Table of Contents

1. Getting Started with HFSS

System Requirements 1-2
Windows 1-2
Sun Solaris 1-4
HP-UX 1-5
Red Hat Enterprise Linux v3 1-5
The HFSS Desktop 1-6
Working with the Menu Bar 1-6
Working with the Toolbars 1-8
Working with the Shortcut Menus 1-8
Shortcut Menu in the Toolbars Area 1-8
Shortcut Menu in the 3D Modeler Window 1-9
Shortcut Menus in the Project Manager Window 1-10
Working with the Status Bar 1-11
Working with the Project Manager 1-11
Working with the Project Tree 1-12
Setting the Project Tree to Expand Automatically 1-12
Viewing HFSS Design Details 1-12
Viewing the Design List 1-13
Viewing Material Definitions 1-14
Working with the Properties Window 1-14
Showing and Hiding the Properties Window 1-15
Setting the Properties Window to Open Automatically 1-15
Modifying Object Attributes Using the Properties Window 1-16
Modifying Object Command Properties Using the Properties Window . . . 1-16
Working with the Progress Window 1-16

Working with the Message Manager 1-17
Working with the 3D Modeler Window 1-17
Working with the History Tree 1-18
Controlling the View of Objects in the History Tree 1-20
Using the Password Manager to Control Access to Resources 1-21
Running HFSS from a command line 1-22
Getting Started Guides 1-24
Copyright and Trademark Notices 1-26

2. Getting Help

Conventions Used in the Online Help 2-2
Searching in Help 2-3
Using WebUpdate 2-4

3. Working with HFSS Projects

HFSS Files 3-2
Creating Projects 3-3
Projects 3-4
Opening Recent Projects 3-4
Opening Legacy HFSS Projects 3-4
Legacy HFSS Project Translation 3-4
Closing Projects 3-7
Saving Projects 3-8
Saving a New Project 3-8
Saving the Active Project 3-9
Saving a Copy of a Project 3-9
Renaming a Project 3-10
Saving Project Data Automatically 3-10
Save Before Solve Option 3-11
Recovering Project Data in an Auto-Save File 3-11
Deleting Projects 3-12
Undoing Commands 3-13
Redoing Commands 3-14
Validating Projects 3-15
Exporting Files 3-17
Exporting 2D Geometry Files 3-17
Exporting 3D Model Files 3-17
Exporting Graphics Files 3-18
Exporting Data Table Files 3-19
Importing Files 3-20

Importing 2D Model Files 3-20
Importing 3D Model Files 3-21
Importing Solution Data 3-22
Importing Data Tables 3-22
Inserting a Documentation File 3-24
Printing 3-25
Saving Project Notes 3-26
Setting Options in HFSS 3-27
Setting General Options 3-27
General Options: Project Options Tab 3-27
General Options: Default Units Tab 3-28
General Options: Analysis Options Tab 3-28
General Options: WebUpdate Options Tab 3-29
General Options: Distributed Analysis Options Tab 3-29
Setting HFSS Options 3-29
HFSS Options: General Options Tab 3-29
HFSS Options: Solver Tab 3-30
Setting Fields Reporter Options 3-30
Setting Report2D Options 3-31
Report2D Options: Table Tab 3-31
Report2D Options: General Options Tab 3-31
Report2D Options: Line Style Tab 3-32
Report2D Options: Color Tab 3-32
Report2D Options: Graphics Font Tab 3-32
Setting 3D Modeler Options 3-32
3D Modeler Options: Operation Tab 3-33
3D Modeler Options: Display Tab 3-33
3D Modeler Options: Drawing Tab 3-33
Working with Variables 3-35
Adding a Project Variable 3-35
Deleting Project Variables 3-36
Adding a Design Variable 3-36
Deleting Design Variables 3-37
Defining an Expression 3-37
Using Valid Operators for Expressions 3-38
Using Intrinsic Functions in Expressions 3-38
Using Piecewise Linear Functions in Expressions 3-40
Using Dataset Expressions 3-40
Adding Datasets 3-40
Modifying Datasets 3-41
Defining Mathematical Functions 3-41
Assigning Variables 3-42

Choosing a Variable to Optimize 3-42
Including a Variable in a Sensitivity Analysis 3-43
Choosing a Variable to Tune 3-43
Including a Variable in a Statistical Analysis 3-44

4. Setting up an HFSS Design

Inserting an HFSS Design 4-2
Selecting the Solution Type 4-3
Setting the Model's Units of Measurement 4-4

5. Drawing a Model

Drawing Objects 5-2
Drawing a Straight Line Segment 5-2
Drawing a Three-Point Arc Line 5-3
Drawing a Center-Point Arc Line 5-4
Drawing a Spline 5-4
Drawing a Polyline 5-5
Inserting Line Segments 5-6
Drawing an Equation-Based Curve 5-7
Drawing a Circle 5-7
Drawing an Ellipse 5-8
Drawing a Rectangle 5-9
Drawing a Regular Polygon 5-10
Drawing an Equation-Based Surface 5-10
Drawing a Sphere 5-11
Drawing a Cylinder 5-11
Drawing a Box 5-12
Drawing a Regular Polyhedron 5-13
Drawing a Cone 5-13
Drawing a Torus 5-14
Drawing a Helix 5-15
Drawing a Spiral 5-16
Drawing a Spiral using User Defined Primitives 5-17
Drawing a Bondwire 5-18
Drawing a Point 5-19
Drawing a Plane 5-19
Drawing Non-Model Objects 5-20
Selecting Non-Model Drawing Mode 5-20
Changing an Object to Non Model 5-21
Model Analysis 5-22
Analysis Options Dialog 5-22

Model Analysis dialog 5-23
Objects Tab 5-23
Object Misalignment Tab 5-24
Surface Mesh (Single/Pairs) Tab 5-24
Last Simulation Mesh Tab 5-25
Align Faces 5-25
Heal 5-26
Healing Stage One: Validation Check 5-26
Healing Stage Two: Fix Intersections Between Objects 5-27
Healing Stage Three: Fix Object Pair Intersections 5-28
Healing Stage Four: Analysis Phase 5-28
Healing Non-Manifold Objects 5-28
Setting the Healing Options 5-28
Creating a User Defined Primitive 5-30
Modifying Objects 5-33
Assigning Color to Objects 5-34
Setting the Default Color of Objects 5-34
Setting the Default Color of Object Outlines 5-34
Assigning Transparency to an Object 5-34
Setting the Default Transparency of Objects 5-35
Copying and Pasting Objects 5-35
Copying Objects to the Clipboard 5-35
Deleting Objects 5-36
Deleting Polyline Segments 5-36
Deleting Start Points and Endpoints 5-36
Delete Last Operation 5-37
Moving Objects 5-37
Rotating Objects 5-37
Mirroring Objects 5-38
Offsetting Objects 5-38
Duplicating Objects 5-39
Duplicating Objects Along a Line 5-39
Duplicating Objects Around an Axis 5-40
Duplicating and Mirroring Objects 5-40
Scaling Objects 5-41
Sweeping Objects 5-41
Sweeping Around an Axis 5-41
Sweeping Along a Vector 5-42
Sweeping Along a Path 5-43
Covering Lines 5-44
Covering Faces 5-44
Uncovering Faces 5-44

Detaching Faces 5-44
Creating a Cross-Section 5-45
Connecting Objects 5-45
Moving Faces 5-45
Moving Faces Along the Normal 5-46
Moving Faces Along a Vector 5-46
Uniting Objects 5-47
Subtracting Objects 5-48
Creating Objects from Intersections 5-49
Creating an Object from a Face 5-50
Creating an Object from an Edge 5-51
Splitting Objects 5-51
Separating Bodies 5-52
Converting Polyline Segments 5-53
Rounding the Edge of Objects (Fillet Command) 5-54
Flattening the Edge of Objects (Chamfer Command) 5-54
Purge History 5-54
Generate History 5-55
Selecting Items in the 3D Modeler Window 5-57
Selecting Objects 5-57
Selecting Multiple Objects 5-57
Selecting Objects by Name 5-58
Setting the Default Color and Transparency of Selected Objects 5-58
Setting the Default Color of Highlighted Objects 5-59
Creating an Object List 5-59
Reassigning Objects to Another Object List 5-59
Selecting Faces 5-60
Selecting All Faces of an Object 5-60
Selecting Faces by Name 5-61
Selecting Faces by Plane 5-61
Face Selection Toolbar Icons 5-62
Creating a Face List 5-62
Selecting Edges 5-62
Selecting Vertices 5-63
Clearing a Selection 5-64
Selecting the Face or Object Behind 5-64
Selecting Cartesian Coordinates 5-64
Selecting Cylindrical Coordinates 5-65
Selecting Spherical Coordinates 5-65
Selecting Absolute Coordinates 5-66
Selecting Relative Coordinates 5-66

Choosing the Movement Mode 5-68
Moving the Cursor In Plane 5-68
Moving the Cursor Out of Plane 5-68
Moving the Cursor in 3D Space 5-69
Moving the Cursor Along the X-Axis 5-70
Moving the Cursor Along the Y-Axis 5-71
Moving the Cursor Along the Z-Axis 5-71
Choosing Snap Settings 5-72
Snap Setting Guidelines 5-73
Measure Modes for Objects 5-74
Measuring Position 5-74
Measuring Length 5-75
Measuring the Distance Between Two Points 5-76
Measuring Area 5-77
Measuring Volume 5-77
Setting Coordinate Systems 5-78
Setting the Working Coordinate System 5-78
Creating a Relative Coordinate System 5-79
Creating an Offset Relative CS 5-79
Creating a Rotated Relative CS 5-79
Creating an Offset and Rotated Relative CS 5-80
Creating a Face Coordinate System 5-81
Automatically Creating Face Coordinate Systems 5-81
Modifying Coordinate Systems 5-82
Deleting Coordinate Systems 5-82

6. Assigning Boundaries

Zoom to Selected Boundary 6-2
Assigning Perfect E Boundaries 6-3
Assigning Perfect H Boundaries 6-4
Assigning Impedance Boundaries 6-5
Assigning Radiation Boundaries 6-6
Assigning PML Boundaries 6-8
Creating PMLs Automatically 6-8
Creating PML Boundaries Manually 6-10
Guidelines for Assigning PML Boundaries 6-11
Modifying PML Boundaries 6-12
Assigning Finite Conductivity Boundaries 6-13
Assigning Symmetry Boundaries 6-14
Assigning Master Boundaries 6-15

Assigning Slave Boundaries 6-16
Assigning Lumped RLC Boundaries 6-18
Assigning Layered Impedance Boundaries 6-19
Designating Infinite Ground Planes 6-21
Modifying Boundaries 6-22
Deleting Boundaries 6-23
Reassigning Boundaries 6-24
Reprioritizing Boundaries 6-25
Global Material Environment 6-26
Duplicating Boundaries and Excitations with Geometry 6-27
Showing and Hiding Boundaries and Excitations 6-28
Showing and Hiding Boundaries and Excitations in the Active View Window 6-28
Showing and Hiding Boundaries and Excitations in Every View Window 6-29
Reviewing Boundaries and Excitations in the Solver View 6-30
Setting Default Values for Boundaries and Excitations 6-31

7. Assigning Excitations

Zoom to Selected Excitation 7-2
Assigning Wave Ports 7-3
Assigning Wave Ports for Modal Solutions 7-3
Assigning Wave Ports for Terminal Solutions 7-4
Assigning Lumped Ports 7-7
Assigning Lumped Ports for Modal Solutions 7-7
Assigning Lumped Ports for Terminal Solutions 7-8
Defining an Integration Line 7-10
Guidelines for Defining Integration Lines 7-10
Duplicating Integration Lines 7-11
Modifying Integration Lines 7-11
Defining a Terminal Line 7-12
Guidelines for Defining Terminal Lines 7-12
Duplicating Terminal Lines 7-13
Modifying Terminal Lines 7-13
Setting up Differential Pairs 7-13
Assigning Incident Waves 7-15
Incident Plane Wave 7-15
Incident Hertzian-Dipole Wave 7-17
Incident Cylindrical Wave 7-18
Incident Gaussian Beam Wave 7-19
Incident Linear Antenna Wave 7-20

Far Field Wave 7-21
Near Field Wave 7-23
Assigning Voltage Sources 7-27
Modifying Voltage Sources 7-27
Assigning Current Sources 7-28
Modifying Current Sources 7-28
Assigning Magnetic Bias Sources 7-29
Modifying Excitations 7-30
Deleting Excitations 7-31
Reassigning Excitations 7-32
Duplicating Excitations with Geometry 7-33
Showing and Hiding Excitations 7-34
Setting the Impedance Multiplier 7-35
Renormalizing S-Matrices 7-36
De-embedding S-Matrices 7-37

8. Assigning Materials

Solving Inside or on the Surface 8-3
Searching for Materials 8-4
Searching by Material Name 8-4
Searching by Material Property 8-4
Adding New Materials 8-5
Assigning Material Property Types 8-5
Defining Anisotropic Relative Permeability Tensors 8-6
Defining Anisotropic Relative Permittivity Tensors 8-6
Defining Anisotropic Conductivity Tensors 8-6
Defining Anisotropic Dielectric Loss Tangent Tensors 8-7
Defining Magnetic Loss Tangent Tensors 8-7
Defining Variable Material Properties 8-8
Defining Frequency-Dependent Material Properties 8-8
Defining Frequency-Dependent Material Properties for Lossy Dielectrics . 8-9
Enter Frequency Dependent Data Points 8-10
Defining Material Properties as Expressions 8-11
Defining Functional Material Properties 8-11
Viewing and Modifying Material Attributes 8-12
Validating Materials 8-13
Copying Materials 8-14
Removing Materials 8-15
Exporting Materials to a Library 8-16
Sorting Materials 8-17

Filtering Materials 8-18
Working with Material Libraries 8-19
Working with Ansoft's System Material Library 8-19
Working with User Material Libraries 8-19
Editing Libraries 8-19
Configuring Libraries 8-19

9. Assigning DC Thickness

10. Modifying the Model View

Rotating the View 10-2
Panning the View 10-3
Zooming In and Out 10-4
Zooming In or Out on a Rectangular Area 10-4
View Options: 3D UI Options 10-6
Fitting Objects in the View Window 10-7
Fitting All Objects in a View Window 10-7
Fitting a Selection in a View Window 10-7
Hiding Objects from View 10-8
Showing Objects 10-9
Rendering Objects as Wireframes or Solids 10-10
Setting the Default View Rendering Mode 10-10
Modifying the View Orientation 10-11
Applying a Default View Orientation 10-11
Applying a New View Orientation 10-11
Removing an Orientation 10-12
Modifying the Lighting 10-13
Setting the Projection View 10-14
Setting the Background Color 10-15
Modifying the Coordinate System Axes View 10-16
Showing or Hiding the Axes 10-16
Show the Axes for Selected Objects 10-16
Enlarging or Shrinking the Axes 10-16
Choosing Grid Settings 10-17
Setting the Grid Type 10-17
Setting the Grid Style 10-17
Setting the Grid Density and Spacing 10-17
Setting the Grid's Visibility 10-18
Setting the Grid Plane 10-18

11. Defining Mesh Operations

Assigning Length-Based Mesh Refinement on Object Faces 11-2
Assigning Length-Based Mesh Refinement Inside Objects 11-3
Assigning Skin Depth-Based Mesh Refinement on Object Faces 11-4
Modifying Surface Approximation Settings 11-5
Specifying the Model Resolution 11-6
Reverting to the Initial Mesh 11-7
Applying Mesh Operations without Solving 11-8

12. Specifying Solution Settings

Add Dependent Setup 12-2
Setting the Solution Frequency 12-3
Solving for Ports Only 12-4
Setting the Minimum Frequency 12-5
Setting the Number of Modes 12-6
Setting Adaptive Analysis Parameters 12-7
Setting the Maximum Number of Passes 12-7
Setting the Maximum Delta S Per Pass 12-8
Setting the Maximum Delta Energy Per Pass 12-8
Setting the Maximum Delta Frequency Per Pass 12-8
Specifying Convergence on Real Frequency Only 12-9
Specifying Output Variable Convergence 12-9
Specifying a Source for the Initial Mesh 12-10
Clearing Linked Data 12-11
Setting Lambda Refinement 12-11
Setting the Percent Maximum Refinement Per Pass 12-11
Setting the Maximum Refinement 12-11
Setting the Minimum Number of Passes 12-12
Setting the Minimum Number of Converged Passes 12-12
Setting Matrix Convergence Criteria 12-12
Using a Low-Order Solution Basis 12-13
Use Absorbing Boundary (ABC) on Ports 12-14
Waveport Adapt Options 12-14
Adding a Frequency Sweep 12-15
Selecting the Sweep Type 12-15
Options for Discrete Sweeps 12-16
Options for Fast Sweeps 12-16
Options for Interpolating Sweeps 12-17
Setup Interpolation Convergence 12-17
Setting the Error Tolerance 12-18

Setting the Maximum Number of Solutions 12-18
Interpolation Basis Convergence 12-19
Specifying the Frequency Points to Solve 12-20
Specifying Frequency Points with a Linear Step Size 12-20
Specifying a Linear Count of Frequency Points 12-20
Specifying Single Frequency Points 12-21
Deleting Frequency Points 12-21
Inserting Frequency Points 12-21
Choosing Frequencies for Full-Wave SPICE 12-22
Guidelines for Calculating Frequencies for Full-Wave SPICE 12-22
Disabling a Frequency Sweep 12-24
Disabling an Analysis Setup 12-25
Specifying the Number of Processors 12-26
Specifying the Desired RAM Limit 12-27
Specifying the Hard Memory Limit 12-28

13. Running Simulations

Solving Remotely (Windows only) 13-3
Setup for remote machine 13-3
Preconfiguring the Remote Machine 13-3
Configure Distributed COM for Remote Machine 13-3
Windows 2000 for Remote Machine 13-3
Windows XP/Server 2003 for Remote Machine 13-4
Enable Firewall Access for Remote Machine 13-6
Configure the Temporary File Directory for Remote Machine 13-7
Set up Security Permissions on Remote Machine 13-7
Solve a Test Design as a Remote User on a Remote Machine 13-7
Setup for local machine 13-8
Pre-configuring the Local Machine 13-8
Configuring Distributed COM for callback 13-8
Windows 2000 for Local Machine 13-8
Windows XP/Server 2003 for Local Machine 13-8
Enable Firewall Access for Local Machine 13-9
Configure Remote Analysis Dialog for Local Machine 13-10
Troubleshooting 13-11
Distributed Solve with Licensing 13-12
Monitoring the Solution Process 13-14
Monitoring Queued Simulations 13-14
Changing the Solution’s Priority 13-15
Aborting Analyses 13-16
Re-solving a Problem 13-17

14. Using Optimetrics for Design Analysis

Setting up a Parametric Analysis 14-3
Adding a Variable Sweep Definition 14-3
Specifying Variable Values for a Sweep Definition 14-4
Synchronizing Variable Sweep Definitions 14-4
Modifying a Variable Sweep Definition Manually 14-5
Overriding a Variable's Current Value in a Parametric Setup 14-5
Specifying a Solution Setup for a Parametric Setup 14-6
Specifying the Solution Quantity to Evaluate 14-6
Specifying a Solution Quantity's Calculation Range 14-7
Setting up an Optimization Analysis 14-9
Setting the Maximum Iterations for an Optimization Analysis 14-10
Adding a Cost Function 14-10
Specifying a Solution Quantity for a Cost Function Goal 14-11
Setting the Calculation Range of a Cost Function Goal 14-12
Setting a Goal Value 14-12
Specifying a Single Goal Value 14-13
Specifying an Expression as a Goal Value 14-13
Specifying a Variable-Dependent Goal Value 14-13
Overriding the Min. and Max. Variable Values for a Single Optimization Setup 14-15
Changing the Min. and Max. Variable Values for Every Optimization Setup 14-15
Setting the Min. and Max. Step Sizes 14-15
Setting the Min and Max Focus 14-16
Solving a Parametric Setup Before an Optimization 14-17
Solving a Parametric Setup During an Optimization 14-17
Automatically Updating a Variable's Value After Optimization 14-17
Setting up a Sensitivity Analysis 14-19
Setting the Maximum Iterations Per Variable 14-19
Setting up an Output Parameter 14-20
Specifying a Solution Quantity for an Output Parameter 14-20
Setting the Calculation Range of an Output Parameter 14-21
Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup 14-22
Changing the Min. and Max. Variable Values for Every Sensitivity Setup 14-22
Setting the Initial Displacement 14-23
Solving a Parametric Setup Before a Sensitivity Analysis 14-23
Solving a Parametric Setup During a Sensitivity Analysis 14-23
Setting up a Statistical Analysis 14-24
Setting the Maximum Iterations for a Statistical Analysis 14-24
Setting the Distribution Criteria 14-26
Overriding the Distribution Criteria for a Single Statistical Setup 14-26
Changing the Distribution Criteria for Every Statistical Setup 14-27

Solving a Parametric Setup During a Statistical Analysis 14-28
Tuning a Variable 14-29
Applying a Tuned State to a Design 14-29
Saving a Tuned State 14-30
Reverting to a Saved Tuned State 14-30
Resetting Variable Values after Tuning 14-30
Saving Field Solutions for Optimetrics Analyses 14-31
Saving Field Solutions for a Parametric Setup 14-31
Saving Field Solutions for an Optimization Setup 14-31
Saving Field Solutions for a Sensitivity Setup 14-32
Saving Field Solutions for a Tuning Analysis 14-32
Saving Field Solutions for a Statistical Setup 14-33
Copying Meshes in Optimetrics Sweeps 14-34
Excluding a Variable from an Optimetrics Analysis 14-37
Modifying the Value of a Fixed Variable for Statistical Analysis 14-38
Setting a Linear Constraint 14-39
Modifying a Linear Constraint 14-39
Deleting a Linear Constraint 14-40

15. Post Processing and Generating Reports

Viewing Solution Data 15-2
Viewing Convergence Data 15-2
Viewing the Number of Completed Passes 15-3
Viewing the Max Magnitude of Delta S Between Passes 15-3
Viewing the Output Variable Convergence 15-4
Viewing the Delta Magnitude Energy 15-4
Viewing the Magnitude Margin 15-5
Viewing the Phase Margin 15-5
Viewing the Max Delta (Mag S) 15-5
Viewing the Max Delta (Phase S) 15-6
Viewing the Maximum Delta Frequency 15-6
Plotting Convergence Data 15-6
Viewing a Solution's Profile 15-7
Viewing Matrix Data 15-8
Selecting the Matrix Display Format 15-9
Exporting Matrix Data 15-9
Renaming Matrix Data 15-11
Exporting Equivalent Circuit Data 15-11
Exporting W-Element Data 15-13
Viewing Eigenmode Solution Data 15-13
Deleting Solution Data 15-14

Viewing Analysis Results for Optimetrics Solutions 15-15
Viewing Solution Data for an Optimetrics Design Variation 15-15
Viewing an Optimetrics Solution's Profile Data 15-15
Viewing Results for Parametric Solution Quantities 15-16
Plotting Solution Quantity Results vs. a Swept Variable 15-16
Viewing Cost Results for an Optimization Analysis 15-17
Plotting Cost Results for an Optimization Analysis 15-17
Viewing Output Parameter Results for a Sensitivity Analysis 15-17
Plotting Output Parameter Results for a Sensitivity Analysis 15-18
Viewing Distribution Results for a Statistical Analysis 15-18
Plotting Distribution Results for a Statistical Analysis 15-18
Scaling a Source's Magnitude and Phase 15-21
Guidelines for Scaling a Source's Magnitude and Phase 15-23
Creating Animations 15-24
Creating Phase Animations 15-24
Creating Frequency Animations 15-24
Creating Geometry Animations 15-25
Controlling the Animation's Display 15-26
Exporting Animations 15-27
Plotting Field Overlays 15-29
Plotting Derived Field Quantities 15-30
Creating Scalar Field Plots 15-30
Modifying SAR Settings 15-31
Creating Vector Field Plots 15-31
Modifying Field Plots 15-31
Setting Field Plot Attributes 15-32
Modifying Field Plot Colors 15-33
Setting the Color Key Visibility 15-33
Moving the Color Key 15-34
Modifying the Field Plot Scale 15-34
Modifying Vector Field Plot Arrows 15-35
Setting the Mesh Visibility on Field Plots 15-36
Modifying Scalar Field Plot Isovalues 15-36
Mapping Scalar Field Plot Transparency to Field Values 15-37
Modifying Markers on Point Plots 15-37
Modifying Line Plots 15-38
Setting a Plot's Visibility 15-39
Saving a Field Overlay Plot 15-39
Opening a Field Overlay Plot 15-40
Deleting a Field Overlay Plot 15-40
Setting Field Plot Defaults 15-40

Using the Fields Calculator 15-42
Opening the Fields Calculator 15-42
Context Area 15-44
The Calculator Stack 15-44
Registers 15-44
Enlarging the Register Display Area 15-45
Units of Measure 15-45
Stack Commands 15-45
Input Commands 15-46
Quantity Command 15-46
Geometry Command 15-47
Constant Command 15-48
Number Command 15-49
Function Command 15-49
Geom Settings Command 15-50
Read Command 15-50
General Commands 15-50
Scalar Commands 15-52
Vec? Command 15-53
1/x (Inverse) Command 15-53
Pow Command 15-53
(Square Root) Command 15-53
Trig 15-53
d/d? (Partial Derivative) Command 15-54
(Integral) Command 15-54
Min Command 15-54
Max Command 15-55
∇ (Gradient) Command 15-55
Ln Command 15-55
Log Command 15-55
Vector Commands 15-56
Scal? Command 15-56
Matl Command 15-56
Mag Command 15-57
Dot Command 15-57
Cross Command 15-57
Divg Command 15-57
Curl Command 15-57
Tangent Command 15-57
Normal Command 15-58
Unit Vec Command 15-58
Output Commands 15-59
Value Command 15-59

Eval Command 15-59
Write Command 15-60
Export Command 15-61
Calculating Derived Field Quantities 15-62
Named Expression Library 15-63
Exiting the Fields Calculator 15-64
Radiated Fields Post Processing 15-65
Setting up a Near-Field Sphere 15-65
Setting up a Near-Field Line 15-67
Computing Maximum Near-Field Parameters 15-67
Setting up a Far-Field Infinite Sphere 15-68
Defining Antenna Arrays 15-70
Defining a Regular Antenna Array 15-70
Defining a Custom Antenna Array 15-71
Computing Antenna Parameters 15-71
Exporting Antenna Parameters and Maximum Field Data 15-73
Plotting the Mesh 15-74
Setting Mesh Plot Attributes 15-74
Creating Reports 15-77
Creating a Quick Report 15-77
Creating a Custom Report 15-77
Modifying Reports 15-78
Modifying the Background Properties of a Report 15-78
Selecting the Report Type 15-80
Selecting the Display Type 15-81
Creating 2D Rectangular Plots 15-81
Creating 3D Rectangular Plots 15-82
Creating 2D Polar Plots 15-83
Reviewing 2D Polar Plots 15-84
Creating 3D Polar Plots 15-84
Creating Smith Charts 15-86
Creating Data Tables 15-86
Creating Radiation Patterns 15-87
Delta Markers in 2D Reports 15-88
Plotting in the Time Domain 15-89
Working with Traces 15-90
Removing Traces 15-90
Replacing Traces 15-91
Adding Blank Traces 15-91
Sweeping a Variable in a Report 15-91
Sweeping Values Across a Normalized Distance 15-92
Sweeping Values Across a Sphere 15-92

Refresh Sweeps 15-93
Selecting a Function 15-93
Selecting Solution Quantities to Plot 15-95
Selecting a Field Quantity to Plot 15-97
Selecting a Far-Field Quantity to Plot 15-97
Plotting Vertical Cross-Sections of Far Fields 15-101
Plotting Horizontal Cross-Sections of Far Fields 15-101
Selecting a Near-Field Quantity to Plot 15-102
Selecting an Emission Test Quantity to Plot 15-103
Plotting Imported Solution Data 15-104
Specifying Output Variables 15-105
Adding a New Output Variable 15-105
Building an Expression Using Existing Quantities 15-105
Deleting Output Variables 15-106

16. Technical Notes

The Finite Element Method 16-2
Representation of a Field Quantity 16-2
Basis Functions 16-2
Size of Mesh Vs. Accuracy 16-3
The HFSS Solution Process 16-4
The Mesh Generation Process 16-4
Seeding the Mesh 16-5
Guidelines for Seeding the Mesh 16-5
Length-Based Mesh Refinement 16-6
Skin Depth-Based Mesh Refinement 16-6
Surface Approximation Settings 16-7
Guidelines for Modifying Surface Approximation Settings 16-8
Meshing Region Vs. Problem Region 16-8
Model Resolution 16-9
Port Solutions 16-10
Excitation Fields 16-10
Wave Equation 16-10
Mesh Refinement on Ports 16-11
Modes 16-11
Mode Conversion 16-12
Modes, Reflections, and Propagation 16-12
Modal Field Patterns and Frequency 16-12
Multiple Ports on the Same Face 16-12
Port Field Accuracy 16-13
Saving Field Solutions 16-13
The Adaptive Analysis Process 16-13

Maximum Delta S 16-14
Maximum Delta E 16-14
Percent of Tetrahedra Refined Per Pass 16-15
Magnitude Margin 16-15
Phase Margin 16-15
Maximum Delta Frequency 16-16
Max Delta (Mag S) 16-16
Max Delta (Phase S) 16-16
Single Frequency Solution 16-16
Frequency Sweeps 16-17
Fast Frequency Sweeps 16-18
Discrete Frequency Sweeps 16-19
Interpolating Frequency Sweeps 16-20
Solution Types 16-21
Eigenmode Solutions 16-22
Calculating the Resonant Frequency 16-22
Calculating the Quality Factor 16-22
Calculating the Free Space Wave Number 16-23
Field Solutions 16-23
Field Overlay Plots 16-23
Field Quantities 16-24
Specifying the Phase Angle 16-24
Peak Versus RMS Phasors 16-24
Calculating the SAR 16-26
S-Parameters 16-27
Renormalized S-Matrices 16-27
Calculating Characteristic Impedance 16-28
Renormalizing to Zpv or Zvi Impedances 16-29
Calculating the PI Impedance 16-29
Calculating the PV Impedance 16-29
Calculating the VI Impedance 16-29
Impedance Multipliers 16-30
Calculating Terminal Characteristic Impedance Matrix 16-30
Calculating the S-Matrix 16-30
Calculating the Z-Matrix 16-31
Calculating the Y-Matrix 16-32
Calculating the Complex Propagation Constant (Gamma) 16-32
Calculating the Effective Wavelength (Lambda) 16-32
Calculating the Relative Permittivity (Epsilon) 16-32
De-embedded S-Matrices 16-33
Radiated Fields 16-34
Spherical Cross-Sections 16-35
Maximum Near-Field Data 16-36

Maximum Far-Field Data 16-37
Array Factors 16-38
Theory of the Array Factor Calculation 16-39
Regular Uniform Arrays 16-40
Scan Specification for Regular Uniform Arrays 16-40
Custom Arrays 16-41
Power Normalizations 16-42
Antenna Parameters 16-43
Polarization of the Electric Field 16-43
Spherical Polar 16-44
Ludwig-3 Polarization 16-45
Circular Polarization 16-45
Axial Ratio 16-45
Polarization Ratio 16-46
Max U 16-47
Peak Directivity 16-47
Peak Gain 16-47
Peak Realized Gain 16-48
Radiated Power 16-48
Accepted Power 16-49
Incident Power 16-50
Radiation Efficiency 16-50
Calculating Finite Thickness Impedance 16-52
Geometric Objects 16-53
Bondwires 16-53
Healing Models 16-54
Error types 16-54
Error detection 16-55
Boundaries 16-56
Perfect E Boundaries 16-56
Impedance Boundaries 16-56
Units of Impedance Boundaries 16-57
Radiation Boundaries 16-58
PML Boundaries 16-58
Material Tensors Applied at PML Boundaries 16-59
Tensor Entries 16-60
Boundaries at PML Surfaces 16-61
Finite Conductivity Boundaries 16-61
Symmetry Boundaries 16-61
Perfect E Vs. Perfect H Symmetry Boundaries 16-62
Symmetry and Port Impedance 16-63
Symmetry and Multiple Modes 16-63

Master and Slave Boundaries 16-63
Calculating the E-Field on the Slave Boundary 16-64
Lumped RLC Boundaries 16-65
Layered Impedance Boundaries 16-65
Impedance Calculation for Layered Impedance Boundary 16-66
Surface Roughness Calculation for Layered Impedance Boundary 16-67
Infinite Ground Planes 16-67
Frequency-Dependent Boundaries and Excitations 16-68
Default Boundary Assignments 16-68
Excitations 16-69
Wave Ports 16-69
Polarizing the E-Fields 16-69
Lumped Ports 16-70
Setting the Field Pattern Direction 16-70
Differential Pairs 16-71
Computing Differential Pairs 16-71
Differential Admittance and Impedance Matrices 16-73
Differential S-Matrices 16-73
Magnetic Bias Sources 16-74
Uniform Applied Bias Fields 16-74
Non-uniform Applied Bias Fields 16-75
Incident Waves 16-75
Materials 16-77
Relative Permeability 16-77
Relative Permittivity 16-77
Bulk Conductivity 16-78
Dielectric Loss Tangent 16-78
Magnetic Loss Tangent 16-79
Ferrite Materials 16-79
Magnetic Saturation 16-79
Lande G Factor 16-80
Delta H 16-80
Anisotropic Materials 16-80
Anisotropic Relative Permeability Tensors 16-81
Anisotropic Relative Permittivity Tensors 16-81
Anisotropic Conductivity Tensors 16-82
Anisotropic Dielectric Loss Tangent Tensors 16-82
Anisotropic Magnetic Loss Tangent Tensors 16-83
Anisotropic Materials and Ports 16-84
Frequency-Dependent Material Properties 16-85
Frequency Dependent Material Loss Model in HFSS 16-86
Conclusions 16-95

Parametric Overview 16-97
Optimization Overview 16-98
Choosing an Optimizer 16-98
Optimization Variables and the Design Space 16-99
Cost Function 16-99
Acceptable Cost 16-100
Cost Function Noise 16-100
Linear Constraints 16-100
Goal Weight 16-101
Step Size 16-102
Sensitivity Analysis Overview 16-104
Selecting a Master Output 16-104
Tuning Analysis Overview 16-105

17. Scripting

Recording a Script 17-1
Stopping Script Recording 17-1
Running a Script 17-2
Pausing and Resuming a Script 17-2
Stopping a Script 17-3

18. Glossaries

1

Getting Started with HFSS

HFSS is an interactive software package for calculating the electromagnetic behavior of a structure. The software includes post-processing commands for analyzing this behavior in detail.

Using HFSS, you can compute:

- Basic electromagnetic field quantities and, for open boundary problems, radiated near and far fields.
- Characteristic port impedances and propagation constants.
- Generalized S-parameters and S-parameters renormalized to specific port impedances.
- The eigenmodes, or resonances, of a structure.

You are expected to draw the structure, specify material characteristics for each object, and identify ports and special surface characteristics. HFSS then generates the necessary field solutions and associated port characteristics and S-parameters.

Note If you are using the Eigenmode Solution solver, you do not need to specify sources for the problem. HFSS calculates the resonances for the model based on the geometry, materials, and boundaries.

As you set up the problem, HFSS allows you to specify whether to solve the problem at one specific frequency or at several frequencies within a range.

System Requirements

HFSS supports the following operating systems:

- Windows
- Sun Solaris
- HP-UX
- Red Hat Enterprise Linux v3

On all systems when you run HFSS for the first time (that is, with no project directory specified in the registry), or if the project directory or the temp directory does not exist, HFSS displays a dialog that asks you to set the project and temp directories. For the temp directory, there is a comment asking you to ensure that adequate disk space is available.

HFSS 10 requires at least FLEXlm for Ansoft v10.1 license server.

Note If you attempt to run HFSS and get a message reporting a corrupted license file, please contact Ansoft.

Windows

Supported Platforms

- Windows 2000 Professional
- Windows XP Professional
- Windows XP Professional x64 Edition
- Windows Server 2003

32-Bit System Requirements

Minimum System Requirements

Processor: All fully compatible 686 (or later) instruction set processors, 500 MHz

Hard Drive Space (for HFSS software): 200 MB

RAM: 512 MB

Recommended Minimum Configuration

Processor: All fully compatible 786 (or later) instruction set processors, 1 GHz

Hard Drive Space (for HFSS software and temporary files): 500 MB

RAM: 2 GB

Increasing RAM on 32-Bit PC

Users with the appropriate Windows OS can take advantage of potentially all the installed RAM up to a limit of 3 GB on 32-bit PCs. Doing so also requires setting up the appropriate OS

boot.ini switch (/3GB) to tell the OS that 3 GB is to be used for application space and only one GB for the OS kernel and related overhead.

Note If you are using the NVIDIA Quadro2 MXR/EX video card on Windows 2000 or Windows XP, you should also download Version 40.41 or greater video driver, available for download at http://www.nvidia.com..

64-bit System Requirements

Minimum System Requirements:

Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support Hard Drive Space (for HFSS software): 200 MB

RAM: 2 MB

Recommended Minimum Configuration (for Optimal Performance)

Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support Video card: 128-bit SVGA or PCI Express video card Hard Drive Space (for HFSS software and temporary files): 700 MB

RAM: 8 GB

Sun Solaris

Supported Platforms

- Solaris 8
- Solaris 9

Minimum System Requirements

Processor: UltraSparc v9 processor, 450 MHz

Hard Drive Space (for HFSS software): 550 MB

RAM: 1 GB

Recommended Minimum Configuration

Processor: UltraSparc v9 dual processor or better, 900 MHz

Hard Drive Space (for HFSS software and temporary files): 800 MB

RAM: 4 GB

Note You must install Sun OpenGL libraries before installing and running HFSS. This is available for free download at:

http://wwws.sun.com/software/graphics/opengl/download.html .

Note Some older graphics cards may have minor display issues (such as the appearance of check marks or “t” in a report title. .

HP-UX

Supported Platforms

- HP-UX 11.0
- HP-UX 11.11

Minimum System Requirements

Processor: PA-RISC 2.0 processor, 450 MHz

Hard Drive Space (for HFSS software): 750 MB

RAM: 1 GB

Recommended Minimum Configuration

Processor: PA-RISC 2.0 dual processor or better, 900 MHz

Hard Drive Space (for HFSS software and temporary files): 1.5 GB

RAM: 4 GB

Red Hat Enterprise Linux v3

If you attempt to open an HFSS v9 project in Linux, you receive an error message that the project must first be converted to HFSS v10. This must be done using the -BatchSave command on a non-Linux system running HFSS v10. See the discussion here.

32-bit System Requirements

Minimum System Requirements:

Processor: All fully compatible 686 (or later) instruction set processors, 500 MHz Hard Drive Space (for HFSS software): 200 MB

RAM: 512 MB

Recommended Minimum Configuration (for Optimal Performance):

Processor: All fully compatible 786 (or later) instruction set processors, 2 GHz Hard Drive Space (for HFSS software and temporary files): 700 MB

RAM: 4 GB

64-bit System Requirements

Minimum System Requirements:

Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support Hard Drive Space (for HFSS software): 200 MB

RAM: 2 MB

Recommended Minimum Configuration (for Optimal Performance):

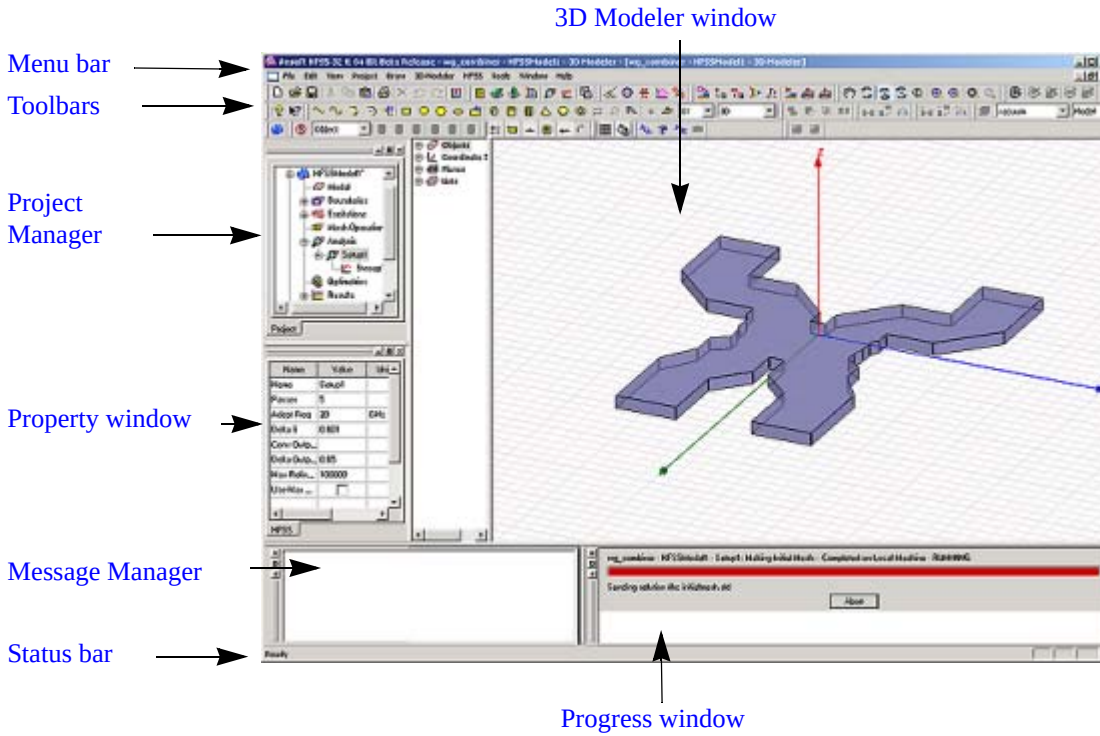
Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support Video card: 128-bit SVGA or PCI Express video card Hard Drive Space (for HFSS software and temporary files): 700 MB

RAM: 8 GB

The HFSS Desktop

The HFSS desktop consists of several windows, a menu bar, toolbars, and a status bar.

Click a link below to view more information about that desktop component.



Working with the Menu Bar

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

To open a help topic about an HFSS menu command, press **Shift**+**F1**, and then click the command or toolbar icon.

HFSS contains the following menus, which appear at the top of the desktop:

File menu	Use the **File** menu commands to manage HFSS project files and printing options.
Edit menu	Use the **Edit** menu commands to modify the objects in the active model and undo and redo actions.
View menu	Use the **View** menu commands to display or hide desktop components and model objects, modify **3D Modeler** window visual settings, and modify the model view.
Project menu	Use the **Project** menu commands to add an HFSS design to the active project, view, define datasets, and define project variables.
Draw menu	Use the **Draw** menu commands to draw one-, two-, or three-dimensional objects, and sweep one- and two-dimensional objects.
3D Modeler menu	Use the **3D Modeler** menu commands to import, export, and copy Ansoft 2D Modeler files and 3D Modeler files; assign materials to objects; manage the **3D Modeler** window's grid settings; define a list of objects or faces of objects; control surface settings; perform boolean operations on objects; and set the units for the active design.
HFSS menu	Use the **HFSS** menu to setup and manage all the parameters for the active project. Most of these project properties also appear in the project tree.
Tools menu	Use the **Tools** menu to modify the active project's material library, arrange the material libraries, run and record scripts, update project definitions from libraries, customize the desktop's toolbars, and modify many of the software's default settings.
Window menu	Use the **Window** menu commands to rearrange the **3D Modeler** windows and toolbar icons.
Help menu	Use the **Help** menu commands to access the online help system and view the current HFSS version information.

Related Topics

Getting Help

Working with the Toolbars

The toolbar buttons and shortcut pull-down lists act as shortcuts for executing various commands.

To execute a command, click a toolbar button or click a selection on the shortcut pull-down list.

To open a help topic about a toolbar button's functionality, press **Shift**+**F1**, and then click the toolbar button or a command in the shortcut pull-down list.

To display a brief description of the toolbar button, move the pointer over the button or shortcut pull-down list.

Hint To modify the toolbars on the desktop, do one of the following:

- On the **Tools** menu, click **Customize**.
- Right-click the history tree, and then click **Customize** on the shortcut menu.

To reset to toolbars to the default positions and settings:

- On the **Tools** menu, click **Customize**. On the **Customize** dialog box, click **Reset All**.

Working with the Shortcut Menus

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

Shortcut menu in the toolbars area	Use the shortcut menu in the toolbars area of the desktop to show or hide windows or toolbars, and customize the toolbars.
Shortcut menu in the 3D Modeler window	Use the shortcut menu in the **3D Modeler** window to select, magnify, and move options (zoom, rotate, etc.), change the view, perform boolean operations, assign materials, boundaries, excitations, or mesh operations to objects, and work with field overlays.
Shortcut menus in the Project Manager window	Use the shortcut menus in the **Project Manager** window to manage HFSS project and design files and design properties; assign and edit boundaries, excitations, and mesh operations; add, analyze, and manage solution setups; add optimetrics analyses; create post-processing reports; insert far- and near-field radiation setups; edit project definitions; and, run Ansoft's Maxwell SPICE.

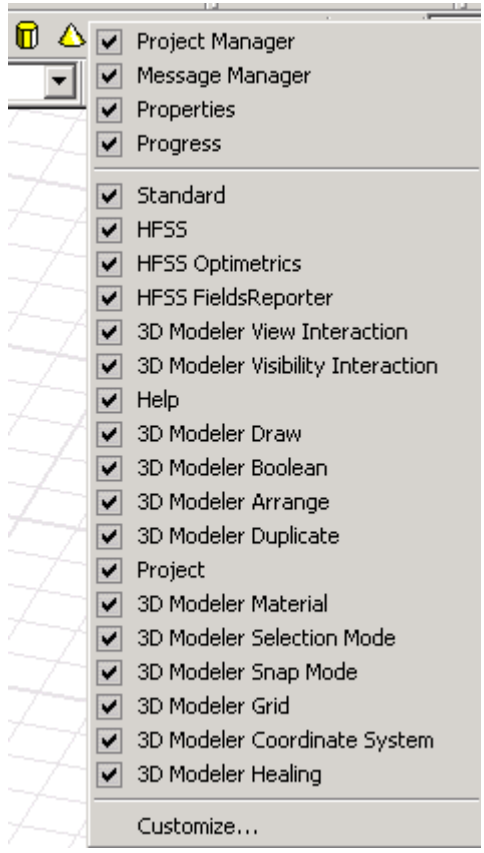
Note All of the commands on the shortcut menus are also available on the menu bar.

Shortcut Menu in the Toolbars Area

Use the shortcut menu in the toolbars area of the desktop to show or hide windows or toolbars, and customize the toolbars.

To access the shortcut menu in the toolbars area:

- Right-click in the toolbars area at the top of the desktop.



A check box will appear next to a command if the item is visible. For example, if a check box appears next to the **Project Manager** command, then the **Project Manager** window is currently visible on the desktop.

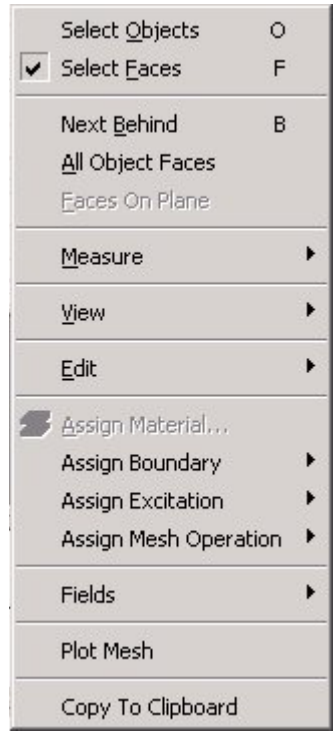
Click **Customize** to open the **Customize** dialog box, which enables you to modify the toolbar settings on the desktop.

Shortcut Menu in the 3D Modeler Window

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

To access the shortcut menu in the **3D Modeler** window:

- Right-click in the **3D Modeler** window (grid area).



Shortcut Menus in the Project Manager Window

Each node, or item, in the project tree has a shortcut menu. For example, from the shortcut menu for the Boundaries icon, you can assign boundaries to selected objects; review information for all the boundary assignments for the active design; remove all boundary assignments; show or hide a boundary's geometry, name, or vectors; change the priority of a previously assigned boundary; and use the **PML Setup** wizard to create a perfectly matched layer (PML) boundary.

Working with the Status Bar

The status bar is located at the bottom of the application window. It displays information about the command currently being performed.

To display or hide the status bar:

- On the **View** menu, click **Status Bar**.

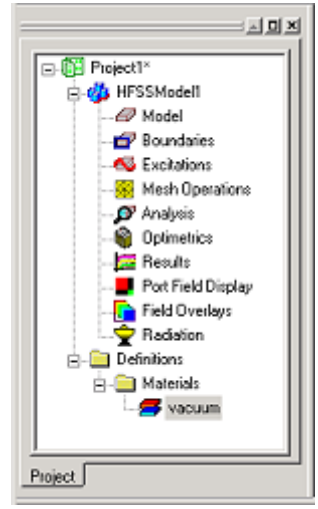
A check box appears next to this command if the status bar is visible.

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

- **X**, **Y**, and **Z** coordinate boxes.
- A pull-down list to enter a point's absolute, relative, cartesian, cylindrical, or spherical coodinates.
- The model's units of measurement.

Working with the Project Manager

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



The **Project Manager** window displays details about all open HFSS projects. Each project ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post-processing information.

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

- On the **View** menu, click **Project Manager**.

  A check box appears next to this command if the **Project Manager** window is visible.
- Right-click in the toolbars area on the desktop, and then click **Project Manager** on the short-

cut menu.

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

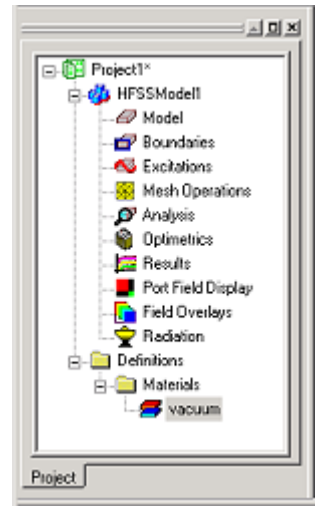
Related Topics

Working with the Project Tree

Shortcut Menus in the Project Manager Window

Working with the Project Tree

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



The top node listed in the project tree is the project name. It is named Project*n* by default, where *n* is the order in which the project was added to the current session of HFSS. Expand the project icon to view all the project's HFSS design information and material definitions.

Related Topics

Viewing HFSS Design Details

Setting the Project Tree to Expand Automatically

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

1. On the **Tools** menu, point to **Options**, and then click **General Options**.

   The **Options** dialog box appears.

2. Under the **Project Options** tab, select **Expand Project Tree on Insert**.

Viewing HFSS Design Details

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

its boundary conditions and material assignments, and field solution and post-processing information.

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

Boundaries	Displays the boundary conditions assigned to an HFSS design, which specify the field behavior at the edges of the problem region and object interfaces.
Excitations	Displays the excitations assigned to an HFSS design, which are used to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces in the design.
Mesh Operations	Displays the mesh operations specified for objects or object faces. Mesh operations are optional mesh refinement settings that are specified before a mesh is generated.
Analysis	Displays the solution setups for an HFSS design. A solution setup specifies how HFSS will compute the solution.
Optimetrics	Displays any Optimetrics setups added to an HFSS design.
Results	Displays any post-processing reports generated.
Port Field Display	Displays all port fields in the active model.
Field Overlays	Displays field overlay plots, which are representations of basic or derived field quantities on surfaces or objects. Plot folders are listed under **Field Overlays.** These folders store the project's plots and can be customized. See *Setting Field Plot Defaults* for information on how to customize the plot folders.
Radiation	Displays far- and near-field setups added to an HFSS design.

Note To edit a project's design details:

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

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

Viewing the Design List

You can use the **HFSS>Design List** command or the Design List icon to view a dialog with tables of the design properties. The Design list is a dialog that with tabs to let you view the following

Model	Displays the objects that comprise the model and their properties.
Boundaries	Displays the boundary conditions assigned to an HFSS design, which specify the field behavior at the edges of the problem region and object interfaces.
Excitations	Displays the excitations assigned to an HFSS design, which are used to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces in the design.
Mesh Operations	Displays the mesh operations specified for objects or object faces. Mesh operations are optional mesh refinement settings that are specified before a mesh is generated.
Analysis Setup	Displays the solution setups for an HFSS design. A solution setup specifies how HFSS will compute the solution.

Viewing Material Definitions

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

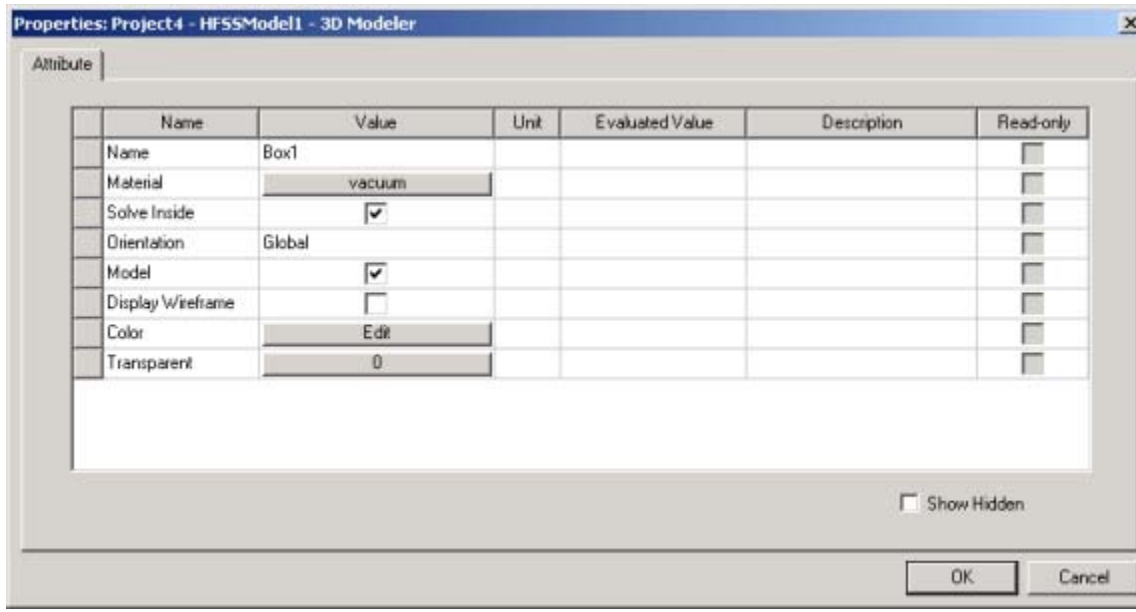
Related Topics

Adding New Materials

Working with the Properties Window

The **Properties** window displays the attributes, or properties, of an item selected in the project tree, the history tree, or the **3D Modeler** window. The **Properties** window enables you to edit an item's properties. The properties, and the ability to edit them in the **Properties** window, will vary, depend-

ing on the type of item selected. The tabs available in the **Properties** window will also vary, depending the selection.



Related Topics

Showing and Hiding the Properties Window

Setting the Properties Window to Open Automatically

Showing and Hiding the Properties Window

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

- On the **View** menu, click **Property Window**.

  A check box appears next to this command if the **Properties** window is visible.
- Right-click in the toolbars area at the top of the desktop, and then click **Properties** on the shortcut menu.

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

Setting the Properties Window to Open Automatically

To set the **Properties** window to open after an object is drawn, enabling you to modify the object's properties, do the following:

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler Options**.

   The **3D Modeler Options** window appears.
2. Click the **Drawing** tab.
3. Select **Edit property of new primitives**.

   Hereafter, after you draw an object, the **Properties** window will open.

Modifying Object Attributes Using the Properties Window

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

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

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

Modifying Object Command Properties Using the Properties Window

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

Not all command properties can be modified. In general, the command properties that you can typically modify are the numeric values, such as position values (base position, normal position, start position, etc.), size values (height, radius, etc.), and various other coordinate values. You can also modify many of the unit settings for a command property.

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

   Hint Press and hold **CTRL** to select multiple commands. If you select multiple commands, only the common, or shared, properties will be displayed under the **Command** tab.

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

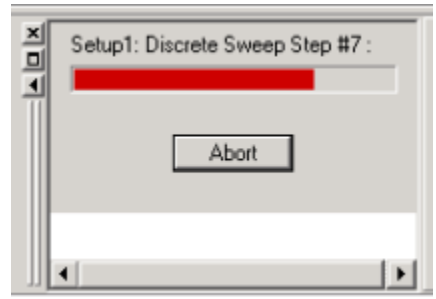
   Depending on the property type, you can edit it by doing one of the following:

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

Working with the Progress Window

The **Progress** window monitors a simulation while it is running.

In the image below, the **Progress** window is displaying the progress of a discrete frequency sweep, which is nearing completion:



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

- On the **View** menu, click **Progress Window**.

  A check box appears next to this command if the **Progress** window is visible.
- Right-click the history tree, and then click **Progress** on the shortcut menu.

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

Working with the Message Manager

The Message Manager displays messages associated with a project's development, such as error messages about the design's setup or informational messages about the progress of an analysis.

To display or hide the Message Manager:

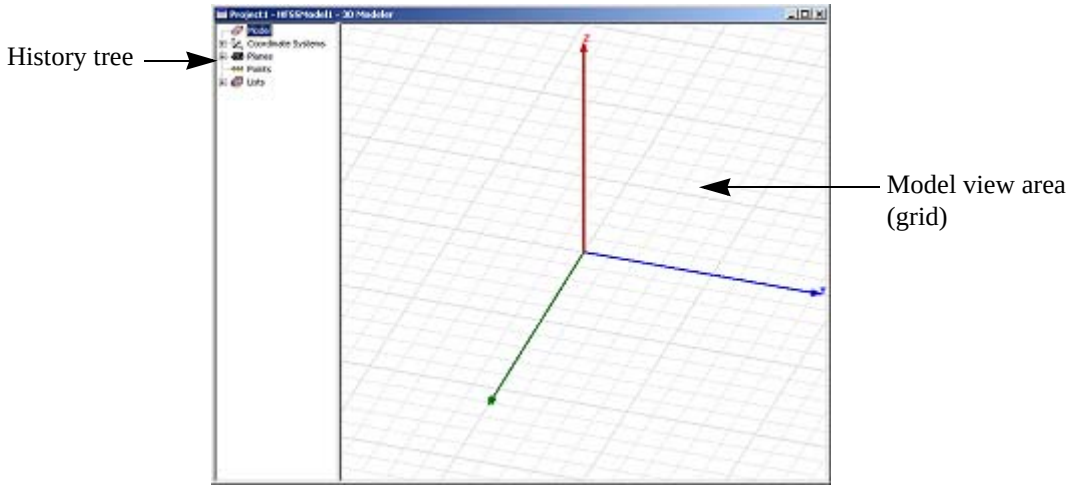
- On the **View** menu, click **Message Manager**.

A check box appears next to this command if the Message Manager is visible.

Working with the 3D Modeler Window

The **3D Modeler** window is the area where you create the model geometry. It appears to the right of the **Project Manager** window *after* you insert an HFSS design to a project.

The **3D Modeler** window consists of the model view area, or grid, and the history tree, as shown below:



To open a new 3D Modeler window, do one of the following:

- Insert a new HFSS design into the current project.
- Double-click an HFSS design in the project tree.

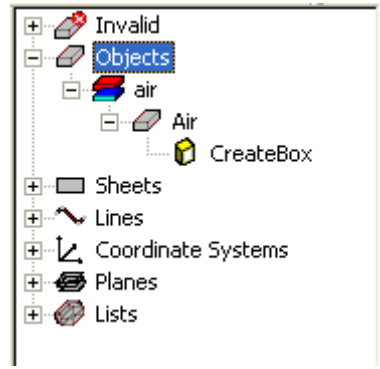
The model you draw is saved with the current project when you click **File>Save**.

Objects are drawn in the **3D Modeler** window. You can create 3D objects by using HFSS's Draw menu commands or you can draw 1D and 2D objects, and then manipulate them to create 3D objects. For more information, see *Drawing a Model*.

You can modify the view of objects in the **3D Modeler** window without changing their actual dimensions or positions. For more information, see *Modifying the Model View.*

Working with the History Tree

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



Right-clicking on any group icon opens a pull-down to expand all groupings or collapse all groupings. In addition, right-clicking on Objects lets you specify whether or not the Objects are sorted by material (the default is to sort by material.) When the objects are sorted by material, 2D and 3D objects are listed separately in the history tree.

The history tree also lists the history of all commands carried out a model's objects. This history is displayed in the order in which it occurred. Notice in the above image the expanded air object and its history of commands.

To view the properties of an item in the history tree:

- Click the item's name in the history tree.

  The item's properties appears in the **Properties** window.

The history tree contains the following model details:

Invalid	Lists all invalid objects
Objects	Displays all the model's objects and a history of the commands carried out on each object. By default HFSS groups objects by material. you can change this by selecting the **Objects** icon in the history tree and right-click to display the shortcut menu with the **Group Objects By Material** checkbox.
Sheets	Displays all the sheets in the model 3D design area. By default, HFSS groups sheet objects by boundary assignment. You can change this by selecting the Sheet icon in the history tree and right-click to display the shortcut menu with the **Group Sheets by Assignment** checkbox.
Lines	Displays all line objects included in the active model. See *Drawing a Line* for information on how to draw a line object.
Points	Displays all point objects included in the active model. See *Drawing a Point* for information on how to draw a point object.
Coordinate Systems	Displays all the coordinate systems for the active model. See *Setting Coordinate Systems* for more information on this model detail.
Planes	Displays the planes for all the coordinate systems. When you create a coordinate system, default planes are created on its xy, yz, and xz planes.
Lists	Displays the object or face lists for the active model. By default, a list called "AllObjects" appears. Creating an object list is a convenient way to identify a group of objects for a field plot or calculation. Creating a face list is a convenient way to identify a specific set of surfaces for a field plot or calculation.

Note While objects created in HFSS can always be classed in the history tree as either a solid, sheet, or wire some imported objects may have mixture of these. HFSS places such objects in an Unclassifed folder in the history tree.

Controlling the View of Objects in the History Tree

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

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

Related Topics

Purge History

Using the Password Manager to Control Access to Resources

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

To Specify a New Password Protected Resource

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

   This displays the **Password Manager** dialog

2. Click the **New** button.

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

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

   This displays the **Enter Passwords** dialog. This dialog has radio buttons to let you:

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

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

   This displays the **Password Manager** with the resource listed.

To Encrypt a Resource

1. Select an existing resource to highlight it and enable the **Encrypt** button.
2. This displays a File browser window
3. Select the appropriate Files of Type filter.

   The choices are Circuit files (*.lib) and Ansoft Library files. For HFSS, chose Ansoft Library files. Any existing resources in the selected directory will appear.

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

   This encrypts the resource.

Running HFSS from a command line

HFSS includes options that can be included when launching from a command line or terminal prompt.

Command-line syntax

hfss ***(command-line options) (file name)***

Command-line options

The following command-line options are available in HFSS. All command-line options are case-insensitive.

-BatchSave <*project file name*>

Saves a named project to the current version. This is primarily intended for converting version 9 projects to version 10 when you intend to subsequently run them on a Linux platform. The conversion from version 9 to version 10 must be done under Windows, HP, or Solaris before those projects can run on a Linux system.

-BatchSolve <*project file name*>

By default, solve all adaptive setups, sweeps, as well as Optimetrics setups found in the project file. Additional parameters for batch solves include the following:

[***designName***] - batch solve all setups for design with the name given under the project.

[***designName***]:**Nominal** - batch solve all nominal setups for design with the name given under the project.

[***designName***]:**Optimetrics** - batch solve all Optimetrics setups for design with the name given under the project.

[***designName***]:**Nominal**:[***setupname***] - batch solve the specified nominal setup for design with the name given under the project.

[***designName***]:**Optimetrics**:[***setupname***] - batch solve the specified Optimetrics setup for design with the name given under the project.

Example:

```
C:\HFSS\hfss.exe -batchsolve HFSSDesign1:Nominal
“C:\Project1.hfss”
```

-Distribute

Batch solve a single setup whenever parallel solve is possible. For now, only parametric setup is implemented for batch solve. This command must be combined with the **Batch-Solve** flag and must be specified before it in the command line.

Example:

```
C:\HFSS\hfss.exe -distribute -batchsolve _
HFSSDesign1:Optimetrics:ParametricSetup1 “C:\Project1.hfss”
```

-Help

Open a window that displays the different command-line options

-Iconic

Run HFSS with the window iconified (minimized)

-LogFile <*log file name*>

Specify a log file(use in conjunction with -BatchSolve or -RunScriptAndExit). If no log file is specified, it will be written to the directory in which the script or HFSS project is located, with the name <project_name>.log.

-ng

Run HFSS in non-graphical mode (use in conjunction with -BatchSave, -BatchSolve or -RunScriptAndExit)

-Queue

Queue licenses (use in conjunction with -BatchSolve or -RunScriptAndExit)

-RunScript <*script file name*>

Run the specified script

-RunScriptAndExit <*script file name*>

Run the specified script and exit

-scriptargs <*scriptArguments*>

Add arguments to the specified script. This command can be used in conjunction with **-RunScript** and **-RunScriptAndExit**.

Example:

```
c:\hfss\hfss.exe -runscriptandexit “c:\project1.vbs”
-scriptargs “Setup1”
```

Related Topics

Running a Script.

Getting Started Guides

Open the PDF: .

This *Getting Started* guide is written for HFSS beginners as well as experienced users who are using HFSS version 9 for the first time. This guide will lead you step-by-step through creating, solving, and analyzing the results of a T-shaped waveguide with an inductive septum. This type of structure is used to split an incoming microwave signal into two outgoing signals. The waveguide's transmission and reflection of the signal will depend on the position of the septum.

By following the steps in this guide, you will learn how to perform the following tasks in HFSS:

- Draw a geometric model.
- Modify a model's design parameters.
- Assign variables to a model's design parameters.
- Specify solution settings for a design.
- Validate a design's setup.
- Run an HFSS simulation.
- Create a 2D x-y plot of S-parameter results.
- Create a field overlay plot of results.
- Create a phase animation of results.

Open the PDF: .

This *Getting Started* guide is written for Optimetrics beginners as well as experienced users who are using Optimetrics version 3 for the first time. You must have completed *Getting Started with HFSS: A Waveguide T-Junction* before you begin this guide.

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

By following the steps in this guide, you will learn how to perform the following tasks in HFSS using Optimetrics:

- Create a basic parametric setup.
- Solve a parametric analysis.
- Create a 2D x-y plot of S-parameter results.
- Create a 2D x-y plot of power distribution results.
- Create a geometry animation.
- Specify a variable to be optimized.
- Create an optimization setup, which includes defining a cost function and setting the range of variable values for an optimization.
- Solve an optimization analysis.
- During an optimization analysis, view a plot of cost values versus solved iterations.

- Run an HFSS simulation using the optimal variable value.
- Update an existing field overlay plot with new results.

Copyright and Trademark Notices

The information contained in the HFSS online help is subject to change without notice.

Ansoft makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Ansoft shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

This document contains proprietary information which is protected by copyright. All rights are reserved.

Ansoft Corporation
225 West Station Square Drive
Suite 200
Pittsburgh, PA 15219
(412) 261 - 3200

Maxwell 3D, Maxwell Strata, HFSS, Full-Wave Spice, ePhysics, and Optimetrics are registered trademarks or trademarks of Ansoft Corporation. All other trademarks are the property of their respective owners.

© 2005 Ansoft Corporation. All rights reserved.

2

Getting Help

Ansoft Technical Support

To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, http://www.ansoft.com, click the **Contact** button, and then click **Support**. Your Ansoft sales engineer may also be contacted in order to obtain this information.

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

Help Menu Commands

To access online help from the menu bar, do the following:

- Click **Help>Contents**
- Click **Help>Index**
- Click **Help>Search**

You can also access help for the scripting commands via the menu bar:

- Click **Help>Scripting Contents**
- Click **Help>Scripting Index**
- Click **Help>Search Scripting**

Context-Sensitive Help

To access online help from the HFSS user interface, do one of the following:

- To open a help topic about an HFSS menu command, press **Shift**+**F1** or click  and then click the command or toolbar icon.
- To open a help topic about an HFSS dialog box, open the dialog box, and then press **F1**.

Conventions Used in the Online Help

The following documentation conventions are used in the HFSS online help.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:

  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.

  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, click **HFSS>Excitations>Assign>Wave Port**.

  - Labeled keys on the computer keyboard. For example, "Press **Return**" means to press the key labeled **Return**.
- Italic type is used for the following:

  - Emphasis.

  - The titles of publications.

  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *file name*" means to type the word **copy**, to type a space, and then to type a file name.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press **Shift+F1**" means to press the **Shift** key and the **F1** key at the same time.
- Toolbar buttons serve as shortcuts for executing commands. Toolbar buttons are displayed after the command they execute. For example,

  "On the **Draw** menu, click **Line**  " means that you can click the Draw Line toolbar button to execute the **Line** command.

Searching in Help

The online help system provides four ways to search for information and navigate quickly:

- A hierarchical table of contents - you can expand or collapse the hierarchy by clicking, and you can jump to selected entries by double-clicking.
- A searchable index - you can search for indexed terms by typing the text field, and jump to topic locations by double-clicking on them.
- A full text search - you can type text, and search the entire online help. Items are listed according to rank in discussing the search text.
- A favorites list - you can select topics that you use frequently to create a favorites list.

Using WebUpdate

To use WebUpdate:

1. Select **Help>Launch WebUpdate**.

   This displays the **WebUpdate** dialogue, which lists the applications available for update.

2. Select the application of interest and click **Next**.

   This displays the application and whether it is currently up to date and whether an update is available

3. If an update is available, enable the application checkbox to select it.

   a. You can choose to enable the checkboxes to install the update automatically and to save the update to disk.

      If you choose to update, the **Next** button is enabled.

   b. Click **Next** to continue the update.

   c. The Webupdate shows the progress of the update.

4. Click **Close** when done

3

Working with HFSS Projects

An HFSS project is a folder that includes one or more HFSS models, or *designs*. Each design ultimately includes a geometric model, its boundary condition and material assignments, and field solution and post-processing information.

A new project called *Projectn* is automatically created when the software is launched. A design named Design*n* is is automatically created for a new project. You can also open a new project by clicking **File>New**. In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

HFSS Files

When you create an HFSS project, it is given an *.hfss* file extension and stored in the directory you specify. Any files related to that project are also stored in that directory.

Some common HFSS file and folder types are listed below:

.hfss	HFSS project.
design_name.**hfssresults**	HFSS folder containing results data for a design.
project_name.**hfssresults**	HFSS folder containing results data for a project.
project_name.**asol**	HFSS field data results for a project. This file's contents may be empty if a solution is not available. This file is stored in the *project_name*.**hfssresults** folder.
.pjt	HFSS version 8.5 and earlier project.
.anfp	Ansoft PCB neutral file

Creating Projects

- On the **File** menu, click **New**  .

  A new project is listed in the project tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current session. A default design named Design*n* is added under the project.

  Project definitions, such boundary and material assignments, are stored under the project name in the project tree.

You specify the name of the project when you save it using the **File>Save** or **File>Save As** commands.

Projects

Open a previously saved project using the **File>Open** command.

1. On the **File** menu, click **Open**  .
2. Use the file browser to find the HFSS version 10 .adsn project file.

   By default, files that can be opened or translated by HFSS are displayed.
3. Select the file you want to open.
4. Click **OK**.

   The project information appears in the project tree.

   If you open another project without editing the automatically-created project, HFSS removes the automatically-created project.

You can also open a saved project by:

- Dragging an HFSS project file icon to the HFSS icon.
- Dragging an HFFF project file icon to the HFSS desktop.
- Double-clicking on an HFSS project file icon.

Related Topics

Opening Legacy HFSS Projects

Opening Recent Projects

To open a project you recently saved in HFSS:

- Click the name of the project file at the bottom of the **File** menu.

If you open another project without editing the automatically-created project, HFSS removes the automatically-created project.

Opening Legacy HFSS Projects

1. On the **File** menu, click **Open**  .
2. Select **Ansoft Legacy EM Projects (*.cls)** from the **Files of type** pull-down list.
3. In the **Look in** pull-down list, click the location of the project. In the folder list, double-click folders to find the one that contains the project.
4. Double-click the project you want to open.

Legacy HFSS Project Translation

When you open a legacy HFSS project - a project created in Ansoft HFSS version 8.5 or earlier - virtually all of the project's pre-processing data is translated. Note that solution results and Optimetrics setup data are unavailable; however, the nominal model created for Optimetrics is translated. Following are additional notes about the translation of various legacy project information.

Model Geometry	• The translated geometry's construction history is unavailable; therefore the original object properties you defined cannot be modified in the **Properties** window. However, you can modify the geometry using version 10's modeling features. • For units unavailable in version 10, such as yards, the nearest available units are used; the model will be scaled slightly to fit the new units.
Excitations and Boundaries	• Port impedance and calibration lines become integration lines in version 10. If the legacy project contained both impedance and calibration lines, impedance lines are translated and calibration lines are ignored. If the project contained both impedance and terminal lines, both are translated. The impedance lines will be ignored for Driven Terminal solutions and terminal lines will be ignored if the project is changed to a Driven Modal solution. • Boundaries assigned to named interface selections or rectangle selections are not translated. • For a boundary assigned to the intersection of two faces, HFSS 10 will create a new 2D sheet object from the intersecting area and assign the boundary to that object.
Materials	• Functions defined in legacy projects become project variables in version 10; therefore, functional material properties are translated. • Perfect conductors become regular materials with conductivity values of of 1E30. • Object coordinate systems are created for objects assigned anisotropic materials in legacy projects. The coordinate system is defined at the same origin as the global coordinate system, with the same orientation defined when the anisotropic material was assigned to the object in the legacy project. • Nonlinear materials from legacy projects that have magnetic saturation values greater than zero are treated as ferrite materials in version 10. Their properties are not modified.
Mesh Operations	• Mesh refinement operations performed on arbitrary boxes in legacy projects are ignored. • Area- and volume-based mesh operations are translated as length-based mesh operations in version 10 by taking their square roots and cube roots, respectively.
Optimetrics	• Setup information, including design variables, is not supported; however, the nominal model can be translated. • Parameterizing a translated model is limited because geometry construction history is unavailable.

Solution Types	• Driven solver projects that contained terminal lines are translated to Driven Terminal solution types in version 10.
Solution Setup	• Impedance-only and emissions-only solutions are not supported in version 10; therefore these selections in legacy projects are ignored. • The design's initial mesh is used for the version 10 solution. Current meshes are not translated. • Saving dominant-only or higher-order-only mode S-matrix entries are not supported in version 10; therefore these mode selections in legacy projects are ignored. • For frequency sweeps, the **Number of Steps** value specified in the legacy project is converted to the corresponding **Step Size** value in version 10. • The total number of requested adaptive passes in the legacy project becomes the **Maximum Number of Passes** value in version 10. For example, if you request 3 adaptive passes, solve them, and then request 2 adaptive passes, 5 will be the value specified for the **Maximum Number of Passes** in version 10.
Solutions	• Solution data is not translated; therefore, you must solve legacy HFSS projects again in version 10.

Closing Projects

To close the current HFSS project, select **HFSS>Close**. This closes the project without exiting HFSS.

Saving Projects

Use the **File>Save As** command to do the following:

- Save a new project.
- Save the active project with a different name or in a different location.
- Save the active project in another file format for use in another program.

Use the **File>Save**  command to save the active project.

HFSS has a "Save before solving" setting located in the **Tools>Options>HFSS Options** menu. By default this is on. However, for efficiency reasons, the project is only saved if it has been modified since its last save.

A prompt appears when you attempt to save a previously-versioned file. If you agree to the prompt, the file is upgraded to the HFSS version in which you are running the software. In this case the file may no longer be compatible with previous versions of HFSS. If you do not agree to the prompt, the file is not saved, so the file retains the previous compatibility.

Note When HFSS is running projects created prior to HFSS v9 in batch mode, an .hfss file will be automatically saved in the default project directory using the same root name as the original .pjt project file. If an error occurs while saving, the message is written to the log file.

Related Topics

Saving a New Project

Saving the Active Project

Saving a Copy of a Project

Deleting Projects

Saving a New Project

1. On the **File** menu, click **Save As**.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Use the correct file extension for the file type.
5. If the window has a **Switch to saved** option, do one of the following:
   - Leave the option selected to display the new file name, and then close the current file.
   - Cancel the **Switch to saved** selection to save the file under the new name without changing which file is displayed.
6. Click **OK**.

HFSS saves the project to the location you specified.

Warning Be sure to save geometric models periodically. Saving frequently helps prevent the loss of your work if a problem occurs.

Although HFSS has an "auto-save" feature, it may not automatically save frequently enough for your needs.

Related Topics

Saving the Active Project

Saving a Copy of a Project

Saving the Active Project

- On the **File** menu, click **Save**  .

  HFSS saves the project over the existing one.

Warning Be sure to save geometric models periodically. Saving frequently helps prevent the loss of your work if a problem occurs.

Although HFSS has an "auto-save" feature, it may not automatically save frequently enough for your needs.

Related Topics

Saving a New Project

Saving a Copy of a Project

Saving a Copy of a Project

To save an existing, active project with a new name, a different file extension, or to a new location:

1. On the **File** menu, click **Save As**.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select the desired file extension for the file type.
5. If the window has a **Switch to saved** field, do one of the following:
   - Leave the field selected to display the new file name, and then close the current file.
   - Cancel the **Switch to saved** selection to save the file under the new name without changing which file is displayed.
6. Click **OK**.

   HFSS saves the project with the new name or file extension to the location you specified.

Related Topics

Saving a New Project

Saving the Active Project

Renaming a Project

To rename an existing, active project:

1. Select the project in the Project tree.
2. Right-click to display the short-cut menu.
3. Select **Rename**.

   This activates the text field for the project name.
4. Type the new project name and press enter.

   The new project name appears in the directory and the project remains in the original location.

Saving Project Data Automatically

HFSS stores recent actions you performed on the active project in an auto-save file in case a sudden workstation crash or other unexpected problem occurs. The auto-save file is stored in the same directory as the project file and is named Project*n*.hfss.auto by default, where n is the order in which the project was added to the current session. HFSS automatically saves all data for the project to the auto-save file, except solution data. By default, HFSS automatically saves project data after every 10 edits. An "edit" is any action you performed which changes data in the project or the design, including actions associated with project management, model creation, and solution analysis.

With auto-save activated, after a problem occurs, you may be able to choose to re-open the original project file (Project*n*.hfss), in an effort to recover the solution data, or open the auto-save file. If the original file is not available, attempting to open the file provides a message that the autosave is being used. If neither file is available, an error message is displayed.

To modify the auto-save settings:

1. On the **Tools** menu, point to **Options**, and then click **General Options**.

   The **Options** dialog box appears.
2. Under the **Project Options** tab, verify that **Do Autosave** is selected.

   This option is selected by default.
3. In the **Autosave interval** box, enter the number of *edits* that you want to occur between automatic saves. By default, this option is set at 10.

Note Auto-save *always* increments forward; therefore, even when you undo a command, HFSS counts it as an edit.

4. Click **OK** to apply the specified auto-save settings.

   Once the specified number of edits is carried out, a "model-only" save will occur. This means that HFSS does not save solutions data or clear any undo/redo history.

   When HFSS auto-saves, an ".auto" extension is appended to the original project file name. For

example, “Project1.hfss” will automatically be saved as “Project1.hfss.auto”.

Warning When you close or rename a project, HFSS deletes the auto-save file. HFSS assumes that you have saved any desired changes at this point.

Related Topics

Recovering Project Data in an Auto-Save File

Save Before Solve Option

The **Tools>HFSS Options** command displays a dialog with a checkbox for an automatic **Save Before Solve** option. If save-before-solve is set, and the user starts a solve while another solve is running, we will ask if the user wants to go ahead and solve without saving first. This lets you do multiple solves, and if you have not edited the project in between solves, crash recovery will work. In any case, we permit you to start a new solve while running another without having to abort the running solve.

Recovering Project Data in an Auto-Save File

Following a sudden workstation crash or other unexpected problem, you can recover the project data in its auto-save file.

Warning When you recover a project’s auto-save file you *cannot* recover any solutions data; recovering an auto-save file means you will lose any solutions data that existed in the original project file.

To recover project data in an auto-save file:

1. If HFSS has unexpectedly crashed, launch HFSS from your desktop.
2. On the **File** menu, click **Open**, and then select the original Project*n*.hfss project file for which you want to recover its Project*n*.hfss.auto auto-save file.

   The **Crash Recovery** window appears, which gives you the option to open the original project file or the auto-save file.
3. Select **Open project using autosave file** to recover project data in the auto-save file, and then click **OK**. HFSS replaces the original project file with the data in the auto-save file.

   HFSS immediately overwrites the original project file data with the auto-save file data, removing the results directory (solutions data) from the original project file as it overwrites to the auto-save file.

Warning If you choose to recover the auto-save file, you cannot recover the original project file that has been overwritten; recovering data in an auto-save file is *not* reversible.

Related Topics

Saving Project Data Automatically

Deleting Projects

To delete a project:

1. Select the project in the project tree.
2. Click either **Edit>Delete**, or right click to display the short-cut menu and select **Delete**.

   A dialog displays the message: “The project selected and all its files will be deleted from the permanent storage medium. Click OK to proceed.”
3. Click **OK** to delete the files or **Cancel** to retain them.

Undoing Commands

Use the **Undo** command on the **Edit** menu to cancel, or undo, the last action you performed on the active project or design. This is useful for undoing unintended commands related to project management, model creation, and post-processing.

1. In the **Project Manager** window, do one of the following:
   - To undo the last action you performed on the *active project*, such as inserting a design or adding project variables, click the project icon.
   - To undo the last action you performed on the *active design*, such as drawing an object or deleting a field overlay plot, click the design icon.

Note You cannot undo an analysis that you've performed on a model, that is, the **HFSS>Analyze** command.

2. On the **Edit** menu, click **Undo**, or click the **Undo** button  on the toolbars.

   Your last action is now undone.

Note When you save a project, HFSS always clears the entire undo/redo history for the project and its designs.

Related Topics

Redoing Commands

Redoing Commands

Use the **Redo** command on the **Edit** menu to reapply, or redo, the last action that was canceled, or undone. You can redo a canceled action related to project management, model creation, and post-processing.

1. In the **Project Manager** window, do one of the following:
   - To redo the last action you canceled on the *active project*, such as inserting a design or adding project variables, click the project icon.
   - To redo the last action you canceled on the *active design*, such as drawing an object or deleting a field overlay plot, click the design icon.
2. On the **Edit** menu, click **Redo**, or click the **Redo** button  on the toolbars.

   Your last canceled action is now reapplied.

Note When you save a project, HFSS always clears the entire undo/redo history for the project and its designs.

Related Topics

Undoing Commands

Validating Projects

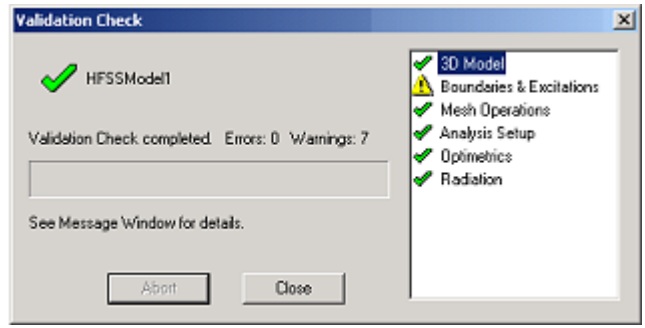
Before you run an analysis on a model, it is very important that you first perform a validation check on the project. When you perform a validation check on a project, HFSS runs a check on all the setup details of the active project to verify that all the necessary steps have been completed and their parameters are reasonable.

To perform a validation check on the active project:

1. On the **HFSS** menu, click **Validation Check**  .

   HFSS checks the project setup, and then the **Validation Check** window appears.

2. View the results of the validation check in the **Validation Check** window.



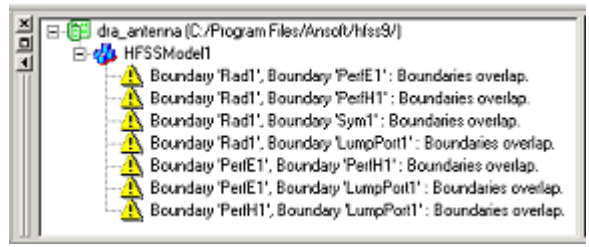
The following icons can appear next to an item:

 Indicates the step is complete.

 Indicates the step is incomplete.

 Indicates the step may require your attention.

3. View any messages in the **Message Manager** window.



4. If the validation check indicates that a step in your project is incomplete or incorrect, carefully review the setup details for that particular step and revise them as necessary.

5. On the **HFSS** menu, click **Validation Check** to run a validation check after you have revised any setup details for an incomplete or incorrect project step.
6. Click **Close**.

Exporting Files

You can export the following types of files from HFSS:

- Ansoft 2D modeler files
- 3D model files
- Graphics files
- Data tables

Related Topics

Exporting Matrix Data

Exporting Equivalent Circuit Data

Exporting 2D Geometry Files

When you export a file in a 2D geometry format (the Ansoft 2D Modeler (**.sm2**) format or the AutoCAD DXF (**.dxf**) format), the geometry located within the xy plane is exported.

Note If you want to export a plane that does not coincide with the global xy plane, you must create a relative coordinate system to redefine the location of the origin.

See *Creating a Relative Coordinate System* for more information.

To export a file to a **.sm2** or **.dxf** format:

1. On the **3D Modeler** menu, click **Export** to save the file in an Ansoft 2D Modeler format.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select **Ansoft 2D Geometry Files (*.sm2)** or **AutoCAD DXF Files (*.dxf)** from the **Save as type** pull-down list.
5. Click **Save**. The file is exported to the specified location with the appropriate file format.

Related Topics

Exporting 3D Model Files

Exporting Graphics Files

Exporting 3D Model Files

You can export HFSS 3D models to the following 3D model file formats:

Extension	Contents
.sat	ACIS geometry solid model files.
.sm3	HFSS 3D Modeler files in ACIS version 2.0 or greater.

Extension	Contents
.step, .stp	Industry standard AP203 STEP files.
.iges, .igs	Industry standard Initial Graphics Exchange Specification (IGES) files.

To export a file to a 3D model format:

1. On the **3D Modeler** menu, click **Export** to save the file in a 3D model format.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select the desired 3D model file format from the **Save as type** pull-down list.
5. If you selected **.sm3**, the **Select Version** dialog box appears. Do the following:
   - Click an ACIS version in which to export the model from the **ACIS SM3 Version** pull-down list, and then click **OK**.
6. Click **Save**. The file is exported to the specified location as a 3D model file.

Related Topics

Exporting 2D Model Files

Exporting Graphics Files

Exporting Graphics Files

You can export the following graphics formats:

Extension	Contents
.bmp	Batch Message Processing files.
.gif	Graphics Interchange Format files.
.jpeg	Joint Photographics Experts Group files.
.tiff	Tagged Image File Format files.
.wrl	Virtual Reality Modeling Language (VRML) files.

To export a file to a graphics format:

1. On the **3D Modeler** menu, click **Export** to save the file in a graphics format.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select the desired graphics file format from the **Save as type** pull-down list.

5. Click **Save**. The file is exported to the specified location as a graphics file.

Related Topics

Exporting 2D Model Files

Exporting 3D Model Files

Exporting Data Table Files

1. On the **Report2D** menu, click **Export to File**.
   - Alternatively, right-click on the data table, and then click **Export to File** on the shortcut menu.

   The **Export plot data to file** dialog box appears.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select one of the following file formats from the **Save as type** pull-down list:

.txt	Post processor format file
.csv	Comma-delimited data file
.tab	Tab-separated file
.dat	Ansoft plot data file

5. Click **Save**. The file is exported to the specified location as a data table file.

Related Topics

Exporting Matrix Data

Exporting Equivalent Circuit Data

Importing Files

You can import the following types of files to HFSS:

- 2D model files
- 3D model files
- Solution data files
- Data table files

The import dialog contains a check box for the **Heal** command which is enabled by default.

Related Topics

Exporting Files

Importing 2D Model Files

You can read the following 2D model files directly into the active **3D Modeler** window:

Extension	Contents
.gds	GDS-II is a standard file format for 2D graphical design layout data.
.sm2	Ansoft 2D Modeler files.

Note If you import a file into an active **3D Modeler** window that contains an existing model, the file will be added to the existing model; it will not replace it.

To import a 2D model file:

1. On the **3D Modeler** menu, click **Import**.

   This opens a file browser. The file browser includes a checkbox for the **Heal** command, which is enabled by default. When enabled, the **Heal** command has radio buttons for Automatic healing (the default) or Manual healing.
2. Use the file browser to find the file you want to import.
3. Select the 2D model file you want import.

Note When importing Ansoft 2D Modeler files (**.sm2**), the geometry located within the xy plane is exported. If you want to export a plane that does not coincide with the global xy plane, you must create a relative coordinate system to redefine the location of the origin.

See *Creating a Relative Coordinate System* for more information.

4. Click **Open**. The file is imported into the active **3D Modeler** window.

   If you have selected **Healing** with the Manual option checked for the import, the Healing options panel opens and lets you set parameters for the heal operation and conduct an analysis.

See Healing.

Related Topics

Importing 3D Model Files

Importing 3D Model Files

You can read the following 3D model files directly into the active **3D Modeler** window:

Extension	Contents
.dxf	AutoCAD Drawing Interchange Format files.
.geo	Agilent HFSS solid model files.
.iges, .igs	Industry standard Initial Graphics Exchange Specification (IGES) files.
.sat	ACIS geometry solid model files.
.sm3	HFSS 3D modeler files.
.step, .stp	Industry standard AP203 STEP files.
.prt*, .asm*	Pro/E model files.

Note If you import a file into an active **3D Modeler** window that contains an existing model, the file will be added to the existing model; it will not replace it.

To import a 3D model file:

1. On the **3D Modeler** menu, click **Import**.

   This opens a file browser. The file browser includes a checkbox for the **Heal** command, which is enabled by default. When enabled, the **Heal** command has options for Automatic healing (the default) or Manual healing.
2. Use the file browser to find the file you want to import.
3. Select the 3D model file you want to import.
4. Click **Open**. The file is imported into the active **3D Modeler** window.

   If you have selected Healing with the Manual option checked for the import, the Healing options panel opens and lets you set parameters for the heal operation. See Healing.

Note While objects created in HFSS can always be classed in the history tree as either a solid, sheet, or wire some imported objects may have mixture of these. HFSS places such objects in an Unclassifed folder in the history tree.

Related Topics

Importing 2D Model Files

Importing Solution Data

1. On the **HFSS** menu, point to **Results**, and then click **Import Solutions**.

   The **Imported Data** dialog box appears.
2. Click **Import Solution**.

   The **S Parameter Import** dialog box appears.
3. In the **File Name** text box, type the name of the solution file you want to import or click **Browse** and use the file browser to locate the file.
4. Click **Load File**. Note that the file has not been imported yet.
5. Optionally, type a new name in the **Source Name** box or accept the default name.
6. Click the solutions you want to import in the **Available Solutions** list, and then click **Import**.

   You return to the **Imported Data** dialog box.
7. Click the solution data you want to import, and then click **OK**.

Importing Data Tables

You can import data table files that contain data in the following formats:

- Tab-separated. HFSS will recognize complex data if the values are separated by a comma (e.g. *real, imaginary*).
- Comma-separated. HFSS will recognize complex data if the values are separated by a space (e.g. *real imaginary*).

1. On the **HFSS** menu, point to **Results**, and then click **Import Solutions**.
   - Alternatively, right-click **Results** in the project tree and then click **Import Solutions** on the shortcut menu.

   The **Imported Data** dialog box appears.
2. Click **Import Table**.

   The **Table Import** dialog box appears.
3. In the **File Name** text box, type the name of the data table file you want to import or click **Browse** and use the file browser to locate the file.
4. If the data in the table is complex, select the format — real/ imaginary, or magnitude/ phase — in which to import the data.

If the data is simple, this option will be ignored.

5. Click **Load File**. Note that the file has not been imported yet.
6. Optionally, type a new name in the **Source Name** box that indicates the origin or the data table, or accept the default name.
7. Optionally, type a new name in the **Table Name** box that describes the data in the table, or accept the default name
8. In the **All Columns** list, the headings of each column in the data file are listed. Optionally, specify a new name for a column heading by doing the following:
   a. In the **All Columns** list, click the heading you want to change. The heading appears in the **Column Name** box.
   b. Type a new name in the **Column Name** box, and then click **Set Column Name**.

      The heading is changed to the new name in every place it appears in the **Imported Data** dialog box.
9. In the **Independent Data Columns** list, the first heading in the data table file is listed by default. In the **Dependent Data Columns** list, the second and subsequent headings in the data table file are listed by default. Optionally, click a heading name and then click an arrow button to move it from one column to another.
10. If the data in the **Dependent Data Columns** list contains matrix data, select **Matrix Data**. If it contains field data, select **Field Data**.
11. Click **Import**.

    You return to the **Imported Data** dialog box.
12. Click the data you want to import in the **Current Imports** list, and then click **OK**.

    The solution data is now available for post processing.

Inserting a Documentation File

You may want to add a documentation file to the project tree.

1. Click **Project>Insert Documentation** File.

   This opens a file browser dialog that lets you navigate your file system.

2. Selecting the file and click OK.

   This places the documentation file in the project tree.

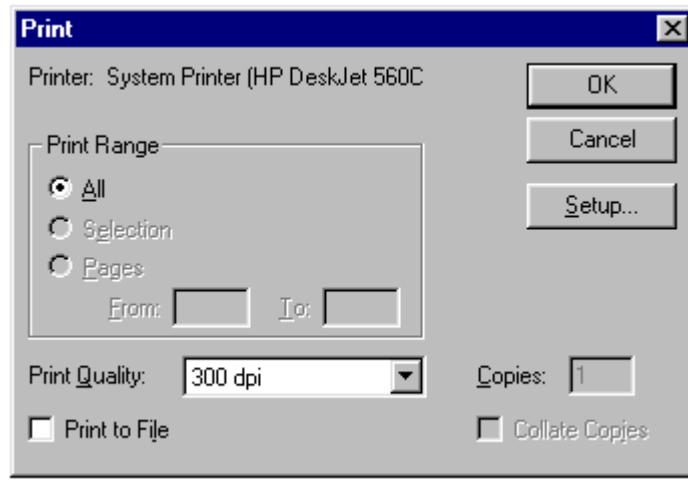
Printing

The printing commands enable you to print the display in the active window.

To print the project:

1. On the **File** menu, click **Print**  .

   A dialog box similar to the following one appears:



2. You can change the print quality (a higher dpi makes a higher quality print, but takes more time and printer memory), or you can send the output to a .prn file.
3. Do one of the following:
   - Click **OK** to print the project.
   - Click **Cancel** to dismiss the window without printing.
   - Click **Setup** to define printer settings.

Saving Project Notes

You can save notes about a project, such as its creation date and a description of the device being modeled. This is useful for keeping a running log on the project.

To add notes to a project:

1. On the **HFSS** menu, click **Edit Notes**  .

   The **Design Notes** window appears.
2. Click in the window and type your notes.
3. Click **OK** to save the notes with the current project.

To edit existing project notes:

- Double-click the **Notes** icon in the project tree.

  The **Design Notes** window appears, in which you can edit the project's notes.

Setting Options in HFSS

You can set the following options from the HFSS Desktop:

- General options, such as project options, units settings, and analysis options.
- HFSS-specific options, such as default solution mode, processor and RAM settings.
- Fields Reporter options, such as field overlay and phase animation settings.
- Report2D options, such as fonts, labels, line styles, and colors.
- 3D Modeler options, such as cloning options, display colors and render settings, snap modes and mouse sensitivity.

Setting General Options

To set general options in HFSS:

1. Click **Tools>Options>General Options**.
   The **General Options** window appears, displaying five available tabs:
   - Project Options
   - Default Units
   - Analysis Options
   - WebUpdate Options
   - Distributed Analysis Options
2. Click each tab, and make the desired selections.
3. Click **OK**.

General Options: Project Options Tab

These options are set on the **Project Options** tab of the **General Options** dialog box.

1. To auto-save your project, do the following in the Autosave section:
   a. Select the **Do Autosave** check box.
   b. Enter the number of edits after which to save in the **Autosave interval** text box. The default is **10**.
2. Enter a directory path in the **Temp Directory** text box, or click the **...** button to find and select the desired directory.
3. Enter a directory path in the **Project Directory** text box, or click the **...** button to find and select the desired directory.
4. Also in the **Temp Directory Settings** section, select or clear the **Set as default temp directory for remote analysis launched as 'This User'** check box.
5. Enter a directory path in the **Library Directory** text box, or click the **...** button to find and select the desired directory.
6. To reset the library directory to the default, click **Reset Library Directory**.
7. Select or clear the **Expand Project Tree on Insert** check box.

8. Select or clear the **Show Message Window on new messages** check box.
9. Select or clear the **Expand Message Window Tree on errors and warnings** check box.
10. Select or clear the **Show Progress Window when starting a simulation** check box.
11. Select or clear the **Update reports on file open** check box.

General Options: Default Units Tab

These options are set on the **Default Units** tab of the **General Options** dialog box.

Select the desired units from each of the following pull-down lists:

- Length
- Angle
- Time
- Temperature
- Torque
- Magnetic Induction
- Frequency
- Power
- Voltage
- Current
- Speed
- Weight
- Resistance
- Inductance
- Capacitance
- Force
- Angular Speed

General Options: Analysis Options Tab

These options are set on the **Analysis Options** tab of the **General Options** dialog box.

1. The type of design is **HFSS**.
2. Select or clear the **Show the Remote Analysis Dialog** check box.
3. Select either **Local Machine** or **Remote Machine** as the **Default Analysis Location**.
4. If you select **Remote Machine**, then specify the location in the **Default Remote Machine Details** section. See Solving Remotely.
5. To start all simulations as a specific user, do the following in the **Launch User Options** section:
   - Select the **Launch simulations as 'this' user** check box.
   - Enter a **User Name**, **Password**, and **Domain/Workgroup** in the corresponding text

boxes.

6. Select or clear the **Queue all simulations** check box.

General Options: WebUpdate Options Tab

These options are set on the **WebUpdate Options** tab of the **General Options** dialog box.

Select one of the following from the **Automatically check for updates every** pull-down list:

- Never
- 30 days
- 120 days
- 180 days

The last time the software was updated, as well as the last attempt, are displayed in the following two fields:

- **Last update date**
- **Last update attempt date**

General Options: Distributed Analysis Options Tab

These options are set on the **Distributed Analysis Options** tab of the **General Options** dialog box.

To add a machine to the **Machines for Distributed Analysis** list:

1. Select one of the following, and enter the information in the text box:
   - IP Address (format: 192.168.1.2)
   - DNS Name (format: www.server.com)
   - UNC Name (format: \\server)
2. Click **Add machine to list**.

To remove a machine from the **Machines for Distributed Analysis** list:

1. Select the machine(s) you want to remove from the list.
2. Click **Remove selected machines**.

Setting HFSS Options

To set HFSS options:

1. Click **Tools>Options>General Options**.
   The **HFSS Options** window appears, displaying two available tabs:
   - General Options
   - Solver
2. Click each tab, and make the desired selections.
3. Click **OK**.

HFSS Options: General Options Tab

These options are set on the **General Options** tab of the **HFSS Options** dialog box.

1. To change the default solution type when you initially insert a project, select one of the follow-

ing from the Default solution type pull-down list:

- Eigenmode
- Driven Modal
- Terminal

2. In the **Material Options** section:
   - Check or uncheck whether to **Include ferrite materials**
   - Set the **Solve Inside threshold** values in Siemens/m.
3. In the **Boundary Options section**, select or clear the following two check boxes:
   - Use Wizards for data input when creating new boundaries
   - Duplicate boundaries with geometry
4. Select or clear the following check boxes:
   - Dynamically update post process data
   - Save before solving
   - Save Optimetrics field solutions
   - Apply variation deletions immediately
5. Set the default Matrix sort order. This affects the order of the Matrix Data, and is of interest depending on how port names are assigned for that design. The default is ascending alphanumeric. This can also be a User Specified order that defaults to creation order.

HFSS Options: Solver Tab

These options are set on the **Solver** tab of the **HFSS Options** dialog box.

To set the solver options for HFSS:

1. Enter the **Number of Processors** to use.
2. Select one of the following from the Default Process Priority pull-down list:
   - Critical (highest) Priority
   - Above Normal Priority
   - Normal Priority
   - Below Normal Priority
   - Idle (lowest) Priority
3. Select one or both of the following check boxes, and enter values in the text boxes:
   - Desired RAM Limit (MB)
   - Maximum RAM Limit (MB)

Setting Fields Reporter Options

The **Fields Reporter** options control two basic areas:

- Whether to **Group Field Overlays by Type** (default, yes).
- Default Phase Animation settings for **Scalar Plots** and **Vector Plots.**

Each of these accepts values for From and To in degrees, and the number of steps.

Setting Report2D Options

To set Report2D options in HFSS:

1. Click **Tools>Options>Report2D Options**.
   The **Report2D Options** window appears, displaying five available tabs:
   - Table
   - General Options
   - Line Style
   - Color
   - Graphics Font
2. Click each tab, and make the desired selections.
3. Click **OK**.

Report2D Options: Table Tab

These options are set on the **Table** tab of the **Report2D Options** dialog box.

1. To specify the font information that appears in the **Font**, **Style**, and **Size** boxes:
   a. Click the **Select** button in the **Font** section.
      The **Font** dialog box appears.
   b. Select a font type from the **Font** list.
   c. Select a font style from the **Font style** list.
   d. Select a font size from the **Size** list.
   e. Select a font script from the **Script** list, as applicable.
   f. Click **OK** to close the **Font** dialog box and return to the **Table** tab of the **Report2D Options** dialog box.
2. In the Format section, enter values for the **Field Width** and **Precision**, and select or clear the **Use Scientific Notation** check box.
3. Select or clear the following check boxes:
   - Copy to clipboard with headers
   - Copy to clipboard with tab separators

Report2D Options: General Options Tab

These options are set on the **General Options** tab of the **Report2D Options** dialog box.

1. Enter the **Accumulate Depth**.
2. Select or clear the following check boxes:
   - Floating Data Cursor
   - Show Polar Grid Labels
   - Show Date/Time/Path

3. Specify the **Clipboard options**:
   - To specify the **Capture aspect size ratio**, select **As Shown** or **Full Screen**.
   - To specify the **Background color**, select **As Shown** or **White**.

Report2D Options: Line Style Tab

These options are set on the **Line Style** tab of the **Report2D Options** dialog box.

1. Select a trace or grid from the **Item** pull-down list.
2. In the **Style** section, select the type of line you want to use, and enter the point size.
3. To show symbols on the traces:
   - Select the **Show Symbols On All Traces** check box.
   - Enter the interval at which to draw each symbol.
4. To revert to the default settings, click **Use Defaults**.

Report2D Options: Color Tab

These options are set on the **Color** tab of the **Report2D Options** dialog box.

To change the color for a specific item:

1. Select the item from the **Item to select** list.
2. Click **Edit Color**.
   The **Color** window appears.
3. Select a color, and click **OK**.
4. To revert to the default settings, click **Use Defaults**.

Report2D Options: Graphics Font Tab

These options are set on the **Graphics Font** tab of the **Report2D Options** dialog box.

To change the font specifications for a specific item:

1. Select the item from the **Item** column of the table.
2. Click **Edit Font**.
   The **Font** window appears.
3. Select a **Font**, **Font style**, and **Size** from the corresponding lists.
4. Click **OK**.
5. To revert to the default settings, click **Use Defaults**.

Setting 3D Modeler Options

To set 3D modeler options in HFSS:

1. Click **Tools>Options>3D Modeler Options**.
   The **3D Modeler Options** window appears, displaying three available tabs:
   - Operation
   - Display
   - Drawing

2. Click each tab, and make the desired selections.
3. Click **OK**.

3D Modeler Options: Operation Tab

These options are set on the **Operation** tab of the **3D Modeler Options** dialog box.

1. To specify when to clone tool objects, select or clear the following check boxes in the **Clone** section:
   - Cone tool objects before uniting
   - Cone tool objects before subtracting
   - Cone tool options before intersecting
2. In the **Coordinate System** section, select or clear the **Automatically switch to face coordinate system** check box.
3. In the **Polyline** section, select or clear the **Automatically cover closed polylines** check box.

3D Modeler Options: Display Tab

These options are set on the **Display** tab of the **3D Modeler Options** dialog box.

1. To specify a default color for a 3D Modeler drawing object or action (such as on select):
   - Select the object or action from the **Default color** pull-down list.
     The **Color** window appears.
   - Select a color, and click **OK**.
2. To specify the default for the **View>Render** setting for new projects, select **WireFrame** or **SmoothShade** from the **Default view render** pull-down list.
3. To set the **Default transparency**, move the slider, or enter a numerical value.
4. Select or clear the **Show orientation of selected objects** check box.
5. Select or clear the **Highlight selection dynamically** check box.
6. Under **Default tree layout**, select or clear the **Group objects by material** check box.
7. Under **History operations visualization**, select or clear the **Visualize history of objects** check box.

3D Modeler Options: Drawing Tab

These options are set on the **Drawing** tab of the **3D Modeler Options** dialog box.

1. To specify snap settings, select or clear the following check boxes in the **Snap Mode** section:
   - Grid
   - Vertex
   - Edge Center
   - Face Center
   - Quadrant
   - Arc Center

2. Enter how near the mouse needs to be to click a grid item in the **Mouse Sensitivity** box, in pixels.
3. Select or clear the **Edit property of new primitives** check box.

   The specifies whether a Properties dislog appears on the creation of a new primitive.
4. The **Operation Data Mode** controls whether you draw new objects directly via the mouse, or whether a **Properties** dialog opens for you to enter dimensions for the object. The **Dialog** mode drawing feature works with the equation based line, and all two and three dimensional objects.
   - Point mode - mouse drawing.
   - Dialog - enter dimensions in the properties dialog.

   You can also use **F3** for Point mode and **F4** for dialog mode.

Working with Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in HFSS. Variables are useful in the following situations:

- You expect to change a parameter often.
- You expect to use the same parameter value often.
- You intend to run a parametric analysis, in which you specify a series of variable values within a range to solve.
- You intend to optimize a parameter value by running an optimization analysis.
- You intend to run a convergence on an output variable.

There are two types of variables in HFSS:

Project Variables A project variable can be assigned to any parameter value in the HFSS project in which it was created. HFSS differentiates project variables from other types of variables by prefixing the variable name with the following symbol: **$**. You can manually include the symbol **$** in the project variable's name, or HFSS will automatically append the project variable's name after you define the variable.

Design Variables A design variable can be assigned to any parameter value in the HFSS design in which it was created.

Adding a Project Variable

A project variable can be assigned to a parameter value in the HFSS project in which it was created. HFSS differentiates project variables from other types of variables by prefixing the variable name with the following symbol: **$**. You can manually include the symbol **$** in the project variable's name when you create it, or HFSS will automatically append the project variable's name with the symbol after you define the variable.

1. On the **Project** menu, click **Project Variables**.
   - Alternatively, right-click the project name in the project tree, and then click **Project Variables** on the shortcut menu.

   The **Properties** dialog box appears.
2. Under the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.
3. In the **Name** text box, type the name of the variable.

   Project variable names must start with the symbol **$** followed by a letter. Variable names may include alphanumeric characters and underscores (_). The names of intrinsic functions and the pre-defined constant pi (π) cannot be used as variable names.
4. In the **Value** text box, type the quantity that the variable represents. Optionally, include the

units of measurement.

Warning If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

The quantity can be a numerical value, a mathematical expression, or a mathematical function. The quantity entered will be the *current*, (or *default) value* for the variable.

5. Click **OK**.

   You return to the **Properties** dialog box. The new variable and its value are listed in the table. If the value is an expression, the evaluated value is shown. Updating the expression also changes the evaluated value display.

6. Optionally, type a description of the variable in the **Description** text box.
7. Optionally, select **Read Only**. The variable's name, value, unit, and description cannot be modified when **Read Only** is selected.
8. Optionally, select **Hidden**. If you clear the **Show Hidden** option, the hidden variable will not appear in the **Properties** dialog box.

The new variable can now be assigned to a parameter value in the project in which it was created.

Realated Topics

Deleting Project Variables

Deleting Project Variables

To delete a project variable:

1. Remove all references to the variable in the project.
2. Save the project to erase the command history.
3. Click **Project>Project Variables** to display the **Properties** dialog with list of variables.
4. Select the variable and click **Remove** and **OK**.

Adding a Design Variable

A design variable is associated with an HFSS design. A design variable can be assigned to a parameter value in the HFSS design in which it was created.

1. On the **HFSS** menu, click **Design Properties**.
   - Alternatively, right-click the design name in the project tree, and then click **Design Properties** on the shortcut menu.

   The **Properties** dialog box appears.

2. Under the **Local Variables** tab, click **Add**.

   The **Add Property** dialog box appears.

3. In the **Name** text box, type the name of the variable.

   Variable names must start with a letter, and may include alphanumeric characters and underscores (_). The names of intrinsic functions and the pre-defined constant pi (π) cannot be

used as variable names.

4. In the **Value** text box, type the quantity that the variable represents. Optionally, include the units of measurement.

Note If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

The quantity can be a numerical value, a mathematical expression, or a mathematical function. The quantity entered will be the *current* (or *default value)* for the variable.

Note Complex numbers are not allowed for variables to be used in an Optimetrics sweep, or for optimization, statistical, sensitivity or tuning setups.

5. Click **OK**.

   You return to the **Properties** dialog box. The new variable and its value are listed in the table. If the value is an expression, the evaluated value is shown. Updating the expression also changes the evaluated value display.

6. Optionally, type a description of the variable in the **Description** text box.

The new variable can now be assigned to a parameter value in the design in which it was created.

Realated Topics

Deleting Design Variables

Deleting Design Variables

To delete a design variable:

1. Remove all references to the variable in the design.
2. Save the project to erase the command history.
3. Click **HFSS>Design Properties** to display the **Properties** dialog with list of local variables.
4. Select the variable and click **Remove** and **OK**.

Defining an Expression

Expressions are mathematical descriptions that typically contain intrinsic functions, such as sin(x), and arithmetic operators, such as +, -, *, and /, but do not reference defined variables.

The symbol, pi (π), is the only available pre-defined constant. It may not be reassigned a new value.

Numerical values may be entered in Ansoft's shorthand for scientific notation. For example, $5x10^7$ could be entered as **5e7**.

Using Valid Operators for Expressions

The operators that can be used to define an expression or function have a sequence in which they will be performed. The following list shows both the valid operators and the sequence in which they are accepted (listed in decreasing precedence):

( )	parenthesis	1
-	unary minus	2
^ and ****	exponentiation	3
*****	multiplication	4
/	division	5
%	modulus	6
+	addition	7
-	subtraction	8
<<	left shift	9
>>	right shift	10
==	equals	11
!=	not equal to	12
>	greater than	13
<	less than	14
>=	greater than or equal to	15
<=	less than or equal to	16
&	bitwise and	17
|	bitwise or	18
^	bitwise xor	19
~	1’s complement	20

Using Intrinsic Functions in Expressions

HFSS recognizes a set of intrinsic trigonometric and mathematical functions that can be used to define expressions. Intrinsic function names are reserved, and may not be used as variable names.

The following intrinsic functions may be used to define expressions:

Function	Description	Syntax
abs	Absolute value	abs(x)
acos	Arccosine	acos(x)
asin	Arcsine	asin(x)

atan	Arctangent (in range of -90 to 90 degrees)	atan(x)
atan2	Arctangent (in range of -180 to 180 degrees)	atan2(y,x)
ceil	Ceiling (round up)	ceil(x)
cos	Cosine	cos(x)
cosh	Hyperbolic Cosine	cosh(x)
deg	Degrees (convert from radians)	deg(x)
dset	Dataset Definition	dset((x_0, y_0), (x_1, y_1), ... (x_n, y_n))
exp	Exponential	exp (x)=e^x
floor	Floor (round down)	floor(x)
if	If	if(cond_exp,true_exp, false_exp)
ln	Natural Log	ln(x)
log	Log to base 10	log(x)
pow	Raise to power (x^y)	pow(x,y)=x^y
pwlx	Piecewise Linear x	pwlx(variable, form_exp, dataset_exp)
pwly	Piecewise Linear y	pwly(variable, form_exp, dataset_exp)
rad	Radians (convert from degrees)	rad(x)
rand	Random Number (between zero and 1)	rand()
round	Round (round to nearest)	round(x)
sign	Sign	sign(x)
sin	Sine	sin(x)
sinh	Hyperbolic Sine	sinh(x)
sqrt	Square Root	sqrt(x)
tan	Tangent	tan(x)
tanh	Hyperbolic Tangent	tanh(x)

Note If you you do not specify units, all trigonometric functions interpret their arguments as radians. Likewise, inverse trigonometric functions' return values are in given in radians. When the argument to a trignometric expression is a variable, the units are assumed to be radians. If you want values interpreted in degrees, supply the argument with the unit name **deg**.

Using Piecewise Linear Functions in Expressions

The following piecewise linear intrinsic functions are accepted in expressions:

```
pwlx (dataset_expression, variable)
pwly (dataset_expression, variable)
```

The **pwlx** function interpolates along the x-axis and returns a corresponding y value. The **pwly** function interpolates along the y-axis and returns a corresponding x value.

Using Dataset Expressions

Dataset expressions take the form:

$$\mathtt{dset}((x_0,y_0), \ldots, (x_n,y_n))$$

These expressions may be used as the first parameter to piecewise linear (**pwlx** and **pwly**) functions, and may also be assigned to variables, in which case the variable may be used as the second parameter to **pwlx** and **pwly** functions.

Dataset expressions are derived from a series of points in a plot created in the **Datasets** dialog box. Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot and an expression is derived from the curve that best fits the segmented plot. The created expression is then used in the piecewise linear intrinsic functions.

Adding Datasets

Datasets are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot and an expression is derived from the curve that best fits the segmented plot. The created expression is then used in piecewise linear intrinsic functions.

1. On the **Project** menu, click **Datasets**.

   The **Datasets** dialog box appears.

2. Click **Add**.

   The **Add Dataset** dialog box appears.

3. Optionally, type a name other than the default for the dataset in the **Name** text box.
4. Type the x- and y-coordinates for the first data point in the row labeled **1**.
5. Type the x- and y-coordinates for the remaining data points in the dataset using the same method.

   After you type a point's coordinates and move to the next row, the point is added to the plot,

adjusting the view with each newly entered point.

6. When you are finished entering the data point coordinates, click **OK**.

The dataset plot is extrapolated into an expression that can be used in parametric analyses or assigned to a material property value.

Modifying Datasets

1. On the **Project** menu, click **Datasets**.

   The **Datasets** dialog box appears.

2. Click the dataset name you want to modify, and then click **Edit**.

   The **Edit Dataset** dialog box appears.

3. Optionally, type a name other than the default for the dataset in the **Name** text box.
4. Type new values for the data points as desired.

   The plot is adjusted to reflect the revised data points.

5. When you are finished entering the data point coordinates, click **OK**.

Defining Mathematical Functions

A mathematical function is an expression that references another defined variable. A function's definition can include both expressions and variables.

The following mathematical functions may be used to define expressions:

Basic Functions	/, +, -, *, **%** (modulus), ****** (exponentiation), **-** (Unary minus), << (left shift), >> right shift, == (equals), **!=** (not equals), > (greater than), < (less than), >= (greater than equals), <= (less than equals), **&** (bitwise and), **|** (bitwise or), ^ (bitwise xor), ~ (1's compliment), **&&** (logical and), **II** (logical or), **!** (factorial)
Intrinsic functions	**if**, **sign** (returns the sign of an argument), **abs**, **exp**, **pow**, **ln** (natural log), **log** (log to the base 10), **lg** (log to the base 2), **sqrt**, **floor**, **ceil**, **round**, **rand** (returns a random number between 0 and 1), **deg**, **rad**
Trigonometric expressions	**sin**, **cos**, **tan**, **asin**, **acos**, **atan**, **sinh**, **cosh**, **tanh**

The predefined variables X, Y, Z, Phi, Theta, and R must be entered as such. X, Y, and Z are the rectangular coordinates. Phi, Theta, and R are the spherical coordinates.

If you do not specify units, all trigonometric expressions expect their arguments to be in radians, and the inverse trigonometric functions' return values are in radians. If you want to use degrees, you must supply the unit name **deg**. When the argument to a trignometric expression is a variable, the units are assumed to be radians. These function names are reserved and may not be used as variable names.

As far as expression evaluation is concerned: units are conversion factors (that is, from the given unit to SI). Note also that the evaluated value of an expression) is always interpreted as in SI units.

Assigning Variables

To assign a variable to a parameter in HFSS:

- Type the variable name or mathematical expression in place of a parameter value in a **Value** text box.

  If you typed a variable name that has not been defined, the **Add Variable to *DesignName*** dialog box will appear, enabling you to define the design variable.

  If you typed a variable name that included the **$** prefix, but that has not been defined, the **Add Variable to Project** dialog box will appear, enabling you to define the project variable.

Note You can assign a variable to nearly any design parameter assigned a numeric value in HFSS. See the HFSS online help about the specific parameter you want to vary to determine if can be assigned a variable.

Choosing a Variable to Optimize

Before a variable can be optimized, you must specify that you intend for it to be used during an optimization analysis in the **Properties** dialog box.

1. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.
2. Click the tab that lists the variable you want to optimize.
3. Click the row containing the variable you want to optimize.

Note Dependent variables cannot be optimized.

4. Select the **Optimization** option above.
5. For the variable you want to optimize, select **Include**.

   The selected variable will now be available for optimization in an Optimetrics setup defined in the current design or project.

Note Complex numbers are not allowed for variables to be used in an Optimetrics sweep, or for optimization, statistical, sensitivity or tuning setups.

6. Optionally, override the default minimum and maximum values that Optimetrics will use for the variable in every optimization analysis. During optimization, the optimizer will not consider variable values that lie outside of this range.

Related Topics

Setting up an Optimization Analysis

Including a Variable in a Sensitivity Analysis

Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** dialog box.

1. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.
2. Click the tab that lists the variable you want to include in the sensitivity analysis.
3. Click the row containing the variable you want to include in the sensitivity analysis.

Note Dependent variables cannot be included in a sensitivity analysis.

4. Select the **Sensitivity** option above.
5. For the variable you want to include in the sensitivity analysis, select **Include**.

   The selected variable will now be available for sensitivity analysis in a sensitivity setup defined in the current design or project.

Note Complex numbers are not allowed for variables to be used in an Optimetrics sweep, or for optimization, statistical, sensitivity or tuning setups.

6. Optionally, override the default minimum and maximum values that Optimetrics will use for the variable in every sensitivity analysis. During sensitivity analysis, Optimetrics will not consider variable values that lie outside of this range.
7. Optionally, override the default initial displacement value that Optimetrics will use for the variable in every sensitivity analysis. During sensitivity analysis, Optimetrics will not consider a variable value for the first design variation that is greater than this step size away from the starting variable value.

Related Topics

Setting up a Sensitivity Analysis

Choosing a Variable to Tune

Before a variable can be tuned, you must specify that you intend for it to be tuned in the **Properties** dialog box.

1. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.
2. Click the tab that lists the variable you want to tune.

3. Click the row containing the variable you want to tune.

Note Dependent variables cannot be tuned.

4. Select the **Tuning** option above.
5. For the variable you want to tune, select **Include**.

Note Complex numbers are not allowed for variables to be used in an Optimetrics sweep, or for optimization, statistical, sensitivity or tuning setups.

The selected variable will now be available for tuning in the **Tune** dialog box.

Related Topics

Tuning a Variable

Including a Variable in a Statistical Analysis

Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** dialog box.

1. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.
2. Click the tab that lists the variable you want to include in the statistical analysis.
3. Click the row containing the variable you want to include in the statistical analysis.

Note Dependent variables cannot be included in a statistical analysis.

4. Select the **Statistical** option above.
5. For the variable you want to include in the statistical analysis, select **Include**.

   The selected variable will now be available for statistical analysis in a statistical setup defined in the current design or project.

Note Complex numbers are not allowed for variables to be used in an Optimetrics sweep, or for optimization, statistical, sensitivity or tuning setups.

6. Optionally, override the distribution criteria that Optimetrics will use for the variable in every statistical analysis.

Related Topics

Setting up a Statistical Analysis

4

Setting up an HFSS Design

To set up an HFSS design, follow this general procedure. Note that after you insert a design, you do not need to perform the steps sequentially, but they must be completed before a solution can be generated.

1. Insert an HFSS design into a project.
2. Select the solution type.
3. Set the model's units of measurement.
4. Draw the model geometry and assign material characteristics to objects.
5. Assign boundaries, which specify the field behavior at the edges of the problem region and object interfaces.
6. For Driven solution-type designs, assign excitations - sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces.
7. Specify how HFSS will compute the solution.

Inserting an HFSS Design

The first step in setting up an HFSS design is to add a design to the active project.

To insert an HFSS design:

- On the **Project** menu, click **Insert HFSS Design**  .

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

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

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

Related Topics

Setting the Project Tree to Expand Automatically

Selecting the Solution Type

Before you draw the model, specify the design's solution type. As you set up your design, options available in the user interface will depend upon the selected solution type.

1. On the **HFSS** menu, click **Solution Type**.

   The **Solution Type** dialog box appears.

2. Select one of the following solution types:

Driven Modal	For calculating the mode-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines, which are "driven" by a source.
Driven Terminal	For calculating the terminal-based S-parameters of passive, high-frequency structures with multi-conductor transmission line ports, which are "driven" by a source. Results in a terminal-based description in terms of voltages and currents.
Eigenmode	For calculating the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

Related Topics

Technical Notes: Solution Types

Setting the Model's Units of Measurement

You can specify the units of measurement for drawing geometric models. After the units of measurement have been specified, they are assigned to the objects in the **3D Modeler** window. You can then choose to display the model's dimensions in the new units, or rescale the model's dimensions to the new units.

To set the model's units of measurement:

1. On the **3D Modeler** menu, click **Units**.

   The **Set Model Units** dialog box appears.
2. Select the new units for the model from the **Select units** pull-down list.
3. Specify how the change in units affects the model:
   - Select the **Rescale to new units** option to rescale the dimensions to the new units. For example, selecting centimeters (cm) as the new unit of measurement results in a dimension of 10 millimeters (mm) becoming 10 cm.
   - Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale. For example, selecting cm as the new unit of measurement results in a dimension of 10 mm becoming 1 cm.
4. Click **OK** to apply the new units to the model.

5

Drawing a Model

After you insert an HFSS design into the current project, you can draw a model of the electromagnetic structure. The general strategy is to build the model as a collection of 3D objects. Each material type is treated as a separate object.

You can create 3D objects by using HFSS's **Draw** commands or you can draw 1D and 2D objects, and then manipulate them to create 3D objects. Objects are drawn in the **3D Modeler** window. You can also import objects from other systems.

To open a new **3D Modeler** window, do one of the following:

- Insert a new HFSS design into the current project.
- Double-click an HFSS design in the project tree.

If there a **3D Modeler** window is not open, do one of the following:

- On the **HFSS** menu, click **3D Model Editor**.
- Right-click the design name in the project tree, and then click **3D Model Editor** on the shortcut menu.

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

Drawing Objects

You can draw one-, two-, or three-dimensional objects using the **Draw** commands. You can alter objects individually or together to create the geometry of your structure. In the **Tools>3 D Modeler Options, Drawing** tab, you can set a default to either draw objects directly with the mouse or by invoking a **Properties** dialog in which you can enter the values for the object dimensions. The **Dialog** mode drawing feature works with the equation based line, and all two and three dimensional objects. You can toggle to **Point** mode via the **F3** function key and to **Dialog** mode via the **F4** function key.

One-dimensional (1D) objects in HFSS include straight line, arc line, and spline segments, or a combination of these - called polylines. One-dimensional objects are open objects; their boundaries do not enclose a region, unless you connect their endpoints. They have length, but no surface or volume. Generally they are used as temporary objects from which to create 2D objects.

Two-dimensional (2D) objects in HFSS include objects such as arcs, rectangles, ellipses, circles, and regular polygons. Two-dimensional objects are closed sheet objects; their boundaries enclose a region. You can create 2D sheet objects by covering the enclosed region. By default, the history tree organizes sheet objects according to their boundary assignments. To change this, select the **Sheets** icon, and right-click to display the **Group Sheets by Assignment** checkbox.

Three-dimensional (3D) objects in HFSS include objects such as boxes, cylinders, regular polyhedrons, cones, spheres, toruses, and helices. These objects have boundaries that enclose a region with volume. You can create 3D objects by manipulating 2D objects along a plane or by using the appropriate **Draw** commands. By default, the history tree groups 3D objects by material. To change this, select the **Objects** icon, and right click to display the **Group Objects by Material** checkbox.

After you draw an object in the **3D Modeler** window, you can modify the object's properties, such as its position, dimensions, or color, in the **Properties** dialog box. Parameters that can be assigned a value can be assigned a variable.

Drawing a Straight Line Segment

To create an object with one or more straight line segments, use the **Draw>Line** command.

1. On the **Draw** menu, click **Line**  .
2. Select the first point of the line in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   To delete the last point that was entered, click **Back Up** on the shortcut menu.
3. Select the endpoint of the line by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.

   The endpoint serves as the start point for a subsequent line segment.

   To delete all points and start over, press **ESC** or click **Escape Draw Mode** on the shortcut menu.

4. Complete the line in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the shortcut menu.
   - Press **Enter**.

   The **Properties** dialog box appears, enabling you to modify the object's attributes.
5. Click **OK**.

Note While drawing a polyline, you can switch between straight line, arc line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Related Topics

Deleting Polyline Segments

Converting Polyline Segments

Drawing a Three-Point Arc Line

In HFSS, a three-point arc line segment is an arced line defined by three points on its curve. Use the **Draw>Arc>3 Point** command to create a polyline object with one or more arc line segments.

1. On the **Draw** menu, point to **Arc**, and then click **3 Point**  .
2. Select the start point of the arc in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** text boxes.
3. Select the midpoint of the arc by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.

   To delete the last point that was entered, click **Back Up** on the shortcut menu.

   To delete all points and start over, press **ESC** or click **Escape Draw Mode** on the shortcut menu.
4. Select the endpoint of the arc by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.

   The endpoint serves as the start point for a subsequent arc line segment.
5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.

   The **Properties** dialog box appears, enabling you to modify the object's attributes.
6. Click **OK**.

   Based on the three points you specified, HFSS calculates the center point and radius of the arc and draws an arced line through the three points.

Note While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Related Topics

Drawing a Center-Point Arc Line

Deleting Polyline Segments

Converting Polyline Segments

Drawing a Center-Point Arc Line

In HFSS, a center-point arc line segment is an arced line defined by a center point, start point and angle. Use the **Draw>Arc>Center Point** command to create a polyline object with one or more center-point arc line segments.

1. On the **Draw** menu, point to **Arc**, and then click **Center Point**  .
2. Select the center point of the arc in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** text boxes.
3. Select the start point, or radius, of the arc by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.

   To delete the last point that was entered, click **Back Up** on the shortcut menu.

   To delete all points and start over, press **ESC** or click **Escape Draw Mode** on the shortcut menu.
4. Select the angle, or endpoint, of the arc by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.
5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.

   The **Properties** dialog box appears, enabling you to modify the object's attributes.
6. Click **OK**.

Note While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Spline

A spline is a curved line defined by three points. HFSS uses a natural spline type: a piecewise cubic spline with an end condition that has a derivative of zero. Use the **Draw>Spline** command to create a polyline object with one or more spline segments.

1. On the **Draw** menu, click **Spline**  .
2. Select the spline's start point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes, and then press **Enter**.

   To delete the last point entered, click **Back Up** on the shortcut menu.

   To delete all selected points and start over, press **ESC** or click **Escape Draw Mode** on the

shortcut menu.

3. Select the midpoint of the spline by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.
4. Select the endpoint of the spline by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.

   The endpoint serves as the start point for a subsequent spline segment.
5. Complete the spline in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the shortcut menu.
   - Press **Enter**.

   The **Properties** dialog box appears, enabling you to modify the object's attributes.
6. Click **OK**.

Note While drawing a polyline, you can switch between spline, straight line, or arc line segments using the **Set Edge Type** commands on the shortcut menu.

Related Topics

Deleting Polyline Segments

Converting Polyline Segments

Drawing a Polyline

A polyline is a single object that includes any combination of straight line, arc line, or spline segments. The endpoint of one segment is the start point for the next segment. Use the shortcut menu's **Set Edge Type** commands to switch between straight line, arc line, or spline segments while drawing a polyline.

1. On the **Draw** menu, click **Line**  .
2. Right-click in the **3D Modeler** window to access the shortcut menu, and then point to **Set Edge Type**.
3. Click **Straight**, **Spline**, **3 Point Arc**, or **Center Point Arc** depending on which type of polyline segment you want to draw.
4. If you clicked **Straight**, follow the procedure for drawing a straight line.

   If you clicked **Spline**, follow the procedure for drawing a spline.

   If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.

   If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.
5. Repeat steps 2 and 3 for each segment of the polyline object. The endpoint of the previous segment serves as the start point for the next segment.
6. Complete the polyline in one of the following ways:
   - Double-click the endpoint of the final segment.

- Click **Done** on the shortcut menu.

Note To connect the polyline's start and endpoints, click **Close Polyline** on the shortcut menu.

The **Properties** dialog box appears, enabling you to modify the object's attributes.

7. Click **OK**.

Related Topics

Deleting Polyline Segments

Converting Polyline Segments

Modifying Lines on Line Plots

Generate History

Inserting Line Segments

You can insert line segments of various kinds for existing line objects.

1. Select the line object in the **History** tree

   This highlights the object and enables the **Insert Line Segment** commands in the **Draw** menu and short-cut menu.

2. Use the cascade menu from the **Draw>Line Segment** command to or the right-click menu to select whether to **Insert Before Line Segment** or **Insert After Line Segment**.
3. Use the next cascade menu to specify the kind of segment to add. These can be: Straight, Spline. 3 Point Arc, or Center Point Arc.
4. If you clicked **Straight**, follow the procedure for drawing a straight line.

   If you clicked **Spline**, follow the procedure for drawing a spline.

   If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.

   If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.

5. Repeat steps 2 and 3 for each segment of the polyline object. The endpoint of the previous segment serves as the start point for the next segment.
6. Complete the polyline in one of the following ways:
   - Double-click the endpoint of the final segment.
   - Click **Done** on the shortcut menu.

Note To connect the polyline's start and endpoints, click **Close Polyline** on the shortcut menu.

The **Properties** dialog box appears, enabling you to modify the object's attributes.

7. Click **OK**.

Related Topics

Drawing a Center-Point Arc Line

Deleting Polyline Segments

Converting Polyline Segments

Drawing an Equation-Based Curve

Any line that can be described by an equation in three dimensions can be drawn.

1. On the **Draw** menu, click **Equation Based Curve** .
2. The **Equation Based Curve** dialog box opens. Enter the equations for the X, Y, and Z components of the curve in terms of parameter **_t**. Use the ellipsis (...) buttons to open an **Edit Equation** dialog box which allows you to select from the available functions, operators, and quantities.
3. Enter the lower and upper bounds for the parameter **_t**.
4. Enter the number of points to be used to analyze or compute values along the line segment.
5. Click **OK** on the **Equation Based Curve** dialog box.

   The **Properties** dialog box appears, enabling you to modify the object's attributes.
6. Click **OK** on the **Properties** dialog box.

Drawing a Circle

Draw a circle by selecting a center point and a radius. Circles are drawn as true surfaces in HFSS.

1. On the **Draw** menu, click **Circle** .
2. Select the center point of the circle in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the radius by selecting a point on the circle's circumference in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   The **Properties** dialog box appears, enabling you to modify the object's properties.
4. Click **OK**.

   If the **Automatically cover closed polyline** option is selected in the **3D Modeler Options** window, the circle will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Related Topics

Covering Lines

Drawing an Ellipse

Draw an ellipse by specifying a center point, base radius, and secondary radius.

1. On the **Draw** menu, click **Ellipse**  .
2. Select the center point of the ellipse in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the base radius of the ellipse. If the current drawing plane is xy, then x is the base radius direction. If the drawing plane is yz, then y is the base radius direction. If the drawing plane is xz, then z is the base radius direction. Select the point in one of the following ways:
   - Click the point. HFSS constrains mouse movement to the base radius direction.
   - Type the coordinates of a point relative to the center point in the **dX**, **dY**, or **dZ** box, where **d** is the distance from the previously selected point.
4. Specify the secondary radius of the ellipse. Select the point in one of the following ways:
   - Click the point. HFSS constrains mouse movement to a point on the plane orthogonal to the base radius direction.
   - Type the coordinates of a point relative to the center point in the **dX**, **dY**, or **dZ** box.

   The **Properties** dialog box appears, enabling you to modify the object's properties.

   The **Ratio** value represents the aspect ratio of the secondary radius to the base radius.
5. Click **OK**.

   If the **Automatically cover closed polyline** option is selected in the **3D Modeler Options** window, the ellipse will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

If the base radius is larger than the secondary radius, the ellipse's longer axis will lie along the default base radius direction. If the secondary radius is larger than the base radius, the ellipse's

longer axis will lie perpendicular to the default base radius direction. To create an ellipse with an arbitrary orientation, rotate or move the ellipse after drawing it.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Related Topics

Covering Lines

Drawing a Rectangle

Draw a rectangle (or square) by selecting two diagonally opposite corners.

1. On the **Draw** menu, click **Rectangle**  .
2. Select the first diagonal corner in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   To delete the selected point and start over, press **ESC** or click **Escape Draw Mode** on the shortcut menu.
3. Select the second corner of the rectangle in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the first diagonal corner in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   The **Properties** dialog box appears, enabling you to modify the object's properties.
4. Click **OK**.

   If the **Automatically cover closed polyline** option is selected in the **3D Modeler Options** window, the rectangle will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Related Topics

Covering Lines

Drawing a Regular Polygon

A regular polygon is a 2D object with three or more equal sides. Regular polygons are useful for drawing faceted 2D objects.

1. On the **Draw** menu, click **Regular Polygon**  .
2. Select the center point of the polygon in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the polygon's radius, the distance from the center point to one of the polygon's vertices, in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.
4. In the **Segment number** dialog box, enter the **Number of segments** in the polygon, and then click **OK**.

   The **Properties** dialog box appears, enabling you to modify the object's properties.
5. Click **OK**.

Note The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

If the **Automatically cover closed polyline** option is selected in the **3D Modeler Options** window, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Related Topics

Covering Lines

Drawing an Equation-Based Surface

Any surface that can be described by an equation in three dimensions can be drawn.

1. On the **Draw** menu, click **Equation Based Surface**  .
2. The **Equation Based Surface** dialog box opens. Enter the equations for the X, Y, and Z components of the surface in terms of parameters **_u** and **_v**. Use the ellipsis (...) buttons to open an **Edit Equation** dialog box which allows you to select from the available functions, operators, and quantities.
3. Enter the lower and upper bounds (start and end values) for the parameter **_u**.
4. Enter the number of points to be used to analyze or compute values on the surface with respect to the **_u** parameter.
5. Enter the lower and upper bounds (start and end values) for the parameter **_v**.
6. Enter the number of points to be used to analyze or compute values on the surface with respect

to the **_v** parameter.

7. Click **OK** on the **Equation Based Surface** dialog box.

   The **Properties** dialog box appears, enabling you to modify the object's attributes.

8. Click **OK** on the **Properties** dialog box.

Drawing a Sphere

Draw a sphere, a 3D circle, by selecting a center point and a radius. Spheres are drawn as true surfaces in HFSS.

1. On the **Draw** menu, click **Sphere**  .
2. Select the center point of the sphere in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the radius by selecting a point on the sphere's circumference in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   The **Properties** dialog box appears, enabling you to modify the object's properties.
4. Click **OK**.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Drawing a Cylinder

Draw a cylinder by selecting a center point, radius, and height. Cylinders are drawn as true surfaces in HFSS.

1. On the **Draw** menu, click **Cylinder**  .
2. Select the center point of the cylinder's base circle in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the radius by selecting a point on the base circle's circumference in one of the follow-

ing ways:

- Click the point.
- Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Specify the cylinder's height by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the **dX**, **dY**, and **dZ** boxes.

Note If you create a cylinder with a height of zero, HFSS draws a circular sheet object.

The **Properties** dialog box appears, enabling you to modify the object's properties.

5. Click **OK**.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Drawing a Box

Draw a box by selecting two diagonally opposite corners of the base rectangle, then specifying the height.

1. On the **Draw** menu, click **Box**  .
2. Select the first diagonal corner of the base rectangle in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   To delete the selected point and start over, press **ESC** or click **Escape Draw Mode** on the shortcut menu.
3. Select the second corner of the base rectangle in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the first diagonal corner in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the height of the box by selecting a point on the axis perpendicular to the base rectangle. Select the point by clicking the point or typing the coordinates in the **dX**, **dY**, and **dZ** boxes.

The **Properties** dialog box appears, enabling you to modify the object's properties.

5. Click **OK**.

Drawing a Regular Polyhedron

In HFSS, regular polyhedrons are 3D objects with regular polygon faces; each face has three or more equal sides. Regular polyhedrons are useful for drawing faceted 3D objects.

1. On the **Draw** menu, click **Regular Polyhedron**  .
2. Select the center point of the polyhedron in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Select the radius of the polyhedron, the distance from the center point to one of the polyhedron's vertices, in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.
4. In the **Segment number** dialog box, enter the **Number of segments** in the polyhedron, and then click **OK**.

   The **Properties** dialog box appears, enabling you to modify the object's properties.
5. Click **OK**.

Drawing a Cone

Draw a cone by selecting the center point and radius of the cone's base circle, then specifying the radius of the cone's top circle and the cone's height. Cones are drawn as true surfaces in HFSS.

1. On the **Draw** menu, click **Cone**  .
2. Select the center point of the cone's base circle in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the radius of the cone's base circle by selecting a point on the base circle's circumference. Select the point in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the radius of the cone's top circle by selecting a point on its circumference. Select the point by clicking it or typing its coordinates in the **dX**, **dY**, and **dZ** boxes.

   To create an apex, select the same center point as the cone's base circle.
5. Specify the height of the cone by selecting a point on the axis perpendicular to the base circle's

plane. Select the point by clicking the point or typing the coordinates in the **dX**, **dY**, and **dZ** boxes.

The **Properties** dialog box appears, enabling you to modify the object's properties.

6. Click **OK**.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Drawing a Torus

Draw a torus by selecting its center point, major radius, and minor radius. HFSS then sweeps a circle around a circular path. Toruses are drawn as true surfaces in HFSS.

1. On the **Draw** menu, click **Torus**  .
2. Select the center point of the torus in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Specify the major radius by selecting a point in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   The major radius determines the diameter of the torus.
4. Specify the minor radius by selecting a point relative to the major radius point.

   The minor radius determines the diameter of the "donut hole".

   The **Properties** dialog box appears, enabling you to modify the object's properties.
5. Click **OK**.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Drawing a Helix

A helix is a 3D spiral object created by sweeping a 1D or 2D object along a vector. Sweeping a 1D object results in a hollow 3D object. Sweeping a 2D sheet object results in a 3D solid object.

1. Select the 1D or 2D object you want to sweep to form a helix.
2. On the **Draw** menu, click **Helix**  .
3. Draw the vector you want to sweep the object along. The two points which describe the vector affect axis direction only and not the helix length. The helix length is determined by entry of the pitch and number of turns.
   a. Select the start point by clicking the point or typing its coordinates in the **X**, **Y**, and **Z** text boxes.
   b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX**, **dY**, and **dZ** boxes.

      The **Helix** dialog box appears.
4. Select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Pitch** text box, type the distance between each turn in the helix, and click a unit in the pull-down list.
6. In the **Turns** text box, type the number of complete revolutions the object will make along the vector.

   The selected object is swept along the vector to form a helix. The original object you swept is deleted.

   The **Properties** dialog box appears, enabling you to modify the object's properties.
7. Click **OK**.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable tradeoff between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Drawing a Spiral

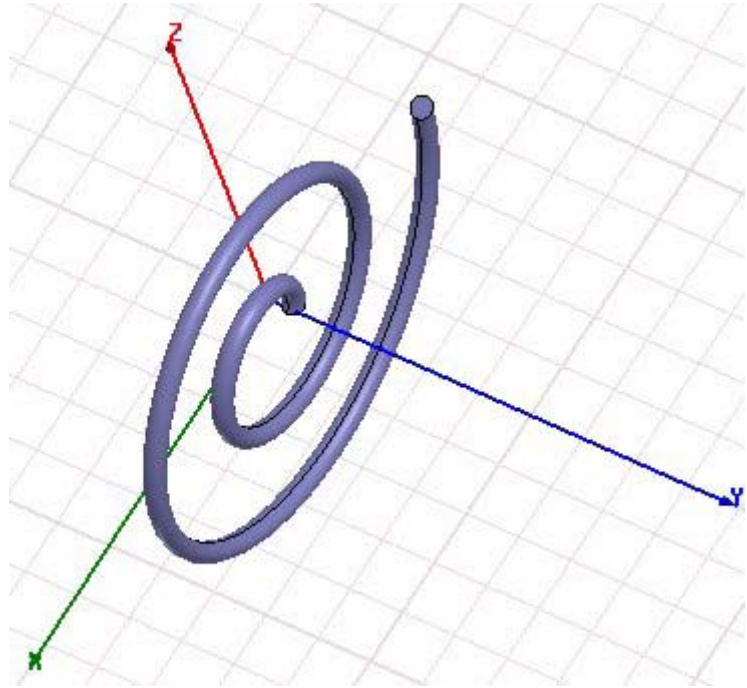
A spiral is a 2D or 3D spiral object created by sweeping an object around a vector. Sweeping a 1D object results in a 2D sheet object. Sweeping a 2D sheet object results in a 3D solid object.

1. Select the 1D or 2D object you want to sweep to form a spiral.
2. On the **Draw** menu, click **Spiral**  .
3. Draw the vector you want to sweep the object around:
   a. Select the start point by clicking the point or typing its coordinates in the **X**, **Y**, and **Z** text boxes.
   b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX**, **dY**, and **dZ** boxes.

   The **Spiral** dialog box appears.
4. Select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Radius Change** text box, type the difference in radius between each turn of the spiral.

   The radius of the first turn is measured from the center point of the 1D or 2D object you are sweeping to the vector you drew.
6. Click a unit for the radius in the pull-down list.
7. In the **Turns** text box, type the number of complete revolutions the object will make around the vector.

   The selected object is swept around the vector to form a spiral. The original object you swept is deleted.The **Properties** dialog box appears, enabling you to modify the object's properties.
8. Click **OK**.



This 3D spiral was created from a 2D circle drawn at z = 0. The turn direction was right hand, the radius change was set at 2, and the number of turns was set at 2.

Note The HFSS 3D Geometry Modeler permits drawing of true-curved objects. However, the solution will be obtained with a tetrahedral mesh which conforms to the true surface only within the limits identified by certain mesh settings. HFSS has default settings for this conformance which is a reasonable trade-off between solution speed and solution quality for most objects, but may not be ideal for all such objects. High-aspect ratio curves structures, such as helices with narrow and curved cross-sections, may benefit from user control of the faceting values. For details about these commands see:

Technical Notes, "Surface Approximations" and related sections, "Modifying Surface Approximations," and "Guidelines for Modifying Surface Approximations"

Drawing a Spiral using User Defined Primitives

Ansoft provides you with a DLL to define the parameters of a rectangular spiral.

1. Click **Draw>User Defined Primitive>SysLib>Examples>RectangularSpiral**.

   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to see edit the parameters. An **Info** tab contains information about the user defined primitive, its pur-

pose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

Xpos	Type the location of the starting point in the X direction.
Ypos	Type the location of the starting point in the Y direction.
TurnSep	Type the separation distance between turns.
Turns	Type the number of complete revolutions the object will make around the vector
Width	Type a value for the width of the spiral.
Height	Type a value for the height of the spiral. If you specify the height as zero, HFSS draws a sheet object.

3. Click **OK**.

   This creates the primitive and displays the **Properties** dialog for the new object.

Hint To see newly created DLLs, click **Draw>User Defined Primitive>Update Menu**.

To see the primitives that you have created, click **Draw>User Defined Primitive>UserLib.**

Related Topics

Creating a User Defined Primitive

Drawing a Bondwire

A bondwire is a thin metal wire that connects a metal signal trace with a chip. Please see the topic *Bondwires* in the *Technical Notes* before drawing a bondwire.

1. On the **Draw** menu, click **Bondwire**  .
2. Select the bond pad point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
3. Select the lead point by clicking the point or typing the coordinates in the **X**, **Y**, and **Z** boxes.

   The **Bondwires** dialog box appears.
4. In the **Type** list, click the JEDEC modeling standard shape you want the bondwire to have: **JEDEC 4-point** or **JEDEC 5-point**.
5. Type the number of facets in the bondwire in the **No. of Facets** text box.

   The minimum value is 3.
6. Enter the height between the bond pad and the top of the loop in the **h1** text box. Include the height's unit of length.

7. The value in the **h2** text box is the height between the bond pad and the lead point. It was calculated by HFSS based on the lead point you selected. If you modify the value of *h2*, the lead point will be modified.

   Optionally, type a new value in the **h2** text box. Include the height's unit of length.

8. If you selected **JEDEC 5-point**, do the following:
   a. Type the angle between the horizontal plane and the wire at the bond pad point in the **alpha** text box.
   b. Type the angle between the horizontal plane and the wire at the lead point in the **beta** text box.
9. Click **OK**.

Related Topics

Technical Notes: Bondwires

Drawing a Point

Drawing a point object within the problem region enables you to plot fields or perform field computations at that point. Points are always considered non-model objects by HFSS.

1. On the **Draw** menu, click **Point**  .
2. Select the point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   The point is listed under **Points** in the history tree.

Related Topics

Modifying Markers on Point Plots

Drawing Non-Model Objects

Drawing a Plane

A plane object is a cutplane through the problem region. You can plot fields or perform field computations on its surface. Planes are always considered non-model objects by HFSS.

1. On the **Draw** menu, click **Plane**  .
2. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   To delete the selected point and start over, press **ESC**.
3. Select a normal point in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the origin in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   The plane is created. Its center point is located at the origin you specified and oriented perpen-

dicular to the normal point you specified. The plane is listed under **Planes** in the history tree.

Note You only need to draw a plane that does not lie on a pre-defined xy, yz, and xz plane. Default planes are created on the xy, yz, and xz planes of the global coordinate system as well as any new coordinate system you create.

Related Topics

Drawing Non-Model Objects

Drawing Non-Model Objects

If you want to create an object that does not affect the geometric model, define the object as *non model*. This ensures that the object is used for analysis only; it will not affect the solution process. After drawing the object, assuming it lies in the problem region, you can plot near fields, electric fields, magnetic fields, or derived field quantities on it.

Following are examples of using non-model objects to analyze a solution:

- Draw a polyline along which to plot fields or perform field computations. Note that when you create a value versus distance plot, by default, the line will be divided into 100 equally spaced points. You can modify the number of points into which the line is divided in the **Edit Sweeps** dialog box.
- Draw a rectangle upon which to plot fields or perform field computations.
- Draw a volume box to analyze fields in areas of the problem region that are not occupied by an object or that consist of parts of several objects.
- Draw a plane, which is always a non-model object, upon which you can plot fields or perform field computations.
- Draw a point object, which is always a non-model object, in order to plot fields or perform field computations at that point.

What do you want to do?

Switch to non-model drawing mode. Objects you draw in non-model mode will not be included in the solution process.

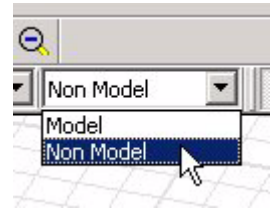
Modify an existing model object to be a non-model object.

Selecting Non-Model Drawing Mode

To switch to non-model drawing mode:

1. On the **3D Modeler** menu, point to **New Object Type**, and then click **Non Model**.
   - Alternatively, click **Non Model** on the drawing model pull-down list in the 3D Modeler

Draw toolbar:



2. Draw the object.

Related Topics

Changing an Object to Non Model

Drawing Non-Model Objects

Changing an Object to Non Model

To modify an existing object to be a non-model object:

1. Select the object you want to modify.
2. In the **Properties** dialog box, clear the **Model** option.

   The object will not be included in the solution process. If the object lies in the problem region, you can plot solution quantities on it.

Related Topics

Selecting Non-Model Drawing Mode

Drawing Non-Model Objects

Model Analysis

For some models it may be beneficial to remove unnecessary small entities and to fix object misalignments to avoid potential mesh issues. HFSS includes Model Analysis functions to help you evaluate models you have imported or created. Select **3D Modeler> Model Analysis** to see the menu options. Depending on the design and the current selection, some features may not be enabled. The menu includes the following commands.

- **Analyze Objects**
- **Interobject Misalignment**
- **Analyze Surface Mesh**
- **Heal**
- **Show Analysis dialog**
- **Align Faces**
- **Remove Faces**
- **Remove Edges**

Note Before running model analysis, you must remove all command history for the selected object by using the Purge History command.

Analysis Options Dialog

Select the objects you want to analyze and click **3D Modeler> Model Analysis>Analyze Objects** to display the **Analysis Options** dialog. Select the types of geometry entities that you want to detect and specify the corresponding tolerance values.

- Small Edges, length less than
- Small Faces, area less than
- Sliver Faces
  - Object Bounding Box scale factor
  - Sliver Edge width

Minimum and maximum edges and face areas are stated at the bottom of the dialog for reference purposes:

- Edge length min and max
- Face area min and max

Clicking **OK** on this dialog displays the **Model Analysis** dialog which contains the results of the analysis.

Related Topics

Heal

Model Analysis dialog.

Model Analysis dialog

This dialog contains results for all model analysis, including diagnostic information relating to mesh issues. To view the analysis options:

1. Select the appropriate **3D Modeler> Model Analysis>Show Analysis Dialog** command to display the **Model Analysis** dialog box.

   This dialog also appears automatically after selecting **OK** to the **Analysis Options** dialog.

   - Objects tab
   - Object Misalignment tab
   - Surface Mesh (Single/Pairs) tab
   - Last Simulation Mesh tab - displays a log of the error type and error details.
2. Select the **Auto Zoom to Selection** check box to automatically zoom to the item selected in the Objects tab.

Related Topics

Heal

Objects Tab

All results relating to model analysis of specific objects are presented under the **Objects** tab. The **Objects** tab for the **Model Analysis** dialog lets you view the following information:

1. The results table contains the following information.
   - Name - column listing the objects in the current design.
   - Last Analysis status - column giving the analysis status of the listed objects. Objects can have the following status:
     - Good - the object contains no invalid geometry entities given the tolerance values specified in the **Analysis Options** dialog.
     - Null Body - the object is non-existent.
     - Analysis not performed - the object was not selected for analysis.
     - Invalid entity errors - these are api_check_entity() errors and non-manifold errors which must be fixed prior to meshing.
     - Small entity errors - small faces, sliver faces and small edges that are optionally detected based on the tolerance limits specified in the **Analysis Options** dialog.
2. Select any object name in the table which contains errors to display a set of radio buttons in the panel and a list of corresponding faces, edges and vertices.

Note **Auto Zoom to Selection** -- if this option is checked, HFSS automatically zooms to the item selected in the **Model Analysis** dialog box.

3. Select the face, edge or vertex entity from the list to view the error description in the **Description** field.
4. Select the **Delete** button if you want to remove a selected face or edge entity.

5. Select the **Perform** button to list the commands that you can execute on the selected objects in the Results table.
    - **Heal Objects -** repairs invalid geometry entities for the selected objects within the specified tolerance settings. The **Healing Analysis** dialog will appear.
    - **Analyze Objects** - evaluates the object status. Selecting this displays the Analysis Options dialog.
    - **Analyze Surface Mesh** - invokes a mesh for each selected object and reports analysis results under the **Surface Mesh (Single/Pairs)** tab. Selecting this option displays a dialog with radio buttons to select.
        - Perform Object Pairs Analysis - evaluates mesh for all combinations of the selected objects.
        - Ignore objects separated by greater than a specified value - object pairs are disregarded from analysis if their separation is greater than the specified value.
        - Click **OK** to perform the analysis with the selected options.
    - Analyze Interobject Misalignment - determines any misalignments between two selected objects in the results table. The results are reported under the **Objects Misalignment** tab.

- **Display Healing Log** -- checking this causes the **Model Analysis** dialog to display a healing log which includes information about operations performed on an object during the healing process.

Object Misalignment Tab

The table in this panel displays results of an Interobject Misalignment analysis. All misaligned face pairs corresponding to the analyzed objects are listed in the table.

- **Align Faces** - select a face pair in the table and click the **Align Faces** button to align selected faces.
- **Clear All Analysis Data** - this button removes all information from the tables.
- **Auto Zoom to Selection** -- if this option is checked, HFSS automatically zooms to the item selected in the table.

Surface Mesh (Single/Pairs) Tab

The panel displays the results of a surface mesh analysis.

1. You can display results for:
    - Individual Objects
    - Object Pairs

Note **Auto Zoom to Selection** -- if this option is checked, HFSS automatically zooms to the object or object pair selected.

2. The results table contains the following information:

- Object - column listing object name or a pair of object names.
- Last Analysis Status - column stating the meshing status of the object or object pair.

- Mesh Success
- Mesh Failure

- Error Type - this column gives the category of error that caused the mesh failure.
- Error Detail - provide specific geometry information regarding mesh error location.

Display options include:

- **Display Mesh Analysis** log checkbox -checking this displays further details concerning each error to be listed.
- **Auto Zoom to Selection** -- checking this causes HFSS to automatically zoom to objects or faces corresponding to the error.

Last Simulation Mesh Tab

The table in this panel lists all model errors as viewed by the mesher.

- Error Type - this column gives the category of error that caused the mesh failure.
- Error Detail - provide specific geometry information regarding mesh error location.

Display options include:

- **Display Mesh Analysis** log checkbox -checking this displays further details concerning each error to be listed.
- **Auto Zoom to Selection** -- checking this causes HFSS to automatically zoom to objects or faces corresponding to the error.

Align Faces

Use this **3D Modeler>Model Analysis** command to align the selected faces. You can also use the toolbar icon when you have made an appropriate face selection 

Remove Faces

Use this **3D Modeler>Model Analysis** command to remove the selected faces. You can also use the toolbar icon when you have made an appropriate face selection 

Remove Edges

Use this **3D Modeler>Model Analysis** command to remove the selected edges. You can also use the toolbar icon when you have made an appropriate edge selection 

Related Topics

Technical Notes: Healing Models

Technical Notes: Error Types

Technical Notes: Error Detection

Analyze Objects

Analyze Interobject Misalignment

Analyze Surface Mesh

Healing

Heal

Use this command to heal an imported object. Imported objects, which have only one operation on the history tree, can be healed. (Use the Purge History command to remove unwanted history operations before using **Heal**.) When models are imported, two types of errors can occur – geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors are errors in how the underlying components like faces, edges and vertices are connected. These must be fixed before mesh analysis can be performed.

Basic Steps in the Heal Process

There are four steps that are performed for healing objects.

1. Validation check.
2. Basic healing. This is done for all selected objects. Basic healing consists of fixing surface normals in the object and updating the orientation of (to avoid having an object with negative volume).
3. Advanced healing. This is auto-heal. This is invoked on objects that require healing, that is. bodies that have failed api_check_entity() errors or have non-manifold errors.
4. Small feature removal. If you choose to remove small edges, small faces and sliver faces, the actions are performed on all selected objects. There is no guarantee that small feature removal will be successful.

The above actions are performed on the selected objects. If you choose objects for healing which have not been analyzed, analysis is performed to determine its state (that is, whether it has invalid entities, small entities, and so forth). Invalid objects have all the above steps performed. Advanced healing is not performed on objects that do not require it.

While working on analyzing complex bodies, it is sometimes useful to examine faces, edges and vertices. In particular it is useful to find the connected faces for a face or edge or vertex, connected edges for a face/edge/vertex and connected vertices for a face/edge/vertex. The additional selection modes are available under **Edit->Select** and via the toolbar icons.

Related Topics

Validation Check

Fix Intersections Between Objects

Fix Object Pair Intersections

Analysis Phase

Healing Non-manifold Objects

Healing Options

Technical Notes: Healing Models

Technical Notes: Error Types

Technical Notes: Error Detection

Healing Stage One: Validation Check

1. After import, perform validation check. This lets you focus on objects and object pairs that pre-

vent the mesh from being invoked. The objects that fail api_check_entity() should be analyzed via **Analyze Objects** menu item.

2. Select the objects and invoke **3D Modeler->Model Analysis->Analyze Objects**.

   This displays an options dialog to allow you to perform small feature detection and on completion, the **Model Analysis** dialog is displayed.

3. Choose the objects that have "Invalid Entities Found" and **Perform->Heal Objects**.

   In most cases, the objects will be healed and the errors fixed.

4. If errors still persist, choose the edges and faces and click on **Delete**.

   This will replace the selected face/edge object by a tolerant edge/vertex respectively. In some cases the replacement of the face/edge by tolerant edge/vertex will fail.

When models pass the initial validity checks, mesh generation could still fail. The following errors can be present in models: (See Error Detection.)

1. Non-manifold topology. These are non-manifold edges and vertices that are present in the model.
2. Object pair intersection. This detects whether pairs of objects intersect.
3. Small feature detection – small edge length, small face area and sliver face detection.
4. Mis-aligned entities detection – detects pairs of faces from objects that can be aligned to remove object intersections. This improves the probability of mesh success.
5. Mesh failure error display. This is available for single object, object pairs and last simulation run (all objects in a model). Errors reported by the meshing module are reported to the user.

Errors of type 3 and 4 must be resolved before the mesh can be invoked on the model.

By default, the **Heal** command is automatically applied to imported objects.

Healing Stage Two: Fix Intersections Between Objects

The second stage in healing is to fix intersections between objects. After validation check is performed, the pairs of objects that intersect are chosen for analysis. Use the analysis results to find whether objects have faces that can be aligned.

Choose all the bodies that intersect with another body.

1. From the **Model Analysis** dialog choose perform/Analyze Interobject misalignment. Or you can run **3D Modeler->Model Analysis->Analyze Interobject Misalignment**.

   If the analysis finds object pairs that can be aligned, they will be displayed in the **Objects Misalignment** tab.

2. You can select individual or multiple rows and perform Align Faces. In some cases, face alignment will fail if the topology of the body changes by a large factor after alignment.
3. Identify individual bodies and body pairs that fail to mesh.
4. Perform Mesh analysis on individual objects and object pairs.
5. Review the reports and fix the errors.

Healing Stage Three: Fix Object Pair Intersections

The next stage of healing is to fix object-pair intersections. The healing process in stage two might fix some of these errors (by alignment).

1. If these errors are still present, you must remove them either by using the **Delete Faces** command (**3d Modeler->Model Analysis->Delete Faces**) or by performing Boolean subtract.
2. Overlap between objects is too large to be fixed by healing or by face alignment. Boolean intersect shows the common portion between the bodies. In this case, use a subtract operation to remove overlaps.

Healing Stage Four: Analysis Phase

This is an analysis phase. If the last simulation run has errors, these are displayed in the **Model Analysis->Last Simulation Run** tab.

Healing Non-Manifold Objects

If more than two faces meet along an edge, the edge is non-manifold. Normally, if you collect every face at a vertex that can be reached from a given face by crossing one or more edges starting or ending at the vertex, the collection contains all of the faces that meet at that vertex. If this is not the case, the vertex is non-manifold. One or more wires may be attached to a vertex that is already on the boundary of one or more faces. This again makes the vertex non-manifold.

To heal non-manifold objects:

1. Identify an edge that is non-manifold.
2. Select the connected faces.

   You can use the Face selection toolbar icons.
3. Create a face coordinate system on the planar face.
4. Create a small box to cover the non-manifold edge.
5. Either do a union or a subtraction to remove the faces that contain the non-manifold edge.

   The non-manifold edge is now removed. You may also remove or add a small portion of the model.
6. Do for all the non-manifold edges

Setting the Healing Options

1. Click **3D Model>Model Analysis>Heal** to open the **Healing Options** dialog. You can also open the **Healing Options** dialog from the **Model Analysis** dialog via the **Objects** tab drop down menu. The **Healing Options** dialog allows you to set parameters to remove:
   - Small Edges, less than a specified value.
   - Small faces, less than a specified value.
   - Sliver faces, less than either:
     - Bounding box, less than a specified scale factor
     - Sliver Edge width, less than a specified value.

   Sliver faces have a maximum distance among the long edges that is smaller than the specified

tolerance and have at least one short edge and at most three long edges. A short edge has a length less than the specified tolerance. A long edge has a length greater than the specified tolerance. You can give the tolerance as a absolute value or a factor of the bounding box containing the face.

2. Click **OK** to apply the specified Healing options and to open the Analysis dialog.

Creating a User Defined Primitive

HFSS allows you to generate user-defined primitives, primitives customized to suit any application. User-defined primitives are accessed using DLLs that you build and compile.

HFSS includes example C++ source and header files that can be used to generate DLLs. The files are located in the **`UserDefinedPrimitives/Examples`** subdirectory under the **`hfss10`** directory.

As an example, create the primitive **myUDP.dll** using Microsoft Visual C++ Developer Studio:

1. Create a directory to store all of the workspace information, call it `UDPDir`.
2. Use the sample workspace **RectangularSpiral.dsw** as a template:
   a. Copy **RectangularSpiral.dsw** and **RectangularSpiral.dsp** from the **`UserDefinedPrimitives/Examples`** directory to this new directory.
   b. Make sure the new files have write permissions.
   c. Rename the files to **myUDP.dsw** and **myUDP.dsp** respectively.
   d. Open the **.dsw** and **.dsp** files in a text editor, and replace every occurrence of **RectangularSpiral** with **myDLL**.
   e. Save **myUDP.dsp** and **myUDP.dsw**.
3. In the **`UDPDir`** directory, create a **`Headers`** subdirectory.
4. Copy the **UserDefinedPrimitiveStructures.h** and **UserDefinedPrimitiveDLLInclude.h** files from the **`UserDefinedPrimitives/Headers`** directory.

Note The header files include information on the methods that are available for use in your source code. They must be included when you compile the DLL.

5. In the **`UDPDir`** directory, create a **`Source`** subdirectory.
6. Use the sample source file **RectangularSpiral.cpp** as a template:
   a. Copy **RectangularSpiral.cpp** from the **`UserDefinedPrimitives/Examples`** directory to this new directory.
   b. Make sure the new file has write permission.
   c. Rename the file to **myUDP.cpp**.

   The resulting directory structure will appear similar to the following:

UDPDir/

myUDP.dsw

myUDP.dsp

Headers/

`UserDefinedPrimitiveDLLInclude.h`

`UserDefinedPrimitiveStructures.h`

Sources/

myUDP.cpp

7. Open **myUDP.dsw** using Microsoft Visual C++ Developer Studio, and edit the source code to create your desired primitive. You may also add additional headers and source files as appropriate.

   The UDP dll contains a data structure called UDPPrimitiveTypeInfo. This contains information about the udp, its purpose, company/author who created it, date created and the version number. When you select a primitive from your library, you see the **Create Primitive** dialog with a **Parameters** tab for setting the parameters, and an **Info** tab with the information from this data structure.
8. Build **myUDP.dll** using the **Win32 Release** configuration.
9. Copy the resulting file **myUDP.dll** to the **`hfss10/userlib/UserDefinedPrimitives`** directory.
10. To view your primitives in HFSS, clicking **Draw>User Defined Primitive>UserLib**.

Note On UNIX, you may use the same example directory structure, source, and header files to build and compile a shared library using C++. The resulting shared library will have a **.so** extension for Solaris and a **.sl** extension for HP-UX, and needs to be placed in the same **`hfss10/userlib/UserDefinedPrimitives`** directory.

As with the Windows DLL, the compiled library will work only on the operating system on which it was built.

Modifying Objects

You can quickly modify the position, dimensions, and other characteristics of objects created in the **3D Modeler** window.

What do you want to do?

- Assign color to an object.
- Assign transparency to an object.
- Copy and paste objects.
- Delete objects.
- Delete Last Operation
- Move objects.
- Rotate objects.
- Mirror objects about a plane.
- Offset an object (move every face of an object).
- Duplicate objects.
- Scale the size of objects.
- Sweep objects.
- Cover lines.
- Cover faces.
- Uncover faces.
- Detach faces.
- Create a new object by taking a cross-section of a 3D object.
- Connect objects.
- Move faces.
- Unite objects.
- Subtract objects.
- Create objects from intersections.
- Create an object from a face.
- Split objects.
- Separate objects.
- Convert polyline segments.
- Rounding the edge of an object (Fillet)
- Flattening the edge of an object (Chamfer)
- Purge History
- Generate History

Assigning Color to Objects

1. Select the object to which you want to assign a color.
2. In the **Properties** dialog box, click the **Attribute** tab.
3. Click **Edit** in the **Color** row.

   The **Color** palette appears.
4. Select a color from the **Color** palette, and then click **OK**.

   The color is assigned to the selected object.

Related Topics

Setting the Default Color of Objects

Setting the Default Color of Objects

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler** options.
2. Click the **Display** tab.
3. Select **Object** from the **Default color** pull-down list.
4. Click the color button beside the **Default color** pull-down list.

   The **Color** palette appears.
5. Select a color from the **Color** palette, and then click **OK**.

   Any objects you draw after this point will be assigned the default color you selected.

Setting the Default Color of Object Outlines

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler** options.
2. Click the **Display** tab.
3. Select **Object Wire** from the **Default color** pull-down list.
4. Click the color button beside the **Default color** pull-down list.

   The **Color** palette appears.
5. Select a color from the **Color** palette, and then click **OK**.

   The outlines of any objects you draw after this point will be assigned the default color you selected.

Assigning Transparency to an Object

1. Select the object to which you want to assign a transparency.
2. In the **Properties** dialog box, click the **Attribute** tab.
3. Click the value in the **Transparency** row.

   The **Set Transparency** window appears.
4. Move the slider to the right to increase the transparency of the object. Move the slider to the left to decrease the transparency of the object.
5. Click **OK**.

Related Topics

Setting the Default Transparency of Objects

Setting the Default Transparency of Objects

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler** options.
2. Click the **Display** tab.
3. Move the **Default transparency** slider to the right to increase the transparency of objects.

   Move the slider to the left to decrease the transparency of objects.

   Any objects you draw after this point will be assigned the default transparency you selected.

Copying and Pasting Objects

To copy objects and paste them in the same design or another design, use the **Edit>Copy** and **Edit>Paste** commands.

1. Select the objects you want to copy.
2. On the **Edit** menu, click **Copy**  .

   The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted.

   To cut an item to the clipboard and deleting the original, use the sissors icon on the toolbar.
3. Select the design into which you want to paste the objects. It can be the same design from which you copied the items.
4. Click in the **3D Modeler** window.
5. Select the working coordinate system. Objects are pasted relative to the current working coordinate system.
6. On the **Edit** menu, click **Paste**  .

   The objects appear in the new window.

Items on the Clipboard can be pasted repeatedly. The items currently stored on the Clipboard are replaced by the next items that are cut or copied.

Related Topics

Duplicating Boundaries and Excitations with Geometry

Copying Objects to the Clipboard

You can import images of the **3D Modeler** window into any other application. The image has to be copied to the clipboard, so that it can be imported into the other application.

To copy from HFSS and paste into another application:

1. Click **Edit>Copy To Clipboard**.

   The **3D Modeler** window is copied to the Clipboard as an image.
2. Select and open the application into which you want to paste the objects, and paste the image.

Deleting Objects

1. Select the objects to delete.
2. On the **Edit** menu, click **Delete**  .
   - Alternatively, press **Delete**.

   The objects are deleted.

Note To maintain valid boundaries, excitations, or other parameters that were associated with the deleted object, reassign them to other objects.

Related Topics

Deleting Polyline Segments

Deleting Startpoints and End points

Deleting Polyline Segments

A polyline is a single object that includes any combination of straight line, arc line, or spline segments. You can delete the first or last segment of a polyline by selecting it in the history tree and pressing **Delete**.

1. In the history tree, locate the polyline that contains the segment you want to delete. Expand this part of the history tree.
2. In the history tree, select the polyline segment operation you want to delete.
3. On the **Edit** menu, click **Delete**  .
   - Alternatively, press **Delete**.

   The polyline segment you selected is deleted.

Note You may delete one polyline segment at a time.

Deleting Start Points and Endpoints

If you select a polyline in the history tree, the **Delete Start Point** and **Delete End Point** commands may be enabled. These permit you to delete portions of the line.

1. In the history tree, locate the polyline that contains the segment you want to delete. Expand this part of the history tree.
2. In the history tree, select the polyline you want to edit.

   The segment is highlighted.
3. On the **Edit** menu or the shortcut menu, click either **Delete Start Point** to remove the leading segments or **Delete End Point** to remove the following segments.

   The designated segment is removed, and the line changes.

Delete Last Operation

To delete the last operation on an object:

1. Select the object.
2. Click **3D Modeler>Delete Last Operation.**

   This undoes the last operation, including removing that operation from the history, and updating the context for the **Undo** and **Redo** commands.

Related Topics

Undoing Commands

Redoing Commands

Moving Objects

1. Select the objects to move.
2. On the **Edit** menu, point to **Arrange**, and then click **Move**  .
3. Select an arbitrary anchor point in one of the following ways:
   - Click the point.
   - Enter the point's coordinates in the **X**,**Y**, and **Z** boxes.
4. Select a target point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the anchor point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   All selected objects move the distance determined by the offset between the anchor point and the target point.

Rotating Objects

Rotate objects about the x-, y-, or z-axis using the **Edit>Arrange>Rotate** command.

To rotate objects about an axis:

1. Select the objects to rotate.
2. On the **Edit** menu, point to **Arrange**, and then click **Rotate**  .

   The **Rotate** dialog box appears.
3. Select the axis about which to rotate the objects: **X**, **Y**, or **Z**.
4. Type the angle to rotate the objects in the **Angle** box.

   A positive angle causes the object to be rotated in the counter-clockwise direction. A negative angle causes the object to be rotated in the clockwise direction.
5. Click **OK**.

   The selected objects are rotated about the axis.

To rotate *and copy* objects, use the **Edit>Duplicate>Around Axis** command.

Mirroring Objects

Mirror an object about a plane using the **Edit>Arrange>Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to move an object and change its orientation.

To mirror an object about a plane:

1. Select the object you want to mirror. You can select multiple objects.
2. On the **Edit** menu, point to **Arrange**, and then click **Mirror**  .
3. Select a point on the plane on which you want to mirror the object. The distance between the point on mirror plane and point along the normal does not matter; only the vector direction matters

   A line drawn from this point to the mirror plane will be perpendicular to the plane.
4. Select a normal point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   The selected object is moved to the plane you specified and oriented according to the normal point you specified.

To mirror *and copy* objects about a plane, use the **Edit>Duplicate>Mirror** command.

Related Topics

Duplicating and Mirroring Objects

Offsetting Objects

Move every face of a 3D object in a direction normal to its surface using the **Edit>Arrange>Offset** command. The faces are moved a specified distance normal to their original planes. This command enables you to move every face of a solid object without having to individually select and move each face. Use the **Surfaces>Move Faces>Along Normal** command if you want to move just one or more faces of an object.

To offset every face of an object:

1. Select the object you want to offset.
2. On the **Edit menu**, point to **Arrange**, and then click **Offset**.

   The **Offset** dialog box appears.
3. Type the distance you want to move the object faces from their origins, and then select a unit from the pull-down list.
4. Click **OK**.

   The selected object's faces are moved the distance you specified.

Duplicating Objects

You can duplicate objects within a design using the **Edit>Duplicate** commands. Duplicates are dependent upon the parameters of their *parent* object at the time they were created, that is, they share the parent object's history at the time of creation. The command hierarchy in the history tree will show the duplication command, illustrating which commands affect all duplicates (those performed before the duplication) and which commands would not affect the duplicates (those performed after the duplication). For example, if you modify the radius of a parent object's hole, the change is applied to the holes of the object's duplicates because they share the radius specification history, but if you move the faces of the parent object, its duplicates are not affected because this operation took place after the duplicates were created.

Operations performed on duplicates are independent. For example, if you duplicate a cylinder twice, creating a row of three, and then split the second cylinder, the first and third cylinders are not affected by the split.

When creating duplicates, the parent object is duplicated along a line or around an axis the number of times you specify. You can also create a single duplicate that mirrors the parent object about a plane.

Choose from the following commands:

Edit>Duplicate>Along Line	Duplicates the parent object along a straight line.
Edit>Duplicate>Around Axis	Duplicates the parent object around an axis.
Edit>Duplicate>Mirror	Duplicates a mirror image of the parent object about a plane.

To copy objects to another design, use the **Edit>Copy** and **Edit>Paste** commands.

Note There is currently no method for dissolving the parent/duplicate relationship once a duplicate has been created.

Duplicating Objects Along a Line

To duplicate an object along a straight line, use the **Edit>Duplicate>Along Line** command. The line along which the object is duplicated can be vertical, horizontal, or lie at an angle.

1. Select the object you want to duplicate.
2. On the **Edit** menu, point to **Duplicate**, and then click **Along Line**  .
3. Specify the vector along which the object will be duplicated:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the duplication line.
   b. Select a second point in one of the following ways:

- Click the point.
- Type the coordinates of a point relative to the anchor point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

This point defines the direction and distance from the anchor point to duplicate the object.

The **Duplicate Along Line** dialog box appears.

4. Type the total number of objects, including the original, in the **Total Number** box.
5. Click **OK**.

   The duplicates are placed along the vector you specified.

Duplicating Objects Around an Axis

To duplicate an object around the x-, y-, or z-axis, use the **Edit>Duplicate>Around Axis** command.

1. Select the object you want to duplicate.
2. On the **Edit** menu, point to **Duplicate**, and then click **Around Axis**  .

   The **Duplicate Around Axis** dialog box appears.
3. Select the axis around which you want to duplicate the object: **X**, **Y**, or **Z**.
4. Type the angle between duplicates in the **Angle** box.

   A positive angle causes the object to be pasted in the counter-clockwise direction.

   A negative angle causes the object to be pasted in the clockwise direction.
5. Type the total number of objects, including the original, in the **Total Number** box.
6. Click **OK**.

   The object is duplicated around the axis at the angle you specified.

Duplicating and Mirroring Objects

To duplicate and mirror an object about a plane, use the **Edit>Duplicate>Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to duplicate an object and specify the duplicate's position.

This command is similar to **Edit>Arrange>Mirror**, except that this command duplicates an object, rather than moves it.

1. Select the object you want to mirror.
2. On the **Edit** menu, point to **Duplicate**, and then click **Mirror**  .
3. Select a point on the plane on which you want to mirror the object.

   A line drawn from this point to the mirror plane will be perpendicular to the plane. The distance between the point on mirror plane and point along the normal does not matter; only the vector direction matters
4. Select a normal point on the plane in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX**, **dY**, and **dZ** boxes,

where **d** is the distance from the previously selected point.

A duplicate of the object appears on the plane you specified, oriented according to the normal point you specified.

Related Topics

Mirroring Objects

Scaling Objects

Scale an object's dimensions in one or more directions using the **Edit>Scale** command.

The scale of an object is determined by the distance of each of its vertices from the origin of the model coordinate system. When an object is scaled, the distance of each vertex from the origin is multiplied by the scaling factor, causing the object to be resized and/or moved.

For example, if you specify a scaling factor of 2 in the X direction, each vertex in the model will be moved so that the distance to its origin is doubled. Note that a vertex located at the origin will not move. You can alter an object's proportions by scaling it in one direction.

To scale an object's dimensions in one or more directions:

1. If necessary, set a different working coordinate system to achieve the desired scaling.
2. Select the object to scale.
3. On the **Edit** menu, click **Scale**.

   The **Scale** dialog box appears.
4. Type the scale factor for each axis.
5. Click **OK**.

   The object is scaled about the working coordinate system's origin.

Sweeping Objects

You can sweep a 2D object around an axis, along a vector, or along a path to create a 3D solid object. Objects that can be swept include circles, arcs, rectangles, polylines, or any 2D object created in the **3D Modeler** window. The 2D object need not be orthogonal to the sweep path.

You can also sweep open 1D objects, such as polylines. This results in open 2D sheet objects.

Related Topics

Sweeping Around an Axis

Sweeping Along a Vector

Sweep Along a Path

Sweeping Around an Axis

Sweep a 1D or 2D object around the x-, y-, or z-axis using the **Draw>Sweep>Around Axis** command. Sweeping circles around an axis is a convenient way to create an open coil loop.

Before using this command, keep the following guidelines in mind:

- The object and the axis you are sweeping around must lie in the same plane. For example, if you are sweeping an object around the z-axis, the object must lie in a plane that includes the z-

axis, such as xz or yz.

- The normal of the object's plane faces must be perpendicular to the axis around which you are sweeping.
- The object may not cross the axis around which it is being swept.

To sweep an object around an axis:

1. Select the object you want to sweep.
2. On the **Draw** menu, point to **Sweep**, and then click **Around Axis**.

   The **Sweep Around Axis** dialog box appears.
3. Select the axis you want to sweep the object around: **X**, **Y**, or **Z**.
4. Type the angle to sweep the object through in the **Angle of sweep** box.

   The value must be between **-360** and **360** degrees.
5. Type the draft angle.

   This is the angle to which the object's profile, or shape, is expanded or contracted as it is swept.
6. Select one of the following draft types from the pull-down list. The draft type instructs HFSS how to fill in gaps created by expanding or contracting a profile with a draft angle.

Extended	The edges of the new profile will be extended with straight tangent lines until they intersect. The facetting of the faces will be displayed.
Round	The edges of the new profile will be rounded.
Natural	The edges of the new profile will be extended along their natural curves until they intersect. For example, if the original object had sharp edges, the new profile will have sharp edges.

7. Click **OK**.

   The object is swept around the axis. The new object has the properties of the original object. The **Properties** dialog box appears, enabling you to modify the object's properties.
8. Click **OK**.

Sweeping Along a Vector

Sweep a 1D or 2D object along a vector using the **Draw>Sweep>Along Vector** command.

1. Select the object you want to sweep.
2. On the **Draw** menu, point to **Sweep**, and then click **Along Vector**.
3. Draw the vector you want to sweep the object along:
   a. Select the start point by clicking the point or typing its coordinates in the **X**, **Y**, and **Z** boxes.
   b. Select the endpoint in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the start point in the **dX**, **dY**, and **dZ** boxes,

where **d** is the distance from the previously selected point.

The **Sweep Along Vector** dialog box appears.

4. Type the draft angle.

   This is the angle to which the profile is expanded or contracted as it is swept.

5. Select one of the following draft types from the pull-down list box:

Extended	The new object will have sharp edges like the original object. The facetting of the faces will be displayed.
Round	The new object will have rounded edges.
Natural	The new object will have sharp edges like the original object

   The object is swept along the vector. The new object has the name and color of the original profile. The **Properties** dialog box appears, enabling you to modify the object's properties.

6. Click **OK**.

Sweeping Along a Path

Sweep a 1D or 2D object along a path that is defined by an open or closed polyline using the **Draw>Sweep>Along Path** command.

When you are sweeping an object along a path, keep in mind that one of the path's endpoints must lie in the same plane as the object being swept. The other endpoint must lie in a plane perpendicular to the object being swept.

To sweep an object along a path:

1. Create the polyline you want to use as a path.
2. Select the object you want to sweep, and then select the new polyline.
3. On the **Draw** menu, point to **Sweep**, and then click **Along Path**.

   The **Sweep Along Path** dialog box appears.

4. Type the angle of the twist in the path.

   This is the number of degrees the profile will rotate as it is swept through the complete path.

5. Type the draft angle.

   This is the angle to which the profile is expanded or contracted as it is swept.

6. Select one of the following draft types from the pull-down list box:

Extended	The new object will have sharp edges like the original object. The facetting of the faces will be displayed.
Round	The new object will have rounded edges.
Natural	The new object will have sharp edges like the original object

   The object is swept along the path. The polyline object used as the path is deleted. The new object has the properties of the original object. The **Properties** dialog box appears, enabling you to modify the object's properties.

7. Click **OK**.

Covering Lines

To cover a closed 1D polyline object with a face, use the **3D Modeler>Surface>Cover Lines** command. The polyline object becomes a 2D sheet object.

To convert a polyline object to a sheet object:

1. Select the closed polyline object you want to cover.
2. On the **3D Modeler** menu, point to **Surface**, and then click **Cover Lines**.

   The object is now covered. It is now a 2D sheet object that can be swept to form a 3D solid object.

Note If you want HFSS to automatically cover all closed polyline objects you draw, including circles, ellipses, rectangles, and regular polygons, select the **Automatically cover closed polylines** option in the **3D Modeler Options** dialog box.

Covering Faces

To cover the face of a 2D or 3D object, use the **3D Modeler>Surface>Cover Faces** command.

To cover the faces of objects:

1. Select the faces of the objects you want to cover.
2. On the **3D Modeler** menu, point to **Surface**, and then click **Cover Faces**.

   The object faces are now covered.

Uncovering Faces

Uncover a surface of a 3D object using the **3D Modeler>Surface>Uncover Faces** command. Uncovering the surface of a 3D solid object results in an open 2D sheet object.

To uncover the face of a 3D object:

1. Switch to face selection mode: On the **Edit** menu, point to **Select**, and then click **Faces**.
2. Select a face of the object you want to uncover.
3. On the **3D Modeler** menu, point to **Surface**, and then click **Uncover Faces**.

   The selected face is uncovered, leaving an open face on the object.

Note You can uncover one face of a 3D object at a time. If you select multiple faces, only the first face will be uncovered.

Detaching Faces

The **3D Modeler>Surface>Detach Faces** command enables you to remove the face of a 3D object, resulting in two separate objects.

To detach the face of an object:

1. Switch to face selection mode: On the **Edit** menu, point to **Select**, and then click **Faces**.
2. Select the face of the object you want to detach. You can select multiple faces to detach.

3. On the **3D Modeler** menu, point to **Surface**, and then click **Detach Faces**.

   The selected face is now detached, resulting in two 2D sheet objects.

Creating a Cross-Section

You can take a cross-section of a 3D object to create a new 2D object. This is done using the **3D Modeler>Surface>Section** command.

Use this command to create cross-sections of 3D objects on the xy, yz, or xz plane. The cross-sections are created as 2D closed polyline objects.

To create a cross-section of an object:

1. Make sure the working coordinate system you want to use for the cross-sectioning plane is set.
2. Select the object from which you want to create a cross-section.
3. On the **3D Modeler** menu, point to **Surface**, and then click **Section**.
4. Select the section plane you will use to divide the object: **XY**, **YZ**, or **ZX**.
5. Click **OK**.

   A closed polyline object is created from the object that was sliced by the selected axis. The original, sectioned object is unmodified.

Related Topics

Setting the Working Coordinate System

Connecting Objects

Use the **3D Modeler>Surface>Connect** command to perform the following operations:

- Connect two or more 1D polyline objects. HFSS will modify the first polyline you select to be a 2D sheet object that connects to the second and any subsequently selected polylines. The second and subsequent polylines selected are deleted.
- Connect two or more 2D sheet objects. HFSS will modify the first 2D object you select to be a 3D solid object that connects to the second and any subsequently selected objects. The second and subsequent objects selected are deleted.

To connect objects:

1. Select the objects you want to connect.
2. On the **3D Modeler** menu, point to **Surface**, and then click **Connect**.

   A new object is created that connects the objects you selected. The first object you selected was modified to create the new object and all subsequently selected objects were deleted.

Moving Faces

You can move the faces of a 3D object in a normal direction using the **3D Modeler>Surface>Move Faces** commands. Moving object faces enables you to resize, reshape, or relocate an object.

Related Topics

Moving Faces Along the Normal

Moving Faces Along a Vector

Offsetting Objects

Moving Faces Along the Normal

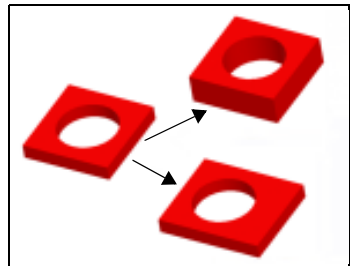
To move a 3D object's face a specified distance in a direction normal to its original plane, use the **3D Modeler>Surface>Move Faces>Along Normal** command. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners, as shown below.

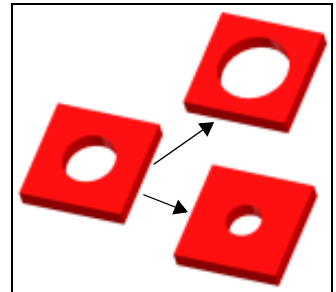
To move an object face in a normal direction:

1. Click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.
3. Click **3D Modeler>Surface>Move Faces**>**Along Normal**.
4. The **Move Faces Along Normal** dialog box appears.
5. Type the distance you want to move the object face from its origin.
6. Click **OK**.

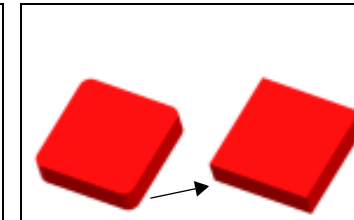
   The face will be moved the distance you specified.



Extruding Faces



Resizing Holes



Removing Rounded Corners

To move every face of an object normal to its surface, use the **Edit>Arrange>Offset** command.

Moving Faces Along a Vector

To move the faces of a 3D object a specified distance along a vector use the **3D Modeler>Surface>Move Faces**>Along Vector command. Each selected face is moved along the vector, normal to its original plane. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for relocating holes in an object, as shown below.

To move an object face along a vector:

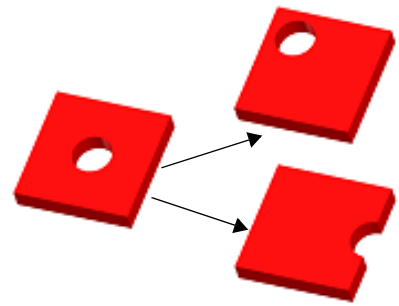
1. Click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

3. Click **3D Modeler>Surface>Move Faces**>**Along Vector**.
4. Specify the vector along which the face will be moved:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the vector.
   b. Select a second point in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the anchor point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

      This point defines the direction and distance from the anchor point to move the face.

   The face is moved along the vector you specified.



Relocating Holes

To move every face of an object normal to its surface, use the **Edit>Arrange>Offset** command.

Uniting Objects

To join two or more objects into one object, use the **3D Modeler>Boolean>Unite** command. The new object has the name, color, boundary, and material assignment of the first object selected. The objects are united at the point of intersection.

To unite two or more objects:

1. Select the objects you want to join.
2. On the **3D Modeler** menu, point to **Boolean**, and then click **Unite**  .

The objects are united.

Note By default, the objects being joined to the first object selected are *not* preserved for later use. If you want to keep a copy of the objects being joined to the first object selected, do one of the following:

- Copy the objects, and then paste them back into the design after uniting them.
- Select **Clone before unite** in the **3D Modeler Options** dialog box. This option instructs HFSS to always keep a copy of the original objects being joined.

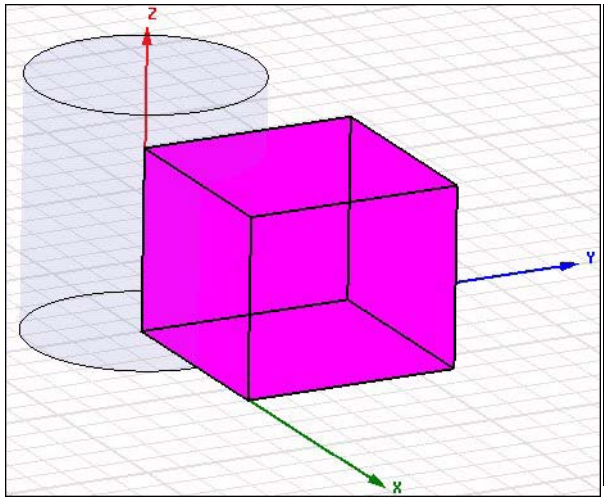
Subtracting Objects

1. Select the object from which you want to subtract other objects.
2. Hold down the **Ctrl** key and select the objects you want to subtract.
3. On the **3D Modeler** menu, point to **Boolean**, and then click **Subtract**  .

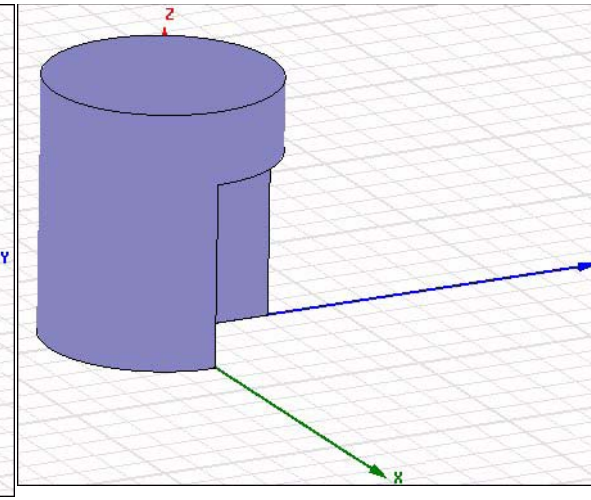
   The **Subtract** dialog box appears.

   Objects listed in the **Tool Parts** list will be subtracted from the object or objects listed in the **Blank Parts** list.
4. Optionally, select an object name in either list and use the left and right arrow buttons to move the object name to the opposite list.
   - Alternatively, type the name of object you want to subtract in the empty text box below the **Tool Parts** list, and then type the name of the object from which you want to subtract it in the empty text box below the **Blank Parts** list.
5. Optionally, select **Clone tool objects before subtract**. This instructs HFSS to always keep a copy of the original objects being subtracted.
6. Click **OK**.

The new object retains the name, color, and material of the first object selected.



An intersecting box and cylinder.



A box subtracted from a cylinder.
The cylinder was selected first.

Note By default, the objects being subtracted from the first object selected are *not* preserved for later use. If you want to keep a copy of the objects being subtracted from the first object selected, do one of the following:

- Copy the objects, and then paste them back into the design after subtracting them.
- Select **Clone before subtract** in the **3D Modeler Options** dialog box. This option instructs HFSS to always keep a copy of the original objects being subtracted.

Creating Objects from Intersections

To create a new object from the intersection of two or more objects, use the **3D Modeler>Boolean>Intersect** command.

To create an object from an intersection:

1. Select the objects from which you want to take the intersection.

Warning If the objects you selected do not overlap, the result is a null object and both objects vanish.

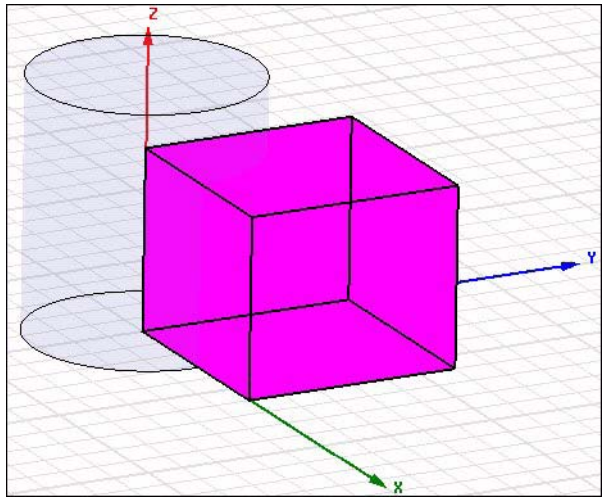
2. On the **3D Modeler** menu, point to **Boolean**, and then click **Intersect**  .

   The original objects vanish, leaving only the new object that was formed from their intersec-

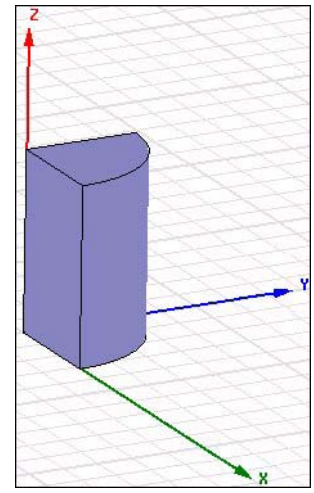
tion.

Note By default, the original intersecting objects are *not* preserved for later use. If you want to keep a copy of the objects that intersect the first object selected, do one of the following:

- Copy the objects, and then paste them back into the design after creating the new object from the intersection.
- Select **Clone before intersect** in the **3D Modeler Options** dialog box. This option instructs HFSS to always keep a copy of the original objects that intersect the first object selected.



An intersecting box and cylinder.



Object formed from the intersection of the box and cylinder.

Creating an Object from a Face

The **3D Modeler>Surface>Create Object from Face** command copies a selected face, resulting in a new 2D sheet object.

To create a new object from a face:

1. Click **Select Faces** on the shortcut menu.
2. Select the object face you want to copy. You can select multiple faces and each will become a new object.
3. On the **3D Modeler** menu, point to **Surface**, and then click **Create Object from Face**.

The face is copied, resulting in a new 2D sheet object.

Hint This command is useful for assigning a boundary to the intersection of two faces. To do this, first select the faces, and then create an object from them using the procedure above. Next, make sure the **Clone before intersect** option is clear in the **3D Modeler Options** window, and then use the **3D Modeler>Boolean>Intersect** command to modify the object so that it includes only the intersection of the two faces. Then assign the boundary to the new object.

Creating an Object from an Edge

Use this command to create wire bodies from edges.

1. Enter edge selection mode and select an edge.

   The edge is highlighted.

2. Click **3D Modeler>Create Object From Edge**.

   This creates a new line object from the edge. The resulting object appears in the history tree as a line object named ObjectfromEdge*N*.

Splitting Objects

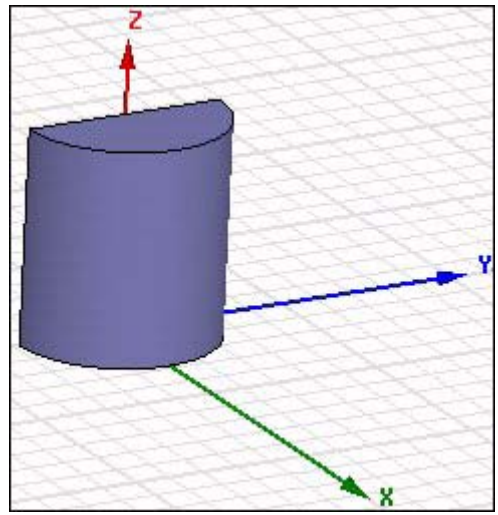
To an object or objects that lie on the xy, yz, or xz plane, use the **3D Modeler>Boolean>Split** command.

1. Select the object you want to split. You can select more than one.
2. On the **3D Modeler** menu, point to **Boolean**, and then click **Split**  .

   The **Split** dialog box appears.

3. Select the **Split plane** that you will use to split the objects.
4. Select the object fragments you want to keep:
   - those on the positive side of the selected plane,
   - those on the negative side of the plane, or
   - all pieces on both sides of the plane.
5. Select the Split Option you want to use:
   - Split entire selection (the default) - if you have multiple objects selected, all objects are split regardless of whether they cross the split plane.
   - Split objects crossing split plane.- if you have multiple objects selected, only those objects that cross the split plane are split.
6. Click **OK**.

The objects are divided as specifed. Any objects that become null are deleted.



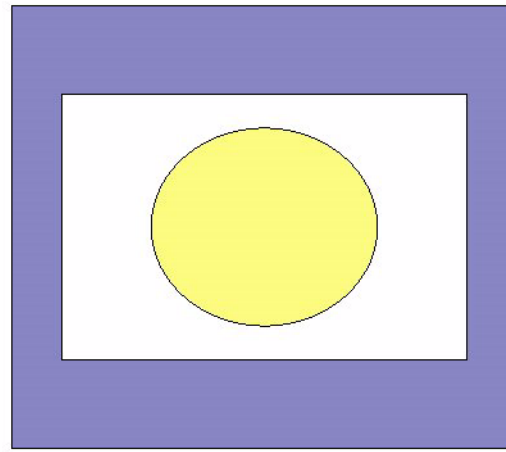
A cylinder split along the positive side of the yz plane.

Separating Bodies

To separate an object with multiple lumps into individual bodies:

1. Select the object you want to separate.
2. On the **3D Modeler** menu, point to **Boolean**, and then click **Separate Bodies**.

The object is separated.



This figure shows two separate bodies, each with one lump, that were created from one object.

Converting Polyline Segments

A polyline is a single object that includes any combination of straight line, arc line, or spline segments. You can convert a polyline segment from one type to another. The following conversions are supported:

- Straight line segments to arc line or spline segments.
- Arc line segments to straight line or spline segments.
- Spline segments to straight line segments.

To convert polyline segments:

1. In the history tree, locate the polyline that contains the segment you want to convert. Expand this part of the history tree.
2. In the history tree, right-click the polyline segment operation you want to change, and then click **Properties**.
3. In the **Properties** dialog box, click in the **Value** text box of the **Segment Type** row.
4. Select the desired polyline segment type from the pull-down list.

   The polyline segment you selected is changed to the new type.

Note Converting an arc line or spline segment to a straight line segment results in two straight line segments; one segment is created between the start point and midpoint and one segment is created between the midpoint and endpoint.

Rounding the Edge of Objects (Fillet Command)

To round the edge of an object:

1. Select the edge you want to change.

   This highlights the edge and enables the **Fillet** command.

   Click **3D Model>Fillet** or click the fillet icon  .

   The **Fillet Properties** dialog is displayed.

2. Enter a value for the **Fillet Radius** in the text field and select units from the dropdown menu. The default is millimeters.
3. Enter a value for the setback distance.

   The setback distance controls the shape of the vertex. It is the distance of the cross curve from the vertex at the end of the edge. If it is less than the fillet radius it has no effect. You will get an error if it is greater then the length of the edge.

4. Click **OK** to apply the change to the edge.

   The dialog closes and the object is rounded by the radius value relative to the edge you selected.

Flattening the Edge of Objects (Chamfer Command)

To flatten the edge of an object.

1. Select the edge you want to change.

   This highlights the edge and enables the **Chamfer** command.

   Click **3D Model>Chamfer** or click the chamfer icon  **.**

   The **Chamfer Properties** dialog is displayed.

2. Enter a value for the **Chamfer Value** in the text field and select units from the dropdown menu. The default is millimeters.
3. Click **OK** to apply the change to the edge.

   The dialog closes and the object is rounded by the radius value relative to the edge you selected.

Purge History

Each object is a sequence of modeler-based operations. The history for each object is shown under its name in the model tree. You can use the **Purge History** command to remove the history of operations while not affecting the geometry itself. This is useful when you wish to perform healing operations on the object.

1. Select the object.
2. Select **3D Modeler>Purge History**.

   The history for the model is purged, and the context for the **Undo** and **Redo** commands is updated.

Related Topics

Working with the History Tree

Generate History

Generate History

If a polyline object is imported or history was previously purged, you can click on the polyline object and select **Generate History** to reproduce the individual line segments used to create the polyline in the model history tree.

Related Topics

Purge History

Draw Polyline

Selecting Items in the 3D Modeler Window

To modify or learn more about an item's properties, you must first select it. All commands you choose while an item is selected are performed on or in reference to the selected item.

What do you want to select?

- Objects.
- Faces.
- Edges.
- Vertices.
- Coordinates in the drawing space.

Selecting Objects

By default, HFSS is in *object selection mode*. Simply click an object in the view window or an object name in the history tree and it will be selected. All other objects become relatively transparent.

When the mouse hovers over an object in the view window, that object is highlighted, which indicates that it will be selected when you click. Selected objects become the color specified under the **Display** tab of the **3D Modeler Options** dialog box.

If HFSS is not currently in object selection mode, you can switch to it using one of the following methods:

- Press the shortcut key **O**.
- Right-click in the view window, and then click **Select Objects**.
- On the **Edit** menu, point to **Select**, and then click **Objects**.
- Select **Object** from the pull-down list in the **3D Modeler Selection** toolbar.

Related Topics

Selecting Multiple Objects

Selecting Objects by Name

Selecting All Faces of an Object

Creating an Object List

Selecting Multiple Objects

1. Make sure that HFSS is in object selection mode by pressing the shortcut key **O**.
2. Select multiple objects in one of the following ways:
   - Hold down **CTRL** and click the objects in the view window that you want to select.
   - Hold down **CTRL** and click the object names in the history tree that you want to select.
   - Click **Edit>Select All** to select all objects that were drawn in the active view window, including objects that are not currently visible.
   - Press **CTRL**+**A** or click **Edit>Select All Visible** to select all objects that are visible in the

active view window.

Selected objects become the color that is specified for selected objects under the **Display** tab of the **3D Modeler Options** dialog box. Use **Tools>Options.3D Modeler Options** to display the dialog and set the default color. By default, the selected objects are opaque and all other objects become relatively transparent.The settings for the relative opacity and transparency of selected and non-selected objects appear in the **3D UI Options** dialog box. Use **View>Options** to display the **3D UI Options** dialog.

Selecting Objects by Name

1. Make sure that HFSS is in object selection mode by pressing the shortcut key **O**.
2. On the **Edit** menu, point to **Select**, and then click **By Name**  or in the toolbar, select Object from the drop-down menu to the right of the  icon, and click the icon.

   The **Select Object** dialog box appears.
3. In the **Name** list, click the name of the object you want to select. Use the **Ctrl** key to select more than one.
   - Alternatively, type the name of an object you want to select in the empty text box.
4. Click **OK**.

   The object is selected.

Setting the Default Color and Transparency of Selected Objects

To set the color of objects when they are selected:

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler** options.
2. Click the **Display** tab.
3. Click **Select** on the **Default color** pull-down list.
4. Click the color button beside the **Default color** pull-down list.

   The **Color** palette appears.
5. Select a color from the **Color** palette, and then click **OK**.

   Any objects you select after this point will temporarily become the default color you selected.

By default, HFSS shows selected objects as nearly opaque and shows non-selected objects as nearly transparent. This features helps you distinguish between selected and non-selected objects.

To set the transparency of selected and non-selected objects:

1. On the **View** menu, select **Options**.

   The **3D UI Options** dialog appears. The **When there is a selection** region contains check-boxes for setting the transparency for selected and non-selected objects.

   Click the checkbox for the value you want to change.

   This enables the value field. The default transparency for selected objects is 0.1, which makes them almost opaque. The default transparency for non-selected objects is 0.9, which makes them highly transparent.
2. Enter a new value, and click **OK** to apply the new transparency values.

Setting the Default Color of Highlighted Objects

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler** options.
2. Click the **Display** tab.
3. Click **Highlight** on the **Default color** pull-down list.
4. Click the color button beside the **Default color** pull-down list.

   The **Color** palette appears.
5. Select a color from the **Color** palette, and then click **OK**.

   After this point, the outlines of objects you hover over with the mouse will temporarily become the default color you selected.

Creating an Object List

Create an object list when you want to define a list of objects. Creating an object list is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a list can still be treated as separate objects. The same object can be included in several different lists.

To create an object list:

1. Make sure that HFSS is in object selection mode by pressing the shortcut key **O**.
2. Select the objects you want to include in the list.
3. Click **3D Modeler>List>Create>Object List**.

   The object list is created. It is listed in the history tree under **Lists**.

The object list will be treated as one volume when you are plotting and performing fields calculations. It will be listed in the **Geometry** window of the Fields Calculator, when you select **Volume**.

Example: To plot the E-field on a surface formed by the intersection of the xy-plane and several objects, first define a list of these objects. Then, when plotting fields, select the object list name from the **Geometry** window of the Fields Calculator. Fields will be plotted only at the intersection of the plane and the objects in the list.

Reassigning Objects to Another Object List

You can assign objects after you have created object lists. Creating an object list is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a list can still be treated as separate objects. The same object can be included in several different lists.

To reassign objects in an object list:

1. Make sure that HFSS is in object selection mode by pressing the shortcut key **O**.
2. Select the objects you want to reassign.
3. Click **3D Modeler>List>Reassign**

   A dialog with the existing object lists is displayed. (They appear in the history tree under **Lists**.
4. Select the list to which you want to assign the select object and click OK.

   The object is reassigned to the selected list.

The object list will be treated as one volume when you are plotting and performing fields calculations. It will be listed in the **Geometry** window of the Fields Calculator, when you select **Volume**.

Related Topics

Creating an Object List

Selecting Faces

If HFSS is in face selection mode, click an object face in the view window to select it. To select multiple faces, hold the **CTRL** key as you click the faces. You also have the option to create face lists, which define a list of object faces.

When the mouse hovers over a face in the view window, that face is highlighted, which indicates that it will be selected when you click. Selected faces become the color specified under the **Display** tab of the **3D Modeler Options** dialog box. All other objects and faces become relatively transparent.

Switch to face selection mode using one of the following methods:

- Press the shortcut key **F**.
- Right-click in the view window, and then click **Select Faces**.
- On the **Edit** menu, point to **Select**, and then click **Faces**.
- Select **Face** from the pull-down list to the right of the select objects icon  in the **3D Modeler Selection** toolbar.

Related Topics

Selecting All Faces of an Object

Selecting the Face or Object Behind

Selecting Faces by Name

Selecting Faces by Plane

Creating a Face List

Face Selection Toolbar Icons

Selecting All Faces of an Object

1. Optionally, select the object (or objects, faces, edges or vertices) with the faces you want to select.
2. Switch to face selection mode by pressing the shortcut key **F**.
3. If an object is not selected, click a face on the object of interest.
4. On the **Edit** menu, point to **Select**, and then click **All Object Faces**.
   - Alternatively, right-click in the view window, and then click **All Object Faces** on the shortcut menu.
   - As another alternative, select use the face selection toolbar icons.

   All the faces of the object are selected. If you selected multiple objects, all faces of those objects are selected.

Related Topics

Selecting Faces

Selecting the Face or Object Behind

Creating a Face List

Face Selection Toolbar Icons

Selecting Faces by Name

1. Make sure that HFSS is in face selection mode by pressing the shortcut key **F**.
2. On the **Edit** menu, point to **Select**, and then click **By Name**  the toolbar, select Face from the dropdown menu to the right of the object selection  icon and click the icon.

   The **Select Face** dialog box appears.
3. In the **Object name** list, click the name of the object with the face you want to select.

   The object's faces are listed in the **Face ID** column.
4. Click the face you want to select in the **Face ID** column. You can select more than one.

   The face is selected in the view window.
5. Click **OK**.

Related Topics

Selecting Faces

Creating a Face List

Selecting Faces by Plane

To select a face that is aligned with a global plane, use one of the following two methods.

1. Make sure that HFSS is in face selection mode by pressing the shortcut key **F**.
2. In the History Tree, expand the **Planes** icon. Left-click on a plane (Global:XY, Global:YZ, or Global:XZ) to display the selected global plane.
3. On the **Edit** menu, point to **Select**, and then click **Faces on Plane**.

   The **selected faces** are highlighted.

Alternative method:

1. In the History Tree, expand the **Planes** icon.
2. **Right-click on a plane** (Global:XY, Global:YZ, or Global:XZ) to select the global plane and display a pull-down menu.
3. On the **pulldown** menu, click **Faces on Plane**.

   The **selected faces** are highlighted.

Related Topics

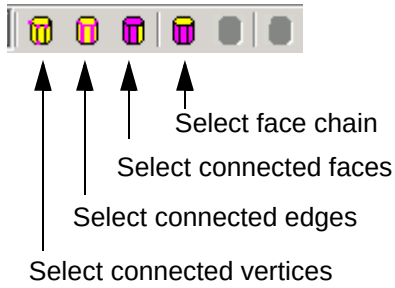
Selecting Faces

Creating a Face List

Face Selection Toolbar Icons

While working on analyzing complex objects, it is sometimes useful to examine faces, edges and vertices. In particular it is useful to find the connected faces for a face or edge or vertex, connected edges for a face/edge/vertex and connected vertices for a face/edge/vertex. The additional selection modes are available under **Edit->Select** and via the toolbar icons.

Selecting an object face enables the face selection icons in the toolbar.



You can use these icons to modify the selection:

- Select face chain selects faces that touch each other. It allows faces that are part of a “protrusion” to be selected.
- Select connected faces selects faces connected to the current selection.
- Select connected edges selects the edges of the selected face or faces.
- Select vertices selects the vertices of the selected face or faces.

Creating a Face List

Create a face list when you want to define a list of object faces. Creating a face list is a convenient way to identify and select a specific set of surfaces for a field plot or calculation. The same face can be included in several different lists.

To create a face list:

1. Make sure that HFSS is in face selection mode by pressing the shortcut key **F**.
2. Select the object faces you want to include in the face list.
3. Click **3D Modeler>List>Create>Face List**.

   The face list is created. It is listed in the history tree under **Lists**.

The face list will be treated as one selection of surfaces when you are plotting and performing fields calculations. The face list will be listed in the **Geometry** window of the Fields Calculator, when you select **Surface**.

Selecting Edges

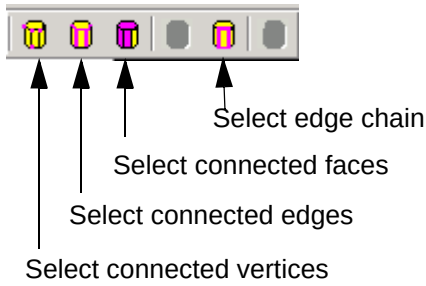
If HFSS is in edge selection mode, simply click an object’s edge in the view window and it will be selected. To select multiple edges, hold the **CTRL** key as you click the edges.

When the mouse hovers over an edge in the view window, that edge is highlighted, which indicates that it will be selected when you click. Selected edges become the color specified under the **Display** tab of the **3D Modeler Options** dialog box. All other objects become relatively transparent.

Switch to edge selection mode using one of the following methods:

- From the menu bar, click **Edit**>**Select**>**Edges**, or click **Edit>Select** and type the E hotkey for Edge select mode.
- Select **Edge** from the pull-down list in the **3D Modeler Selection** toolbar.

Selecting an edge enables the following toolbar icons.



You can use these icons to modify the current selection.

- Select edge chain selects the edges that touch the selected edge.
- Select connected faces selects faces touching to the current selection.
- Select connected edges selects the edges that touch the current selection.
- Select vertices selects the vertices of the selected edge or edges.

Selecting Vertices

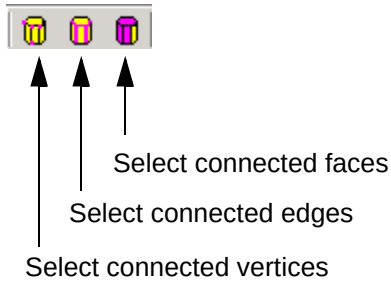
If HFSS is in vertex selection mode, simply click an object's vertex in the view window and it will be selected. To select multiple vertices, hold the **CTRL** key as you click the vertices.

When the mouse hovers over a vertex in the view window, that vertex is highlighted, which indicates that it will be selected when you click. Selected vertices become the color specified under the **Display** tab of the **3D Modeler Options** dialog box. All other objects become relatively transparent.

Switch to vertex selection mode using one of the following methods:

- On the **Edit** menu, point to **Select**, and then click **Vertices**.
- Select **Vertex** from the pull-down list in the **3D Modeler Selection** toolbar.

Selecting a vertex enables the following selection icons.



You can use these icons to modify the current selection.

- Select connected faces selects faces touching to the current selection.
- Select connected edges selects the edges that touch the current selection.
- Select vertices selects the vertices of edges that touch the current selection.

Clearing a Selection

To clear an object, face, edge, or vertex selection, do one of the following:

- Click the view window at a point where an object does not exist.
- To clear an object selection, click a point away from the object name in the history tree.
- On the **Edit** menu, click **Deselect All**.
- Press **Shift+Ctrl+A**.

  The items are no longer selected.

Selecting the Face or Object Behind

To select the face or object behind a selected face or object, do one of the following:

- On the **Edit** menu, point to **Select**, and then click **Next Behind**.
- Right-click in the view window and click **Next Behind**.
- Press the shortcut key **B**.
- Press **Ctrl+B**.

This option is useful when you are trying to select a face or object that is in the interior of a model, or when you do not want to change the model view to select a face or object.

Selecting Cartesian Coordinates

To select a point using Cartesian coordinates, type the point's distance from the origin in the x, y, and z directions in the **X**, **Y**, and **Z** text boxes, respectively. When selecting a second point, specify its distance from the previously selected point in the x, y, and z directions in the **dX**, **dY**, and **dZ** text boxes, respectively.

1. After clicking the desired drawing command, select **Cartesian** from the pull-down list in the

status bar.

2. Type the point's x-, y-, and z-coordinates in the **X**, **Y**, and **Z** text boxes.

Hint Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

- Alternatively, click the point in the view window.

3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the x, y, and z directions in the **dX**, **dY**, and **dZ** text boxes, respectively.

Related Topics

Selecting Cylindrical Coordinates

Selecting Spherical Coordinates

Selecting Cylindrical Coordinates

To select a point using cylindrical coordinates, specify the point's radius, measured from the origin, in the **R** text box, the angle from the x-axis in the **Theta** text box, and the distance from the origin in the z direction in the **Z** text box. When selecting a second point, specify its distance from the previously selected point in the in **dR**, **dTheta**, and **dZ** text boxes.

1. After clicking the desired drawing command, select **Cylindrical** from the pull-down list in the status bar.
2. Type the point's r-, theta-, and z-coordinates in the **R**, **Theta**, and **Z** boxes.

Hint Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

- Alternatively, click the point in the view window.

3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dR**, **dTheta**, and **dZ** text boxes.

Related Topics

Selecting Cartesian Coordinates

Selecting Spherical Coordinates

Selecting Spherical Coordinates

To select a point in spherical coordinates, specify the point's radius, measured from the origin, in the **R** text box, the angle from the x-axis in the **Theta** text box, and the angle from the origin in the z direction in the **Phi** text box. When selecting a second point, specify its distance from the previously selected point in the in **dR**, **dTheta**, and **dPhi** text boxes.

1. After clicking the desired drawing command, select **Spherical** from the pull-down list in the

status bar.

2. Type the point's r-, theta-, and phi-coordinates in the **R**, **Theta**, and **Phi** text boxes.

Hint Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the view window.

3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dR**, **dTheta**, and **dPhi** text boxes.

Related Topics

Selecting Cartesian Coordinates

Selecting Cylindrical Coordinates

Selecting Absolute Coordinates

When entering a point's coordinates, you can specify them in *absolute* or *relative* coordinates. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0). This is the default setting for the first point you select after clicking a drawing command. Relative coordinates are relative to the reference point, or the previously selected point.

To enter a point's absolute coordinates:

1. Click the desired drawing command.
2. Select **Absolute** from the **Absolute/Relative** pull-down list in the status bar.
3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the status bar.

Note When drawing objects other than polylines and helices, by default, the second point you select is relative to the first point; **Relative** is automatically selected in the **Absolute/Relative** pull-down list in the status bar. Be sure to select **Absolute** from the **Absolute/Relative** pull-down list in the status bar if you want the second point to be relative to the working coordinate system.

Related Topics

Selecting Relative Coordinates

Selecting Relative Coordinates

When entering a point's coordinates, you can specify them in *absolute* or *relative* coordinates. Relative coordinates are relative to the reference point, or the previously selected point. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0).

To enter a point's relative coordinates:

1. Click the desired drawing command.
2. Select **Relative** from the **Absolute/Relative** pull-down list in the status bar.
3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the status bar.

Related Topics

Selecting Absolute Coordinates

Choosing the Movement Mode

When drawing objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis:

To change the reference point, move the cursor to the desired point and press **Ctrl+Enter**.

You can move the cursor to one of the following points:

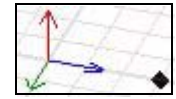
- In the same plane as the reference point (in-plane movement mode).
- Perpendicular to the reference point (out-of-plane movement mode).
- If an object is present to snap to a point in 3D space (3D movement mode).
- Along the x-axis.
- Along the y-axis.
- Along the z-axis.

Moving the Cursor In Plane

To move the cursor to a point *on the same plane* as the reference point

1. Click the desired drawing command.
2. Do one of the following:
   - On the **3D Model** menu, point to **Movement Mode**, and then click **In Plane**.
   - Click **In Plane** in the movement mode pull-down list in the 3D Modeler Draw toolbar.

The next point you select will be on the same plane as the reference point.



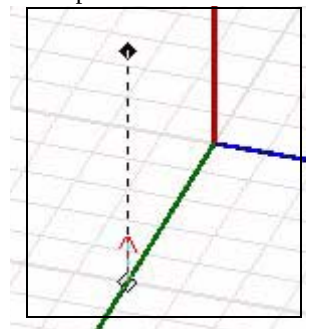
The cursor's location, displayed with a black diamond that indicates it has snapped to the grid, is on the same plane as the reference point.

Moving the Cursor Out of Plane

To move the cursor to a point *perpendicular* to the reference point:

- After clicking the desired drawing command, on the **3D Model** menu, point to **Movement Mode**, and then click **Out of Plane**.

  A dashed line is displayed between the reference point and the cursor's location, which is now perpendicular to the reference point.

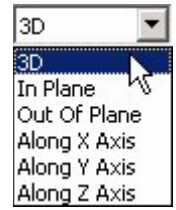


The cursor's location, displayed with a black diamond that indicates it has snapped to a grid point, is perpendicular to the reference point.

Moving the Cursor in 3D Space

To move the cursor to a point in 3D space relative to the reference point:

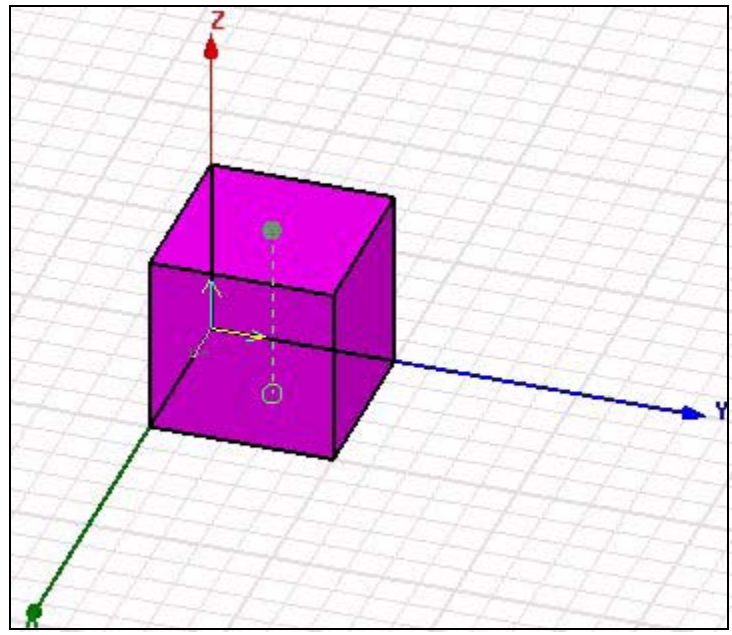
1. Click the desired drawing command.
2. Do one of the following:
   - On the **3D Model** menu, point to **Movement Mode**, and then click **3D**.
   - Click **3D** in the movement mode pull-down list in the 3D Modeler Draw toolbar.



If an object is within snapping range, the cursor will snap to the nearest point in 3D space occupied by the object.

If an object is not within snapping range, 3D movement mode is identical to the in-plane

movement mode.

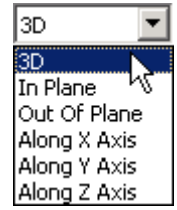


The cursor's location, displayed by a circle that indicates it has snapped to a face center, is (0.5, 0.5, 1.0), a point in 3D space relative to the reference point.

Moving the Cursor Along the X-Axis

To move the cursor to a point away from the reference point in the x direction:

1. Click the desired drawing command.
2. Do one of the following:
   - On the **3D Model** menu, point to **Movement Mode**, and then click **Along X Axis**.
   - Hold the shortcut key **X**.
   - Click **Along X Axis** in the movement mode pull-down list in the 3D Modeler Draw toolbar:



The next point you select will be on the same plane as the reference point in the positive or negative x direction.

Moving the Cursor Along the Y-Axis

To move the cursor to a point away from the reference point in the y direction:

1. Click the desired drawing command.
2. Do one of the following:
   - On the **3D Model** menu, point to **Movement Mode**, and then click **Along Y Axis**.
   - Hold the shortcut key **Y**.
   - Click **Along Y Axis** in the movement mode pull-down list in the 3D Modeler Draw toolbar:

The next point you select will be on the same plane as the reference point in the positive or negative y direction.

Moving the Cursor Along the Z-Axis

To move the cursor to a point away from the reference point in the z direction:

1. Click the desired drawing command.
2. Do one of the following:
   - On the **3D Model** menu, point to **Movement Mode**, and then click **Along Z Axis**.
   - Hold the shortcut key **Z**.
   - Click **Along Z Axis** in the movement mode pull-down list in the 3D Modeler Draw toolbar:

The next point you select will be on the same plane as the reference point in the positive or negative z direction.

Choosing Snap Settings

By default, the selection point and graphical objects are set to “snap to”, or adhere to, a point on the grid when the cursor hovers over it. The coordinates of this point are used, rather than the exact location of the mouse. The cursor changes to the shape of the snap mode when it is being snapped.

To change the snap settings for the active design, you can use either the **3D Modeler** menu or the toolbar icons  :

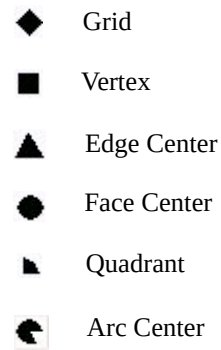
1. On the **3D Modeler** menu, click **Snap Mode** or click the toolbar icons.

   If you select the menu command, the **Snap Mode** window appears.

2. Specify the snap mode settings you want.

- If you want the cursor to snap to a point on the grid, select **Grid** or the icon  .
- To snap to a vertex, select **Vertex** or the icon  .
- To snap to the center point of an edge, select **Edge Center** or the icon  . The center point may be on a 1D, 2D, or 3D object edge.
- To snap to the center of an object face, select **Face Center** or the icon  .
- To snap to the nearest quarter point on an edge, select **Quadrant** or the icon  .
- To snap to the center of an arc, select **Arc Center** or the icon  .

  When the cursor snaps to a point, it will change to one of the following snap mode shapes:



Note By default, the mouse is set to snap to the grid, a vertex, an edge center, a face center, and the nearest quadrant. To modify the default snap settings for the active design and all new designs, modify the selections under the **Drawing** tab in the **3D Modeler Options** dialog box.

Related Topics

Snap Setting Guidelines

Snap Setting Guidelines

In general, select at least one of the snap options in the **Snap Mode** window. If none of these options are selected, the software is in “free mode” and selects whatever point you click, regardless of its coordinates. This can cause problems when you are trying to create closed objects. Although the point you select may appear to be the vertex point of an open object, you may not have actually clicked the exact coordinates of the point.

Measure Modes for Objects

The Measure mode lets you measure the position, length, area, and volume of objects. You can access the Measure mode in two ways:

- Select **3D Modeler>Measure.**
- Right-click and select **Measure** from the short-cut menu.

After selecting **Measure**, a cascading menu appears for **Position**, **Length**, **Area**, and **Volume**. To enter Measure Mode, select the measurement that you want to make. The **Measure Information** dialog box appears, displaying information for the selected entity. With the **Measure Information** dialog box in Measure Position mode, you can also measure the distance between two points. The measure data is preserved until you use the **Clear** button, even if you close the dialog.

Related Topics

Measuring Position

Measuring the Distance Between Two Points

Measuring Length

Measuring Area

Measuring Volume

Setting Coordinate Systems

Modifying the Coordinate System Axes View

Choosing Snap Settings

Measuring Position

To measure an object's position:

1. Select **3D Modeler>Measure>Position**.

   The cursor leads a diamond-shape selection marker that snaps from grid point to grid point on the z-plane of the design area. As you drag the selection marker over an object, it changes shape to provide information about the object at the corresponding coordinate. :

   Grid point

   Vertex

   Edge Center

   Face Center

   Quadrant

2. Clicking on a point lists the entity (face, vertex, or point) in the entity window, and highlights the coordinates of the selected point.

When you click on a point outside the design object, diamond-shaped selection marker changes to depict a reference point displayed as a mini x-y-z-axis:

If you click a point within the object, the selection maker draws a dashed line from the x-y-z coordinate marker along the z-coordinate distance from the x-plane that connects to a diamond marker on the x-plane. This provides a visual reference for selecting in 3-dimensional space. For more information about reference point behavior, see *Choosing a Movement Mode*.

The **Measure Information** dialog box contains an **Entity** list that shows the selected points, vertices, and faces, and a **Measure** list that gives coordinates for the highlighted point, vertex. or face.

When you select additional positions, these are added to the **Entity** list in the **Measure Information** dialog. Each position is described as a Point*n*, Edge center*n*, Vertexn, etc., depending on its location, where *n* increments during the session. The **Measure** list shows the position of each point selected and additional distance information when you have two points selected:

- The Position and identification of the two most recent points selected, with both highlighted. If you select more than two points at once, the Measure list shows their position and ID.
- The linear distance between the two most recently selected points. If you have selected multiple points, you can use Ctrl-click to highlight any two of them to obtain distance information.
  - The X distance.
  - The Y distance.
  - The Z distance.

As you work with the Measure Information dialog:

- Click the **Clear** button to remove a highlighted point, vertex, or face.
- Click the **Clear All** button to remove all points, vertices, or faces from the **Measure Information** dialog box.

To close the dialog box and exit **Measure** mode, click the **Close** button.

Related Topics

Measuring Length

Measuring the Distance Between Two Points

Measuring Area

Measuring Volume

Measuring Length

To measure the length of lines or edges:

1. Select **3D Modeler>Measure>Length** to enter the Measure mode for length.

   The **Measure Information** dialog box appears, showing the **Entity** list and the **Measure** list.
2. Move the cursor across a design object to highlight each line that can be selected.

Selecting a highlighted line changes that line's color and makes the rest of the object transparent. At the same time, the **Measure Information** dialog box displays the Edge_number in the **Entity** list, and the length of the corresponding edge object in the current units.

3. If you select additional edges, their numbers are added to the display in the **Entity** list, and the highlighted edge is depicted in the **Measure** list.

To remove a selected length from the entity list, click the **Clear** button. To remove all selected lengths, click the **Clear All** button.To close the dialog box and exit Measure mode, click the **Close** button.

Related Topics

Measuring Position

Measuring the Distance Between Two Points

Measuring Area

Measuring Volume

Measuring the Distance Between Two Points

To measure the distance between two points:

1. Select **3D Modeler>Measure>Position** to enter Measure Position mode.
2. Click the first point.

   The selected point is highlighted, and its position is shown in the **Measure** list.
3. Click the second point.

   Both points are highlighted in the **Entity** list, and the following values are listed in the **Measure** list:

   - Total distance.
   - X distance
   - Y Distance.
   - Z Distance.
4. In the measure dialog, using the **Ctrl** key, select any two points between which you would like to measure distance. You can select any combination of two points in the measure dialog to get the distance.

Related Topics

Measuring Position

Measuring Length

Measuring Area

Measuring Volume

Measuring Area

Select **3D Modeler>Measure>Area** to enter the measure mode. In the Measure Area mode, the **Measure Information** dialog box appears. To measure the area of a face:

1. To highlight a face to measure, move the cursor over the design.

   A face that can be currently selected is highlighted.

2. Click the highlighted face to obtain a measurement.

   The selected face is highlighted in a different color and the rest of the design becomes transparent.

In the **Measure Information** dialog box, the **Entity** list shows the face number, and the **Measure** list shows the area for the selected face. Select other faces to add their numbers to the **Entity** list. To remove a selected entity and measure, click the **Clear** button. To remove all faces from the **Entity** list, click the **Clear All** button. To close the **Measure Information** dialog box and exit the Measure Area mode, click the **Close** button.

Related Topics

Measuring Position

Measuring Length

Measuring Volume

Measuring Volume

To measure volume:

1. Select **3D Modeler>Measure>Volume** to enter the measure mode.

   The **Measure Information** dialog box appears.

2. To highlight a component whose volume to measure, move the cursor over the design.

   A component that can be currently selected is highlighted.

3. Click on the highlighted component to obtain a measurement.

   In the **Measure Information** dialog box, the **Entity** list shows the object name, and the **Measure** list shows the volume for the selected object.

Select other objects to add their numbers to the **Entity** list. To remove a selected object and measure, click the **Clear** button. To remove all objects from the **Entity** list, click the **Clear All** button. To close the **Measure Information** dialog box and exit the Measure Area mode, click the **Close** button.

Related Topics

Measuring Position

Measuring the Distance Between Two Points

Measuring Length

Measuring Area

Setting Coordinate Systems

HFSS has three types of coordinate systems that enable you to easily orient new objects: a *global* coordinate system, a *relative* coordinate system, and a *face* coordinate system. Every coordinate system (CS) has an x-axis that lies at a right angle to a y-axis, and a z-axis that is perpendicular to the xy plane. The origin (0,0,0) of every CS is located at the intersection of the x-, y-, and z-axes.

The *global coordinate system* (CS) is the fixed, default CS for each new project. It cannot be edited or deleted.

A *relative CS* is user-defined. Its origin and orientation can be set relative to an existing CS. Relative CSs enable you to easily draw objects that are located relative to other objects. If you modify a relative CS, all objects drawn on that CS will be affected and change position accordingly. You can choose to set a relative CS that is *offset* from an existing CS, *rotated* from an existing CS, or *both* offset and rotated from an existing CS.

A *face CS* is also user-defined. Its origin is specified on a planar object face. Face CSs enable you to easily draw objects that are located relative to an object's face.

Switch between global, relative, and face CSs by changing the *working CS*. Simply click the CS you want to use in the history tree. The working CS is indicated by a red *W* that appears at the lower-left corner of the CS name in the history tree. The **Properties** dialog box lists the CS associated with an object.

User-defined CSs are saved with the active project.

Related Topics

Creating a Relative Coordinate System

Creating a Face Coordinate System

Setting the Working Coordinate System

Modifying the Coordinate System Axes View

Setting the Working Coordinate System

The working coordinate system (CS) is the current CS with which objects being drawn are associated. The working CS can be the global CS or a user-defined relative CS or face CS. Select the working CS by clicking its name in the history tree, or follow this procedure:

1. On the **3D Modeler** menu, point to **Coordinate System**, and then click **Set Working CS**.

   The **Select Coordinate System** window appears.

2. Click a CS in the list.
3. Click **Select**.

   A red *W* appears at the lower-left corner of the CS name in the history tree, indicating that it is the working CS.

   Objects that you draw hereafter will be associated with the CS you selected.

Related Topics

Setting Coordinate Systems

Creating a Relative Coordinate System

When creating a relative CS, you have the following options:

- You can create an *offset relative CS*, that is, a relative CS whose origin lies a specified distance from another CS's origin. By moving a CS's origin, you can enter coordinates relative to an existing object, without having to add or subtract the existing object's coordinates.
- You can create a *rotated relative CS*, that is, a relative CS whose axes are rotated away from another CS's axes. By rotating the axes of a CS, you can easily add an object that is turned at an angle relative to another object.
- You can also create a relative CS that is *both offset and rotated*.

Creating an Offset Relative CS

To create a relative CS with an origin that lies a specified distance from another CS's origin:

1. In the history tree, click the CS upon which you want to base the new relative CS, making it the working CS.
2. Point to **3D Modeler>Coordinate System>Create>Relative CS**.
3. On the **Relative CS** menu, click **Offset**  .
4. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   To select a point that does not lie in the current plane, use the **Movement Mode** commands on the shortcut menu.

   The new relative CS is created. Its origin has moved from the previous working CS, but its axes remain the same. It is listed in the history tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.

Related Topics

Creating a Relative Coordinate System

Creating an Offset and Rotated Relative CS

Creating a Rotated Relative CS

To create a new relative CS with its axes rotated away from another CS's axes:

1. In the history tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Point to **3D Modeler>Coordinate System>Create>Relative CS**.
3. On the **Relative CS** menu, click **Rotated**  .
4. Specify the x-axis by selecting a point on the axis in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

To select a point that does not lie in the current plane, use the **Movement Mode** commands on the shortcut menu.

5. Specify the xy plane by selecting any point on it in one of the following ways:
   - Click the point.
   - Type the coordinates of a point that is relative to the previously selected point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   You do not need to specify the z-axis. It is automatically calculated to be at a right angle to the y-axis.

   The new relative CS is created. It has the same origin as the previous working CS, but its axes are rotated. It is listed in the history tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.

Related Topics

Creating a Relative Coordinate System

Creating an Offset and Rotated Relative CS

Creating an Offset and Rotated Relative CS

To create a new relative CS that is both offset and rotated from an existing CS:

1. In the history tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Point to **3D Modeler>Coordinate System>Create>Relative CS**.
3. On the **Relative CS** menu, click **Both**  .
4. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.

   To select a point that does not lie in the current plane, use the **Movement Mode** commands on the shortcut menu.
5. Specify the x-axis by selecting a point on the axis in one of the following ways:
   - Click the point.
   - Type the coordinates of a point that is relative to the origin in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.
6. Specify the xy plane by selecting any point on it in one of the following ways:
   - Click the point.
   - Type the coordinates of a point that is relative to the previously selected point in the **dX**, **dY**, and **dZ** boxes.

   You do not need to specify the z-axis. It is automatically calculated to be at a right angle to the y-axis.

   The new relative CS is created. It is listed in the history tree under **Coordinate Systems**. It

automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.

Related Topics

Creating a Relative Coordinate System

Creating a Face Coordinate System

1. Select the object face upon which you want to create the face CS.
2. Click **3D Modeler>Coordinate System>Create>Face CS**  .
3. Select the origin in one of the following ways:
   - Click the point on the face.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
4. Specify the x-axis by selecting a point on the object face in one of the following ways:
   - Click the point.
   - Type the coordinates of a point that is relative to the previously selected point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

   You do not need to specify the y- or z-axes. HFSS assumes that the z-axis is normal to the object face and the y-axis is automatically calculated to be at a right angle to the z-axis.

   The new face CS is listed in the history tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be referenced to the coordinates of this face CS. Default planes are created on its xy, yz, and xz planes.

Only operations listed in the history tree *before* the face CS's creation will affect the face CS, and in turn, affect objects dependent upon that face CS. A face CS, or objects created on it, is *not* affected by operations that occur after it is created.

For example, suppose you create a box, then a face CS on a face of the box, and then a cylinder on the face CS. If you then edit the box's dimensions in the **Properties** dialog box, the cylinder will move accordingly. But if you rotate the box using the **Edit>Arrange>Rotate** command, the box will move, but the cylinder will not move because the operation occurs later in the history tree.

Related Topics

Automatically Creating Face Coordinate Systems

Setting the Working Coordinate System

Modifying Coordinate Systems

Setting Coordinate Systems

Automatically Creating Face Coordinate Systems

You can instruct HFSS to automatically create a new face CS every time you draw on an object's face.

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler Options**.

   The **3D Modeler Options** dialog box appears.

2. Select **Automatically switch to face coordinate system**.
3. Click **OK**.

Now, when you select a face, and then click a drawing command, a new face CS will be created on the face. HFSS automatically sets the new face CS as the working CS. The object you draw is oriented according to the new face CS.

Note HFSS will not automatically create a new face CS if a face CS has already been assigned to the selected face.

Related Topics

Creating a Face Coordinate System

Modifying Coordinate Systems

Keep in mind that when you edit a CS, the following will also be affected:

- All objects drawn on the CS.
- All CSs that were defined relative to that CS.
- All objects drawn on a CS that was defined relative to that CS.

1. On the **3D Modeler** menu, point to **Coordinate System**, and then click **Edit**.

   The **Select Working CS** window appears.
2. Click the CS you want to modify.
3. Click **Select**.
4. If you selected a relative CS, follow the directions for creating a relative CS.

   If you selected a face CS, follow the directions for creating a face CS.

Related Topics

Setting Coordinate Systems

Creating a Relative Coordinate System

Creating a Face Coordinate System

Modifying the Coordinate System Axes View

Deleting Coordinate Systems

1. Click the name of the CS you want to delete in the history tree.
2. On the **Edit** menu, click **Delete**  .
   - Alternatively, press **Delete**.

   The CS will be deleted and all objects drawn on it will be deleted. Further, any CS that was dependent upon the deleted CS will be deleted and any objects that were drawn on the dependent CS will also be deleted.

Related Topics

Setting Coordinate Systems

Creating a Relative Coordinate System

Creating a Face Coordinate System

Modifying the Coordinate System Axes View

6

Assigning Boundaries

Boundary conditions specify the field behavior at the edges of the problem region and object interfaces. You may assign the following types of boundaries to an HFSS design:

Perfect E	Represents a perfectly conducting surface.
Perfect H	Represents a surface on which the tangential component of the H-field is the same on both sides.
Impedance	Represents a resistive surface.
Radiation	Represents an open surface from which energy can radiate.
PML	Represents several layers of specialized materials that absorb outgoing waves.
Finite Conductivity	Represents an imperfect conductor.
Symmetry	Represents a perfect E or perfect H plane of symmetry.
Master	Represents a surface on which the E-field at each point is matched to another surface (the slave boundary) to within a phase difference.
Slave	Represents a surface on which the E-field at each point has been forced to match the E-field of another surface (the master boundary) to within a phase difference.
Lumped RLC	Represents any combination of lumped resistor, inductor, and/or capacitor in parallel on a surface.
Layered Impedance	Represents a structure with multiple layers as one impedance surface.

You may also choose to designate a perfect E, finite conductivity, or impedance boundary as an infinite ground plane if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.

For convenience, you can access the **Edit Global Materials** command from the Boundaries menu.

Note By default, the history tree in the 3D modeler window groups sheet objects according to boundary assignment. To change this, select the **Sheets** icon and right-click to display the **Group Sheets by Assignment** checkbooks.

Related Topics

Technical Notes: Boundaries

Zoom to Selected Boundary

Zoom to Selected Boundary

You can select on a boundary name in the Project tree, and right-click, the popup menu includes a **Zoom to** command. This zooms the view in the 3D Modeler view in or out to show the selected boundary. This can be very useful in looking at problem areas.

Assigning Perfect E Boundaries

A perfect E boundary is used to represent a perfectly conducting surface in a structure.

1. Select the object or object face to which you want to assign the perfect E boundary.
2. Click **HFSS>Boundaries>Assign>Perfect E**.

   The **Perfect E Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.
5. Click **OK**.

   The new boundary is listed under **Boundaries** in the project tree.

Related Topics

Technical Notes: Perfect E Boundaries

Assigning Perfect H Boundaries

A perfect H boundary represents a surface on which the tangential component of the H-field is the same on both sides. For internal surfaces, this results in a natural boundary through which the field propagates. For surfaces on the outer surface of the model, this results in a boundary that simulates a perfect magnetic conductor in which the tangential component of the H-field is zero.

1. Select the object or object face to which you want to assign the perfect H boundary.
2. Click **HFSS>Boundaries>Assign>Perfect H**.

   The **Perfect H Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Click **OK**.

   The new boundary is listed under **Boundaries** in the project tree.

Assigning Impedance Boundaries

An impedance boundary represents a resistive surface. The behavior of the field at the surface and the losses generated by the currents flowing inside the resistor are computed using analytical formulas; HFSS does not actually simulate any fields inside the resistor.

1. Select the object or object face to which you want to assign the impedance boundary.
2. Click **HFSS>Boundaries>Assign>Impedance**.

   The **Impedance Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Enter the **Resistance** in ohms/square.
5. Enter the **Reactance** in ohms/square.
6. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.

   Note that if you select **Infinite Ground Plane**, the effect of the impedance boundary will be incorporated into the field solution in the usual manner, but the radiated fields will be computed as if the lossy ground plane is perfectly conducting.
7. Click **OK**.

   The new boundary is listed under **Boundaries** in the project tree.

Note You can assign a variable as the resistance and reactance values.

Related Topics

Technical Notes: Impedance Boundaries

Assigning Radiation Boundaries

For Driven Modal or Driven Terminal Designs

A radiation boundary is used to simulate an open problem that allows waves to radiate infinitely far into space, such as antenna designs. HFSS absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure.

A radiation surface does not have to be spherical, but it must be exposed to the background, convex with regard to the radiation source, and located at least a quarter wavelength from the radiating source. In some cases the radiation boundary may be located closer than one-quarter wavelength, such as portions of the radiation boundary where little radiated energy is expected.

1. Select the object or object face to which you want to assign the radiation boundary.
2. Click **HFSS>Boundaries>Assign>Radiation**.

   The **Radiation Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Optionally, select **Advanced Options** to display a list of radio buttons showing boundary properties. If your project uses a fieldsolution from another source, your "target" project must have radiation boundaries with **Advanced Options** defined defined in order to specify where the fields from the "source" project enter. See the discussion here.
5. These can be:
   - **Radiating Only** - this refers to the original radiating surface properties (the default).
   - **Incident Field** - the incident field source patterns are projected on these surfaces and are backed by ABC or PML. This is like a generalized space port. HFSS knows the incident field pattern, applies it to the port and expects a reflected field pattern which radiates back. In other words, it behaves as if you excited the project by a Norton or Thevenin generator using an impedance which is the free space wave impedance.
   - **Enforced Field** - this has the H tangential component of the incident field directly applied on these surfaces. It is an inhomogeneous Newmann BC. In other words, it behaves as if you excited the project by an ideal current source (enforced current).
   - **Reference for Frequency Selective Surface (FSS)** - this surface become the input surface for calculations of the reflection/transmission coefficients. The other radiating surface automatically becomes output. Only one FSS can be defined in a given model. Reflection/Transmission coefficients for FSS designs can be viewed in the solution data

panel as S-parameters or you can create an S-parameter report.

Note

If you select either **Enforced Field** or **Incident Field** you should run a validation check in order to avoid an invalid setup. The setup is invalid if any of these surfaces are internal.

If you select either **Enforced Field** or **Incident Field** in most cases, you should avoid internal surfaces. In order to do that, internal objects with Enforced/Incident Field BC should be substructed to become background, or PEC material should be assigned to these objects to become “NoSolveinside”.

6. Click **OK**.

   The new boundary is listed under **Boundaries** in the project tree.

Note

Do not define a surface that cuts through an object to be a radiation boundary.

In general, do not define the interface of two internal objects to be a radiation boundary. The only exception is when one object is a perfectly matched layer boundary (PML) and the other is the PML base object.

Related Topics

Technical Notes: Radiation Boundaries

Assigning PML Boundaries

A perfectly matched layer (PML) boundary is used to simulate materials that absorb outgoing waves. Setting up a PML boundary is similar to setting up a radiation boundary. You start by drawing a virtual object around the radiating structure. However, instead of placing a radiation boundary on its surfaces, you add PMLs to fully absorb the electromagnetic field.

HFSS can create PMLs automatically, or you can create them manually. Create PMLs automatically if the base object touching the PML is planar and its material is homogenous. HFSS creates a separate PML object for each covered face. The PML boundaries are grouped in the **Project** tree under the **Boundaries** icon. Within these groupings, you can edit the radiation parameters (for example, as Incident Wave Port) in order to set up the right total field excitation based on the physical optics approach. PML radiation boundaries are not generated in eigenmode projects.

In creating PMLs, you can select non rectangular faces as long as they do not touch any other selected face. The underlying object does not have to be a box. If there are faces that touch, the touching faces must locally be box-like.

What do you want to do?

Create PMLs automatically.

Create PMLs manually.

Guidelines for Assigning PML Boundaries

Creating PMLs Automatically

1. Draw a PML base object at the radiation surface.
2. Select the faces of the PML base object to turn into PMLs.

   Select only external, planar faces and exclude faces defined as symmetry boundaries.
3. On the **HFSS** menu, point to **Boundaries**, and then click **PML Setup Wizard**.

   The **PML Setup** wizard appears.
4. Select **Create PML Cover Objects on Selected Faces**.
5. Type the thickness of each layer in the **Uniform Layer Thickness** text box. You can assign a variable as the thickness value.

   Note The layer thickness cannot be modified directly after PML objects have been created. If you want to be able to modify the thickness, assign a variable as the thickness value.

   If you do not assign a value, you can select **Use Default Formula** to have HFSS calculate a value for you.
6. Optionally, select **Create joining corner and edge objects**.

   Edge and corner PML objects will be created to join adjacent PML surfaces together, ensuring complete coverage. This option is only available if the selected faces are on a box object.
7. For non-planar, you can select **Use Selected Object, '***objectName***', as PML Cover**.
8. Under **Base Face Radiation Properties**, click a radio button to specify one of the following:

- **Radiating Only** - the radiation surface (default).
- **Incident Field** - the incident field source patterns are projected on these surfaces and are backed by ABC or PML. This is like a generalized space port. HFSS knows the incident field pattern, applies it to the port and expects a reflected field pattern which radiates back. In other words, it behaves as if you excited the project by a Norton or Thevenin generator using an impedance which is the free space wave impedance.
- **Reference for Frequency Selective Surface (FSS)** - this surface becomes the input surface for calculations of the reflection/transmission coefficients. The other radiating surface automatically becomes output. Only one FSS can be defined in a given model. Reflection/Transmission coefficients for FSS designs can be viewed in the solution data panel as S-parameters or you can create an S-parameter report.

  If you check **Reference for Frequency Selective Surface (FSS)**, the PML objects will stay visible.

9. Click **Next**.

   HFSS creates PMLs from the faces you selected. Names are automatically given to the layers. that start with *PML*, which is necessary for HFSS to recognize them as PMLs.

10. Specify how the PMLs terminate by selecting one of the following:

    a. **PML Objects Accept Free Radiation** if the PMLs terminate in free space.

       - Then enter the lowest frequency in the frequency range you are solving for in the **Min Frequency** text box.

    b. **PML Objects Continue Guided Waves** if the PMLs terminate in a transmission line.

       - Then specify the propagation constant at the minimum frequency.

11. Specify the minimum distance between the PMLs and any of the radiating bodies in the **Minimum Radiating Distance** text box. You may choose to have HFSS calculate the value by clicking **Use Default Formula**.

    The PML material characteristics depend on the cumulative effect of their near fields at the location of the PML surfaces.

12. Click **Next**.

    HFSS calculates the appropriate PML materials based on the settings you specified and the material of the base object, and assigns these materials to the objects in the PML group.

    A summary dialog box appears, enabling you to modify the settings you specified.

13. Click **Finish**.

Related Topics

Creating PML Boundaries Manually

Modifying PML Boundaries

Guidelines for Assigning PML Boundaries

Technical Notes: PML Boundaries

Creating PML Boundaries Manually

See Guidelines for Assigning PML Boundaries.

1. Draw the PML object at the radiation surface, and then select it.
2. In the **Properties** window, give the object a name with the prefix *PML*.

   Object names that start with *PML* are necessary for HFSS to recognize them as PMLs.
3. On the **HFSS** menu, point to **Boundaries**, and then click **PML Setup Wizard**.

   The **PML Setup** wizard appears.
4. Select **Use Selected Object as PML Cover**.
5. Select the **Corresponding Base Object**, the object touching the PML, from the pull-down list.
6. Type the thickness of each layer in the **Uniform Layer Thickness** text box. You can assign a variable as the thickness value.

   If you do not assign a value, you can select **Use Default Formula** to have HFSS calculate a value for you.
7. Select the orientation of the PML object, the direction of outward propagation, in the relative, or local, coordinate system.
8. Under **Base Face Radiation Properties**, click a radio button to specify one of the following:
   - **Radiating Only** - the radiation surface (default).
   - **Incident Field** - the incident field source patterns are projected on these surfaces and are backed by ABC or PML. This is like a generalized space port. HFSS knows the incident field pattern, applies it to the port and expects a reflected field pattern which radiates back. In other words, it behaves as if you excited the project by a Norton or Thevenin generator using an impedance which is the free space wave impedance.
   - **Reference for Frequency Selective Surface (FSS)** - this surface becomes the input surface for calculations of the reflection/transmission coefficients. The other radiating surface automatically becomes output. Only one FSS can be defined in a given model. Reflection/Transmission coefficients for FSS designs can be viewed in the solution data panel as S-parameters or you can create an S-parameter report.

     If you check **Reference for Frequency Selective Surface (FSS)**, the PML objects will stay visible.
9. Click **Next**.
10. Specify how the PML terminates by selecting one of the following:
    a. **PML Objects Accept Free Radiation** if the PML terminates in free space.
       - Enter the lowest frequency in the frequency range you are solving for in the **Min Frequency** text box.

    b. **PML Objects Continue Guided Waves** if the PML terminates in a transmission line.
       - Specify the propagation constant at the minimum frequency.
11. Specify the minimum distance between the PML and the radiating body in the **Minimum Radiating Distance** text box. You may choose to let HFSS calculate the value by clicking **Use**

Default Formula.

The PML material characteristics depend on the cumulative effect of their near fields at the location of the PML surfaces.

12. Click **Next**.

    HFSS calculates the appropriate PML material based on the settings you specified and the material of the base object, and assigns this material to the PML.

    A summary dialog box appears, enabling you to modify the settings you specified.

13. Click **Finish**.

Related Topics

Guidelines for Assigning PML Boundaries

Modifying PML Boundaries

Technical Notes: PML Boundaries

Guidelines for Assigning PML Boundaries

Keep the following guidelines in mind when assigning PML boundaries:

- When automatically creating PMLs, HFSS creates a new relative coordinate system for each PML object. This results in the z direction of the PML object coinciding with the normal direction of the base object's face.
- HFSS treats PMLs uniformly with regard to thickness. If the PMLs in your design vary in thickness, create a separate PML group for each thickness.

You should manually create a PML in the following situations:

- The base object is curved.

  HFSS calculates the PML material properties using the normal vector at the center of the base object's face. If the face is curved, the normal vector changes with position. The PML materials will only be good approximations if the normal vector at each point on the face is close to the normal vector at the face center.

  It is a good idea to segment the curved surface of the base object for greater accuracy. Create separate PMLs for each segment. Note that each segment's thickness is treated uniformly. The view angle of the segments should be no wider than 45 degrees. The smaller the angle of each segment, the greater the accuracy of the corresponding PML.

- The material of the corresponding base object touching the PML is not homogenous. An example is a metal-shielded microstrip line with a substrate. One PML could be drawn to terminate the microstrip and another could correspond to the substrate.

  Create as many PML objects as there are subsections of material properties in the base object.

Related Topics

Creating PML Boundaries Manually

Technical Notes: PML Boundaries

Modifying PML Boundaries

1. Make sure that nothing is selected in the **3D Modeler** window.
2. On the **HFSS** menu, point to **Boundaries**, and then click **PML Setup Wizard**.

   The summary dialog box of the **PML Setup** wizard appears.
3. If more than one group of PMLs were defined, select the PML group you want to modify from the table.
4. Modify the PML settings.
5. Click **Recalculate**.

   HFSS automatically recalculates and assigns the appropriate PML materials to the objects in the PML group.
6. Click **Finish**.

Note If objects are modified after PMLs are created, the PML materials will be invalid and must be recalculated in the **PML Setup Wizard**. For example, if the material of the PML base object is modified, the associated PML materials must be recalculated in the **PML Setup Wizard**.

Related Topics

Assigning PML Boundaries

Assigning Finite Conductivity Boundaries

A finite conductivity boundary represents an imperfect conductor. It approximates the behavior of the field at the object surface; HFSS does not compute the field inside the object.

The finite conductivity boundary is valid only if the conductor being modeled is a good conductor, that is, if the conductor's thickness is much larger than the skin depth in the given frequency range.

1. Select the object or object face to which you want to assign the finite conductivity boundary.
2. Click **HFSS>Boundaries>Assign>Finite Conductivity**.

   The **Finite Conductivity Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Do one of the following:
   - Enter the conductivity in inverse ohm-meters, and then enter the permeability.
   - Select **Use Material**, click the default material name, and then choose a material from the material editor. The conductivity and permeability values of the material you select will be used for the boundary. Note that selecting a perfectly conducting material for a finite conductivity boundary triggers a validation error.
5. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.

   Note that if you select **Infinite Ground Plane**, the effect of the finite conductivity boundary will be incorporated into the field solution in the usual manner, but the radiated fields will be computed as if the lossy ground plane is perfectly conducting.
6. To specify the roughness of surfaces such as the interface between the conductor and the substrate for a microstrip line, enter a value for **Surface Roughness** and select the units (default, microns) from the pull down menu.

   (This may be more intuitive than using a layered impedence boundary to model the effects.)
7. To specify a layer thickness, click the checkbox to enable the **Layer Thickness** field, and enter a value and select units.
8. Click **OK**.

Note You can assign a variable as the conductivity or permeability values.

Related Topics

Technical Notes: Finite Conductivity Boundaries

Assigning Symmetry Boundaries

For Driven Modal or Eigenmode Designs

A symmetry boundary represents a perfect E or perfect H plane of symmetry. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design.

1. Select the object face to which you want to assign the symmetry boundary.
2. Click **HFSS>Boundaries>Assign>Symmetry**.

   The **Symmetry Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Select the type of symmetry plane the boundary represents: **Perfect E** or **Perfect H**.
5. Click **Impedance Multiplier**.

   If the design includes a port, you must adjust the impedance multiplier or the computed impedances will not be for the full structure.

   The **Port Impedance Multiplier** dialog box appears.
6. Type a value in the **Impedance Multiplier** box, and then click **OK**.
7. Click **OK**.

Related Topics

Technical Notes: Symmetry Boundaries

Setting the Impedance Multiplier

Technical Notes: *Impedance Multipliers*

Assigning Master Boundaries

Master and slave boundaries enable you to model planes of periodicity where the E-field at every point on the slave boundary surface is forced to match the E-field of every corresponding point on the master boundary surface to within a phase difference. The transformation used to map the E-field from the master to the slave is determined by specifying a coordinate system on both the master and slave boundaries.

1. Select the face to which you want to assign the master boundary.
2. Click **HFSS>Boundaries>Assign>Master**.

   The **Master Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. HFSS uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.
   a. Select **New Vector** from the **U Vector** pull-down list.

      The **Master Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.

      The **Master Boundary** dialog box reappears
   d. To reverse the direction of the V vector, select **Reverse Direction**.
5. Click **OK**.

   HFSS will compute the E-field on this boundary and map it to the slave boundary using the transformation defined by the master and slave coordinate systems.

Related Topics

Technical Notes: Master and Slave Boundaries

Assigning Slave Boundaries

Assigning Slave Boundaries

Master and slave boundaries enable you to model planes of periodicity where the E-field at every point on the slave boundary surface is forced to match the E-field at every corresponding point on the master boundary surface to within a phase difference. The transformation used to map the E-field from the master to the slave is determined by specifying a coordinate system on both the master and slave boundaries.

1. Select the face to which you want to assign the slave boundary.
2. Click **HFSS>Boundaries>Assign>Slave**.

   The **Slave Boundary** wizard appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Select the corresponding master boundary from the **Master Boundary** pull-down list.

   If a master boundary has not yet been defined, return to make this selection when it has been defined.
5. You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. HFSS uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.
   a. Select **New Vector** from the **U Vector** pull-down list.

      The **Slave Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.

      The **Slave Boundary** dialog box reappears.
   d. To reverse the direction of the V vector, select **Reverse Direction**.
6. Click **Next**.
7. You have the option to relate the slave boundary's E-fields to the master boundary's E-fields in one of the following ways:
   - For driven designs, select **Use Scan Angles to Calculate Phase Delay** to enable the **Scan Angle** fields. Then enter the ϕ scan angle in the **Phi** box and the θ scan angle in the **Theta** box. The scan angles apply to whole model, in the global coordinate system. The phase delay is calculated from the scan angles; however, if you know the phase delay, you may enter it directly in the **Phase Delay** box below.

Note For Eigenmode problems, the **Use Scan Angles to Calculate Phase Delay** fields are disabled.

   - Select **Field Radiation**, and then enter the phase difference, or phase delay, between the

boundaries' E-fields in the **Phase Delay** box. The phase delay applies only to this boundary.

Note You can assign a variable as the phi, theta, or phase delay values.

HFSS will compute the E-field on the master boundary and map it to this boundary using the transformation defined by the master and slave coordinate systems.

Related Topics

Technical Notes: Master and Slave Boundaries

Assigning Master Boundaries

Assigning Lumped RLC Boundaries

A lumped RLC boundary represents any combination of lumped resistor, inductor, and/or capacitor in parallel on a surface. Different circuit types can be modeled by varying the combination of circuit element types. For example, a lumped RLC serial circuit connection can be modeled with three sequential circuit elements: one element surface with only resistance present, one element surface with only inductance present, and one element surface with only capacitance present.

1. Select the object or object face to which you want to assign the lumped RLC boundary.
2. Click **HFSS>Boundaries>Assign>Lumped RLC**.

   The **Lumped RLC Boundary** dialog box appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Do the following:
   - If a resistor is present, select **Resistance** and type the resistance value in ohms.
   - If an inductor is present, select **Inductance** and type the inductance value in henrys.
   - If a capacitor is present, select **Capacitance** and type the capacitance value in farads.

     Optionally, you can assign a variable to these values.

   If an element type is not present, do not select it.
5. Draw a current flow line, which represent the start and end points of the circuit element as it was measured:
   a. Select **New Line** from the **Current Flow Line** pull-down list.

      The **Lumped RLC Boundary** dialog box disappears while you draw the current flow line.
   b. Select the start point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select the endpoint using the mouse or the keyboard.

      This point defines the direction and length of the line.

Once the line has been defined, you can edit it as follows:

- Select **Swap End Points** from the **Current Flow Line** pull-down list to switch the start and endpoints of the line, reversing the line's direction.

Related Topics

Technical Notes: Lumped RLC Boundaries

Assigning Layered Impedance Boundaries

A layered impedance boundary is used to model multiple thin layers in a structure as one impedance surface. The effect is the same as an impedance boundary condition, except that HFSS calculates the impedance of the surface based on data you enter for the layered structure. Surface roughness is also taken into account. The layered impedance boundary is supported for single-frequency solutions and for Discrete and Interpolating frequency sweeps.

1. Select the face to which you want to assign the layered impedance boundary.
2. Click **HFSS>Boundaries>Assign>Layered Impedance**.

   The **Layered Impedance Boundary** wizard appears.
3. Type the boundary's name in the **Name** text box or accept the default name.
4. Enter the **Surface Roughness** for the layered structure.

   If the layered structure is internal to the design, enter the average surface roughness of the two outermost sides.

   You can assign a variable as this value.
5. Click **Next**.
6. If the layered structure is external to the design, do the following:
   - By default, HFSS assumes the layered structure is external to the design; the outermost layer of the structure is listed. Select whether this layer is an **Infinite**, **Perfect E**, or **Perfect H** layer from the **Thickness/Type** list.

   If the layered structure is within the 3D model, do the following:

   a. Select the **Internal** option.

   b. Enter a thickness for the first layer in the **Thickness/Type** column. You can assign a variable as this value.
7. To change the first layer's material, click **vacuum** and follow the procedure for assigning a material.
8. To add a new layer to the structure:

   a. Click **New Layer**.

      The new layer is added at the end of the list.

   b. Enter a thickness for the layer in the **Thickness/Type** column. You can assign a variable as this value.

   c. To change the layer's material, click **vacuum** and follow the procedure for assigning a material.
9. Optionally, to reorder layers, drag the rows in the list to the desired position.
10. Optionally, to view the impedance values that will be calculated based on the data provided, do the following:

    a. Enter the frequency at which the solution is being solved in the **Test Frequency** text box.

    b. Click **Calculate**.

The real and imaginary components of the HFSS-calculated layered impedance value appear.

11. Click **Finish**.

    The layered impedance boundary is assigned to the selected face.

Note A warning will be posted if a fast sweep is defined in a design that contains a layered impedance boundary, since the impedance may only be accurate for the center frequency.

Related Topics

Technical Notes: Layered Impedance Boundaries

Designating Infinite Ground Planes

To simulate the effects of an infinite ground plane:

- Select the **Infinite ground plane** check box when setting up a perfect E, finite conductivity, or impedance boundary condition.

This selection only affects the calculation of near- and far-field radiation during post processing. HFSS models the boundary as a finite portion of an infinite, perfectly conducting plane.

Related Topics

Technical Notes: Infinite Ground Planes

Modifying Boundaries

To change the properties of a boundary, do one of the following:

- Double-click the boundary's icon in the project tree.

  The boundary's dialog box appears, in which you can edit its properties.
- Right-click the boundary in the project tree, and then click **Properties** on the shortcut menu.

  The boundary's dialog box appears, in which you can edit its properties.
- On the **HFSS** menu, click **List**.

  The **Design List** dialog box appears, in which you can modify the properties of one or more boundaries.

Deleting Boundaries

To delete *one boundary*:

1. Select the boundary you want to delete by selecting its icon in the project tree.
2. On the **Edit** menu, click **Delete**  .

To delete *all boundaries*:

- On the **HFSS** menu, point to **Boundaries**, and then click **Delete All**.

You can also *delete one or more boundaries* in the **Design List** dialog box:

1. On the **HFSS** menu, click **List**.

   The **Design List** dialog box appears.
2. Under the **Boundaries** tab, click the row of the boundary you want to delete.
3. Click **Delete**.

Reassigning Boundaries

You can reassign a boundary to another surface. This is useful when you have modified objects with assigned boundaries, invalidating the boundaries. For example, if you unite two objects with assigned boundaries, the second object's boundary will become invalid because united objects maintain the characteristics of the first object selected. In this case, you would need to reassign the boundary or delete it

1. Select the object or object face to which you want to assign an existing boundary.
2. Click **HFSS>Boundaries>Reassign**.

   The **Reassign Boundary** window appears.
3. Select an existing boundary from the list, and then click **OK**.

   The boundary is reassigned to the object or object face.

Note When reassigning a boundary that includes vectors in its definition, HFSS attempts to preserve the vectors with the new assignment, but this is not always possible.

Alternatively, select the object or object face to which you want to assign an existing boundary. Right-click the existing boundary in the project tree, and then click **Reassign** on the shortcut menu.

Reprioritizing Boundaries

Each boundary you assign overwrites any existing boundary which it overlaps. You can change the priority of a previously assigned boundary to be greater than a more recently assigned boundary.

The order of boundaries is important because, for any given triangle of the mesh, only one boundary or excitation can be visible to the solvers. When two boundary definitions overlap, the one with the higher priority is visible to the solvers. .

1. Click **HFSS>Boundaries>Reprioritize** to reprioritize boundaries.

   The **Reprioritize Boundaries** window appears. The order the boundaries and excitations appear in the list indicates the order in which they were defined. The lowest priority assignment appears at the top of the list.

   Ports are automatically placed at the bottom (highest priority) of the list; you cannot move a boundary to a higher priority than a port. Magnetic Bias Excitations (if any) have the lowest priority. Other boundaries and excitations appear between these two extremes.

2. Drag the boundary you want to change to the desired order of priority.

Note The order of boundaries and excitations in the project tree is alphabetical. The order does not correspond to the order of boundaries and excitations visible to the solvers.

Related Topics

Reviewing Boundaries and Excitations in the Solver View

Global Material Environment

The **HFSS>Boundaries>Edit Global Material Environment** command displays the **Global Material Environment** dialog. By clicking the **Material** button, you can access the Select Definition dialog. This lets you work with the materials library.

Duplicating Boundaries and Excitations with Geometry

To duplicate a boundary or excitation when its geometry is pasted or duplicated:

1. Open the **HFSS Options** dialog box: On the **Tools** menu, point to **Options**, and then click **HFSS Options**.
2. Select **Duplicate boundaries with geometry**.

   All boundaries and excitations will be duplicated with their associated geometries until you choose to clear this option.

Hint Use this option to copy and paste boundaries. For example:

1. Select the face to which you want to assign the boundary.
2. On the **3D Modeler** menu, point to **Surface**, and then click **Create Object From Face**.
3. Assign the boundary to the new face object.
4. Copy and paste the new face object to copy and paste the boundary.

Related Topics

Copying and Pasting Objects

Showing and Hiding Boundaries and Excitations

You can choose to show or hide a boundary or excitation's geometry, name, or vectors, in the active view window or in all view windows.

What do you want to do?

Show or hide a boundary or excitation in the active view window.

Show or hide a boundary or excitation in every view window.

Showing and Hiding Boundaries and Excitations in the Active View Window

1. On the **View** menu, click **Active View Visibility**  or select the **Active View Visibility** icon in the toolbar.

   The **Active View Visibility** dialog box appears.
2. Select the tab for the objects you want to show or hide. The dialog contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.
4. Click the **Boundaries** tab if you want to show or hide boundaries.

   Click the **Excitations** tab if you want to show or hide excitations.

   - For designs with large numbers of objects, you can resize the dialog for easier selection.
   - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
   - You can slso use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

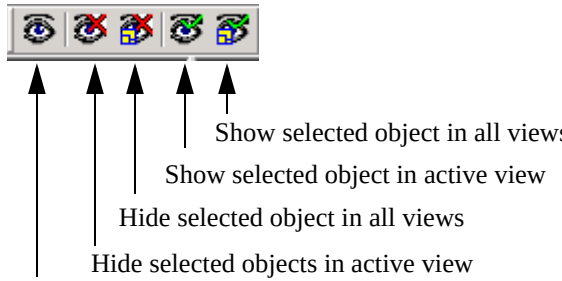
   The objects you select and designate as Visible (by selecting the property or using Show) appear.
5. Clear the **Visibility** selection of boundaries or excitations that you want to hide from view.

   The boundary or excitation will only be visible in the active view window if it is selected.
6. Select the **Visibility** option for boundaries or excitations that you want to show in the active view window.

   The boundary or excitation will be visible in the active view window when it is selected or when it is not selected.

   You can also use the toolbar icons to **Show/Hide selected objects in all views** and **Show/Hide**

selected objects in active views.



Showing and Hiding Boundaries and Excitations in Every View Window

1. Click **HFSS>Boundaries>Visualization** if you want to show or hide boundaries.

   Click **HFSS>Excitations>Visualization** if you want to show or hide excitations.
2. Clear the **View Geometry**, **View Name**, or **View Vector** selection of boundaries and excitations that you want to hide from view. Select the options you want to show.

   The options affect all view windows.

Reviewing Boundaries and Excitations in the Solver View

After you have assigned all the necessary boundaries and excitations to a model, you should review their order of priority according to the HFSS solver. Reviewing the solver's view of the model's boundaries and excitations enables you to verify that their order during the solution process will be as you intended.

To check the solver's view of boundaries and excitations:

1. On the **HFSS** menu, click **Boundary Display (Solver View)**.

   HFSS generates an initial mesh and determines the locations of the boundaries and excitations on the model.

   The **Solver View of Boundaries** window appears, which lists all the boundaries and excitations for the active model in the order specified in the **Reprioritize Boundaries and Excitations** dialog box.

2. Select the **Visibility** option for the boundary or excitation you want to review.

   The selected boundary or excitation will appear in the **3D Modeler** window in the color it has been assigned.

   - **Visible to Solver** will appear in the **Solver Visibility** column for each boundary or excitation that is valid.
   - **Overridden** will appear in the **Solver Visibility** column for each boundary or excitation that will be ignored by the solver as a result of it overlapping an existing boundary or excitation with a higher priority.

3. Verify that the boundaries or excitations you assigned to the model are being displayed as you intended for solving purposes.
4. If the order of priority is not as you intended, reprioritize the boundaries and excitations.

Related Topics

Technical Notes: Default Boundary Assignments

Setting Default Values for Boundaries and Excitations

When assigning a boundary or excitation, many of the fields in the boundary and excitation dialog boxes have default values associated with them. These default values are initially set by HFSS, but can be overridden.

To modify the default values associated with a specific boundary or excitation type:

1. Assign a boundary or excitation.
2. Modify any default values.
3. Close the boundary or excitation's dialog box.
4. Re-open the new boundary or excitation's dialog box. It now includes a **Defaults** tab.
5. Under the **Defaults** tab, click **Save Defaults**.

   The values assigned to this boundary are saved as the default values and will be assigned when new boundaries of this type are created.
6. Optionally, click **Revert to Standard Defaults**.

   The default values you set for this boundary type will be cleared and will revert to the default values set by HFSS.

Note For PLM boundaries, the defaults are set via a formula, rather than a value.

7

Assigning Excitations

Excitations in HFSS are used to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces in the design. You may assign the following types of excitations to a Driven solution-type HFSS design:

Wave Port	Represents the surface through which a signal enters or exits the geometry.
Lumped Port	Represents an internal surface through which a signal enters or exits the geometry.
Incident Wave	Represents a propagating wave impacting the geometry.
Voltage Source	Represents a constant electric field across feed points.
Current Source	Represents a constant electric current across feed points.
Magnetic Bias	Used to define the net internal field that biases a 3D ferrite material object.

After assigning an excitation, you can modify it in some of the following ways, if applicable to the excitation type:

- Change its properties.
- Delete it.
- Reassign it to another surface.
- Reprioritize it.
- Hide it from view.
- Modify the impedance multiplier.

Related Topics

Technical Notes: Excitations

Zoom to Selected Excitation

Zoom to Selected Excitation

When you select on an excitation name in the Project tree, and right-click, the popup menu includes a **Zoom to** command. Selecting this command zooms the view in the 3D Modeler view in or out to show the selected excitation. This can be very useful in looking at problem areas.

Assigning Wave Ports

Wave ports represent places in the geometry through which excitation signals enter and leave the structure. They are used when modeling strip lines and other waveguide structures. The setup of wave ports varies slightly depending on whether your solution is modal or terminal.

Note Use lumped ports to represent an *internal* surface through which an excitation signal enters or exits the geometry.

Related Topics

Assigning Wave Ports for Modal Solutions

Assigning Wave Ports for Terminal Solutions

Technical Notes: Wave Ports

Assigning Wave Ports for Modal Solutions

1. Select the object face to which you want to assign the port.
2. Click **HFSS>Excitations>Assign>Wave Port**.

   The **Wave Port** wizard appears.
3. Type the port's name in the **Name** text box or accept the default name.
4. Click **Next**.

   This shows the **Modes** window. Here you specify the number of modes for a port, define integration lines for each mode, and choose whether to renormalize the port.
5. To specify more than one mode to analyze at the port, type a new value in the **Number of Modes** box.

   The mode spreadsheet is updated to include the total number of modes.
6. To specify an integration line for a port mode, follow the directions for defining an integration line.

   When you have defined an integration line, the table cell under the Integration line heading changes from "None" to "Defined." Clicking on the cell now shows a drop down list of available options. These are:

   - Defined - acknowledges a current definition.
   - None - no line defined.
   - Swap Endpoints - inverts the end points of a line.
   - Duplicate Line... - lets you duplicate a currently defined line.
   - New Line... - lets you create a new line.

   If you change an existing integration line, use the options for this line. If you need to define an integration line for one or more modes, repeat the process for each. If a solution exists adding or changing integration lines invalidates them, and issues a warning.
7. If you want to align the E-field of the modes with the integration line, select **Polarize E Field**.

If solutions exist, changing this selection invalidates them and issues a warning message.

8. Select the method with which to calculate the characteristic impedance by selecting **Zpi**, **Zpv**, or **Zvi** from the **Char Imp. (Zo)** pull-down list.

   For definitions of how HFSS defines these values, see Calculating the PI Impedance, Calculating the PV Impedance, and Calculating the VI Impedance.

9. Click **Next**.

   This displays the **Wave Port: Post Processing** window. Values here affect S-Parameters only.

   The **Port Renormalization** choices include:

   - **Do Not Renomalize** (the default)
   - **Renormalize All Modes**. This enables the **Full Port Impedance** text box. The default impedance for re-normalization of each port is 50 ohms.

     If you want to enter a complex impedence, enter it in the following form:

     <re> + <im>j
   - If there are multiple modes, the **Renormalize Specific Modes** is enabled. Click this to enable the **Edit Mode Impedances** button. This opens a editable table with the impedances for each mode.

10. To deembed the port, select **Deembed**, and then type the distance of the transmission line to add and select the units to use. You can assign a variable as this value. After you enter the value, a blue arrow depicts the embedding distance in the graphics window when the port is selected.

    Note A positive distance value will de-embed into the port. A negative distance value will de-embed out of the port. When you

    - Alternatively, click **Get Distance Graphically** to draw a line with a length representing the de-embed distance. After you draw the line in the 3D window, the Distance field shows the specifed distance. You can edit this value.

11. Click **Finish**.

Related Topics

Defining Integration Lines

Technical Notes: Wave Ports

Technical Notes: Polarizing the E-Fields

Technical Notes: Calculating Characteristic Impedance

Assigning Wave Ports for Terminal Solutions

1. Select the object face to which you want to assign the port.
2. Click **HFSS>Excitations>Assign>Wave Port**.

   The **Wave Port** wizard appears.
3. Type the port's name in the **Name** text box or accept the default name.

4. Click **Next**.

   This shows the **Terminals** window. Here you specify the number of terminals for a port, and define integration lines for each terminal.

   The number of terminals on a wave port must be equal to the number of modes set for the port.

5. To specify more than one terminal to analyze at the port, type a new value in the **Number of Terminals** box.

   The terminal spreadsheet is updated to include the total number of terminals.

6. To specify a terminal line for a port, follow the directions for defining a terminal line.

   When you have defined a terminal line, the table cell under the Terminal line heading changes from “None” to “Defined.” Clicking on the cell shows the change to available options after defining a terminal line. The options for the terminal line now are:

   - Defined - acknowledges a current definition.
   - None - no line defined.
   - Swap Endpoints - inverts the end points of a line.
   - Duplicate Line... - lets you duplicate a currently defined line.
   - New Line... - lets you create a new line.

   If you choose to make changes to an existing terminal line, use the options for this line. If you need to define a terminal line for one or more modes, repeat step 5 for each. If a solution exists, adding or changing a terminal line invalidates the solution and causes a warning message.

7. When you are finished in the **Wave Port: Terminals** window, click **Next**.

   This displays the **Wave Port: Differential Pairs** window. If you have previously defined differential two terminal lines on a single port, the differential pairs fields are enabled. You have the option of defining differential pairs.

8. Optionally, follow the directions for defining differential pairs. Each terminal can be used with only one differential pair.

   To specify a reference impedance value:

   a. Type an impedance value in the **Reference Impedance** text box, and select an associated Ohms value.

   b. To apply the value to all terminal lines, click **Set All Zref**. To apply the value to Different terminal lines, click **Set All Diff Zref**. To apply the value to common terminal lines, click **Set All Comm. Zref.**

9. When you are finished in the **Wave Port: Differential Pairs** window, click **Next.**

   This displays the **Wave Port: Post Processing** window. Values here affect S-Parameters only.

   The **Port Renormalization** choices include:

   - **Do Not Renormalize** (the default)
   - **Reference Impedance for All Terminals**. This enables the **Full Port Impedance** text box. The default impedance for re-normalization of each port is 50 ohms.

     If you want to enter a complex impedence, enter it in the following form:

<re> + <im>j

- If there are multiple modes, **Reference Impedance for Specific Terminals** is enabled. Click this to enable the **Edit Terminal Impedances** button. This opens a editable table with the impedances for each terminal.

10. To deembed the port, select **Deembed**, and then type the distance of the transmission line to add. You can assign a variable as this value. After you enter the value, a blue arrow depicts the embedding distance in the graphics window when the port is selected.

Note A positive distance value will de-embed into the port. A negative distance value will de-embed out of the port.

- Alternatively, click **Get Distance Graphically** to draw a line with a length representing the de-embed distance. After you draw the line in the 3D window, the Distance field shows the specifed distance. You can edit this value.

11. Click **Finish**.

Related Topics

Technical Notes: Wave Ports

Assigning Lumped Ports

Lumped ports are similar to traditional wave ports, but can be located internally and have a complex user-defined impedance. Lumped ports compute S-parameters directly at the port. They are used when modeling microstrip structures. Their setup varies slightly depending on whether the solution is modal or terminal.

Note Use wave ports to model *exterior* surfaces through which a signal enters or exits the geometry.

A lumped port can be defined as a rectangle from the edge of the trace to the ground or as a traditional wave port. The default boundary is perfect H on all edges that do not come in contact with the metal.

The following restrictions apply:

- The complex full port impedance must be non-zero and the resistance must be non-negative.
- Only one port mode is allowed, or one terminal if it is a terminal solution.
- An integration or terminal line must be defined.

Related Topics

Assigning Lumped Ports for Modal Solutions

Assigning Lumped Ports for Terminal Solutions

Technical Notes: Lumped Ports

Technical Notes: Calculating Characteristic Impedance

Assigning Lumped Ports for Modal Solutions

1. Select the object face to which you want to assign the port.
2. Click **HFSS>Excitations>Assign>Lumped Port**.

   The **Lumped Port** wizard appears.
3. Type the port's name in the **Name** text box or accept the default name.
4. Define the complex **Full Port Impedance**:
   a. Enter the resistance or real part of the impedance in the **Resistance** text box.
   b. Enter the reactance or imaginary part of the impedance in the **Reactance** text box,.

      You can assign a variable as these values.
5. Click **Next.**

   This displays the **Lumped Port: Modes** window. The number of Modes is not editable.
6. For the Integration Line column, follow the directions for defining an integration line.
7. For the **Char. Imp. (Zo)** column, set the method with which to calculate the characteristic impedance by selecting **Zpi**, **Zpv**, or **Zvi** from the **Char. Imp. (Zo)** pull-down list.

   For definitions of how HFSS defines these values, see Calculating the PI Impedance, Calculating the PV Impedance, and Calculating the VI Impedance.

8. Click **Next**.

   This displays the **Lumped Port: Post Processing** window. Values here affect S-Parameters only. By default, lumped ports are renormalized to a 50 Ohm full port impedance. To specify a renormalization impedance, select **Renormalize All Modes** and type a value in the **Full Port Impedance** text box. Select the corresponding unit in the drop down menu.

   If you want to enter a complex impedence, enter it in the following form:

   <re> + <im>j

   If you do not want to renormalize the port impedance, select **Do Not Renormalize**.

9. Click **Finish**.

Related Topics

Defining an Integration Line

Technical Notes: Lumped Ports

Assigning Lumped Ports for Terminal Solutions

1. Select the object face to which you want to assign the port.
2. Click **HFSS>Excitations>Assign>Lumped Port**.

   The **Lumped Port** wizard appears.
3. Type the port's name in the **Name** text box or accept the default name.
4. Define the complex **Full Port Impedance** of the port:
   a. Enter the resistance or real part of the impedance in the **Resistance** text box.
   b. Enter the reactance or imaginary part of the impedance in the **Reactance** text box.

   You can assign a variable as these values.
5. Click **Next**.

   This displays the **Lumped Port: Terminals** window. The number of terminals is not editable.
6. If desired, follow the directions for defining a terminal line.
7. If desired, you can edit the name of the terminal line by clicking on the default name and editing.
8. Click **Next**.

   This displays the **Lumped Port: Post Processing** window. Values here affect S-Parameters only. Port processing operations do not affect field plots. By default, the port reference impedance for all terminals is 50 Ohms.

   If you want to enter a complex impedence, enter it in the following form:

   <re> + <im>j
9. To specify a different full port impedance, type a value in the **Full Port Impedance** text box. Select the corresponding unit in the drop down menu.
10. Click **Finish** when done.

Related Topics

Defining a Terminal Line

Technical Notes: Lumped Ports

Defining an Integration Line

An integration line is a vector that can represent the following:

- A calibration line that specifies the direction of the excitation field pattern at a port.
- An impedance line along which to compute the Zpv or Zvi impedance for a port.

To define an integration line:

1. In the **Wave Port** or **Lumped Port** dialog boxes, click the **Modes** tab.
2. Select **New Line** from the mode's **Integration Line** list.

   The dialog box disappears while you draw the vector.
3. Select the start point of the vector in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
4. Select the endpoint of the vector using the mouse or the keyboard.

   The endpoint defines the direction and length of the integration line.

   The **Wave Port** or **Lumped Port** dialog box reappears.

Related Topics

Guidelines for Defining Integration Lines

Duplicating Integration Lines

Modifying an Integration Line

Technical Notes: Polarizing the E-Fields

Technical Notes: Setting the Field Pattern Direction

Guidelines for Defining Integration Lines

An integration line is a vector that can represent the following:

- A calibration line that specifies the direction of the excitation field pattern at a port. If you are analyzing more than one mode at a port, define a separate set of integration lines for each mode; the orientation of the electric field differs from mode to mode.
- An impedance line along which to compute the Zpv or Zvi impedance for a port. In this case, select two points at which the voltage differential is expected to be at a maximum. For example, on a microstrip port, place one point in the center of the microstrip, and the other directly underneath it on the ground plane. In a rectangular waveguide, place the two points in the center of the longer sides.

  For definitions of how HFSS defines these Zpv and Zvi values, see Calculating the PV Impedance, and Calculating the VI Impedance.

Duplicating Integration Lines

After you have defined an integration line for a mode, you can duplicate it along a vector one or more times. You can then assign the duplicates to additional modes at the port.

1. In the **Wave Port** dialog box, click the **Modes** tab.
2. Select the mode row containing the integration line you want to duplicate.
3. Select **Duplicate Line** from the row's **Integration Line** list.

   The dialog box disappears while you draw the vector along which to paste the duplicate.
4. Draw the vector along which the duplicate will be pasted:
   a. Select an arbitrary anchor point on the edge of the port face in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   b. Select a second point using the mouse or the keyboard.

      This point defines the direction and distance from the anchor point to duplicate the line.

      The **Duplicate Port Line** dialog box appears.
5. Type the total number of lines, including the original and duplicates, to make in the **Number of Duplicates** box.

   If you type a value that is greater than the number of assigned modes, the extra duplicates will appear as gray integration lines until they are assigned to a mode.
6. Optionally, select **Assign to existing modes**. The duplicates will be assigned to the modes defined for the port, beginning with the mode after the one with the line that was duplicated.
7. Click **OK**.

   The duplicates are pasted along the vector you specified.

Modifying Integration Lines

Modify an existing integration line under the **Modes** tab in the **Wave Port** or **Lumped Port** dialog boxes.

To *swap the coordinates* of an integration line's start point and endpoints:

- Select **Swap Endpoints** from the mode's **Integration Line** list.

  The line's direction will be reversed.

To *copy* a previously defined integration line's points:

- Select **Copy from Mode*n*** from the mode's **Integration Line** list.

  The new integration line will have the same start and endpoints as the selected mode's integration line.

To *delete* a defined integration line for a mode:

- Select **None** from the mode's **Integration Line** list.

Defining a Terminal Line

Terminal voltage lines are used to define voltages on port boundaries.

1. In the **Wave Port** or **Lumped Port** dialog box, click the **Terminals** tab.
2. Select **New Line** from the terminal's **Terminal Line** list.

   The dialog box disappears while you draw the terminal line.
3. Select the start point of the line in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the **X**, **Y**, and **Z** text boxes.
4. Select the endpoint of the line using the mouse or the keyboard.

   The endpoint defines the direction and length of the terminal line.

   The dialog box reappears.

Note In circuit analysis, the polarity reference for a voltage is designated with "+" and "-" symbols. The voltage polarity reference on a terminal voltage line is established by an arrow; the arrow head is synonymous with "+" and the arrow base is synonymous with "-".

Related Topics

Guidelines for Defining Terminal Lines

Duplicating Terminal Lines

Modifying Terminal Lines

Guidelines for Defining Terminal Lines

When defining terminal lines, keep the following guidelines in mind:

- In general, draw a single terminal voltage line from the reference, or ground conductor to each port-plane conductor.
- Solve for all present TEM modes. One terminal voltage line must be created for each port mode in the device.
- Be consistent with the setup of terminal lines. For example, the setup on port 1 should usually be the same as that on port 2.
- Voltage loops are not permitted because voltages are not independent.
- Each terminal voltage line is currently restricted to a single line segment. In certain geometries, this restriction may force you to draw the terminal voltage line through a second conductor. This is permissible; however, you cannot draw more than one terminal voltage line connecting a given reference conductor and port-plane conductor, nor draw a terminal voltage line with its entire length along a perfect conductor.

Duplicating Terminal Lines

After you have defined a terminal line, you can duplicate it along a vector one or more times. You can then assign the duplicates to additional terminals at the port.

1. In the **Wave Port** dialog box, click the **Terminals** tab.
2. Select the terminal row containing the terminal line you want to duplicate.
3. Select **Duplicate Line** from the row's **Terminal Line** list.

   The dialog box disappears while you draw the vector along which to paste the duplicate.
4. Draw the vector along which the duplicate will be pasted:
   a. Select an arbitrary anchor point on the edge of the port face in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   b. Select a second point using the mouse or the keyboard.

      This point defines the direction and distance from the anchor point to duplicate the line.

      The **Duplicate Port Line** dialog box appears.
5. Type the number of copies to make in the **Number of Duplicates** box.

   If you type a value greater than the number of assigned terminals, the extra duplicates will appear as gray terminal lines until they are assigned to a terminal.
6. Optionally, select **Assign to existing terminals**. The duplicates will automatically be assigned to the terminals defined for the port, beginning with the first terminal that does not yet have a defined terminal line.

   Clear this option if you want to manually assign the duplicates.

Note To manually assign a duplicate terminal line to a terminal, select **Assign Line #*n*** from the terminal's **Terminal Line** list.

7. Click **OK**.

   The duplicates are pasted along the vector you specified.

Modifying Terminal Lines

To *swap the coordinates* of a terminal line's start point and endpoints:

- Select **Swap Endpoints** from the terminal's **Terminal Line** pull-down list.

  The line's direction will be reversed.

To *delete* a defined terminal line:

- Select **None** from the terminal's **Terminal Line** pull-down list.

Setting up Differential Pairs

A differential pair represents two circuits, one positive and one negative, routed close together so they will pick up nearly the same amount of noise. The two signals are subtracted from each other by a receiver, yielding a much more noise-free version of the signal.

You can define one or more differential pairs from terminal lines defined on existing wave ports.

To set up a differential pair:

1. In the **Wave Port** dialog box, click the **Differential Pairs** tab.
2. Click **New Pair**.

   This adds existing pairs to the list of terminals, and sets default values for the Differential Mode and Common mode.
3. Select the positive terminal of the pair from the pull-down list.
4. Select the negative terminal of the pair from the pull-down list.
5. Under **Differential Mode**, do the following:
   a. Type a name for the differential mode in the **Name** text box.
   b. Either specify a real valued reference impedance for the differential mode in the **Ref. Z** text box or use the **Full Port Reference Impedance** text box and the **Set All Diff. Zref.** button or the **Set All Zref** button to set the values.

Note The value fields in the table support Ctrl/C to copy selected text from a cell, and Ctrl/V to paste text to a selected cell.

6. Under **Common Mode**, do the following:
   a. Type a name for the common mode in the **Name** text box.
   b. Either specify a real valued reference impedance for the common mode in the **Ref. Z** text box, or use the **Full Port Reference Impedance** text box and the **Set All Comm. Zref.** button or the **Set All Zref** button to set the values.

After HFSS has generated a solution, view the common and differential quantities of the differential pair under the **Matrix** tab of the **Solution Data** window.

Related Topics

Technical Notes: Computing Differential Pairs

Assigning Incident Waves

HFSS allows you to assign seven different types of incident wave sources.

1. Click **HFSS>Excitations>Assign>Incident Wave**.

The cascade menu allows you to select one of the following types of incident waves:

Plane Wave

Hertzian-Dipole Wave

Cylindrical Wave

Gaussian Beam

Linear Antenna Wave

Far Field Wave

Near Field Wave

Using Field Solutions from Other Simulations

HFSS can use field solutions from other simulations as sources for new simulations. The other simulations can be done in HFSS, in SIwave or in Maxwell3D. Some examples are (1) a detailed and optimized design of a cell phone radiating in a larger environment (HFSS-HFSS), (2) a complicated printed circuit board causing EMC/EMI problems in and around its housing (SIwave - HFSS) or (3) an electromechanical component causing EMC/EMI problems in a vehicle (Maxwell3D - HFSS). In all cases, radiated fields from the “source” project are imposed as an incident wave in the “target” project.

These radiated fields can both be far fields and near fields, depending on your judgment of what fits a particular situation. In the “target” project, they are defined through Incident Wave / Far Field Wave and Incident Wave / Near-Field Wave. There, the link to the “source” project can be established.

Also, in the “target” project, radiation boundaries with Advanced Options must be defined in order to specify where the fields from the “source” project enter the “target” project.

Incident Plane Wave

An incident Plane wave is a wave that propagates in one direction and is uniform in the directions perpendicular to its direction of propagation.

1. Click **HFSS>Excitations>Assign>Incident Wave>Plane Wave**.

   The **Incident Wave Source: General Data** page appears.
2. Type the source’s name in the **Name** text box or accept the default name.
3. Select the **Vector Input Format** as **Cartesian** or **Spherical** coordinates.
4. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
5. Click **Next**.

6. If you selected **Cartesian**, the **Incident Wave Source: Cartesian Vector Setup** page appears. Define the propagation vector, *k*, and the E-field polarization vector, E_0:
   a. Enter the X-, Y-, and Z-components for **k vector** in the **X**, **Y**, and **Z** boxes.
   b. Enter the coordinates for $\boldsymbol{E_0}$ **vector** in the **X**, **Y**, and **Z** boxes.

   A single incident wave will be defined. Continue with Step 8 below.

Note When entering the propagation vector, *k*, and E-field polarization vector, E_0, using Cartesian coordinates, keep the following guidelines in mind:

- To define an incident wave traveling in the positive z direction, enter (0, 0, 1) as the **k vector** coordinates.
- The magnitude of the E_0 vector cannot be zero.
- *k* must be orthogonal to E_0.

7. If you selected **Spherical**, the **Incident Wave Source: Spherical Vector Setup** page appears.
   a. Under **IWavePhi**, enter the following:

Start	The point where the rotation of ϕ begins.
Stop	The point where the rotation of ϕ ends.
Points	The number of points on the sweep of ϕ.

   Click **View Point List** to see the values of ϕ.
   b. Under **IWaveTheta**, enter values for **Start**, **Stop**, and **Points**.

   Click **View Point List** to see the values of θ.
   c. Enter the ϕ and θ components of $\boldsymbol{E_0}$ in the **Phi** and **Theta** boxes.

   A spherical grid is created when θ is swept through each ϕ point. At each grid point, an incident wave is present traveling towards the origin of the coordinate system for the design. The number of incident waves and grid points can be calculated by multiplying the number of ϕ points by the θ points.

Note Only a single incident wave angle can be defined for periodic structures which are defined with master and slave boundaries

8. Click **Next**. the **Incident Wave Source: Plane Wave Options** page appears.
9. Select the **Type of Plane Wave**.
   a. If you select **Regular/Propagating**, no other fields are active.
   b. If you select **Evanescent**, the **Propagation Constant** fields become active. Enter the **Real** and **Imaginary** parts of the Propagation Constant.
   c. If you select **Elliptically Polarized**, the **Polarization Angle** and **Polarization Ratio**

fields become active.

d. To restore the default (Regular/Propagating), click the **Use Defaults** button.

10. Click **Finish**.The incident wave you defined is added to the **Excitations** list in the **Project**.

Related Topics

Technical Notes: *Incident Waves*

Incident Hertzian-Dipole Wave

An incident Hertzian-Dipole wave simulates the field of an elementary short dipole antenna placed at the origin.

1. Click **HFSS>Excitations>Assign>Incident Wave>Hertzian-Dipole Wave**.

   The **Incident Wave Source: General Data** page appears.

2. Type the source's name in the **Name** text box or accept the default name.
3. Select the **Vector Input Format** as **Cartesian** or **Spherical** coordinates.
4. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
5. Click **Next**.
6. If you selected **Cartesian**, the **Incident Wave Source: Cartesian Vector Setup** page appears. Enter the X-, Y-, and Z-components for the vector **I*Dipole Length** in the **X**, **Y**, and **Z** boxes. **I** is the current amplitude (peak value). Units are Amp-meters (A*m).

   A single incident wave will be defined. Continue with Step 8 below.

7. If you selected **Spherical**, the **Incident Wave Source: Spherical Vector Setup** page appears.

   a. Under **IWavePhi**, enter the following:

Start	The point where the rotation of ϕ begins.
Stop	The point where the rotation of ϕ ends.
Points	The number of points on the sweep of ϕ.

   Click **View Point List** to see the values of ϕ.

   b. Under **IWaveTheta**, enter values for **Start**, **Stop**, and **Points**.

   Click **View Point List** to see the values of θ.

   c. Enter the ϕ and θ components of the vector **I*Dipole Length** in the **Phi** and **Theta** boxes. **I** is the current amplitude (peak value). Units are Amp-meters (A*m).

   A spherical grid is created when θ is swept through each ϕ point. At each grid point, an incident wave is present traveling towards the origin of the coordinate system for the design. The number of incident waves and grid points can be calculated by multiplying the number of ϕ points by the θ points.

Note Only a single incident wave angle can be defined for periodic structures which are defined with master and slave boundaries

8. Click **Next**. the **Incident Wave Source: Hertzian-Dipole Wave Options** page appears.
9. Select the **Radius of Surrounding Sphere**. Inside this sphere, the field magnitude will be made equal to the field magnitude calculated on the surface of the sphere. To restore the default (10 mm), click the **Use Defaults** button.
10. Click **Finish**.The incident wave you defined is added to the **Excitations** list in the **Project**.

Related Topics

Technical Notes: *Incident Waves*

Incident Cylindrical Wave

An incident Cylindrical wave is a wave that simulates the far field of an infinite line current placed at the origin.

1. Click **HFSS>Excitations>Assign>Incident Wave>Cylindrical Wave**.

   The **Incident Wave Source: General Data** page appears.
2. Type the source's name in the **Name** text box or accept the default name.
3. Select the **Vector Input Format** as **Cartesian** or **Spherical** coordinates.
4. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
5. Click **Next**.
6. If you selected **Cartesian**, the **Incident Wave Source: Cartesian Vector Setup** page appears. Enter the X-, Y-, and Z-components for the **I Vector** in the **X**, **Y**, and **Z** boxes. **I** is the current amplitude (peak value). Units are Amps (A).

   A single incident wave will be defined. Continue with Step 8 below.
7. If you selected **Spherical**, the **Incident Wave Source: Spherical Vector Setup** page appears.
   a. Under **IWavePhi**, enter the following:

Start	The point where the rotation of ϕ begins.
Stop	The point where the rotation of ϕ ends.
Points	The number of points on the sweep of ϕ.

   Click **View Point List** to see the values of ϕ.

   b. Under **IWaveTheta**, enter values for **Start**, **Stop**, and **Points**.

   Click **View Point List** to see the values of θ.

   c. Enter the ϕ and θ components of the **I Vector** in the **Phi** and **Theta** boxes. **I** is the current amplitude (peak value). Units are Amps (A).

   A spherical grid is created when θ is swept through each ϕ point. At each grid point, an

incident wave is present traveling towards the origin of the coordinate system for the design. The number of incident waves and grid points can be calculated by multiplying the number of ϕ points by the θ points.

Note Only a single incident wave angle can be defined for periodic structures which are defined with master and slave boundaries

8. Click **Next**. the **Incident Wave Source: Cylindrical Wave Options** page appears.
9. Select the **Radius of Surrounding Cylinder**. Inside this cylinder, the field magnitude will be made equal to the field magnitude calculated on the surface of the cylinder. To restore the default (10 mm), click the **Use Defaults** button.
10. Click **Finish**.The incident wave you defined is added to the **Excitations** list in the **Project**.

Related Topics

Technical Notes: *Incident Waves*

Incident Gaussian Beam Wave

An incident Gaussian Beam wave is a wave that propagates in one direction and is of Gaussian distribution in the directions perpendicular to its direction of propagation.

1. Click **HFSS>Excitations>Assign>Incident Wave>Gaussian Beam**.

   The **Incident Wave Source: General Data** page appears.
2. Type the source's name in the **Name** text box or accept the default name.
3. Select the **Vector Input Format** as **Cartesian** or **Spherical** coordinates.
4. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
5. Click **Next**.
6. If you selected **Cartesian**, the **Incident Wave Source: Cartesian Vector Setup** page appears. Define the propagation vector, *k*, and the E-field polarization vector, E_0:
   a. Enter the X-, Y-, and Z-components for **k vector** in the **X**, **Y**, and **Z** boxes.
   b. Enter the coordinates for $\boldsymbol{E_0}$ **vector** in the **X**, **Y**, and **Z** boxes.

   A single incident wave will be defined. Continue with Step 8 below.

Note When entering the propagation vector, *k*, and E-field polarization vector, E_0, using Cartesian coordinates, keep the following guidelines in mind:

- To define an incident wave traveling in the positive z direction, enter (0, 0, 1) as the **k vector** coordinates.
- The magnitude of the E_0 vector cannot be zero.
- *k* must be orthogonal to E_0.

7. If you selected **Spherical**, the **Incident Wave Source: Spherical Vector Setup** page appears.
   a. Under **IWavePhi**, enter the following:

Start	The point where the rotation of ϕ begins.
Stop	The point where the rotation of ϕ ends.
Points	The number of points on the sweep of ϕ.

      Click **View Point List** to see the values of ϕ.
   b. Under **IWaveTheta**, enter values for **Start**, **Stop**, and **Points**.
      Click **View Point List** to see the values of θ.
   c. Enter the ϕ and θ components of $\boldsymbol{E_0}$ in the **Phi** and **Theta** boxes.
      A spherical grid is created when θ is swept through each ϕ point. At each grid point, an incident wave is present traveling towards the origin of the coordinate system for the design. The number of incident waves and grid points can be calculated by multiplying the number of ϕ points by the θ points.

Note Only a single incident wave angle can be defined for periodic structures which are defined with master and slave boundaries

8. Click **Next**. The **Incident Wave Source: Gaussian Beam Options** page appears.
9. Select the **Beam Width at Focal Point**. To restore the default (10 mm), click the **Use Defaults** button.
10. Click **Finish**.The incident wave you defined is added to the **Excitations** list in the **Project**.

Related Topics

Technical Notes: *Incident Waves*

Incident Linear Antenna Wave

An incident linear antenna wave is a wave that simulates the far field of a linear antenna placed at the origin.

1. Click **HFSS>Excitations>Assign>Incident Wave>Linear Antenna Wave**.
   The **Incident Wave Source: General Data** page appears.
2. Type the source's name in the **Name** text box or accept the default name.
3. Select the **Vector Input Format** as **Cartesian** or **Spherical** coordinates.
4. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
5. Click **Next**.
6. If you selected **Cartesian**, the **Incident Wave Source: Cartesian Vector Setup** page appears. Enter the X-, Y-, and Z-components for the **I Vector** in the **X**, **Y**, and **Z** boxes. **I** is the antenna current amplitude (peak value). Units are Amps (A).

A single incident wave will be defined. Continue with Step 8 below.

7. If you selected **Spherical**, the **Incident Wave Source: Spherical Vector Setup** page appears.
   a. Under **IWavePhi**, enter the following:

Start	The point where the rotation of ϕ begins.
Stop	The point where the rotation of ϕ ends.
Points	The number of points on the sweep of ϕ.

   Click **View Point List** to see the values of ϕ.
   b. Under **IWaveTheta**, enter values for **Start**, **Stop**, and **Points**.
   Click **View Point List** to see the values of θ.
   c. Enter the ϕ and θ components of the **I Vector** in the **Phi** and **Theta** boxes. **I** is the antenna current amplitude (peak value). Units are Amps (A).
   A spherical grid is created when θ is swept through each ϕ point. At each grid point, an incident wave is present traveling towards the origin of the coordinate system for the design. The number of incident waves and grid points can be calculated by multiplying the number of ϕ points by the θ points.

Note Only a single incident wave angle can be defined for periodic structures which are defined with master and slave boundaries

8. Click **Next**. The **Incident Wave Source: Linear Antenna Wave Options** page appears.
9. Select the **Length of the Antenna**.
10. Select the **Radius of Surrounding Cylinder**. Inside this cylinder, the field magnitude will be made equal to the field magnitude calculated on the surface of the cylinder.
11. To restore the defaults (10 mm), click the **Use Defaults** button.
12. Click **Finish**.The incident wave you defined is added to the **Excitations** list in the **Project**.

Related Topics

Technical Notes: *Incident Waves*

Far Field Wave

A Far field wave is sufficiently far (that is, usually more than a wave length distance) from an antenna to approximate as a plane wave. Far field waves are mostly homogeneous.

1. Click **HFSS>Excitations>Assign>Far Field Wave**.
   The **Incident Wave Source:General Data** page appears.
2. Type the source name in the **Name** text box or accept the default name.
   If the coordinate system you are using in the source design (the project/design to which you are linking) is different from that in the target design (the design in which you are creating the link), you must define the relationship between those coordinate systems. The relationship

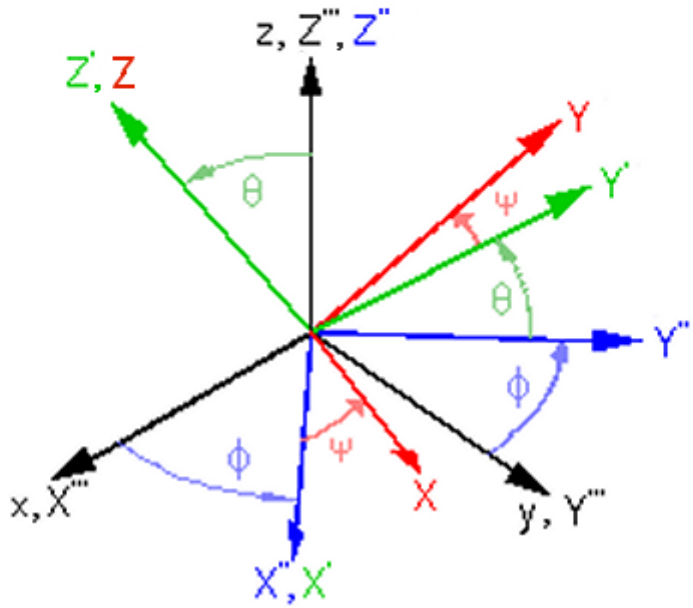
between two coordinate systems can always be defined as a translation and a rotation. The translation is the offset between the origins of the two coordinate systems, and the rotation can be defined through the use of Euler angles.

3. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave). This represents the translation of the source design's origin with respect to the target design's origin. For instance, if the source design's origin is located in the target design at (-2, -2, 1), then the translation between the two coordinate systems is (-2, -2, 1).
4. Click **Next**.

   A page appears that contains a **Setup Link** button to browse for a Source of Field and fields for specifying the linked design orientation relative to this design.
5. To specify the Source of Field, click the **Setup Link** button.

   This displays an **HFSS** window. It has three fields under the General tab: Project File, Design, and Solution.
6. Specify the Project file for the design that is the source of the Far Field wave. A browse button [...] lets you look through your file system. If you do not specify a project file, but select the current model, the current Project File is automatically filled in.
7. Specify the Design for the source of the Far Field wave. If the source is in the current design, you can select this from a drop down menu. If you select the current model, the Project File is automatically filled in.
8. Specify the Solution to use. A drop down list lets you select from the available solutions.
9. To view the External Field Link configuration and parameters, select each tab. Under the **External Field Link** configuration tab, you can set excitation magnitudes and phases in the source design. Under the **Parameters** tab, you can set the desired variable values in the source design.
10. Click **OK** to close the **HFSS** window and return to the **Incident Wave Source** window.
11. You can define the rotation of the source's coordinate system with respect to the target's coordinate system through the use of Euler angles. Similarly to the definition of translation, these angles represent the three rotations that the source design must undergo to align with the target design's coordinate system. Enter the Euler angles in the respective text fields and use the pull-down menus to specify the units (degrees or radians):
    - Phi (rotation about the Z-axis).
    - Theta (rotation about the X-axis)
    - Psi (rotation about resultant Z-axis.

x, y, z: Source coordinate system
X''', Y''', Z''': Target coordinate system (same as source)
X'', Y'', Z'' : Target coordinate system after rotation about Z''' axis
X', Y', Z' : Target coordinate system after rotation about X'' axis
X, Y, Z : Target coordinate system after rotation about Z' axis



12. Click **Finish** to close the dialog. The Far Field wave source point and direction is highlighted in the modeler window, and the wave appears in the Excitations list in the **Project**.

Related Topics

Technical Notes: *Incident Waves*

Clear Linked Data

Using Field Solutions from Other Simulators

Near Field Wave

A Near Field wave is close enough to the antenna source for near field effects to occur, typically within a wave length. Near field waves tend to be evanescent, that is, non-homogeneous.

1. Click **HFSS>Excitations>Assign>Near Field Wave**.

   The **Incident Wave Source:Near Field Wave** page appears.

2. Type the source name in the **Name** text box or accept the default name.

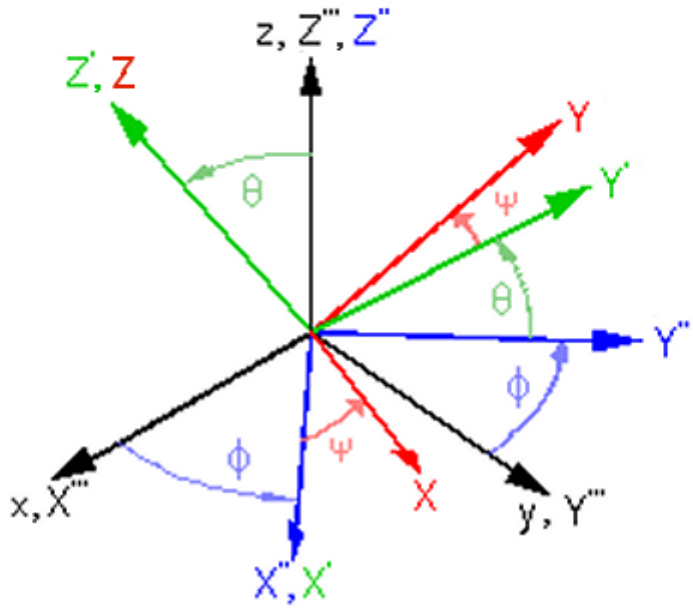
If the coordinate system you are using in the source design (the project/design to which you are linking) is different from that in the target design (the design in which you are creating the link), you must define the relationship between those coordinate systems. The relationship between two coordinate systems can always be defined as a translation and a rotation. The translation is the offset between the origins of the two coordinate systems, and the rotation can be defined through the use of Euler angles.

3. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave). Select the units for the coordinate values from the drop-down lists. This represents the translation of the source design's origin with respect to the target design's origin. For instance, if the source design's origin is located in the target design at (-2, -2, 1), then the translation between the two coordinate systems is (-2, -2, 1).
4. Click **Next**.

   A page appears that contains a **Setup Link** button to browse for a Source of Field and fields for specifying the linked design orientation relative to this design.
5. To specify the Source of Field, click the **Setup Link** button.

   This displays an **HFSS** window. It has three fields under the General tab: Project File, Design, and Solution.
6. Specify the Project file for the design that is the source of the Near Field wave. A drop down menu lets you select the current file, and a browse button [...] lets you look through your file system. If you select the current model, the current Design is automatically filled in.
7. Specify the Design for the source of the Near Field wave. If you select the current Project File, the Design is automatically filled in.
8. Specify the Solution to use. A drop down list lets you select from the available solutions.
9. To view the External Field Link configuration and parameters, select each tab. Under the **External Field Link** configuration tab, you can set excitation magnitudes and phases in the source design. Under the **Parameters** tab, you can set the desired variable values in the source design.
10. Click **OK** to close the **HFSS** window and return to the **Incident Wave Source** window.
11. You can define the rotation of the source's coordinate system with respect to the target's coordinate system through the use of Euler angles. Similarly to the definition of translation, these angles represent the three rotations that the source design must undergo to align with the target design's coordinate system. Enter the Euler angles in the respective text fields and use the pull-down menus to specify the units (degrees or radians):
    - Phi (rotation about the Z-axis).
    - Theta (rotation about the X-axis)
    - Psi (rotation about resultant Z-axis.

x, y, z: Source coordinate system
X''', Y''', Z''': Target coordinate system (same as source)
X'', Y'', Z'' : Target coordinate system after rotation about Z''' axis
X', Y', Z' : Target coordinate system after rotation about X'' axis
X, Y, Z : Target coordinate system after rotation about Z' axis



12. Click **Finish** to close the dialog. The Near Field wave source point and direction is highlighted in the modeler window, and the wave appears in the Excitations list in the **Project**.

Note

The Near field link uses a default mesh density on the surfaces that link to the other design. If this default mesh density is not sufficient to obtain a desired accuracy, you can select these surfaces and assign a surface mesh seeding.

Once the Near Field link has obtained the near fields from the other design, it continues to work with this information regardless of later mesh changes tthat resulted from adaptive passes or mesh operations. To enforce the Dynamic Link to use a newly seeded mesh, clear the linked data by using *Clear Linked Data*.

Related Topics

Technical Notes: *Incident Waves*

Clear Linked Data

Using Field Solutions from Other Simulators

Assigning Voltage Sources

Assign a voltage source when you want to specify the voltage and direction of the electric field on a surface. A voltage source is used when the feed structure is very small compared to the wavelength and a constant electric field may be assumed across the feed points. In this case, HFSS assigns a constant electric field across the gap on which you specified the voltage.

1. Select the object face to which you want to assign the voltage source.
2. Click **HFSS>Excitations>Assign>Voltage**.

   The **Voltage Source** dialog box appears.
3. Type the source's name in the **Name** text box or accept the default name.
4. Type the value of the source, in volts or amps, in the **Magnitude** box. You can assign a variable as this value.
5. Specify the direction of the electric field by drawing a vector:
   a. Select **New Line** from the **E-Field Direction** pull-down list.

      The **Voltage Source** dialog box disappears while you draw the vector.
   b. Select the start point of the line in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
   c. Select the endpoint of the line using the mouse or the keyboard.

      The endpoint defines the direction and length of the line.

   The **Voltage Source** dialog box reappears.
6. Click **OK**.

When the source is selected, an arrow indicates the direction and a letter (v or i) indicates the type of source.

Modifying Voltage Sources

To change the name, value, or electric field direction of an assigned voltage source:

1. Double-click the source's icon under **Excitations** in the project tree.

   The **Voltage Source** dialog box appears.
2. Edit the name or value of the source.
3. To reverse the direction of the e-field:
   - Select **Swap Endpoints** from the **E-Field Direction** pull-down list.

   The start and endpoints of the E-field line are switched; the line's direction is reversed.

Assigning Current Sources

Assign a current source when you want to define the magnitude and direction of the current flow through a surface. A current source is used when the feed structure is very small compared to the wavelength and the electric current on the surface is assumed to be constant across the feed points.

1. Select the object face to which you want to assign the current source.
2. Click **HFSS>Excitations>Assign>Current**.

   The **Current Source** dialog box appears.
3. Type the source's name in the **Name** text box or accept the default name
4. Type the value of the source, in volts or amps, in the **Magnitude** box. You can assign a variable as this value.
5. Specify the current flow direction by drawing a vector:
   a. Select **New Line** from the **Current Flow Direction** pull-down list.

      The **Current Source** dialog box disappears while you draw the vector.
   b. Select the start point of the line in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
   c. Select the endpoint of the line using the mouse or the keyboard.

      The endpoint defines the direction and length of the line.

   The **Current Source** dialog box reappears.
6. Click **OK**.

When the source is selected, an arrow indicates the direction and a letter (v or i) indicates the type of source.

Modifying Current Sources

To change the name, value, or current flow direction of an assigned current source:

1. Double-click the source's icon under **Excitations** in the project tree.

   The **Current Source** dialog box appears.
2. Edit the name or value of the source.
3. To reverse the direction of the current flow:
   - Select **Swap Endpoints** from the **Current Flow Direction** pull-down list.

   The start and endpoints of the current flow line are switched; the line's direction is reversed.

Assigning Magnetic Bias Sources

When you create a ferrite material, you must define the net internal field that biases the ferrite by assigning a magnetic bias source. The bias field aligns the magnetic dipoles in the ferrite, producing a non-zero magnetic moment.

1. Select the 3D ferrite object to which you want to assign the magnetic bias source.
2. Click **HFSS>Excitations>Assign>Magnetic Bias**.

   The **Magnetic Bias** wizard appears.
3. Type the source's name in the **Name** text box or accept the default name.
4. Specify whether the applied bias field is **Uniform** or **Non-uniform**.
5. If you selected the **Uniform** radio button, click **Next** and do the following:

   a. In the **Internal Bias** field, type the value of the ferrite in amperes/meters. You can assign a variable as this value.

   b. Enter the rotation of the permeability tensor with respect to the xyz-coordinate system in the **X Angle**, **Y Angle**, and **Z Angle** boxes. You can assign variables to these values.

   If you selected **Non-uniform**, select the **Setup** button to display the **Setup Link** dialog. Under the **General** tab, do the following:

   a. Type the name of a Maxwell 3D Field Simulator project in the **Project** box, or click the ellipsis [...] browse button display a file browser to select the project.

      HFSS uses the Maxwell 3D project as the source of the non-uniform magnetostatic field information during solution generation. Linking invokes a Maxwell 3D window to provide the solution for the targeted HFSS project.

   b. If there are multiple designs available for the project, you can select from the drop down menu.

   c. If there are multiple solutions available, you can select form the drop-down menu.

   The **Setup Link** dialog also contains a **Parameters** tab.

   The Parameter is available within the Maxwell 3D Field Simulator and the Value can (and often will) be a parameter in the HFSS Setup.
6. Click **Finish**.

   The magnetic bias source is assigned to the selected object. If you have set up a link, HFSS invokes a Maxwell 3D window to provide the solution for the targeted HFSS project.

You can also access and edit the magnetic bias source information via the **Properties** dialog for the source. Magnetic bias sources always have the lowest priority compared to boundaries and other excitations in the solver view.

Related Topics

Reprioritizing Boundaries and Excitations.

Technical Notes: *Magnetic Bias Sources*

Technical Notes: *Magnetic Saturation*

Modifying Excitations

To change the properties of an excitation, do one of the following:

- Double-click the excitation's icon under **Excitations** in the project tree.

  The excitation's dialog box appears, in which you can modify its properties.
- Right-click the excitation in the project tree, and then click **Properties** on the shortcut menu.

  The excitation's dialog box appears, in which you can modify its properties.
- On the **HFSS** menu, click **List**.

  The **Design List** dialog box appears. Under the **Excitations** tab, you can modify the properties of one or more boundaries.

Deleting Excitations

To delete *one excitation*:

1. Select the excitation you want to delete by clicking its icon in the project tree.
2. On the **Edit** menu, click **Delete**  .

To delete *all excitations*:

- On the **HFSS** menu, point to **Excitations**, and then click **Delete All**.

You can also *delete one or more excitations* in the **Design List** dialog box:

1. On the **HFSS** menu, click **List**.

   The **Design List** dialog box appears.
2. Under the **Excitations** tab, click the row of the excitation you want to delete.
3. Click **Delete**.

Reassigning Excitations

You can reassign an excitation to another surface. This is useful when you have modified objects with assigned excitations, invalidating the excitations. For example, if you unite two objects with assigned excitations, the second object's excitation will become invalid because united objects maintain the characteristics of the first object selected. In this case, you would need to reassign the excitation or delete it.

1. Select the object or object face to which you want to assign an existing excitation.
2. Click **HFSS>Excitations>Reassign**.

   The **Reassign Excitation** window appears.
3. Select an existing excitation from the list, and then click **OK**.

   The excitation is reassigned to the object or object face.

Note When reassigning an excitation that includes vectors in its definition, HFSS attempts to preserve the vectors with the new assignment, but this is not always possible.

Duplicating Excitations with Geometry

See *Duplicating Boundaries and Excitations with Geometry*.

Showing and Hiding Excitations

See *Setting Boundary and Excitation Visualization Options*.

Setting the Impedance Multiplier

For designs with ports.

If one or more symmetry planes have been defined or if only a wedge of a structure is modeled, you must adjust the impedance multiplier or the computed impedances will not be for the full structure.

1. Click **HFSS>Excitations>Edit Impedance Mult**.

   The **Port Impedance Multiplier** dialog box appears.

2. Type a value in the **Impedance Multiplier** box.

   You can assign a variable as this value.

3. Click **OK**.

Related Topics

Technical Notes: *Impedance Multipliers*

Renormalizing S-Matrices

You can renormalize an S-matrix to a specific port impedance when you set up a wave port. (It is the final step in the **Wave Port** wizard.) Or you can return to the **Wave Port** dialog box by double-clicking the wave port icon in the project tree, and then clicking the **Post Processing** tab.

To renormalize an S-matrix to a specific port impedance:

1. If you have already set up the wave port on the desired object face, double-click the wave port's icon in the project tree.

   The **Wave Port** dialog box appears.

2. Click the **Post Processing** tab.

   The **Port Renormalization** choices include:

   - **Do Not Renormalize** (the default)
   - **Renormalize All Modes**. This enables the **Full Port Impedance** text box. The default impedance for re-normalization of each port is 50 ohms.

     If you want to enter a complex impedence, enter it in the following form:

     <re> + <im>j

   - If there are multiple modes, **Renormalize Specific Modes**. This enables the Edit Mode Impedances button. This opens a editable table with the impedances for each mode.

3. Click **OK** to apply the selected values and close the dialog.

Note You do not need to re-run a simulation in order to renormalize a port. Post-processing reports are automatically updated to reflect the renormalized S-matrix.

Related Topics

Technical Notes: *Renormalized S-Matrices*

De-embedding S-Matrices

You can de-embed the port to a specific port impedance when you set up a wave port. (It is the final step in the **Wave Port** wizard.) Or you can return to the **Wave Port** dialog box by double-clicking the wave port icon in the project tree, and then clicking the **Post Processing** tab.

To compute a de-embedded S-matrix:

1. If you have already set up the wave port on the desired object face, double-click the wave port's icon in the project tree.

   The **Wave Port** dialog box appears.
2. Click the **Post Processing** tab.
3. Select **Deembed**, and then enter the length of the transmission line to be added in the **Distance** text box. A positive value de-embeds into the port. A negative value de-embeds out of the port.

   You can assign a variable as this value. After you enter the value, a blue arrow depicts the embedding distance in the graphics window while the port is selected.
4. Click **OK** to assign that length to the selected port.

Note You do not need to re-run a simulation in order to de-embed the S-matrix. Post-processing reports are automatically updated to reflect the de-embedded S-matrix.

Related Topics

Technical Notes: *De-embedded S-Matrices*

8

Assigning Materials

To assign a material to an object, follow this general procedure:

1. Select the object to which you want to assign a material.
2. On the **3D Modeler** menu, click **Assign Material**  .

   The **Select Definition** window appears. When the **Show all libraries** checkbox is selected, the window lists all of the materials in Ansoft's global material library as well as the project's local material library.

   You can also open the **Select Definition** window in one of the following ways:

   - In the **Properties** dialog box for the object, click the material name under the **Attributes** tab.
   - Right-click **Model** in the project tree, and then click **Assign Material** on the shortcut menu.
   - Right-click the object in the history tree, and then click **Assign Material** on the shortcut menu.
3. Select a material from the list.

Note You can search the listed materials by name or property value.

   If the material you want to assign is not listed, add a new material to the global or local material library, and then select it.
4. Click **OK**.

   The material you chose is assigned to the object.

Note In the history tree, by default, HFSS groups objects by material. To change the default, select the object icon and right-click to display the **Group Objects by Material** checkbox.

Solving Inside or on the Surface

When you assign a material to an object, you can specify whether to generate a field solution inside the object or on the surface of the object. If you elect to generate a solution inside the object, HFSS will create a mesh inside the object and generate a solution from the mesh. If you elect to generate a solution on the surface of the object, HFSS will create only a surface mesh for the object.

If you want a solution to be generated inside an object, select **Solve Inside** in the **Properties** window. Conversely, if you want a solution to only be generated on the surface of an object, clear the **Solve Inside** option in the **Properties** window.

By default, **Solve Inside** is selected for all objects with a bulk conductivity less than 10^5 siemens/meter and for perfect insulators. By default, the **Solve Inside** option in the **Properties** window is clear for perfect conductors.

To change the threshold for solving inside objects, do the following:

1. Under the **Tools** menu, point to **Options**, and then click **HFSS Options**.
2. Under the **General** tab, enter a new value in the **Solve Inside threshold** text box.

Searching for Materials

If there is a specific material or material property value that you want to assign to an object, you can search the materials in the **Select Definition** window by name or by material property.

Searching by Material Name

1. In the **Search Criteria** area of the **Select Definition** window, select **by Name**.
2. In the **Search Parameters** area, type a material name in the **Search by Name** text box.

   The row containing the material name most similar to the one you typed will be selected.

If the selected material is not the one you are searching for, do one of the following:

- Use the keyboard's arrow keys to scroll up or down the list of materials.
- Type a new material name in the **Search by Name** text box.

Searching by Material Property

1. In the **Search Criteria** area of the **Select Definition** window, select **by Property**.
2. Select a material property from the pull-down list:
3. In the **Search Parameters** area, type a value for the property in the **Search by Property** text box, and then click **Search**.

   The materials are sorted according to the value you entered. The material with the property value closest to the one you typed will be selected.

If the selected material is not the one you are searching for, do one of the following:

- Use the keyboard's arrow keys to scroll up or down the list of materials.
- Type a new value in the **Search by Property** text box.

Adding New Materials

You can add a new material to a project or global user-defined material library. To make the new project material available to all projects, you must export the material to a global user-defined material library.

1. In the **Select Definition** window, click **Add Material**.

   The **View/Edit Material** window appears.
2. Type a new name for the material in the **Material Name** text box or accept the default.
3. Select a material property type - **Simple** or **Anisotropic** - for each property from the **Type** pull-down list.
4. If the material is *linear*, enter values for the following material properties in the **Value** boxes:
   - Relative Permeability
   - Relative Permittivity
   - Bulk Conductivity
   - Dielectric Loss Tangent,
   - Magnetic Loss Tangent

   If the material is a *ferrite*, enter a value greater than 0 in the Magnetic Saturation **Value** box. You may also choose to enter values in the Lande G Factor and Delta H **Value** boxes.

Note You may enter a variable name or mathematical expression in the **Value** box.

5. If one or more of the material properties are dependent on frequency, click **Set Frequency Dependency**, and then follow the directions for defining frequency dependent materials.
6. To modify the units for a material property, double-click the **Units** box, and then select a new unit system.
7. Click **OK**.

   The new material is added to the material library.

Related Topics

Defining Variable Material Properties

Assigning Material Property Types

Defining Frequency-Dependent Material Properties

Assigning Material Property Types

Each material property can be assigned one of the following material property types:

Simple	The material is homogeneous and linear.
Anisotropic	The material's characteristics vary with direction.

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis.

Defining Anisotropic Relative Permeability Tensors

1. In the **Relative Permeability** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** pull-down list.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Relative Permeability** row.
2. Enter the relative permeability along one axis of the material's permeability tensor in the **Value** box of the **T(1,1)** row.
3. Enter the relative permeability along the second axis in the **Value** box of the **T(2,2)** row.
4. Enter the relative permeability along the third axis in the **Value** box of the **T(3,3)** row.

If the relative permeability is the same in all directions, use the same values for each axis.

These values can also be defined as variables.

Related Topics

Technical Notes: Anisotropic Relative Permeability Tensors

Defining Anisotropic Relative Permittivity Tensors

1. In the **Relative Permittivity** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** pull-down list.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Relative Permittivity** row.
2. Enter the material's relative permittivity along one tensor axis in the **Value** box of the **T(1,1)** row.
3. Enter the relative permittivity along the second axis in the **Value** box of the **T(2,2)** row.
4. Enter the relative permittivity along the third axis in the **Value** box of the **T(3,3)** row.

If the relative permittivity is the same in all directions, use the same values for each axis.

These values can also be defined as variables.

Related Topics

Technical Notes: Anisotropic Relative Permittivity Tensors

Defining Anisotropic Conductivity Tensors

1. In the **Bulk Conductivity** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** pull-down list.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Bulk Conductivity** row.
2. Enter the relative conductivity along one axis of the material's conductivity tensor in the **Value** box of the **T(1,1)** row.
3. Enter the relative conductivity along the second axis in the **Value** box of the **T(2,2)** row.
4. Enter the relative conductivity along the third axis in the **Value** box of the **T(3,3)** row.

The values of the relative conductivity along the first and second axis apply to all axes that lie in the xy cross-section being modeled. The values of the relative conductivity along the third axis applies to the z-component. These values affect current flowing in dielectrics between the conductors.

These values can also be defined as variables.

Related Topics

Technical Notes: Anisotropic Conductivity Tensors

Defining Anisotropic Dielectric Loss Tangent Tensors

If electric loss tangent is anisotropic, do the following:

1. In the **Dielectric Loss Tangent** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** pull-down list.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Dielectric Loss Tangent** row.
2. Enter the ratio of the imaginary relative permittivity to the real relative permittivity in one direction in the **Value** box of the **T(1,1)** row.
3. Enter the ratio of the imaginary relative permittivity to the real relative permittivity in the second direction in the **Value** box of the **T(2,2)** row.
4. Enter the ratio of the imaginary relative permittivity to the real relative permittivity in the third orthogonal direction in the **Value** box of the **T(3,3)** row.

If the electric loss tangent is the same in all directions, use the same values for each direction.

These values can also be defined as variables.

Related Topics

Technical Notes: Anisotropic Dielectric Loss Tangent Tensors

Defining Magnetic Loss Tangent Tensors

1. In the **Magnetic Loss Tangent** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** pull-down list.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Magnetic Loss Tangent** row.
2. Enter the ratio of the imaginary relative permeability to the real relative permeability in one direction in the **Value** box of the **T(1,1)** row.
3. Enter the ratio of the imaginary relative permeability to the real relative permeability in the second direction in the **Value** box of the **T(2,2)** row.
4. Enter the ratio of the imaginary relative permeability to the real relative permeability in the third direction in the **Value** box of the **T(3,3)** row.

If the magnetic loss tangent is the same in all directions, use the same values for each direction. These values can also be defined as variables.

Related Topics

Technical Notes: Anisotropic Magnetic Loss Tangent Tensors

Defining Variable Material Properties

When defining or modifying a material's properties, each material property value in the **View/Edit Material** window can be assigned a project variable. Simply type the project variable's name in the appropriate **Value** box. Project variables are used for material properties because materials are stored at the project level.

For example, define a project variable with the name **MyPermittivity** and define its value as **4**. To assign this property value to a material, type **$MyPermittivity** in the **Relative Permittivity Value** box for the material. Be sure to include the prefix **$** before the project variable name, which notifies HFSS that the variable is a project variable.

Defining Frequency-Dependent Material Properties

1. With respect to a material selected in the **Select Definition** window, in the **View/Edit Material** window, click **Set Frequency Dependency**.
2. In the **Frequency Dependent Material Setup Option** window, do one of the following:
   - Select **Piecewise Linear Input** to define a property value as a piecewise linear function. You will specify the property's values at frequencies below and above a frequency range. Based on these values, HFSS will automatically create a linear dataset that specifies the property's values at the desired frequencies during solution generation. This dataset can be modified with additional points if desired.
   - Select **Loss Model Input** if you are working with a lossy dielectric material with a lower frequency near DC. HFSS will enable you to specify the material's conductivity at DC, rather than its loss tangent, and specify the high frequency/optical permittivity.
   - Select **Enter Frequency Dependent Data Points** if you want to add frequency dependent data points or edit existing frequency dependent data points.
3. Click **OK**.
   - If you selected **Piecewise Linear Input**, the **Piecewise Linear Frequency Dependent Material Input** dialog box appears.
   - If you selected **Loss Model Input**, the **Loss Model Frequency Dependent Material Input** dialog box appears. In this case, follow the directions for specifying frequency dependence for a lossy dielectric material.
   - If you selected **Enter Frequency Dependent Data Points**, that dialog box appears. In this case, follow the directions for entering frequency dependent data points.
4. In the **Piecewise Linear Frequency Dependent Material Input** dialog box, enter a **Lower Frequency** value.

   HFSS assumes that the material's property values remain constant below this frequency.
5. Enter an **Upper Frequency** value.

   HFSS assumes that the material's property values remain constant above this frequency.
6. Enter the permittivity of the material at frequencies below the lower frequency in the **At Lower Frequency** text box.
7. Enter the permittivity of the material at frequencies above the upper frequency in the **At Upper**

Frequency text box.

If the permittivity of the material does not vary with frequency, enter the same value you entered for the permittivity's lower frequency.

8. Follow steps 5 and 6 for entering values for permeability, dielectric loss tangent, and magnetic loss tangent.
9. Click **OK**.

   You return to the **View/Edit Material** window. New default function names appear in the material property text boxes. HFSS automatically created a dataset for each material property. Based on a varying property's dataset, HFSS can interpolate the property's values at the desired frequencies during solution generation.

To modify the dataset with additional points, see *Modifying Datasets*.

Related Topics

Defining Frequency-Dependent Material Properties for Lossy Dielectrics

Enter Frequency Dependent Data Points

Technical Notes: Frequency-Dependent Material Properties

Modifying Datasets

Defining Frequency-Dependent Material Properties for Lossy Dielectrics

1. In the **View/Edit Material** window, click **Set Frequency Dependency**.
2. In the **Frequency Dependent Material Setup Option** window, select **Loss Model Input** if you are working with a lossy dielectric material with a lower frequency near DC. HFSS will enable you to specify the material's conductivity at DC, rather than its loss tangent, and specify the high frequency/optical permittivity.
3. Click **OK**.

   The **Loss Model Frequency Dependent Material Input** dialog box appears.
4. Enter a **Lower Frequency** value.

   HFSS assumes that the material's property values remain constant below this frequency.
5. Enter an **Upper Frequency** value.

   HFSS assumes that the material's property values remain constant above this frequency, unless otherwise specified for relative permittivity.
6. Under **Relative Permittivity**, do the following:
   a. Enter the permittivity of the material at frequencies below the lower frequency in the **At Lower Frequency** text box.
   b. Enter the permittivity of the material at frequencies above the upper frequency in the **At Upper Frequency** text box.

      If the material does not vary with frequency, enter the same value you entered for the permittivity's lower frequency.
   c. Optionally, to specify the high frequency/optical permittivity, select **At High/Optical**

Frequency, and then type the value in the text box.

7. Enter the permittivity of the material at frequencies below the lower frequency in the **At Lower Frequency** text box.
8. Under **Conductivity or Dielectric Loss Tangent**, do the following:
   a. If you prefer to specify the material's conductivity at DC, rather than its loss tangent value at the lower frequency, select **At DC (Conductivity)**, and then type the conductivity value at DC in the text box.
   b. If you prefer to specify the loss tangent value of the material at the lower frequency, rather than its conductivity at DC, select **At Lower Frequency (Loss Tangent)**, and then type the loss tangent value in the text box.
   c. Enter the **Upper Frequency (Loss Tangent)** value of the material in the text box.
9. Click **OK**.

   You return to the **View/Edit Material** window. New default function names appear in the material property text boxes.

Related Topics

Technical Notes: Frequency-Dependent Material Properties

Enter Frequency Dependent Data Points

Enter Frequency Dependent Data Points

1. When you click **OK** on the on after selecting **Enter Frequency Dependent Data Points** on the **Frequency Dependent Material Setup** dialog box, the **Enter Frequency Dependent Data** points dialog box appears. It shows a table with four columns:
   - **Name**: the name of the selected material property.
   - **Freq Dependent**: Check the box to indicate if the property is expressed as frequency-dependent dataset. If a property can not be set as frequency-dependent dataset, the cell is disabled.
   - **Dataset column**: this is disabled unless **Freq Dependent** is checked or the property cannot be set as frequency dependent. When enabled, it contains a dropdown menu with a list of existing datasets and the **Add/Import dataset...**to add or import new dataset.
   - **Freq As**: after a dataset is successfully imported or added, there are two choices available: "X datapoint" or "Y datapoint".
2. If you select **Add/Import dataset**, the **Add Dataset** dialog appears.

   This contains the following fields:
   - The name field for the current dataset. The default is ds1.
   - The **Import from File** button. This opens a file browser for you to select an existing dataset.
   - The **Coordinates** table. This contains X and Y text fields in which you can enter data points. The values you add are interactively displayed on the graph to the right of the table. You can also Add rows above or below a selected row, Delete rows, or Append a

specified number rows.

3. After you have specified or imported the data points, and **OK** the dialog, the **Enter Frequency Data Points** dialog shows the **Dataset Name** and the **Freq As** value.
4. After you **OK** the **Enter Frequency Dependent Data Points** dialog shows the new values.

Defining Material Properties as Expressions

When defining or modifying a material's properties, each material property value in the **View/Edit Material** window can be assigned a mathematical expression. Simply type the expression in the appropriate **Value** box. Expressions typically contain intrinsic functions, such as sin(x), and arithmetic operators, such as +, -, *, and /, but do not include project variables.

Defining Functional Material Properties

Any material property that can be specified by entering a constant can also be specified using a mathematical function. This is useful when you are defining a material property whose value is given by a mathematical relationship — for instance, one relating it to frequency or another property's value. When defining or modifying a material's properties, simply type the name of the function in the appropriate **Value** box.

Related Topics

Defining Mathematical Functions

Viewing and Modifying Material Attributes

1. In the **Select Definition** window, select the material you want to view or modify, and then click **View/Edit Materials**.

   The **View/Edit Material** window appears. The material name and its property values are listed.
2. If **Show all libraries** has not been selected, you mat need to select the libraries you want to view.
3. You can modify the material as follows:
   a. Type a new name for the material in the **Material Name** text box.
   b. Type new material property values in the **Value** boxes.
   c. Specify whether a material property is **Simple** or **Anisotropic**.
   d. Change the units for a material property.

   Note: Materials stored in Ansoft's global material library cannot be modified.
4. Click **OK** to save the changes and return to the **Select Definition** window.

Warning If you modify a material that is assigned in the active project after generating a solution, the solution will be invalid.

Validating Materials

HFSS can validate a material's property parameters for an Ansoft software product. For example, it will check if the range of values specified for each material property is reasonable.

If a material's property parameters are invalid, an error message will appear in the lower-right corner of the **View/Edit Material** window. If the parameters are valid, a green check mark will appear there.

To validate the material attributes listed in the **View/Edit Material** window:

- Select a product from the **Select Ansoft Product** area, and then click **Validate Now**.

Copying Materials

1. In the **Select Definition** window, select the material you want to copy, and then click **Clone Material**.
2. To modify the material's attributes, follow the directions for modifying materials.
3. Click **OK** to save the copy in the active project's material library.

Removing Materials

1. In the **Select Definition** window, select a material you want to remove from the active project's material library.
2. Click **Remove Material**.

   The material is deleted from the project material library.

Note The following materials cannot be deleted:

- Materials stored in Ansoft's global material library.
- Materials that have been assigned to objects in the active project.

In a project library, you may want to use the **Project>Remove Unused Definitions** command to remove selected materials definitions that your project does not require.

Exporting Materials to a Library

1. In the **Select Definition** window, select the material you want to export.
2. Click **Export Material to Library**.

   The **Export to material library** file browser appears.
3. Click **PersonalLib** to export the material to a local project directory, accessible only to the user that created it.

   Click **UserLib** to export the material to a a library that is shared by more than one user, usually in a central location.
4. Type the library's file name and then click **Save**.

Sorting Materials

You can change the order of the materials listed in the **Select Definition** window. You can sort the list of materials by name, library location, or material property value.

To change the order of the listed materials:

- Click the column heading by which you want to order the materials.

  If the arrow in the column heading points up, the material data will be listed in ascending order (1 to 9, A to Z) based on the values in the column you chose. If you want the material data to be listed in descending order (9 to 1, Z to A), click the column heading again. The arrow will point down.

Filtering Materials

If you want to remove certain materials or material properties from the list in the **Select Definition** window, use the filter options under the **Material Filters** tab. You can filter out materials based upon the product or library with which they are associated. You can also filter out material properties and types of material properties.

To filter materials or material properties listed in the **Select Definition** window:

1. Click the **Material Filters** tab.
2. Select one or more Ansoft products under **Filter Material by Product**. Only materials associated with the products you select will be listed in the **Select Definition** window.
   - Click **Select All** to select all of the products listed. Click **Clear** to clear all product selections.
3. Select one or more property types under **Filter Property Types**. Only the property types you select will be listed.
4. Select one or more material properties under **Select Material Properties**. Only the material properties you select will be listed.
5. Select one or more material types under **Filter Material Types**. Only the material types you select will be listed.
6. Select one or more material libraries under **Filter Material by Location**. Only the libraries you select will be listed.
7. Click the **Materials** tab to save your selections.

   Click **Cancel** to revert back to the last saved selections.

Working with Material Libraries

There are two different kinds of materials libraries in HFSS, a system library and a user library.

Working with Ansoft's System Material Library

HFSS provides you with a global, *or system library* of predefined materials. Global materials in the Ansoft system library are available in every HFSS project. They cannot be modified.

You can create a global system library that is stored in a common location and available to multiple users.

Working with User Material Libraries

You can create your own personalized global material library, or *user library*, that can be used in any HFSS project only by the user that created it. User-defined global materials can be modified at any time.

You can also create a personalized local user library that is used only in the active HFSS project.

Editing Libraries

There are two different methods of editing libraries.

- Using right-click on Materials in the project window to display the **Edit All Libraries** shortcut menu. Clicking displays the **Edit Libraries** window.

  Editing definitions from the project window does not modify the configured libraries for any particular design, since this is editing in general.

- Using **Tools>Edit Configured Libraries>Materials** from the menu bar takes the current design into account and adds any new libraries to the configured list for the design.

Configuring Libraries

Use **Tools>Configure Libraries** to display the **Configure Design Libraries** window. From this window you can view the available libraries for System, User, and Project, and which of these libraries has been configured. Set of selection arrows allows you to move a highlighted library to the Configured list. A checkbox permits you to specify a configured library as default.

Related Topics

Exporting Materials to a Library.

9

Assigning DC Thickness

You can select the **Assign DC Thickness** option to more accurately compute DC resistance of a thin conducting object for which **Solve Inside** is not selected. Skin impedance of the object will be calculated using the defined finite thickness. Otherwise, standard skin impedance calculations assuming infinite thickness will be applied to the object. This option also exists for finite conductivity boundaries.

The **Assign DC Thickness** option on the **HFSS** menu is enabled if at least one object contains a good conducting isotropic material (such as copper), and the **Solve Inside property** is not selected. If the object meets these conditions, you can assign a DC thickness.

1. Select **HFSS>Assign DC Thickness**.

   This displays the **Thickness of Objects for DC Resistance** dialog. Objects to which the thickness can be applied are listed in the **Object Name** column.

2. Select the objects to assign a value. You can select objects either by:

   - Clicking on the **Object Name** to highlight it.
   - Use the **Select By Name** field to type the object name, and click the **Search** button.

     The first object to match the name is highlighted.

   Selecting an object highlights the **Thickness** field and the **Set Thickness** button.

3. Enter a thickness value and select the units.

   This applies the value to the selected object and checks the **Use Thickness** property.

4. To change the value and uncheck the **Use Thickness** property, select the **Clear** button and then enter a different value. You can also manually select or deselect the box and manually enter or delete a thickness value in the table.

5. When you have assigned the values you need, click **OK** to close the dialogue.

Related Topics

Technical Notes: Calculating Finite Thickness Impedance

10

Modifying the Model View

You can modify the view of contents in the **3D Modeler** window without changing their actual dimensions or positions.

What do you want to do?

- Rotate the view.
- Pan the view.
- Zoom in or out.
- Fit contents in the view window.
- Show or hide objects.
- Show or hide boundaries or excitations.
- Render objects as wireframes, flat-shaded, or smooth-shaded solids.
- Modify the view orientation.
- Modify the lighting.
- Set the projection view.
- Set the background color.
- Modify the appearance of the coordinate system axes.
- Modify the appearance of the grid.

Related Topics

Assigning Color to an Object

Assigning Transparency to an Object

Rotating the View

To rotate the view

1. On the **View** menu, click **Rotate**  .
2. Drag the mouse in the direction you want to rotate the view.

   The view rotates until you release the mouse button.
3. To exit **Rotate** mode, click **Rotate** on the **View** menu again or press **ESC**.

Hint Alternatively, rotate the view using one of the following methods:

- Hold down the **ALT** key as you drag the mouse.
- Right-click in the view window, and then click **View>Rotate** on the shortcut menu.

To rotate the view around the vertical axis:

1. On the **View** menu, click **Spin**.
   - Alternatively, right-click in the view window, and then click **View>Spin**.
   - Or, click the spin icon on the toolbar 
2. Drag the mouse left or right at the speed you want to spin the view.

   The view spins continually in the direction and at the speed you dragged the mouse.
3. To stop spinning the view, click in the view window.
4. To end **Spin** mode, click **Spin** again on the **View** menu or press **ESC**.

To rotate the view around the screen center:

1. Click the rotate icon on the toolbar  .
2. Drag the mouse up and down at the speed you want to rotate the view.
3. To end **Rotate** mode, click the icon again or press **ESC**.
4.

Panning the View

To move (pan) the view:

1. On the **View** menu, click **Pan**  .
2. Drag the mouse in the direction you want to pan the view.

   The view will pan until you release the mouse button.
3. To exit **Pan** mode, click **Pan** on the **View** menu again or press **ESC**.

Hint Alternatively, pan the view using one of the following methods:

- Hold down the **SHIFT** key as you drag the mouse.
- Right-click in the view window, and then click **View>Pan** on the shortcut menu.

Zooming In and Out

You can magnify (zoom in) or shrink (zoom out) the contents in the view window using hot keys or mouse zoom mode.

To zoom in using hotkeys:

- Press the plus sign (+) or (=) keys or press Ctrl-E keys.

  The view zooms in 5 percent.

To zoom out using hotkeys:

- Press the minus sign (-) key or press the Ctrl-F keys.

  The view zooms out 5 percent.

To zoom using the mouse.

1. On the **View** menu, click **Zoom**  .
2. To zoom in, left click and hold, and drag the mouse towards the top of the view window. The objects in view expand as you drag.

   To zoom out, left click and hold and drag the mouse towards the bottom of the view window. The objects in view decrease in size as you drag.

   The absolute size of the model does not change.
3. To end **Zoom** mode, click **Zoom** on the **View** menu again or press **ESC**.

Hint Alternatively, zoom in or out on the view using one of the following methods:

- Hold down the **ALT**+**SHIFT** keys as you drag the mouse.
- Right-click in the view window, and then click **View>Zoom** on the shortcut menu.

Related Topics

Zooming In or Out on a Rectangular Area

Zooming In or Out on a Rectangular Area

To magnify or shrink a specific rectangular area in the view window:

1. On the **View** menu, click **Zoom In**  or **Zoom Out**  .
   - Alternatively, right-click in the view window, and then click **View>Zoom In** or **View>Zoom Out** on the shortcut menu.
2. Use the mouse to draw a rectangle (or square) by selecting two diagonally opposite corners.

   This is the area of magnification that will be increased or decreased.

   The rectangular area is magnified or decreases in size. The absolute size of the model does not change.
3. To end **Zoom** mode, click **Zoom In** or **Zoom Out** on the **View** menu again or press **ESC**.

Related Topics

Zooming In and Out

View Options: 3D UI Options

Use the **View>Options** command to open the **3D UI Options** dialog. This lets you set defaults for the following view options:

- Stereo Mode (default, disabled)
- Drag Optimization (default, disabled)
- Show Ansoft Logo in Prints (default, disabled)
- Default Color Key Height (the maximum number of values displayed)
- Where there is a selection options:
- Selection always visible (default, enabled)
    - Selection always visible
    - Set transparency of selected objects
    - Set transparency of non-selected objects.
- Default screen rotation about
        - Screen center (default)
        - Current axis
        - Model center.

Fitting Objects in the View Window

What do you want to do?

- Fit all objects in a view window.
- Fit selected objects in a view window.

Fitting All Objects in a View Window

1. On the **View** menu, point to **Fit All**.
2. On the **Fit All** menu, click one of the following commands:
   - **All Views**.

     All view windows displaying the active design will change to include all model objects.
   - **Active View**  .

     The view in the active 3D Modeler window changes to include all model objects.

Hint Alternatively, fit all objects in the *active* view window using one of the following methods:

- Press **CTRL**+**D**.
- Right-click in the view window, and then click **View>Fit All** on the shortcut menu.

Related Topics

Fitting a Selection in a View Window

Fitting a Selection in a View Window

1. Select the objects you want to fit in the view.
2. On the **View** menu, point to **Fit Selection**.
3. On the **Fit Selection** menu, click one of the following commands:
   - **Active View**  to fit the selected objects in the active view window.
   - **All Views** to fit the selected objects in every open view window of the active design.

Related Topics

Fitting All Objects in a View Window

Hiding Objects from View

To hide selected objects.

1. Select the object you want to hide from view.
2. On the **View** menu, point to **Hide Selection**.
3. On the **Hide Selection** menu, click one of the following commands:
   - **All Views** to hide the selected object in every open view window.
   - **Active View** to hide the selected object in the active view window.

   You can also use the **Hide** icons in the toolbar to hide selected objects in all views or the active view.

Hide selected objects in all views

Hide selected objects in active view

Hide/Show overlaid visualization in the active view icon

   The objects you selected are hidden.

If there are many objects, it may be easier to hide objects using the **Active View Visibility** dialog

1. On the **View** menu, point to **Active View Visibility**, or click the **Hide/Show** icon on the menu bar.

   The **Active View Visibility** dialog box appears.
2. Select the tab for the objects you want to show or hide. The dialog contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
   - For designs with large numbers of objects, you can resize the dialog for easier selection.
   - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
   - You can slso use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.
3. Under the tab, clear the **Visibility** option for the objects you want to hide in the active view window.

   The objects you designate are hidden.

Related Topics

Showing Objects

Showing Objects

To show *one or more objects* that are currently hidden:

1. On the **View** menu or on the menu bar icon, click **Active View Visibility**  .

   The **Active View Visibility** dialog box appears.
2. Select the tab for the objects you want to show or hide. The dialog contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.
   - For designs with large numbers of objects, you can resize the dialog for easier selection.
   - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
   - You can slso use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

   The objects you select and designate as Visible (by selecting the property or using Show) reappear.

To show *all objects* that are currently hidden:

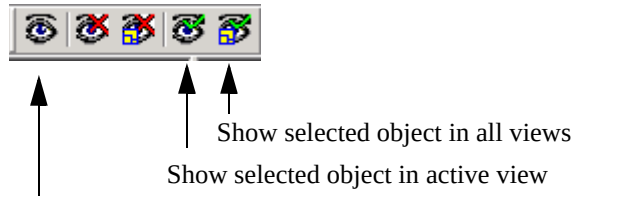
1. On the **View** menu, point to **Show All**.
2. On the **Show All** menu, click one of the following commands:
   - **All Views** to show all objects in every open view window
   - **Active Views** to show all objects in the active view window.

   The selected objects reappear.

To show *selected objects* that are currently hidden:

1. Select the object. Hidden items are selected once the node corresponding to them is clicked in the history pane
2. On the **View** menu, select **Show Selection**, and these click on of the following.
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window.

   You can also use the toolbar icons to **Show selected objects in all views** and **Show selected objects in active views**.



The selected objects reappear.

Rendering Objects as Wireframes or Solids

To render (display) *all* objects in the view window as wireframe outlines, flat-shaded solids, or smooth-shaded solids:

1. On the **View** menu, point to **Render**.
2. On the **Render** menu, click one of the following:
   - **Wireframe**.

     The objects in the view window are displayed as skeletal structures, enabling you to see all sides of the objects at one time.

     You can also use the **F6** key or the shade icon  to toggle the display to wireframe.
   - **Smooth Shaded**.

     The objects in the view window are displayed as shaded objects with smooth edges.

     You can also use the **F7** key or the shade icon  to toggle the display to smooth shaded.

To render *a single* object in the view window as a *wireframe outline*:

1. Select the object you want to render as a wireframe:
2. In the **Properties** dialog box, under the **Attribute** tab, select **Display Wireframe**.

Related Topics

Setting the Default View Rendering Mode

Setting the Default View Rendering Mode

To set a default rendering mode for all objects created in the active design and in future designs:

1. On the **Tools** menu, point to **Options**, and then click **3D Modeler** options.
2. Click the **Display** tab.
3. Select one of the following from the **Default View Render Mode** pull-down list.
   - **Wireframe**.

     The objects in the view window will be displayed as skeletal structures, enabling you to see all sides of the objects at one time.
   - **Smooth Shaded**.

     The objects in the view window will be displayed as shaded objects with smooth edges.

   The rendering mode will be applied to all new objects you create.

Modifying the View Orientation

To change the orientation of the view (the viewing direction) in the view window:

1. On the **View** menu, point to **Modify Attributes** and then click **Orientation**.

   A dialog box with orientation settings appears.
2. Apply a default orientation to the view or create and apply a new orientation.
3. Click **Apply** for the selected view to appear in the view window.
4. Click **Make Default** if you want the selected viewing direction to be the initial viewing direction when a **3D Modeler** window is opened, either in the current project or future projects.
5. Click **Close** to dismiss the dialog box.

   The orientation you set will be saved with the design. New orientations assigned to other designs after this point will not affect this orientation.

Related Topics

Applying a Default View Orientation

Applying a New Orientation

Applying a Default View Orientation

To apply a default viewing direction to the active view window:

1. On the **View** menu, point to **Modify Attributes** and then click **Orientation**.

   A dialog box with orientation settings appears.
2. Click one of the orientation names listed in the viewing directions list.
3. To view the associated vector components for the orientation you clicked, select **Input vector components** under **Add Orientation to List**.

   The Vx, Vy, and Vz components will be displayed in the text boxes to the right.
4. To view the associated input angles for the orientation you clicked, select **Input angles** under **Add Orientation to List**.

   The phi and theta components of the selected orientation will be listed in the text boxes to the right.
5. Click **Apply**.

   The viewing direction will be applied to the active view window.

Applying a New View Orientation

To apply a new viewing direction to the active view window:

1. On the **View** menu, point to **Modify Attributes** and then click **Orientation**.

   A dialog box with orientation settings appears.
2. To create a viewing direction that is based on a default viewing direction, click the existing orientation name in the viewing directions list.

   To create a viewing direction based on the current view in the **3D Modeler** window, click **Get**

Current View Direction.

3. To modify the selected orientation's vector components, select **Input vector components** under **Add Orientation to List**, and then modify the values in the **Vx**, **Vy**, or **Vz** text boxes.
4. To modify the selected orientation's input angles, select **Input angles** under **Add Orientation to List**, and then modify the values in the phi and theta text boxes.
5. Type a name for the new orientation in the **Name** text box.
6. Click **Add/Edit**.

   The new orientation is added to the list of viewing directions.
7. Click **Make Default** if you want the new viewing direction to be the initial viewing direction when a **3D Modeler** window is opened in the current project or future projects.

Removing an Orientation

To remove a viewing direction from the list in the orientation settings dialog box:

1. On the **View** menu, point to **Modify Attributes** and then click **Orientation**.

   A dialog box with orientation settings appears.
2. Click the viewing direction you want to delete from the list of names.
3. Click **Remove**.

   The viewing direction is removed from the list.

This operation cannot be undone.

Modifying the Lighting

You have the option to emit the following types of light on a design:

- *Ambient* lighting surrounds the model evenly with light. All objects are lit evenly in every direction by a color of light that you specify.
- *Distant* lighting directs a ray of light at the model in a direction you specify. By default, two distant light vectors are in effect for every new view window.

To modify the lighting:

1. On the **View** menu, point to **Modify Attributes** and then click **Lighting**.

   The **Lighting Properties** dialog box appears.
2. Select **Do Not Use Lighting** to turn off ambient and distant lighting.

   Clear this option to activate ambient and distant lighting.
3. To surround the model with light, click the **Ambient Light Properties** color button, and then select a color for the surrounding light from the **Color** palette.
4. To modify the distant light on a model, do one of the following:
   a. Add a new distant light by clicking **Add**.
   b. Copy an existing distant light that you intend to modify by first selecting it in the **Distant Light Vectors** table, and then clicking **Clone**.
   c. Select a default distant light to modify by selecting it in the **Distant Light Vectors** table.
5. For the selected distant light vector, specify the vector direction:
   a. To modify the direction by specifying Cartesian coordinates, do one of the following:
      - Enter the new Cartesian coordinates in the **X**, **Y**, and **Z** boxes.
      - Use the **Vx**, **Vy**, and **Vz** sliders to specify the Cartesian coordinates dynamically.
   b. To modify the direction by specifying the spherical coordinates, do one of the following:
      - Enter the new spherical coordinates in the ϕ and θ boxes.
      - Use the ϕ and θ sliders to specify the spherical coordinates dynamically.
6. Click **Reset** to revert to the default ambient and distant light settings.
7. Click **Save As Default** if you want the new lighting settings to be the defaults for all **3D Modeler** windows, either in the current project or future projects.
8. Click **Close** to dismiss the dialog box.

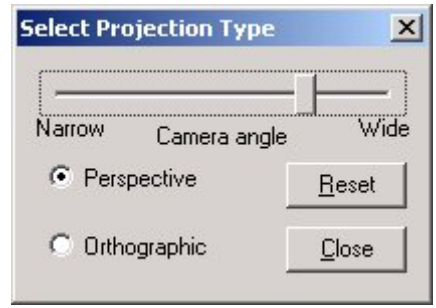
The lighting settings will be saved with the design. New lighting applied to other designs after this point, including new default settings, will not affect these lighting settings.

Setting the Projection View

To modify the projection of model objects (the camera angle) in the view window:

1. On the **View** menu, point to **Modify Attributes** and then click **Projection**.

   The **Select Projection Type** window appears:



2. Select **Perspective** to change the angle of the view.
   - Move the slider to the right to increase the proximity, or widen, the view. Move the slider to the left to decrease the proximity, or flatten, the view.

   Objects that are closer appear larger relative than objects that are farther away.
3. Select **Orthographic** to view the model without distortion.

   The slider is disabled because a distortion scale is no longer applicable.
4. Click **Reset** to return the model to its original view.
5. Click **Close** to accept the projection setting and dismiss the window.

   The **Select Projection Type** window closes. The last view you specified in the projection window remains visible in the view window.

The projection view you set will be saved with the design. New projection views assigned to other designs after this point will not affect this projection setting.

Setting the Background Color

To set the color of the background in the view window:

1. On the **View** menu, point to **Modify Attributes** and then click **Background color**.

   The **Select Background Color** window appears.

2. To assign a solid background color, do the following:
   a. Select **Plain Background**.
   b. Modify the background color in one of the following ways:
      - Click the **Background Color** button and then select a color from the **Color** palette.
      - Use the RGB sliders under **Change View Color Dynamically** to specify the color's red, green, and blue values.
3. To assign a background color that gradually changes from one color to another, do the following:
   a. Select **Gradient Background**.
   b. Specify the background color at the top and bottom of the view window in one of the following ways:
      - Under **Select Background Type**, click the **Top Color** button and select a color from the **Color** palette. Then click the **Bottom Color** button and select a color from the **Color** palette.
      - Under **Change View Color Dynamically**, click **Top Color** or **Bottom Color** and use the RGB sliders to specify the color's red, green, and blue values.
4. Click **Reset** to revert to the default background colors.
5. Click **Save As Default** if you want the new background color to be the background color for all **3D Modeler** windows in either the current project or future project.

The background color you set will be saved with the design. New background color settings assigned to other designs after this point, including new default settings, will not affect this design.

Modifying the Coordinate System Axes View

What do you want to do?

- Show or hide the coordinate system axes.
- Show the coordinate system axes for selected objects.
- Enlarge or shrink the size of the coordinate system axes.

Showing or Hiding the Axes

1. On the **View** menu, point to **Coordinate System**.
2. On the **Coordinate System** menu, click one of the following:
   - **Hide** to hide the x-, y-, and z-axes in the active view window.
   - **Show** to display the x-, y-, and z-axes in the active view window.

Show the Axes for Selected Objects

1. On the **Tools** menu, point to **Options**, and then click **HFSS Options**.
2. Select **Show orientation of selected objects**.

Enlarging or Shrinking the Axes

1. On the **View** menu, point to **Coordinate System**.
2. On the **Coordinate System** menu, click one of the following:
   - **Large** to display the x-, y-, and z-axes as extending to the edges of the active view window.
   - **Small** to display the x-, y-, and z-axes in a smaller size in relative to the edges of the active view window.

Choosing Grid Settings

The grid displayed in the **3D Modeler** window is a drawing aid that helps to visualize the location of objects. The points on the grid are divided by their local x-, y-, and z-coordinates for Cartesian grids, or by their local radius and angle coordinates for polar grids. Grid spacing is set according to the current project's drawing units.

You can control the following aspects of the grid:

- Type (rectangular or circular)
- Style (dots or lines)
- Density
- Spacing
- Visibility
- Snap settings
- Grid plane

Setting the Grid Type

1. On the **View** menu, click **Grid Settings**.

   The **Grid Settings** window appears.

2. Select a grid type for the active view window: **Cartesian** for a rectangular grid or **Polar** for a circular grid.

   The grid in the active view window is centered at the origin of the working coordinate system.

For Cartesian grids, you will define a coordinate by specifying its distance from the origin along each axis in the **X**, **Y**, and **Z** text boxes or its relative distance from the previously selected point in the **dX**, **dY**, and **dZ** text boxes.

For polar grids, you will define a coordinate by specifying its radius from the origin in the **R** text box and its angle from the x-axis in the **Theta** text box or its relative distance from the previously selected point in the **dR** and **dTheta** text boxes.

Setting the Grid Style

1. On the **View** menu, click **Grid Settings**.

   The **Grid Settings** window appears.

2. Select one of the following grid styles for the active view window:

Dot	Displays each grid point as a dot.
Line	Displays lines between grid points.

Setting the Grid Density and Spacing

1. On the **View** menu, click **Grid Settings**.

   The **Grid Settings** window appears.

2. If you want to change the density of the grid in the active view window as you zoom in or out on objects, do the following:
   a. Select **Auto adjust density to**.
   b. Specify a distance between grid points by typing a value in the **pixels** box.

      The default is set to 30 pixels, which is generally the best setting for displaying objects.
3. If you do not want the grid density to change when you zoom in or out, but instead want to specify a constant grid spacing, do the following:
   a. Clear the **Auto adjust density to** option.
   b. Specify the grid's spacing in the active design's units.

      If you selected a Cartesian grid type, type the values of **dX**, **dY**, and **dZ**. These values represent the difference between one grid point and the next in the x, y, and z directions, respectively.

      If you selected a polar grid type, type the values for **dR** and **dTheta**. **dR** represents the difference between each radius. **dTheta** is the difference between angles.

   The distance between grid points will increase and decrease proportionately as you zoom in and out in the active view window.

Setting the Grid's Visibility

- To hide the grid, click the Grid toolbar icon:  . Click it again to show the grid.

Alternatively:

1. On the **View** menu, click **Grid Settings**.

   The **Grid Settings** window appears.
2. Select **Grid Visible** to make the grid visible in the active **3D Modeler** window.

   Clear the selection to make the grid invisible.

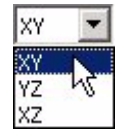
Related Topics

Setting the Grid Plane

Setting the Grid Plane

To specify the plane on which you want to display the grid in the active view window, do one of the following:

- On the **3D Modeler** menu, point to **Grid Plane**, and then select a grid plane: **XY**, **YZ**, or **XZ**.
- Click a grid plane on the pull-down list on the 3D Modeler Draw toolbar:



11

Defining Mesh Operations

In HFSS, mesh operations are optional mesh refinement settings that provide HFSS with mesh construction guidance. This technique of guiding HFSS's mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Mesh Operations** commands on the **HFSS** menu.

You can instruct HFSS to refine the length of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can instruct HFSS to refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement). These types of mesh operations are performed on the *current mesh*, that is, the most recently generated mesh.

In a few circumstances, you may also want to create a mesh operation that modifies HFSS's surface approximation settings for one or more faces. Surface approximation settings are only applied to the *initial mesh*, that is, the mesh that is generated the first time a design variation is solved.

See the technical notes for more details about HFSS's application of mesh operations.

What do you want to do?

- Perform length-based mesh refinement on object faces.
- Perform length-based mesh refinement inside objects.
- Perform skin depth-based mesh refinement on object faces.
- Modify surface approximation settings for one or more faces.
- Specify the model resolution for a selection.

Related Topics

Technical Notes: The Mesh Generation Process

Technical Notes: Seeding the Mesh

Technical Notes: Guidelines for Seeding the Mesh

Technical Notes: Surface Approximation Settings

Assigning Length-Based Mesh Refinement on Object Faces

1. Select the faces you want HFSS to refine.

   Alternatively, select an object if you want HFSS to refine every face on the object.
2. Click **HFSS>Mesh Operations>Assign>On Selection>Length-Based**.

   The **Element Length-Based Refinement** dialog box appears.
3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of tetrahedra edges touching the faces:
   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the tetrahedral edges touching the faces in the **Maximum Length of Elements** text box.

      HFSS will refine the element edges touching the selected faces until their lengths are equal to or less than this value.

      The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

      A maximum length of $\frac{\sqrt{2\lambda}}{10}$ is recommended for radiation boundary surfaces.
5. To restrict the number of elements added during refinement of the faces:
   a. Select **Restrict the Number of Elements**.
   b. Enter the **Maximum Number of Elements** to be added.
   c. Click **OK**.

When the mesh is generated, the refinement criteria you specified is used. When the maximum number of elements is reached, some elements may exceed the requested maximum element length.

Related Topics

Technical Notes: Length-Based Mesh Refinement

Technical Notes: Seeding the Mesh

Technical Notes: Guidelines for Seeding the Mesh

Assigning Length-Based Mesh Refinement Inside Objects

Applying Mesh Operations without Solving

Technical Notes: The Mesh Generation Process

Assigning Length-Based Mesh Refinement Inside Objects

To instruct HFSS to refine every face of an object and its interior:

1. Select the object you want HFSS to refine.
2. Click **HFSS>Mesh Operations>Assign>Inside Selection>Length-Based**.

   The **Element Length-Based Refinement** dialog box appears.
3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of the tetrahedral element edges inside the object:
   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the edges inside the object in the **Maximum Length of Elements** text box.

      The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected object's faces.

      HFSS will refine the element edges inside the object until they are equal to or less than this value.
5. To restrict the number of elements added during the refinement inside the object:
   a. Select **Restrict the Number of Elements**.
   b. Enter the **Maximum Number of Elements** to be added.
   c. Click **OK**.

When the mesh is generated, the refinement criteria you specified will be used. When the maximum number of elements are reached, it may result in some elements exceeding the requested maximum element length.

Related Topics

Technical Notes: Length-Based Mesh Refinement

Technical Notes: Seeding the Mesh

Technical Notes: Guidelines for Seeding the Mesh

Assigning Length-Based Mesh Refinement on Object Faces

Applying Mesh Operations without Solving

Technical Notes: The Mesh Generation Process

Assigning Skin Depth-Based Mesh Refinement on Object Faces

1. Select the faces you want to be refined.

   Alternatively, select an object if you want HFSS to refine every face on the object.
2. Click **HFSS>Mesh Operations>Assign>On Selection>Skin-Depth-Based**.

   The **Skin Depth-Based Refinement** dialog box appears.
3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. Type the skin depth within which to refine the mesh in the **Skin Depth** text box.

   Alternatively, calculate the skin depth based on the object's material permeability and conductivity and the frequency at which the mesh will be refined:

   a. Click **Calculate Skin Depth**.

      The **Calculate Skin Depth** dialog box appears.

   b. Enter the material's **Relative Permeability** and **Conductivity**.

   c. Specify the **Frequency** at which to refine the mesh.

   d. Click **OK**.

      HFSS calculates the skin depth and enters its value in the **Skin Depth** text box.
5. In the **Number of Layers of Elements** text box, type the number of layers to add perpendicular to the object's surface.

   HFSS will add an equivalent number of mesh points to each layer. For example, if HFSS added 10 points to satisfy the **Surface Triangle Length**, it will add 10 points to each layer.
6. Type the maximum edge length of the surface mesh in the **Surface Triangle Length** text box.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

   HFSS will refine the surface triangle mesh (the faces of the tetrahedra touching the surface) until their edge lengths are equal to or greater than the specified value.
7. To restrict the number of elements added during refinement on the faces:

   a. Select **Restrict the Number of Surface Elements**.

   b. Enter the **Maximum Number of Surface Elements** to be added.

   c. Click **OK**.

When the mesh is generated, the refinement criteria you specified will be used.

Related Topics

Technical Notes: Skin Depth-Based Mesh Refinement

Technical Notes: Seeding the Mesh

Technical Notes: Guidelines for Seeding the Mesh

Applying Mesh Operations without Solving

Technical Notes: The Mesh Generation Process

Modifying Surface Approximation Settings

HFSS applies surface approximation settings when it generates the initial mesh. If you modify HFSS's default settings after the initial mesh has been generated, they will not affect the mesh for that design variation.

1. Select the faces for which you want to modify the surface approximation settings.
   - Alternatively, select an object if you want to modify the surface approximation settings of every face on the object.
2. Click **HFSS>Mesh Operations>Assign>Surface Approximation**.

   The **Surface Approximation** dialog box appears.
3. Type a name for the group of settings in the **Name** text box or accept the default name.
4. Under **Surface Deviation**, do one of the following:
   - Select **Ignore** if you do not want to use surface deviation settings for the selected faces.
   - Select **Set maximum surface deviation (length)**, and then type the distance between the true surfaces of the selected faces and the meshed faces in the text box.
5. Under **Normal Deviation**, do one of the following:
   - Select **Ignore** if you do not want to use HFSS's default normal deviation settings for the selected faces.
   - Select **Use defaults** if you want to use HFSS's default normal deviation setting for the selected faces, which is 22.5 degrees.
   - Select **Set maximum normal deviation (angle)**, and then type the angular distance between the normal of the true surface and the corresponding mesh surface in the text box.
6. Under **Aspect Ratio**, do one of the following:
   - Select **Ignore** if you do not want to use HFSS's default aspect ratio settings for the selected faces.
   - Select **Use defaults** if you want to use HFSS's default aspect ratio settings for the selected faces, which are 10 for curved surfaces and 200 for planar surfaces.
   - Select **Set aspect ratio**, and then type a value in the text box. This value determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.
7. Click **OK**.

   The settings will be applied to the initial mesh generated on the selected surface. The group of settings is listed in the project tree under **Mesh Operations**.

Related Topics

Technical Notes: Surface Approximation Settings

Technical Notes: Guidelines for Modifying Surface Approximation Settings

Technical Notes: The Mesh Generation Process

Specifying the Model Resolution

You can set Model Resolution on one or more objects to remove unnecessary details from the mesh representation. This can be used to reduce the mesh complexity of the selected objects.

1. Select the object or objects on which to specify a Model Resolution length.
2. Click on **HFSS>Mesh Operations>Assign>Model Resolution**.
3. This displays the **Model Resolution Mesh Operation** dialog.

   Alternatively, you can display the same dialog if you:

   a. Right-click on either **Mesh Operations** in the Project Tree, or right-click in the **3D Modeler** window to display the respective shortcut menu.

   b. Click on **Assign>Model Resolution** in the **Project Tree** menu or click on **Assign Mesh Operation>Model Resolution** on the shortcut menu.

   The **Model Resolution Mesh Operation** dialog contains text fields for the Name and Model Resolution length, and it has a drop down menu for the units.
4. Specify the name, the value, and the units.
5. Click **OK**.

   This adds the Model Resolution under the **Mesh Operations** icon in the Project Tree.

Note

Setting Model Resolution will invalidate any existing solutions.

When two objects in contact have different model resolution lengths, the smaller length will apply for the common regions.

Related Topics

Technical Notes: *Model Resolution*

Reverting to the Initial Mesh

The initial mesh is the mesh that is generated the first time a design variation is solved. It includes surface approximation settings, but does not include lambda refinement or defined mesh operations.

If you have modified the design setup, and do not want to use the existing current mesh, revert to the initial mesh prior to solving.

- On the **HFSS** menu, point to **Analysis Setup**, and then click **Revert to Initial Mesh**.

Reverting to the initial mesh is useful when you want to evaluate how a different solution frequency affects the mesh generated during an adaptive analysis.

Related Topics

Technical Notes: The Mesh Generation Process

Applying Mesh Operations without Solving

If you want to refine the mesh on a face or volume, but do not want to generate a solution, do the following after defining mesh operations:

- On the **HFSS** menu, point to **Analysis Setup**, and then click **Apply Mesh Operations**.

  If a current mesh has been generated, HFSS will refine it using the defined mesh operations.

  If a current mesh has not been generated, HFSS will apply the mesh operations to the initial mesh.

  If an initial mesh has not been generated, HFSS will generate it and apply the mesh operations to the initial mesh.

  If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.

Hint Define a new mesh operation rather than modify an existing mesh operation. HFSS will not re-apply a modified mesh operation.

Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them.

Related Topics

Technical Notes: The Mesh Generation Process

12

Specifying Solution Settings

Specify how HFSS will compute a solution by adding a *solution setup* to the design. You can define more than one solution setup per design. Each solution setup includes the following information:

- General data about the solution's generation.
- Adaptive mesh refinement parameters, if you want the mesh to be refined iteratively in areas of highest error.
- Frequency sweep parameters, if you want to solve over a range of frequencies.

To add a new solution setup to a design:

1. Select a design in the project tree.
2. On the **HFSS** menu, point to **Analysis Setup**, and then click **Add Solution Setup**  .
   - Alternatively, right click **Analysis** in the project tree, and then click **Add Solution Setup** on the shortcut menu.
   - If you have already created a solution and you want to use an existing mesh, you can click **Add Dependent Setup**.

   The **Solution Setup** dialog box appears. It is divided among the following tabs:

General	Includes general solution settings.
Options	Includes settings for lambda refinement, adaptive analysis and solution options.
Advanced	Includes settings for mesh linking, output variable convergence, absorbing boundaries on ports, and waveport adapt options.
Defaults	Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

3. Click the **General** tab.

4. Enter a **Setup Name** or accept the default.
5. For Driven solution types, do the following:
   a. Enter the **Solution Frequency** in the frequency units.
   b. Optionally, select **Solve Ports Only**.

   For Eigenmode solution types, do the following:
   a. Enter the **Minimum Frequency** in the frequency units.
   b. Enter the **Number of Modes**. The number must be greater than 0 and less than 20.
6. If you are performing an adaptive analysis, enter **2** or more passes in the **Maximum Number of Passes** box, and then specify the remaining adaptive analysis parameters.

   If you are not performing an adaptive analysis, entering **0** will enable you to bypass the adaptive analysis process and just perform a frequency sweep. Entering **1** will also bypass adaptive analysis, generating a solution only at the solution frequency you specified.
7. Click **OK**.
8. Optionally, add a frequency sweep to the solution setup.

Related Topics

Add Dependent Setup

Technical Notes: The HFSS Solution Process

Add Dependent Setup

To apply all settings from an existing setup to a child setup:

1. Select an existing setup in the project tree.
2. On the **HFSS** menu, right click on the setup in the project tree, and then click **Add Dependent Solution Setup** on the shortcut menu.

   A dependent setup icon appears, which has an altered graphic to distinguish it from the parent setup icon. The child setup name is "*parent_setup name*_1. All of the settings from the parent setup are copied to the child setup. The dependent setup uses the mesh from the parent setup. This is shown under the **Advanced** tab of the **Solution Setup** dialog, **Specifying a Source** for the initial mesh. You can add a dependent setup to another dependent setup, and form of the name shows the hierarchical dependence by appending "_1" to show further dependence.

If you intend to change any of the settings, you do this just as you would for a new setup.

Related Topics

Specifying Solution Settings

Setting the Solution Frequency

For Driven solution types.

For every Driven solution setup, specify the frequency at which to generate the solution. If you want to solve over a range of frequencies, define a frequency sweep. If a frequency sweep is solved, an adaptive analysis is performed only at the solution frequency.

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Solution Frequency** in the frequency units.

Note For Fast sweeps, HFSS uses the solution frequency as the center frequency if it is within the frequency range (greater than the start frequency and less than the stop frequency.) Otherwise the middle of the frequency range is used as the center frequency.

Solving for Ports Only

For Driven solution types with ports.

To quickly compute only the 2D excitation field patterns, impedances, and propagation constants at each port:

- Under the **General** tab of the **Solution Setup** dialog box, select **Solve Ports Only**.

  This disables the remaining settings for **Maximum number of passes** and **Convergence per pass**.

HFSS calculates the natural field patterns (or modes) that can exist inside a transmission structure with the same cross-section as the port. These 2D field patterns serve as boundary conditions for the full 3D problem.

Related Topics

Technical Notes: Port Solutions

Setting the Minimum Frequency

For Eigenmode solution types.

For every Eigenmode solution setup, specify the minimum frequency at which to search for eigenmodes. HFSS searches for the user-specified number of modes with a higher resonant frequency than the **Minimum Frequency** value.

- Under the **General** tab of the **Solution Setup** dialog box, type a **Minimum Frequency** in the frequency units.

Warning Because the minimum frequency is used to normalize some matrices, if the frequency is set too low, HFSS tries to solve a nearly-singular matrix, which may erode the accuracy of the calculations. As a general rule, do not enter a frequency less than 0.01 times the suggested, or default, value for **Minimum Frequency**.

Setting the Number of Modes

For Eigenmode solution types.

For every Eigenmode solution setup, specify the number of eigenmode solutions that the solver finds. If you enter **5**, the solver calculates the first 5 eigenmode solutions above the minimum frequency.

The Eigenmode solver can find up to 20 eigenmode solutions.

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Number of Modes**.

Setting Adaptive Analysis Parameters

When you set up an adaptive analysis, define the following parameters under the **General** tab of the **Solution Setup** dialog box:

- Maximum Number of Passes
- Maximum Delta S (for designs with ports) or Use Matrix convergence (here you can set a matrix values for convergence, including maximum delta for Mag S and Phase S).
- Maximum Delta Energy per Pass (for designs with voltage sources, current sources, incident waves, or magnetic bias).
- For Eigenmode solutions, specify Maximum Delta Frequency per pass and, if desired, Converge on Real frequency Only.

Under the **Options** tab of the **Solution Setup** dialog box, you can edit the following settings:

- Lambda Refinement
- Maximum Refinement Per Pass
- Maximum Refinement
- Minimum Converged Passes
- Minimum Number of Converged Passes
- Use Low Order solution Basis

Under the **Advanced** tab of the **Solution Setup**, depending on the solution type, you can edit the following settings.

- Initial Mesh Options for mesh linking
- Output Variable Convergence (output variables must be defined for this to be enabled.)
- Use Absorbing Boundary on Ports
- Waveport Adapt options (Port Field Accuracy and Min/Max Port Triangle settings)

Setting the Maximum Number of Passes

The **Maximum Number of Passes** value is the maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis will continue unless the convergence criteria are reached.

To set the maximum number of passes for an adaptive analysis:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Number of Passes**.

Note The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in HFSS requesting more memory than is available or taking excessive time to compute solutions.

Setting the Maximum Delta S Per Pass

For designs with ports.

The delta S is the magnitude of the change of the S-parameters between two consecutive passes. The value you set for **Maximum Delta S** is a stopping criterion for the adaptive solution. If the magnitude of the change of all S-parameters are less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.

To set the maximum delta S per adaptive pass:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Delta S**.

Delta S data is available only after HFSS completes two iterations of the adaptive analysis process.

Note Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been de-embedded and renormalized.

Related Topics

Viewing the Maximum Magnitude of Delta S Between Passes

Technical Notes: Maximum Delta S

Setting the Maximum Delta Energy Per Pass

For designs with voltage sources, current sources, incident waves or magnetic bias.
Not applicable to designs with ports.

The delta Energy is the difference in the relative energy error from one adaptive solution to the next. The value you set for **Maximum Delta Energy** is a stopping criterion for the adaptive solution. If the delta Energy falls below this value, the adaptive analysis stops. Otherwise, it continues until the convergence criteria are reached.

To set the maximum delta Energy per adaptive pass:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Delta Energy**.

Delta Energy data is available only after HFSS completes two iterations of the adaptive analysis process.

Related Topics

Viewing the Delta Magnitude Energy

Technical Notes: Maximum Delta Energy

Setting the Maximum Delta Frequency Per Pass

For Eigenmode solution types

The delta Frequency is the percentage difference between calculated eigenmode frequencies from one adaptive pass to the next. The value you set for **Maximum Delta Frequency Per Pass** is a stopping criterion for the adaptive solution. If the eigenmode frequencies change by a percentage

amount less than this value from one pass to the next, the adaptive analysis stops. Otherwise, it continues until the maximum number of passes is completed.

To set the **Maximum Delta Frequency Per Pass**:

- Under the **General** tab of the **Solution Setup** dialog, enter a value for **Maximum Delta Frequency Per Pass**.

  Delta Frequency data is available only after HFSS completes two iterations of the adaptive analysis.

Related Topics

Specifying Convergence on Real Frequency Only

Specifying Convergence on Real Frequency Only

For Eigenmode solution types.

Selecting **Converge on Real Frequency Only** causes the percent difference calculation among a set of frequencies to be based only on the real parts of the frequencies; the imaginary parts of the frequencies are ignored.

- Under the **General** tab of the **Solution Setup** dialog box, select **Converge on Real Frequency Only**.

Specifying Output Variable Convergence

You can specify additional convergence criteria through the use of output variables. The **Max Delta Per Pass** defined for output variable convergence represents the difference in values of the output variable between consecutive adaptive passes. If the difference in the value of the output variable between consecutive passes is less than the **Max Delta Per Pass** value this part of the convergence criteria is satisfied.

For driven solutions, if the **Maximum Delta S**, **Maximum Delta E**, or alternate matrix convergence criteria are achieved in addition to any specified output variable convergence criteria, the adaptive analysis stops. Otherwise, the solution continues until the requested number of passes is completed.

For eigenmode solutions, if the **Maximum Delta Frequency Per Pass** criteria is achieved in addition to any specified output variable convergence criteria, the adaptive analysis stops. Otherwise, the solution continues until the requested number of passes is completed.

To set the **Output Variable Convergence** criteria:

1. Ensure that the desired output variable to use for convergence exists. See Specifying Output Variables.
2. Under the **Advanced** tab of the **Solution Setup** dialog box, select the **Also Use Output Variable Convergence** checkbox. If no output variables have been defined in the design then this option is disabled.
3. Select the desired output variable from the drop down list and specify the **Max Delta Per Pass** criteria.

   The **Setup Context** button is enabled if the output variable represents a field quanity that requires a specific geometric context on which to calculate its value. For example, to converge

on a far field quantity such as antenna Gain, you must select a radiation setup and the corresponding theta/phi point at which to calculate the gain value.

a. For output variables that require a geometric context, select the **Setup Context** button.

   This displays the **Output Variable Context** dialog.

   - Geometry Selection - a drop down list of appropriate geometry domains corresponding to the output variable quantity.

   The Evaluation Context fields are enabled as required for the variable. These can be

   - Phi and Theta - values for spherical geometries.
   - IWave Phi and IWave Theta values - intrinsic variables necessary for designs which include incident wave excitations.
   - Linear Distance - a drop down list of linear points for polyline geometries.

b. Select a geometry and associated **Evaluation Context** values.

c. Click **OK** to close the **Setup Context** dialog.

Related Topics

Viewing Convergence Data

Viewing the Output Variable Convergence

Specifying a Source for the Initial Mesh

You may choose to specify a source for the initial mesh from either the current design or another design. The source mesh should represent a geometrically equivalent model. To specify a source for the initial mesh:

1. Under the **Advanced** tab of the **Solution Setup** dialog box, click the checkbox for **Use Current Mesh From**.

   This enables the **Other Design** option and the **Current Design** option, provided another solution setup exists in the current design. Note that the **Lamda refinement** option is deselected under the **Options** tab to avoid over-refinement of the linked mesh

   - If you click **Current Design**, you can select from available solution setups via the drop down menu.
   - If you click **Other Design**, the **Setup Link** button becomes active. Click **Setup Link** to display the **Setup Link** dialog box.

     Under the **General** tab, the **Setup Link** dialog box contains fields for the Project File, Design, and Solution.

     a. To specify a Project File, either click the drop down menu to select from available projects, or click the ellipsis [...] button to open a file browser window.

        When you select a Project File, the Design field and the Solution field are filled in with default values, and the drop down menus contain any available Projects and solutions.

        The **Parameters** tab lets you view any variables contained in the Project you select.

     b. Click the **OK** button to accept the project file for the setup.

2. Continue with other settings or click **OK** to accept the setup and close the **Setup** dialog box.

Related Topics

Clear Linked Data

Clearing Linked Data

If you have previously setup links to a design, the **HFSS>Analysis Setup** menu contains an option to **Clear Linked Data**. This removes the linked data for all links in a design, therefore invalidating the solutions. Clearing linked data for some link type requires HFSS to revert to the initial mesh. Thus in some cases, this command removes the current mesh of the target design.

Setting Lambda Refinement

Lambda refinement is the process of refining the initial mesh based on the material-dependent wavelength. It is recommended and selected by default. If you select the **Use Current Mesh From** option under the **Advanced** tab, the **Do Lamda Refinement** deselected, but the fields remain enabled so that you can select it if desired.

To specify the size of wavelength by which HFSS will refine the mesh:

1. Under the **Options** tab of the **Solution Setup** dialog box, select **Do Lambda Refinement**.

   This enables the **Target** field and the **Use free space lambda** check box.

2. Enter a value for the wavelength in the **Target** field.

   For Driven solutions, the default target is 0.3333, which means that HFSS will refine the mesh until most element lengths are approximately one-third wavelength.

   For eigenmode solutions, the default target is 0.2

3. If you want the initial mesh to be refined based on the wavelength in free space, select **Use free space lambda**. Material-dependent lambda refinement will be deactivated.

Note Changing the Lambda refinement target invalidates any solutions that were performed with the previous lambda refinement.

Setting the Percent Maximum Refinement Per Pass

The value you set for percent **Maximum Refinement Per Pass** determines how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error will be refined. The default value is 20%.

To set the percent refinement per adaptive pass:

- Under the **Options** tab of the **Solution Setup** dialog box, enter a value for percent **Maximum Refinement Per Pass**.

Related Topics

Technical Notes: Percent of Tetrahedra Refined Per Pass

Setting the Maximum Refinement

This specifies the maximum number of tetrahedra that can be added during an adaptive pass. The default is set at 100000.

To set a new value for the Maximum Refinement:

1. Under the **Options** tab of the **Solution Setup** dialog box, click the **Maximum Refinement** checkbox to enable the text field.
2. Enter the number of tetrahedra for **Maximum Refinement**.

   You can also control these values in the docked properties window for the setup. Click the checkbox for **Use Max Refinement**, to apply the value in the Max Refinement text field.

Setting the Minimum Number of Passes

An adaptive analysis will not stop unless the minimum number of passes you specify has been completed, even if convergence criteria have been met.

- Under the **Options** tab of the **Solution Setup** dialog box, enter a value for **Minimum Number of Passes**.

Note For a solve setup with zero passes, no sweeps, and that is not ports only, validation produces a warning message.

Setting the Minimum Number of Converged Passes

An adaptive analysis will not stop unless the minimum number of converged passes you specify has been completed.

- Under the **Options** tab of the **Solution Setup** dialog box, enter a value for **Minimum Converged Passes**.

  The convergence criteria must be met for at least this number of passes before the adaptive analysis will stop.

Setting Matrix Convergence Criteria

For designs with ports.

You can specify different stopping criteria for specific entries in the S-matrix. This is done in the **Matrix Convergence** dialog box. The adaptive analysis will continue until the magnitude and phase of the entries change by an amount less than the specified criteria from one pass to the next, or until the number of requested passes is completed.

To set the matrix convergence:

1. Under the **General** tab of the **Solution Setup** dialog box, select **Use Matrix Convergence**.
2. Click **Set Magnitude and Phase**.

   The **Matrix Convergence** dialog box appears.

3. Select one of the following from the **Entry Selections** pull-down list:

All	Sets all of the matrix entries at once. (The default).
Diagonal/Off-Diagonal	Sets all of the diagonal matrix entries at once, all off-diagonal matrix entries at once, or both diagonal and off diagonal entries at once.
Selected Entries	Sets individual matrix entries that you will select.

   For the selection **All**, enter the convergence criteria for the **Maximum Delta (Mag S)** and the **Maximum Delta (Phase S)** in the fields to the right.

   For the selection **Diagonal**/**Off-Diagonal**, first check **Diagonal Entries**, **Off-Diagonal Entries**, or both, to enable the convergence criteria field or fields. Then enter the convergence criteria for the **Maximum Delta (Mag S)** and the **Maximum Delta (Phase S)** in the fields to the right.

4. If you chose **Selected Entries**, the Matrix Convergence dialog displays some new fields:
   - two lists of ports and associated modes (or terminals).
   - a table showing columns for Matrix Entry 1, Matrix Entry 2, and the Delta Mag and Delta Phase.
   - an **Insert Entries** button with which to move selections from the port list to the table

   To select the desired ports and mode (or terminal) pairs, do the following:

   a. Select the first matrix entry (or entries) from the top list of matrix entries.

   b. Select the second matrix entry (or entries) from the bottom list of matrix entries.

   c. Click **Insert Entries**.

      The entries appear in the table to the right. If you have selected multiple entries, all combinations of matrix entry1 and matrix entry2 populate the table.

   d. In the **Magnitude** box, enter the maximum change in magnitude from pass to pass from the first set of matrix entries to the second set of matrix entries.

   e. In the **Phase** box, enter the maximum change in phase, in degrees, from pass to pass from the first set of matrix entries to the second set of matrix entries.

5. Click **OK** to close apply the values and close the dialog.

Related Topics

Viewing the Magnitude Margin

Viewing the Phase Margin

Viewing Delta (Mag S)

Viewing Delta (Phase S)

Using a Low-Order Solution Basis

To reduce the basis functions HFSS uses from second order to linear:

- Under the **Options** tab of the **Solution Setup** dialog box, select **Use Low-Order Solution Basis**.

This option is useful when a model requires a mesh that produces more than 100,000 tetrahedra, but the model size is small compared to wavelength.

Warning If you select **Use Low-Order Solution Basis**, all tetrahedra in the model must have edge lengths less than $1/20^{th}$ wavelength. Thus, this option is usually selected in combination with a specific lambda refinement setting.

Related Topics

Technical Notes: Basis Functions

Use Absorbing Boundary (ABC) on Ports

If the design includes waveports, the **Use Absorbing Boundary (ABC) on Ports** option is enabled under the **Advanced** options tab of the **Solution Setup** dialog box.

- If you select this setting, edges which are assigned to ABC and touch a port have an absorbing boundary condition applied during the port solution.
- If you do not select the setting, a perfect conducting boundary condition is used during the port calculations.

In most cases this setting has a limited affect on the overall fields or post processed quantities.

Note If you apply this setting to a port on an object that contains anisotropic materials, an error message is generated during the solution.

Waveport Adapt Options

If the design includes waveports, the **Waveport Adapt Options** options appear under the **Advanced** options tab of the **Solution Setup** dialog box. These options include:

- **Port field accuracy** - specified as a target percentage.
- **Set Min/Max Triangles -** unchecked by default.

If you check **Set Min/Max Triangles**, the **Minimum Number of triangles** and **Maximum Number of Triangles** fields are enabled. You can edit the default values of 100 for the minimum and 500 for the maximum.

For designs with lumped ports, this option is not active. Higher numbers of triangles would not benefit a solution setup in this case.

Adding a Frequency Sweep

For Driven solution types.

To generate a solution across a range of frequencies, add a frequency sweep to the solution setup. HFSS performs the sweep after the adaptive solution, if one is defined. If an adaptive solution is not requested, the sweep is the only solution generated. You can also disable a sweep, so that you can run only the adaptive solution (or a ports-only solution) without the sweep, then later reactivate the sweep definition.

To add a frequency sweep:

1. On the **HFSS** menu, point to **Analysis Setup**, and then click **Add Sweep**  .
2. Select the solution setup to which the sweep applies and click **OK**.

   The **Edit Sweep** dialog box appears.
3. Specify the following sweep parameters:
   - Sweep type - Discrete, Fast, or Interpolating.
   - Frequency points to solve.
   - The solved frequencies for which you want to save the fields.
4. If you plan to perform a Full-Wave SPICE analysis, click Time Domain Calculation tool to obtain assistance determining a suitable frequency sweep range for the solutions.
5. Click **OK**.

Note For a solve setup with zero passes, no sweeps, and that is not ports only, validation produces a warning message.

Related Topics

Disabling a Frequency Sweep

Technical Notes: Frequency Sweeps

Selecting the Sweep Type

For Driven solution types.

Specify the type of sweep you want to perform in the **Edit Sweep** dialog box. Choose one of the following sweep types:

Fast	Generates a unique full-field solution for each division within a frequency range. Best for models that will abruptly resonate or change operation in the frequency band. A Fast sweep will obtain an accurate representation of the behavior near the resonance.
Discrete	Generates field solutions at specific frequency points in a frequency range. Best when only a few frequency points are necessary to accurately represent the results in a frequency range.
Interpolating	Estimates a solution for an entire frequency range. Best when the frequency range is wide and the frequency response is smooth, or if the memory requirements of a Fast sweep exceed your resources.

When you select Fast and Interpolating sweeps in the **Edit Sweep** dialog, the **Setup Interpolation Basis** button is activated Selecting this button displays the **Setup Interpolation Basis** dialog, which lets you specify a maximum number of solutions, and other convergence values.

Related Topics

Technical Notes: Frequency Sweeps

Options for Discrete Sweeps

For Discrete sweeps, the **Edit Sweep** dialog options you can set include

- Sweep Name
- Frequency Setup
- Whether to Save Fields (for all Frequencies). By default, all frequencies are saved.

Options for Fast Sweeps

For Fast sweeps, the **Edit Sweep** dialog options you can set include:

- Sweep Name
- Frequency Setup
- Whether to **Save Fields**. By default, all fields are saved.
- DC Extrapolation options
  1. Select **Extrapolate to DC** to enable the DC Extrapolation options.
  2. Enter a value for the **Minimum Solved Frequency**. This value represents the smallest frequency in the sweep for which a full solution is generated.
  3. The **Snap Magnitude to 0 or 1** check box is enabled by default. Enter a value for **Snapping Tolerance**. If the magnitude of a particular S-parameter value at DC falls within this tolerance greater than 0 or less than 1, then the magnitude is snapped to 0 or 1 respectively. Uncheck the selection box to disable this option.

Options for Interpolating Sweeps

For Interpolating sweeps, the **Edit Sweep** dialog options you can set include:

- Sweep Name
- Max Solutions
- Error Tolerance
- Frequency Setup
- Interpolation Convergence

  Click the **Setup Interpolation Convergence** button to open the Setup Interpolation Convergence dialog.
- DC Extrapolation options
  1. Select **Extrapolate to DC** to enable the DC Extrapolation options.
  2. Enter a value for the **Minimum Solved Frequency**. This value represents the smallest frequency in the sweep for which a full solution is generated.
  3. The **Snap Magnitude to 0 or 1** check box is enabled by default. Enter a value for **Snapping Tolerance**. If the magnitude of a particular S-parameter value at DC falls within this tolerance greater than 0 or less than 1, then the magnitude is snapped to 0 or 1 respectively. Uncheck the selection box to disable this option.

Setup Interpolation Convergence

For Interpolated sweeps, the **Setup Interpolation Convergence** dialog lets you specify the following settings for a sweep:

- Whether to use all or selected entries in the matrix of data types for the convergence. To choose, click the **Select Entries** button to display the Interpolation Basis Convergence dialog.
- The data types for convergence. You can select **Use All Entries** (the default) or to **Use Selected Entries**. If you select **Use Selected**, only the check box for S-Matrix is enabled. If you select **Use All Entries** for the convergence, as many of the data types as are available for the kind of solution under consideration have check boxes enabled.

  For Driven Model, 3D Solution Interpolating sweeps:
  - S-Matrix - checked and disabled.
  - T-Matrix - disabled and unchecked.
  - Port Impedance - enabled and unchecked.
  - Propagation constants - enabled and unchecked.

  For Driven Terminal, 3D Solution Interpolating sweeps:
  - S-Matrix - checked and disabled.
  - T-Matrix - enabled and unchecked.
  - Port Impedance - enabled and unchecked.
  - Propagation constants - enabled and unchecked.

  For Driven modal, ports-only, interpolating

- S-Matrix - unchecked and disabled
- T-Matrix - disabled and unchecked
- Port impedance - enabled and unchecked
- Propagation constants - enabled and checked

For Driven terminal, ports-only, interpolating

- S-Matrix - unchecked and disabled
- T-Matrix - enabled and unchecked
- Port impedance - enabled and unchecked
- Propagation constants - enabled and checked

Note If a driven setup's ports-only setup changes and then the problem type switches between driven modal and driven terminal, HFSS resets the interpolation basis data types for the interpolating sweep.

Setting the Error Tolerance

For Fast and Interpolating sweeps.

The **Error Tolerance** value is the maximum relative difference allowed between two successive interpolation solutions. The default 0.5 percent for interpolating sweeps and 0.2 percent for fast sweeps is usually satisfactory.

To set the error tolerance for an Interpolating sweep:

- In the **Edit Sweep** dialog box, click the **Setup Interpolation Basis** button.
- This displays the **Data Type for Convergence** dialog.
- Type a value in the **Error Tolerance** box.

Setting the Maximum Number of Solutions

For Fast and Interpolating sweeps.

The **Max Solutions** value is the maximum number of solutions that will be solved for the frequency range. For fast sweeps and for interpolating sweeps the default is 50. To change the value:

- In the **Edit Sweep** dialog box, click the **Setup Interpolation Basis** button.

  This displays the **Data Type for Convergence** dialog.
- Type a value in the **Max Solutions** box and click **OK**.

Note HFSS automatically subdivides the interpolating sweep range so that no single subrange gets too many basis elements. The effect is that you can now (if appropriate) request hundreds of basis elements in the Max Solutions box for interpolating sweep setup, without incurring any basis seeding performance penalty.

Interpolation Basis Convergence

From the **Setup Interpolations Basis** dialog, select the U**se Selected Entries** radio button to enable the **Select Entries** button. Select this to display the **Interpolation Basis Convergence** dialog. This dialog permits you to specify the convergence basis

1. Select one of the following from the **Entry Selections** pull-down list:

All	Sets all of the matrix entries at once. (The default).
Diagonal	Sets all of the diagonal matrix entries at once.
Off-Diagonal	Sets all of the off-diagonal matrix entries at once.
Selected Entries	Sets individual matrix entries that you will select.

2. If you chose **All**, **Diagonal**, or **Off-Diagonal**, you may fine-tune the matrix entry selection process by selecting one of the following options from the **Mode Selection** pull-down list:

All	Sets all of the mode matrix entries. Select in conjunction with **All**, **Diagonal**, or **Off-Diagonal** entry selections.
Dominant Only	Sets only the dominant mode matrix entries. Select in conjunction with **All**, **Diagonal**, or **Off-Diagonal** entry selections.
Higher Order Only	Sets only the higher-order mode matrix entries. Select in conjunction with **All**, **Diagonal**, or **Off-Diagonal** entry selections.

3. If you chose **Selected Entries**, the dialog displays some new fields:
   - two lists of waveports in terms of row and column
   - an empty table showing rows and columns.
   - an **Add Selection** button and a **Remove Selection** button with which to move selections from the wave port list to and from the table

   To select the waveports for convergence, do the following:

   a. Select the first matrix entry or entries from the top list of matrix entries.

   b. Select the second matrix entry or entries from the bottom list of matrix entries.

   c. Click **Insert Entries**.

      The table location corresponding to the selection, the dash in the display is replaced by ON. For example, selecting the first element in the row list and the fourth element in the column list, and then **Add Selection** places an ON in the first row, fourth column. You can **Remove Selection** one entry at a time, or clear the entire table with the **Clear** button.

4. Click **OK** to close apply the selections and close the dialog.

Specifying the Frequency Points to Solve

You can specify the following types of frequency points to solve within a frequency sweep:

Linear Step	A linear range of frequency points in which you specify a constant step size.
Linear Count	A linear range of frequency points in which you specify the number, or count, of points within the frequency range.
Single Points	Individual frequency points. *For Discrete sweeps*.

Select the type of frequency point entry from the **Type** pull-down list **The Edit Sweep** dialog contains a Time Domain Calculation tool that you can use to help calculate frequency step sizes and maximum frequencies, particularly if you intend to perform Full-Wave Spice analysis.

Specifying Frequency Points with a Linear Step Size

1. In the **Edit Sweep** dialog box, click **Linear Step** in the **Type** pull-down list.
2. In the **Start** text box, type the starting frequency of the frequency sweep.

   HFSS solves the solution beginning with the frequency entered in the **Start** box and ending with the frequency entered in the **Stop** box.
3. In the **Stop** text box, type the ending frequency of the frequency sweep.
4. In the **Step Size** box, type the difference between frequency points.

   HFSS will solve the frequency point at each step in the specified frequency range, including the start and stop frequencies.

   For example, specifying **10** for the start frequency, **20** for the stop frequency, and **2.5** for the step size for a Discrete sweep instructs HFSS to compute a solution for frequencies of 10, 12.5, 15, 17.5, and 20. The step size specified for an Interpolating sweep dictates the amount of information that will be viewed on a post-processing plot.
5. For Fast sweeps, select **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

   For Discrete sweeps, select **Save Fields (All Frequencies)** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

   If want to save the fields for just one or a few Discrete sweep frequencies, select **Single Points** from the **Type** pull-down list, and then select the **Save Fields** check box for the desired frequency.

Specifying a Linear Count of Frequency Points

1. In the **Edit Sweep** dialog box, click **Linear Count** in the **Type** pull-down list.
2. In the **Start** text box, type the starting frequency of the frequency sweep.

   HFSS solves the solution beginning with the frequency entered in the **Start** box and ending with the frequency entered in the **Stop** box.
3. In the **Stop** text box, type the ending frequency of the frequency sweep.
4. In the **Count** text box, type the number of points in the sweep. The count value includes the start and stop values.

HFSS will divide the frequency range into the count you specify and solve each frequency point in the count.

5. For Discrete and Fast sweeps, select **Save Fields (All Frequencies)** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

   If you want to save the fields for just one or a few Discrete sweep frequencies, select **Single Points** from the **Type** pull-down list, and then select the **Save Fields** check box for the desired frequency.

6. For Fast sweeps, select **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

   For Discrete sweeps, select **Save Fields (All Frequencies)** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

   If want to save the fields for just one or a few Discrete sweep frequencies, select **Single Points** from the **Type** pull-down list, and then select the **Save Fields** check box for the desired frequency.

Specifying Single Frequency Points

For Discrete sweeps.

1. In the **Edit Sweep** dialog box, click **Single Points** in the **Type** pull-down list.
2. In the **Single** text box, type a desired frequency point, and then select the frequency units.
3. Select **Save Fields** if you want to save the calculated 3D field solution associated with all port modes at that frequency.
4. Click **Insert**.

   The point is added to the **Frequency** column to the right. A check mark in the **Save Fields** column indicates that the fields for the point will be saved. Optionally, click the check box in the **Save Fields** column.

5. Repeat steps 2 - 4 for each frequency point you want to solve.

Deleting Frequency Points

1. Select **Single Points** from the **Type** pull-down list.
2. Select the row containing the frequency you do not want to solve.
3. Click **Delete**.

Inserting Frequency Points

For Discrete sweeps, you can insert specific frequency points that you want to solve in the frequency range. They can be inserted after you have added uniform frequency points to solve.

1. Select **Single Points** from the **Type** pull-down list.
2. Select a row before which you want to add a frequency point.
3. In the **Single** text box, type a desired frequency point in the frequency units.
4. Select **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at that frequency.

5. Click **Insert**.

Choosing Frequencies for Full-Wave SPICE

If you plan to perform a full-wave SPICE analysis, use the **Time Domain Calculation** dialog box to help determine a suitable frequency sweep range for the solution.

To perform the calculation of suitable frequencies to solve:

1. In the **Edit Sweep** dialog box, click **Time Domain Calculation**.

   The **Time Domain Calculation** dialog box appears.

2. Type a minimum rise time value in the **Signal Rise Time** box.

   This value represents the time scale that will characterize the rate of change of the input time signal, which will be applied in the circuit simulator.

3. Type a value in the **Time Steps Per Rise Time** box.

   The time sampling increment for the entire signal is calculated using

   $$\Delta t = \frac{\tau}{N_\tau}$$

   where

   - Δt is the time sampling increment.
   - τ is the signal rise time.
   - N_τ is the number of time steps per signal rise time.

4. Type a value in the **Number of Time Points** box.

   Note that the input time signal duration is determined using $N \times \Delta t$, where N is the number of time points.

5. Click **Calculate**.

   - HFSS now determines the **Maximum Frequency** using

     $$F_{max} = \frac{0.5}{\Delta t}$$

     where F_{max} is the maximum frequency.

     HFSS determines the **Frequency Step Size** using $\frac{F_{max}}{N}$.

6. Click **OK** to transfer the data to the frequency sweep fields in the **Edit Sweep** dialog box.

Related Topics

Guidelines for Calculating Frequencies for Full-Wave SPICE

Guidelines for Calculating Frequencies for Full-Wave SPICE

Keep the following guidelines in mind when you set up the calculation for the suggested frequency step size and maximum frequency:

- The maximum frequency should be at least five times the inverse of the rise and fall times. If the specified frequency band is too wide, an HFSS frequency sweep may have conver-

gence problems. If this happens, try to decrease the maximum frequency until the solution converges.

- It is recommended, though not required, that the minimum frequency be less than the maximum frequency divided by the number of frequency steps. It is usually recommended to have at least 500 frequency steps. A higher number will slightly improve the full-wave SPICE solution accuracy, but will also increase CPU and memory requirements to solve the problem. For most cases, using 1000 frequency steps provides a good trade-off between the accuracy and computational requirements.

Warning Occasionally, HFSS can fail to solve for the minimum frequency during a Discrete or Interpolating frequency sweep due to a failure of the port solver to converge. If this happens, try to increase the minimum frequency until the solution process completes successfully. However, the minimum frequency should be as low as possible because the low-frequency response determines the steady-state time response.

- The suggested frequency sweep ranges are estimates. You may have a pulse with a wider frequency content and HFSS's recommended frequency sweep range may miss some of the high frequencies.

Disabling a Frequency Sweep

To disable a sweep definition without deleting it:

1. Expand the tree hierarchy under the **Analysis** icon in the project tree.
2. Expand the tree hierarchy under the icon for the analysis setup that includes the sweep.
3. Right-click on the icon for the sweep definition. In the shortcut menu that appears, toggle the **Enabled** selector to the OFF setting (check mark not displayed). .

To reactivate the sweep, open theshortcut menu again and toggle the **Enabled** selector to the ON setting (check mark displayed).

Disabling an Analysis Setup

To disable an Analysis setup definition without deleting it:

1. Expand the tree hierarchy under the **Analysis** icon in the project tree.
2. Right-click on the icon for the setup definition. In the shortcut menu that appears, toggle the **Enabled** selector to the OFF setting (check mark not displayed).

To reactivate the analysis, open the shortcut menu again and toggle the **Enabled** selector to the ON setting (check mark displayed).

Specifying the Number of Processors

If you want to use more than one processor, you can specify the number of processors in the **HFSS Options** dialog box. By default, this value is set to 1.

To modify the number of processors used when solving:

1. Under the **Tools** menu, point to **Options**, and then click **HFSS Options**.
2. Click the **Solver** tab.
3. Type a value for **Number of Processors**.

   This setting affects all HFSS projects for the current user and machine. However the solver considers the number of processors to be optional and reverts to a single processor solution if a multiple processor licence is not available.

Specifying the Desired RAM Limit

The **Desired RAM Limit** setting requests a restriction on the amount of physical random access memory (RAM) the MPS solver used in HFSS may use before it must stop solving *on-core*—solving processes entirely in RAM—and start solving *off-core*. In off-core mode, HFSS creates temporary solution files to which it spills, or shifts, data from RAM, instead of forcing the operating system to start disk swapping. Unlike off-core mode, disk swapping halts some processes and temporarily writes them to disk files to free up physical memory to run other processes. The MPS solver is much more efficient at handling its own memory, although if the process grows large enough, the operating system may be forced into disk swapping.

The MPS solver used in HFSS automatically detects the amount of installed physical RAM. If the **Desired RAM Limit** setting is active, and its value is less than the amount of installed physical RAM, the solver will automatically switch to off-core mode once the **Desired RAM Limit** value is reached. The solver ignores the setting if the amount of physical RAM specified is greater than the amount of installed physical RAM, or if the amount of physical RAM required by the solver is greater than the amount specified by the **Desired RAM Limit** setting.

Note Regardless of the **Desired RAM Limit** setting, if no more physical RAM is free, the HFSS solver will automatically switch to off-core mode.

To specify the **Desired RAM Limit** of the machine on which HFSS is installed:

1. Under the **Tools** menu, point to **Options**, and then click **HFSS Options**.
2. Click the **Solver** tab.
3. Select **Desired RAM Limit (MB)**.
4. Type a value for the amount of available memory, in megabytes, in the text box. This setting affects all HFSS projects for the current user and machine.

Note Allocation of greater than 2 GB of RAM on 32-bit hardware platforms is only possible with the appropriate operating system and boot settings, even if more than 2 GB are physically installed. See Increasing PC RAM.

Specifying the Hard Memory Limit

The **Maximum RAM Limit** setting specifies the absolute limit on the amount of physical memory that the solver can use. If the solver attempts to allocate more memory than this setting, the solution process will terminate abnormally, and incorrect error messages may appear.

To specify the **Maximum RAM Limit** for the current user and machine:

1. Under the **Tools** menu, point to **Options**, and then click **HFSS Options**.
2. Click the **Solver** tab.
3. Select **Maximum RAM Limit (MB)**.
4. Type a value for the maximum amount of memory, in megabytes, in the text box.

   The solver will only use the amount of memory specified and no more. This setting affects all HFSS projects for the current user and machine.

Note Ansoft recommends that you use the **Desired RAM Limit** setting if you wish to limit the RAM that the solver may allocate.

Note Allocation of greater than 2 GB of RAM on 32-bit hardware platforms is only possible with the appropriate operating system and boot settings, even if more than 2 GB are physically installed. See Increasing PC RAM.

13

Running Simulations

After you specify how HFSS will compute the solution, begin the solution process.

1. Select a single solution setup in the project tree.

2. On the **HFSS** menu, click **Analyze All**  or right click on the setup to use and select **Analyze** from the short-cut menu.

   HFSS computes the 3D field solution inside the structure.

To run more than one analysis at a time, follow the same procedure while a simulation is running. The next solution setup will be solved when the previous solution is complete.

Note If a linked dependency in the setup is already simulating (for example, due to setup links to the same external source for a near or far field wave, or a magnetic bias), HFSS won't allow another dependent simulation to start until the first use of the source has completed.

To solve *every enabled solution setup in a design*:

1. In the project tree, under the design you want to solve, select **Analysis**.

2. On the **HFSS** menu, click **Analyze All**  or right click on **Analysis** in the project tree and from the short-cut menu select **Analyze All**.

   Each enabled solution setup will be solved in the order it appears in the project tree. The docked properties window for each setup and sweep contains an **Enabled** checkbox. To disable a setup or a sweep, uncheck the **Enabled** checkbox.

3. To view the solution queue, click **Tools>Show Queued Simulations**.

   This displays a dialog that displays each simulation and its current status. You select and

remove any simulation from the queue.

To solve *a single sweep under a setup*:

1. Click on a sweep to run under the setup of interest in the project tree.
2. Right-click on the sweep, and select **Analyze** from the short-cut menu.

   The selected sweep runs.

To solve *two or more sweeps or two or more parametric analyses under a setup*:

1. Configure two or more machines for a parallel solve. For details, see Solving Remotely for configuration issues and Distributed Solve for license issues.
2. Right-click on the setup of interest in the project tree.

   This displays the short-cut menu. If there are two or more sweeps and two or more machines are configured for parallel solving, the **Distribute Analysis** command is enabled on the menu.
3. Select **Distribute Analysis** from the short-cut menu.

   Each solution setup will be solved in the order it appears in the project tree, using the available machines.

Related Topics

Technical Notes: *The Solution Process*

Solving Remotely (Windows only)

It is possible to solve a project on a different machine from the one on which you set up the problem. This is particularly useful when you want to take advantage of a more powerful machine, but it is not convenient to access that machine. This process involves configuring the machine that will perform the solving (the **remote machine**) as well as the machine on which the design was set up (the **local machine**).

Setup for remote machine

This section describes how to setup the remote machine.

- Preconfiguring the Machine
- Configure Distributed COM
- Set up Security Permissions on the Remote Machine
- Set up the local machine
- Distributed solve with licensing

Preconfiguring the Remote Machine

1. Log in as an administrator to the machine on which you want to solve designs – the remote machine.
2. Install HFSS.
3. Determine an account that will act as the remote user - the user who will solve designs. This user must be an administrator on any remote machines.

Note The permissions for the remote user should be the same or fewer than any local user.

Configure Distributed COM for Remote Machine

1. Click **Start>Run**, and type **dcomcnfg**.
2. Click **OK** to start the DCOM Configuration window.

   The DCOM Configuration window looks different depending on which operating system you are using - Windows 2000 or Windows XP, or Windows Server 2003.

Windows 2000 for Remote Machine

You will need to verify that the following steps have been followed:

a. Under the **Default Properties** tab, make sure **Enable Distributed COM on this computer** is checked.

b. Under the **Applications** tab, verify that the **HFSSEngine** class corresponding to the version of HFSS you are configuring is listed. (For instance, if you are configuring HFSS 10, look for the **HFSSEngineV10** class.) If the desired HFSS Engine class is not listed, you will need to reregister the corresponding HFSSComEngine. Once it is listed, click on the **Properties** button.

c. Under the **General** tab, set the Authentication Level to either **Connect** or **None**. **Connect**

is more secure.

d. Under the **General** tab, view the path listed in **Local Path**. This is the path to the COM engine you are configuring for remote analysis. If the installation path includes a space, Windows 2000 does not show the full path. In this case, you will need to determine the path manually; the COM engine should be located in the version-specific product subdirectory of the installation directory.

e. Under the **Location** tab, verify that the **Run application on this computer** checkbox is checked.

f. Under the **Identity** tab, select **This User** radio button to permit a specific user to analyze models on the remote machine (known as the "remote user"). Enter the user name and password for the remote user, and click **Apply**.

g. Under the **Security** tab, verify the following:

   1. For **Access Permissions**, verify **Allow Access** permissions for the network and remote users.
   2. For **Launch Permissions**, verify **Allow Launch** permissions for the network and remote users.
   3. For **Configuration Permissions**, verify access for any users who should be able to configure settings for the COM engine.

   If a given user/group is not listed:

   - Choose **Add** to add a new user/group.
   - Select the domain in which the user/group you want to add is located, from the **List Names From** pull-down list.
   - If you want to add an individual user, click the **Show Users** button.
   - Select the user or group you want to add, and verify that **Type of Access** lists full access for the user.
   - Click **OK**.
   - The user/group appears in the **Registry Value Permissions** dialog box.

   4. Click **OK** to apply the changes.

h. Click **OK** to apply all the changes to the product and version-specific DCOM configuration.

i. Click **OK** to close the DCOM configuration window.

Windows XP/Server 2003 for Remote Machine

You will need to verify that the following steps have been followed:

a. Under **Console Root>Component Services>Computers**, right-click **My Computer** and

then click **Properties**.

Note If you receive a message about unblocking this application from the Windows Firewall, click **OK** to unblock the Microsoft Management Console and proceed with the configuration.

b. Under **Default Properties**, verify that **Enable Distributed COM on this computer** is selected. Click **OK**.

c. Under **Console Root>Component Services>Computers>My Computer>DCOM Config**, verify that the **HFSSEngine** class corresponding to the version of HFSS you are configuring is listed. (For instance, if you are configuring HFSS 10, look for the **HFSSEngineV10** class.) If the desired HFSSEngine version is not listed, you will need to reregister the corresponding HFSSComEngine. Once it is listed, right-click on the class and choose **Properties.**

Note If the version-specific **HFSSEngine** class is not registered:

- Click on Start/Run, and type “command” to open a command prompt.
- Change the directory to the **<installation directory>\hfss<version number>** directory.
- Type **hfsscomengine –RegServer** to register the HFSS COM Engine.

Note If a warning appears concerning the CLSID of the desired version-specific **HFSSEngine** class, select OK to register the CLSID key.

d. Under the **General** tab, set the Authentication Level to either **Connect** or **None**. **Connect** is more secure.

e. Under the **General** tab, view the path listed in **Local Path**. This is the path to the COM engine you are configuring for remote analysis.

f. Under the **Location** tab, verify that the **Run application on this computer** checkbox is checked.

g. Under the **Identity** tab, choose **This User** to permit a specific user to analyze models on the remote machine (known as the “remote user”). Enter the user name and password for the remote user, and click **Apply** to validate the entries.

h. Under the **Security** tab, verify the following:

1. Click the **Customize** radio button, and then click **Edit**.
2. For **Launch and Activation Permissions**, verify that **Remote Launch** and **Remote Activation** for the remote user are selected.
3. Under **Access Permissions**, verify that **Local Access** and **Remote Access** for the network and remote users are selected.

4. For **Configuration Permissions**, verify access for any users/groups who should be able to configure settings for the COM engine.

   If a given user/group is not listed:

   - Click **Add** to add a new user/group.
   - Click **Locations** and select the domain/workgroup in which the user you want to add is located.
   - Enter the name of the user/group.
   - Click **Check Names** to verify that the name is correct.
   - Click **OK** to close the **Add Users** window.
   - The user/group appears in the corresponding permissions dialog box.

5. Click **OK** to apply the changes.

i. Click **OK** to apply all the changes to the product and version-specific DCOM configuration.

Enable Firewall Access for Remote Machine

If you have a firewall installed on the remote machine, the COM Engine configured above needs access through it. You also need access to TCP port 135, which is used for Remote Procedure Call (RPC) End Point Mapping.

If the remote machine is using Windows XP Service Pack 2, Windows Firewall was automatically installed and enabled during the service pack installation. You will need to add the COM engine and TCP port 135 to the list of exceptions:

1. Select **Start > Settings > Control Panel** (or **Start > Control Panel**, depending on your Windows display settings).
2. In the Control Panel window, double-click on **Security Center**. The **Security Center** window appears.
3. Click the **Windows Firewall** option.
4. Click the **Exceptions** tab.
5. Add the COM engine:
   a. Click the **Add Program** button.
   b. Click the **Browse** button. Browse to or type the path to the COM engine.
   c. Click **OK** to confirm the program as an exception.
6. Add TCP port 135:
   a. Click the **Add Port** button.
   b. For the **Name**, enter descriptive text to identify this exception, (e.g. RPC End Point Mapping).
   c. For the **Port number**, enter 135 for the number, and choose **TCP**.
   d. Click **OK** to confirm the port as an exception.
   e. Click **OK** to close the Windows Firewall window.

f. Exit the Security Center and the Control Panel.

Configure the Temporary File Directory for Remote Machine

After the DCOM configuration is complete, you need to configure the temporary file directory that will be used by the remote machine while solving.

1. Start the product you plan to use to solve a remote analysis.
2. After the software has started, click **Tools > Options > General Options**.
3. Under **Temp Directory Settings**, in the **Temp Directory** field, enter the name of a temporary directory that all users can see (e.g. C:\Temp). If the directory does not exist, the software creates it automatically.
4. Select the **Select as default temp directory for remote analysis launched as 'This User'** check box.

Note If this check box is disabled, you are not logged in as a user with administrative privileges. You must be logged in as a user with administrative privileges to proceed.

5. Click **OK** to apply the changes. If the **Temp Directory** entered above does not exist, the software prompts you to confirm creation of the directory.

Set up Security Permissions on Remote Machine

Once you have installed HFSS on the remote machine and configured DCOM, you need to configure the security permissions for the temporary file directory and all of the program files (by default, file permissions are inherited)

To configure security permissions on a remote machine:

1. Log into the machine as administrator.
2. Edit the permissions for both the temporary file directory (configured above) as well as the directory in which the COM engine is located. The COM engine path was determined during the DCOM configuration listed above. To edit the permissions of a particular directory:
   a. In Windows Explorer, right-click the directory you are configuring, and choose **Properties**.
   b. Click the **Security** tab. If it is not shown, then you may not be using the NTFS file system. You may be able to skip the following steps for this directory.
   c. If the remote user is not listed, click **Add**, and enter the name of the user.
   d. Choose **Full Control** to give the remote user all access rights for this directory.

Solve a Test Design as a Remote User on a Remote Machine

Finally, you should try to solve a test design as the remote user on the remote machine. If you have any problems solving the test design, resolve these issues before configuring the local machine.

Setup for local machine

Pre-configuring the Local Machine

1. Log in as an administrator on the machine on which you want to solve designs – the local machine.
2. Install HFSS.

Configuring Distributed COM for callback

1. Click **Start>Run**, and type **dcomcnfg**.
2. Click **OK** to start the DCOM Configuration window.

   The DCOM Configuration window looks different depending on which operating system you are using - Windows 2000, Windows XP, or Windows Server 2003.

Windows 2000 for Local Machine

Verify that the following steps have been followed:

a. Under the **Default Properties** tab, make sure **Enable Distributed COM on this computer** is checked.

b. Under the **Default Security** tab, make sure the following permissions are set:

   1. Under **Default Access Permissions**, verify **Allow Access** permissions for the **Network** user.
   2. Under **Default Launch Permissions**, verify **Allow Launch** permissions for the **System** user.
   3. Click **OK** to apply the changes.

c. Click **OK** to close the DCOM configuration window.

d. Reboot the machine if any changes were made during this process.

Windows XP/Server 2003 for Local Machine

a. Under **Console Root>Component Services>Computers**, right-click **My Computer**, and click **Properties**.

Note If you receive a message about unblocking this application from the Windows Firewall, click **OK** to unblock the Microsoft Management Console, and proceed with the configuration.

b. Under **Default Properties**, verify that **Enable Distributed COM on this computer** is selected. Click **OK**.

c. Under the **COM Security** tab, verify that the following permissions are set:

   1. Under **Access Permissions**:
      - Choose **Edit Default**, and verify **Remote Access** permissions are enabled for the **Network** user and **Local Access** permissions are enabled for the **Self** user.
      - Choose **Edit Limits**, and verify all permissions are enabled for **Everyone**.

2. Under **Launch and Activation permissions**, click **Edit Default**, and verify **Local Launch** and **Local Activation** permissions are enabled for the **System** user.

If a given user/group is not listed:

- Click **Add** to add a new user.
- Click **Locations**, and select the domain/workgroup in which the user you want to add is located.
- Type the name of the user.
- Click **Check Names** to verify that the name is correct.
- Click **OK**.

The user/group appears in the corresponding permissions dialog.

d. Click **OK** to apply all the changes, and reboot the machine if any changes were made during this process. Otherwise, exit the Component Services menu. Click **OK** to apply the changes to the product and version-specific DCOM configuration.

Enable Firewall Access for Local Machine

If you have a firewall installed on the remote machine, the runtime application needs access through it. You will also need access to TCP port 135, which is used for Remote Procedure Call (RPC) End Point Mapping.

If the remote machine is using Windows XP Service Pack 2, Windows Firewall was automatically installed and enabled during the service pack installation. You will need to add the COM engine and TCP port 135 to the list of exceptions:

1. Select **Start > Settings > Control Panel** (or **Start > Control Panel**, depending on your Windows display settings).
2. In the Control Panel window, select **Security Center**.

   The **Security Center** window appears.
3. Select the **Windows Firewall** option.
4. Select the **Exceptions** tab.
5. Add the installed program:
   a. Click the **Add Program** button.
   b. If the installed application is listed (e.g. HFSS 10), select the program from the list. Otherwise, choose the **Browse** button, and browse to or type the path to the COM engine.
   c. Click **OK** to confirm the program as an exception.
6. Add TCP port 135:
   a. Click the **Add Port** button.
   b. In the **Name** field, enter descriptive text to identify this exception, e.g. RPC End Point Mapping.
   c. For the **Port number**, enter 135 for the number, and click **TCP**.
   d. Click **OK** to confirm the port as an exception.

e. Click **OK** to close the Windows Firewall window.

f. Exit the Security Center and the Control Panel.

Configure Remote Analysis Dialog for Local Machine

1. Start HFSS on the local machine as one of the users that will be using the remote solving capability.
2. Once the software has started, choose **Tools**/**Options**/**General Options**.
3. Select the **Analysis Options** tab.
4. Under **General Analysis Options**, click the **Show the Remote Analysis Dialog** check box. If you plan to use a specific machine, you may change the Default Analysis Location to **Remote Machine** and provide details on the remote machine under **Default Remote Machine Details**.
5. Under **Launch as User Options**, check the **Launch simulations as 'this user'** option, and enter the user name, password, and domain/workgroup of the remote user.
6. Click **OK**.
7. Set up a project to test the remote solution capability.
8. Once the project setup is complete, click **HFSS>Analyze**.

   The **Remote Analysis** dialog box appears.
9. Select the remote analysis option you want to use, and click **OK**.

The problem begins to solve on the remote machine.

Troubleshooting

Issue	Solution
The following error appears when I try to analyze my design after a long hang: **Unable to locate or start COM engine on machine_name** **Error: The RPC server is unavailable.**	This indicates that the local machine was unable to launch a process on the remote machine. Check the following, in order: • Verify that the machine name is correct. • Verify that the machine is turned on. • For **Interactive user mode**, verify that someone is logged onto the machine. • Verify the installation on the remote machine. • Verify that the COM engine is registered on the remote machine. To do this, follow the steps listed in the **Configuring Distributed COM** under the Remote Machine setup. • Verify launch permissions on the remote machine.
The following error appears immediately when I try to analyze my design: **Unable to locate or start COM engine on machine_name** **Error: The RPC server is unavailable.**	This may be a sign of security issues on the remote machine. Check the following, in order: • Verify that proper security permissions are enabled on the remote machine. • Check if there is a firewall installed on the remote machine.
The following message appears immediately when I try to analyze my design: **Simulation completed with execution error on machine_name**	This indicates that the callback does not have proper permissions. Check the **Configure Distributed COM for callback** section under the local machine setup to correct the problem.

The following error appears immediately when I try to analyze my design: **Unable to locate or start COM engine on machine_name.** **Error: The server process could not be started because the configured identity is incorrect. Check the user name and password.**	The user listed as the “this user” on the remote machine’s DCOM configuration has the wrong password entered. To correct the problem, enter the correct password for the user listed as “this user”.
The following error appears immediately when I try to analyze my design: **Unable to locate or start COM engine on \\HOUSE Error: Access is denied.**	This indicates that the remote machine COM engine has an incorrect password or permissions. To correct the problem: • Verify that the password entered for the remote user on the local machine is correct. • Verify that the COM engine and the directory in which the COM engine resides both include permissions for the remote user.
The following errors appear immediately when I try to analyze my design: **Unable to create simulation working directory within temp dir: temporary_directory** **Error decoding model data or writing it to disk.**	The temporary file directory has insufficient permissions for the remote user to write temporary files. Check the permissions by **Setting up Security Permissions on Remote Machine**.

Distributed Solve with Licensing

If you have purchased the appropriate license, HFSS supports two forms of distributed solve:

- Distributing rows of a parametric table during Optimetrics solve.
- Distributing sub-sweeps within a solve setup.

In these scenarios, if you do a distributed solve, HFSS launches solver engines on multiple machines, assuming that you have configured your machines appropriately. See Solving Remotely for configuration issues

The number of hfss_solve licenses controls how many solves (regular or distributed) can be done at a time. The number of hfss_distrib license determines how many distributed solves can be done at a time. Each hfss_distrib license supports 10 distributed engines. If an installation has more than one distributed solve license, the owners of those licenses can each access up to the total number of available licensed engines. For example, with two distributed solve licenses, all 20 licensed engines are available to divide between those two license owners. HFSS permits you to split the engines 10:10 or 5:15 or 2:18, or even if one person uses all 20 engines, leaving the second hfss_distrib

license unused. However, they cannot divide the 20 machines among more than two distributed solves, since they only have two hfss_distrib licenses available.

Monitoring the Solution Process

While a simulation is running, you can monitor the solution's progress in the **Progress** window. Above the red progress bar, messages describe the setup and step. The progress bar shows the relative progress of each step. Under the bar, messages note the part of the design being solved, and give memory estimates during the factoring process.

You can also view the following solution data at any time during or after the solution:

- The convergence data:
- The matrices computed for the S-parameters, impedances, and propagation constants.
- A profile of status of the adaptive analysis, including the number of valid passes completed.

To view the **Solutions** window:

1. Select the solution **Setup** in the **Project** tree.
2. Right-click to display the shortcut menu.
3. Select one of **Convergence**, **Matrix Data**, or **Profile** from the shortcut menu.

   This displays the **Solutions** window with the corresponding tab selected and the current data displayed.

For out of core problems, quite different amounts of memory may be used for factorization and for solution. So if the amount for factorization is displayed under the progress bar and the amount used is calculated for the profile at the end of the solution, they may be quite different numbers.

Note If HFSS looses its license, it waits for the license to be regained, checking every 2 minutes or until you abort.

Monitoring Queued Simulations

If you have multiple setups for a design, and have selected **Analyze All**, the simulations are queued until there is a machine available. Setups are solved in the order that they appear in the project tree.

1. To view the solution queue, click **Tools>Show Queued Simulations** or click the Show Queue icon on the toolbar  .

   This displays a dialog that displays each simulation and its current status. You select and remove any simulation from the queue.
2. To remove a simulation from the queue, select the simulation, and click **Remove from Queue**.

   This removes the selected simulation from the queue.

Changing the Solution's Priority

You can reduce the priority of HFSS simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of HFSS simulations, your other software tools will respond as they normally would, but HFSS simulations may take longer.

Note The Windows Task Manager will not indicate a reduced priority for the HFSS solver. It only lists the priority of the engine manager, which will appear normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations:

1. While a solution is running, right-click on the **Progress** window, and click **Change Priority** on the shortcut menu.

   Alternatively, use the **Tools>Options>HFSS Options** to display the **HFSS Options** dialog and select the **Solver** tab.

2. From the **Change Priority** menu (or the Default Process Priority dropdown menu) select one of the following priorities:

   Lowest Priority

   Below Normal

   Normal The default.

   Above Normal

   Highest

Aborting Analyses

To end the solution process before it is complete:

- In the **Progress** window, click **Abort**.

  HFSS will end the analysis immediately.

If you aborted the solution in the middle of an adaptive pass or a frequency sweep, the data for that pass or current frequency point is deleted. Any solutions that were completed prior to the one that was aborted are still available.

The solutions that are available are dependent upon when you aborted. For example, if you stopped the solution while a post-processing macro was executing, the field solution computed for that setup is still available.

To abort the solution process after the current adaptive pass or solved frequency point is complete:

- Right-click on the **Progress** window, and click **Clean Stop** on the shortcut menu.

  HFSS will end the analysis after the next solved pass or frequency point.

If you request a clean stop between the third and fourth adaptive pass, the solutions for the third and fourth pass will be available. Note that if you are performing an Interpolating sweep, all of the information from the analysis will be deleted.

Re-solving a Problem

If you modify a design after generating a solution, by adding another port for example, the solution in memory will no longer match the design. The solution setup with the invalid solution is marked with an *X* in the project tree and in the **Results** window.

To generate a new solution after modifying a design, follow the procedure for running a simulation:

1. Select a solution setup in the project tree.
2. On the **HFSS** menu, click **Analyze**.

14

Using Optimetrics for Design Analysis

Optimetrics is a set of analysis tools that enables you to determine the best design variation among a model's possible variations. You create the original model, the *nominal design*, and then define the design parameters that vary, which can be nearly any design parameter assigned a numeric value in HFSS. (See the online help topic for the specific parameter you want to vary.) For example, you can parameterize the model geometry, material properties, or boundary conditions. You can then perform the following types of analyses on your nominal HFSS design:

Parametric	Define one or more *variable sweep definitions*, each specifying a series of variable values within a range. Optimetrics solves the design at each variation. You can then compare the results to determine how each design variation affects the performance of the design. Parametric analyses are often used as precursors to optimization solutions because they help to determine a reasonable range of variable values for the optimization analysis.
Optimization	Identify the cost function and the optimization goal. Optimetrics changes the design parameter values to meet that goal. The cost function can be based on any solution quantity that HFSS can compute, such as field values, S-parameters, and eigenmode data.
Sensitivity	Optimetrics explores the vicinity of the design point to determine the sensitivity of the design to small changes in variables.
Tuning	Variable values are changed interactively and the performance of the design is monitored.
Statistical	Optimetrics determines the distribution of a design's performance, which is caused by a statistical distribution of variable values.

Related Topics

Setting up a Parametric Analysis

Setting up an Optimization Analysis
Setting up a Sensitivity Analysis
Tuning a Variable
Setting up a Statistical Analysis

Setting up a Parametric Analysis

A *parametric setup* specifies all of the design variations that Optimetrics will drive HFSS to solve. A parametric setup is made up of one or more *variable sweep definitions*, which is a set of variable values within a range that you want HFSS to solve when you run the parametric setup.

You can define more than one parametric setup per design.

To add a parametric setup to a design:

1. On the **HFSS** menu, point to **Optimetrics Analysis**, and then click **Add Parametric**  .
   - Alternatively, right-click **Optimetrics** in the project tree, and then click **Add>Parametric** on the shortcut menu.

   The **Setup Sweep Analysis** dialog box appears.
2. Add a variable sweep definition.

Related Topics

Adding a Variable Sweep Definition

Technical Notes: Parametric Overview

Adding a Variable Sweep Definition

A parametric setup is made up of one or more *variable sweep definitions*. A variable sweep definition is a set of variable values within a range that Optimetrics drives HFSS to solve when the parametric setup is analyzed. You can add one or more sweep definitions to a parametric setup.

1. On the **HFSS** menu, point to **Optimetrics Analysis**, and then click **Add Parametric**  .
   - Alternatively, right-click **Optimetrics** in the project tree, and then click **Add>Parametric** on the shortcut menu.

   The **Setup Sweep Analysis** dialog box appears.
2. Under the **Sweep Definitions** tab, click **Add**.

   The **Add/Edit Sweep** dialog box appears.

   All of the independent variables associated with the design are listed in the **Variable** pull-down list.
3. Click the variable for which you are defining the sweep definition in the **Variable** pull-down list.

   If you do not define a sweep definition for a variable in the list, the variable's current value in the nominal design will be used in the parametric analysis.

Note Complex numbers are not allowed for variables to be used in an Optimetrics sweep, or for optimization, statistical, sensitivity or tuning setups.

4. Specify the variable values to be included in the sweep.
5. Click **Add**, and then click **OK**.

You return to the **Setup Sweep Analysis** dialog box. The variable sweep is listed in the top half of the window.

View the design variations that will be solved in table format under the **Table** tab. Viewing the sweep definition in table format enables you to visualize the design variations that will be solved and manually adjust sweep points if necessary.

Related Topics

Synchronizing Variable Sweep Definitions

Specifying Variable Values for a Sweep Definition

To specify the variable values to include in a sweep definition:

1. Select one of the following in the **Add/Edit Sweep** dialog box:

Single Value	Specify a single value for the sweep definition.
Linear Step	Specify a linear range of values with a constant step size.
Linear Count	Specify a linear range of values and the number, or count, of points within the variable range.
Decade Count	
Octave Count	
Exponential Count	

2. If you selected **Single Value**, type the value of the sweep definition in the **Value** text box.

   If you selected another sweep type, do the following:

   a. Type the starting value of the variable range in the **Start** text box.

   b. Type the final value of the variable range in the **Stop** text box.

3. If you selected **Linear Step** as the sweep type, type the step size in the **Step** text box.

   The step size is the difference between variable values in the sweep definition. The step size determines the number of design variations between the start and stop values. HFSS will solve the model at each step in the specified range, including the start and stop values.

   If you selected another sweep type, type the number of points, or variable values, in the sweep definition in the **Count** text box. The total number of points includes the start and stop values.

Note You can also edit the sweep Start, Stop, and Step values in the docked properties window for the sweep.

Synchronizing Variable Sweep Definitions

By default, variable sweep definitions are nested. Alternatively, you can synchronize the variable sweep definitions if they have the same number of sweep points.

For example, if you synchronize a sweep definition that includes values of 1, 2, and 3 inches with a second sweep definition that includes values of 4, 5, and 6 inches, HFSS will solve 3 design variations. The first variation is solved at the variable values of 1 and 4; the second variation is solved at the variable values 2 and 5; and the third variation is solved at the final variable values 3 and 6.

To synchronize variable sweep definitions:

1. Under the **Sweep Definitions** tab of the **Setup Sweep Analysis** dialog box, select the rows containing the sweep definitions you want to synchronize.
2. Click **Sync**.

   The synchronized sweeps are given a group number, which is listed in the **Sync #** column.

Optionally, view the design variations that will be solved in table format under the **Table** tab.

Modifying a Variable Sweep Definition Manually

You can manually modify the variable values that will be solved for a parametric setup by explicitly changing, adding, or deleting existing points in a variable sweep definition under the **Table** tab of the **Setup Sweep Analysis** dialog box.

To manually modify a variable sweep definition:

1. Click the **Table** tab of the **Setup Sweep Analysis** dialog box.

   The design variations HFSS will solve for the parametric setup are listed in table format.
2. Do one of the following:
   - To modify a variable value, click a value text box in the table and type a new value.
   - To delete a variable value from the sweep definition, click the row you want to delete, and then click **Delete**.
   - To add a new variable value to the sweep definition, click **Add**. Then click in the value text box and type a new value.

Your modifications are tracked and available for viewing at the bottom of the **Setup Sweep Analysis** dialog box under the **Sweep Definitions** tab. The operations you performed are listed with descriptions.

Warning If you want to modify an original sweep definition using the **Add/Edit Sweep** dialog box after you have manually modified its table of design variations, your manual modifications will become invalid and will be removed.

Overriding a Variable's Current Value in a Parametric Setup

If you choose not to sweep a variable, HFSS uses the variable's current value set for the nominal design when it solves the parametric setup. To override the current variable value for a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.

   Under **Starting Point**, all of the current independent design variable values are listed.
2. Click the **Value** text box of the variable with the value you want to override for the parametric setup.

3. Type a new value in the **Value** text box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for the parametric setup. For this parametric setup, the new value will override the current value in the nominal design.

Note Alternatively, you can select the **Override** option first, and then type a new variable value in the **Value** text box.

4. Optionally, click a new unit in the **Units** text box.

To revert to the current variable value, clear the **Override** option.

Specifying a Solution Setup for a Parametric Setup

To specify the solution setup that HFSS will analyze when it solves a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.
2. Select the solution setup you want HFSS to use when it solves the parametric setup.

   HFSS will solve the parametric setup using the solution setup you select. If you select more than one, results will be generated for all selected solution setups.

Specifying the Solution Quantity to Evaluate

When you add a parametric setup, you can identify one or more solution quantities to be presented in the **Post Analysis Display** dialog box. The solution quantities are specified by mathematical expressions that are composed of basic quantities, such as S-parameters, and output variables. When you view the results, HFSS extracts the solution quantities and lists them in the results table.

1. In the **Setup Sweep Analysis** dialog box, click the **Calculations** tab.
2. Click **Add**.
3. In the **Solution** text box, click the solution from which the solution quantity will be extracted.
4. In the **Calculation** text box, specify the solution quantity in one of the following ways:
   - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
   - If you want to create an output variable that represents the solution quantity, do the following:
     a. Click **Edit Calculation**.

        The **Output Variables** dialog box appears.

     b. Add the expression you want to evaluate, and then click **Done**.

        The most recently created output variable appears in the **Calculation** text box.

     c. To specify a different defined output variable, click the **Calculation** text box. It becomes a pull-down list that displays all of the defined output variables. Click a new output variable in the pull-down list.

Note The calculation you specify must be able to be evaluated into a single, real number.

Specifying a Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity will be extracted. For a parametric setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you selected to extract the solution from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you selected to extract the solution quantity from a frequency sweep solution, Optimetrics by default will use the starting frequency in the sweep.

1. In the **Setup Sweep Analysis** dialog box, click the **Calculations** tab.
2. Click **Edit Cal. Range**.

   The **Edit Calculation Range** dialog box appears.
3. In the **Variable** list, click an intrinsic variable.
4. **Single Value** is selected by default. In the **Value** text box, click a value at which the solution quantity will be extracted.
5. Click **Update**, and then click **OK**.

Setting up an Optimization Analysis

Following is the general procedure for setting up an optimization analysis.

1. Set up the variables you want to optimize in the **Properties** dialog box.
2. On the **HFSS** menu, point to **Optimetrics Analysis**, and then click **Add Optimization**  .

   The **Setup Optimization** dialog box appears.
3. Under the **Goals** tab, select an optimizer by clicking Sequential Non-Linear Programming, or Sequential Mixed Integer Non-Linear Programming, Quasi Newton, or Pattern Search Optimizer. in the **Optimizer** pull-down list. (Click the linked text for a discussion and recommendations.)

   Selecting either the Quasi Newton or Pattern Search optimizers enables the display of the **Acceptable Cost** and **Noise** fields. These fields do not appear for the SNLP or SMINLP optimizers.

   For the Quasi Newton and Patterns Search optimizers, under the **Variables** tab of the **Setup** dialog, selecting the **Advanced** button lets you specify a **Min Step** and **Max Step**. For the SNLP and SMINLP optimizers, under the **Variables** tab of the **Setup** dialog, selecting the **Advanced** button lets you specify values for Min focus and Max focus instead.
4. Type the maximum number of iterations you want HFSS to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. If you want to save the field solution data for every solved design variation in the optimization analysis, select **Save Fields**.
6. Under **Cost Function**, add a cost function.
7. For the Quasi Newton and Pattern Search optimizers:
   - Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box.
   - Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   1. Check the **Advanced options** checkbox.

      This displays the **Cost Function Norm Type** pull down menu.
   2. Select one of **L1**, **L2**, or **Maximum**.

Options for optimization analysis setup:

Modify the starting variable value.

Modify the minimum and maximum values of variables that will be optimized.

Exclude variables from optimization.

Modify the values of fixed variables that are not being optimized.

Set the minimum and maximum step size between solved design variations.(For the Quasi Newton and Patterns Search optimizers, Variables tab, Advanced button)

Set the minimum and maximum focus size. (For the SNLP and SMINLP optimizers, Variables tab, Advanced button).

Set linear constraints.

Request that Optimetrics solve a parametric sweep before an optimization analysis.

Request that Optimetrics solve a parametric sweep during an optimization analysis.

Automatically update optimized variables to the optimal values after an optimization analysis is completed.

Related Topics

Technical Notes: Optimization Overview

Technical Notes: Acceptable Cost

Setting the Maximum Iterations for an Optimization Analysis

The **Max. No. of Iterations** value is the maximum number of design variations that you want Optimetrics to solve during an optimization. This value is a stopping criterion; if the maximum number of iterations has been completed, the optimization analysis stops. If the maximum number of iterations has not been completed, the optimization continues by performing another iteration, that is, by solving another design variation. It performs iterations until the acceptable cost function is reached or until the optimizer cannot proceed as a result of other optimization setup constraints, such as when it searches for a variable value with a step size smaller than the minimum step size.

To set the maximum number of iterations for an optimization analysis:

- Under the **Goals** tab of the **Setup Optimization** dialog box, type a value in the **Max. No. of Iterations** text box.

Adding a Cost Function

A cost function can include one or more goals for an optimization analysis. Optimetrics manipulates the model's design variable values to fulfill the cost function.

Following is the general procedure for adding a cost function with a single goal.

1. Under the **Goals** tab of the **Setup Optimization** dialog box, click **Add**.

   A new row is added to the **Cost Function** table.
2. In the **Solution** column, click the solution from which the cost function will be extracted.
3. Specify a solution quantity on which to base the cost function goal.
4. In the **Calc. Range** text box, set the range within which the cost function will be computed.

5. In the **Condition** text box, click a condition from the pull-down list:

<=	less than or equal to
=	equal to
>=	greater than or equal to
Minimize	reduce the cost function to a minimum value.
Maximize	identify a maximized condition.

6. In the **Goal** text box, type the value of the solution quantity that you want to be achieved during the optimization analysis. If the solution quantity is a complex calculation, the goal value must be complex; two goal values must be specified. The **Minimize** and **Maximize** options do not require you to specify a Goal value.
7. Optionally, if you have multiple goals and you want to assign higher or lower priority to a goal, type a different value for the goal's weight in the **Weight** text box. The goal with the greater weight is given more importance. If the goal is a complex value, the weight value must be complex; two weight values must be specified. The weight value cannot be variable dependent.

The optimization will stop when the solution quantity meets the acceptable cost criterion.

Related Topics

Technical Notes: Cost Function

Technical Notes: Acceptable Cost

Setting a Goal Value

Technical Notes: Goal Weight

Specifying a Solution Quantity for a Cost Function Goal

When setting up a cost function, you must identify the solution quantity on which to base each goal. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as S-parameters, and output variables.

1. Add a row (a goal) to the cost function table:
   a. Under the **Goals** tab of the **Setup Optimization** dialog box, click **Add**.

      A new row is added to the **Cost Function** table.
   b. In the **Solution** column, click the solution from which the cost function will be extracted.
2. In the **Solution** text box, click the solution from which the solution quantity will be extracted.
3. In the **Calculation** text box, specify the solution quantity in one of the following ways:
   - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
   - If you want to create an output variable that represents the solution quantity, do the following:
     a. Click **Edit Calculation**.

The **Output Variables** dialog box appears.

b. Add the expression you want to evaluate, and then click **Done**.

HFSS entered the most recently created output variable in the **Calculation** text box.

c. To specify a different defined output variable, click the **Calculation** text box. It becomes a pull-down list that displays all of the defined output variables. Click an output variable in the pull-down list.

Setting the Calculation Range of a Cost Function Goal

The calculation range is the range within which you want a cost function goal to be calculated. It can be a single value or a range of values, depending on the solution or solution quantity selected for the goal.

1. Under the **Goals** tab in the **Setup Optimization** dialog box, click **Edit Calc. Range**.
2. In the **Variable** pull-down list, click a variable.

   If you chose to solve a parametric setup during the optimization analysis, the variables swept in that parametric setup are available in the **Variable** pull-down list. If you sweep a variable in the parametric setup that is also being optimized, that variable will be excluded from the optimization.

   Other examples of available variables include frequency, if the solution quantity is an S-parameter quantity, and phi or theta, if the solution quantity is a radiated field quantity.
3. After you select a variable in the **Variable** pull-down list, you can select a range of values for the calculation range as follows:
   a. Select **Range**.
   b. In the **Start** text box, type the starting value of the range.
   c. In the **Stop** text box, type the final value of the range.
4. To select a single value for the calculation range:
   a. Select **Single Value**.
   b. In the **Value** text box, type the value of the variable at which the cost function goal will be extracted.
5. Click **Update**, and then click **OK**.

Setting a Goal Value

A goal is the value you want a solution quantity to reach during an optimization analysis. It can be a real value or a complex value. If the solution quantity is a complex calculation, the goal value must be complex. You can type the goal value in the **Goal** text box. Alternatively, you can use the **Edit Goal/Value Weight** dialog box to specify the goal value as a single value, a mathematical expression, or dependent on a variable such as frequency.

What do you want to do?

Specify a single goal value.

Specify an expression as the goal value.

Specify a variable-dependent goal value.

Specifying a Single Goal Value

1. Under the **Goals** tab in the **Setup Optimization** dialog box, click **Edit Goal/Weight**.

   The **Edit Goal/Weight** dialog box appears.
2. Under the **Goal Value** tab, click **Simple Numeric Value** in the **Type** list.
3. If the goal value is complex, click **real/imag** in the pull-down list to the right if you want to specify the real and imaginary parts of the goal value.

   Alternatively, click **mag/ang** if you want to specify the magnitude and angle of the goal value.
4. Type the goal value in the **Goal Value** table.

   If the goal value is complex, type both parts of the goal value in the text box below the **Goal Value** heading. For example, type **1, 1** to specify the real part of the goal value as 1 and the imaginary part as 1.

   If the goal value is real, type a real goal value in the text box below the **Goal Value** heading.
5. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying an Expression as a Goal Value

1. Under the **Goals** tab in the **Setup Optimization** dialog box, click **Edit Goal/Weight**.

   The **Edit Goal/Weight** dialog box appears.
2. Under the **Goal Value** tab, click **Expression** in the **Type** list.
3. If you know the syntax of the mathematical expression or the existing output variable's name, type it in the text box below the **Goal Value** heading.

   Alternatively, if you want to create an output variable that represents the goal value, do the following:

   a. Click **Edit Expression**.

      The **Output Variables** dialog box appears.

   b. Add the expression you want to be the goal value, and then click **Done**.

      HFSS has entered the most recently created output variable in the text box below the **Goal Value** heading.
4. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying a Variable-Dependent Goal Value

1. Under the **Goals** tab in the **Setup Optimization** dialog box, click **Edit Goal/Weight**.

   The **Edit Goal/Weight** dialog box appears.
2. Under the **Goal Value** tab, click **Variable Dependent** in the **Type** list.

3. Click a variable in the pull-down list to the left of the table.
4. Type the value of that variable in the first column of the table.
5. Type a corresponding goal value for that variable value in the text box below the **Goal Value** heading.
6. Click **Add** to add another row to the reference curve.
7. Repeat steps 4, 5, and 6 until you have specified the reference curve.
8. Click **OK**.

   The goal value is listed as being variable dependent in the **Goal** text box.

StartingVariableValueOptimization

Modifying the Starting Variable Value for Optimization

A variable's starting value is the first value that will be solved during the optimization analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each optimization setup.

Note If you choose to solve a parametric setup before an optimization analysis, a variable's starting value will be ignored if a more appropriate starting value is calculated for it during the parametric analysis.

1. In the **Setup Optimization** dialog box, click the **Variables** tab.

   All of the variables that were selected for the optimization analysis are listed.
2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for this optimization analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.
3. Optionally, click a new unit system in one of the **Units** text boxes.

Note To revert to the default starting value, clear the **Override** option.

MinMaxValuesOptimization

Setting the Min. and Max. Variable Values for Optimization

For every optimization setup, Optimetrics automatically sets the minimum and maximum values that it will consider for a variable being optimized. Optimetrics sets a variable's minimum value equal to approximately one-half its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately one and one-half the starting value. During optimization analysis, variable values that lie outside of this range are not considered.

What do you want to do?

Override the default minimum and maximum variable values for a single optimization setup.

Change the default minimum and maximum variable values for every optimization setup.

Overriding the Min. and Max. Variable Values for a Single Optimization Setup

1. In the **Setup Optimization** dialog box, click the **Variables** tab.

   All of the variables that were selected for optimization analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for this optimization analysis; the variable's current **Min** or **Max** value in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default minimum and maximum values, clear the **Override** option.

Changing the Min. and Max. Variable Values for Every Optimization Setup

1. Make sure that the variable's minimum and maximum values are not being overridden in any optimization setup.
2. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.

3. Select **Optimization**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   When Optimetrics solves an optimization setup, it will not consider variable values that lie outside of this range.

Setting the Min. and Max. Step Sizes

For the Quasi Newton and Pattern Search optimizers, the step size is the difference in a variable's value between one solved design variation and the next. The step size is determined when Optimetrics locates the next design variation that should be solved in an effort to meet the cost function.

1. In the **Setup Optimization** dialog box, click the **Variables** tab.
2. Click **Advanced**.

   Optimetrics displays **Min. Step** and **Max. Step** columns, with default values for each variable to be optimized.

3. In the **Min. Step** text box, type the minimum step size value. Optionally, modify the unit system in the **Units** text box.

4. In the **Max. Step** text box, type the maximum step size value. Optionally, modify the unit system in the **Units** text box.

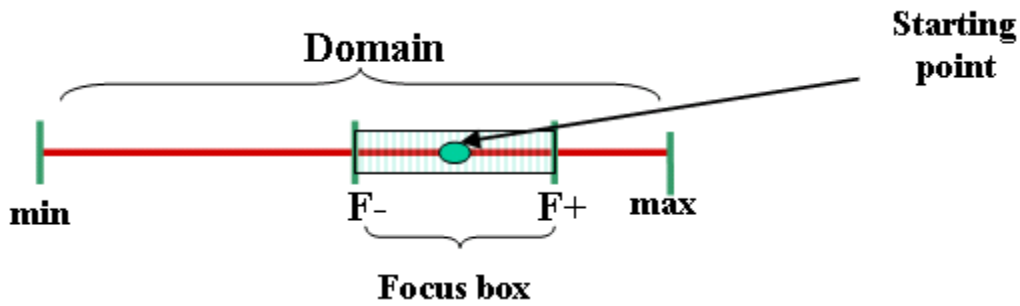
Hint A value of zero is recommended for the minimum step size.

Related Topics

Technical Notes: Step Size

Setting the Min and Max Focus

For the SNLP and SMINLP optimizers, the min focus and max focus criteria allow you to specify a subrange of parameter values where the optimizer should look when performing the optimization. This focus box is where you suspect the optimal solution will be, so it is a hint for the optimizer.



- The domain limits the search. The domain = physical limits.
- The focus box doesn't limit the search. Rather, the Focus box = an initial guess of optimum search domain. The starting point is the center of the focus box, but the search does extend beyond the box.
- This focus must be inside the domain limits. Consequently, it has to be equal or smaller size.

Equalizing the influence of different optimization variables.

The optimizer seeks optimal values for the optimization variables. These variables are usually quantities with specified units. The change in one variable could be measured in [mm] and the change in other variable could be [mA]. Instead of those units, the optimizer uses internal abstract units, so that a change in one variable changes the design behavior about as much as the same change in another variable, where changes are measured in the respective internal abstract units. Now, when you define the focus box, the unit of the abstract internal unit is defined as the difference of the upper and lower focus limits. This way you can use the focus box to equalize the influence of different optimization variables on the design behavior.

To set the Min and Max Focus values:

1. In the **Setup Optimization** dialog box, click the **Variables** tab.
2. Click **Advanced**.

   Optimetrics displays **Min. Focus** and **Max. Focus** columns, with default values for each vari-

able to be optimized.

If you don't have an initial guess based on your knowledge of the problem, make focus box equal to domain; that is, the physical limits. This tells SNLP to search the entire decision space.

- In the **Min. Focus** text box, type the minimum value of the focus range. Optionally, modify the unit system in the **Units** text box.
- In the **Max. Focus** text box, type the maximum value of the focus range. Optionally, modify the unit system in the **Units** text box.

Solving a Parametric Setup Before an Optimization

Solving a parametric setup before an optimization setup is useful for guiding Optimetrics during an optimization.

To solve a parametric setup before an optimization setup:

1. In the **Setup Optimization** dialog box, click the **General** tab.
2. In the **Parametric Analysis** pull-down list, click the parametric setup you want Optimetrics to solve before optimization.

Note The parametric setup must include sweep definitions for the variables you are optimizing.

3. Select **Solve the parametric sweep before optimization**.

If the parametric setup has not yet been solved, Optimetrics will solve it. Optimetrics uses the cost value evaluated at each parametric design variation to determine the next step in the optimization analysis. This enables you to guide the direction in which the optimizer searches for the optimal design variation.

Solving a Parametric Setup During an Optimization

Solving a parametric setup during an optimization analysis is useful when you want Optimetrics to solve every design variation specified in the parametric setup at each optimization iteration. A cost function goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during an optimization analysis:

1. In the **Setup Optimization** dialog box, click the **General** tab.
2. In the **Parametric Analysis** pull-down list, click the parametric setup you want Optimetrics to solve during an optimization.
3. Select **Solve the parametric sweep during optimization**.

Automatically Updating a Variable's Value After Optimization

When Optimetrics finds an optimal variable value by solving an optimization setup, it can automatically update that variable's current value set for the nominal model to the optimal value.

1. In the **Setup Optimization** dialog box, click the **General** tab.
2. Select **Update design parameters' values after optimization**.

When optimization is complete, the current variable value for each optimized variable will be changed to the optimal value.

Setting up a Sensitivity Analysis

Following is the general procedure for setting up a sensitivity analysis.

1. Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** dialog box.
2. On the **HFSS** menu, point to **Optimetrics Analysis**, and then click **Add Sensitivity**  .

   The **Setup Sensitivity Analysis** dialog box appears.
3. Under the **Calculations** tab, type the maximum number of iterations per variable value that you want HFSS to perform in the **Max. No. of Iterations/Sensitivity Variable** text box.
4. If you want to save the field solution data for every design variation solved during the sensitivity analysis, select **Save Fields**.
5. Under **Output Parameters**, set up an output parameter.
6. Specify the value of the design point at which the sensitivity analysis should stop in the **Approximate Error in Master Output** text box.

Optional sensitivity analysis setup options:

Modify the starting variable value.

Modify the minimum and maximum values of variables that will be solved.

Exclude variables from the sensitivity analysis.

Set the initial displacement.

Modify the values of fixed variables that are not being modified during the sensitivity analysis.

Set linear constraints.

Request that Optimetrics solve a parametric sweep before a sensitivity analysis.

Request that Optimetrics solve a parametric sweep during a sensitivity analysis.

Related Topics

Technical Notes: Sensitivity Analysis Overview

Setting the Maximum Iterations Per Variable

The **Max. No. of Iterations/Sensitivity Variable** value is the maximum number of design variations that Optimetrics will solve per variable during a sensitivity analysis. This value is a stopping criterion; if the maximum number of iterations has been completed, the sensitivity analysis stops. If the maximum number of iterations has not been completed, the sensitivity analysis continues by performing another iteration, that is, by solving another design variation. It performs iterations until the approximate error in master output value is reached or Optimetrics cannot proceed as a result of other sensitivity setup constraints, such as when it searches for a variable value that is larger than the maximum value.

To set the maximum number of iterations for a sensitivity analysis:

- Under the **Calculations** tab of the **Setup Sensitivity Analysis** dialog box, type a value in the **Max. No. of Iterations/Sensitivity Variable** text box.

Setting up an Output Parameter

Following is the general procedure for adding an output parameter to a sensitivity setup.

1. Under the **Calculations** tab of the **Setup Sensitivity Analysis** dialog box, click **Add**.

   A new row is added to the **Output Parameters** table.
2. In the **Solution** column, click the solution from which the output parameter will be extracted.
3. Specify a solution quantity on which to base the output parameter.
4. In the **Calc. Range** text box, set the value of the variable at which the output parameter will be computed.
5. If you have more than one output parameter, select **Master Output** if you want Optimetrics to use the output parameter to base its selection of solved design variations.

Related Topics

Technical Notes: Selecting the Master Output

Specifying a Solution Quantity for an Output Parameter

When setting up an output parameter, you must identify the solution quantity on which to base the output parameter. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as S-parameters; and output variables.

1. Add a row to the output parameters table: Under the **Calculations** tab of the **Setup Sensitivity Analysis** dialog box, click **Add**.

   A new row is added to the **Output Parameters** table.
2. In the **Solution** text box, click the solution from which the solution quantity will be extracted.
3. In the **Calculation** text box, specify the solution quantity in one of the following ways:
   - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
   - If you want to create an output variable that represents the solution quantity, do the following:
     a. Click **Edit Calculation**.

        The **Output Variables** dialog box appears.
     b. Add the expression you want to evaluate, and then click **Done**.

        HFSS enters the most recently created output variable in the **Calculation** text box.
     c. To specify a different defined output variable, click the **Calculation** text box. It becomes a pull-down list that displays all of the defined output variables. Click a new

output variable in the pull-down list.

Note The solution quantity you specify must be able to be evaluated to a single, real number.

Setting the Calculation Range of an Output Parameter

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity will be extracted. For a sensitivity setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you specified that the solution quantity be extracted from a frequency sweep solution, by default Optimetrics will use the starting frequency in the sweep.

1. Under the **Calculations** tab of the **Setup Sensitivity Analysis** dialog box, click **Edit Calc. Range**.

   The **Edit Calculation Range** dialog box appears.

2. In the **Variable** pull-down list, click a variable.

   If you chose to solve a parametric setup during the sensitivity analysis, the variables swept in that parametric setup are available in the **Variable** pull-down list. If you sweep a variable in the parametric setup that is also a sensitivity variable, that variable will be excluded from the sensitivity analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if you selected a radiated field quantity as the solution quantity.

3. The **Single Value** option is selected by default. In the **Value** text box, type the value of the variable at which the solution quantity will be extracted.

4. Click **Update**, and then click **OK**.

StartingVariableValueSensitivity

Modifying the Starting Variable Value for Sensitivity Analysis

The design point of the sensitivity analysis is the starting value of the sensitivity variable and is usually the first variation to be solved. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify the design point for each sensitivity setup.

1. In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.

   All of the variables that were selected for the sensitivity analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for this sensitivity analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

MinMaxVariableValuesSensitivity

Setting the Min. and Max. Variable Values for Sensitivity

For every sensitivity setup, Optimetrics automatically sets the minimum and maximum values that it will consider for a sensitivity variable. Optimetrics sets a variable's minimum value equal to approximately one-half its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 1 1/2 the starting value. During sensitivity analysis, variable values that lie outside of this range will not be considered.

What do you want to do?

Override the default minimum and maximum variable values for a single sensitivity setup.

Change the default minimum and maximum variable values for every sensitivity setup.

Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup

1. In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.

   All of the variables that were selected for sensitivity analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for this sensitivity analysis; the variable's current **Min** or **Max** value set in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default minimum and maximum values, clear the **Override** option.

Changing the Min. and Max. Variable Values for Every Sensitivity Setup

1. Make sure that the variable's minimum and maximum values are not being overridden in any sensitivity setup.
2. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.
3. Select **Sensitivity**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   When Optimetrics solves a sensitivity setup, it will not consider variable values that lie outside of this range.

Setting the Initial Displacement

The initial displacement is the difference in a variable's starting value and the next solved design variation. During the sensitivity analysis, Optimetrics will not consider an initial variable value that is greater than this step size away from the starting variable value.

1. In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Advanced**.

   Optimetrics displays the **Initial Displacement** column, with default values for each sensitivity variable.
3. In the **Initial Displacement** text box, type the initial displacement value. Optionally, modify the unit system in the **Units** text box.

Solving a Parametric Setup Before a Sensitivity Analysis

Solving a parametric setup before a sensitivity setup is useful for guiding Optimetrics in a sensitivity analysis.

To solve a parametric setup before a sensitivity setup:

1. In the **Setup Sensitivity Analysis** dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve before the sensitivity setup from the **Parametric Analysis** pull-down list.

Note The parametric setup must include sweep definitions for the sensitivity variables.

3. Select **Solve the parametric sweep before analysis**.

If the parametric setup has not yet been solved, Optimetrics will solve it. Optimetrics will use the results (of the solution calculation you requested under the **Goals** tab of the **Setup Sensitivity** dialog box) to determine the next design variation to solve for the sensitivity analysis.

Solving a Parametric Setup During a Sensitivity Analysis

Solving a parametric setup during a sensitivity analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each sensitivity analysis iteration. An output parameter goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during a sensitivity analysis:

1. In the **Setup Sensitivity Analysis** dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the sensitivity analysis from the **Parametric Analysis** pull-down list.
3. Select **Solve the parametric sweep during analysis**.

Setting up a Statistical Analysis

Following is the general procedure for setting up a statistical analysis.

1. Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** dialog box.
2. On the **HFSS** menu, point to **Optimetrics Analysis**, and then click **Add Statistical**  .

   The **Setup Statistical Analysis** dialog box appears.
3. Under the **Calculations** tab, type the maximum number of iterations that you want HFSS to perform in the **Maximum Iterations** text box.
4. If you want to save the field solution data for every design variation solved during the statistical analysis, select **Save Fields**.
5. Specify a solution quantity to evaluate.
6. In the **Calc. Range** text box, set the value at which the solution quantity will be computed.
7. Optionally, modify the distribution criteria that will be used.

Other optional statistical analysis setup options:

Modify the starting variable value.

Exclude variables from the statistical analysis.

Modify the values of fixed variables that are not being modified during the statistical analysis.

Request that Optimetrics solve a parametric sweep during a statistical analysis.

Setting the Maximum Iterations for a Statistical Analysis

The **Maximum Iterations** value is the maximum number of design variations that Optimetrics will solve during a statistical analysis. This value is a stopping criterion; if the maximum number of iterations has been completed, the analysis stops. If the maximum number of iterations has not been completed, Optimetrics continues by performing another iteration, that is, by solving another design variation.

To set the maximum number of iterations for a statistical analysis:

- Under the **Calculations** tab of the **Setup Sensitivity Analysis** dialog box, type a value in the **Maximum Iterations** text box.

SolutionQuantityStatistical

Specifying the Solution Quantity to Evaluate

When you add a statistical setup, you can identify one or more solution quantities to evaluate in the **Post Analysis Display** dialog box. The solution quantities are specified by mathematical expres-

sions that are composed of basic quantities, such as S-parameters; and output variables. When you view the results, HFSS displays the distribution of the solution quantities.

1. In the **Setup Statistical Analysis** dialog box, click the **Calculations** tab.
2. Click **Add**.
3. In the **Solution** text box, click the solution from which the solution quantity will be extracted.
4. In the **Calculation** text box, specify the solution quantity in one of the following ways:
   - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
   - If you want to create an output variable that represents the solution quantity, do the following:
     a. Click **Edit Calculation**.

        The **Output Variables** dialog box appears.

     b. Add the expression you want to evaluate, and then click **Done**.

        The most recently created output variable appears in the **Calculation** text box.

     c. To specify a different defined output variable, click the **Calculation** text box. It becomes a pull-down list that displays all of the defined output variables. Click a new output variable in the pull-down list.

Note The solution quantity you specify must be able to be evaluated into a single, real number.

CalculationRangeStatistical

Setting the Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity will be extracted. For a statistical setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you specified that the solution quantity be extracted from a frequency sweep solution, Optimetrics will use the starting frequency in the sweep by default.

1. Under the **Calculations** tab of the **Setup Statistical Analysis** dialog box, click **Edit Calc. Range**.

   The **Edit Calculation Range** dialog box appears.

2. In the **Variable** pull-down list, click a variable.

   If you chose to solve a parametric setup during the statistical analysis, the variables swept in that parametric setup are available in the **Variable** pull-down list. If you sweep a variable in the parametric setup that is also a statistics variable, that variable will be excluded from the statistics analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. **Single Value** is selected by default. In the **Value** text box, type the value of the variable at which the solution quantity will be extracted.
4. Click **Update**, and then click **OK**.

Setting the Distribution Criteria

For every statistical setup, Optimetrics automatically sets the distribution criteria to be uniform within a 10% tolerance of the variable's starting value. You can modify the distribution type and criteria for a single statistical setup or for every statistical setup.

What do you want to do?

Override the default distribution criteria for a single statistical setup.

Change the default distribution criteria for every statistical setup.

Overriding the Distribution Criteria for a Single Statistical Setup

To override the default distribution criteria for a single statistical setup:

1. In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

   All of the variables that were selected for statistical analysis are listed.
2. Select **Uniform** or **Gaussian** in the **Distribution** column for the variable you want to override.

   If you changed the distribution type, the **Override** option is now selected. This indicates that the distribution type you selected will be used for this optimization analysis; the current distribution type selected for the variable in the nominal design will be ignored in this statistical analysis.

   - Alternatively, you can select the **Override** option first, and then select a different distribution type in the **Distribution** text box.
3. Optionally, if you want to change the distribution criteria, click in **Distribution Criteria** column for the variable you want to override.

   The **Edit Distribution** dialog box appears.
4. If the distribution type is **Gaussian**, do the following:
   a. Type the standard deviation in the **Std. Dev** text box.
   b. Type the lower limit of the distribution in the **Low Cutoff** text box.
   c. Type the upper limit of the distribution in the **High Cutoff** text box.

      HFSS will solve design variations using a Gaussian distribution within the low and high cutoff values.
5. If the distribution type is **Uniform**, do the following:
   - Type a tolerance value in the text box.

     HFSS will solve design variations within the tolerance range of the starting value, using an even distribution.
6. Click **OK**.

To revert to the default distribution settings, clear the **Override** option.

Changing the Distribution Criteria for Every Statistical Setup

To change the default distribution criteria for every statistical setup:

1. Make sure that the variable's distribution criteria are not being overridden in any statistical setup.
2. If the variable is a design variable, do the following: On the **HFSS** menu, click **Design Properties**.

   If the variable is a project variable, do the following: On the **Project** menu, click **Project Variables**.

   The **Properties** dialog box appears.
3. Select **Statistics**.
4. Click in the **Distribution** column for the variable you want to change, and then select **Uniform** or **Gaussian**.
5. Optionally, if you want to change the distribution criteria, click in the **Distribution Criteria** column for the variable you want to change.

   If the distribution type is **Gaussian**, the **Gaussian Distribution** dialog box appears. If the distribution type is **Uniform**, the **Uniform Distribution** dialog box appears.
6. If the distribution type is **Gaussian**, do the following:
   a. Type the standard deviation in the **Std. Dev** text box.
   b. Type the lower limit of the distribution in the **Low Cutoff** text box.
   c. Type the upper limit of the distribution in the **High Cutoff** text box.

      HFSS will solve design variations using a Gaussian distribution within the low and high cutoff values.
7. If the distribution type is **Uniform**, do the following:
   - Type a tolerance value in the text box.

     HFSS will solve design variations within the tolerance range of the starting value, using an even distribution.
8. Click **OK**.

StartingVariableValueStatistical

Modifying the Starting Variable Value for Statistical Analysis

A variable's starting value is the first value that will be solved during the statistical analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each statistical setup.

1. In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

   All of the variables that were selected for the statistical analysis are listed.
2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for this statistical analysis; the current value set for the nominal model will be ignored.

- Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Solving a Parametric Setup During a Statistical Analysis

Solving a parametric setup during a statistical analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each statistical analysis iteration.

To solve a parametric setup during a statistical analysis:

1. In the **Setup Statistical Analysis** dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the statistical analysis from the **Parametric Analysis** pull-down list.
3. Select **Solve the parametric sweep during analysis**.

Tuning a Variable

1. Before a variable can be tuned, you must specify that you intend for it to be used during a tuning analysis in the **Properties** dialog box.
2. On the **HFSS** menu, point to **Optimetrics Analysis**, and then click **Tune**  .

   The **Tune** dialog box appears.
3. Clear the **Real Time** option.

   If this option is selected, a simulation begins immediately after you move the slider.
4. If you want to save the field solution data for the design variations solved during a tuning analysis, select **Save Fields**.
5. In the **Sim. Setups** column, select the solution setup you want HFSS to use when it solves the specified design variation.

   HFSS will solve the analysis using the solution setup you select. If you select more than one, results are generated for all selected solution setups.
6. In the **Nominal** text box for the variable you want to tune, type the value of the variable you want HFSS to solve, or drag the slider to increase or decrease its value.

   Alternatively, if you want HFSS to solve a range of values, specify a linear range of values with a constant step size:

   a. Select **Sweep**.

   b. In the text box below the **Step** value, type the starting value in the variable range.

   c. Type the step size, or difference between variable values in the sweep definition, in the **Step** text box. The step size determines the number of design variations between the start and stop values. HFSS will solve the model at each step in the specified range, including the start and stop values.

   d. In the text box just below the variable name, type a stopping value in the variable range.
7. Click **Tune**.

Related Topics

Applying a Tuned State to a Design

Technical Notes: Tuning Analysis Overview

Resetting Variable Values after Tuning

Applying a Tuned State to a Design

You can apply the variable values solved during a tuning analysis to the nominal design in one of the following three ways:

- When closing the **Tune** dialog box:
  1. Click **Close** to exit the **Tune** dialog box.

     The **Apply Tuned Variation** dialog box appears.
  2. Click the design variation you want to apply, and then click **OK**.

The variable values from the solved design variation become the current variable values for the nominal design.

- When saving a tuned state.
- When reverting to a tuned state.

Saving a Tuned State

You can save the settings in the **Tune** dialog box, including the variable values you specified for a tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. After tuning a variable, click **Save** in the **Tune** dialog box.

   A **Save As** dialog box appears.
2. Type a name for the tuned state in the text box.
3. Select **Apply tuned values to design** if you want to update the model to the new variable values.
4. Click **OK** to return to the **Tune** dialog box.

Related Topics

Reverting to a Saved Tuned State

Reverting to a Saved Tuned State

You can revert to a group of saved settings in the **Tune** dialog box, including the variable values you specified for a specific tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. In the **Tune** dialog box, click **Revert**.

   The **Revert** dialog box appears.
2. Type the name of the tuned state you want to apply or click a name in the pull-down list.
3. Select **Apply tuned values to design** if you want to update the model to the selected tuned state's variable values.
4. Click **OK** to return to the **Tune** dialog box.

Related Topics

Saving a Tuned State

Resetting Variable Values after Tuning

If you want to reset variable values to the values they were set to when you started the current session of the **Tune** dialog box:

- After tuning a variable, click **Reset** in the **Tune** dialog box.

  Solutions for the design variations solved during tuning analyses remain available for post processing.

Saving Field Solutions for Optimetrics Analyses

In order to preserve disk space, by default HFSS does not save field solution data for every solved design variation in an optimization analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

To save the fields for all design variations, change the default setting for all projects:

1. On the **Tools** menu, point to **Options**, and then click **HFSS Options**.

   The **HFSS Options** dialog box appears.

2. Under the **General** tab, select **Save Optimetrics field solutions**.

   Save Fields is selected by default when you create a new Optimetrics setup.

Related Topics

Saving Field Solutions for a Parametric Setup

Saving Field Solutions for an Optimization Setup

Saving Field Solutions for a Sensitivity Setup

Saving Field Solutions for a Tuning Analysis

Saving Field Solutions for a Statistical Setup

Copy Geometrically Equivalent Meshes

Saving Field Solutions for a Parametric Setup

In order to preserve disk space, by default HFSS does not save field solution data for every solved design variation in a parametric setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the parametric setup, by default field solutions will not be available.

To save the fields for all design variations solved during a parametric analysis:

1. Either **Add Sweep** or right click on an existing sweep to open the **Setup Sweep Analysis** dialog box.
2. Select the **Options** tab.
3. Click the **Save Fields And Mesh** check box. Optionally, select **Copy geometrically equivalent meshes**.

   HFSS will save the field solution data for every solved design variation in the parametric setup.

Related Topics

Saving Field Solutions for Optimetrics Analyses

Saving Field Solutions for an Optimization Setup

In order to preserve disk space, by default HFSS does not save field solution data for every solved design variation in an optimization analysis. It only saves the field solutions for the nominal design

when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

To save the fields for all design variations solved during an optimization analysis:

1. Open an **Edit Sweep** dialog by either adding a sweep or right-click on a an existing sweep to view the short cut menu and selecting Properties.
2. Select the **Options** tab.
3. Click the **Save Fields And Mesh** check box. Optionally, select **Copy geometrically equivalent mashes**.

   HFSS will save the field solution data for every solved design variation in the optimization setup.

Related Topics

Saving Field Solutions for Optimetrics Analyses

Saving Field Solutions for a Sensitivity Setup

In order to preserve disk space, by default HFSS does not save field solution data for every solved design variation in a sensitivity analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the sensitivity analysis, all field solutions are deleted.

To save the fields for all design variations solved during a sensitivity analysis:

1. Open the **Setup Sensitivity Analysis** dialog box.
2. Select the **Options** tab.
3. Click the **Save Fields And Mesh** check box. Optionally, select **Copy geometrically equivalent mashes**.

   HFSS will save the field solution data for every solved design variation in the sensitivity analysis.

Related Topics

Saving Field Solutions for Optimetrics Analyses

Saving Field Solutions for a Tuning Analysis

In order to preserve disk space, by default HFSS does not save field solution data for every design variation solved in a tuning analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the tuning analysis, all field solutions are deleted.

To save the fields for all design variations solved during a tuning analysis:

- In the **Tuning** dialog box, select **Save Fields**.

  HFSS will save the field solution data for every solved design variation in a tuning analysis.

Related Topics

Saving Field Solutions for Optimetrics Analyses

Saving Field Solutions for a Statistical Setup

In order to preserve disk space, by default HFSS does not save field solution data for every design variation solved in a statistical analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the statistical analysis, all field solutions are deleted.

To save the fields for all design variations solved during a statistical analysis:

1. Open the **Setup Statistical Analysis** dialog box.
2. Select the **Options** tab.
3. Click the **Save Fields And Mesh** check box. Optionally, select **Copy geometrically equivalent mashes**.

   HFSS will save the field solution data for every solved design variation in the statistical setup.

Related Topics

Saving Field Solutions for Optimetrics Analyses

Copying Meshes in Optimetrics Sweeps

An option in the Optimetrics Analysis setups allows you to request HFSS to copy a mesh that was calculated for one sweep variation for reuse on a geometrically-equivalent sweep variation. For example, with this option selected a sweep on a scan angle would not need to generate meshes for each solution. The option is available on the setups for sweeps on parametrics, optimization, sensitivity, and statistics.

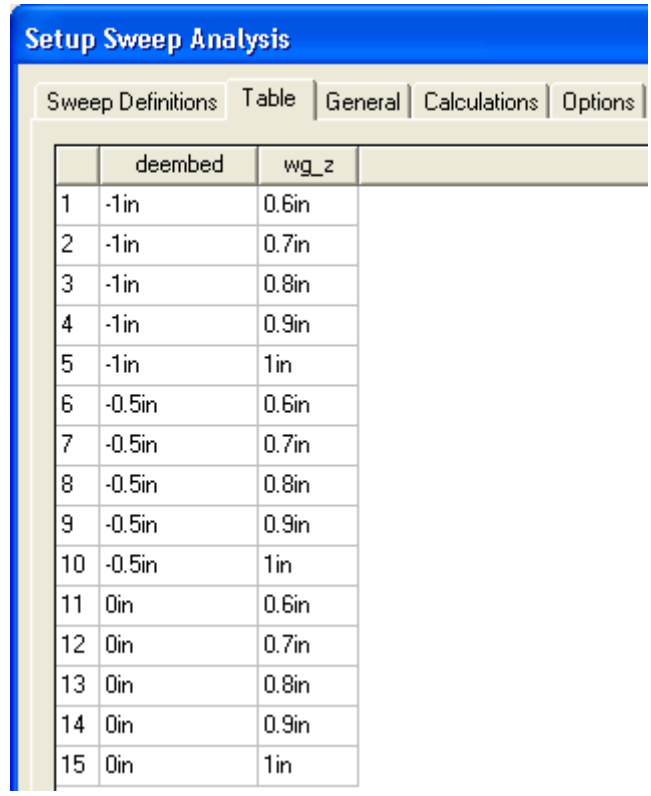
To copy and reuse meshes on geometrically-equivalent parametric variations:

1. Define a variable for the kind of Optimetrics sweep you intent to setup.
2. Select the appropriate **HFSS>Optimetrics>Add** command to display a **Setup** dialog box.
3. Click the **Options** tab in the Setup dialog box.
4. Select **Copy geometric equivalent meshes**.

   HFSS will copy the mesh solution calculated for a particular parametric sweep for reuse on each geometrically-equivalent sweep variation.

Note This option is available with all Optimetrics setups, and is applied when these analyses generate geometrically-equivalent values. However, it is most relevant to parametric sweep, where such equivalences are more likely to occur.

Here is an example. Suppose the sweep is over two parameters: **deembed** (a non-geometric parameter), and **wg_z** (a geometric parameter). The setup table below shows the variations (combinations of values) of these two parameters that will be swept:



	deembed	wg_z
1	-1in	0.6in
2	-1in	0.7in
3	-1in	0.8in
4	-1in	0.9in
5	-1in	1in
6	-0.5in	0.6in
7	-0.5in	0.7in
8	-0.5in	0.8in
9	-0.5in	0.9in
10	-0.5in	1in
11	0in	0.6in
12	0in	0.7in
13	0in	0.8in
14	0in	0.9in
15	0in	1in

Since **deembed** is not a geometric parameter, the five values of **wg_z** represent geometrically-equivalent variations. Each value of **wg_z** occurs three times, so there are five sets of geometric equivalences, each set containing three sweep variations. The first set contains variations #1, #6, and #11 (**wg_z** = 0.6 in). When the **Copy geometric equivalent meshes** option is in effect, the mesh that is calculated for variation #1 will be copied and reused for geometrically-equivalent variations #6 and #11.

Note In previous releases of HFSS, the copy mesh operation was applied only when the sweep combinations were on adjacent lines in the Sweep Setup table (that is, with the above table sorted by values of **wg_z**). With HFSS v10, the copy mesh operation affects geometrically-equivalent combinations in any table sequence.

Note The mesh that is copied and reused is the one calculated for the first of the geometrically-equivalent configurations. In the example above, variation #11 will use the mesh generated for variation #1, not the mesh that is finally calculated for variation #6 (which may have required additional passes).

The **Copy geometrically equivalent mesh** option is not recommended for use when the frequency is varying, since meshing is frequency-dependent. You may wish to turn this option off when the first geometrically equivalent variation requires numerous passes after the initial mesh, but the other geometrically-equivalent variations require fewer additional passes, so that it is cheaper to start with the initial mesh each time.

Excluding a Variable from an Optimetrics Analysis

To exclude a variable from being optimized or included in a sensitivity or statistical analysis:

1. Do one of the following:
   - In the **Setup Optimization** dialog box, click the **Variables** tab.
   - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
   - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

   All of the independent variables that were selected for the optimization analysis are listed.
2. Clear the **Include** option for the variable you want to exclude from the analysis.

   The **Override** option is now selected. This indicates that, for this optimization analysis, the variable will not be included.

Note Alternatively, you can select the **Override** option first, and then clear the **Include** option for the variable you want to exclude.

Modifying the Value of a Fixed Variable for Statistical Analysis

If you are not including a variable in an optimization, sensitivity, or statistical analysis, Optimetrics uses that variable's current value during the analysis.

To override the current value of a fixed variable for an Optimetrics setup:

1. Do one of the following:
   - In the **Setup Optimization** dialog box, click the **Variables** tab.
   - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
   - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.
2. Click **Set Fixed Variables**.

   The **Setup Fixed Variables** dialog box appears. Under **Fixed Variables**, all of the current independent variable values are listed.
3. Click the **Value** text box of the variable with the value you want to override.
4. Type a new value in the **Value** text box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for this Optimetrics setup; the current variable value set for the nominal design will be ignored.

Note Alternatively, you can select the **Override** option first, and then type a new value in the **Value** text box.

5. Optionally, click a new unit system in the **Units** text box.

To revert to a default variable value, clear the **Override** option.

Setting a Linear Constraint

A linear constraint defines the linear relationship between variables. Setting linear constraints in Optimetrics is useful for establishing limitations involving linear combinations of variable values.

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click **Add**.

   The **Edit Linear Constraint** dialog box appears.
4. Click a **Coeff** text box and type a positive or negative coefficient value.
5. Click a condition, < (less than) or > (greater than), in the pull-down list.
6. Type the inequality value, which should be a constant value, in the text box to the right of the condition.
7. Click **OK**.

   You return to the **Linear Constraint** dialog box. The left-hand side of the constraint appears in the **LHS** (left-hand side) column. The condition is listed in the **Condition** column, and the inequality value is listed in the **RHS** (right-hand side) column.

Related Topics

Modifying a Linear Constraint

Deleting a Linear Constraint

Technical Notes: Linear Constraints

Modifying a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click the row listing the constraint you want to modify, and then click **Edit**.

   The **Edit Linear Constraint** dialog box appears.
4. Optionally, click a **Coeff** text box and type a new coefficient value.
5. Optionally, click a different condition, < (less than) or > (greater than), in the pull-down list.

6. Optionally, type a different inequality value in the text box to the right of the condition, and then click **OK**.

   You return to the **Linear Constraint** dialog box. The new coefficient value, the condition, and the inequality value appear in the **LHS** (left-hand side), **Condition**, and **RHS** (right-hand side) columns, respectively.

Deleting a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click the row listing the constraint you want to delete, and then click **Delete**.

   The constraint is deleted.

15

Post Processing and Generating Reports

When HFSS has completed a solution, you can display and analyze the results in the following ways:

- View solution data including the following: convergence information, computing resources that were used during the solution process, and matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution. For eigenmode solutions, you can view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode. Solution data can also be viewed while HFSS is generating a solution.
- View analysis results for Optimetrics solutions.
- Plot field overlays - representations of basic or derived field quantities - on surfaces or objects.
- Create 2D or 3D reports of S-parameters, basic and derived field quantities, and radiated field data.
- Plot the finite element mesh on surfaces or within 3D objects.
- Create animations of field quantities, the finite element mesh, and defined project variables.
- Scale an excitation's magnitude and modify its phase.

Note Except in the case of non-model boxes drawn in the global coordinate system (CS), non-model objects cannot be used for any fields post processing operation You can use non-model boxes drawn in the global CS for post processing operations, including integration and solution domaining.

Viewing Solution Data

While HFSS is generating a solution, or when it is complete, you can view the following information about the solution:

- Convergence information.
- Computing resources, or profile information, that were used during the solution process.
- Matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution.
- For eigenmode solutions, view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode.
- The state of solved solutions.

To access the **Solution Data** window, in which the information above can be accessed, do one of the following:

- On the **HFSS** menu, point to **Results**, and then click **Solution Data**  .
- Right-click **Results** in the project tree, and then click **Solution Data** on the shortcut menu.

Related Topics

Viewing Solution Data for an Optimetrics Design Variation

Viewing Convergence Data

To view an adaptive solution's convergence information, either during or after the solution process:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

   The **Solution Data** window appears. The **Convergence** tab is selected.

2. From the **Simulation** list, select the solution setup for which you want to view convergence data.

   By default, the most recently solved solution is selected.

3. Under the **Convergence** tab, depending on your design setup, you can review the following convergence data:

   - Number of adaptive passes completed and remaining.
   - The number of tetrahedra created at each adaptive pass.
   - Maximum magnitude of delta S between two passes.
   - Maximum delta Energy between two passes.
   - Magnitude margin between passes.
   - Phase margin (deg) between passes.
   - Maximum delta frequency between passes.

   If for the Solution Setup, you elected to **Use Matrix Convergence**, and selected specific table entries for the Magnitude and Phase, the **Convergence** tab also shows the following values with the Magnitude Margin and Phase Margin:

- Max Delta (Mag S)
- Max Delta (Phase S)

4. Select **Table** to display the convergence data in table format or **Plot** to plot the convergence data on a rectangular (X - Y) plot.

Related Topics

Viewing Solution Data for an Optimetrics Design Variation

Viewing the Number of Completed Passes

At any time during the solution process, you can view the number of adaptive passes (solve — error analysis — refine cycles) that have been completed and that have yet to be completed. When the solution is complete, you can view the number of adaptive passes that were performed. If the solution converged within the specified stopping criteria, fewer passes than requested may have been performed.

To view the number of passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The number of completed and remaining passes is listed in the **Number of Passes** area.

Viewing the Max Magnitude of Delta S Between Passes

For solutions with ports.

At any time during or after the solution process, you can view the maximum change in the magnitude of the S-parameters between two consecutive passes. This information is available after two or more passes are completed.

To view the maximum magnitude of delta S between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Max. Mag. Delta S** column lists the maximum magnitude of delta S from one pass to the next.

  The **Max. Mag. Delta S** area lists the target change in magnitude of delta S and the change in magnitude of delta S between the last two solved passes.

Note Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been de-embedded and renormalized.

Note You can renormalize mathematically, without having to re-solve, by accessing the postprocessing tab on the lumped port definition panel and de-selecting the **Deembed** selection box.

Related Topics

Setting the Maximum Delta S Per Pass

Technical Notes: Maximum Delta S

Viewing the Output Variable Convergence

At any time during or after the solution process, you can view the real and imaginary values of the output variable.

To view the output variable convergence:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Output Var (real)** column lists the real value of the output variable for each pass. The **Output Var (imag)** column lists the imaginary value of the output variable for each pass. If output variable convergence is not used, the columns are not used in the table.

Related Topics

Specifying Output Variable Convergence

Viewing the Delta Magnitude Energy

For designs with voltage sources, current sources, or incident waves. Not applicable to designs with ports.

At any time during or after the solution process, you can view the difference in the relative energy error from one adaptive pass to the next. The change in the magnitude of delta energy is available after two or more passes are completed.

To view the delta magnitude E between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Delta Mag. Energy** column lists the delta energy from one pass to the next.

  The **Delta Mag. Energy** area lists the target change in delta energy and the change in delta Energy between the last two solved passes.

Related Topics

Setting the Maximum Delta Energy Per Pass

Technical Notes: Maximum Delta Energy

Viewing the Magnitude Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the solution's proximity to the target delta magnitude, which was specified in the **Matrix Convergence** dialog box. The magnitude margin is available after two or more passes are completed.

To view the magnitude margin between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Magnitude Margin** column lists the magnitude margin from one pass to the next.

Related Topics

Setting Matrix Convergence Criteria

Technical Notes: Magnitude Margin

Viewing the Phase Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the solution's proximity to the target delta phase, which was specified in the **Matrix Convergence** dialog box. The phase margin is available after two or more passes are completed.

To view the phase margin between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Phase Margin** column lists the phase margin from one pass to the next.

Related Topics

Setting Matrix Convergence Criteria

Technical Notes: Phase Margin

Viewing the Max Delta (Mag S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the maximum difference of the S matrix magnitudes between two consecutive passes. The Max Delta (Mag S) is available after two or more passes are completed.

To view the between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Max Delta (Mag S)** column lists the Max Delta (Mag S) from one pass to the next.

Related Topics

Setting Matrix Convergence Criteria

Technical Notes: Max Delta (Mag S)

Viewing the Max Delta (Phase S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the maximum difference of the S Matrix phase between two consecutive passes. The Max Delta (Phase S) is available after two or more passes are completed.

To view the Max Delta (Phase S) between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Max Delta (Phase S)**column lists the Max Delta (Phase S) from one pass to the next.

Related Topics

Setting Matrix Convergence Criteria

Technical Notes: Max Delta (Phase S)

Viewing the Maximum Delta Frequency

For Eigenmode solutions.

At any time during the solution process, you can view the maximum delta frequency, the largest percent difference in the resonant frequencies from one adaptive pass to the next. It is a measure of the stability of the computed frequencies from pass to pass and is available after two or more passes are completed.

To view the maximum delta frequency between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

  The **Solution Data** window appears. The **Convergence** tab is selected.

  The **Max Delta Freq. %** column lists the maximum delta frequency from one pass to the next.

  The **Max Delta Freq. %** area lists the target maximum delta frequency and the maximum delta frequency between the last two solved passes.

Related Topics

Technical Notes: Maximum Delta Frequency

Plotting Convergence Data

To display convergence data vs. pass on a rectangular (x - y) plot:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

   The **Solution Data** window appears. The **Convergence** tab is selected.

2. In the lower-left corner of the window, select **Plot** as the view type.
3. Select the data you want to plot on the x-axis from the **X** pull-down list.
4. Select the data type you want to plot on the y-axis from the **Y** pull-down list.

   The x -y plot appears in the view window.

Viewing a Solution's Profile

At any time during or after the solution process, you can examine the computing resources - or profile data - that were used by HFSS during the analysis. The profile data is essentially a log of the tasks performed by HFSS during the solution. The log indicates the length of time each task took and how much RAM/disk memory was required.

- In the project tree, right-click the solution setup of interest, and then click **Profile** on the short-cut menu.

  The **Solution Data** dialog box appears. The **Profile** tab is selected.

  The displayed data depends on the type of problem and solution setup. In general, it includes the following information:

Task	Lists the software module that performed a task during the solution process, and the type of task that was performed. For example, for the task *mesh3d_adapt*, Mesh3d is the software module that adaptively refined the mesh. The profile ine for the matrix solver has the following format: "Solver *pds*", where: p, the precision type is: M (mixed) or D (double) d, the matrix data type is: R (real) or C (complex) s, the symmetry type is: S (symmetric), A (asymmetric), H (hermitian)
Real Time	The amount of real time (clock time) required to perform the task.
CPU Time	The amount of CPU time required to perform the task.
Memory	The peak amount of memory used by your machine while performing the task. This value includes all of the applications running at the time; it is not limited to HFSS. The memory is freed for other uses after each task is complete.
Information	General information about the solution, including the number of tetrahedra used in the mesh.

Related Topics

Viewing an Optimetrics Solution's Profile Data

Viewing Matrix Data

To view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution:

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

   The **Solution Data** dialog box appears. The **Matrix Data** tab is selected.

2. In the **Design Variation** text box, specify the design with the matrices you want to view.

   Optionally, choose a design variation solved during an Optimetrics analysis from the **Set Design Variation** dialog box.

3. In the **Simulation** pull-down list, click the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.

4. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Z_o** (characteristic impedance.) The available types depend on the solution type.

5. Select the format — **Magnitude/ Phase (deg)**, **Real/ Imaginary**, **dB/ Phase (deg)**, **Magnitude**, **Phase (deg)**, **Real**, **Imaginary**, or **dB** — in which to display the matrix information. The available formats depend on the matrix type being displayed.

6. Select the solved frequencies to display:

   - To display the matrix entries for all solved frequencies, select **All Freqs**. It is selected by default.
   - To show the matrix entries for one solved frequency, clear **All Frequencies** and then select the solved frequency for which you want to view matrix entries.

     For adaptive passes, only the solution frequency specified in the **Solution Setup** dialog box is available. For frequency sweeps, the entire frequency range is available.
   - To insert of delete one or more displayed frequencies, click **Edit Freqs**.

     This command is only available if the sweep type is Fast or Interpolating. If you choose to export the matrix data for the Fast or Interpolating sweep after modifying the frequencies in the **Edit Frequencies** dialog box, only those frequencies displayed under the **Matrix Data** tab will be exported.

   The data is displayed in the table. By default, Waveports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. To change the port order, change setting for Default Matrix sort order in the HFSS General options.

Related Topics

Selecting the Matrix Display Format

Viewing Solution Data for an Optimetrics Design Variation

Exporting Matrix Data

Renaming Matrix Data

Selecting the Matrix Display Format

You can display matrix data in the following formats. The available formats depend on the type of matrix being displayed.

Magnitude, Phase (deg)	Displays the magnitude and phase (in degrees) of the matrix type.
Real, Imaginary	Displays the real and imaginary parts of the matrix type.
dB, Phase (deg)	Displays the magnitude in decibels and phase in degrees of the matrix type.
Magnitude	Displays the magnitude of the matrix type.
Phase (deg)	Displays the phase in degrees of the matrix type.
Real	Displays the real parts of the matrix type.
Imaginary	Displays the imaginary parts of the matrix type.
dB	Displays the magnitude in decibels of the matrix type.

Exporting Matrix Data

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

   The **Solution Data** window appears. The **Matrix Data** tab is selected.
2. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Z_o** (characteristic impedance.)
3. Click **Export Matrix Data**.

   A file browser appears.
4. Type the name of the file you are exporting to in the **File name** text box.

5. Select one of the following file formats from the **Save as type** pull-down list:

Format	Type	Description
(spreadsheet) *.tab	data table	A text file in which the elements of the S-matrix are arranged in a series of columns that are tab-separated and include a first row of headings. The file may be imported into a spreadsheet or similar utility.
***.sNp**	Touchstone/Libra	A Touchstone S-parameter file in which the number of ports is indicated by *n*. For example, a Touchstone file with one port would have the file extension .s1p. When you export this format, you can specify: • the export reference impedance, • whether to renormalize the solution. If you want to export raw S-Parameter data for later use, you may choose to not renormalize the solution. If all ports and associated modes/terminals are normalized to the same impedance and you choose **Do Not Renormalize Solution** during export, the Touchstone file header will indicate the normalized impedance.
***.szg**	Ensemble/Planar EM or HFSS version 6 or later	A solution file read by Ensemble or Planar EM version 6 and later, Ansoft HFSS version 6 and later, and Maxwell Strata version 1.1.
***.nmf**	Neutral file format	Neutral file format defined by the MAFET Consortium.
***.m**	MATLAB	The Mathworks' MATLAB file format in which the elements of the S-, Y-, or Z-matrix are arranged in a series of rows.
***.cit**	Citifile format	

6. For Touchstone files, you see a **Combine Sweeps** option on the Export Network Data solution dialog. This lets you combine sweeps into a single output file if:
   - The sweeps must contain interpolated data, so internally it must come from interpolating or fast sweeps.
   - The files must not have overlaps or gaps in the frequencies. (They can meet at a single frequency. For example, you can combine sweeps from 8 to 10 GHz with sweeps from 10 to 12 GHz, but not sweeps from 9 to 11 GHz and 10 to 11 GHz, and not 8-10 GHz and 11-

13GHz.)

1. Select the **Combine Sweeps** button to display a **Combine Interpoloating Sweeps For Export** dialog with a list of of sweeps.
2. Select the sweeps to combine and click **Combine**.

   This closes the **Combine Interpolating Sweeps for Export** dialog.

3. Click **Save**.

   The data is exported to the file.

   - By default, waveports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. You can change this order to creation order and back without invalidating the solution on the HFSS Options dialog.
   - If you select Touchstone format, you are first presented with a dialog that asks you to specify the export reference impedance (an integer value) and whether to renormalize the solution.

Note If you modify the display of solved frequencies in an Interpolating or Fast sweep under the **Matrix Data** tab (by clicking **Edit Freqs** and then modifying the values in the **Edit Sweep** dialog box,) only those frequencies listed will be exported to the file.

Renaming Matrix Data

In the project tree, you can right-click on a port excitation to rename it. When you rename a port excitation, the associated data is reordered so that it can be presented in the same manner. The reordering is done to match the tree-sort order presented for the ports (renamed matrix data is reordered so that alphabetic values appear before numeric values).

Exports of the matrix data are ordered in the same manner. This reordering is conducted as part of post processing and does not force a re-solve.

Exporting Equivalent Circuit Data

You can export S-parameter data from a Driven Terminal solution to PSpice, HSPICE, or Maxwell Spice format. Importing the new data file to PSpice, HSPICE, or Maxwell Spice will enable you to include wave effects in the circuit simulations. You can also export a W-Element model for a port..

Note You must have a frequency sweep solution and five or more frequency points to successfully export an equivalent circuit data file. See the *Choosing Frequencies for Full-Wave SPICE* topic of the online help for suggestions about the frequency range of the sweep. Exporting to an equivalent circuit data file format is appropriate for solutions matrices that are not based on differential pairs.

Certain discrete sweeps for Full-Wave SPICE exports. It is allowed if the discrete data is evenly spaced, includes DC, and has at least 500 frequency points.

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

   The **Solution Data** window appears. The **Matrix Data** tab is selected.

2. Click **Equivalent Circuit Export**.

   The **Equivalent Circuit Export Options** dialog box appears.
3. Type the name or browse to the directory in which you want to store the data.
4. Click one of the following formats in the **Format** list:

   PSpice (*.lib)

   Star HSpice (*.sp)

   Spectre (*.spc)

   Maxwell Spice (*.spc)

   Your format selection affects the options available under **Full Wave Spice Export**.
5. If the **Full-Wave Spice Export** checkbox is enabled, you can select it. Checking the box enables the text field for the file name, and depending on the format selection, other options may be enabled.
   - For PSpice and Maxwell Spice formats only the file name field is enabled.
   - For Spectre and Star HSPice formats, the following fields are enabled:
     - **Desired Fitting Error (percent)**
     - **Minimum Order**
     - **Maximum Order**
6. Optionally, select **Use Command Ground** to combine references in the exported file.
7. Optionally, select **Passivity Error Check**. Selecting this enables the **Passivity Error Tolerance field**.

   The passivity check tests whether the S-parameter data from HFSS is passive or not. Passive devices can only dissipate or temporarily store energy, but never generate it. The mathematical definition of passivity is based upon the following condition:

   Q = I - conjugate(transpose(S)) * S must be a positive semidefinite matrix.

   where:
   - S is the S-parameter matrix
   - I is an identity matrix.

   A positive semidefinite matrix has only non-negative eigenvalues. The passivity test computes the eigenvalues of the matrix Q above at each frequency in the sweep. If any of the eigenvalues is negative, and larger (in magnitude) than the specified passivity tolerance, then a violation of passivity is reported to the user. The default value for passivity tolerance is 5 percent tolerance above 1.0 magnitude.
8. Optionally, select **Lumped Element Export (Low Bandwidth)** if you want to save the data as a low-frequency circuit model using simple lumped elements (resistors, capacitors, inductors, and dependent current sources). This option is not enabled for Spectre export.
9. **Partial Fraction Expansion for Matlab**

10. Click **OK**.

    The S-matrices are written to the data file that you specified in the equivalent circuit data format.

Exporting W-Element Data

It is possible to extract a W-element model for a port. This W-element model can be used in a SPICE model to represent a length of transmission line of the same cross section as the port. A W-element model can be extracted for a port only solution and for a full 3D solution.

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

   The **Solution Data** window appears. The **Matrix Data** tab is selected.
2. Click **Equivalent Circuit Export**.

   The **Equivalent Circuit Export Options** dialog box appears. At the bottom of the dialog you see the W-element model check box.
3. Click the W-element model check box to enable the W-element fields.
4. The W-element model name field has the project name by default. You can change this if desired.
5. In the **Model name** field, provide a model name.
6. Select the port from the **Port Name** pull down.
7. To export a W-element model for all ports, check the **Export for All Ports** check box.
8. Click **OK**.

   The W-element model is written to the data file that you specified.

Viewing Eigenmode Solution Data

To view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode:

1. In the project tree, right-click the solution setup of interest, and then click **Eigenmode Data** on the shortcut menu.

   The **Solution Data** window appears. The **Eigenmode Data** tab is selected.
2. In the **Simulation** pull-down list, select the solution setup and solved pass - adaptive or single frequency solution - for which you want to view data.

   The solved eigenmodes are listed in the table below.

   The **Frequency** column lists the real and imaginary parts of the frequency (or resonant frequency) for each solved eigenmode.

   For lossy Eigenmode solutions, a **Q** column appears, which lists the unloaded quality factor Q computed for each eigenmode.

Related Topics

Technical Notes: *Eigenmode Solutions*

Technical Notes: *Calculating the Resonant Frequency*

Technical Notes: *Calculating the Quality Factor*

Technical Notes: *Calculating the Free Space Wave Number*

Deleting Solution Data

You can use **Clean Up Solitions** to selectively make deletions, or use **Delete All Reports** to remove all solutions from the results.

To use **Clean Up Solutions**:

1. On the **HFSS** menu, point to **Results**, and then click **Clean Up Solutions**.

   The **Clean Up Solutions** dialog box appears.

2. Under **Data Deletion Options**, select whether you want to delete only fields data, only fields and mesh data, or all solution data. Deleting all solution data erases all mesh, matrix, and fields data for all adaptive passes and frequency sweeps for the selected **Variation Selection Options**.

3. Under **Variation Selection Options**, select which solution data you want to delete:

   - Select **All Variations Except Current Variation** to delete all solution data that do not correspond to the current project and design variable values for the current design.
   - Select **All Variations** to delete all solution data for the current design.
   - Select **Select Variations** to specify the variations you wish to delete. Click **Variations** to select the variations for deletion.

4. Click **Do Deletions**.

   The solution data you selected are deleted. Any post processing reports or field overlays you created that included data you deleted will be marked with an X in the project tree. They will be invalid until new solution data are generated.

To use **Delete All Reports**:

1. On the **HFSS** menu, point to **Results**, and then click **Delete All Reports**

   All items under the Results folder in the Project tree are removed.

Warning Solution data that have been deleted cannot be recovered!

Viewing Analysis Results for Optimetrics Solutions

To view data specific to an Optimetrics solution, in general, do the following:

- In the project tree, right-click the Optimetrics setup for which you want to view the results, and then click **View Analysis Result** on the shortcut menu.

  The **Post Analysis Display** dialog box appears.

See the help topics in this section for more details about viewing Optimetrics analysis results.

Viewing Solution Data for an Optimetrics Design Variation

To view the convergence information, computing resources used, or matrices computed for any design variation solved during an Optimetrics analysis, you must first select the design variation in the **Set Design Variation** dialog box. This dialog box is accessible from the **Solution Data** window.

1. On the **HFSS** menu, point to **Results**, and then click **Solution Data**  .

   The **Solution Data** window appears.
2. Click the browsing dots beside the **Design Variation** text box.

   The **Set Design Variation** dialog box appears.
3. Clear the **Use nominal design** option.
4. Click the design variation for which you want to view the solution data, and then click **OK**.

   The solution data is displayed in the table.

Related Topics

Viewing an Optimetrics Solution's Profile Data

Viewing an Optimetrics Solution's Profile Data

At any time during or after the Optimetrics solution process, you can see an overview of the computing resources - or profile data - that were used by HFSS as it solved each design variation. The profile data indicates the length of time each design variation took to solve.

1. In the project tree, right-click the Optimetrics solution setup of interest, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.
2. Click the **Profile** tab.
3. Select the Optimetrics setup with the results you want to view from the pull-down list at the top of the dialog box.
4. Optionally, to examine more detailed profile data for a specific design variation, do the following:
   a. Click a design variation in the table.
   b. Click **Solver Profile**.

      The **Solution Data** dialog box appears. The profile data for the selected design variation is displayed in the table.

Related Topics

Viewing a Solution's Profile

Viewing Results for Parametric Solution Quantities

1. In the project tree, right-click the parametric setup for which you want to view the results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.
2. Select the parametric setup with the results you want to view from the pull-down list at the top of the dialog box.
3. If it is not already selected, select **Table** as the view type.

   The results for the selected solution quantities are listed in table format for each solved design variation.
4. Optionally, select **Show complete output name**.

   The complete name of the solution for which the results are being displayed will be listed in the column headings.
5. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **3D Modeler** window is changed to represent the selected design variation.

Related Topics

Plotting Solution Quantity Results vs. a Swept Variable

Plotting Solution Quantity Results vs. a Swept Variable

To plot solution quantity results versus a swept variable's values on a rectangular (x - y) plot:

1. In the project tree, right-click the parametric setup for which you want to view the results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.
2. Select the parametric setup with the results you want to plot from the pull-down list at the top of the dialog box.
3. If it is not already selected, select **Plot** as the view type.
4. Select the variable with the swept values you want to plot on the x-axis from the **X** pull-down list.
5. Only one sweep variable at a time can be plotted against solution quantity results. Any other variables that were swept during the parametric analysis remain constant.

   Optionally, to modify the constant values of other swept variables, do the following:

   a. Click **Set Other Sweep Variables Value**.

      The **Setup Plot** dialog box appears. All of the other solved variable values are listed.

   b. Click the row with the variable value you want to use as the constant value in the plot, and then click **OK**.

6. Select the solution quantity results you want to plot on the y-axis from the **Y** pull-down list.

   The x -y plot appears in the view window.

Viewing Cost Results for an Optimization Analysis

To view cost values versus completed iterations in data table format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.
2. Select the optimization setup with the results you want to view from the pull-down list at the top of the dialog box.
3. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The cost value at each solved design variation is listed in table format.
4. Optionally, click a design variation in the table, and then click **Apply**.

   The design displayed in the **3D Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Cost Results for an Optimization Analysis

To view cost values versus completed iterations in rectangular (x-y) plot format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.
2. Select the optimization setup with the results you want to view from the pull-down list at the top of the dialog box.
3. Under the **Result** tab, select **Plot** as the view type.

   A plot of the cost value at each iteration appears.

Viewing Output Parameter Results for a Sensitivity Analysis

To view actual output parameter values versus design point in data table format:

1. In the project tree, right-click the sensitivity setup for which you want to view the cost results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.
2. Select the sensitivity setup with the results you want to view from the pull-down list at the top of the dialog box.
3. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The following values are listed in table format:

   - The regression value of the output parameter at the design point is listed in the **Func. Value** column.
   - The first derivative of the regression is listed in the **1st D** column.

- The second derivative of the regression is listed in the **2nd D** column.

4. Click **Apply**.

   The design displayed in the **3D Modeler** window is changed to represent the design point of the sensitivity analysis.

   Click **Revert** to return the design in the view window to the original variable values.

Plotting Output Parameter Results for a Sensitivity Analysis

To plot output parameter results versus sensitivity variable values on a rectangular (x - y) plot:

1. In the project tree, right-click the sensitivity setup for which you want to view the output parameter results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the sensitivity setup with the results you want to view from the pull-down list at the top of the dialog box.
3. Under the **Result** tab, select **Plot** as the view type.
4. Select the sensitivity variable with the sweep values you want to plot on the x-axis from the **X** pull-down list.
5. Select the output parameter results you want to plot on the y-axis from the **Y** pull-down list.

   The x -y plot appears in the **Post Analysis Display** dialog box.

   The plot displays actual output parameter results for each solved design variation. It also displays a parabola that best fits these results. The parabola is a more accurate representation of sensitivity around the design point than any individual solved design variation.

Viewing Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the pull-down list at the top of the dialog box.
3. If it is not already selected, select **Table** as the view type.

   The distribution results for the selected solution quantities are listed in table format for each solved design variation.

4. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **3D Modeler** window is changed to represent the selected design variation.

Plotting Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and then click **View Analysis Result** on the short-

cut menu.

The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the pull-down list at the top of the dialog box.
3. If it is not already selected, select **Plot** as the view type.
4. Type the number of bins you want to plot on the x-axis.
5. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** pull-down list.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

Scaling a Source's Magnitude and Phase

Scale the magnitude and set the phase of ports, voltage and current sources, Eigenmodes, and incident waves in the **Edit Sources** dialog box.

1. On the **HFSS** menu, point to **Fields**, and then click **Edit Sources**.

   The **Edit Sources** dialog box appears. It displays information for design sources in table format.

Source	Type	Solved Magnitude	Solved Phase	Scaling Factor	Offset Phase	Units
static name	static text	static value	static value	editable value	editable value	menu

N/A Terminated is checked (for Terminal Solutions only)

   In the case of terminal solutions, the table contains some additional columns with a scroll bar.

Terminated	Resistance	Unit	Reactance	Unit
Unchecked. (Default)	N/A unless Terminated is checked.			
Checked	editable value	menu	editable value	menu

   If incident waves are present, the dialog contains a row of radio buttons to select the type of incident wave.

   Note that in the **modal** case a unit stimulation means 1 Watt of incident power at the port; in the **terminal** case a unit stimulation means 1 volt of total voltage at the terminal. After converting the voltage stimulation to the equivalent power stimulation the antenna results agree perfectly. In particular, the "ratioed" antenna parameters such as gain, directivity, and efficiency agree between the modal and terminal projects, while absolute antenna quantities such as incident power, accepted power may initially appear different. This is a direct result of the difference in edit-sources stimulations in the two types of projects.

2. Select the source whose magnitude and phase you want to scale.

   If your solution type is driven terminal, a voltage source magnitude and phase may be set for the selected terminal.

3. In the **Scaling Factor** text box, enter the factor by which the value of the source is to be scaled. Design variables can be used as source scalings.

Note You may not enter a negative voltage. To obtain the equivalent of a negative magnitude, add or subtract 180 degrees from the phase value.

If you use a design variable as a scaling factor note that solutions are invalidated if the variable is changed.

4. In the **Offset Phase** text box, enter the new phase for the source.

   The phase of the source is changed by the value that you enter.

5. Optionally, if your solution type is driven terminal, you may specify a complex reference

impedance:

a. For the selected terminal, select **Terminated**.

   This disables the values to the left of the checkbox, and enables the Resistance and Reactance text boxes. Use the scroll bar to view them.

b. Enter the real part of the impedance in the **Resistance** text box and select the units. Ohms/square is the default.

c. Enter the imaginary part of the impedance in the **Reactance** text box and select the units. Ohms/square is the default.

6. If an incident wave is present, use the radio buttons at the button of the panel to select one of the following field types to use:

Scattered Fields	The differential field formed by subtracting the incident field from the total field.
Total Fields	The physically measurable field that exists with the model present and a non-zero incident field.
Incident Fields	The plane-wave field that would exist in the absence of the model.

7. Click **OK** to apply the changes and close the dialog, or click **Apply** to view the changes without closing the dialog.

   The magnitude and phase are assigned to the selected excitation.

Note When you scale an excitation, keep in mind that the original value of the excitation remains unchanged.

Guidelines for Scaling a Source's Magnitude and Phase

When specifying the factor by which the value of the source is scaled, keep the following guidelines in mind:

For ports

- The excitation's magnitude specifies time-averaged incident power in watts.
- If you are using a symmetry plane, remember to scale the input signal appropriately. For example, if you have one symmetry plane, use an input value of 0.5 watts to excite the full structure with 1 watt; if you have two symmetry planes, use an input value of 0.25 watts to excite the full structure with 1 watt, and so forth.
- Generally, use the default value of **1**. This specifies that the solution's E- and H-fields be scaled such that the excitation wave delivers 1 watt of power. To view the solution at some other power, enter a positive value.
- Only port-mode combinations with non-zero magnitudes will be used.

For voltage and current sources

- The source magnitude for voltage and current sources specifies peak value volts and peak value amperes, respectively.
- If you have defined multiple voltage and current sources, you can "remove" them by setting their magnitudes to **0**. This enables you to easily observe the effects that individual or specific groups of sources have on the problem.

For incident waves

- Source magnitude specifies peak value E-field in volts per meter.
- When you scale the incident E-field, the scattered E-field and the total E-field are scaled as well.
- This scaling factor affects all incident angles in the incident wave setup.

For Eigenmodes

- Source magnitude is unitless and represents a relative value.
- When you enter a scaling factor for an eigenmode the relative source magnitude is amplified by this value. Exactly one eigenmode must be excited by setting its scaling factor to a non-zero positive number.

When specifying the new phase for ports, generally use zero. This zero-phase solution results from excitations phased in such a way that, at $\omega t = 0$, peak values occur at the port faces.

Creating Animations

An animated plot is a series of frames that displays a field, mesh, or geometry at varying values. To create an animated plot, you specify the values of the plot that you want to include, just as an animator takes snapshots of individual drawings that make up a cartoon. Each value is a frame in the animation. You specify how many frames to include in the animation.

You can export the animation to animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.

Creating Phase Animations

To animate a plot with respect to the phase of the plotted field:

1. Create a field overlay plot to animate.
2. On the **HFSS** menu, point to **Fields**, and then click **Animate**  .
3. If you already created an animation, the **Select Animation** dialog box appears. Click **New**.

   The **Setup Animation** dialog box appears.
4. Type a name for the animation in the **Name** text box or accept the default name.
5. Optionally, type a description of the animation in the **Description** text box.
6. Under the **Swept Variable** tab, select **Phase** from the **Swept Variable** list.
7. Specify the phase values you want to include in the animation:
   a. Type the starting value of the phase in the **Start** text box.
   b. Type the stopping value of the phase in the **Stop** text box.
   c. Type the number of **Steps** to include in the animation.

      For example, if the **Start** value is **10**, the **Stop** value is **160**, and the number of steps is **10**, the animation will display the plot at 10 phase values between 10 and 160. The start value will be the first frame displayed, resulting in a total of 11 frames in the animation.
8. Click **OK**.

   The animation begins in the view window. The play panel appears in the upper-left corner of the desktop, enabling you to stop, restart, and control the speed and sequence of the frames.

Related Topics

Controlling the Animation's Display

Creating Frequency Animations

1. Create a field overlay plot to animate.

   In the **Create Field Plot** dialog box, make sure to select a sweep solution to plot from the **Solution** pull-down list.
2. On the **HFSS** menu, point to **Fields**, and then click **Animate**  .
3. If you already created an animation, the **Select Animation** dialog box appears. Click **New**.

   The **Setup Animation** dialog box appears.

4. Type a name for the animation in the **Name** text box or accept the default name.
5. Optionally, type a description of the animation in the **Description** text box.
6. Under the **Swept Variable** tab, select **Frequency** from the **Swept Variable** list.
7. Select the frequency values you want to include in the animation from the **Select values** list.

   Use the **Shift** key to select a series of values, and the **Ctrl** key to select values that are not in sequence.
8. Click **OK**:

   The animation begins in the view window. It will display one frame for each frequency value you selected.

   The play panel appears in the upper-left corner of the desktop, enabling you to stop, restart, and control the speed and sequence of the frames.

Related Topics

Controlling the Animation's Display

Creating Geometry Animations

Following is the general procedure for creating an animation that varies a part of the model geometry.

1. Right-click in the view window, point to **View**, and then click **Animate**.

   The **Select Drawing** dialog box appears.
2. Select the object you want to animate.

   The **Setup Animation** dialog box appears.
3. Type a name for the animation in the **Name** text box or accept the default name.
4. Optionally, type a description of the animation in the **Description** text box.
5. Under the **Swept Variable** tab, the **Swept Variable** list includes all of the defined geometric project and design variables. Select the geometry variable that you want to animate from the **Swept Variable** list.
6. Specify the values of the variable that you want to include in the animation:
   a. Type the starting value of the variable in the **Start** text box.
   b. Type the stopping value of the variable in the **Stop** text box.
   c. Type the number of **Steps** to include in the animation.

      For example, if the **Start** value is **0.15in**, the **Stop** value is **0.45in**, and the number of steps is **15**, the animation will display the geometry at 15 values between 0.15 inches and 0.45 inches. The animation will also include the start value, which will be the first frame displayed, resulting in a total of 16 frames in the animation.
7. Click **OK**.

   The animation begins in the view window. It will display one frame for each variable value.

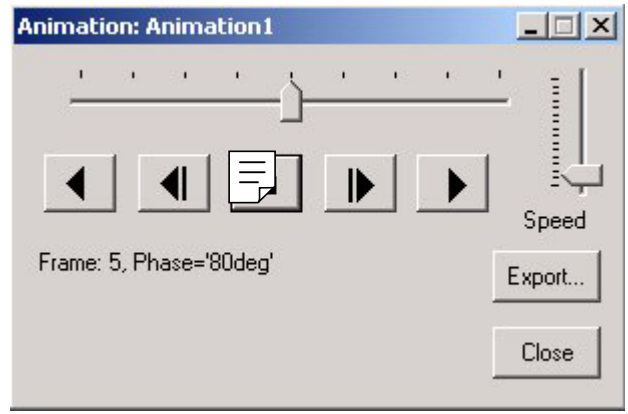
   The play panel appears in the upper-left corner of the desktop, enabling you to stop, restart, and control the speed and sequence of the frames.

Related Topics

Controlling the Animation's Display

Controlling the Animation's Display

When an animation is displayed in the view window, the **Animation** window, also called the *play panel*, appears in the upper-left corner of the desktop. It has buttons that enable you to control the speed and sequence of the frames, start and stop the animation and export the animation. Click an area of the window below to learn its function.



Animation slider Each dot on the slider represents a frame in the animation. Drag the slider to the right to display the next frame in the animated plot. Drag the slider to the left to display the previous frame in the animation.


Plays the plot's animation sequence backwards.


Steps backward through the animated plot one frame at a time.


Stops the animation.


Steps forward through the animated plot one frame at a time.


Plays the plot's animation sequence forwards.

Drag the **Speed** slider to the top to increase the speed of the animation. Drag the **Speed** slider to the bottom to decrease its speed.

Frame information The current frame and phase at which the plot is being displayed is listed below the control buttons.

Enables you to export the animation to an animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.

Closes the animation window.

Exporting Animations

1. Create the animation you want to export.
2. In the play panel, click **Export**.

   The **Save As** dialog box appears.
3. Follow the procedure for saving a new file. Select **Animated GIF File (.gif)** or **AVI File (.avi)** as the file type.

   The **Animation Options** dialog box appears.
4. To replace colors in the file with 256 shades of gray, select **Grayscale**.

   Grayscale animations tend to use less memory than full color animations.
5. Specify the **Compression factor**.
6. Specify one of the following **Compression types**:

   INTEL Indeo

   Cinepak

   Microsoft Video 1

   None

   The animation is exported to the file format you specified.

Plotting Field Overlays

Field overlays are representations of basic or derived field quantities on surfaces or objects.

To plot a basic field quantity:

1. Select a point, line, surface, or object to create the plot on or within.

   If it does not exist, create it.
2. On the **HFSS** menu, point to **Fields**, and then point to **Plot Fields**.
3. On the **Plot Fields** menu, click the field quantity you want to plot.

   If you select a scalar field quantity, a scalar surface or volume plot will be created. If you select a vector field quantity, a vector surface or volume plot will be created. If the quantity you want to plot is not listed, see *Calculate a Derived Field Quantity*.

   The **Create Field Plot** dialog box appears.
4. To specify a name for the plot other than the default, select **Specify Name**, and then type a new name in the **Name** text box.
5. Select the solution to plot from the **Solution** pull-down list.
6. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the **Plot Folder** pull-down list. Plot folders are listed under **Field Overlays** in the project tree.
7. Under **Intrinsic Variables**, select the frequency and phase angle at which the field quantity is evaluated.
8. Select the field quantity to plot from the **Quantity** list.
9. Select the volume (region) in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume with the selected object.
10. Click **Done**.

    The field quantity is plotted on the surfaces or within the objects you selected. The plot uses the attributes specified in the **Plot Attributes** dialog box.

    The new plot appears in the view window. It is listed in the specified plot folder in the project tree.

Related Topics

Plotting Derived Field Quantities

Using the Fields Calculator

Technical Notes: Field Overlays

Technical Notes: Field Quantities

Technical Notes: Specifying the Phase Angle

Plotting Derived Field Quantities

Derived field quantities are field quantity representations that have been deduced from the original field solution using the Fields Calculator.

1. Select a point, line, surface, or object to create the plot on or within.

   If it does not exist, create it.
2. On the **HFSS** menu, point to **Fields**, and then click **Plot Calculated Expression**.
3. Select the derived quantity you want to plot, and then click **OK**.

   The **Create Field Plot** dialog box appears.
4. To specify a name for the plot other than the default, select **Specify Name**, and then type a new name in the **Name** text box.
5. Select the solution to plot from the **Solution** pull-down list.
6. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the **Plot Folder** pull-down list. Plot folders are listed under **Field Overlays** in the project tree.
7. Under **Intrinsic Variables**, select the frequency and phase angle at which the field quantity is evaluated.
8. Select the derived field quantity to plot from the **Quantity** list.
9. Select the volume, or region, in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume and the selected object.
10. Click **Done**.

    The derived field quantity you created in the Fields Calculator is plotted on the surfaces or objects you selected. The new plot is listed in the project tree under **Field Overlays**.

Related Topics

Using the Fields Calculator

Technical Notes: Field Quantities

Technical Notes: Specifying the Phase Angle

Creating Scalar Field Plots

A scalar plot uses shaded colors or contoured lines to illustrate the magnitude of field quantities on surfaces or volumes.

1. Do one of the following:
   a. To create a scalar surface plot, select the faces on which you want to plot the fields.
   b. To create a scalar volume plot, select the objects within which you want to plot the fields.
2. On the **HFSS** menu, point to **Fields**, and then point to **Plot Fields**.
3. On the **Plot Fields** menu, click the scalar field quantity you want to plot.

   The **Create Field Plot** dialog box appears.
4. Follow the procedure for plotting field overlays.

The plot uses the attributes specified in the **Plot Attributes** dialog box.

The new plot will be listed in the specified plot folder in the project tree.

Related Topics

Modifying Field Plot Attributes

Modifying SAR Settings

HFSS uses default specific absorption rate (SAR) settings when creating a local SAR or average SAR field overlay plot. To change the default settings:

1. On the **HFSS** menu, point to **Fields**, and then click **SAR Setting**.

   The **Specific Absorption Rate Setting** dialog box appears.
2. In the **Material Density** text box, enter the mass density of the dielectric material in g/cm^3.
3. In the **Mass of Tissue** text box, enter the mass of the material that surrounds each mesh point. This can be a value between 1 and 10.
4. Click **OK**.

Hint The SAR settings will apply to the entire model. To plot the SAR inside a volume with multiple dielectric objects, each with their own mass density, set the mass density, and then plot the SAR only in the object of interest.

Related Topics

Technical Notes: Calculating the SAR

Creating Vector Field Plots

A vector plot uses arrows to illustrate the magnitudes of the x-, y-, and z-components of field quantities. Vector plots can be created on surfaces or volumes.

1. Do one of the following:
   a. To create a vector surface plot, select the faces on which you want to plot the fields.
   b. To create a vector volume plot, select the objects within which you want to plot the fields.
2. On the **HFSS** menu, point to **Fields**, and then point to **Plot Fields**.
3. On the **Plot Fields** menu, click the vector field quantity you want to plot.
4. Follow the procedure for plotting field overlays.

Modifying Field Plots

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot**  .

   The **Select Field Plot(s)** dialog box appears.
2. Select the plot you want to modify in the **Select** column, and then click **OK**.
3. Optionally, click a different solution to plot in the **Solution** pull-down list.
4. Optionally, specify a different **Plot Folder** in which to store the plot.
5. Under **Intrinsic Variables**, specify the frequency and phase angle at which the field quantity

will be evaluated.

6. Optionally, select a different field quantity to plot from the **Quantity** list.
   - To choose a calculated expression, select **Calculator** from the **Category** pull-down list.
   - To choose a default field quantity, select **Standard** from the **Category** pull-down list.
7. Select the volume, or region, in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume and the selected object.
8. Click **Done**.

   The field quantity is plotted on the surfaces or within the objects you selected. The new plot is listed in the specified plot folder in the project tree.

   The plot uses the attributes specified in the **Plot Attributes** dialog box.

Related Topics

Technical Notes: Specifying the Phase Angle

Setting Field Plot Attributes

After creating a vector or scalar field overlay on a surface or volume, you can modify its appearance by changing the settings in the **Plot Attributes** dialog box. You will modify the settings for a plot folder and all plots in that folder will use the same attributes.

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Under the following tabs in the dialog box, you can control the following plot attributes:

Color map	The number of colors used and how they are displayed.
Scale	The scale of field quantities.
Marker/Arrow	• The appearance of points (for scalar point plots). • The appearance of arrows (for vector plots).
Plots	• The spacing of arrows (for vector plots). • To display or hide the mesh on the plot's surface or volume. • The type of isovalue display (for scalar plots.) • The transparency based on solution value.

4. Under each tab, click **Save as default** if you want the tab's settings to apply to field overlay plots created after this point.
5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

6. Click **Close** to dismiss the dialog box.

Modifying Field Plot Colors

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Color map** tab.
4. Select one of the following color types:

Uniform	Field quantities are plotted in a single color. Choose the plot color from the **Color** palette.
Ramp	Field quantities are plotted in shades of a single color. Choose the plot color from the **Color** palette. The shade of the color corresponds to its field value.
Spectrum	Field quantities are plotted in multiple colors. Choose a color spectrum from the pull-down list. Each field value is assigned a color from the selected spectrum.

5. Enter the **Number of colors** to use in the plot.
6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
7. Click **Close** to dismiss the dialog box.

Related Topics

Setting the Color Key Visibility

Moving the Color Key

Setting the Color Key Visibility

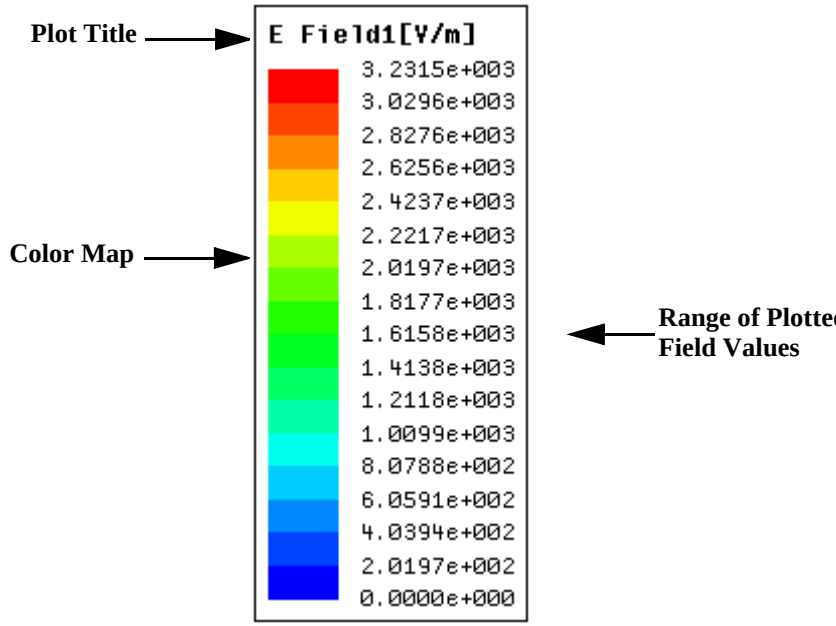
The color key (shown below) displays the range of plotted field values for a field overlay plot. It displays the colors that correspond to the range of field values on the plot.

1. On the **View** menu, click **Active View Visibility**  .

   The **Active View Visibility** dialog box appears.
2. Click the **Color Keys** tab.
3. In the **Visibility** column, select the field overlay or mesh plots in which you want to display the color key. Clear the plots in which you want to hide the color key from view.
4. Click **Done** to dismiss the dialog box.

Alternatively, to hide the color key, right-click on the color key in the view window, and then click **Hide** from the shortcut menu.

Only the color keys in the selected plots will be visible.



Related Topics

Modifying Field Plot Colors

Moving the Color Key

Click on the active field overlay plot's color key and drag it to a new location.

Modifying the Field Plot Scale

To change how field quantities are scaled on the field overlay plot:

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Scale** tab.

4. Select one of the following scale options:

Auto	The full range of field values will be plotted on the selected surface or volume.
Use Limits	Only the field values between the minimum and maximum values will be plotted. Field values below or above these values will be plotted in the colors assigned to the minimum or maximum limits, respectively.

5. If you selected **Use Limits**, enter the lowest field value to be plotted in the **Min.** text box and the highest field value to be plotted in the **Max.** text box.
6. Select one of the following options:

Linear	Field values will be plotted on a linear scale.
Log	Field values will be plotted on a logarithmic scale.

7. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
8. Click **Close** to dismiss the window.

Modifying Vector Field Plot Arrows

To change the appearance of a vector field plot's arrows:

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Marker/Arrow** tab.
4. Under **Arrow Options**, select one of the following arrow types:

Line	The arrows are displayed as 2D/flat.
Cylinder	The arrow tails are displayed as cylinders. The arrowheads are displayed as 3D/round.
Umbrella	The arrow tails are displayed as 1D lines. The arrowheads are displayed as 3D/round.

5. Use the **Size** slider to increase (move to the right) or decrease (move to the left) the length and dimensions of the arrows. The arrows are resized relative to the size of the model geometry.
6. Select **Map Size** to scale the size of the arrows to the magnitude of the field quantity being plotted.
7. Select **Arrow tail** to include tails on all arrows.
8. Click the **Plots** tab.

9. HFSS plots arrows on a grid that is superimposed on the surface or object you selected for the plot. Under **Vector plot**, use the **Spacing** slider to increase (move to the right) or decrease (move to the left) the distance between arrows (grid points.)
   - Select **Uniform** if you want the arrows to be spaced equally.
10. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.
11. Click **Close** to dismiss the window.

Setting the Mesh Visibility on Field Plots

To display or hide the mesh on field plots, or change the mesh's color:

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Plots** tab.
4. Select **Add Grid** to display the mesh.
5. Optionally, select a color for the mesh from the **Color** palette.
6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
7. Click **Close** to dismiss the window.

Modifying Scalar Field Plot Isovalues

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Plots** tab.
4. If the plot is a scalar surface plot, do the following:
   a. Select one of the following isosurface display types in the **IsoValType** pull-down list:

Line	Lines are drawn along the isovalues.
Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

b. Optionally, if you selected **Fringe** or **Tone**, select **Outline** to add a border line between isovalues.

5. If the plot is a scalar volume plot, do the following:
   a. Select one of the following display types:

IsoValSurface	Color is drawn on the isovalues.
Cloud	Field values are represented by points that illustrate the spatial distribution of the solution. The higher the solution value, the greater the cloud density.

   b. Optionally, if you select **Cloud**, use the **Cloud density** slider to increase or decrease the number of points that represent the density on the volume.
   c. Optionally, if you select **Cloud**, enter a point size for the clouds in the **Point size** text box.
6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
7. Click **Close** to dismiss the window.

Mapping Scalar Field Plot Transparency to Field Values

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Plots** tab.
4. Use the **Map transp.** slider to increase (move to the right) or decrease (move to the left) the transparency of the plot.
   - If you select **Map transp.**, the transparency of field values increases as the solution values decrease.
5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
6. Click **Close** to dismiss the window.

Modifying Markers on Point Plots

For scalar point plots, a marker is used to represent a field quantity at a selected point. (For vector point plots, arrows are used.) Modify the shape and size of markers in the plot attributes window.

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

All plots in the selected folder will be modified.

A dialog box with attribute settings for the selected folder appears.

3. Click the **Marker/Arrow** tab in the plot attributes window.
4. Under **Marker options**, select one of the marker types to represent the field quantity at the point:
   - **Sphere**
   - **Box**
   - **Tetrahedron**
   - **Octahedron**
5. Use the **Size** slider to increase (move to the right) or decrease (move to the left) the size of the marker.
6. Select **Map size** to scale the size of the marker to the magnitude of the quantity being plotted.
7. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
8. Click **Close** to dismiss the window.

Related Topics

Drawing a Point

Modifying Line Plots

Field quantities can be plotted directly on a line object. Scalar quantities are plotted as 3D color-shaded lines. Vector quantities are plotted as arrows that are based on the line.

To modify the appearance of line plots:

1. On the **HFSS** menu, point to **Fields**, and then click **Modify Plot Attributes**  .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Plots** tab.
4. Select one of the following isosurface display types in the **IsoValType** pull-down list:

Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

5. Select one of the following styles for the line object in the **Line style** pull-down list:

Cylinder	The line object is shaped like a cylinder.
Solid	The line object is a 3D solid.
Dash-Dash	The line object is represented by dashed black line segments.
Dot-Dot	The line object is represented by a series of dots.
Dash-Dot	The line object is represented by a a series of alternating dashed black line segments and dots.

6. Use the **Line width** slider to increase (move to the right) or decrease (move to the left) the thickness of the line.
7. By default, a polyline object is divided into 100 equally spaced points for post processing. To modify the number of points on the line, type a new value in the **Number of points** text box.
8. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
9. Click **Close** to dismiss the window.

Related Topics

Drawing a Polyline

Setting a Plot's Visibility

To display or hide a field overlay or mesh plot from view in the **3D Modeler** window:

1. On the **View** menu, click **Active View Visibility**  . Alternatively, you can select the **Active View Visibility** icon from the toolbar.

   The **Active View Visibility** dialog box appears.
2. Click the **FieldsReporter** tab.
3. In the **Visibility** column, select the field overlay or mesh plots you want to display. Clear the plots you want to hide from view.

   Only the selected plots will be visible.

Saving a Field Overlay Plot

Field overlay and mesh plots are saved in the project file (.adsn); however, you can save a plot to HFSS Field Plot File format (.dsp) and then open it in HFSS.

To save field overlay or mesh plot data to a .dsp file:

1. In the project tree, click the plot you want to export.
2. On the **HFSS** menu, point to **Fields**, and then click **Save as**  .

   The **Select Field Plot(s)** dialog box appears.
3. Select the plots you want to export by checking the **Select** box, and then click **OK**.

The file browser appears.

4. Make sure that **Field Plot Files (.dsp)** is the selected file type.
5. Specify the name of the .dsp file and the location in which to save it.
6. Click **Save**.

   The plot is exported to the specified .dsp file.

The file you created can be opened in HFSS version 9 and later. Simply click **HFSS>Fields>Open**.

Opening a Field Overlay Plot

To open a field overlay or mesh plot that you have saved to HFSS Field Plot File format (.dsp) in HFSS version 9 and later:

1. On the **HFSS** menu, point to **Fields**, and then click **Open**  .

   The file browser appears.
2. Make sure that **Field Plot Files (.dsp)** is the selected file type.
3. Browse to the location of the .dsp file you want to open, and then click the file name.
4. Click **Open**.

   The plot appears in the view window. It is listed under **Field Overlays** in the project tree.

Deleting a Field Overlay Plot

1. On the **HFSS** menu, point to **Fields**, and then click **Delete Plot**  .

   The **Delete Plots** dialog box appears.
2. Select the plots you want to delete by checking the **Delete** check box.
3. Click **OK**.

   The selected plots are deleted.

Alternatively, click the plot in the project tree that you want to delete, and then press **Delete**  .

Setting Field Plot Defaults

Each new field plot uses the default plot settings specified in the **Set Plot Defaults** dialog box.

To modify the default plot settings:

1. If a plot folder has not been created, click **Field Overlays** in the project tree.
2. On the **HFSS** menu, point to **Fields**, and then click **Set Plot Defaults**  .

   The **Set Plot Defaults** dialog box appears.
3. Select the solution to plot from the **Solution** pull-down list.
4. Select the plot folder in which new plots will be stored from the **Quantity type** pull-down list.

Choose one of the following options:

New Folder	Each new plot will be stored in a separate folder in the project tree.
Automatic	Each new plot will be stored in a folder determined by HFSS as the most appropriate based on the plotted field quantity. For example, all surface magnitude E plots will be stored in the same folder.
An existing folder	Select the existing folder in which you want to store new plots.

Note Plots stored in the same folder will use the same color key. The **Auto** scale setting will be based on the maximum field solution value present in a plot.

5. Under **Intrinsic Variables**, specify the frequency and phase angle at which the field quantity is evaluated.
6. Click **OK**.

Related Topics

Technical Notes: Specifying the Phase Angle

Using the Fields Calculator

The Fields calculator enables you to perform computations using basic field quantities. The calculator will compute derived quantities from the general electric field solution; write field quantities to files, locate maximum and minimum field values, and perform other operations on the field solution.

The calculator does not actually perform the computations until a value is needed or is forced for a result. This makes it more efficient, saving computing resources and time; you can do all the calculations without regard to data storage of all the calculated points of the field. It is generally easier to do all the calculations first, then plot the results.

Related Topics

Opening the Fields Calculator

Context Area

Calculator Stack

Registers

The Stack Commands

Input Commands

General Commands

Scalar Commands

Vector Commands

Output Commands

Calculating Derived Output Quantities

Named Expression Library

Opening the Fields Calculator

To open the Fields Calculator, do one of the following:

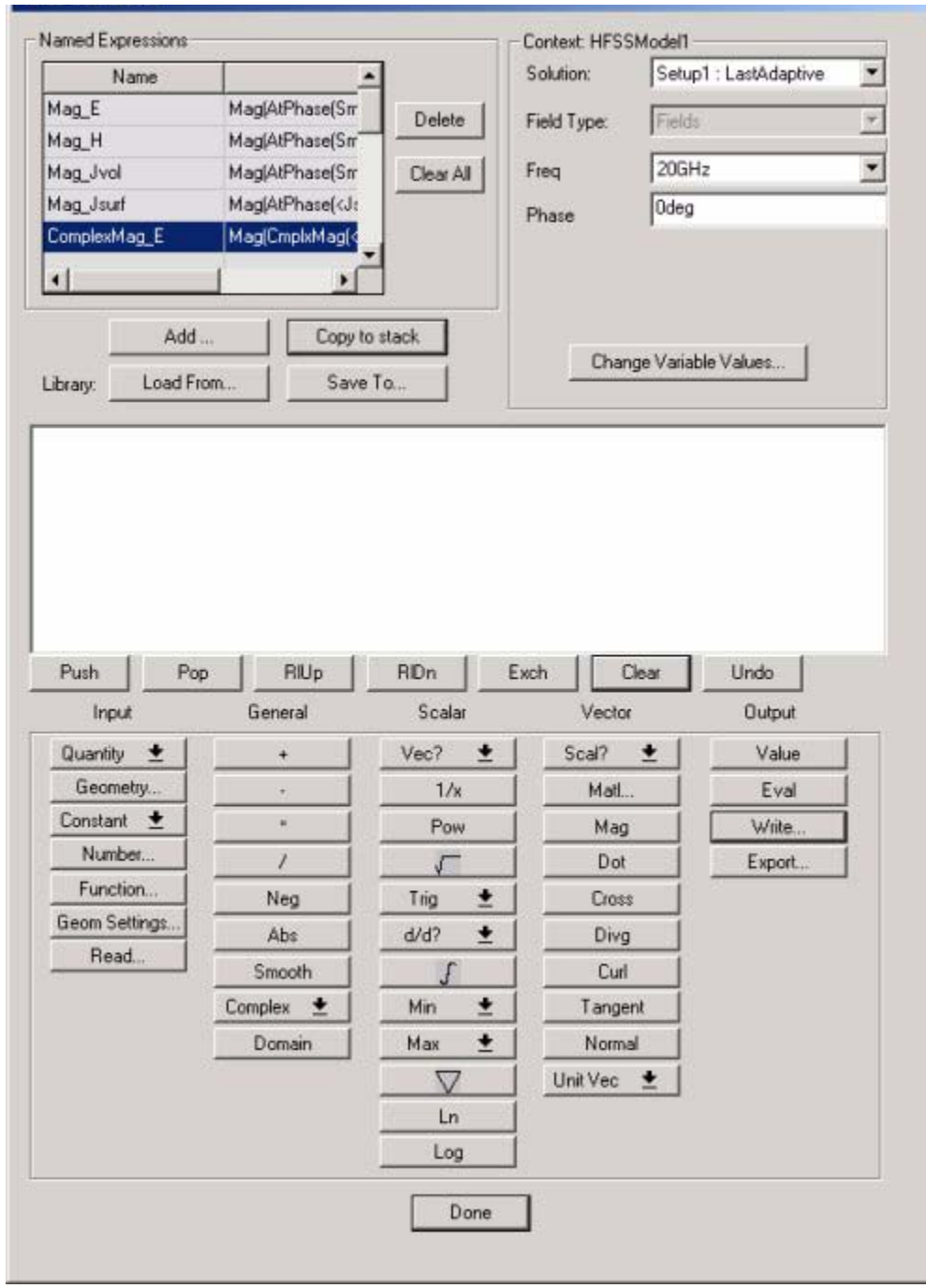
- On the **HFSS** menu, point to **Fields**, and then click **Calculator** 

or

- Right-click **Field Overlays** in the project tree, and then click **Calculator** on the shortcut menu.

  The Fields Calculator window appears.

To view information on a command or screen area, click over the button or screen area on the illustration below.


Named Expressions
Name
Mag_E
Mag_H
Mag_Jvol
Mag_Jsurf
ComplexMag_E
Delete
Clear All
Add ...
Copy to stack
Library:
Load From...
Save To...
Context: HFSSModel1
Solution:
Setup1 : LastAdaptive
Field Type:
Fields
Freq
20GHz
Phase
0deg
Change Variable Values...
Push
Pop
RlUp
RlDn
Exch
Clear
Undo
Input
General
Scalar
Vector
Output
Quantity
Geometry...
Constant
Number...
Function...
Geom Settings...
Read...
+
-
*
/
Neg
Abs
Smooth
Complex
Domain
Vec?
1/x
Pow
Trig
d/d?
Min
Max
Ln
Log
Scal?
Matl...
Mag
Dot
Cross
Divg
Curl
Tangent
Normal
Unit Vec
Value
Eval
Write...
Export...
Done

Context Area

The panel at the upper right of the window identifies the context to be used for the calculations. The top line identifies the design. Depending on the design, text entry boxes allow you to select a **Solution**, **Field Type**, **Freq**,**Phase**, .**IWavePh**i and **IWaveTheta**. The **Field Type** is available only when HFSS Transient Fields are used and the **IWavePhi** and **IWaveTheta** are available only for incident wave projects in which the wave is defined with spherical coordinates.)

The Calculator Stack

The calculator is made up of a stack of registers. Registers are displayed in the register display area at the center of the calculator window. Each register can hold:

- Field quantities such as the H-field or E-field.
- Functional or constant scalars and vectors.
- Geometries — points, lines, surfaces, or volumes — on which a field quantity is to be evaluated.

To perform a computation on the field solution, you must first load a basic field quantity into a register on the stack. Once a quantity is loaded into a register, it can be:

- Manipulated using mathematical operations such as curls, gradients, cross products, divergences, and dot products.
- Integrated over lines, surfaces, or subvolumes of the solution region — either predefined surfaces, volumes, and lists, or lines, surfaces, and volumes that were defined using the **Draw** commands.
- Plotted on a point, line, surface, or volume.
- Exported to a file, allowing you to superimpose saved solutions.

Registers

Calculator registers hold field quantities, numbers, vectors, and geometries. No registers are created until you load something into the calculator; therefore, this part of the window is initially blank. As items are loaded into the calculator, it creates new registers to hold them.

Each register is labeled with its contents as follows:

Vec	Vector quantities, which have both direction and magnitude at each point in space. The x-, y-, and z-components of these quantities are stored in the register.
Scl	Scalar quantities, which have a magnitude only.
Cvc	Complex vector quantities.
Csc	Complex scalar quantities.
Pnt	Points.
Lin	Lines.
Srf	Surfaces.
Vol	Volumes.

SclLin	Scalar value on a line.
VecLine	Vector value on a line.
SclSrf	Scalar value on a surface.
VecSrf	Vector value on a surface.

When examining calculator registers, keep the following in mind:

- To move or delete calculator registers, use the stack commands.
- To save a register to a disk file, use the **Write** command.

Enlarging the Register Display Area

If there are too many registers to fit into the display area, do one of the following:

- Use the scroll bars to view the hidden registers.
- Enlarge the calculator window using the window's borders or its maximize button.

Units of Measure

Unless you are prompted specifically for the unit of measure, all measurements should be assumed to be in SI base units, not model units.

Stack Commands

Use these commands to manipulate the registers in the calculator stack.

Push

Reloads the quantity in the top register onto the top of the stack, creating a new register. The contents of the top two registers are identical.

Pop

Deletes the top register from the stack.

RlUp

Rolls the top register to the bottom of the stack, moving the other registers up the stack.

RlDn

Rolls the bottom register to the top of the stack, moving the other registers down the stack.

Exch

Exchanges the top two registers in the stack.

Clear

Clears the contents of the stack.

Undo

Use this command to undo the effect of the last operation you performed on the contents of the top register. Successive **Undo** commands act on any previous operations.

Note You cannot undo a simple operation such as loading a field quantity, constant, function, or geometry into the calculator. Instead, use the **Pop** or **Clear** commands to delete these items from the calculator stack.

Input Commands

Use the following commands to load data onto the top of the calculator stack:

Quantity	Basic field quantities, such as E and H, and simple derived quantities such as volume current.
Geometry	Geometries such as planes, points, polylines, and volumes
Constant	Predefined constants such as π, ε_0, and conversion factors between various units of measurement.
Number	Vector and scalar constants, including complex numbers.
Function	User-defined or intrinsic variables
Geom Settings	Properties of polylines, surfaces, or volumes used in the Fields Calculator
Read	Previously-saved calculator registers containing field quantities.

These quantities can be manipulated using the **Stack** commands, **General** commands, **Scalar** commands, and **Vector** commands. The results of these calculations can then be examined using the **Output** commands.

Quantity Command

The **Input** command loads a field quantity into the top register of the calculator. Phasors in the calculator are peak phasors. The **Poynting** command in the calculator therefore implements the Poynting vector for peak phasors. Calculations which compute either average or instantaneous time domain quantities must adhere to the peak phasor conventions.

The available quantities are:

E	The electric field, E
H	The magnetic field, H
Jvol	The volume current density, J_{vol}
Jsurf	The surface current density, J_{surf}
Poynting	The Poynting vector, defined as 0.5E x H*
LocalSAR	The local Specific Absorption Rate
AverageSAR	The average Specific Absorption Rate

Certification SAR IEEE standard Specific Absorption Rate certification number

SurfaceLossDensity This contains the surface impedance (if any) loss at every node in every triangle. This is calculated as:

$$p_s = Re(S \cdot n)$$

where p_s is the surface impedance loss density, S is the Poynting vector on the boundary, and n is the out unit normal of the boundary.

To export a REG file containing the surface loss density, place the SurfaceLossDensity in the top register and use the **Write...** command.

The Reg file can be used to for coupled solutions with ePhysics. To setup HFSS - Transient Thermal coupling, use the ePhysics **Solve Setup** window for static thermal solutions by choosing the **Import** button next to HFSS Loss.

VolumeLossDensity The volume loss density p is calculated as:

$$p_v = \frac{1}{2}Re(E \cdot \tilde{J} + j\omega B \cdot \tilde{H}) = \frac{1}{2}Re(E \cdot \tilde{J} - curlE \cdot \tilde{H})$$

where E is the electric field, $\tilde{J}$ is the conjugate of the volumentric current density, B is the magnetic flux density, and H is the conjugate of the magnetic field.

To export a Reg file containing the volume loss density, place the VolumeLossDensity into the top register, and use the **Write...**.command.

The Reg file can be used to for coupled solutions with ePhysics. To setup HFSS - Transient Thermal coupling, use the ePhysics **Solve Setup** window for static thermal solutions by choosing the **Import** button next to HFSS Loss.

Geometry Command

The **Geometry** command loads a geometry into the top register of the calculator. Do this to:

- Find the value of derived field quantities on any point, line, surface, or volume.
- Plot quantities directly from the calculator.
- Display a previously defined isosurface, maximum or minimum field point using the **Draw** command.

The following types of geometries are available:

Point

Line

Surface

Volume

Coord

To load a geometry into the calculator:

1. In the Fields Calculator, click **Geometry**.
   The **Geometry** dialog box appears.
2. Select a geometry type.
   A list of all available geometries appears.
3. Click the geometry.
4. Click **OK** to load the geometry.

Note Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1.

If you explicitly subtract Box2 from Box1, any calculation on the surface (faces) of Box1 will use the 6 exterior faces and the 6 interior faces. Any calculation on the volume of Box1 will use the difference in volume between Box1 and Box2.

If you do not explicitly subtract Box2 from Box1, the inner box is only implicitly subtracted. Any calculation on the surface of Box1 in this case will use only the 6 exterior faces of Box1. Any calculation on the volume of Box1 will use the entire volume without subtracting the volume of Box2.

Constant Command

The **Constant** command loads one of these predefined constants into the top register of the calculator:

Pi	π
Epsi0	The permittivity of free space, $\varepsilon_0 = 8.85418782 \times 10^{-12}$ C^2/Nm^2
Mu0	The permeability of free space, $\mu_0 = 4\pi \times 10^{-7}$ Wb/Am
c	The speed of light in vacuum, $c = 2.99792458 \times 10^8$ m/s

Number Command

The **Number** command enters one of the following into the top register of the calculator:

Scalar

A scalar constant. To enter a constant scalar number:

1. Click **Number**.

   The **Input Number** dialog box appears.
2. Select **Scalar**.
3. Type the scalar value in the **Value** text box.
4. Click **OK** to load the number into the top register.

Vector

A vector constant.

To enter a constant vector:

1. Click **Number**.

   The **Input Number** dialog box appears.
2. Select **Vector**.
3. Enter the x-, y-, and z-components of the vector.
4. Click **OK** to load the vector into the top register.

Complex

A complex constant. Complex constants are entered in the form $C=A+jB$, where A represents the real part of the constant and B represents the imaginary part.

1. Click **Number**.

   The **Input Number** dialog box appears.
2. Select **Scalar** or **Vector**.
3. Select **Complex**.
4. Enter the real and imaginary components of the number.
5. Click **OK** to load the number into the top register.

Function Command

Any functions you use must be defined prior to using this operation.

Enters one of the following into the top register of the calculator:

Scalar A scalar function.

To enter a function:

1. Click **Function**.

   The **Function** dialog box appears.
2. Select **Scalar**.
3. Select the function from the list.
4. Click **OK** to load the functional scalar into the top register.

Vector A vector function, in which the values of the vector's x-, y-, and z-components are given by functions.

To enter a functional vector:

1. Click **Function**.

   The **Function** dialog box appears.
2. Select **Vector**.
3. Select the function from the list.
4. For each component of the vector, click **SetX**, **SetY**, and **SetZ**.
5. Click **OK** to load the functional vector into the top register.

Note The predefined variables **X**, **Y**, **Z**, **RHO**, **THETA**, **R**, and **PHI** and any functions that you created can be used to define functional scalar and vector quantities.

Geom Settings Command

Clicking the **Geom Settings** button opens the **Geometric Settings** dialog box. The dialog box allows you to specify the line discretization, the number of equally-spaced points used to plot fields and other quantities on a line. The default is 1000 points.

Read Command

This command copies the contents of a disk file into the top register. The register must be one that has been saved using the Write output command.

To read in a register:

1. Click **Read**.
2. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assumed for register files.
3. Click **OK**.

   The contents of the file are copied to the top register in the stack.

General Commands

Use these commands to perform operations on both vector and scalar quantities.

+ (Add)

Adds the quantities in the top two registers of the calculator.

– (Subtract)

Subtracts the quantity in the top register from the quantity in the second register. The two registers must hold the same type of quantity (both scalar or both vector). You cannot subtract a scalar from a vector (or vice versa).

*** (Multiply)**

Multiplies the quantity in the top register by the quantity in the second register. One of the two registers must contain a scalar value; the other register can be either a scalar or a vector.

/ (Divide)

Divides the quantity in the second register by the quantity in the top register. The second register must contain a scalar value; the top register can be either a scalar or a vector.

Neg

Changes the sign of the quantity in the top register.

Abs

Takes the absolute value of the quantity in the top register.

Smooth

Smooths the quantity in the top register. Because of the numerical solution technique used, field values are not always continuous across the boundaries of the individual elements that make up the finite-element mesh. Smoothing makes the values continuous. In general, use smoothing before plotting a quantity.

Complex

These commands perform operations on a complex quantity in the top register. Complex quantities are indicated by a C at the beginning of the register label. They can be represented in terms of real and imaginary components, or in terms of magnitude and phase:

$$C = A + jB = Me^{j\phi}$$

where:

- A is the real part of the complex number.
- B is the imaginary part of the complex number.
- M is its magnitude, which is equal to $\sqrt{A^2 + B^2}$.
- ϕ is its phase, which is equal to $\text{atan}(B/A)$.

The **Complex** commands let you do the following:

Real	Takes the real part of the complex quantity (A).
Imag	Takes the imaginary part of the complex quantity (B).
CmplxMag	Takes the magnitude of the complex quantity (M).
CmplxPhase	Takes the phase of the complex quantity (ϕ).

Conj	Takes the complex conjugate of the quantity in the top register. If a complex number is given by $C = A + jB$, its complex conjugate is given by $C^* = A - jB$.
AtPhase	Lets you specify the phase angle, θ, at which an field quantity is evaluated. These quantities can be represented in the form $$\boldsymbol{A}(x, y, z, t) = \boldsymbol{A}(x, y, z)\cos\omega t + \theta(x, y, z).$$ where • ω is the angular frequency at which the quantities are oscillating, specified during the solution. • θ(x,y,z) is the phase angle (the offset from a cosine wave that peaks at t=0). Entering the phase angle lets you compute the real part of the field's magnitude at different points in its cycle.
CmplxReal	Converts the real scalar of the top register to the real part of a complex number.
CmplxIImag	Converts the real scalar of the top register to the imaginary part of a complex number.

Domain

This limits a calculation to the volume you specify. This operation requires the top two entries of the stack to be a volume geometry and a numeric field quantity. To do this:

1. Load the field quantity into the top register, and perform any necessary operations on it.
2. Load the volume using the **Geometry** command.
3. Click **Domain**.

The **Domain** command is often used to limit a calculation or plot to the intersection of a surface and an object or group of objects.

Scalar Commands

Use these commands to perform operations on scalar quantities.

Vec?	Makes the scalar quantity in the top register a vector component.
1/x	Takes the inverse of the scalar quantity in the top register.
Pow	Raises a scalar quantity to the power you specify.
√ (Square Root)	Takes the square root of the quantity in the top register.
Trig	Takes a selected trigonometric value of the value in the top register of the calculator stack
d/d?	Takes the partial derivative of the quantity in the top register.
∫ (Integral)	Takes the integral of a scalar quantity over a volume, surface, or line.

Min	Computes the minimum of a scalar field quantity on a line, surface, or volume.
Max	Computes the maximum of a scalar field quantity on a line, surface, or volume.
∇ (Gradient)	Takes the gradient of the scalar quantity in the top register.
ln	Takes the natural logarithm (base e) of the scalar quantity in the top register.
log	Takes the logarithm (base 10) of the scalar quantity in the top register

Vec? Command

Makes the scalar quantity in the top register a vector component. Choose from the following:

VecX	The x-component of a vector.
VecY	The y-component of a vector.
VecZ	The z-component of a vector.

1/x (Inverse) Command

Takes the inverse of the scalar quantity in the top register.

Pow Command

Raises a scalar quantity to the power you specify.

To raise a scalar quantity to a power:

1. Enter the quantity into the calculator.
2. Enter the exponent to which it is to be raised into the calculator.
3. Click **Pow**.

   The results are displayed in the top register.

 (Square Root) Command

Takes the square root of the quantity in the top register.

Trig

Takes one of the following trigonometric values of the value in the top register of the calculator stack:

Sin	Sine.
Cos	Cosine.
Tan	Tangent.
Asin	Arcsine.

Acos	Arccosine.
Atan	Arctangent.
Atan2	Arctangent squared.

d/d? (Partial Derivative) Command

Takes the partial derivative of the quantity in the top register:

d/dx	Takes the partial derivative of the quantity with respect to x.
d/dy	Takes the partial derivative of the quantity with respect to y.
d/dz	Takes the partial derivative of the quantity with respect to z.

∫ (Integral) Command

Takes the integral of a scalar quantity over a volume, surface, or line. The top register must contain a geometry and the second register must contain the scalar quantity to be integrated.

To perform an integration:

1. Load a quantity into the top register of the calculator, and perform any required operations on it.
2. Use one of the **Geometry** commands to load the line, surface, or volume over which the quantity is to be integrated.

Note If you computed the tangent or normal of the quantity to be integrated, you do not have to load a geometry onto the calculator stack. HFSS integrates the tangential or normal component of the quantity over the line on which you computed its tangent, or the surface on which you computed its normal.

3. Choose the ∫ command to integrate the scalar quantity over the geometry.

To find the numerical results of an integration, use the **Eval** command.

Min Command

Computes the minimum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the minimum value of the field.
Position	Finds the point where the minimum field value occurs. You can then: • Plot the minimum field value at the point using the **Plot** command. • Plot basic field quantities at the point. • Load the point into the calculator. • Change the point's location.

These commands operate in the same way as the **Max** commands. Use the **Eval** command to display the actual minimum field value or the coordinates of the point where it occurs.

Max Command

Computes the maximum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the maximum value of the field.
Position	Finds the point where the maximum field value occurs. You can then: • Plot the maximum field at the point using the **Plot** command. • Plot field quantities at the point. • Load the point into the calculator. • Change the point's location.

To compute the maximum field value:

1. Load a field quantity into the calculator, and perform any necessary operations on it. Keep the following in mind:
   - You cannot find the maximum value of a vector quantity. Therefore, make sure that the result is a scalar.
   - Before computing the maximum value of a complex quantity, you must find the real part of the quantity using the **Cmplx/Real** or **Cmplx/AtPhase** commands.
2. Load a point, line, or volume into the calculator using one of the **Geometry** commands.
3. Do one of the following:
   - Choose **Max/Value** to compute the maximum field value on the geometry.
   - Choose **Max/Position** to identify the point at which this value occurs.

Use the **Eval** command to display the actual maximum field value or the coordinates of the point where it occurs.

∇ (Gradient) Command

Takes the gradient of the scalar quantity in the top register.

Ln Command

Takes the natural logarithm (base e) of the scalar quantity in the top register.

Log Command

Takes the logarithm (base 10) of the scalar quantity in the top register.

Vector Commands

Use these commands to perform operations on vector quantities.

Scal?	Replaces the vector in the top register with a scalar quantity whose value is a component of the vector.
Matl	Multiplies or divides the vector field quantity in the top register by a material property
Mag	Takes the magnitude of the vector quantity in the top register
Dot	Takes the dot product of the vector quantities in the top two registers.
Cross	Takes the cross product of the vector quantities in the top two registers.
Divg	Takes the divergence of the vector quantity in the top register.
Curl	Takes the curl of the vector quantity in the top register.
Tangent	Computes the tangential component of a vector quantity along a line
Normal	Computes the normal component of a vector quantity on a surface such as a cutplane or object surface.
Unit Vec	Computes the normal or tangent unit vector. The unit vector is a “wild card” entry. The context is specified at the time of plotting, integrating, or report generation.

Scal? Command

Replaces the vector in the top register with a scalar quantity whose value is a component of the vector. Choose from the following:

ScalarX	Returns the x-component of the vector.
ScalarY	Returns the y-component of the vector.
ScalarZ	Returns the z-component of the vector.

Matl Command

Multiplies or divides the vector field quantity in the top register by a material property. At each tetrahedron, the field quantity is multiplied or divided by the value of the selected material property — taking the different material attributes of each object into account.

To multiply or divide a vector quantity by a material property:

1. Click **Matl**.

   The **Material Operation** window appears.

2. Select a material property. Available properties are:

Permittivity (epsi)	The relative permittivity, ε_r.
Permeability (mu)	The relative permeability, μ_r.
Conductivity	The conductivity, σ.
Omega (w)	The angular frequency, ω. The angular frequency is equal to $2\pi f$, where f is the frequency at which the solution was generated.

3. Select an operation — **Multiply** or **Divide**.
4. Choose **OK** to multiply or divide the field quantity by a material property or **Cancel** to stop the operation.

Mag Command

Takes the magnitude of the vector quantity in the top register.

Dot Command

Takes the dot product of the vector quantities in the top two registers.

Cross Command

Takes the cross product of the vector quantities in the top two registers.

Divg Command

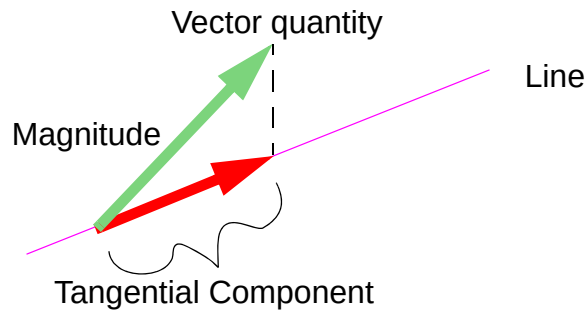
Takes the divergence of the vector quantity in the top register.

Curl Command

Takes the curl of the vector quantity in the top register.

Tangent Command

.



To take the tangent of a vector:

1. Load a vector quantity into the top register.
2. Load a line into the top register using the **Geometry/Line** command.

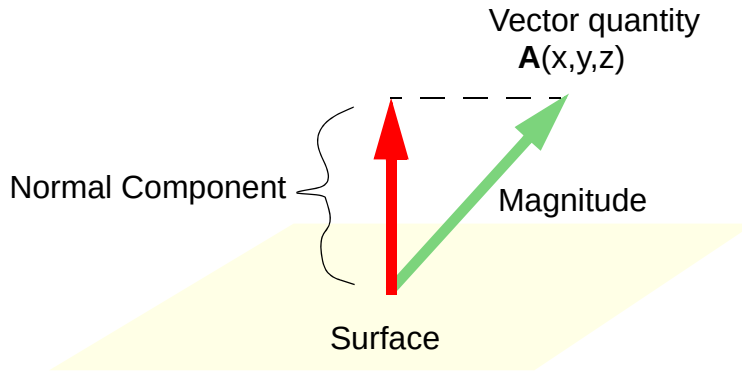
3. Click **Tangent**.

Normal Command

Computes the normal component of a vector quantity on a surface such as a cutplane or object surface. This is the equivalent of taking the dot product of the quantity with the surface's unit normal

$$Normal = \boldsymbol{A}(x, y, z) \bullet \hat{\boldsymbol{n}}$$

vector:



To take the normal of a vector:

1. Load a vector quantity into the top register.
2. Load a surface into the top register using the **Geometry/Surface** command.
3. Click **Normal**.

Note

Because surface normals of sheets are not well defined the fields calculator can produce incorrect results if an expression is evaluated on a sheet. To enforce the correct direction of the surface normal of a sheet, a faceted 3D object (such as a box) can be defined such that one of its planar faces is coincident with the sheet. Because surface normals of a valid object are always defined in an outward direction in HFSS, the fields calculator uses the surface normal of the face of the 3D object that is coincident with the sheet.î

Unit Vec Command

Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation.

Select from the following:

Tangent	Computes the unit vector tangent to the line specified at the time of plotting, integrating, or report generation based on the context.
Normal	Computes the unit vector normal to the surface specified at the time of plotting, integrating, or report generation based on the context.
CoordSys(X)	Computes the unit vector in the X-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys(Y)	Computes the unit vector in the Y-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys(Z)	Computes the unit vector in the Z-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.

Output Commands

Use these commands to compute or evaluate expressions and to output the data in the calculator.

Value command	Computes the value of a field quantity at a point.
Eval command	Numerically evaluates and displays the results of calculator operations.
Write command	Saves the contents of the top register to a disk file.
Export command	Saves field quantities in a format that can be read by other modeling or post-processing software packages.

Value Command

This computes the value of a field quantity at a point. Use it to find:

- The magnitude of a scalar field quantity at that point.
- The x-, y-, and z-components of a vector field quantity at that point.

To find the value of a field quantity at a point:

1. Load the field quantity into the top register, and perform any needed operations on it.
2. Load the appropriate point into the calculator using the **Geometry/Point** command.
3. Click **Value**.

To view the numerical results of this operation, use the **Eval** command.

Eval Command

This command numerically evaluates and displays the results of calculator operations such as integrations, maximum or minimum field computations, field values at points, and so forth. The quan-

tity to be evaluated must be in the top register. The **Eval** command computes the numerical results of the operation, which replace the contents of the register.

For instance, to find the current around a loop, you must numerically evaluate the following integral for that loop: $I = \oint H \bullet dl$.

Since **H** and **I** are complex quantities, you will need to evaluate the real part of **H** to obtain the real part of **I**, then evaluate the imaginary part of **H** to obtain the imaginary part of **I**. To do this:

1. Load **H** into the calculator using the **Qty** command.
2. Take the real part of **H** using the **Cmplx/Real** command.
3. Load the rectangular loop using the **Geom/Line** command. Create the loop, a closed polyline, to integrate over.
4. Click **Tangent** to get the component of **H** along the line.
5. Take the integral around the loop using the $\int$ command.
6. Click **Eval** to evaluate the integral. The real part of **I** appears in the top register.
7. Repeat this process using the imaginary part of **H** (found with the **Cmplx/Imag** command) to obtain the imaginary part of **I**.

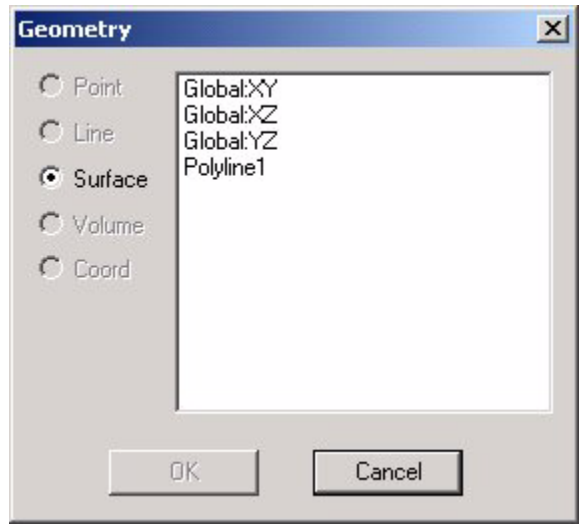
Write Command

This command saves the contents of the top register to a disk file. Use this command to:

- Save registers for use during a later post-processing session.
- Save a field quantity for use when post processing a different model.

To save a register:

1. Click **Write**.
2. If the register includes numeric with a constrained quantity (such as jsurf), you see a dialog that gives a choice of constraining geometries. For example:



3. Select the geometry of interest, and select **OK**.

   This displays a file browser.

4. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assigned to register files.

5. Click **OK**.

   The contents of the register are saved to the file you specified.

Export Command

This command opens the **Export Solution** dialog, from which you can export the field quantity in the top register to a file, mapping it to a grid of points. Use this command to save field quantities in a format that can be read by other modeling or post-processing software packages. Two options are available for defining the grid points on which to export:

Input grid points from file	Maps the field quantity to a customized grid of points. Before using this command, you must create a file containing the points. If you input grid points from a .pts file, the **Export** command takes the units from the model.
Calculate grid points	Maps the field quantity to a three-dimensional cartesian grid. You specify the dimensions and spacing of the grid in the x, y, and z directions, with units that you specify. The initial units are taken from the model.

To export a field quantity to a customized grid:

1. Load the quantity into the top register for the fields calculator, and perform any operations on

it.

2. Click the **Export** button in the Fields Calculator.

   This opens the Export Solution dialog.

3. Type or select the name of the file in which the field quantity is to be saved in the **Output File Name** text box. You can use the file icon to open the file browser to specify the file name and directory path. A **.reg** extension is automatically assigned to this file.

4. Click either the **Input grid points from file** button if you have a created a .pts file containing the grid points, or click the **Calculate grid points** button. For each grid dimension (**X**, **Y**, and **Z**), enter the following:

   Minimum The minimum x-, y-, or z-coordinate of the grid, and unit of measure.

   Maximum The maximum x-, y-, or z-coordinate of the grid, and unit of measure.

   Spacing The distance between grid points, and unit of measure.

- If you select **Input grid points from file**, either type the name and directory of the file containing the points on which the field is to be mapped, or, click on the file icon and use the file browser to locate the point file (**.pts** extension).
- If you select **Calculate grid points** button. For each grid dimension (**X**, **Y**, and **Z**), enter the following:

  Minimum The minimum x-, y-, or z-coordinate of the grid, and unit of measure.

  Maximum The maximum x-, y-, or z-coordinate of the grid, and unit of measure.

  Spacing The distance between grid points, and unit of measure.

5. Click **OK** to export the file.

   The field quantity is mapped to the grid and saved to the file you specified (**.reg** extension.).

Calculating Derived Field Quantities

The **Named Expressions** panel displays expressions that can be included in register definitions by name.

When an HFSS design is open and a Solution Setup has been performed, the following predefined named expressions are available:

Expression Name	Expression Definition
Mag_E	Mag(AtPhase(Smooth(<Ex,Ey,Ez>),Phase))
Mag_H	Mag(AtPhase(Smooth(<Hx,Hy,Hz>),Phase))
Mag_Jvol	Mag(AtPhase(Smooth(<JVx,JVy,JVz>),Phase))
Mag_Jsurf	Mag(AtPhase(Smooth(<Jsurfx,Jsurfy,Jsurfz>),Phase))
ComplexMag_E	Mag(CmplxMag(Smooth(<Ex,Ey,Ez>))

ComplexMag_H	Mag(CmplxMag(Smooth(<Hx,Hy,Hz>))
ComplexMag_Jvol	Mag(CmplxMag(Smooth(<JVx,JVy,JVz>))
ComplexMag_Jsurf	Mag(CmplxMag(Smooth(<Jsurfx,Jsurfy,Jsurfz>))
Vector_E	AtPhase(Smooth(<Ex,Ey,Ez>),Phase)
Vector_H	AtPhase(Smooth(<Hx,Hy,Hz>),Phase)
Vector_Jvol	AtPhase(Smooth(<JVx,JVy,JVz>),Phase)
Vector_Jsurf	AtPhase(Smooth(<Jsurfx,Jsurfy,Jsurfz>),Phase)
Vector_RealPoynting	Real(Poynting)
Local_SAR	LocalSAR
Average_SAR	AverageSAR
Surface_Loss_Density	SurfaceLossDensity. See further discussion here.
Volume_Loss_Density	VolumeLossDensity See further discussion here.

Click on a named expression to select it. When a named expression has been selected, the **Copy to Stack** button is activated. Click **Copy to Stack** to push the expression on the top of the stack.

Related Topics

Named Expression Library

Named Expression Library

To add a named expression of your own to the Fields Calculator list:

1. In the register display area, create the expression you want to plot.
2. When you are finished creating the expression, click **Add** in the **Named Expressions** panel.

   The **Named Expression** dialog box appears.
3. Type a name for the expression in the **Name** text box.

   The new expression is added to the list of named expressions.

When the **Named Expression** list contains one or more user-defined expressions, the **Delete** and **Clear All** buttons are active (you cannot delete or clear the predefined named expressions.) Click **Delete** to delete the selected user-defined named expression. Click **Clear All** to delete all user-defined named expressions.

To save one or more named expressions for the Fields Calculator to a personal Library:

1. Click the **Save To** button on the Fields Calculator.

   The **Select Expressions for Saving** dialog displays.
2. If any new named expressions exist, you can select one or more to save to a file.
3. Give a file name, and click **OK** to save the file.

To load named expressions for the Fields Calculator from a personal library:

1. From the Fields Calculator, click **Load From**.

   This displays a file browser that you can use to search for existing .clc files.

2. Select the library to load and click OK.

   This loads the expression file you have selected.

Related Topics

Calculating Derived Field Quantities

Exiting the Fields Calculator

Click **Done** to exit the Fields Calculator.

Radiated Fields Post Processing

To analyze the radiated fields associated with a design, define a radiation surface over which the fields will be calculated. The values of the fields over this surface are used to compute the fields in the space surrounding the device. This space is typically split into two regions — the near-field region and the far-field region. The near-field region exists at less than a wave-length from an energy source. The far field is where radiation occurs. See *Radiated Fields* for the specific equations used in HFSS for calculating the near and far field regions.

You can define a spherical surface over which to analyze the near or far fields by specifying a range and step size for phi and theta. This defines the spherical direction in which radiated fields will be evaluated. You can also draw a line along which to calculate the near fields.

Optionally, after defining the radiation surface, HFSS can compute antenna array radiation patterns and antenna parameters for designs that have analyzed a single array element. HFSS models the array radiation pattern by applying an “array factor” to the single element’s pattern when far fields are calculated. You set up the array factor information by defining either a finite, 2D array geometry of uniformly spaced, equal-amplitude elements (a regular array) or an arbitrary array of identical elements distributed in 3D space with individual complex weights (a custom array.)

HFSS can also compute antenna parameters, such as the maximum intensity, peak directivity, peak gain, and radiation efficiency. For near-field analysis, HFSS can also compute maximum parameters, such as the maximum of the total E-field and the maximum E-field in the x-direction.

Note When computing near and far fields, keep in mind that you must have defined at least one radiation or PML boundary in the design. At any time you may change the radiation surfaces that HFSS uses when calculating the radiated fields without needing to re-solve the problem, but the radiation-type boundary is still required.

Related Topics

Technical Notes: Radiated Fields

Setting up a Near-Field Sphere

To evaluate near fields on a spherical surface, set up a near-field sphere. To plot near-field values across the sphere, you will select the sphere object from the **Geometry** list in the **Traces** dialog box when you create a report.

1. Click **HFSS**>**Radiation**>**Insert Near Field Setup**>**Sphere**.

   The **Near Field Radiation Sphere Setup** window appears.
2. Under the **Sphere** tab, type a a name for the sphere in the **Name** text box.
3. Type the radius at which to compute the radiated fields in the **Radius** text box.

   The radius is measured from the origin of the sphere’s coordinate system, which is specified under the **Coordinate System** tab.
4. Specify the range of angles to include in the sphere:

a. Specify the following for **Phi**, in degrees (**deg**) or radians (**rad**):

Start	The point where the rotation of phi begins.
Stop	The point where the rotation of phi ends.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of phi.

b. Specify the following for **Theta**, in degrees (**deg**) or radians (**rad**):

Start	The point where the rotation of theta begins.
Stop	The point where the rotation of theta ends.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of theta.

See *Spherical Cross-Sections* in the *Technical Notes* for guidelines for setting phi and theta.

5. Click the **Coordinate System** tab, and then specify the orientation of the sphere in one of the following ways:
   - To orient the sphere according to the global coordinate system (CS), select **Use global coordinate system**.
   - To orient the sphere according to a user-defined CS, select **Use local coordinate system** and then select a defined CS from the **Choose from existing coordinate systems** list.
6. To specify a surface other than an assigned radiation or PML boundary over which to integrate the radiated fields, do the following:
   a. Click the **Radiation Surface** tab.
   b. Select **Use Custom Radiation Surface**.
   c. Select a defined face list from the list below.

      HFSS will use the surfaces in the face list as the radiating surfaces when calculating the near fields. The face list cannot include a face that lies on a PML object.
7. Click **OK**.

   The sphere is created. It is listed in the project tree under **Radiation**.

You must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Near Field Radiation Sphere Setup** window.

Note For parts of the sphere outside of the model region, near-field approximation is calculated. However, if parts of the sphere are inside the model region, the model fields are used to compute interpolated values. A section of the sphere is considered to overlap the model if it lies in the enlarged model region after accounting for symmetry planes.

Related Topics

Technical Notes: Spherical Cross-Sections

Setting up a Near-Field Line

To evaluate the near field along a line, set up a near-field line. The near-field line can be a polyline with two or more segments. To plot near-field values along the line, you will select the line object from the **Geometry** list in the **Traces** dialog box when you create a report.

1. Draw a polyline in post-processing mode.
2. Click **HFSS**>**Radiation**>**Insert Near Field Setup>Line**.

   The **Near Field Line Setup** dialog box appears.
3. Under the **Near Field Line Setup** tab, type a a name for the line in the **Name** text box.
4. Select the polyline along which you want to evaluate the near fields from the **Choose Line** list.
5. Specify the **Number of points** in the line.

   This is the total number of equally spaced points on the line. Specifying points on the line will enable you to plot the near-field values across a normalized distance, that is, to create a value versus distance plot of a near-field quantity on the line.
6. To specify a surface other than an assigned radiation or PML boundary over which to integrate the radiated fields, do the following:
   a. Click the **Radiation Surface** tab.
   b. Select **Use Custom Radiation Surface**.
   c. Select a defined face list from the list below.

      HFSS will use the surfaces in the face list as the radiating surfaces when calculating the near fields. The face list cannot include a face that lies on a PML object.
7. Click **OK**.

You must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Near Field Line Setup** window.

Note For parts of the near-field line lying outside of the model region, near-field approximation is calculated. However, if parts of the line lie inside the model region, the model fields are used to compute interpolated values. A section of the near-field line is considered to overlap the model if it lies in the enlarged model region after accounting for symmetry planes.

Computing Maximum Near-Field Parameters

You must have defined at least one radiation or PML boundary in the design for HFSS to compute maximum field data for the near-field region.

1. Right-click the **Sphere** or **Line** icon in the project tree, and then click **Compute Max Param-**

eters on the shortcut menu.

The **Select Solution** dialog box appears.

2. Under the **Solutions** tab, select the solution for which you want HFSS to compute the near-field parameters.
3. Under the **Intrinsic Variables** tab, select the solved frequency point at which you want HFSS to compute the near-field parameters.

   The **Max Field Data** window appears, listing the following information:

   Total

   X

   Y

   Z

   Phi

   Theta

   LHCP

   RHCP

   Ludwig 3/X dominant

   Ludwig 3/Y dominant

Note When calculating the maximum far-field values, the distance *r* is factored out of the E-field. Therefore, the units for the maximum field data values are given in volts.

Related Topics

Technical Notes: Maximum Near-Field Data

Setting up a Far-Field Infinite Sphere

To evaluate radiated fields in the far-field region, you must set up an infinite sphere that surrounds the radiating object. To plot far-field values across the sphere, you will select the sphere object from the **Geometry** list in the **Traces** dialog box when you create a report.

1. Click **HFSS**>**Radiation**>**Insert Far Field Setup**>**Infinite Sphere**.

   The **Far Field Radiation Sphere Setup** window appears.
2. Under the **Infinite Sphere** tab, type a a name for the sphere in the **Name** text box.
3. Specify the range of angles to include in the sphere:

a. Specify the following for **Phi**, in degrees (**deg**) or radians (**rad**):

Start	The point where the rotation of phi begins.
Stop	The point where the rotation of phi ends.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of phi.

b. Specify the following for **Theta**, in degrees (**deg**) or radians (**rad**):

Start	The point where the rotation of theta begins.
Stop	The point where the rotation of theta ends.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of theta.

See *Spherical Cross-Sections* in the *Technical Notes* for guidelines for setting phi and theta.

4. Click the **Coordinate System** tab, and then specify the orientation of the sphere in one of the following ways:
   - To orient the sphere according to the global coordinate system (CS), select **Use global coordinate system**.
   - To orient the sphere according to a user-defined CS, select **Use local coordinate system** and then select a defined CS from the **Choose from existing coordinate systems** list.
5. To specify a surface other than an assigned radiation or PML boundary over which to integrate the radiated fields, do the following:
   a. Click the **Radiation Surface** tab.
   b. Select **Use Custom Radiation Surface**.
   c. Select a defined face list from the list below.

      HFSS will use the surfaces in the face list as the radiating surfaces when calculating the far fields. The face list cannot include a face that lies on a PML object.

Note Do not use a sheet-object based face list as the radiation computation surface.

6. Click **OK**.

   The infinite sphere is created. It is listed in the project tree under **Radiation**.

Note You must have defined at least one radiation or PML boundary in the design for HFSS to compute far-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the far fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Far Field Radiation Sphere Setup** window.

Related Topics

Technical Notes: Spherical Cross-Sections

Creating a Face List

Defining Antenna Arrays

Define a regular or custom antenna array when you want HFSS to compute antenna array radiation patterns and antenna parameters for designs that have analyzed a single array element. HFSS models the array radiation pattern by applying an "array factor" to the single element's pattern when far fields are calculated.

The "regular uniform array" geometry defines a finite 2D array of uniformly spaced, equal-amplitude elements. This is a natural specification after analyzing a single-unit cell of an infinite array. The "custom array" geometry defines an arbitrary array of identical elements distributed in 3D space with individual user-specified complex weights.

Related Topics

Defining a Regular Antenna Array

Defining a Custom Antenna Array

Defining a Regular Antenna Array

A regular antenna array is a finite 2D array geometry of uniformly spaced, equal-amplitude elements.

1. On the **HFSS** menu, point to **Radiation**, and then click **Antenna Array Setup**.

   The **Antenna Array Setup** window appears.
2. Under the **Array Type** tab, select **Regular Array Setup**.
3. Click the **Regular Array** tab.
4. Under **First Cell Position**, enter the xyz-coordinates where the first cell is placed.
5. Under **Directions**, do the following:
   a. To the right of **U Vector**, enter the vector coordinates in the **X**, **Y**, and **Z** text boxes along which the cells in the U-direction are placed.
   b. To the right of **V Vector**, enter the vector coordinates in the **X**, **Y**, and **Z** text boxes along which the cells in the V-direction are placed.
6. Under **Distance Between Cells**, enter the distance between cells in the U-direction and the distance between cells in the V-direction in the design units.
7. Under **Number of Cells**, enter the number of unit cells in the U-direction and the number of unit cells in the V-direction.
8. Under **Scan Definition**, specify the scan direction in one of the following ways:
   - Select **Use Scan Angles**, and then enter the spherical coordinate angles, in degrees, in the radiation coordinate system in the **Theta** and **Phi** text boxes.
   - Select **Use Differential Phase Shift**, and then enter the phase difference between adjacent elements, in degrees, in the **In U direction** and **In V direction** text boxes.
9. Click **OK**.

   The array factor will be applied, using the information you specified, when far fields are calculated.

Related Topics

Technical Notes: Array Factor Calculation

Technical Notes: Regular Uniform Arrays

Technical Notes: Scan Specification for Regular Uniform Arrays

Defining a Custom Antenna Array

A custom antenna array is a an arbitrary array of identical elements distributed in 3D space with individual user-specified complex weights. The array is defined in a text file that includes the element positions, voltage amplitude weights, and phases (degrees). See *Custom Arrays* in the *Technical Notes* for examples of custom array geometry text files.

1. On the **HFSS** menu, point to **Radiation**, and then click **Antenna Array Setup**.

   The **Antenna Array Setup** window appears.
2. Under the **Array Type** tab, select **Custom Array Setup**.
3. Click the **Custom Array** tab.
4. Click **Import Definition**.

   The **Open** dialog box appears.
5. Follow the procedure for opening a file. Select **.txt** as the file type. When you are finished, click **Open**.
6. Optionally, review the definition in the text file by clicking **View Definition** under the **Custom Array Setup** tab.
7. Click **OK**.

   The array factor will be applied, using the information specified in the text file, when far fields are calculated.

Related Topics

Technical Notes: Custom Arrays

Technical Notes: Array Factor Calculation

Computing Antenna Parameters

You must have defined at least one radiation or PML boundary in the design for HFSS to compute antenna parameters and maximum field data for the far-field region.

1. Right-click the **Infinite Sphere** icon in the project tree, and then click **Compute Antenna Parameters** on the shortcut menu.

   The **Antenna Parameters** dialog box appears.
2. Under the **Solutions** tab, select the solution for which you want HFSS to compute antenna parameters.
3. Under the **Intrinsic Variables** tab, select the solved frequency point at which you want HFSS to compute antenna parameters.

   The **Antenna Parameters** window appears. If the design includes ports, the following antenna

parameters are listed:

Maximum intensity (Max U)

Peak directivity

Peak gain

Peak realized gain

Radiated power

Accepted power

Incident power

Radiation efficiency

Warning The computed values of max U and peak directivity depend on the user-determined set of aspect angles chosen for the computation of the radiated fields. If this set does not encompass the actual peak intensity of the radiated pattern, the displayed results for these three parameters will be inaccurate.

If the design does not have ports, the following antenna parameters are listed:

Maximum intensity (Max U)

Peak directivity

Radiated power

4. Click **More** to view the following maximum far-field data:

Total

X

Y

Z

Phi

Theta

LHCP

RHCP

Ludwig 3/X dominant

Ludwig 3/Y dominant

Note When calculating the maximum far-field values, the distance *r* is factored out of the E-field. Therefore, the units for the maximum field data values are given in volts.

Related Topics

Technical Notes: Antenna Parameters

Technical Notes: Maximum Far-Field Data

Exporting Antenna Parameters and Maximum Field Data

The **Antenna Parameters** dialog displays the calculated antenna parameters and Maximum Field data for a setup. The dialog also includes a buttons to **Export** antenna parameters and to **Export Fields**. The fields can be exported in the.csv format and imported into reporter as a table.

To export the antenna parameters to a text file:

1. Click the **Export** button on the **Antenna Parameters** dialog.
   This displays a file browser.
2. Specify the file name and location (or accept the defaults).
3. Click **Save**.
   This saves the text file and closes the browser.

To export the maximum field data to a comma separated format file:

1. Click the **Export Fields** button on the **Antenna Parameters** dialog
   This displays a file browser
2. Specify the file name and location (or accept the defaults.
3. Click **Save**.
   This saves the comma separated text file and closes the browser.

Far fields format:

```
[Point index] [Phi] [Theta] [rEPhi(mag ang)] [rETheta(mag ang)]
```

Near fields format:

```
[Point index] [X] [Y] [Z] [Ex(mag ang)] [Ey(mag ang)] [Ez(mag ang)]
```

Related Topics

Computing Antenna Parameters

Technical Notes: Antenna Parameters

Technical Notes: Maximum Far-Field Data

Plotting the Mesh

After the solution is complete, you can plot the finite element mesh on surfaces or within 3D objects.

1. Select a surface or object to create the mesh plot on or within.

   If it does not exist, create it.
2. On the **HFSS** menu, point to **Fields**, and then click **Plot Mesh**.

   The mesh appears on the surface or object you selected.

Setting Mesh Plot Attributes

1. On the **HFSS** menu, point to **Fields**, and then **Modify Plot Attributes**  .

   The **Select Folder** window appears.
2. Select the folder containing the mesh plot you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with mesh plot attribute settings appears.
3. Click the mesh plot you want to modify in the **Plot** list.
4. Use the **Scale factor** slider to increase (move to the right) or decrease (move to the left) the percentage of the tetrahedra size.

   For example, a scale factor of 80% draws the tetrahedra at 80% of their original size.
5. Use the **Transparency** slider to increase (move to the right) or decrease (move to the left) the transparency of the plot.

   This is useful for viewing objects or plots behind the current plot.
6. Select one of the following display options:

Wire frame	Draws wireframe outlines of the tetrahedra.
Shaded	Draws shaded tetrahedra.
Add Grid	Displays the mesh.

7. Under **Mesh color**, click the **Line** color box, and then select a color for the outline of the tetrahedra from the **Color** palette.
8. Under **Mesh color**, click the **Filled** color box, and then select a color to fill the tetrahedra with from the **Color** palette.
9. Select **Surface Only** to only display the faces of tetrahedra that lie on object surfaces.

   Clear this option to draw all tetrahedra inside selected objects.
10. Click **Save as default** if you want the tab's settings to apply to mesh plots created after this point.
11. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.

12. Click **Close** to dismiss the dialog box.

Creating Reports

After HFSS has generated a solution, all of the results for that solution are available for analysis. One of the ways you can analyze your solution data is to create a 2D or 3D report, or graphical representation, that displays the relationship between a design's values and the corresponding analysis results. Reports are created using either the **Quick Report** command, or by using the **Traces** dialog box to create a custom report. The **Quick Report** feature lets you select from a list of predefined categories (such as S-parameters) from which to create a rectangular plot. The **Traces** dialog box offers more options for display formats, more categories to select (including, for example, output variables). In both cases, the report content depends on the current setup and solution data.

Note Remember the evaluated value of an expression is always interpreted as in SI units. However, when a quantity is plotted in a report, you have the option to plot values in units other than SI. For example, the expression "1+ang_deg(S11)" represents an 'angle''quantity evaluated in radians) though plotted in degrees units. To represent an angle quanity in degrees, you would specify units as "1 deg + ang_deg(S11)".

Creating a Quick Report

Following is the procedure for creating a quick report.

1. On the project tree, select a setup of interest.
2. Right-click to display the shortcut menu and select **Quick Report**.

   The **Quick Report** dialog appears.
3. Select the one or more categories for the report from the list and click OK.

   A rectangular plot of for each selected category displays.

Related Topics

Creating Reports

Modifying Reports

Creating a Custom Report

Following is the general procedure for creating a custom report:

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** dialog box appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.
3. In the **Report Type** list, click the data type you want to plot. There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, select the type of report you want to create.
5. Click **OK**.

The **Traces** dialog box appears.

6. In the **Solution** list, click the solution containing the data you want to plot.
7. In the **Domain** list, click a domain.

   For modal and terminal S-parameter reports, the domain can be frequency or time.

   If you select **Time**, following the directions for time-domain plotting.
8. To create a new mathematical expression to plot, do the following:
   a. Click **Output Variables**.

      The **Output Variables** dialog box appears.
   b. Add the expression you want to plot, and then click **Done**.
9. Add one or more traces to include in the report.
10. Click **Done**.

    The report appears in the view window. It will be listed in the project tree. Some plots may take time to complete. Performing a **File>Save** in such cases after the plot has been created will permit you to review the plot later without having to repeat the calculation time.
11. To speed redraw times when plots are changed, perform a **Save**. This saves the data that comprises expressions.For example if re(S11)*re(S22) is requested over multiple width, each of the S11 and S22 are stored when you save.If you do not do a save, it is not stored.

Related Topics

Creating Reports

Modifying Reports

Modifying Reports

To modify the data that is plotted in a report:

1. In the project tree, click the report you want to modify.
2. On the **Report2D** or **Report3D** menu, click **Modify Report**.

   The **Traces** dialog appears.
3. Modify the selections in the **Traces** dialog box as needed.
4. Click **Done**.

   The updated report appears in the view window.

Related Topics

Modifying Background Properties of a Report

Modifying the Background Properties of a Report

To modify the appearance of a report.

1. Select the report you want to modify, either in the Project tree, or as an open report.
2. Click **Report2D>Properties** or right-click to display the shortcut menu and click **Properties**.

   This displays the Background Properties dialog. This contains tabs for editing the following

properties: You must select an editable object in the report to be able to edit its properties. Each tab contains the following commands:

- A **Reset** button cancels any changes you make.
- A **Save As Default** button saves the current scheme.
- An **OK** button applies the changes to the current selection in the report.
- A **Close** button closes the dialog.

The tabs are as follows:

- **Color** - this tab contains a square to show the current color of the selected object. It also contains red, green and blue color sliders you can use to create colors. It also contains sixteen preset colors that you can select.
- **Font** - this tab lets you change the font type, style, and size of text currently selected in the report. It contains a preview field that shows how the currently select font type, style and size will look.
- **Line Style** - this tab lets you select a trace line-style, width, symbol drawing intervals, and includes check boxes to specify whether to apply the style to all traces, and whether to show symbols on all traces.

Note You can also change the line style selecting a trace and then right-clicking on the report to display the shortcut menu. Under **Trace>Type**, the menu contains entries for Continuous, Discrete, Bar-Zero, Bar Infinity, Stick Zero, Stick Infinity, Histogram, Step, and Stair.

- **Scaling** - this tab applies to the Report graph. It contains check boxes for autoscale, autounits, show the minor grid (all default), and use scientific notation (off by default).
  - Uncheck autoscale to enable Manual Scaling. With this enabled, you can specify Min and Max values, and the size of the Major and Minor divisions in the grid.

    Autoscale may create plots that look different from earlier versions of HFSS, but with manual scaling, earlier displays can be replicated, if desired.
  - Uncheck autounits to enable the Format Fields. These include Unit, Field Width, and Precision values.
  - Check Scientific Notation to display values in scientific notation. The default is standard notation.
  - Radio buttons show the selection for a Linear scale (the default) or Log scale.
  - A Label field shows the default name for the X-axis. By default this is taken from the Traces dialog X-tab Quantity setting.
- **Title** - this tab lets you edit the text displayed at the top center of the report.
- **Legend** - this tab lets you select whether to automatically generate the race description (the default), or lets you enter your own text.

3. Edit the properties, and **OK** the dialog to apply the changes.

Related Topics

Modifying Reports

Selecting the Report Type

The **Report Types** available for creating a report depends on the simulation setup. Whereas a modal setup will permit you to create a report on modal s-parameters and fields, a terminal setup permits you to also create reports based on modal s-parameters, fields, far fields, and near fields.

Depending on the setup, you can make selection from the following report types:

Modal Solution Data	S-, Y-, and Z-parameter data will be available to plot, as well as propagation constant, characteristic port impedance, reflection/ transmission coefficients for FSS designs, and voltage standing wave ratio (VSWR) data. **Note:** For FSS calculations, phase is currently assigned zero value.
Terminal Solution Data	Terminal S-, Y-, and Z-parameter data will be available to plot, as well as terminal characteristic port impedance, common and differential voltage quantities, power, and VSWR data.
Fields	Basic or derived field quantities calculated on surfaces or objects will be available to plot.
Far Fields	Radiated fields computed in the far-field region. The following quantities will be available to plot: rE, gain, realized gain, directivity, axial ratio, polarization ratio, antenna parameters, and normalized antenna calculated by HFSS. Note: You must have defined an infinite sphere geometry and at least one radiation or PML boundary to create a far-fields report.
Near Fields	Radiated fields computed in the near-field region. These include: variables, output variables, near E, max near field parameters, and near normalized antenna. Note: You must have defined a near-field line or near-field sphere and at least one radiation or PML boundary to create a near-fields report.
Emission Test	You can conduct an emission test under the same conditions as for a near field report except that.an emission test cannot be conducted for a ports-only solution. You must have defined a near-field line or near-field sphere and at least one radiation or PML boundary.

Selecting the Display Type

The information in a report can be displayed in several formats. Select from the following **Display Type** formats in the **Create Report** window:

Rectangular Plot	A 2D rectangular (x-y) graph.
3D Rectangular Plot	A 3D rectangular (x-y-z) graph.
Polar Plot	A 2D circular chart divided by spherical coordinates.
3D Polar Plot	A 3D circular plot divided by spherical coordinates.
Smith Chart	A 2D polar chart of S-parameters upon which a normalized impedance grid has been superimposed.
Data Table	A spreadsheet with rows and columns that displays, in numeric form, selected quantities against a swept variable or another quantity.
Radiation Pattern	A 2D polar plot of radiated fields.

Creating 2D Rectangular Plots

A rectangular plot is a 2D, x-y graph of results.

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.
3. In the **Report Type** list, click the data type you want to plot. There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, click **Rectangular Plot**, and then click **OK**.

   The **Traces** dialog box appears. The **Y** tab is selected by default.
5. Under the **Y** tab, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
6. Under the **X** tab, specify the quantity to plot along the x-axis in one of the following ways:
   - Select **Use Primary Sweep**.

     The first (primary) sweep variable listed under the **Sweeps** tab will be plotted along the x-axis.
   - Clear the **Use Primary Sweep** option, and then select the **Category**, **Quantity**, and **Function** of the quantity to plot on the x-axis. The quantity will be plotted against the primary sweep variable listed under the **Sweeps** tab.
7. Under the **Sweeps** tab, confirm or modify the sweep variables that will be plotted.
8. Click **Add Trace**.

A trace represents one or more lines connecting data points on the graph. The trace is added to the traces list at the top of the **Traces** dialog box. Each column lists an axis on the report and the information that will be plotted on that axis.

9. Optionally, add another trace by following the procedure above.
10. Click **Done**.

    The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree.

Related Topics

Sweeping a Variable

Working with Traces

Delta Markers in 2DPlots

Modifying Background Properties of a Report

Creating 3D Rectangular Plots

A rectangular plot is a 3D, x-y-z graph of results.

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.
3. In the **Report Type** list, click the data type you want to plot.There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, click **3D Rectangular Plot**, and then click **OK**.

   The **Traces** dialog box appears. The **Z** tab is selected by default.
5. Under the **Z** tab, specify the information to plot along the z-axis:
   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
6. Under the **Y** tab, specify the information to plot along the y-axis in one of the following ways:
   - Select **Use Secondary Sweep**.

     The second (secondary) sweep variable listed under the **Sweeps** tab will be plotted along the y-axis.
   - Clear the **Use Secondary Sweep** option, and then select the **Category**, **Quantity**, and **Function** of the quantity to plot on the y-axis. The quantity you select will be plotted against the secondary sweep variable listed under the **Sweeps** tab.
7. Under the **X** tab, specify the information to plot along the x-axis in one of the following ways:
   - Select **Use Primary Sweep**.

     The first (primary) sweep variable listed under the **Sweeps** tab will be plotted along the x-axis.

- Clear the **Use Primary Sweep** option, and then select the **Category**, **Quantity**, and **Function** of the quantity to plot on the x-axis. The quantity you select will be plotted against the primary sweep variable listed under the **Sweeps** tab.

8. Under the **Sweeps** tab, confirm or modify the swept variables that will be plotted.
9. Click **Add Trace**.

   A trace represents one or more lines connecting data points on the graph. The trace is added to the traces list at the top of the **Traces** dialog box. Each column lists an axis on the report and the information that will be plotted on that axis.
10. Optionally, add another trace by following the procedure above.
11. Click **Done**.

    The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the project tree.

Related Topics

Sweeping a Variable

Working with Traces

Creating 2D Polar Plots

In HFSS, a polar plot is a 2D circular chart divided by the spherical coordinates R and theta, where R is the radius, or distance from the origin, and theta is the angle from the x-axis. Following is the general procedure for drawing a polar graph of results:

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.
3. In the **Report Type** list, click the data type you want to plot.There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, click **Polar Plot**, and then click **OK**.

   The **Traces** dialog box appears. The **Polar** tab is selected by default.
5. Under the **Polar** tab, specify the information to plot:
   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
6. Under the **Sweeps** tab, specify the values against which to plot the information you specified in step 5.
7. Click **Add Trace**.

   A trace represents one or more lines connecting data points on the graph. The trace is added to the traces list at the top of the **Traces** dialog box. Each column lists an axis on the report and the information that will be plotted on that axis.

8. Optionally, add another trace by following the procedure above.
9. Click **Done**.

   The function of the selected quantity will be plotted against the values you specified on a polar graph. The plot is listed under **Results** in the project tree.

Related Topics

Reviewing 2D Polar Plots

Sweeping a Variable

Working with Traces

Reviewing 2D Polar Plots

For a polar plot of S-parameters, HFSS displays in the lower-left corner the following derived information about the cursor's location:

MP The magnitude and phase of the point.

RX The normalized resistance (**R**) and reactance (**X**).

GB An alternate view of the normalized resistance and reactance in the form of

$$R + jX = \frac{1}{G + jB}$$

where

- ***G*** = conductance
- ***B*** = susceptance

Q The quality factor.

VSWR The voltage standing wave ratio, calculated from the equation $\frac{1 + |S_{ij}|}{1 - |S_{ij}|}$.

A scale below the plot displays the scale of points along the R-axis.

Related Topics

Creating 2D Polar Plots

Creating 3D Polar Plots

A 3D polar plot is a 3D circular chart divided by the spherical coordinates R, theta, and phi, where R is the radius, or distance from the origin, theta is the angle from the x-axis, and phi is the angle from the origin in the z direction. Following is the general procedure for drawing a 3D polar plot of results:

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.
3. In the **Report Type** list, click the data type you want to plot. There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions

(modal and fields).

4. In the **Display Type** list, click **3D Polar Plot**, and then click **OK**.

   The **Traces** dialog box appears. The **Mag** tab is selected by default.

5. Under the **Mag** tab, specify the information to plot along the R-axis, or the axis measuring magnitude:

   a. In the **Category** list, click the type of information to plot.

   b. In the **Quantity** list, click the value to plot.

   c. In the **Function** list, click the mathematical function of the quantity to plot.

6. Under the **Theta** tab, specify the information to plot along the theta-axis:

   - To plot the secondary sweep information (the second sweep variable listed under the **Sweeps** tab) along theta, select **Use Secondary Sweep**.
   - To plot a quantity along theta, clear the **Use Secondary Sweep** option, and then select the **Category**, **Quantity**, and **Function** of the quantity to plot.

     The quantity will be plotted against the secondary sweep variable specified under the **Sweeps** tab.

7. Under the **Phi** tab, specify the information to plot along the phi-axis:

   - To plot the primary sweep information (the first sweep variable listed under the **Sweeps** tab) along phi, select **Use Primary Sweep**.
   - To plot a quantity along phi, clear the **Use Primary Sweep** option, and then click the **Category**, **Quantity**, and **Function** of the quantity to plot.

     The quantity will be plotted along phi against the primary sweep variable specified under the **Sweeps** tab.

8. Under the **Sweeps** tab, specify the values across which to plot the information you specified in steps 5, 6, and 7.

9. Click **Add Trace**.

   The trace is added to the traces list at the top of the **Traces** dialog box. Each column lists an axis on the report and the information that will be plotted on that axis.

10. Click **Done**.

    The function of the selected quantity or quantities will be plotted against the R-, phi-, and theta-axes on a 3D polar graph. The plot is listed under **Results** in the project tree.

Related Topics

Sweeping a Variable

Working with Traces

Creating Smith Charts

A Smith chart is a 2D polar plot of S-parameters upon which a normalized impedance grid has been superimposed. Following is the general procedure for creating a Smith chart of results:

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.
3. In the **Report Type** list, click the data type you want to plot. There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, click **Smith Chart**, and then click **OK**.

   The **Traces** dialog box appears. The **Polar** tab is selected by default.
5. Under the **Polar** tab, specify the information to plot:
   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
6. Under the **Sweeps** tab, specify the values across which to plot the information you specified in step 5.
7. Click **Add Trace**.

   A trace represents one or more lines connecting data points on the graph. The trace is added to the traces list at the top of the **Traces** dialog box. Each column lists an axis on the report and the information that will be plotted on that axis.
8. Optionally, add another trace by following the procedure above.
9. Click **Done**.

   The function of the selected quantity will be plotted against the values you specified on a polar plot. In addition, each circle on the plot is labeled with values of *R*, measuring normalized resistance, and each line is labeled with values of *X*, measuring normalized reactance. The plot is listed under **Results** in the project tree.

Related Topics

Reviewing 2D Polar Plots

Sweeping a Variable

Working with Traces

Creating Data Tables

A data table is a spreadsheet with rows and columns that displays, in numeric form, selected quantities against a swept variable or other quantities.

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.

3. In the **Report Type** list, click the data type you want to plot. There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, click **Data Table**, and then click **OK**.

   The **Traces** dialog box appears. The **Y** tab is selected by default.
5. Under the **Y** tab, select the quantity you are interested in and its associated function:
   a. In the **Category** list, click the type of information to display.
   b. In the **Quantity** list, click the value to display.
   c. In the **Function** list, click the mathematical function to use for the quantity.
6. Under the **X** tab, select the values you want to plot the quantity against in one of the following ways:
   - Select **Use Primary Sweep**.

     The quantity you selected in step 5 will be displayed against the first (primary) sweep variable listed under the **Sweeps** tab.
   - Clear the **Use Primary Sweep** option, and then select the **Category**, **Quantity**, and **Function** of the quantity to plot against the quantity you selected in step 5. This quantity will be plotted against the primary swept variable listed under the **Sweeps** tab.
7. Under the **Sweeps** tab, confirm or modify the swept variables that will be plotted.
8. Click **Add Trace**.

   In the context of a data table, a trace represents a quantity's value at another quantity's value or at selected swept variable values. The trace is added to the traces list at the top of the **Traces** dialog box.
9. Optionally, add another trace by following the procedure above.
10. Click **Done**.

    The quantity you selected in step 5 will be listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the project tree.

Related Topics

Sweeping a Variable

Working with Traces

Creating Radiation Patterns

A radiation pattern is a 2D polar plot displaying the intensity of near- or far-field radiation patterns. It is divided by the spherical coordinates R and theta, where R is the radius, or distance from the origin, and theta is the angle from the x-axis. Following is the general procedure for drawing a radiation pattern of results:

1. On the **HFSS** menu, point to **Results**, and then click **Create Report**.

   The **Create Report** window appears.
2. In the **Target Design** list, click the design containing the solution data you want to plot.

3. In the **Report Type** list, click the data type you want to plot. There are more types available for terminal solutions (terminal, model, fields, near fields, and far fields) than for modal solutions (modal and fields).
4. In the **Display Type** list, click **Radiation Pattern**, and then click **OK**.

   The **Traces** dialog box appears. The **Mag** tab is selected by default.
5. Under the **Mag** tab, specify the information to plot along the R-axis, or the axis measuring magnitude:
   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
6. Under the **Ang** tab, specify the information to plot along the theta-axis in one of the following ways:
   - Select **Use Primary Sweep**.

     The first (primary) sweep variable listed under the **Sweeps** tab will be plotted along the theta-axis.
   - Clear the **Use Primary Sweep** option, and then select the **Category**, **Quantity**, and **Function** of the quantity to plot on the theta-axis. The quantity you select will be plotted against the primary sweep variable listed under the **Sweeps** tab.
7. Under the **Sweeps** tab, specify the values across which to plot the information you specified in steps 5 and 6.
8. Click **Add Trace**.

   A trace represents one or more lines connecting data points on the graph. The trace is added to the traces list at the top of the **Traces** dialog box. Each column lists an axis on the report and the information that will be plotted on that axis.
9. Optionally, add another trace by following the procedure above.
10. Click **Done**.

    The function of the selected quantity or quantities will be plotted against the values you specified on a 2D polar plot. The plot is listed under **Results** in the project tree.

Related Topics

Sweeping a Variable in a Report

Working with Traces

Delta Markers in 2D Reports

To view the difference between any two marker points in a report:

1. Set the first marker by left-clicking and holding the mouse button.
2. Move the mouse without releasing left button to another position, and then release the left button to create second marker.

In the marker text window, you see the difference between the two markers instead of the X, Y value of marker.

Plotting in the Time Domain

1. For a design with an existing sweep setup, follow steps 1 - 4 for creating a report for design.
2. In the **Traces** dialog box, in the **Domain** list, click **Time**.
3. Click **TDR Options**.

   The **TDR Options** dialog box appears.
4. Select the input signal type, **Step** or **Impulse**.
5. Enter the rise time of the pulse in the **Rise Time** text box.
6. Enter the total time on the plot in the **Maximum Plot Time** text box.
7. Specify the number of time points that will be on the plot in the **Delta Time** text box. By default, this is set to the number of points in the frequency sweep.
8. Optionally, under **TDR Window**, modify the window type and width.
9. Click **OK**.

   Optionally, to plot TDR impedance (that is, rather than calculate the S-parameter for waveport1 versus frequency, calculate the delay versus time at a particular impedance), do the following:

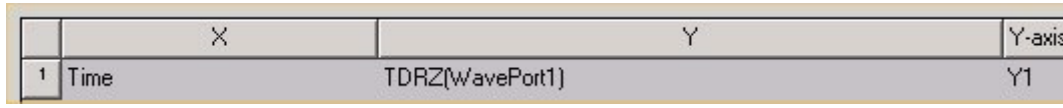
   a. In the **Category** list, click **TDR Impedance**.

   b. In the **Quantity** list, click a TDRZ quantity to plot.

   The TDRZ quantity represents the port impedance translated into the time domain. Its formula is Z_0*(1+ re(S11))/(1-re(S11)).

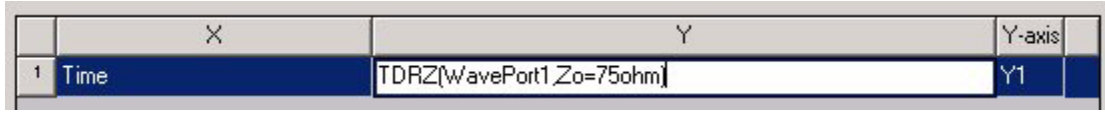
   The default impedance (Zo) is 50 Ohms. If you need a different impedance value:

   1. Click the **Add Trace** button to place the default TDRZ value in the Y-column of the selected traces table.



   2. Edit the value by clicking on the Y column in the **Add Traces** dialog box.

      In this example, the value for Zo is changed from the default to 75 Ohms by typing ‘,Zo=75ohm’ in the Y-column field.



   c. In the **Function** list, click the mathematical function of the quantity to plot.
3. Click **Done**.

   The report appears in the view window. It will be listed in the project tree.

Working with Traces

A trace in a 2D or 3D report defines one or more curves on a graph. A trace in a data table defines part of the displayed matrix of text values.

The values used for a plot's axes (which may be X, Y, Z, phi, theta, or R depending on the display type) can be variables in the design, such as frequency, or functions and expressions based on the design's solutions. If you have solved one or more variables at several values, you can "sweep" over some or all of those values, resulting in a curve in 2D or 3D space.

A report can include any number of traces and, for rectangular graphs, up to four independent y-axes.

In general, to add a trace to a report:

1. In the **Traces** dialog box, specify the information you want to plot along the appropriate axes.
2. Click **Add Trace**.

   A trace is added to the traces list at the top of the **Traces** dialog box. The trace represents the function of the quantity you selected and will be plotted against other quantities or swept variable values. Each column lists an axis on the report and the information that will be plotted on that axis.

   You can modify the information to be plotted by typing the name of the quantity or sweep variable to plot along an axis directly in the text boxes.

   The trace will be visible in the report when you click **Done**.

Note If you click **Done** without adding a trace, an **Edit Traces** box opens to note that a trace is required for a report. Click **Add Trace** to add the selected variable in the **Traces** dialog box as a trace and display the report. Click **Discard Changes** to terminate the **Create Report** dialog. Click **Return to Editor** to continue entering values in the **Traces** dialog box.

Removing Traces

You can remove traces from the traces list in the following ways:

To *remove one trace* from the report:

- Select the trace you want to remove from the traces list, and then click **Remove Trace**.

To *remove all traces* from the report:

- Click **Remove All Traces**.

Related Topics

Working with Traces

Replacing Traces

To replace a trace in the traces list with a different trace definition:

1. Select the trace you want to remove from the traces list.
2. In the **Traces** dialog box, specify the information you want to plot along the appropriate axes.
3. Select **Replace Trace**.

   The trace you selected will be removed and the new trace information you specified will replace it in the traces list.

Related Topics

Working with Traces

Adding Blank Traces

To add a blank trace to the traces list:

- Click **Add Blank Trace**.

You can now type the quantities to plot in the appropriate axes text boxes.

Related Topics

Working with Traces

Sweeping a Variable in a Report

In HFSS, a swept variable is a variable that typically has more than one value. You can plot any calculated or derived quantity against one or more of the swept variable's values.

To specify the swept variable values to plot a selected quantity against:

1. Under the **Sweeps** tab in the **Traces** dialog box, select one of the following:

Use current design and project variable values	The variables you can select to sweep will be the intrinsic variables available in the nominal project. If a new solution is generated with changed design variables, the report will be updated to reflect the new values.
Sweep design and project variable values	The variables you can select to sweep will include the design or project variable values that were solved during the active project's Optimetrics analyses.

2. The first sweep variable listed is the "primary sweep". If you are creating a 3D report, the second sweep variable listed is the "secondary sweep". Any additional sweep variables will be represented as additional curves on the graph.

   To modify which variable is the primary sweep variable:

   - Click the **Name** text box for the primary sweep variable, and then click the variable name you want to be the primary sweep variable.

   To modify the secondary sweep variable or any additional sweep variable, follow the same procedure.
3. To modify the values that will be plotted for a variable:

a. Click a variable in the table.

   To the right, all of the possible values for the selected variable are listed.

b. Select **All Values**.

   All of the selected variable's values will be plotted.

   - Alternatively, clear **All Values** and select the specific values to plot against the selected quantity.

Sweeping Values Across a Normalized Distance

1. If you are plotting a field quantity along a line, define a polyline object in the problem region.

   If you are plotting a near-field quantity along a line, set up a near-field line.
2. In the **Traces** dialog box, click the line geometry of interest in the **Geometry** list.
3. Under the appropriate axes tabs, specify the quantities you want to plot along the axes.
4. Under the **Sweeps** tab, select the **NormalizedDistance** variable.

   The values at which the selected quantity or quantities will be plotted are listed to the right. By default, a post-processing polyline object is divided into 100 equally spaced points. A near-field line is divided into the number of points you specified when you set it up. To modify the number of points on the line, do the following:

   a. Click **Edit Sweep**.

   b. Type the number of points in the line in the **Number of Values** text box. This is the total number of equally spaced points on the line.

   c. Click **Update Values**, and then click **OK**.

   The values listed are updated to reflect the new number of points.
5. To plot the selected quantity or quantities at every point on the line, select **All Values**.

   To plot the selected quantity or quantities at specific points on the line, clear the **All Values** option, and then select the point values on which you want to plot.

Note All maximum near-field data calculated by HFSS is at its maximum over the selected line object; if you plot the parameter over a sweep of values, the parameter will have the same value at each point on the plot.

Related Topics

Sweeping a Variable in a Report

Sweeping Values Across a Sphere

1. Set up a near-field sphere or a far-field infinite sphere.
2. In the **Traces** dialog box, click the sphere geometry of interest in the **Geometry** list.
3. Under the appropriate axes tabs, specify the quantities you want to plot along the axes.
4. Under the **Sweeps** tab, click the **phi** row in the table.

   To the right, all of the possible values for the phi variable are listed. The values are the result of the range of phi you specified during the infinite sphere's setup. To modify the values of phi to

be plotted across the sphere, do the following:

a. Click **Edit Sweep**.

b. Specify the following information:

Start Value	The point where the rotation of phi begins.
End Value	The point where the rotation of phi ends.
Number of Values	The number of values between the start value and the end value.

c. Click **Update Values**, and then click **OK**.

The values listed are updated to reflect the new number of points.

5. To plot the selected quantity or quantities at every value of phi, select **All Values**.

   To plot the selected quantity or quantities at specific values of phi, clear the **All Values** option, and then select the phi values at which you want to plot.

6. Under the **Sweeps** tab, click the **theta** row in the table. Follow steps 4 and 5 for modifying the values of theta, if necessary, and specifying the theta values at which to plot the selected quantity or quantities.

Note All antenna parameters and maximum far-field data calculated by HFSS is at its maximum over the selected object; if you plot the parameter over a sweep of values, the parameter will have the same value at each point on the plot.

Refresh Sweeps

If you change the sweep variable range in HFSS, the plot does not automatically update. For example, you may run a sweep from 1-10 GHz and create an S-parameter plot. If you modify the sweep to run from 1 - 60 GHz, the plot still shows data from 1-10 GHz. To update the plot to show the new sweep range:

1. Select a trace or traces.
2. Click the **Refresh Sweep** button under the **Sweeps** tab on the **Traces** dialog box to update the sweeps to the current values of the corresponding variables.

   A Warning message appears, saying that clicking **OK** will re-write all the sweeps.
3. Click on **OK** to get the latest values of the sweeps.

   The plots are updated.

Selecting a Function

The value of a quantity being plotted depends upon its mathematical function, which you select from the **Function** list in the **Traces** dialog box. The available, valid functions depends on the type of quantity (real or complex) that is being plotted. The function is applied to the quantity which is implicitly defined by all the swept and current variables. For example, “S(11)” is the value of the S-parameter for every swept combination of variables (e.g., “height”, “frequency” and so forth).

Some of these functions can operate along an entire curve. These are: deriv, min, max, integ, avg, rms, pk2pk, cang. These functions have syntax as follows:

- deriv(*quantity*) implicitly implies derivative over the primary sweep
- deriv(*quantity*, *SweepVariable*) explicitly means derivative over the sweep variable specified in the second argument (such as “Freq”).

You can select from the following functions in the **Function** list:

abs	Absolute value
acos	Arc cosine
acosh	Hyperbolic arc cosine
ang_deg	Angle (phase) of a complex number, cut at +/-180
ang_rad	Angle in radians
asin	Arc sine
asinh	Hyperbolic arc sine
atan	Arc tangent
atanh	Hyperbolic arc tangent
avg	Average
cang_deg	Cumulative angle (phase) of a complex number, cut at +/-180
cang_rad	Cumulative angle in radians
conjg	Conjugate of the complex number.
cos	Cosine
cosh	Hyperbolic cosine
dB(x)	20*log10(|x|)
dBm(x)	10*log10(|x|) +30
dBW(x)	10*log10(|x|)
deriv	Derivative.
even	Returns 1 if integer part of the number is even; returns 0 otherwise
exp	Exponential function (the natural anti-logarithm)
im	Imaginary part of the complex number
int	Truncated integer function
integ	Integral.
j0	Bessel function of the first kind (0^{th} order)
j1	Bessel function of the first kind (1^{st} order)

ln	Natural logarithm
log10	Logarithm base 10
mag	Magnitude of the complex number
max	Maximum value along a curve.
max_swp	Maximum value of a sweep.
min	Minimum value along a curve.
min_swp	Miniumum value of a sweep.
nint	Nearest integer
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise
pk2pk	Peak to peak.
polar	Converts the complex number in rectangular to polar
re	Real part of the complex number
rect	Converts the complex number in polar to rectangular
rem	Fractional part
rms	Root mean square.
sgn	Sign extraction
sin	Sine
sinh	Hyperbolic sine
sqrt	Square root
tan	Tangent
tanh	Hyperbolic tangent
y0	Bessel function of the second kind (0^{th} order)
y1	Bessel function of the second kind (1^{st} order)

Selecting Solution Quantities to Plot

When you create a report of Modal or Terminal solution data, each trace in the report includes a quantity that is plotted along an axis. The quantity being plotted can be a value that was calculated by HFSS such as S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.The valid categories available depend on the type of quantity (real or complex) that is being plotted, the setup, the solution type, and the plot domain.

To select an S-parameter quantity to plot:

1. In the **Traces** dialog box, select one of the following categories:

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	Derived quantities from the original field solution.
S-parameter	S-parameters from the S-matrix.For designs which include a Frequency Selective Surface (FSS)-referenced radiation boundary, S11 and S21 represent the extracted reflection and transmission coefficients, respectively.
Y-parameter	Admittance matrix parameters computed from the S-parameters and port impedances.
Z-parameter	Impedance matrix parameters computed from the S-parameters and port impedances.
Power	
Gamma	Propagation constants for the S-parameters.
Port Z_o	Characteristic port impedances.
Voltage transform	
TDR Impedance.	TDR impedance for non-terminal problems.
VSWR	Voltage standing wave ratio, calculated from the equation $\frac{1+\left\|S_{ij}\right\|}{1-\left\|S_{ij}\right\|}$.
Group Delay	Quantity calculated as rate of change of the total phase shift with respect to angular frequency, $\frac{d(\phi)}{d(\omega)}$
Active S-parameter	Assume a_k is a complex number representing magnitude and phase for the k^{th} source. If there are n ports in the problem, for the m^{th} mode: $$ActiveS_{mm} = \sum_{k=1}^{n}\left[\frac{a_k}{a_m} \times S_{mk}\right]$$

Active Y-parameter

$$ActiveY_{mm} = \frac{1}{ActiveZ_{mm}}$$

Active Z-parameter

$$ActiveZ_{mm} = Z_0^m \times \left[\frac{1 + ActiveS_{mm}}{1 - ActiveS_{mm}}\right]$$

Active VSWR

$$ActiveVSWR = \frac{1 + mag(ActiveS_{11})}{1 - mag(ActiveS_{11})}$$

2. Select a quantity to plot from the **Quantity** list. The available quantities will depend upon the selected category and the setup of the design.

Selecting a Field Quantity to Plot

When plotting field quantities, the quantity can be a value that was calculated by HFSS such as the magnitude of S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or phase.

To select a field quantity to plot:

1. When you create the report, specify the Report Type as “Fields.”
2. In the **Traces** dialog box, select one of the following categories:

Variables	Intrinsic variables, such as frequency or phase, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	Derived quantities from the original field solution.
Calculator Expressions	Includes scalar and vector field quantities automatically calculated by HFSS, as well as derived field quantities that are defined by calculated expressions you set up in the Fields Calculator.

3. Select a quantity to plot from the **Quantity** list. The available quantities will depend upon the selected category and the setup of the design.

Selecting a Far-Field Quantity to Plot

When plotting far-field quantities, the quantity can be a value that was calculated by HFSS such as antenna gain, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

To select a far-field quantity to plot:

1. When you create the report, specify the Report Type as “Far Fields.”
2. In the **Traces** dialog box, select one of the following categories:

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	Derived quantities from the original field solution.
rE	The selected component of the radiated electric field, which is multiplied by the radial distance, *r*.
Normalized rE	The rE quantity divided by its category's maximum.
Normalized Near E	The Near E quanity divided by its category’s maximum.
Gain	Gain is four pi times the ratio of an antennaís radiation intensity in a given direction to the total power accepted by the antenna.
Normalized Gain	The Gain quantity divided by its category’s maximum.
Directivity	Directivity of the antenna.
Normalized Directivity	The Directivity divided by its category;s maximum.
Realized Gain	Realized gain is four pi times the ratio of an antennaís radiation intensity in a given direction to the total power incident upon the antenna port(s).
Normalized Realized Gain	The Realized Gain divided by its category’s maximum.
Axial Ratio	Axial ratio of the electric field.
Polarization Ratio	Polarization ratio of the electric field.
Antenna Params	HFSS-calculated quantities that include antenna directivity, beam area, radiated power, accepted power, radiation efficiency, max U, and array factor. For far-field setups, the decay factor for lossy materials is calculated as a constant for all far fields.
Normalized Antenna	The resultant plot is for: field quantity / (maximum field quantity value over the entire infinite sphere).
Normalized Bistatic RCS	The normalized radar cross-section, $\frac{\sigma}{\lambda_0^2}$ where λ_0 is the wavelength of free space. *For designs with Plane incident waves. (RCS is not supported for other types of incident waves).*

Radar Cross-Section (Bistatic RCS)

For designs with Plane Incident Waves. (RCS is not supported for other types of incident waves).

The radar cross-section (RCS) or echo area, σ, is measured in meters squared and represented for a bistatic arrangement (that is, when the transmitter and receiver are in different locations as shown in the linked figure). This is represented by

$$\sigma = \frac{4\pi r^2 \left|\boldsymbol{E}_{scat}\right|^2}{\left|\boldsymbol{E}_{inc}\right|^2}$$

where

- E_{scat} is the scattered E-field.
- E_{inc} is the incident E-field.

Complex (Bistatic) RCS

For designs with Plane Incident Waves. (RCS is not supported for other types of incident waves)

The equation for complex (bistatic) RCS is calculated as:

$$\sqrt{\sigma} = 2\sqrt{\pi} R \frac{E_{scat}}{\left|\boldsymbol{E}_{inc}\right|}$$

where

- E_{scat} is the scattered E-field.
- E_{inc} is the incident E-field.

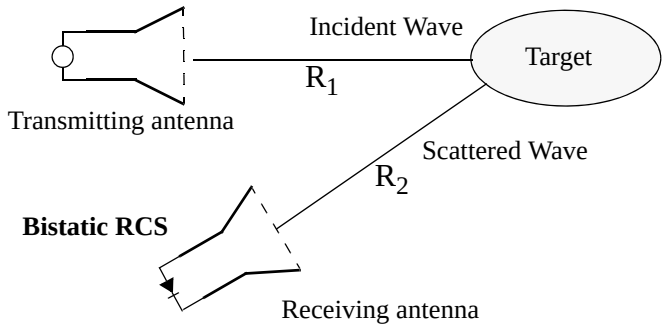
This form retains the phase information.

Monostatic RCS

For designs with Plane Incident waves. (RCS is not supported for other types of incident waves) A proper incident angle sweep should exist at the incident wave source setup before HFSS can plot Monstatic RCS.

The radar cross-section (RCS) or echo area when the transmitter and receiver are at the same location.

The following diagram shows the bistatic RCS concept, with separate transmitting and receiving antennas.



3. Depending on the **Category** you selected, you will need to specify the polarization of the electric field by selecting one of the following types of quantities from the **Quantity** list:

Total	The combined magnitude of the electric field components.
Phi	The phi component.
Theta	The theta component.
X	The x-component.
Y	The y-component.
Z	The z-component.
LHCP	The dominant component for a left-hand, circularly polarized field.
RHCP	The dominant component for a right-hand, circularly polarized field.
CircularLHCP	The polarization ratio for a predominantly left-hand, circularly polarized antenna.
CircularRHCP	The polarization ratio for a predominantly right-hand, circularly polarized antenna.
SphericalPhi	The polarization ratio for a predominantly ϕ-polarized antenna.
SphericalTheta	The polarization ratio for a predominantly θ-polarized antenna.
L3X	The dominant component for an x-polarized aperture using Ludwig's third definition of cross polarization.
L3Y	The dominant component for a y-polarized aperture using Ludwig's third definition of cross polarization.

Related Topics

Technical Notes: Antenna Parameters

Technical Notes: *Polarization of the Electric Field*

Technical Notes: *Spherical Cross-Sections*

Plotting Vertical Cross-Sections of Far Fields

When plotting far fields, a vertical cross-section plot results from holding phi fixed and sweeping theta through a range of values.

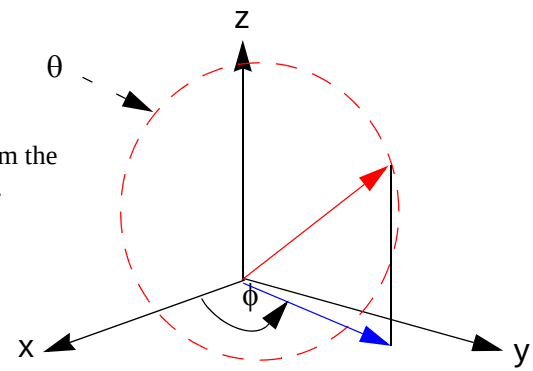
1. Click the **Sweeps** tab in the **Traces** dialog box.
2. Click the **phi** row in the table.

   To the right, all of the possible values for the phi variable are listed. The values are the result of the range of phi you specified during the infinite sphere's setup.
3. Clear the **All Values** option.
4. Select the fixed value that phi should take in the plot.

   HFSS will display values for the vertical cross-section at selected phi cuts of the problem region at a set of theta rotations.

The figure shown below demonstrates the orientation of the vertical cross-section when ϕ is the fixed variable:

θ values are an infinite radial distance away from the origin for far-field plots.



Plotting Horizontal Cross-Sections of Far Fields

When plotting far fields, a horizontal cross-section results from holding theta fixed and sweeping phi through a range of values.

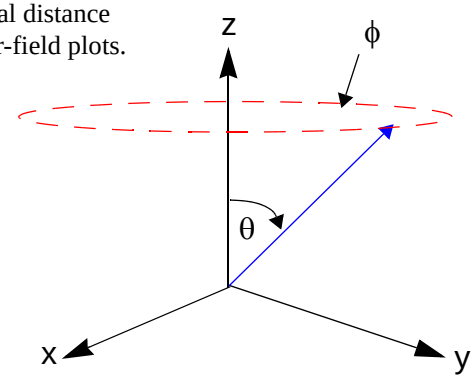
1. Click the **Sweeps** tab in the **Traces** dialog box.
2. Click the **theta** row in the table.

   To the right, all of the possible values for the theta variable are listed. The values are the result of the range of theta you specified during the infinite sphere's setup.
3. Clear the **All Values** option.
4. Select the fixed value that theta should take in the plot.

   HFSS will display values for the horizontal cross-section at selected theta cuts of the problem region at a set of phi rotations.

The figure shown below demonstrates the orientation of the sphere on which the field is computed when θ is the fixed variable:

ϕ values are an infinite radial distance away from the origin for far-field plots.



Selecting a Near-Field Quantity to Plot

When plotting near-field quantities, the quantity can be a value that was calculated by HFSS, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

To select a near-field quantity to plot:

1. When you create the report, specify the Report Type as "Near Fields."
2. In the **Traces** dialog box, select one of the following categories:

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	Derived quantities from the original field solution.
Near E	The radiated electric field in the near region.
Max Near Field Params	The maximum radiated electric field in the near region.
Near Normalized Antenna	The resultant plot is: field quantity / (maximum field quantity value over the entire infinite sphere).

3. If you selected the **Near E** category, specify the polarization of the electric field by selecting one of the following types of quantities from the **Quantity** list:

NearETotal	The combined magnitude of the electric field components.
NearEPhi	The phi component of the electric field.
NearETheta	The theta component of the electric field.
NearEX	The x-component of the electric field.
NearEY	The y-component of the electric field.

NearEZ	The z-component of the electric field.
NearELHCP	The dominant component for a left-hand, circularly polarized electric field.
NearERHCP	The dominant component for a right-hand, circularly polarized electric field.
NearECircularLHCP	The polarization ratio for a predominantly left-hand, circularly polarized antenna.
NearECircularRHCP	The polarization ratio for a predominantly right-hand, circularly polarized antenna.
NearEL3X	The dominant component for an x-polarized aperture using Ludwig's third definition of cross polarization.
NearEL3Y	The dominant component for a y-polarized aperture using Ludwig's third definition of cross polarization.

If a Near-field plot takes a long time to plot, be sure to perform **File>Save** when the plot is displayed. This saves the calculated data and permits fast display on subsequent viewings of the plot.

Related Topics

Technical Notes: Polarization of the Electric Field

Selecting an Emission Test Quantity to Plot

1. When you create the report, select Emission Test.
2. In the **Traces** dialog box, select one of the following categories and apply an appropriate Quantity.

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	Derived quantities from the original field solution.
Sphere	A sphere of 1, 3, 10, or 30 meters, or of the same dimensions and PRBS Simple or PRBS exact (where PRBS is pseudorandom binary [bit] sequence).
Cylinder	A cylinder of 3 or 10 meters, or of the same dimensions and PRBS Simple or PRBS exact.

3. Select a **Function** for the quantity from the function list.
4. For Emission Test, the **Traces** dialog also contains a button for specifying the digital signal options. The default values are a rise time of 0 seconds, and a hold time of 1 second. To specify other values, click **Digital Signal Options**.

   This displays the **Digital Signal Options** dialog. It contains fields for the rise time, hold time, and buttons.

5. **OK** the specified values or Cancel, Use Defaults, or Save As Default as appropriate.

Plotting Imported Solution Data

1. In the **Solution** pull-down list in the **Traces** dialog box, click the imported data you want to plot.
2. If the imported data is in table format, click **Table Data** in the **Category** list.
3. Follow the procedure for creating a report.

Specifying Output Variables

The **Output Variables** window contains three sections:

- **Output Variables** section, where you can specify the name and expression for a new output variable.
- **Calculation section**, where you can insert quantities into the Expression area of the **Output Variables** section.
- **Function section**, where you can insert completed expressions into the Expression area of the **Output Variables** section.

Adding a New Output Variable

To add an output variable:

1. Click **HFSS>Results>Output Variables** or, in the Project tree, right-click on **Results** and select **Output Variables** from the short-cut menu.

   The **Output Variables** window appears. Variables defined using the **HFSS>Results>Output Variables** command appear in the list at the top of the window.
2. In the **Output Variables** section, enter a name for the new variable in the **Name** box.
3. Click in the **Expression** area.
4. To enter an expression, do one or both of the following:
   a. Type part or all of the expression directly in the **Expression** area.
   b. Insert part or all of the expression using the options in the **Calculation** and **Function** sections.
5. Click **Add** to add the new variable to the list.
6. Repeat steps 2 through 5 to add additional variables.
7. When you are finished adding output variables, click **Done** to close the **Output Variables** window.

Building an Expression Using Existing Quantities

When you are entering an expression for a new output variable, you can insert part or all of the expression using the options in the **Calculation** and **Function** sections of the **Output Variables** window.

To add an input variable by inserting part or all of the expression:

1. Click **HFSS>Results>Output Variables** or, in the Project Tree, right-click on **Results** and select **Output Variables** from the short-cut menu.

   The **Output Variables** window appears.
2. In the **Output Variables** section, enter a name for the new variable in the **Name** box.
3. Click in the **Expression** area.
4. To insert a quantity:
   a. From the **Design** pull-down list, select the design from which you want to select the quan-

tity.

b. From the **Report Type** pull-down list, select the type of report from which you want to select the quantity.

c. From the **Solution** pull-down list, select the solution from which you want to select the quantity.

d. From the **Category** list, select the type of quantity you want to enter.

e. From the **Quantity** list, select the quantity.

f. From the **Function** list, select a ready-made function (this option is the same as inserting the function from the Function section).

g. If applicable, from the **Domain** list, select the solution domain.

h. Click **Insert Quantity Into Expression**.

The selected quantity is entered into the **Expression** area of the **Output Variables** section.

5. To insert a function:

a. In the **Function** section, select a ready-made function from the pull-down list.

b. Click **Insert Function**.

The function appears in the **Expression** area of the **Output Variables** section.

6. When you are finished defining the variable in the **Expression** area, click **Add** to add the new variable to the list.
7. Repeat steps 2 through 6 to add additional variables.
8. When you are finished adding output variables, click **Done** to close the **Output Variables** window.

Note Remember the evaluated value of an expression is always interpreted as in SI units. However, when a quantity is plotted in a report, you have the option to plot values in units other than SI. For example, the expression “1+ang_deg(S11)” represents an ‘angle''quantity evaluated in radians) though plotted in degrees units. To represent an angle quanity in degrees, you would specify units as “1 deg + ang_deg(S11)”.

Deleting Output Variables

To delete output variables:

1. Remove all references to the output variable in the project.
2. Save the project to erase the command history.
3. Click **HFSS>Results>Output Variables** or, in the Project Tree, right-click on **Results** and select **Output Variables** from the short-cut menu to open the **Output Variables** dialog.
4. Select the variable and click the **Delete** button.
5. Click **OK** to close the dialog.

16

Technical Notes

The simulation technique used to calculate the full 3D electromagnetic field inside a structure is based on the finite element method. Although its implementation is largely transparent, a general understanding of the method is useful in making the most effective use of HFSS.

The HFSS Technical Notes provide an overview of the finite element method and its implementation in HFSS. They also describe how modal S-parameters are computed from the simulated electric and magnetic fields and how they can be converted to “nodal” or “voltage” based pseudo-S-parameters used in circuit theory.

Information is included on the following:

- The Finite Element Method
- The HFSS Solution Process
- S-Parameters
- Radiated Fields
- Geometric Objects
- Boundaries
- Excitations
- Materials
- Parametric Analysis
- Optimization Analysis
- Sensitivity Analysis
- Tuning Analysis

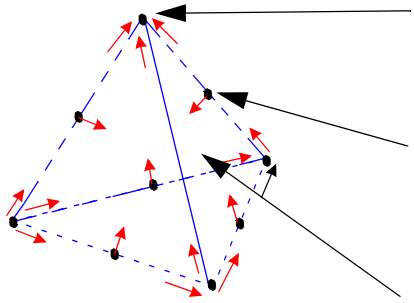
The Finite Element Method

In order to generate an electromagnetic field solution, HFSS employs the finite element method. In general, the finite element method divides the full problem space into thousands of smaller regions and represents the field in each sub-region (element) with a local function.

In HFSS, the geometric model is automatically divided into a large number of tetrahedra, where a single tetrahedron is a four-sided pyramid. This collection of tetrahedra is referred to as the finite element mesh.

Representation of a Field Quantity

The value of a vector field quantity (such as the H-field or E-field) at points inside each tetrahedron is interpolated from the vertices of the tetrahedron. At each vertex, HFSS stores the components of the field that are tangential to the three edges of the tetrahedron. In addition, HFSS can store the component of the vector field at the midpoint of selected edges that is tangential to a face and normal to the edge (as shown below). The field inside each tetrahedron is interpolated from these nodal values.



The components of a field that are tangential to the edges of an element are explicitly stored at the vertices.

The component of a field that is tangential to the face of an element and normal to an edge is explicitly stored at the midpoint of selected edges.

The value of a vector field at an interior point is interpolated from the nodal values.

By representing field quantities in this way, the system can transform Maxwell's equations into matrix equations that are solved using traditional numerical methods.

Basis Functions

Various interpolation schemes, or basis functions, can be used to interpolate field values from nodal values.

- A first order tangential element basis function interpolates field values from both nodal values at vertices and on edges.

  First order tangential elements have 20 unknowns per tetrahedron.

- A zero order basis function makes use of nodal values at vertices only — and therefore assumes that the field varies linearly inside each tetrahedron.

  Zero order tangential elements have six unknowns per tetrahedron.

Size of Mesh Vs. Accuracy

There is a trade-off among the size of the mesh, the desired level of accuracy, and the amount of available computing resources.

The accuracy of the solution depends on the size of each of the individual elements (tetrahedra). Generally speaking, solutions based on meshes using thousands of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each element must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.

However, generating a field solution involves inverting a matrix with approximately as many elements as there are tetrahedra nodes. For meshes with a large number of elements, such an inversion requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power.

To produce the optimal mesh, HFSS uses an iterative process, called an adaptive analysis, in which the mesh is automatically refined in critical regions. First, it generates a solution based on a coarse initial mesh. Then, it refines the mesh in areas of high error density and generates a new solution. When selected parameters converge to within a desired limit, HFSS breaks out of the loop.

The HFSS Solution Process

To calculate the S-matrix associated with a structure with ports, HFSS does the following:

- Divides the structure into a finite element mesh.
- Computes the modes on each port of the structure that are supported by a transmission line having the same cross-section as the port.
- Computes the full electromagnetic field pattern inside the structure, assuming that one mode is excited at a time.
- Computes the generalized S-matrix from the amount of reflection and transmission that occurs.

The resulting S-matrix allows the magnitude of transmitted and reflected signals to be computed directly from a given set of input signals, reducing the full 3D electromagnetic behavior of a structure to a set of high frequency circuit parameters.

The Mesh Generation Process

Following is the general mesh generation process:

1. HFSS generates an initial mesh, which includes surface approximation settings.
2. If lambda refinement was requested, HFSS refines the initial mesh based on the material-dependent wavelength.
3. Any mesh operations that were defined are used to refine the mesh.
4. If ports were defined, HFSS iteratively refines the 2D mesh at the ports.
5. Using the resulting mesh, HFSS computes the electromagnetic fields that exist inside the structure when it is excited at the solution frequency.
6. If you are performing an adaptive analysis, HFSS uses the current finite element solution to estimate the regions of the problem domain where the exact solution has strong error. Tetrahedra in these regions are refined.
7. HFSS generates another solution using the refined mesh.
8. HFSS recomputes the error, and the iterative process (solve — error analysis — adaptive refinement) repeats until the convergence criteria are satisfied or the maximum number of adaptive passes is completed.
9. If a frequency sweep is being performed, then HFSS solves the problem at the other frequency points without further refining the mesh. An adaptive solution is performed only at the specified solution frequency.

Note HFSS does not generate an initial mesh each time it starts the solution process. The initial mesh is generated only if a current mesh is unavailable.

Related Topics

Reverting to the Initial Mesh

Seeding the Mesh

Guidelines for Seeding the Mesh

Length-Based Mesh Refinement

Skin Depth-Based Mesh Refinement

Surface Approximation Settings

Guidelines for Modifying Surface Approximation Settings

Meshing Region Vs. Problem Region

Mesh Refinement on Ports

Model Resolution

Seeding the Mesh

In HFSS, mesh operations are optional mesh refinement settings that enable you to provide HFSS with engineering guidance based on your knowledge of the parts of the model geometry that are critical to the structure's electromagnetic performance. Providing such guidance to HFSS prior to beginning the adaptive analysis process can reduce (sometimes extensively) the number of passes necessary to converge upon a field solution as well as the final number of tetrahedra in the mesh for that solution. Although adaptive analysis convergence targets areas where field behavior is found, refining the mesh using more than the standard criteria, such as material characteristics, can result in finding areas of critical field behavior as soon as the first few passes are solved.

The technique of guiding HFSS's mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Mesh Operations** commands on the **HFSS** menu.

You can instruct HFSS to refine the length of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can instruct HFSS to refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement.) These types of mesh operations can be defined at any time. If you apply them before the adaptive solution process, they are used to refine the initial mesh after it has been generated. You can also choose to apply mesh operations without generating a solution, in which case the mesh operations are applied to the current mesh.

In a few circumstances, you may also want to define a mesh operation that modifies HFSS's surface approximation settings for one or more faces. Surface approximation settings are only applied to the initial mesh.

Related Topics

Defining Mesh Operations

Technical Notes: The Mesh Generation Process

Guidelines for Seeding the Mesh

While seeding the mesh is not required, it is useful in the following conditions:

- Seeding the mesh inside a volume in the model geometry where regions of strong electric or magnetic fields (with strong capacitive or inductive loading) are expected. Examples include a capacitively loaded gap in a resonant structure, sharp waveguide angles or corners, or gaps between multi-coupled lines in filter structures.
- Seeding the mesh on every face of higher aspect ratio boundaries, such as long PCB traces or on the surfaces of long wires. Spacing the mesh points roughly equal to the trace width

of the wire diameter enables you to more accurately capture the behavior of the high-aspect structure from the first adaptive pass.

Related Topics

Defining Mesh Operations

Length-Based Mesh Refinement

When you request length-based mesh refinement, you instruct HFSS to refine the length of tetrahedral elements until they are below a specified value. The length of a tetrahedron is defined as the length of its longest edge.

You can specify the maximum length of tetrahedra on faces or inside of objects. You can also specify the maximum number of elements that are added during the refinement. When the initial mesh has been generated, the refinement criteria you specified will be used to refine the initial mesh.

Related Topics

Assigning Length-Based Mesh Refinement on Object Faces

Assigning Length-Based Mesh Refinement Inside Objects

Skin Depth-Based Mesh Refinement

When you request skin depth-based mesh refinement, you instruct HFSS to refine the surface triangle length of all tetrahedral elements on a face to within a specified value. A layered mesh is created based on the surface mesh. The layers are graded based on the skin depth and number of layers you specify.

During skin depth-based mesh refinement, HFSS creates a series of layers that are planes parallel to the object face, and that are spaced within the specified skin depth. For each point on the surface of the face, a series of points (P0, P1, P2, ..., Pn) are added to the mesh, where n is the number of layers. P0 is the point on the surface and the distance from P0 to Pn is the skin depth. The points are spaced in a non-uniform manner, with the distance between them decreasing in a geometric progression, as you move from Pn to P0.

For example, if

Skin Depth:	12 mm
Number of Layers of Elements:	4

then

Distance [P0,P1]:	0.8 mm.
Distance [P1,P2]:	1.6 mm.
Distance [P2,P3]:	3.2 mm.
Distance [P3,P4]:	6.4 mm.
Distance [P0,P4]:	0.8 + 1.6 + 3.2 + 6.4 = 12 mm

The skin depth-based refinement first satisfies the surface triangle edge length criterion, then introduces the series of points to each additional layer. If a limit has been placed on mesh growth, one of the following happens:

- The limit is set high enough to complete the skin depth refinement.
- The limit is set high enough to satisfy the surface triangle edge length criterion, but not high enough to complete the depth seeding.
- The limit is not set high enough to satisfy even the surface triangle edge length criterion.

Because refining by skin depth can add many seeding points, you should first refine the surface of the object using length-based mesh refinement to obtain an accurate count of the number of points HFSS will add when refining by skin depth. This allows you to reach the surface edge length criterion and approximate the number of elements in the mesh and the number of points on the surfaces before proceeding to skin depth seeding.

The refinement criteria you specified are used to refine the current mesh.

Related Topics

Assigning Skin Depth-Based Mesh Refinement on Object Faces

Surface Approximation Settings

Object surfaces in HFSS may be planar, cylindrical or conical, toroidal, spherical, or splines. The original model surfaces are called *true surfaces*. To create a finite element mesh, HFSS first divides all true surfaces into triangles. These triangulated surfaces are called faceted surfaces because a series of straight line segments represents each curved or planar surface.

For planar surfaces, the triangles lie exactly on the model faces; there is no difference in the location or the normal of the true surface and the meshed surface. When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.

The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees.

The *aspect ratio* of triangles used in planar surfaces is based on the ratio of circumscribed radius to the in-radius of the triangle. It is unity for an equilateral triangle and approaches infinity as the triangle becomes thinner.

You can modify the surface deviation, the maximum permitted normal deviation, and the maximum aspect ratio of triangles settings on one or more faces at a time in the **Surface Approximation** dialog box. (Click **HFSS>Mesh Operations>Assign>Surface Approximation**.)

The surface approximation settings are applied to the initial mesh.

Note For the initial mesh, all the vertices of the triangles lie on the true surfaces. During adaptive meshing, the vertices are added to the meshed surfaces, not to the true surfaces.

Related Topics

Modifying Surface Approximation Settings

Technical Notes: Guidelines for Modifying Surface Approximation Settings

Technical Notes: The Mesh Generation Process

Guidelines for Modifying Surface Approximation Settings

If you intend to modify the surface approximation settings for an object face or faces, keep the following guidelines in mind:

- When necessary, override the default surface approximation settings to represent curved surfaces more accurately. More accurate representation will increase the mesh size and consume more CPU time and memory. The default settings are adequate for most circumstances.
- If you want to obtain a faster solution by using a cruder representation of curved surfaces, set the coarser setting for the whole object, not just a single face.
- It is difficult for HFSS to satisfy aspect ratio demands if the aspect ratio value is set close to 1 because an arbitrary shape cannot be filled with only equilateral triangles. Therefore, setting the aspect ratio to 1 can lead to unreasonably large meshes. HFSS limits the aspect ratio to 4 for planar objects and 1.2 for curved objects.

Related Topics

Modifying Surface Approximation Settings

Technical Notes: Surface Approximation Settings

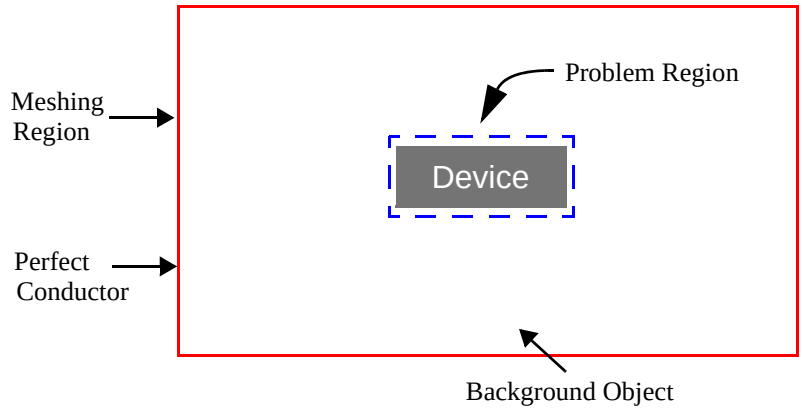
Meshing Region Vs. Problem Region

HFSS distinguishes between the problem region and the meshing region. The *problem region* is the region in which the solution is generated and the mesh is refined. The *meshing region*, which includes the problem region, is the area in which an initial mesh is generated. After an initial mesh is generated, the mesh is refined only in the problem region.

The problem region encompasses an area that is just large enough to include the entire design, but no larger. HFSS automatically defines the problem region during the solution process. If you are interested in effects outside of the structure, such as radiated effects, then you can create a virtual object to expand the size of the problem region to include these areas.

The meshing region, like the problem region, is a box that completely encloses the structure. However the meshing region must be at least 10 times larger than the model. The part of the meshing region not occupied by objects is considered to be the *background* object. The background extends to the boundaries of the meshing region and fills in any voids not occupied by objects. Since the background object is defined as a perfect conductor, no solution is generated inside the background even though an initial mesh is generated for it. HFSS automatically defines the meshing region during the solution process.

The problem region and the meshing region are illustrated below.



Model Resolution

Model Resolution is a setting that determines the smallest details of a model that the mesher should capture and represent in the mesh.

Many times the analysis starts with the geometry already drawn in a different tool for different purpose. Some tools are designed for manufacturing and the resulting models contain lots of extra details not needed for electromagnetic analysis. If the user removes such details in the original tool the results will be better. But if the user does not have access to the original drawing tool or redrawing the model without these details is not possible, **Model Resolution** is another way to remove the details from analysis.

When the user sets the model resolution length to be L, the mesher will start with a surface representation of the model accurate to the modeler's tolerance limit. Then it will progressively remove edges, move points, merge points etc.. within the allowable model resolution limit and simplify the surface mesh. During this process, tiny fillets, rounds, and chamfer protrusions are removed.

Other common model translation anomalies are also handled using **Model Resolution**. For example, some geometry engines will blindly export all of the surfaces as splines. When a user imports such a model for analysis, it would result in very large number of triangles. If the surface can be represented by a smaller set of triangles using Model Resolution, this procedure would reduce the number of triangles in the surface mesh.

The user can start with a model resolution length around 0.1*wavelength. If the model resolution length chosen by the user is too large, the mesher will detect it and report it as an error. The model resolution length is specified in the user units of the modeler. It can be set on selected bodies only. The default value is 100* the tolerance limit of the ACIS modeler.

Related Topics

Specifying Model Resolution

Port Solutions

The excitation field pattern at each port must be calculated before the full 3D electromagnetic field inside a structure can be calculated. HFSS calculates the natural field patterns (or modes) that can exist inside a transmission structure with the same cross-section as the port. The resulting 2D field patterns serve as boundary conditions for the full 3D problem.

Excitation Fields

HFSS assumes that each port is connected to a uniform waveguide that has the same cross-section as the port. Therefore, the excitation field is the field associated with traveling waves propagating along the waveguide to which the port is connected,

$$\boldsymbol{E}(x, y, z, t) = \Re[\boldsymbol{E}(x, y)e^{j\omega t - \gamma z}].$$

where

- $\Re$ is the real part of a complex number or function.
- **E**(x,y) is a phasor field quantity.
- $\gamma=\alpha + j\beta$ is the complex propagation constant, where
  - α is the attenuation constant of the wave.
  - β is the propagation constant associated with the wave that determines, at a given time *t*, how the phase angle varies with z.
- ω is angular frequency, $2\pi f$.
- *j* is the imaginary unit, $\sqrt{-1}$.

In this context, the x- and y-axes are assumed to lie in the cross-section of the port; the z-axis lies along the direction of propagation.

Wave Equation

The field pattern of a traveling wave inside a waveguide can be determined by solving Maxwell's equations. The following equation that the Wave module solves is derived directly from Maxwell's equation

$$\nabla\times\left(\frac{1}{\mu_r}\nabla\times\boldsymbol{E}(x, y)\right) - k_0{}^2\varepsilon_r\boldsymbol{E}(x, y) = 0 \quad ,$$

where

- **E**(x,y) is a phasor representing an oscillating electric field.
- k_0 is the free-space wave number,

  $\omega\sqrt{\mu_0\varepsilon_0} = \omega/c$.
- ω is the angular frequency, $2\pi f$.
- ε_0 is the permittivity of free space, $1/(c^2\mu_0)$
- $\mu_r(x, y)$ is the complex relative permeability.
- $\varepsilon_r(x, y)$ is the complex relative permittivity.

When the Wave module solves this equation, it obtains an excitation field pattern in the form of a phasor solution, **E**(x,y). It also solves independently for **H**(x,y) using the corresponding wave equation in **H**. These phasor solutions are independent of *z* and *t*; only after being multiplied by $e^{-\gamma z}$ do they become traveling waves.

Also note that the excitation field pattern computed by the Wave module is valid only at a given frequency. A different excitation field pattern is computed for each frequency point of interest.

Mesh Refinement on Ports

The Wave module treats its computation of the excitation field pattern as a 2D finite element problem. The mesh associated with each port is simply the 2D mesh of triangles corresponding to the face of tetrahedra that lie on the port surface. The Wave module performs an iterative refinement of this 2D mesh without calling the Meshmaker.

The refinement procedure is as follows:

1. Using the triangular mesh formed by the tetrahedra faces of the initial mesh, Wave calculates solutions for both the magnetic field, **H**, and the electric field, **E**.
2. To determine if the 2D solution is accurate, **wave** uses the following equations:

$$\nabla \times \boldsymbol{H} = \sigma \boldsymbol{E} + j\omega\varepsilon \boldsymbol{E}$$

$$\nabla \times \boldsymbol{E} = -j\omega\mu \boldsymbol{H}$$

   where **H**(x,y) and **E**(x,y) are phasors.
3. Wave first calculates **E** and **H** independently using the appropriate wave equations. Next, it computes $\nabla \times \boldsymbol{H}$ and compares the results to the solved **E**. It then computes $\nabla \times \boldsymbol{E}$ and compares the results to the solved **H**.
4. If the reciprocal comparison falls within an acceptable tolerance, the solution is accepted. Otherwise, the 2D mesh on the port face is refined and Wave performs another iteration.
5. Any mesh points that have been added to the face of a port are read out to the existing mesh files. These points are incorporated into the full 3D mesh the next time the Meshmaker is called.

For a detailed understanding of the theory implemented by the Wave module, refer to the following:

Jin-Fa Lee, Din-Kow Sun, and Zoltan J. Cendes, “Full-Wave Analysis of Dielectric Waveguides Using Tangential Vector Finite Elements,” *IEEE Transactions on Microwave Theory and Techniques*, vol. 39, No 8, August 1991.

Related Topics

Technical Notes: The Mesh Generation Process

Modes

For a waveguide or transmission line with a given cross-section, there is a series of basic field patterns, or modes, that satisfy Maxwell’s equations at a specific frequency. Any linear combination of these modes can exist in the waveguide. By default, HFSS computes only the dominant mode field pattern.

Mode Conversion

In some cases it is necessary to include the effects of higher order modes because the structure acts as a mode converter. For example, if the mode 1 (dominant) field at one port is converted (as it passes through a structure) to a mode 2 field pattern at another, then it is necessary to obtain the S-parameters for the mode 2 field.

Modes, Reflections, and Propagation

It is also possible for a 3D field solution generated by an excitation signal of one specific mode to contain reflections of higher order modes which arise due to discontinuities in the structure. If these higher order modes are reflected back to the excitation port or transmitted to another port, the S-parameters associated with these modes should be calculated.

If the higher order mode decays before reaching any port — either because of attenuation from losses or because it is a non-propagating evanescent mode — there is no need to obtain the S-parameters for that mode. Therefore, one way to avoid the need for computing the S-parameters for a higher order mode is to include a length of waveguide in the model that is long enough for the higher order mode to decay in. For example, if the mode 2 wave associated with a certain port decays to near zero in 0.5 mm, then the “constant cross-section” portion of the geometric model leading up to the port should be at least 0.5 mm long. Otherwise, for accurate S-parameters, the mode 2 S-parameters must be included in the S-matrix. The length of the constant cross-section segment that has to be included in the model depends on the value of the mode’s attenuation constant, α.

Modal Field Patterns and Frequency

The field patterns associated with each mode generally vary with frequency. However, the propagation constants and impedances always vary with frequency. When performing frequency sweeps, be aware that as the frequency increases, the likelihood of higher order modes being propagating modes also increases.

Multiple Ports on the Same Face

Visualize a port face on a microstrip that contains two conducting strips side by side as two separate ports. If the two ports are defined as being separate, the system simulates the case in which the two ports are connected to uncoupled transmission structures. It is as if a conductive wall separates the excitation waves. However, in actuality, there will be electromagnetic coupling between the two strips.

To model this coupling accurately, analyze the two ports as a single port with multiple modes. In general, if there are N disconnected conductors in the port cross-section, at least N - 1 modes are required for an accurate solution. For example, if the port consists of two adjacent microstrip lines surrounded by a conducting enclosure, N = 3; therefore at least two modes should be defined on the port. Assign an equal number of terminals as modes. Refer to Defining Terminals for more information.

If the multi-conductor port plane is near discontinuities within the 3D model, additional modes beyond N - 1 may be necessary. However, if you define terminals on a multi-conductor port, the presence of non-quasi transverse electromagnetic (TEM) modes will adversely affect the

entries of any computed terminal matrices. Therefore, rather than increase the number of modes beyond the required *N* - 1, extend the port outward until any higher-order modes have sufficient attenuation to be omitted from consideration.

Port Field Accuracy

Generally, the default **Port Field Accuracy** value, specified under the **Ports** tab of the **Solution Setup** dialog box, is adequate. You may want improved port accuracy under the following conditions:

- You are interested primarily in the port impedances. Port impedances are computed as part of the port solution.
- You need to lower the noise floor to catch S-parameters that are expected to be in the –70 dB range.

While HFSS uses the **Port Field Accuracy** value each time you request a ports-only solution, it only uses this value for the first full-field solution. This happens because a set of port solutions is computed at the beginning of the field solution process and then that set is used for all subsequent field solutions. Therefore, to specify a new port field accuracy for a field solution, add another solution setup and generate a new solution.

Refining the mesh at the ports causes HFSS to refine the mesh for the entire structure as well. This occurs because it uses the port field solutions as boundary conditions when computing the full 3D solution. Therefore, specifying too small a port field accuracy can result in an unnecessarily complex finite element mesh.

Saving Field Solutions

When specifying the frequency points to be solved during a sweep, you can specify whether you want to save the 3D field solutions associated with all port modes at each frequency. Because each additional field solution — those associated with higher order modes — increases the amount of required disk space by several megabytes, by default HFSS does not save the data for higher order modes unless you specifically request it do so. If you do not save the field solution, the associated mode will not be available as a source stimulation during post processing.

The Adaptive Analysis Process

An adaptive analysis is a solution process in which the mesh is refined iteratively in regions where the error is high, which increases the solution's precision. You set the criteria that control mesh refinement during an adaptive field solution. Many problems can be solved using only adaptive refinement.

Following is the general process followed during an adaptive analysis:

1. HFSS generates an initial mesh.
2. Using the initial mesh, HFSS computes the electromagnetic fields that exist inside the structure when it is excited at the solution frequency. (If you are running a frequency sweep, an adaptive solution is performed only at the specified solution frequency.)
3. Based on the current finite element solution, HFSS estimates the regions of the problem

domain where the exact solution has strong error. Tetrahedra in these regions are refined.

4. HFSS generates another solution using the refined mesh.
5. HFSS recomputes the error, and the iterative process (solve — error analysis — refine) repeats until the convergence criteria are satisfied or the requested number of adaptive passes is completed.
6. If a frequency sweep is being performed, HFSS then solves the problem at the other frequency points without further refining the mesh.

Maximum Delta S

For designs with ports.

The delta S is the change in the magnitude of the S-parameters between two consecutive passes. If the magnitude and phase of all S-parameters change by an amount less than the **Maximum Delta S Per Pass** value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.

For example, if you specify **0.1** as the **Maximum Delta S Per Pass**, HFSS continues to refine the mesh until the number of requested passes is completed or until the magnitude of the complex delta of all S-parameters changes by less than **0.1**.

The maximum delta S is defined as

$$Max_{ij}\left[mag\left(S_{ij}^{N} - S_{ij}^{N-1}\right)\right]$$

where:

- i and j cover all matrix entries.
- N represents the pass number.

Note Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been de-embedded and renormalized.

Related Topics

Viewing the Maximum Magnitude of Delta S Between Passes

Maximum Delta E

For designs with voltage sources, current sources, or incident waves.
Not applicable to designs with ports.

The delta E is the difference in the relative energy error from one adaptive solution to the next. It is a measure of the stability of the computed field values from pass to pass. As the solution converges, delta E approaches zero.

The **Maximum Delta E Per Pass** value is a stopping criterion for the adaptive solution. If the delta E falls below this value, the adaptive analysis stops. Otherwise, it continues until the convergence criteria are reached.

The data represents the delta E for all tetrahedra.

Percent of Tetrahedra Refined Per Pass

The value you set for **Percent Refinement Per Pass** determines how many tetrahedra are added at each iteration of the adaptive refinement process. For instance, entering **10** causes the mesh to increase approximately 10 percent each pass. The tetrahedra with the highest error will be refined. If your mesh consisted of 1000 elements, the tetrahedra would be refined so that 100 new elements are added to the mesh. Generally, you can accept the default value.

Magnitude Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

For each element in the S-matrix, the magnitude margin is the difference between the S-parameter delta magnitude and the target delta magnitude, which was specified in the **Matrix Convergence** dialog box. The magnitude margin reported under the **Convergence** tab is the maximum of these values over the entire matrix. The magnitude margin is defined as

$$Max_{ij}\left[\left|magS_{ij}^{N} - magS_{ij}^{N-1}\right| - magM_{ij}\right]$$

where M_{ij} is the matrix convergence entry. It indicates the solution's proximity to the target delta magnitude. If the solution has converged within the target delta magnitude, a value of zero will be reported for the pass.

Phase Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

For each element in the matrix, the phase margin is the difference between the S-parameter delta phase and the target delta phase, which was specified in the **Matrix Convergence** dialog box. The phase margin reported under the **Convergence** tab is the maximum of these values over the entire matrix. The phase margin is defined as

$$Max_{ij}\left[\left|phaseS_{ij}^{N} - phaseS_{ij}^{N-1}\right| - phaseM_{ij}\right]$$

where M_{ij} is the matrix convergence entry. The phase margin indicates the solution's proximity to the target delta phase. If the solution has converged within the target delta phase, a value of zero will be reported for the pass.

Maximum Delta Frequency

For Eigenmode solutions.

At any time during the solution process, you can view the percent difference in the resonant frequencies from one adaptive solution to the next, or the maximum delta frequency. This is a measure of the stability of the computed frequencies from pass to pass and is available only two or more adaptive passes are completed.

For lossless problems, the maximum delta frequency is the largest percent change in the real part of the frequency for any of the calculated modes. For lossy problems, the maximum delta frequency is the greater of two quantities: the largest percent change in the real part of the frequency over all the modes, and the largest percent change in the imaginary part of the frequency.

Max Delta (Mag S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

The Max delta(Mag S) is the maximum difference of S-Matrix magnititues between two consecutive passes. If the difference in magnitudes of the S matrices change by an amount less than the **Maximum Delta Mag S** value from one pass to the next, this satifies the part of the convergence criteria.

$$Max\left(MagS_{ij}^{N} - MagS_{ij}^{N-1}\right)$$

Max Delta (Phase S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

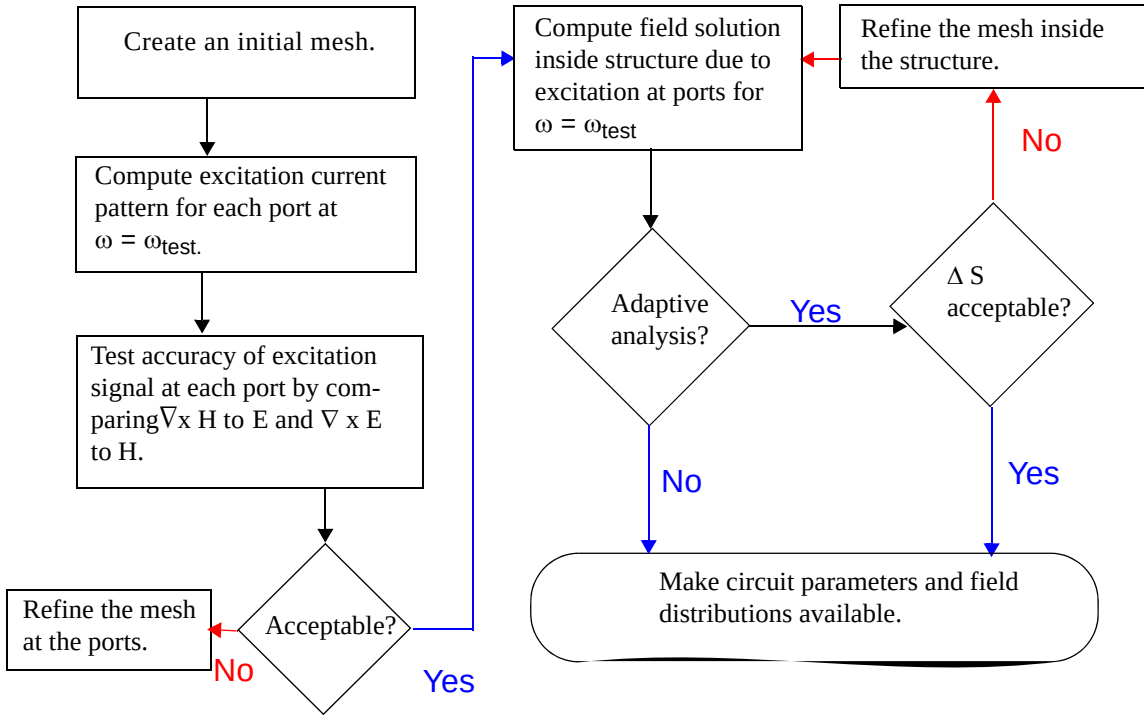
The Max delta(Phase S) is the maximum difference of S-Matrix phase between two consecutive passes. If the difference in phase of the S matrices change by an amount less than the **Maximum Delta Phase S** value from one pass to the next, this satifies this part of the convergence criteria.

$$Max(PhaseS_{n} - PhaseS_{n-1})$$

Single Frequency Solution

A single frequency solution generates an adaptive or non-adaptive solution at a single frequency, the solution frequency specified in the **Solution Setup** dialog box, and is often the first step in performing a frequency sweep. An adaptive solution is one in which a finite element mesh is created

and automatically refined in the areas of highest error — increasing the accuracy of succeeding adaptive solutions. The procedure for performing a single frequency solution is shown below.



Frequency Sweeps

Perform a frequency sweep when you want to generate a solution across a range of frequencies. You may choose one of the following sweep types:

Fast	Generates a unique full-field solution for each division within a frequency range. Best for models that will abruptly resonate or change operation in the frequency band. A Fast sweep will obtain an accurate representation of the behavior near the resonance.
Discrete	Generates field solutions at specific frequency points in a frequency range. Best when only a few frequency points are necessary to accurately represent the results in a frequency range.
Interpolating	Estimates a solution for an entire frequency range. Best when the frequency range is wide and the frequency response is smooth, or if the memory requirements of a Fast sweep exceed your resources.

Fast Frequency Sweeps

A Fast sweep generates a unique full-field solution for each division within a frequency range. Choose a Fast sweep if the model will abruptly resonate or change operation in the frequency band. A Fast sweep will obtain an accurate representation of the behavior near the resonance.

HFSS uses the center frequency of the frequency range to select an appropriate eigenvalue problem with which to generate a solution for the entire Fast sweep. It then uses an Adaptive Lanczos-Padé Sweep (ALPS)- based solver to extrapolate the field solution across the requested frequency range from the center frequency field solution.

HFSS uses the solution frequency as the center frequency if it is within the frequency range (greater than the Start frequency and less than the Stop frequency). Otherwise the middle of the frequency range is used as the center frequency.

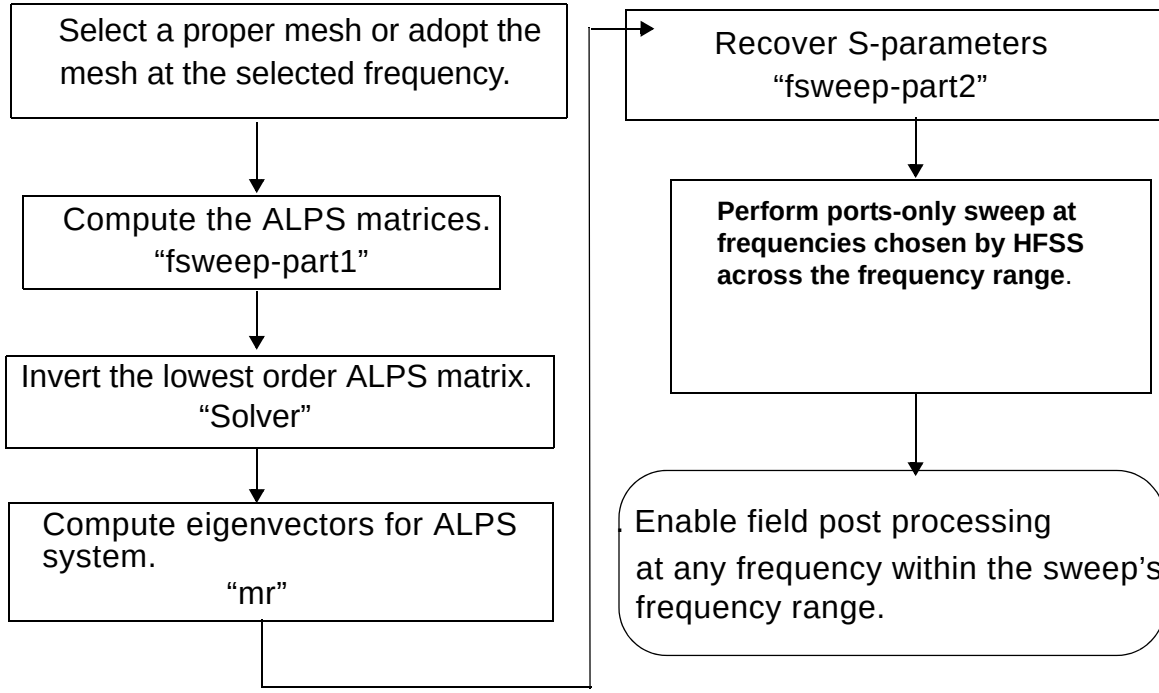
Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or, if you did not request an adaptive solution, the initial mesh generated for the problem. It uses this mesh without further refinement. Also, the field solution at the center frequency is the most accurate. Depending upon the desired level of accuracy you require throughout the frequency range, you may wish to perform additional Fast sweeps at other center frequencies.

The full-field solution is saved only at the center frequency, while the S-parameters are saved for every frequency point; however, the Fast sweep allows the you to post process fields for any frequency entries to the sweep range.

The time required for a Fast sweep may be significantly greater than the time required for a single frequency solution.

Note When performing a Fast sweep, no port mode may cross cut-off in the frequency range. If this occurs, an error message appears listing the port and mode violating this condition.

The procedure for a Fast frequency sweep is shown below.



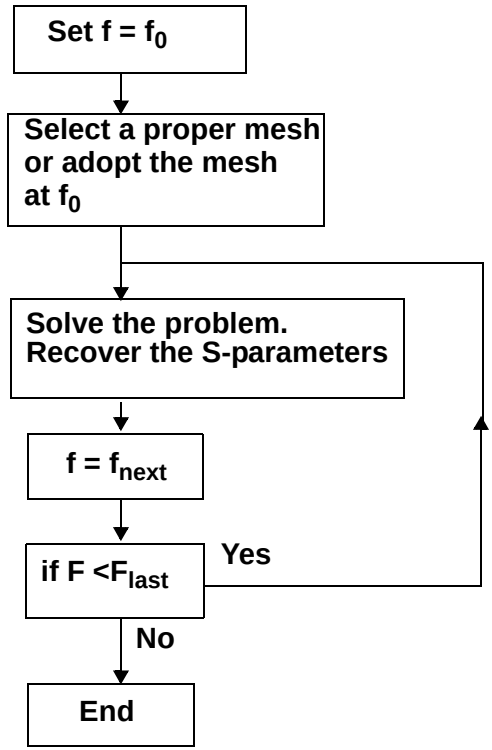
Discrete Frequency Sweeps

A Discrete sweep generates field solutions at specific frequency points in a frequency range. For example, if you specify a range of 1000 MHz to 2000 MHz, then a Step Size of 2.5, the result would be solutions at 1000, 1250, 1500, 1750, and 2000 MHz. By default, the field solution is only saved for the final frequency point computed, which would be at 2000 MHz in this case. Select the **Save Fields** option when setting up the points to solve if you want to save the field solution for a specific point. The S-parameters are saved for every frequency point. The more steps you request, the longer it takes to complete the frequency sweep.

Choose a Discrete sweep if only a few frequency points are necessary to accurately represent the results in a frequency range.

Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or, if you did not request an adaptive solution, the initial mesh generated for the problem. It uses this mesh without further refinement. Because the mesh for the adaptive solution is optimized only for the solution frequency, it is possible that the accuracy of the results could vary at frequencies significantly far away from this frequency. If you wish to minimize the variance, you can opt to use the center of the frequency range as the solution frequency. Then, after inspecting the results, run additional solutions with the solution frequency set to the critical frequencies.

The procedure for a Discrete frequency sweep is shown below, where *n* equally spaced frequencies are included in the sweep.



Interpolating Frequency Sweeps

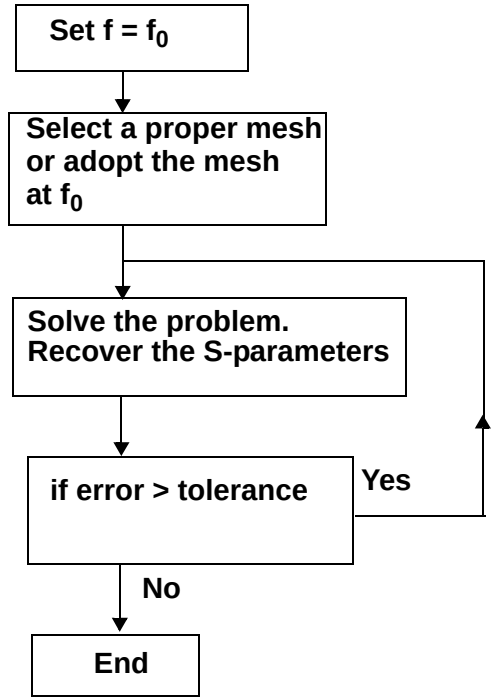
An Interpolating sweep estimates a solution for an entire frequency range. HFSS chooses the frequency points at which to solve the field solution so that the entire interpolated solution lies within a specified error tolerance. The sweep is complete when the solution meets the error tolerance criterion or generates the maximum number of solutions. To view more information about the solution, increase the number of steps and perform the sweep again.

The field solution for each point is deleted so that a new field solution can be generated for the next point. The full-field solution is only saved for the final frequency point computed. The S-parameters are saved for every solved frequency point.

Choose an Interpolating sweep if the frequency range is wide and the frequency response is smooth, or if the memory requirements of a Fast sweep exceed your resources. An Interpolating sweep's time requirement is much less than a Discrete sweep's because a solution for the entire frequency range is interpolated based on solutions for a minimal number of frequency points. The maximum time required for an Interpolating sweep is the time required for a single frequency solution multiplied by the maximum number of solutions.

Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or, if you did not request an adaptive solution, the initial mesh generated for the problem. It uses this mesh without further refinement.

The procedure for an Interpolating sweep is shown below, where *n* frequencies determined by the system are included in the sweep.



Solution Types

Driven Modal Solution

Choose the **Driven Modal** solution type when you want HFSS to calculate the modal-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines. The S-matrix solutions will be expressed in terms of the incident and reflected powers of waveguide modes.

Driven Terminal Solution

Choose the **Driven Terminal** solution type when you want HFSS to calculate the terminal-based S-parameters of multi-conductor transmission line ports. The S-matrix solutions will be expressed in terms of terminal voltages and currents.

Eigenmode Solution

Choose the **Eigenmode** solution type to calculate the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

Eigenmode Solutions

The Eigenmode solver can find the eigenmodes of lossy as well as lossless structures, and can calculate the unloaded Q of a cavity. Q is the quality factor, and is a measure of how much energy is lost in the system. Unloaded Q is the energy lost due to lossy materials. Because ports and other sources are restricted for eigenmode problems, the Q calculated does not include losses due to those sources.

The following restrictions apply to Eigenmode solution designs:

- The following excitations may not be defined: port, incident wave, voltage source, current source, and magnetic bias source.
- Radiation boundaries may not be defined.
- Frequency sweeps are not available.
- You may not view or plot the S-matrix data.
- Designs cannot include ferrite materials.

Related Topics

Calculating the Resonant Frequency

Calculating the Quality Factor

Calculating the Free Space Wave Number

Calculating the Resonant Frequency

The frequency of the eigenmode (or resonant frequency of the structure) is calculated from

$$f = \frac{k_o c}{2\pi}$$

where:

- c is the speed of light.
- f is the frequency of the wave.

Calculating the Quality Factor

Q is the unloaded quality factor, and is a measure of how much energy is lost in the structure due to lossy materials. Because ports and other sources are restricted for Eigenmode solutions, the Q calculated does not include losses due to those sources.

HFSS uses the following equation to calculate the approximate quality factor:

$$Q = \left| \frac{Mag(freq)}{2 \cdot Im(freq)} \right|$$

The Fields Calculator can also be used to calculate Q. In general, the equation for Q is

$$Q = (2\pi)(freq)\frac{U}{P}$$

where:

- U is the total energy stored in the cavity.
- P is the power lost, from resistive losses, for example.

Calculating the Free Space Wave Number

The free space wave number k_o is related to the frequency of the resonant modes and, for lossless problems, is calculated from

$$S\boldsymbol{x} + k_o^{\,2} T\boldsymbol{x} = 0$$

where:

- S and T are matrices that depend on the model geometry and the mesh.
- $\boldsymbol{x}$ is the electric field solution.
- k_o is the free space wave number.

Field Solutions

During the iterative, adaptive solution process, the S-parameters typically stabilize before the full field solution. Therefore, when you are interested in analyzing the field solution associated with a structure, it may be desirable to use convergence criteria that are tighter than usual.

In addition, for any given number of adaptive iterations, the magnetic field (H-field) is less accurate than the solution for the electric field (E-field) because the H-field is computed

from the E-field using the relationship $\boldsymbol{H} = \frac{\nabla \times \boldsymbol{E}}{-j\omega\mu}$.

Field Overlay Plots

In HFSS, field overlays are representations of basic or derived field quantities on surfaces or objects. The objects on which you plot the fields may be pre-existing parts of the model geometry or they may be objects that you draw in post-processing mode.

If you select a surface, HFSS will plot the field quantities on the surface. If you select an object, HFSS will plot the field quantities within the volume of the object.

You can choose to create a scalar plot or a vector plot of the fields. A scalar plot uses shaded lines to illustrate the magnitude of field quantities on surfaces or volumes. A vector plot uses arrows to illustrate the magnitudes of the x-, y-, and z-components of field quantities.

Field Quantities

The default field quantities that can be plotted, their definitions, and associated units are as follows:

Field Quantity	Definition	Units
Mag E	The magnitude of the electric field, |E|(x,y,z,t).	V/m
Mag H	The magnitude of the magnetic field, |H|(x,y,z,t).	Amps/m
Mag Jvol	The magnitude of the current density, |J|(x,y,z,t), over the volume.	$Amps/m^2$
Mag Jsurf	The magnitude of the current density, |J|(x,y,z,t), on the surface.	Amps/m
Complex Mag E	The complex magnitude of the electric field, |E|(x,y,z).	V/m
Complex Mag H	The complex magnitude of the magnetic field, |H|(x,y,z).	Amps/m
Complex Mag Jvol	The complex magnitude of the current density, |J|(x,y,z), over the volume.	$Amps/m^2$
Complex Mag Jsurf	The complex magnitude of the current density, |☺|(x,y,z), on the surface.	Amps/m
Vector E	The electric field, E(x,y,z,t).	V/m
Vector H	The magnetic field, H(x,y,z,t).	Amps/m
Vector Jvol	The current density, J(x,y,z), over the volume.	$Amps/m^2$
Vector Jsurf	The current density, J(x,y,z), on the surface.	Amps/m
Vector Real Poynting	The Poynting vector, defined as E x H*.	W/m^2
Local SAR	The specific absorption rate.	W/kg
Average SAR	The average specific absorption rate.	W/kg
Certification SAR	The IEEE specific absorption rate certification number.	W/kg

Specifying the Phase Angle

Specifying the phase angle at which the field quantity is calculated enables you to compute the real part of the field's magnitude at different points in its cycle. These quantities can be represented in the form $\boldsymbol{A}(x, y, z, t) = \boldsymbol{A}(x, y, z)\cos(\omega t + \theta(x, y, z))$, where

- ω is the angular frequency at which the quantities are oscillating, specified during the solution.
- θ (x,y,z) is the phase angle (the offset from a cosine wave that peaks at t = 0).

Peak Versus RMS Phasors

This section concerns how field quantities are represented within HFSS. Some users will not need this information, such as those who wish to know port S-parameters or relative amplitudes of field

solutions. Those that wish to find absolute field values, for example, will need to review the difference between the two types of field representation, peak and RMS.

HFSS solves in the frequency domain and obtains a phasor representation of the steady-state finite element field solution. Physical quantities such as the instantaneous (time domain) electric field are then obtained as derived quantities from the phasor representation.

If E_x is the x-component of a "peak" phasor quantity representing a time-harmonic electric field, the physical electric field x-component at time t, denoted $E_x(t)$, is computed from

$$E_x(t) = \Re[E_x e^{j\omega t}]$$

where

- $\Re$ is the real part of a complex number or function.
- ω is angular frequency, $2\pi f$.
- j is the imaginary unit, $\sqrt{-1}$.
- t is the time.

On the other hand, if E_x is an "RMS" phasor, an additional factor of $\sqrt{2}$ is required as follows:

$$E_x(t) = \Re[\sqrt{2}E_x e^{j\omega t}]$$

As a consequence of these equations, the peak physical field, max ($E_x(t)$) observed over a full time cycle is $max(E_x(t)) = |E_x|$ for peak phasors and $max(E_x(t)) = \sqrt{2}|E_x|$ for RMS phasors.

Additionally, given field phasors E and H, to compute the time-averaged power flow through a surface, the normal component of the real part of the complex Poynting vector is integrated over the surface. The correct form of the complex Poynting vector *S* depends on which phasor representation is used. For peak phasors, $S = \frac{1}{2}E \times H^*$.

For RMS phasors, $S = E \times H^*$.

The conventions used by HFSS are as follows:

- Each propagating mode incident on a port contains 1 watt of time-averaged power.
- Circuit gap sources are specified in a peak sense. That is, if a voltage gap source magnitude is 5 volts, then the time domain circuit source behaves as $v(t) = 5\cos\omega t$. Likewise for a current gap source.

- Plane wave sources are specified in a peak sense. That is, if the plane wave magnitude is 5 V/m, then the plane wave incident field magnitude is $E(t) = 5\cos(\boldsymbol{k} \cdot \boldsymbol{r} + \omega t)$.
- Radiated power, as computed by the fields post processor, is a time-averaged quantity computed using the complex Poynting vector.
- Phasors in the Fields Calculator are peak phasors. The Poynting vector button in the calculator therefore implements the Poynting vector for peak phasors, $S = \frac{1}{2}E \times H^*$. Calculations that compute either average or instantaneous time domain quantities must adhere to the peak phasor conventions.

Calculating the SAR

The specific absorption rate (SAR) is a measure of the amount of electromagnetic energy absorbed in a lossy dielectric material. The SAR is a basic scalar field quantity that can be plotted on surfaces or objects in HFSS.

HFSS uses the following equation to calculate the SAR: $\sigma * E^2/(2\rho)$.

where

- σ = the material's conductivity. This is defined as: $\sigma_{bulk} + \omega\varepsilon_o\varepsilon_r tg\delta$
- ρ = the mass density of the dielectric material in mass/unit volume.

There are three types of SAR plots available in HFSS: local SAR, average SAR, and certification SAR. When calculating the local SAR, HFSS uses the equation above to calculate the SAR at each mesh point on an overlay plot. HFSS interpolates the values between the mesh points across the plot.

When plotting the average SAR, for each mesh point on the plot, HFSS reports the SAR averaged over a volume that surrounds that point. The volume is determined by the settings for the material's mass density and mass of the material surrounding each mesh point set in the **Specific Absorption Rate Setting** dialog box.

When plotting the certification SAR, HFSS applies an IEEE standard procedure. The IEEE procedure makes the following assumptions:

- The peak E-field will reside on the surface of the phantom.
- The volume used for the integration will be an equal sided cube contained completely inside the phantom with an axis normal to the surface of the location of the peak value of SAR

Related Topics

Modifying SAR Settings

S-Parameters

Please see the following topics in this section:

Renormalized S-Matrices

Calculating Characteristic Impedance

Renormalizing to Zpv or Zvi Impedances

Calculating the PI Impedance

Calculating the PV Impedance

Calculating the VI Impedance

Impedance Multipliers

Calculating the S-Matrix

Calculating the Z-Matrix

Calculating the Y-Matrix

Calculating the Complex Propagation Constant (Gamma)

Calculating the Effective Wavelength (Lambda)

Calculating the Relative Permittivity (Epsilon)

De-embedded S-Matrices

Renormalized S-Matrices

Before a structure's generalized S-matrix can be used in a high frequency circuit simulator to compute the reflection and transmission of signals, it must be normalized to the appropriate impedance. For example, if a generalized S-matrix has been normalized to 50 ohms, it can be used to compute reflection and transmission directly from signals that are normalized to 50 ohms.

To renormalize a generalized S-matrix to a specific impedance, HFSS first calculates a unique impedance matrix *Z*, associated with the structure defined as follows:

$$Z = \sqrt{Z_0}(I - S)^{-1}(I + S)\sqrt{Z_0}$$

where

- *S* is the *n x n* generalized S-matrix.
- *I* is an *n x n* identity matrix.
- Z_0 is a diagonal matrix having the characteristic impedance (Z_0) of each port as a diagonal value.

The renormalized S-matrix is then calculated from the unique impedance matrix using this relationship:

$$S_\Omega = \sqrt{Y_\Omega}(Z - Z_\Omega)(Z + Z_\Omega)^{-1}\sqrt{Z_\Omega}$$

where

- *Z* is the structure's unique impedance matrix.

- Z_Ω and Y_Ω are diagonal matrices with the desired impedance and admittance as diagonal values. For example, if the matrix is being renormalized to 50 ohms, then Z_Ω would have diagonal values of 50.

Visualize the generalized S-matrix as an S-matrix that has been renormalized to the characteristic impedances of the structure. Therefore, if a diagonal matrix containing the characteristic impedances of the structure is used as Z_Ω in the above equation, the result would be the generalized S-matrix again.

For information about renormalized *terminal* S-matrices, see *Differential Pairs* in the *Technical Notes*.

HFSS needs to calculate the characteristic impedance of each port in order to compute a renormalized S-matrix.

Related Topics

Renormalizing S-Matrices

Calculating Characteristic Impedance

Each port in a structure being analyzed can be viewed as a cross-section of a transmission line. HFSS computes the characteristic impedance of each port in three ways — as Zpi, Zpv, and Zvi impedances. You have the option of specifying which impedance will be used in the renormalization calculations.

- For TEM waves, the Zvi impedance converges on the port's actual impedance and should be used.
- When modeling microstrips, it is sometimes more appropriate to use the Zpi impedance.
- For slot-type structures (such as finline or coplanar waveguides), Zpv impedance is the most appropriate.

HFSS will always calculate Zpi impedance, the impedance calculation using power and current, which are well-defined for a port because they are computed over the area of the port. Zpv and Zvi are not calculated by default. This is because V is computed by integrating along a user-defined integration line. To renormalize the solution to a Zpv or Zvi characteristic impedance, you must have defined an impedance line.

Under the **Matrix Data** tab of the **Solution Data** dialog box, the characteristic impedance can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

For more information on the computation of impedances, refer to the following:

Bruno Bianco, Luigi Panini, Mauro Parodi, and Sandro Ridella, "Some Considerations about the Frequency Dependence of the Characteristic Impedance of Uniform Microstrips," *IEEE Transactions on Microwave Theory and Techniques*, vol. MTT-26 No. 3, March 1978.

Edward F. Kuester, David C. Chang, and Leonard Lewin, "Frequency-Dependent Definitions of Microstrip Characteristic Impedance," *International URSI Symposium on Electromagnetic Waves*, Munich, 26-29 August 1980, pp. 335 B/1-3.

Renormalizing to Zpv or Zvi Impedances

The S-matrices initially calculated by HFSS are generalized S-matrices that have been normalized to the impedances of each port; however, you can compute S-matrices that are normalized to specific impedances, such as 50 ohms.

To convert a generalized modal S-matrix to a renormalized modal S-matrix, HFSS first needs to compute the characteristic impedance at each port. There are several ways to compute characteristic impedance. Two methods — the Zpv and Zvi methods — require an impedance, or *integration*, line.

HFSS will always calculate Zpi impedance, the impedance calculation using power and current, which are well-defined for a port because they are computed over the area of the port. Zpv and Zvi are not calculated by default. This is because v is computed by integrating along a user-defined integration line. To renormalize the solution to a Zpv or Zvi characteristic impedance, you must define an integration line.

Calculating the PI Impedance

The Z_{pi} impedance is the impedance calculated from values of power (P) and current (I):

$Z_{pi} = \frac{P}{I \cdot I}$.

The power and current are computed directly from the simulated fields. The power passing through a port is equal to the following: $P = \oint_s E \times H ds$, where the surface integral is over the surface of the port.

The current is computed by applying Ampere's law to a path around the port: $I = \oint_l H \bullet dl$.

While the net current computed in this way will be near zero, the current of interest is that flowing into the structure, I^-, or that flowing out of the structure, I^+. In integrating around the port, HFSS keeps a running total of the contributions to each and uses the average of the two in the computation of impedances.

Calculating the PV Impedance

The Z_{pv} impedance is the impedance calculated from values of power (P) and voltage (V):

$Z_{pv} = \frac{V \bullet V}{P}$, where the power and voltage are computed directly from the simulated fields.

The power is computed in the same way as the Z_{pi} impedance. The voltage is computed as follows: $V = \oint_l E \bullet dl$, over which HFSS integrates is referred to as the impedance line — which is defined when the ports are set up. To define the impedance line for a port, select the two points across which the maximum voltage difference occurs. You must define an integration line to specify where the maximum voltage difference will be.

Calculating the VI Impedance

The Z_{vi} impedance is given by $Z_{vi} = \sqrt{Z_{pi} Z_{pv}}$.

For TEM waves, the Z_{pi} and Z_{pv} impedances form upper and lower boundaries to a port's actual characteristic impedance. Therefore, the value of Z_{vi} approaches a port's actual impedance for TEM waves.

Impedance Multipliers

If a symmetry plane has been defined (allowing the model of a structure to be cut in half), the impedance computations must be adjusted by specifying an impedance multiplier. The need for this multiplier can be understood by looking at how the use of symmetry affects the computation of Z_{pv}.

In cases where a perfect E plane of symmetry splits a structure in two, only one-half of the voltage differential and one-half of the power flow can be computed by the system. Therefore, since the Z_{pv} impedance is given by $Z_{pv} = \frac{V \bullet V}{P}$, the computed value is one-half the desired value. An impedance multiplier of 2 must be specified in such cases.

In cases where a perfect H plane of symmetry splits a structure in two, only one-half of the power flow is seen by the system but the full voltage differential is present. Therefore, structures split in half with perfect H symmetry planes result in computed impedances that are twice those for the full structure. An impedance multiplier of 0.5 must be specified in such cases.

If multiple symmetry planes are used or if only a wedge of a structure is modeled, you must adjust the impedance multiplier accordingly.

If you have defined a symmetry plane, the computed impedances will not be for the full structure. Generally, use one of the following values for the impedance multiplier:

- If the structure has a perfect E plane of symmetry, use 2. Such models have one-half of the voltage differential and one-half of the power flow of the full structure, resulting in impedances that are one-half of those for the full structure.
- If the structure has a perfect H plane of symmetry, enter 0.5. Such models have the same voltage differential but half the power flow of the full structure, resulting in impedances that are twice those for the full structure.
- If the structure has a combination of perfect H and perfect E boundaries, adjust accordingly. For example, you do not have to enter an impedance multiplier for a structure with both a perfect E and perfect H boundary since you would be multiplying by 2 and 0.5.

Related Topics

Setting the Impedance Multiplier

Calculating Terminal Characteristic Impedance Matrix

See Converting Modes to Nodes.

Calculating the S-Matrix

A generalized S-matrix describes what fraction of power associated with a given field excitation is transmitted or reflected at each port.

The S-matrix for a three-port structure is as follows:

where

- All quantities are complex numbers.

$$\begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix}$$

- The magnitudes of *a* and *b* are normalized to a field carrying one watt of power.
  - $|ai|^2$ represents the excitation power at port i.
  - $|bi|^2$ represents the power of the transmitted or reflected field at port i.
- The full field pattern at a port is the sum of the port's excitation field and all reflected/transmitted fields.
- The phase of *ai* and *bi* represent the phase of the incident and reflected/transmitted field at t=0.
  - $\angle a_i$ represents the phase angle of the excitation field on port i at t = 0. (By default, it is zero for lossy port modes and lossless propagating modes. For lossless cut-off modes, it is 90.)
  - $\angle b_i$ represents the phase angle of the reflected or transmitted field with respect to the excitation field.
- S_{ij} is the S-parameter describing how much of the excitation field at port j is reflected back or transmitted to port i. For example, S_{31} is used to compute the amount of power from the port 1 excitation field that is transmitted to port 3. The phase of S_{31} specifies the phase shift that occurs as the field travels from port 1 to port 3.

Note When the Wave module computes the excitation field for a given port, it has no information indicating which way is "up" or "down." Therefore, if the port mode has not been calibrated, the calculated S-parameters may be 180 degrees out of phase with the expected solution.

Under the **Matrix Data** tab of the **Solution Data** dialog box, the S-matrix can be displayed as magnitude/ phase, real/ imaginary, dB/ phase, magnitude, phase, real, imaginary, or dB.

Calculating the Z-Matrix

The impedance matrix, *Z*, is calculated from the S-matrix as follows:

$$Z = \sqrt{Z_0}(I - S)^{-1}(I + S)\sqrt{Z_0}$$

where

- *S* is the *n x n* generalized S-matrix.
- *I* is an *n x n* identity matrix.

- Z_0 is a diagonal matrix having the characteristic impedance (Z_0) of each port as a diagonal value.

Under the **Matrix Data** tab of the **Display Items Dialog**, the Z-matrix can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

Calculating the Y-Matrix

The admittance matrix, *Y,* is simply the inverse of the impedance matrix, *Z.*

Under the **Matrix Data** tab of the **Solution Data** dialog box, the Y-matrix can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

Calculating the Complex Propagation Constant (Gamma)

Each port is assumed to be connected to a transmission structure that has the same cross-section as the port. The complex propagation constant, γ, of these transmission lines is computed by HFSS, and is given by $\gamma = \alpha + j\beta$, where:

- α is the attenuation constant of a signal in the transmission structure. It is the real component of the propagation constant and has units of nepers per meter.
- β is the phase constant associated with the wave. It is the imaginary component of the propagation constant and has units of radians per meter.

Under the **Matrix Data** tab of the **Display Items Dialog**, gamma can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

Calculating the Effective Wavelength (Lambda)

The effective wavelength, λ_{eff}, is calculated from

$$\lambda_{eff} = \frac{2\pi}{\beta}$$

where β is the phase constant associated with the wave.

Under the **Matrix Data** tab of the **Solution Data** dialog box, lambda is displayed when **Gamma** is selected as the matrix type.

Calculating the Relative Permittivity (Epsilon)

The relative permittivity, ε_r, is calculated using

$$\lambda_{eff} = \frac{c}{\sqrt{\varepsilon_r} f}$$

where

- λ_{eff} is the effective wavelength given in meters.
- c is the speed of light.
- f is the frequency of the wave.

Under the **Matrix Data** tab of the **Solution Data** dialog box, epsilon is displayed when **Gamma** is selected as the matrix type.

De-embedded S-Matrices

If a uniform length of transmission line is added to (or removed from) a port, the S-matrix of the modified structure can be calculated using the following relationship $[S'] = [e^{\gamma l}][S][e^{\gamma l}]$, where

- $[e^{\gamma l}]$ is a diagonal matrix with the following entries:

$$\begin{bmatrix} e^{\gamma_1 l_1} & 0 & 0 \\ 0 & e^{\gamma_2 l_2} & 0 \\ 0 & 0 & e^{\gamma_3 l_3} \end{bmatrix}$$

- $\gamma = \alpha + j\beta$ is the complex propagation constant, where:
  - α is the attenuation constant of the wave.
  - β is the propagation constant of the uniform transmission line at port i.
- l_i is the length of the uniform transmission line that has been added to or removed from the structure at port *i*. A positive value indicates that a length of transmission line has been removed from the structure.

The value of γ for each port is automatically calculated by HFSS.

Related Topics

De-embedding S-Matrices

Radiated Fields

When HFSS calculates radiation fields, the values of the fields over the radiation surface are used to compute the fields in the space surrounding the device. This space is typically split into two regions — the near-field region and the far-field region. The near-field region is the region closest to the source. In general, the electric field E(x,y,z) external to the region bounded by a closed surface may be written as

$$\boldsymbol{E}(x, y, z) = \int_s (\langle j\omega\mu_0 \boldsymbol{H}_{tan} \rangle G + \langle \boldsymbol{E}_{tan} \times \nabla G \rangle + \langle \boldsymbol{E}_{normal} \nabla G \rangle) dS$$

where

- *s* represents the radiation boundary surfaces.
- *j* is the imaginary unit, $\sqrt{-1}$.
- ω is the angular frequency, $2\pi f$.
- μ_0 is the relative permeability of the free space, $4\pi \times 10^{-7}$ Wb/Am.
- $\boldsymbol{H}_{tan}$ is the component of the magnetic field that is tangential to the surface.
- $\boldsymbol{E}_{normal}$ is the component of the electric field that is normal to the surface.
- $\boldsymbol{E}_{tan}$ is the component of the electric field that is tangential to the surface.
- *G* is the free space Green's function, given by

$$G = \frac{e^{-jk_0|\boldsymbol{r}-\boldsymbol{r}'|\sqrt{\mu_r \varepsilon_r}}}{|\boldsymbol{r}-\boldsymbol{r}'|}$$

where

- k_0 is the free space wave number,

$\omega\sqrt{\mu_0\varepsilon_0} = \omega / c$.

- $\boldsymbol{r}$ and $\boldsymbol{r}'$ represent field points and source points on the surface, respectively.
- ε_0 is the permittivity of free space, $1/(c^2\mu_0)$
- ε_r is the relative permittivity of a dielectric.
- μ_r is the relative permeability of a dielectric.

This *r* dependence is characteristic of a spherical wave, a key feature of far fields. The far field is a spherical TEM wave with the following equation:

$$\boldsymbol{E} = \eta_0 \boldsymbol{H} \times \hat{\boldsymbol{r}}.$$

where η_0 is the intrinsic impedance of free space.

When calculating the near fields, HFSS uses the general expressions given in (**eq. 1**). You must specify the radial coordinate *r*. Because it can be used to compute fields at an arbitrary radius from the radiating structure, this command can be useful in EMC applications.

Note If HFSS calculates the near fields in a problem containing an incident wave, the radius at which the fields are calculated is very important. If the radius is within the solution region, then the fields calculated are either the total fields or the scattered fields depending upon which is selected. If the radius is outside the solution region, then the fields calculated are only the scattered fields.

When calculating the far fields, the previously discussed far-field approximations are used, and the result is valid only for field points in the far-field region.

Warning A radiation or PML boundary must have been defined in the design for HFSS to calculate radiated fields.

Spherical Cross-Sections

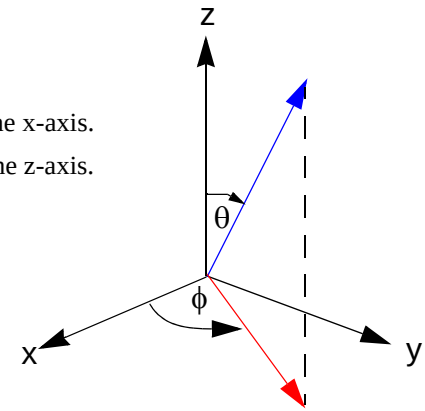
When you set up a spherical surface over which to analyze near or far fields, you specify a range and step size for phi and theta. These indicate the spherical direction in which you want to evaluate the radiated fields. For every value of phi there is a corresponding range of values for theta, and vice versa. This creates a spherical grid. Each grid point indicates a unique direction along a line that extends from the center of the sphere through the grid point. The radiated field is evaluated in this direction. The number of grid points is determined by the step size for phi and theta.

The sphere can be defined according to any defined coordinate system and before or after a solution has been generated.

The relationship between phi and theta is shown below.

ϕ is rotated away from the x-axis.

θ is rotated away from the z-axis.



When HFSS evaluates the radiated fields, it needs at least two directions along which to plot the fields. Therefore, if the step size for phi is zero, then the step size for theta must be greater than zero, and vice versa. This ensures that the fields are plotted in at least two directions.

When setting up the sphere, phi and theta angles must be specified between -360 degrees (**deg**) and 360 degrees (**deg**), or the equivalents in radians (**rad**). If **deg** nor **rad** is specified, HFSS assumes the value to be in degrees.

Following are additional guidelines for specifying **Phi** in the **Near Field Radiation Sphere Setup** window or the **Far Field Radiation Sphere Setup** window:

Start	The point where the rotation of phi begins. The **Start** value must be equal to or greater than one.
Stop	The point where the rotation of phi ends. The **Stop** value must be greater than the **Start** value and less than 360. If the **Stop** value is equal to the **Start** value, then HFSS assumes that only one angle should be used and the **Step Size** value will be ignored.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of phi. For example, to divide a sweep from 0° to 180° into 10° increments, you would enter **10**. Entering zero for the **Step Size** causes the sweep to consist of one point, the start value. If the **Step Size** value is zero, then HFSS assumes that only one angle should be used.

Following are additional guidelines for specifying **Theta**:

Start	The point where the rotation of theta begins. The **Start** value must be greater than -90 degrees, or the equivalent in radians.
Stop	The point where the rotation of theta ends. The **Stop** value must be greater than the **Start** value and less than 90 degrees, or the equivalent in radians. If the **Stop** value is equal to the **Start** value, HFSS assumes that only one angle should be used and the **Step Size** value will be ignored.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of theta. For example, to divide a sweep from -60 degrees to 60 degrees into 10-degree increments, you would enter **10deg**. Entering zero for the number of steps causes the sweep to consist of one point, the **Start** value. If the **Step Size** value is zero, then HFSS assumes that only one angle should be used.

Related Topics

Setting up a Far-Field Infinite Sphere

Setting up a Near-Field Sphere

Maximum Near-Field Data

The parameters listed in the **Max Field Data** window remain the same regardless of the geometry over which they were calculated. However, the coordinates displayed change depending on the geometry.

On a sphere, the coordinates — phi and theta — of the maximum value are listed under **Phi** and **Theta**. The values are given in volts per meter. Along a line, the coordinates — x, y, and z — of the

maximum values are listed under **X**, **Y**, and **Z**. The values are given in volts per meter, and the coordinates are given in meters.

The following parameters are listed:

Total	The maximum of the total E-field.
X	The maximum E-field in the x-direction.
Y	The maximum E-field in the y-direction.
Z	The maximum E-field in the z-direction.
Phi	The maximum E-field in the ϕ-direction.
Theta	The maximum E-field in the θ-direction.
LHCP	The maximum left-hand circularly polarized component, which is equal to $\left\|\frac{1}{\sqrt{2}}(E_\theta - jE_\phi)\right\|$.
RHCP	The maximum right-hand circularly polarized component, which is equal to $\left\|\frac{1}{\sqrt{2}}(E_\theta + jE_\phi)\right\|$.
Ludwig 3/X dominant	The maximum of the dominant component, *Vmain*, for an x-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta cos\phi - E_\phi sin\phi\|$.
Ludwig 3/Y dominant	The maximum of the dominant component, *Vmain*, for a y-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta sin\phi + E_\phi cos\phi\|$.

Maximum Far-Field Data

When HFSS calculates antenna parameters, the following maximum field data is calculated:

Total	The maximum of the total rE-field.
X	The maximum rE-field in the x-direction.
Y	The maximum rE-field in the y-direction.
Z	The maximum rE-field in the z-direction.
Phi	The maximum rE-field in the ϕ-direction.
Theta	The maximum rE-field in the θ-direction.
LHCP	The maximum left-hand circularly polarized component, which is equal to $\left\|\frac{1}{\sqrt{2}}(E_\theta - jE_\phi)\right\|$.

RHCP	The maximum right-hand circularly polarized component, which is equal to $\left\|\frac{1}{\sqrt{2}}(E_\theta + jE_\phi)\right\|$.
Ludwig 3/X dominant	The maximum of the dominant component, *Vmain*, for an x-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta cos\phi - E_\phi sin\phi\|$.
Ludwig 3/Y dominant	The maximum of the dominant component, *Vmain*, for a y-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta sin\phi + E_\phi cos\phi\|$.

When calculating the maximum far field values, the distance *r* is factored out of the E-field. Therefore, the units for the maximum field data values are given in volts.

Array Factors

HFSS enables you to compute antenna array radiation patterns and antenna parameters for designs that have analyzed a single array element. You can define array geometry and excitation. HFSS models the array radiation pattern by applying an "array factor" to the single element's pattern.

Two array geometry types are supported. The "regular uniform array" geometry defines a finite 2D array of uniformly spaced, equal-amplitude elements. This is a natural specification after analyzing a single-unit cell of an infinite array. The regular array type may be scanned to a user-specified direction. Scan direction can be specified in terms of spherical coordinate angles in the radiation coordinate system. The regular array geometry type also allows scan specification in terms of differential phase shifts between elements.

The "custom array" geometry allows for greater flexibility. It defines an arbitrary array of identical elements distributed in 3D space with individual user-specified complex weights.

Cautionary Note for Array Factor Use

The field factorization (eq. 1) and consequent use of an array factor are useful tools for analyzing the radiated fields of antenna arrays; however, the analysis can yield incorrect results if used improperly. An HFSS single array element solution does not generally take into account the effects of the element's hypothetical neighbors.

For closely spaced array elements, these proximity effects (mutual coupling) may be significant. Consequently the patterns of the array elements vary with their position in the array and may depart significantly from the isolated element pattern. In such cases, the primary assumption in the use of the array factor is violated and the results will be inaccurate.

Note in particular that the array power expressions (eq. 13) and (eq. 14) neglect mutual coupling between elements of the finite array. Unless mutual coupling effects are negligible or have been implicitly included in the single element solution, the normalizations (eq. 13) and (eq. 14) gain and directivity are incorrect.

Related Topics

Defining Antenna Arrays

Theory of the Array Factor Calculation

The composite far-field pattern, $\boldsymbol{E}_{array}(\phi, \theta)$ from an array of N identical radiating sources, each with far-field pattern $\boldsymbol{E}_{element}(\phi, \theta)$, may be factored into the form

$$\boldsymbol{E}_{array}(\phi, \theta) = AF(\phi, \theta)\boldsymbol{E}_{element}(\phi, \theta) \quad (1)$$

where the "array factor" AF (ϕ, θ) is defined as

$$AF(\phi, \theta) = \sum_{n=1}^{N} W_n e^{jk\boldsymbol{r}_n \cdot \hat{\boldsymbol{r}}} \quad (2)$$

and where

- (ϕ, θ) are the field-point spherical angles.
- W_n is the complex weight assigned to element n.
- j is $\sqrt{-1}$.
- k is $2\pi/\lambda$.
- $\mathbf{r}_n$ is the position vector of element n, $<xn, yn, zn>$.
- $\hat{\boldsymbol{r}}$ is the pattern angle unit vector, $<\sin\theta\cos\phi, \sin\theta\sin\phi, \cos\theta>$.

The complex weights W_n in (eq. 2) may be written in terms of a (real) voltage amplitude A_n and (real) phase ψ_n as:

$$W_n = A_n e^{j\psi n}. \quad (3)$$

To scan a regular array in the direction (ϕ_0, θ_0), the element phases ψ_n are set to

$$\psi_n = -k\boldsymbol{r}_n \cdot \hat{\boldsymbol{r}}_o \quad (4)$$

where

$$\hat{\boldsymbol{r}}_0 = <\sin\theta_0\cos\phi_0, \sin\theta_0\sin\phi_0, \cos\theta_0> \quad (5)$$

is the scan-angle unit vector.

Regular Uniform Arrays

Let us define a uniform array as an array with unity amplitude weights for all elements, i.e., $A_n = 1$ for all n. For the case in which a uniform array is scanned to direction $\hat{\boldsymbol{r}}_0$, the array factor (eq. 2) becomes

$$AF(\phi, \theta) = \sum_{n=1}^{N} e^{jk\boldsymbol{r}_n \cdot (\hat{\boldsymbol{r}} - \hat{\boldsymbol{r}}_0)}. \quad (6)$$

For a "regular" uniform array with element spacing defined by lattice vectors $\boldsymbol{u}$ and $\boldsymbol{v}$, the element position vectors $\boldsymbol{r}_n$ may be written in the doubly-indexed form

$$\boldsymbol{r}_{mn} = (m-1)\boldsymbol{u} + (n-1)\boldsymbol{v} \quad (7)$$

with $m = 1, 2, ..., Nu$ and $n = 1, 2, ..., Nv$. The total number of elements in the array is given by $N = NuNv$.

The array factor (eq. 6) for the $Nu \times Nv$ array becomes

$$AF(\phi, \theta) = \sum_{m=1}^{Nu} \sum_{n=1}^{Nv} e^{jk\boldsymbol{r}_{mn} \cdot (\hat{\boldsymbol{r}} - \hat{\boldsymbol{r}}_0)}. \quad (8)$$

Scan Specification for Regular Uniform Arrays

The scanning phase (eq. 4) is written in terms of the scan direction $\hat{\boldsymbol{r}}_0$. Alternatively, for a regular uniform array, the scanning phase may be written in terms of the differential phase shift between elements. This may be more natural in cases where the individual array element was analyzed using linked boundaries with user-specified phase shifts applied between master and slave boundaries.

To develop this alternate scanning phase description, (eq. 7) is used to rewrite the expression (eq. 4) in doubly-indexed form as follows:

$$\psi_{mn} = -k\boldsymbol{r}_{mn} \cdot \hat{\boldsymbol{r}}_0 = -k(m-1)\boldsymbol{u} \cdot \hat{\boldsymbol{r}}_0 - k(n-1)\boldsymbol{v} \cdot \hat{\boldsymbol{r}}_0 \quad (9)$$

Let us define ψ_u as the differential phase between adjacent elements in the **u** direction. Similarly, let us define ψ_v as the differential phase between adjacent elements in the **v** direction. Then

$$\psi_u \equiv \psi_{m+1,\,n} - \psi_{m,\,n} = -k\boldsymbol{u} \cdot \hat{\boldsymbol{r}}_0 \qquad (10)$$

and

$$\psi_v \equiv \psi_{m,\,n+1} - \psi_{m,\,n} = -k\boldsymbol{v} \cdot \hat{\boldsymbol{r}}_0 \qquad (11)$$

The scanning phase (eq. 4) may now be rewritten in terms of ψ_u and ψ_v as

$$\psi_{mn} = (m-1)\psi_u + (n-1)\psi_v \qquad (12)$$

Thus in the case of a regular uniform array, the angle pair (ψ_u, ψ_v) may act as a substitute scan definition for the more general (ϕ_0, θ_0).

Custom Arrays

Once you have imported the array factor information from a text file, HFSS uses (eq. 8) to compute the array factor. When a custom array is defined, no scan direction is set and the array factor phase weights are those specified on an element-by-element basis in the geometry file.

The text file must have the following format:

```
N
x_1 y_1 z_1 A_1 P_1
x_2 y_2 z_2 A_2 P_2
...
...
x_N y_N z_N A_N P_N
```

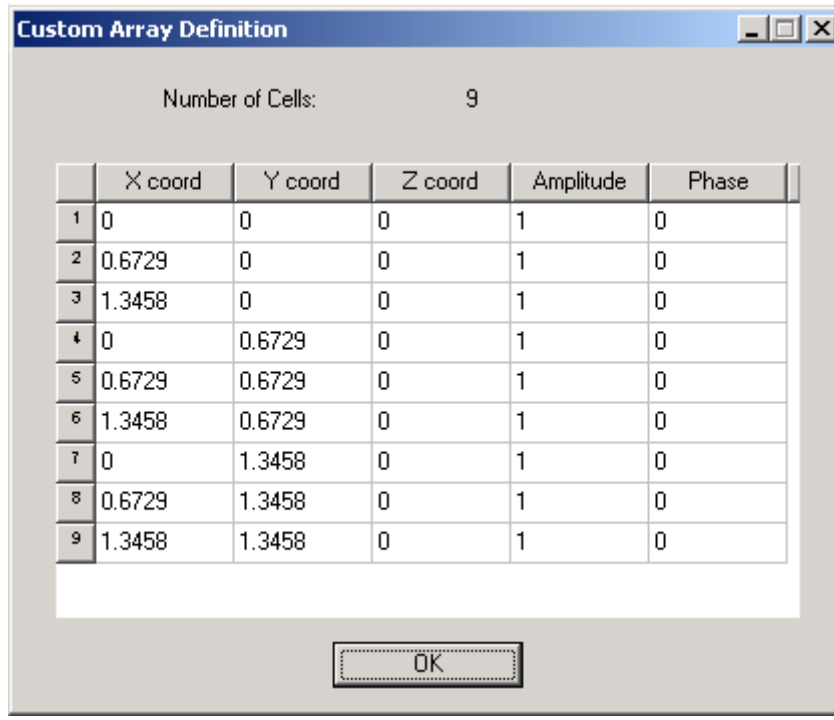
where

- x_1 is the x-coordinate position of the first element, which will take the model drawing unit.
- y_1 is the y-coordinate position of the first element, which will take the model drawing unit.
- z_1 is the y-coordinate position of the first element, which will take the model drawing unit.
- A_1 is the amplitude weight of the first element.
- P_1 is the phase weight for the first element.

Following is an example of a square 3 x 3 custom array geometry defined in a text file. The array elements are uniformly weighted and separated from one another in the x- and y-directions by 0.6729 user units.

```
9
0.0      0.0 0.0 1.00 .0
0.6729   0.0 0.0 1.00 .0
1.3458   0.0 0.0 1.00 .0
0.0      0.6729 0.0 1.00 .0
0.6729   0.6729 0.0 1.00 .0
1.3458   0.6729 0.0 1.00 .0
0.0      1.3458 0.0 1.00 .0
0.6729   1.3458 0.0 1.00 .0
1.3458   1.3458 0.0 1.00 .0
```

The information will appear as follows in the **Custom Array Definition** window:



Power Normalizations

When the array factor feature is in use, the power normalizations used to compute the gain and directivity are modified as follows.

Let $P_{element}^{rad}$ and $P_{element}^{accepted}$ denote the radiated power and the accepted power of the single array element. $P_{element}^{rad}$ is computed by integrating the Poynting vector

on the radiation boundary surface and $P_{element}^{accepted}$ is computed by integrating the Poynting vector on the union of port boundary surfaces.

When the array factor feature is invoked for an array of *N* elements, the array radiated power P_{array}^{rad} and array accepted power $P_{array}^{accepted}$ will be computed simply as the sums of element-radiated and element-accepted powers, respectively, as follows:

$$P_{array}^{rad} = \left(\sum_{n=1}^{N} A_n^2 \right) P_{element}^{rad} \quad (13)$$

$$P_{array}^{accepted} = \left(\sum_{n=1}^{N} A_n^2 \right) P_{element}^{accepted} \quad (14)$$

Here A_n, as defined in (eq. 3), is the real amplitude weight applied to element *n*.

Antenna Parameters

Generally, when dealing with radiated fields, you are also interested in the antenna properties of the radiated bodies. HFSS calculates the following antenna properties:

Maximum intensity (Max U)

Peak directivity

Peak gain

Peak realized gain

Radiated power

Accepted power

Incident power

Radiation efficiency

Warning A radiation or PML boundary must have been defined in the design for HFSS to calculate radiated fields.

Related Topics

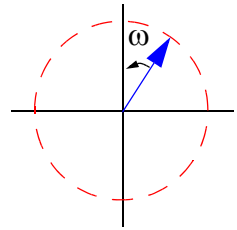
Computing Antenna Parameters

Polarization of the Electric Field

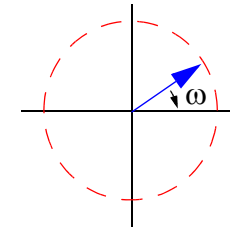
At each aspect angle in the far field of a radiating source, the electric and magnetic field vectors lie in a fixed plane. Over time, the instantaneous electric field vector traces out a figure or shape in this plane. This figure defines the polarization state of the field.

In general, this figure is an ellipse and is called the *polarization ellipse*. The wave is said to be *elliptically polarized* when the instantaneous electric field traces out an ellipse. As a special case, the polarization ellipse may be a circle, in which case the wave is *circularly polarized*. Elliptical and circular polarization have two different states, *left* and *right*, distinguished by the sense of rotation of the electric field vector. Some of these figures, or states, are shown below. In each case the direction of propagation is off the screen.

Circular

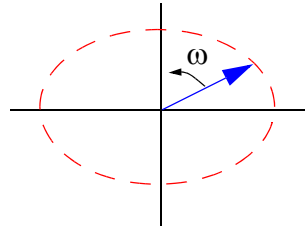


Right-hand circular polarization

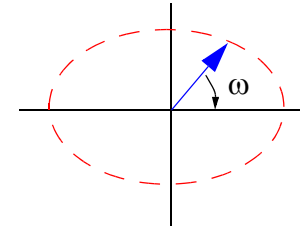


Left-hand circular polarization

Elliptical



Right-hand elliptical polarization



Left-hand elliptical polarization

where ω is the rotation radian frequency.

A second special case occurs when the polarization ellipse degenerates to a straight line. In this case the wave is *linearly polarized*.

To completely describe the polarization state of a radiated field, two independent components are required. HFSS supports three types of descriptions:

- Spherical polar
- Ludwig-3
- Circular

Spherical Polar

The most fundamental description of the polarization state of a radiated field is spherical polar, which is the electric field phasor resolved in the directions of unit theta and phi vectors of the reference coordinate system. In this description, the field may be written as $\boldsymbol{E} = (E_\theta, E_\phi)$.

The polarization ratio for a predominantly ϕ-polarized antenna is equal to

$$\left|\frac{E_\phi}{E_\theta}\right| .$$

The polarization ratio for a predominantly θ-polarized antenna is equal to

$$\left|\frac{E_\theta}{E_\phi}\right| .$$

Ludwig-3 Polarization

Arthur C. Ludwig wrote a classic paper [Ref. 1]on the definition of cross polarization. In particular, his third definition is often used since it describes the field components that are typically measured on a far-field antenna test range. Using his definition, the radiated field may be written as $\boldsymbol{E} = (E_x, E_y)$ where

$$E_x = E_\theta \cos\phi - E_\phi \sin\phi$$

$$E_y = E_\theta \sin\phi + E_\phi \cos\phi$$

and phi is the usual azimuthal angle in the reference spherical coordinate system.

[1] Arthur C. Ludwig, The Definition of Cross Polarization, IEEE Transactions on Antennas and Propagation, vol. AP-21 num. 1, pp. 116 -119, Jan. 1973.

Circular Polarization

For antennas designed to receive or transmit circularly polarized fields, a meaningful description is in terms of pure left and right circular states. In this description, the field may be written as $\boldsymbol{E} = (E_R, E_L)$ where

$$E_R = \frac{1}{\sqrt{2}}(E_\theta + jE_\phi)$$

$$E_L = \frac{1}{\sqrt{2}}(E_\theta - jE_\phi) .$$

Axial Ratio

Axial ratio is defined as the ratio of the major to the minor axis of the polarization ellipse.

1. E_x and E_y are orthogonal complex-valued field components. If either is zero, HFSS treats the field as linearly polarized. However, if neither E_x and E_y is zero:
2. Compute circular components E_- and E_+ from:

   $E_- = E_x - jE_y$

   $E_+ = E_x + jE_y$
3. If $E_+ = 0$ or $E_- = 0$ HFSS understands the field as perfectly circular, the axial ratio is 1. Otherwise, for the ellipical polarization case, HFSS determines the tilt angle τ from:

   $\text{phase}(E_-/E_+) = 2\tau$
4. Rotate the orginal data to coincide with the axes of the polarization ellipse.

$$E'_x = E_x \cos\tau - E_y \sin\tau$$

$$E'_y = E_x \sin\tau - E_y \cos\tau$$

5. The Axial Ratio *AR* is given by:

$$AR = \left| \frac{E'_y}{E'_x} \right|$$

Because the above definition does not discriminate between major and minor ellipse axes, to enforce the convention that $AR \leq 1$, it is necessary to check this condiction and if necessary invert the value obtained.

Polarization Ratio

The IEEE defines the (complex) polarization ratio as, “For a given field vector at a point in space, the (magnitude of the) ratio of the complex amplitudes of two specified orthogonally polarized field vectors into which the given field vector has been resolved.” [Ref. 2] HFSS computes the following six polarization ratios at each selected aspect angle:

$$Circular/LHCP = \left| \frac{E_L}{E_R} \right|$$

$$Circular/RHCP = \left| \frac{E_R}{E_L} \right|$$

$$Spherical/Phi = \left| \frac{E_\phi}{E_\theta} \right|$$

$$Spherical/Theta = \left| \frac{E_\theta}{E_\phi} \right|$$

$$Ludwig\ 3/X = \left| \frac{E_x}{E_y} \right|$$

$$Ludwig\ 3/Y = \left| \frac{E_y}{E_x} \right|$$

[2] IEEE Standard Definitions of Terms for Antennas, IEEE Transactions on Antennas and Propagation, vol. AP-31 num. 6, Nov. 1983.

Max U

The radiation intensity, *U*, is the power radiated from an antenna per unit solid angle. HFSS calculates the radiation intensity in the direction in which it has the maximum value. The maximum intensity of the radiation is measured in watts per steradian and is calculated by

$$U(\theta,\phi) = \frac{1}{2}\frac{|\boldsymbol{E}|^2}{\eta_0}r^2$$

where

- $U(\theta,\phi)$ is the radiation intensity in watts per steradian.
- $|E|$ is the magnitude of the E-field.
- η_0 is the intrinsic impedance of free space — 376.7 ohms.
- r is the distance from the antenna, in meters.

Related Topics

Computing Antenna Parameters

Peak Directivity

Directivity is defined as the ratio of an antenna's radiation intensity in a given direction to the radiation intensity averaged over all directions. Peak directivity, in turn, is the maximum directivity over all the user-specified directions of the far-field infinite sphere.

Directivity is a dimensionless quantity represented by

$$directivity = \frac{4\pi U}{P_{rad}}$$

where

- U is the radiation intensity in watts per steradian in the direction specified.
- P_{rad} is the radiated power in watts.

Note The peak directivity displayed in the **Antenna Parameters** window is the directivity in the direction of maximum radiation intensity, U_{max}.

- For a lossless antenna, the directivity will be equal to the gain. However, if the antenna has inherent losses, the directivity is related to the gain by the radiation efficiency of the antenna.

Related Topics

Setting up a Far-Field Infinite Sphere

Computing Antenna Parameters

Peak Gain

Gain is four pi times the ratio of an antenna's radiation intensity in a given direction to the total power accepted by the antenna. Peak gain, in turn, is the maximum gain over all the user-specified directions of the far-field infinite sphere.

The following equation is used to calculate gain in HFSS:

$$gain = 4\pi\frac{U}{P_{acc}}$$

where

- U is the radiation intensity in watts per steradian in the direction specified.
- P_{acc} is the accepted power in watts entering the antenna.

Gain can be confused with directivity, since they are equivalent for lossless antennas. Gain is related to directivity by the radiation efficiency of the antenna. If the radiation efficiency is 100%, they are equal.

Note Because the gain is calculated from the input signal at the port, a port must be defined for this quantity to be displayed.

Related Topics

Setting up a Far-Field Infinite Sphere

Computing Antenna Parameters

Peak Realized Gain

Realized gain is four pi times the ratio of an antenna's radiation intensity in a given direction to the total power incident upon the antenna port(s). Peak realized gain, in turn, is the maximum realized gain over all the user-specified directions of the far-field infinite sphere.

The following equation is used to calculate realized gain in HFSS:

$$\text{realized gain} = 4\pi\frac{U}{P_{incident}}$$

where

- U is the radiation intensity in watts per steradian in the direction specified.
- $P_{incident}$ is the incident power in watts.

Note Because the gain is calculated from the input signal at the port, a port must be defined for this quantity to be displayed.

Related Topics

Setting up a Far-Field Infinite Sphere

Computing Antenna Parameters

Radiated Power

Radiated power is the amount of time-averaged power (in watts) exiting a radiating antenna structure through a radiation boundary.

For a general radiating structure in HFSS, radiated power is computed as

$$P_{rad} = \Re \int_{s} \boldsymbol{E} \times \boldsymbol{H}^{*} \cdot ds$$

where

- P_{rad} is the radiated power in watts.
- $\Re$ is the real part of a complex number.
- s represents the radiation boundary surfaces.
- E is the radiated electric field.
- H^{*} is the conjugate of **H**.
- ds is the local radiation-boundary unit normal directed out of the 3D model.

Note The accuracy of the computed radiated power depends on the accuracy of ***E*** and ***H***. In some cases it is possible that the computed radiated power may deviate slightly from the actual radiated power.

The accuracy of the computed radiated power depends on the accuracy of E and H on the absorbing boundary. In some cases it is possible that the computed radiated power may deviate slightly from the actual radiated power. To increase the accuracy of the radiated power, seed the mesh on the absorbing boundary. As a check, you can use the S-parameters — if ports have been defined — to calculate the radiated power.

Related Topics

Computing Antenna Parameters

Accepted Power

The accepted power is the amount of time-averaged power (in watts) entering a radiating antenna structure through one or more ports. For antennas with a single port, accepted power is a measure of the incident power reduced by the mismatch loss at the port plane.

For a general radiating structure in HFSS, accepted power is computed as

$$P_{acc} = \Re \int_{A} \boldsymbol{E} \times \boldsymbol{H}^{*} \cdot ds$$

where

- P_{acc} is the accepted power in watts.
- $\Re$ is the real part of a complex number.
- A is the union of all port boundaries in the model.
- E is the radiated electric field.
- H^{*} is the conjugate of **H**.

- *ds* is the local port-boundary unit normal directed into the 3D HFSS model.

For the simple case of an antenna with one lossless port containing a single propagating mode, the above expression reduces to

$$P_{acc} = |a|^2(1 - |s_{11}|^2)$$

where

- *a* is the complex modal excitation specified.
- s_{11} is the single-entry generalized scattering matrix (without renormalization) computed by HFSS.

Note Because the accepted power is calculated from the input signal at the port, a port must be defined for this quantity to be displayed.

Related Topics

Computing Antenna Parameters

Incident Power

Incident power is the total amount of time-averaged power (in watts) incident upon all port boundaries of an antenna structure. Incident power is set at your discretion in the **Edit Sources** window.

For the simple case of an antenna with one lossless port containing a single propagating mode, the incident power $P_{incident}$ is given by

$$P_{incident} = |a|^2$$

where

- *a* is the complex modal-project excitation specified in the **Edit Sources** window.

Note Because input power is calculated from the input signal at the port, a port must be defined for incident power to be displayed.

Related Topics

Computing Antenna Parameters

Radiation Efficiency

The radiation efficiency is the ratio of the radiated power to the accepted power given by

$$e = \frac{P_{rad}}{P_{acc}}$$

where

- P_{rad} is the radiated power in watts.

- P_{acc} is the accepted power in watts.

Note Because the radiation efficiency is calculated from the accepted power, a port must be defined for radiation efficiency to be displayed.

Related Topics

Computing Antenna Parameters

Calculating Finite Thickness Impedance

The **Assign DC Thickness** option on the **HFSS** menu is enabled if at least one object contains a good conducting isotropic material (such as copper), and the **Solve Inside property** is not selected. If the object meets these conditions, you can assign a DC thickness. If the thickness of the layer is finite, the skin impedance is calculated as:

$$Z = R = jX$$

$$R = \frac{1}{\sigma\delta}\frac{sh(2v) + \sin(2v)}{ch(2v) - \cos(2v)}$$

$$X = \frac{1}{\sigma\delta}\frac{sh(2v) - \sin(2v)}{ch(2v) - \cos(2v)}$$

where

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

$$v = \frac{h}{\delta}$$

where h is the layer thickness.

Similar skin impedance is assigned to surfaces of 3D objects of good conductors, which are of NoSolveInside and Thickness for DC Resistance is set

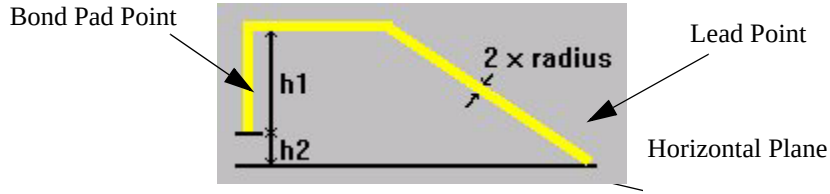
Geometric Objects

Following are supplemental technical details about the following geometric objects:

- Bondwires
- Healing Models

Bondwires

A bondwire is a thin metal wire that connects a metal signal trace with a chip. You can choose to draw a standard JEDEC 4-point bondwire, as shown below:



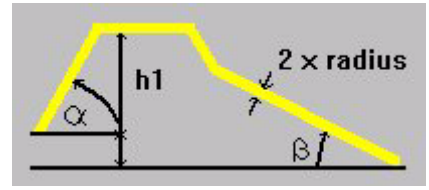
where

h1 = the height between the bond pad point and the top of the loop.

h2 = the height between the lead point and the bond pad point.

radius = half the diameter, or thickness of the wire.

Or you can choose to draw a standard JEDEC 5-point bondwire, as shown below:



where

α = the angle between the horizontal plane and the wire at the bond pad point.

β = the angle between the horizontal plane and the wire at the lead point.

When drawing the bondwire, you will first select the bond pad point, a point in 3D space that defines the bond pad position in a horizontal plane. Then you will select the lead point, which indicates the distance the wire covers in the horizontal plane. HFSS will use the distance between the bond pad and lead points to calculate the height between the bond pad and the lead point, or *h2*, a value that you can modify in the **Bondwires** dialog box.

Related Topics

Drawing Bondwires

Healing Models

The underlying solid modeling technology used by Ansoft products is provided by ACIS geometric modeler. Users can create directly models using primitives and operations on primitives. In addition, users can import models saved in a variety of formats (Step, IGES, etc.) All the models are stored internally in ACIS native format (sat format). When users import models into Ansoft products, translators are invoked that convert the models to sat format. ACIS regularly upgrades their solid modeling system and different versions of Ansoft products like HFSS, Maxwell, Q3D use different versions of ACIS. Third party vendors also create files in ACIS sat format but the data in the files is in some cases not robust.

All of the above contribute to errors when a model is read. When translating from (say) IGES to sat, some accuracy is lost. Not all third party vendors write files to the supported formats with a high level of accuracy. Third party vendors sometimes create invalid sat files – the sat files are either targeted at earlier versions of ACIS or incorporate invalid elements in the sat file. Sometimes there are errors in reading a file generated by an earlier version of ACIS with the current version of ACIS (eg) reading HFSS 9.2 files in HFSS 10 sometimes causes errors.

There is a final issue with using ACIS as the modeler when working with imported models. ACIS can handle mixed dimensionality models. It works with manifold and non-manifold bodies as well as sheet bodies, wire bodies and solid bodies. One of the goals of our use of the ACIS modeling system is to create a valid volumetric mesh for simulation. Mixed dimensionality models will not yield a valid volumetric mesh. If users use Ansoft products exclusively to create geometry models, they are less likely to create invalid models. Imported models are more likely to have errors in geometry and topology definition.

Related Topics

Technical Notes: Error Types

Technical Notes: Error Detection

Analyze Objects

Analyze Interobject Misalignment

Analyze Surface Mesh

Healing

Validating Projects

Error types

When models are imported there are two types of errors – geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors are errors in how the underlying components like faces, edges and vertices are connected. These have to be fixed before mesh analysis can be performed.

When models pass the initial validity checks, mesh generation could still fail. If bodies in the model overlap, mesh will not be invoked. If bodies are very close to each other, mesh might fail. Small features (small edges, small edges, sliver edges) might be present in the bodies which might cause mesh to fail.

The following are common errors present in models :-

1. api_check_entity() errors. These are errors detected by ACIS and are geometry and topology errors.
2. non-manifold topology. These are non-manifold edges and vertices that are present in the model.
3. Body pair intersection. This detects if pairs of bodies intersect.
4. Small feature detection – small edge length, small face area and sliver face detection.
5. Mis-aligned entities detection – detects pairs of faces from bodies that can be aligned to remove interbody intersections. This improves the odds of mesh success.
6. Mesh failure error display. This is available for single body, body pairs and last simulation run (all bodies in model). Errors reported by the meshing module are reported to the user.

Errors of type 1, 2 and 3 must be resolved before the mesh can be invoked on the model.

Error detection

Errors in the model can be detected by the following methods.

1. You can perform validation check (**HFSS->Validation check**). This will return the following types of errors -- api_check_entity() errors, non-manifold errors and intersection errors. This works on all the bodies present in the model.
2. You can invoke Object Analysis (**3D Modeler ->Model Analysis->Analyze Objects**) on a subset of bodies in the model. This will return api_check_entity() errors and non-manifold edge/vertex errors, and optionally small edge, small face and sliver face detection errors.
3. You can invoke Interobject Misalignment Analysis (**3D Modeler ->Model Analysis->Analyze Interobject Misalignment**). This will yield face pairs from different bodies that are slightly misaligned with respect to each other. This misalignment causes the faces to intersect each other or to create small volumetric portion between the bodies that might cause mesh to fail. If these faces are aligned--that is, the faces are made to share the same surface definition--the gap between the faces is eliminated and there is a higher likelihood that mesh will succeed.
4. You can invoke Surface mesh analysis (**3D Modeler ->Model Analysis->Analyze Surface Mesh**). For the selected bodies, mesh in invoked on each individual body and optionally body pairs. Errors from running mesh analysis are displayed. This will help the user to focus on bodies/body pairs that fail meshing. If these are fixed, there is a higher probability that mesh will succeed.
5. You can display mesh errors from last simulation run. Errors from the last simulation run are displayed to the user.

Errors for 2,3,4 and 5 are displayed in the **Model analysis** dialog. Invoking the menu items for 2, 3 and 4 will display the dialog. The dialog can also be invoked by 3**D Modeler->Model Analysis->Show Analysis Dialog.**

Boundaries

Boundary conditions specify the field behavior on the surfaces of the problem region and object interfaces. This area of the technical notes includes information about the following boundary types:

- Perfect E
- Impedance
- Radiation
- PML
- Finite Conductivity
- Symmetry
- Master and Slave
- Lumped RLC
- Layered Impedance
- Infinite Ground Planes

and the following subjects:

- Frequency-Dependent Boundaries
- Default Boundary Assignments

Perfect E Boundaries

In HFSS, perfect E boundaries represent perfectly conducting surfaces in a structure.

By default, all HFSS model surfaces exposed to the background are assumed to have perfect E boundaries; HFSS assumes that the entire structure is surrounded by perfectly conducting walls. The electric field is assumed to be normal to these surfaces. The final field solution must match the case in which the tangential component of the electric field goes to zero at perfect E boundaries.

The surfaces of all model objects that have been assigned perfectly conducting materials are automatically assigned perfect E boundaries.

Impedance Boundaries

In HFSS, impedance boundaries represent surfaces of known impedance. The behavior of the field at the surface and the losses generated by the currents flowing on the surface are computed using analytical formulas; HFSS does not actually simulate any fields inside the resistor.

Similar to finite conductivity boundaries, the following condition applies at impedance boundaries:

$$\boldsymbol{E}_{tan} = Z_s(\hat{n} \times \boldsymbol{H}_{tan})$$

where

- $\hat{n}$ is the is the unit vector that is normal to the surface.
- E_{tan} is the component of the E-field that is tangential to the surface.

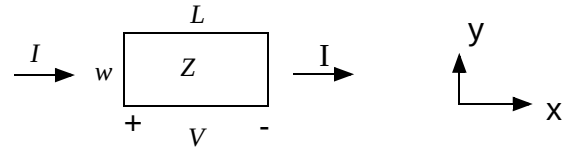
- H_{tan} is the component of the H-field that is tangential to the surface.
- Z_s is the surface impedance of the boundary, $Rs + jXs$, where
  - Rs is the resistance in ohms/square.
  - Xs is the reactance in ohms/square.

For example, assume that a structure contains two dielectrics separated by a thin-film resistor. This resistor could be represented by an impedance boundary at the surface between the two objects.

Units of Impedance Boundaries

Impedance on the surface of objects, Z_s, has units of ohms per square. The units *ohms per square* indicate that the impedance, Z_s, is equal to the equivalent circuit impedance, Z, measured between the edges of a square sheet of the material.

For example, a rectangle of length *L* and width *w* has a uniform current, *I*, applied to it. It has a voltage drop, *V*, across it and an equivalent circuit impedance of *Z* ohms.



If the current density, *J*, is uniform over the rectangle then the equation $\hat{n} \times \vec{E} = Z_s \hat{n} \times \vec{J}$ becomes

$$E = Z_s J \tag{3}$$

where

- $E = \left|\vec{E}\right|$ on the rectangle, and
- $J = \left|\vec{J}\right|$ on the rectangle.

The circuit quantities and fields are related as follows:

$$V = \int_{x=0}^{L} \vec{E} \cdot \overrightarrow{dL} = EL$$

$$I = \int_{y=0}^{w} \vec{J} \cdot \hat{x} dy = Jw$$

$$Z = \frac{V}{I} = \frac{EL}{Jw}$$

Substituting equation (1) into equation (2) results in the following equation:

$$Z = Z_s \frac{L}{w}$$

Thus, when $L = w$, the equivalent circuit impedance is equal to the impedance on one square. Hence the units *ohms per square*.

If in this example $L = 2w$, the impedance would be equal to one-half of the circuit equivalent impedance for the rectangle, or the circuit equivalent impedance of one "square" of the rectangle is equal to the impedance of that square. Therefore, when entering the surface impedance for an object, you must enter the impedance per square.

Radiation Boundaries

In HFSS, radiation boundaries are used to simulate open problems that allow waves to radiate infinitely far into space, such as antenna designs. HFSS absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure.

At radiation boundary surfaces, the second-order radiation boundary condition is used:

$$(\nabla \times \boldsymbol{E})_{tan} = jk_0 \boldsymbol{E}_{tan} - \frac{j}{k_0}\nabla_{tan} \times (\nabla_{tan} \times \boldsymbol{E}_{tan}) + \frac{j}{k_0}\nabla_{tan}(\nabla_{tan} \bullet \boldsymbol{E}_{tan})$$

where

- E_{tan} is the component of the E-field that is tangential to the surface.
- k_0 is the free space phase constant,

  $\omega\sqrt{\mu_0 \varepsilon_0}$.
- j is $\sqrt{-1}$.

The second-order radiation boundary condition is an approximation of free space. The accuracy of the approximation depends on the distance between the boundary and the object from which the radiation emanates.

PML Boundaries

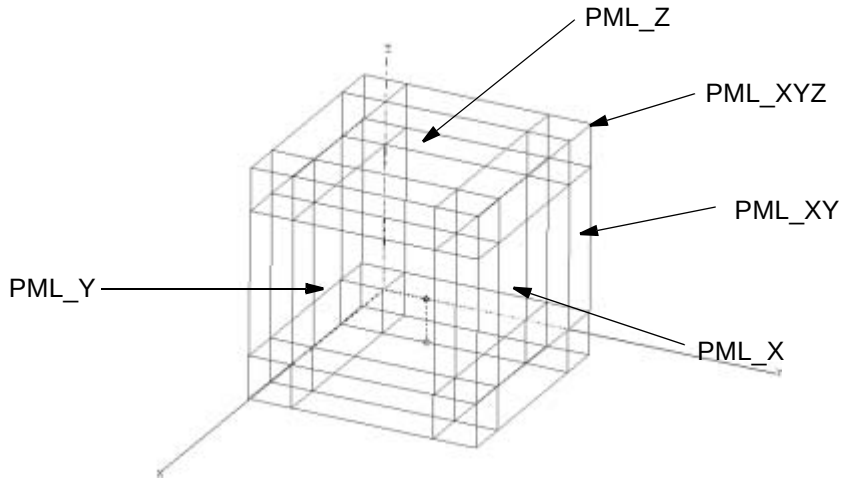
Perfectly matched layers (PMLs) are fictitious materials that fully absorb the electromagnetic fields impinging upon them. These materials are complex anisotropic.

There are two types of PML applications: free space termination and reflection-free termination. With free space termination, PMLs are associated with a surface that radiates into free space equally in every direction. PMLs are more appropriate than radiation boundaries in this case because PMLs enable radiation surfaces to be located closer to radiating objects, reducing the problem domain. Any homogenous isotropic material, including lossy materials like ocean water, can surround the design.

With reflection-free termination of guided waves, the structure continues uniformly to infinity. Its termination surface radiates in the direction in which the wave is guided. Reflection-free PMLs are appropriate for simulating phased array antennas because the antenna radiates in a certain direction.

Material Tensors Applied at PML Boundaries

PMLs materials are complex anisotropic. An example is shown below.



To ensure that there will not be any reflection at the PML/air interface, the bi-axial diagonal material tensors for x-, y- and z-directed PMLs (PML_X, PML_Y, and PML_Z) are as follows.

For PML_X:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} \frac{1}{C} & C & C \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} \frac{1}{C} & C & C \end{bmatrix}$$

For PML_Y:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} C & \frac{1}{C} & C \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} C & \frac{1}{C} & C \end{bmatrix}$$

For PML_Z:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} C & C & \frac{1}{C} \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} C & C & \frac{1}{C} \end{bmatrix}$$

where $C = a - jb$.

The tensors designated as PML_X characterize an x-directed PML corresponding to a PML wall in the yz plane. Similarly, PML_Y and PML_Z are designated tensors for y- and z-directed PMLs.

PMLs of different directions must be joined in order to construct a box with PML walls. To ensure complete coverage where the edges and corners of two PMLs meet, create edge and corner PML objects. The tensors of an edge object joining PML_X and PML_Y are as follows for PML_XY:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} 1 & 1 & C^2 \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} 1 & 1 & C^2 \end{bmatrix}$$

A similar tensor construction rule is valid for joining x- and z-directed and y- and z-directed PMLs. The tensor for a corner object is a follows for PML_XYZ:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} C & C & C \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} C & C & C \end{bmatrix}$$

Tensor Entries

Entering the matrices of the anisotropic materials doesn't require a special procedure. The usual anisotropic material definitions can be used for any PML structure. However, keep in mind that the efficiency of the PMLs depends on the material values assigned to them.

Setting the complex parameter C ensures that the electromagnetic field decays strongly in the PMLs. Back reflections from the bounding PECs are then kept below a prescribed bound. To accomplish this, the following inequalities have to be satisfied:

$$e \geq \frac{-\ln \rho}{2 D_{min} H} = e_{min}$$

$$e \leq \frac{-\ln d}{2 D_{max} h} = e_{max}$$

where

$$D_{min} = \alpha_{min} + \beta_{min} = \frac{1}{r_{max}} + \frac{\omega_{min}}{c}$$

$$D_{max} = \alpha_{max} + \beta_{max} = \frac{1}{r_{min}} + \frac{\omega_{max}}{c}$$

- $e = a = b$
- a and b are the real and imaginary parts of C.
- H is the thickness of the PML object.
- ω_{max} and ω_{min} are the minimum and maximum angular frequencies.
- r_{max} and r_{min} are the minimum and maximum distance of a radiating object to the PML surface.
- ρ is the bound for back reflection.
- d is the maximum decay characterizing the element. (d is approximately $3 \cdot 10^{-3}$.)
- h is the thickness of one finite element.
- c is the velocity of light in vacuum.

Boundaries at PML Surfaces

After embedding a structure in PMLs, the next step is to specify boundaries on the outer surface of the box. The simplest way is to bound the box either with perfect electric conductors (PECs) or perfect magnetic conductors (PMCs.) In general, use PECs because they reduce the problem size.

Finite Conductivity Boundaries

In HFSS, finite conductivity boundaries represent imperfect conductors. At such boundaries, the following condition holds:

$$\boldsymbol{E}_{tan} = Z(\hat{n} \times \boldsymbol{H}_{tan})$$

where

- E_{tan} is the component of the E-field that is tangential to the surface.
- H_{tan} is the component of the H-field that is tangential to the surface.
- Z_s is the surface impedance of the boundary, $(1 + j)/(\delta\sigma)$, where
  - δ is the skin depth, $\sqrt{2/(\omega\sigma\mu)}$, of the conductor being modeled.
  - ω is the frequency of the excitation wave.
  - σ is the conductivity of the conductor.
  - μ is the permeability of the conductor.

The fact that the E-field has a tangential component at the surface of imperfect conductors simulates the case in which the surface is lossy.

The surfaces of any objects defined to be non-perfect conductors are automatically set to finite conductivity boundaries. Note that HFSS does not attempt to compute the field inside these objects; the finite conductivity boundary approximates the behavior of the field at the surfaces of the objects.

The finite conductivity boundary condition is valid only if the conductor being modeled is a good conductor, that is, if the conductor's thickness is much larger than the skin depth in the given frequency range. If the conductor's thickness is in the range or larger than the skin depth in the given frequency range, HFSS's layered impedance boundary condition must be used.

Symmetry Boundaries

In HFSS, symmetry boundaries represent perfect E or perfect H planes of symmetry. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time.

When you are defining a symmetry plane, keep the following requirements in mind:

- A plane of symmetry must be exposed to the background.
- A plane of symmetry must not cut through an object drawn in the **3D Modeler** window.
- A plane of symmetry must be defined on a planar surface.
- Only three orthogonal symmetry planes can be defined in a problem.

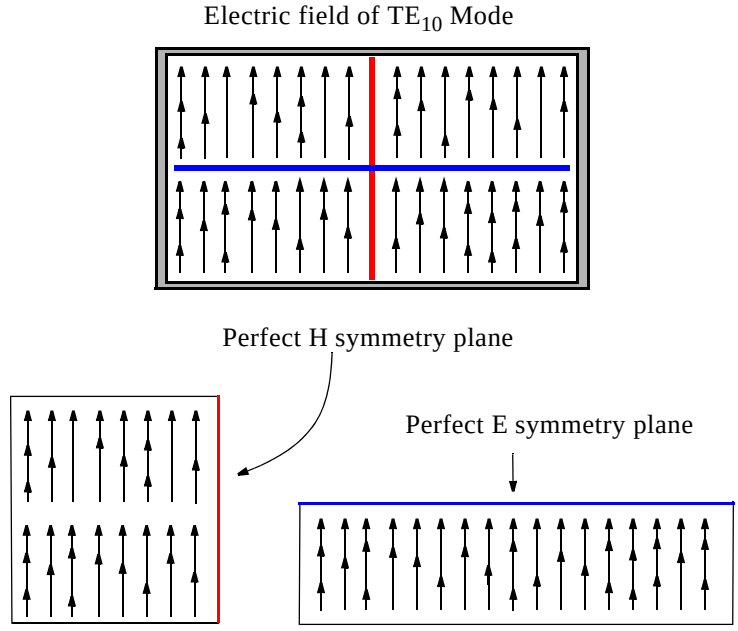
Perfect E Vs. Perfect H Symmetry Boundaries

In general, use the following guidelines to decide which type of symmetry boundary to use, a perfect E or a perfect H:

- If the symmetry is such that the E-field is normal to the symmetry plane, use a perfect E symmetry plane.
- If the symmetry is such that the E-field is tangential to the symmetry plane, use a perfect H symmetry plane.

The simple rectangular waveguide shown below illustrates the differences between the two types of boundaries. The E-field of the dominant mode signal (TE_{10}) is shown. The waveguide has two planes of symmetry, one vertically through the center and one horizontally.

The horizontal plane of symmetry is a perfect E surface. The E-field is normal and the H-field is tangential to that surface. The vertical plane of symmetry is a perfect H surface. The E-field is tangential and H-field is normal to that surface.



For common problems, you can usually decide which symmetry boundary to use by reviewing the geometry. For example, if the structure is a microstrip, the flux lines of the E-field run between the ground plane and the conductive strip; therefore, the E-field is tangential to any vertical symmetry plane that slices a microstrip in half.

Symmetry and Port Impedance

If a symmetry plane has been defined, the computed port impedances will not match the port impedance of the full structure unless an impedance multiplier is specified.

Note Port impedance is only calculated when a port has been defined. If you are solving a problem without ports, you do not need to specify an impedance multiplier.

Symmetry and Multiple Modes

If you are solving for multiple modes, keep in mind that the orientation of the E- and H-fields may differ from mode to mode. A perfect H symmetry boundary for the dominant mode may be a perfect E symmetry for another mode.

Master and Slave Boundaries

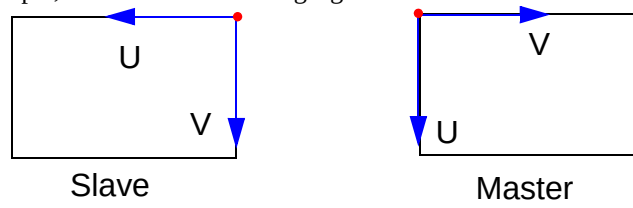
Master and slave boundaries enable you to model planes of periodicity where the E-field on one surface matches the E-field on another to within a phase difference. They force the E-field at each point on the slave boundary match the E-field to within a phase difference at each corresponding point on the master boundary. They are useful for simulating devices such as infinite arrays.

Unlike symmetry boundaries, **E** does not have to be tangential or normal to these boundaries. The only condition is that the fields on the two boundaries must have the same magnitude and direction (or the same magnitude and opposite directions).

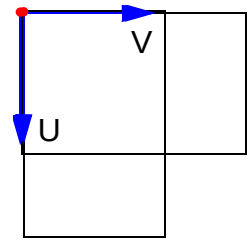
When creating matching boundaries, keep the following points in mind:

- Master and slave boundaries can only be assigned to planar surfaces. These may be the faces of 2D or 3D objects.
- The geometry of the surface on one boundary must match the geometry on the surface of the other boundary. For example, if the master is a rectangular surface, the slave must be a rectangular surface of the same size.
- If the mesh on the master boundary does not match the mesh on the slave boundary exactly, the solution will fail. Normally HFSS automatically forces the mesh to match on each boundary; however, in some cases, the mesh cannot be forced to match. To prevent the solution from failing, create a virtual object on the slave boundary that exactly matches any extra object on the master boundary, or create a virtual object on the master boundary that exactly matches any extra object on the slave boundary.
- To make a surface a master or slave boundary, you must specify a coordinate system that defines the plane on which the selected surface exists. When HFSS attempts to match the two boundaries, the two coordinate systems must also match each other. If they do not, HFSS will transpose the slave boundary to match the master boundary. When doing this, the surface to which the slave boundary is assigned is also transposed. If, after doing this, the two surfaces do not occupy the same position relative to their combined defined coordinate system, an error message appears.

For example, consider the following figure:



To match the coordinate system of the master boundary, the coordinate system on the slave boundary must rotate 90 degrees counterclockwise; however, when this is done, you get the following:



The two surfaces do not correspond and thus the mesh will not match, causing an error message.

- The angle between the axes defined by the u point and v point must be identical for the master and slave boundary.

Calculating the E-Field on the Slave Boundary

The E-field on the slave boundary is forced to match the E-field on the master boundary. The magnitude of the E-field on both boundaries is the same; however, the fields may be out of phase with each other.

The function relating the electric field on the slave boundary, E_S, to the electric field on the master boundary, E_M, depends on the type of problem you are solving. For example, consider an infinite array simulation for a rectangular array. If the array excited to radiate in the direction (θ, ϕ) in spherical coordinates. The fields above the array experience a phase delay of

$$\Psi = k(\hat{r}_o \bullet \boldsymbol{v})$$

where

- $\hat{r}_0$ is the unit vector in the direction of scan.
- $\boldsymbol{v}$ is the vector from the slave boundary to the master boundary.

To simulate this in the finite element solution, HFSS incorporates phase shifts in the relation between the matching boundaries. That is, the electric field values on the master boundary will be

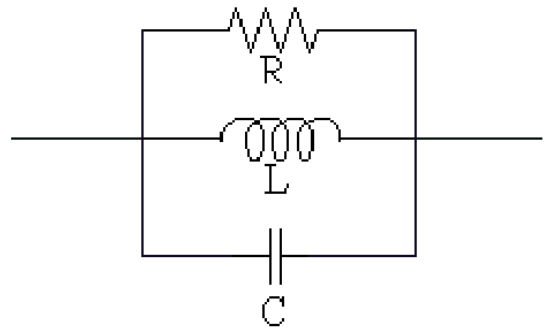
related to the electric field values on the corresponding points on the slave boundary. This equation would be the following:

$$\boldsymbol{E}_S = e^{j\Psi}\boldsymbol{E}_M$$

HFSS gives you the option of entering the scan angles, ϕ and θ, when relating E_S to E_M. The phase delay is calculated from the scan angles. However, if you know the phase delay, you may enter that directly.

Lumped RLC Boundaries

To model any combination of lumped resistor, inductor, and/or capacitor in parallel on a surface, create a lumped RLC boundary. A lumped RLC boundary represents R, L, and C in parallel:



Similar to impedance boundaries, the following condition holds at lumped RLC boundaries:

$$\boldsymbol{E}_{tan} = Z_s(\hat{n} \times \boldsymbol{H}_{tan})$$

where

- $\hat{n}$ is the is the unit vector that is normal to the surface.
- E_{tan} is the component of the E-field that is tangential to the surface.
- H_{tan} is the component of the H-field that is tangential to the surface.
- Z_s is the surface impedance of the boundary, $Rs + jXs$, where
  - Rs is the resistance in ohms/square.
  - Xs is the reactance in ohms/square.

Unlike impedance boundaries, you are not required to supply the impedance per square, but you must supply the actual values for R, L, and C. HFSS then determines the impedance per square of the lumped RLC boundary at any frequency.

A Fast frequency sweep is supported for this boundary condition.

Layered Impedance Boundaries

A layered impedance boundary is used to model multiple layers in a structure as one impedance surface. The effect is the same as an impedance boundary condition, except that HFSS calculates

the reactance and resistance values for the surface based on data you enter for the layered structure. Surface roughness is also taken into account.

The reactance and resistance values are calculated differently for internal and external layered impedance boundaries. For external layered impedance boundaries, HFSS calculates the impedance for the side of the surface in contact with the computational domain and assigns this value to the boundary. For internal layered impedance boundaries, HFSS calculates the average impedance value for the two sides of the surface in contact with the computational domain and assigns this value to the boundary.

The layered impedance boundary is supported for single-frequency solutions and Discrete and Interpolating frequency sweeps.

Impedance Calculation for Layered Impedance Boundary

The impedance of the layered structure is calculated by recursively calling the impedance calculation formulation known from transmission line theory:

$$Z_{inputk} = Z_{wk}\frac{Z_{inputk+1}ch(\gamma_k d_k) + Z_{wk}sh(\gamma_k d_k)}{Z_{inputk+1}sh(\gamma_k d_k) + Z_{wk}ch(\gamma_k d_k)}$$

where

- Z_{inputk} is the input impedance for the k^{th} layer.
- $Z_{wk} = \sqrt{\frac{\mu_0\mu_{rk}}{\varepsilon_0\mu\varepsilon_{rk}}}$
- *ch* is the hyperbolic cosine function.
- *sh* is the hyperbolic sine function.
- $\gamma_k = k_0\sqrt{-\varepsilon_{rk}\mu_{rk}}$

  where
  - γ is the propagation coefficient.
  - *k0* is the free space wave number, $\omega\sqrt{\mu_0\varepsilon_0}$, where ω is the angular frequency, $2\pi f$.
  - ε_{rk} is the relative complex permittivity of the K^{th} layer.
  - μ_{rk} is the relative complex permeability of the K^{th} layer.

  where
  - $\varepsilon_{rk} = epsr_k - j\left(\frac{sigma}{\omega\varepsilon_0} + epsr \cdot \tan de\right)$

- $\mu_{rk} = mur_k - j(mur_k \cdot \tan dm_k)$

- d_k is the thickness of the K^{th} layer.

Surface Roughness Calculation for Layered Impedance Boundary

The surface roughness is measured as the RMS deviation of the conductor surface from a plane. Surface roughness increases conduction losses. Ansoft HFSS calculates surface roughness by modifying the conductivity as follows:

$$\sigma_c = \frac{\sigma}{K_w^2}$$

where

σ is the material's conductivity.

$$K_w = 1 + \exp\left(-\left(\frac{s}{2h}\right)^{1.6}\right)$$

where, further:

- h is the surface roughness.
- s is the skin depth.

Infinite Ground Planes

To simulate the effects of an infinite ground plane, select the **Infinite ground plane** check box when setting up a perfect E, finite conductivity, or impedance boundary condition. The selection only affects the calculation of near- and far-field radiation during post processing. The 3D Post Processor models the boundary as a finite portion of an infinite, perfectly conducting plane.

Conceptually, a boundary condition designated as an infinite ground plane divides the problem region into the half above it, where the entire model resides, and the half below it, where the radiated fields are set to zero. Antenna parameters involving radiated power will be consistent with these properties.

Lossy ground planes may be approximated by selecting the **Infinite ground plane** check box when assigning a finite conductivity or impedance boundary. The effects of these boundaries are incorporated into the field solution in the usual manner, but the radiated fields in the 3D Post Processor are computed as if the lossy ground planes were perfectly conducting.

When defining an infinite ground plane, keep the following requirements in mind:

- An infinite ground plane in a model must be exposed to the background.
- An infinite ground plane must be defined on a planar surface.
- The total number of infinite ground planes and symmetry planes cannot exceed three.
- All infinite ground planes and symmetry planes must be mutually orthogonal.

Frequency-Dependent Boundaries and Excitations

In general, boundary and excitation parameters cannot depend on intrinsic functions. An exception is when a parameter depends on the variable *Freq*, which represents the solution frequency. The following boundary parameters can be assigned an expression that includes *Freq*:

- Impedance boundary - the Resistance and Reactance parameters.
- Finite conductivity boundary - the Conductivity parameter. If a material is specified, the material can be frequency dependent.
- Slave boundary - the Phase parameter.
- Lumped RLC boundary - Resistance, Inductance, and Capacitance parameters.
- Layered impedance boundary - materials assigned on layers can be frequency dependent.

Note Dependence on *Freq* is supported for single-frequency solutions and for Discrete and Interpolating frequency sweeps. If a Fast sweep is requested, the solution will be valid for the center frequency, but may not be valid at other frequencies.

Default Boundary Assignments

If a boundary has not been assigned to a model surface, one of the following default boundaries will be assigned to the surface:

smetal	A single perfect E boundary is assigned to all objects that do not have **Solve Inside** selected in the **Properties** window and that are perfect conductors.
i_<*object name*>	A finite conductivity boundary is assigned to each object that does not have **Solve Inside** selected in the **Properties** window and that is not a perfect conductor. <*object name*> is the name of the object on which the boundary is assigned.
outer	A default boundary applied on the outermost surfaces of the model.

Related Topics

Reviewing Boundaries and Excitations in the Solver View

Excitations

Assigning excitations to an HFSS design enables you to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces. This area of the Technical Notes includes information on the following topics:

- Wave Ports
- Polarizing the E-Fields
- Lumped Ports
- Setting the Field Pattern Direction
- Differential Pairs
- Magnetic Bias Sources
- Incident Waves

Wave Ports

By default, the interface between all 3D objects and the background is a perfect E boundary through which no energy may enter or exit. Wave ports are typically placed on this interface to provide a window that couples the model device to the external world.

HFSS assumes that each wave port you define is connected to a semi-infinitely long waveguide that has the same cross-section and material properties as the port. When solving for the S-parameters, HFSS assumes that the structure is excited by the natural field patterns (modes) associated with these cross-sections. The 2D field solutions generated for each wave port serve as boundary conditions at those ports for the 3D problem. The final field solution computed must match the 2D field pattern at each port.

HFSS generates a solution by exciting each wave port individually. Each mode incident on a port contains one watt of time-averaged power. Port 1 is excited by a signal of one watt, and the other ports are set to zero watts. After a solution is generated, port 2 is set to one watt, and the other ports to zero watts and so forth.

Within the 3D model, an internal port can be represented by a lumped port. Lumped ports compute S-parameters directly at the port. The S-parameters can be renormalized and the Y-matrix and Z-matrix can be computed. Lumped ports have a user-defined characteristic impedance.

Polarizing the E-Fields

In some cases, such as when a port is square or circular, not only is the *positive* and *negative* direction in question, the line with which the E-field is aligned is also arbitrary.

For example, in the case of a square waveguide, the E-field of the dominant mode can be aligned horizontally, vertically, or diagonally within the guide. There is no preferred direction. However, HFSS aligns the field with the defined integration line if you select **Polarize E Field**.

Circular waveguides also require a polarized E-field. The direction of the E-field at $\omega t = 0$ can point in any direction. To align the simulated field with a preferred direction, define an integration line and select **Polarize E Field**. In this case, the integration line must lie in the middle of the port, that is, in the symmetry plane.

When polarizing the E-fields, observe the following guidelines. Otherwise the results may not be as expected.

- Polarize the E-field only on square or circular waveguides.
- Make sure the port on the waveguide only feeds a single conductor (the waveguide wall.)
- Do not polarize the E-fields if you are using a symmetry boundary. The polarization is automatically enforced by the symmetry boundary condition.

Lumped Ports

Lumped ports are similar to traditional wave ports, but can be located internally and have a complex user-defined impedance. Lumped ports compute S-parameters directly at the port. Use lumped ports for microstrip structures.

A lumped port can be defined as a rectangle from the edge of the trace to the ground or as a traditional wave port. The default boundary is perfect H on all edges that do not come in contact with the metal.

The complex impedance Z_S defined for a lumped port serves as the reference impedance of the S-matrix on the lumped port. The impedance Z_S has the characteristics of a wave impedance; it is used to determine the strength of a source, such as the modal voltage *V* and modal current *I*, through complex power normalization. (The magnitude of the complex power is normalized to 1.) In either case, you would get an identical S-matrix by solving a problem using a complex impedance for a lumped Z_S or renormalizing an existing solution to the same complex impedance.

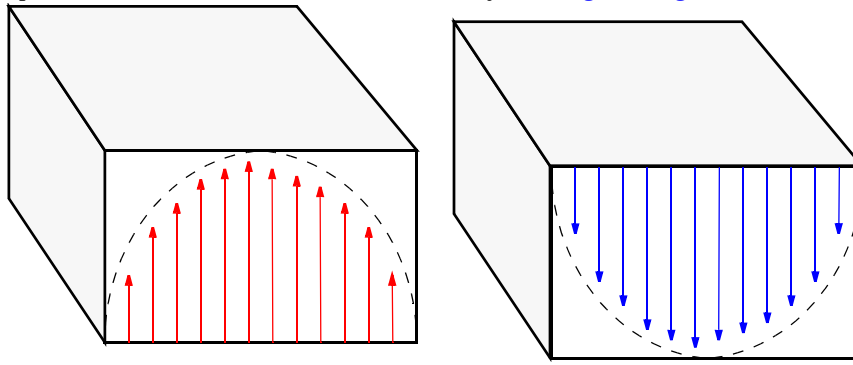
When the reference impedance is a complex value, the magnitude of the S-matrix is not always less than or equal to 1, even for a passive device.

Note When a lumped port is used as an internal port, the conducting cap required for a traditional wave port must be removed to prevent short-circuiting the source.

Setting the Field Pattern Direction

When HFSS computes the excitation field pattern at a port, the direction of the field at $\omega t = 0$ is arbitrary; the field can always point in one of at least two directions.

In the figure below, the mode 1 field at ωt = 0 can either point to the left or to the right. Either direction is correct — unless a preferred direction is specified. To specify a direction, you must calibrate the port relative to some reference orientation by defining an integration line.



In the case of rectangular waveguides, visualize the difference in terms of a physical connection. If the *up* side of a port is aligned with the *up* side of the waveguide carrying the excitation signal, the signal at the port is in phase with what is expected. But if the *up* side of the port is connected to the *down* side of the waveguide, the incoming signal will be out of phase with the expected signal. Likewise, it is desirable to define which way is up at all ports on a structure; otherwise, the resulting S-parameters can be shifted from the expected orientation.

Calibrate a port to define a preferred direction at each port relative to other ports having identical or similar cross-sections. In this way, the results of laboratory measurements, in which the setup is calibrated by removing the structure and connecting two ports together, can be duplicated.

Note Because integration lines can determine the phase of the excitation signal and traveling wave, they are ignored by HFSS when a ports-only solution is requested.

Differential Pairs

A differential pair represents two circuits, one positive and one negative, routed close together so they will pick up nearly the same amount of noise. The two signals are subtracted from each other by a receiver, yielding a much more noise-free version of the signal.

You can compare the noise rejection of a differential pair to that of a conventional “single-ended” signal and alter the differential pair’s terminal Z_o to determine its best reference impedance value.

You can define a series of differential pairs from terminal voltage lines defined on existing ports. You must have defined two terminal lines on a single port for this command to be active.

Computing Differential Pairs

To compute the differential and common voltages v_d and v_c of a terminal pair on a shared port rather than the single-ended voltages v_1 and v_2, define a differential pair in the **Wave Port** dialog box.

The differential and common voltages v_d and v_c are defined by

$$v_d = v_1 - v_2$$

$$v_c = \frac{v_1 + v_2}{2}$$

(1)

Consistent with power conservation, the corresponding differential and common currents, represented as i_d and i_c respectively, are defined by

$$i_d = \frac{i_1 - i_2}{2}$$

$$i_c = i_1 + i_2$$

(2)

Equations (1) and (2) can be concisely represented as

$v = Qe$

$i = Q^{-T}u$

(3)

where

- $v = \begin{bmatrix} v_1 \\ v_2 \end{bmatrix}$
- $i = \begin{bmatrix} i_1 \\ i_2 \end{bmatrix}$
- $e = \begin{bmatrix} v_d \\ v_c \end{bmatrix}$
- $u = \begin{bmatrix} i_d \\ i_c \end{bmatrix}$

- Q is the real, non-singular matrix defined by

$$Q = \begin{bmatrix} \frac{1}{2} & 1 \\ -\frac{1}{2} & 1 \end{bmatrix} \tag{4}$$

- Q^{-T} is the inverse transpose of Q defined by

$$Q^{-T} = \begin{bmatrix} 1 & \frac{1}{2} \\ -1 & \frac{1}{2} \end{bmatrix} \tag{5}$$

Using equations (3), we may easily transform between single-ended and differential quantities.

Differential Admittance and Impedance Matrices

The terminal admittance (Y) and impedance (Z) matrices discussed in the previous topics in the *Technical Notes* relate single-ended voltages and currents as $i = Yv$ and $v = Zi$, respectively. If you defined differential voltages and currents e and u, equations (3) can be used to derive new Y and Z matrices that relate differential quantities.

For example, if $i = Yv$, then substituting equations (3) yields

$Q^{-T}u = YQe$.

Solving for u yields

$u = Q^{T}YQe$

and the matrix Y' relating differential quantities e and u is defined by

$Y' = Q^{T}YQ$.

(6)

A similar procedure applies to the terminal Z matrix.

Differential S-Matrices

It is clear that an S-matrix can be computed for differential signals because it is possible to compute admittance and impedance matrices for differential signals. The differential S-matrix can be envisioned as relating in-going and out-going waves on imaginary transmission lines attached to the differential ports. The characteristic impedance must be specified for these lines.

In the single-ended case, the characteristic impedance for a pair of transmission lines may be written in the form of a matrix relating the voltages and currents on the two (uncoupled) lines,

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} Z^{(1)}{}_{ref} & 0 \\ 0 & Z^{(2)}{}_{ref} \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} \quad (7)$$

where $Z^{(1)}{}_{ref}$ and $Z^{(2)}{}_{ref}$ are the user-specified reference impedances. In the differential case, the matrix equation relating differential and common currents and voltages is written as

$$\begin{bmatrix} v_d \\ v_c \end{bmatrix} = \begin{bmatrix} Z^{(d)}{}_{ref} & 0 \\ 0 & Z^{(c)}{}_{ref} \end{bmatrix} \begin{bmatrix} i_d \\ i_c \end{bmatrix} . \quad (8)$$

In this case, $Z^{(d)}{}_{ref}$ and $Z^{(c)}{}_{ref}$ denote the user-specified differential and common reference impedances, respectively.

Magnetic Bias Sources

When you create a ferrite material, you must define the net internal field that biases the ferrite by assigning a magnetic bias source. The bias field aligns the magnetic dipoles in the ferrite, producing a non-zero magnetic moment.

When the applied bias field is assumed to be uniform, you will specify the tensor coordinate system through a rotation from the global coordinate system. When the applied bias field is non-uniform, specified coordinate system rotations are not allowed. The permeability tensor's local coordinate system is calculated on a tetrahedron by tetrahedron basis, with the direction determined by the field directions calculated in the static solution. HFSS references the static solution project as the source of the non-uniform magnetostatic field information during solution generation.

Uniform Applied Bias Fields

The applied DC bias that causes ferrite saturation is always in the positive z direction of the tensor coordinate system. Initially the tensor coordinate system is assumed to be aligned with the fixed coordinate system; the tensor's z-axis is the same as the model's z-axis. To model other directions of applied bias, the permeability tensor must be rotated so that its z-axis lies in another direction on the fixed coordinate system. This is accomplished by specifying the rotation angles about the axes when you assign a magnetic bias source to a model surface.

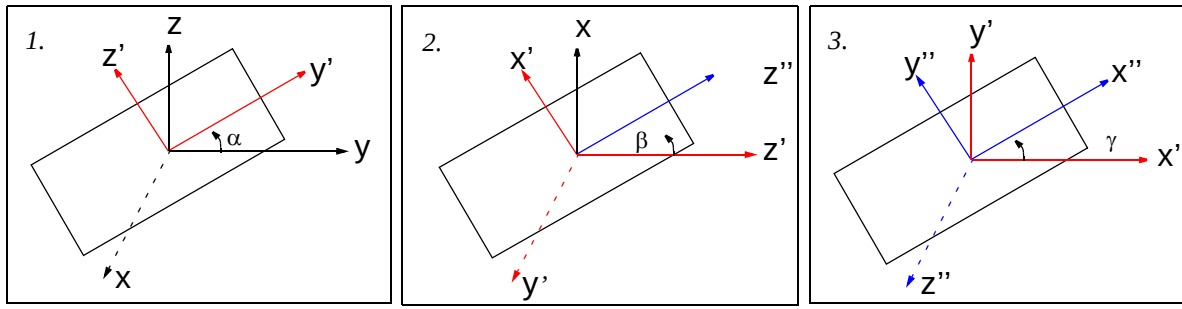
The rotation angles should be defined in the **Magnetic Bias Source** dialog box in such a way that the tensor coordinate system is obtained in the following manner:

1. Rotating the tensor coordinate system by α degrees (from the **X Angle**) around the fixed x-

axis.

2. Rotating the resulting tensor coordinate system by β degrees (from the **Y Angle**) around the new y-axis.
3. Rotating the new tensor coordinate system by γ degrees (from the **Z Angle**) around the new z-axis.

This concept is illustrated in the following graphic. In the first panel, the permeability tensor is rotated α degrees about the x-axis. In the second panel, the tensor is rotated β degrees about the y'-axis (the new y-axis). In the third panel, the tensor is rotated γ degrees about the z"-axis (the new z-axis). The resulting tensor has the coordinate system (x"y"z") relative to the fixed coordinate system.



For example, to model the DC bias in the x direction you would rotate the tensor coordinate system so that its z-axis lies along the x-axis of the fixed coordinate system. To do this you would enter **0** for the **X Angle**, **90** for the **Y Angle**, and **0** for the **Z Angle**.

Non-uniform Applied Bias Fields

To accurately model a ferrite in an applied static magnetic bias field, the non-uniform magnetic bias fields must also be calculated. In HFSS, a ferrite's permeability tensor is a direct result of an applied static magnetic bias field. The static field causes the tensor to assume an hermitian form, with cross coupling terms between field components perpendicular to the bias. However, a uniform bias field is difficult to achieve in practice. Even if the bias field is nearly uniform, a non-ellipsoidal-shaped ferrite material will have non-uniform demagnetization, resulting in non-uniform fields in the ferrite.

Use the magnetostatic solver provided in the Maxwell 3D Field Simulator to generate a solution for non-uniform magnetostatic fields. Once a solution is generated it may be imported into HFSS.

Note To specify the non-uniform bias field, you must have purchased the Maxwell 3D Field Simulator. Refer to the Maxwell 3D Field Simulator documentation for instructions on solving for non-uniform magnetostatic fields.

Incident Waves

An incident wave (or plane wave) is a wave that propagates in one direction and is uniform in the directions perpendicular to its direction of propagation. The angle at which the incident wave

impacts the device is known as the angle of incidence. The equation that HFSS uses to calculate the incident wave is

$$\boldsymbol{E}_{inc} = \boldsymbol{E}_0 e^{-j\boldsymbol{k}_0(\hat{\boldsymbol{k}} \cdot \boldsymbol{r})}$$

where

- E_{inc} is the incident wave.
- E_0 is the E-field polarization vector.
- k_0 is the free space wave number. It is equal to $\omega\sqrt{\mu_0\varepsilon_0}$.
- $\hat{k}$ is the propagation vector. It is a unit vector.
- r is the position vector and is equal to $x\hat{x} + y\hat{y} + z\hat{z}$.

Incident wave excitations are specified in a peak sense. That is, if the incident wave magnitude is 5 V/m, then the real time function of the incident field is $E(t) = 5\cos(\boldsymbol{k} \cdot \boldsymbol{r} + \omega t)$.

Materials

This section of the *Technical Notes* includes information on the following linear material properties:

- Relative Permeability
- Relative Permittivity
- Bulk Conductivity
- Dielectric Loss Tangent
- Magnetic Loss Tangent

and the following ferrite material properties:

- Magnetic Saturation
- Lande G Factor
- Delta H

Information is also included about the following:

- Anisotropic materials and defining anisotropy tensors.
- Frequency-dependent material properties.
- Frequency-dependent Loss Model in HFSS

Relative Permeability

The real portion of the relative permeability is a dimensionless quantity.

The relative permeability of all dielectrics is assumed to be complex, as follows:

$$\mu = \mu' - j\mu''$$

which can also be expressed as

$$\mu = \mu'\left(1 - j\frac{\mu''}{\mu'}\right)$$

where μ' is the real portion of μ and μ''/μ' is the magnetic loss tangent.

Relative Permittivity

The relative permittivity of all dielectrics is assumed to be complex, as follows:

$$\varepsilon = \varepsilon' - j\varepsilon''$$

which can also be expressed as

$$\varepsilon = \varepsilon'\left(1 - j\frac{\varepsilon''}{\varepsilon'}\right)$$

where ε' is the real portion of ε and $\varepsilon''/\varepsilon'$ is the dielectric loss tangent.

If a material's losses due to bulk conductivity will be significant, such as in semiconductor dielectric materials, an additional bulk conductivity value, σ, must be added. From the time harmonic form of Maxwell's equations, the complex permittivity, ε_c is defined as

$$\nabla \times H = (\sigma + j\omega\varepsilon)E = j\omega\varepsilon_c E$$

where

$$\varepsilon_c = \varepsilon(1 - j\tan\delta - j\frac{\sigma}{\omega\varepsilon})$$

When entering the relative permittivity in the **Relative Permittivity** field, use the object's real component, ε'. The real portion of the relative permittivity is a dimensionless quantity and is identical to the material's relative dielectric constant, ε_r.

Bulk Conductivity

HFSS is capable of including conductivity in the model either as a bulk material loss factor, similar to dielectric loss tangent, or as an impedance boundary condition applied to the outer surfaces of the object. The choice of bulk material loss instead of the boundary condition is made by selecting **Solve Inside** in the **Properties** window.

The choice between bulk material loss and the surface boundary condition is problem dependent. The boundary condition should be applied whenever the conductor is much thicker than the skin depth at the solution frequency. In this case, the unknowns within the conductor are not included in the unknown vector, resulting in a smaller matrix and a faster analysis. However, if the conductor is not thick relative to the skin depth, the bulk material conductivity must be used to arrive at an accurate solution. With this assumption, the wave equation reduces to

$$\nabla\times\left(\frac{-1}{j\omega\mu}\nabla\times\boldsymbol{E}\right) = (j\omega\varepsilon + \sigma)\boldsymbol{E}$$

Dielectric Loss Tangent

To represent a dielectric that dissipates the power of a high-frequency electric field, enter a dielectric loss tangent, $\varepsilon''/\varepsilon'$, property value for the material. The smaller the loss tangent, the less lossy the material.

The dielectric loss tangent may vary with frequency. To simulate the variances, define a function for the dielectric loss tangent.

Magnetic Loss Tangent

To represent a dielectric that dissipates the power of a high-frequency magnetic field, enter a magnetic loss tangent, μ''/μ', property value for the material. The smaller the loss tangent, the less lossy the material.

Note If you plan to do a frequency sweep for a design that includes dielectrics, make sure that the dielectric or magnetic loss tangent does not vary significantly over the requested frequency range. If they do, the results may not be what you expect. In cases where the loss tangent does vary significantly over the frequency range in which you are interested, copy and solve the design several times, adjusting the loss tangent and associated frequency range for the copied design so that the loss tangent is relatively stable over the design's requested frequency range.

Ferrite Materials

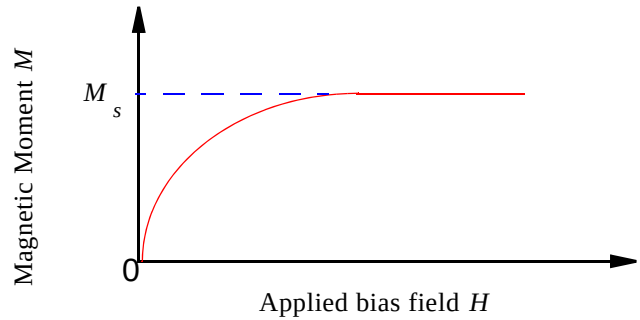
Ferrite materials are used to model the interaction between a microwave signal and a material whose magnetic dipole moments are aligned with an applied bias field. The gyrotropic quality of the ferrite is evident in the permeability tensor which is Hermitian in the lossless case. The Hermitian tensor form leads to the non-reciprocal nature of the devices containing microwave ferrites. If the microwave signal is circularly polarized in the same direction as the precession of the magnetic dipole moments, the signal interacts strongly with the material. When the signal is polarized in the opposite direction to the precession, the interaction will be weaker. Because the interaction between the signal and material depends on the direction of the rotation, the signal propagates through a ferrite material differently in different directions.

If you assign a ferrite material to an object, you must assign a magnetic bias source to the object.

Magnetic Saturation

A material with a non-zero magnetic saturation is considered to be a ferrite. When a ferrite is placed in a uniform magnetic field, the magnetic dipole moments of the material begin to align with the field. As the strength of the applied bias field increases, more of the dipole moments align. The magnetic saturation, M_S, is a property that describes the point at which all of the magnetic dipole moments of the material become aligned. At this point, further increases in the applied bias field

strength do not result in further saturation. The relationship between the magnetic moment, *M*, and the applied bias field, *H*, is shown below.



The magnetic saturation, $4\pi M$, is entered in gauss.

Lande G Factor

The Lande *g* factor is a ferrite property that, on a microscopic level, describes the total magnetic moment of the electrons according to the relative contributions of the orbital moment and the spin moment. When the total magnetic moment is due entirely to the orbital moment, *g* is equal to one. When the total magnetic moment is due entirely to the spin moment, *g* is equal to two. For most microwave ferrite materials, *g* has a range from 1.99 to 2.01. The Lande *g* factor is dimensionless.

Delta H

Delta H is the full resonance line width at half-maximum, which is measured during a ferromagnetic resonance measurement. It relates to how rapidly a precessional mode in the biased ferrite will damp out when the excitation is removed. The factor ΔH doesn't appear in the permeability tensor; instead, the factor α appears. The factor α is computed from

$$\alpha = -\frac{\gamma \mu_0 \Delta H}{2\omega}$$

The factor α changes the κ and χ terms in the permeability tensor from real to complex, which makes the tensor complex non-symmetric (where it had been hermitian for lossless ferrites).

Enter the full resonance line width at half maximum in the **Delta H** value box. The full resonance line width at half maximum is entered in oersteds.

Anisotropic Materials

Anisotropic materials have characteristics that vary with direction. These characteristics are defined by their anisotropy tensors. You must define three diagonals each for anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis.

Anisotropic Relative Permeability Tensors

The relative permeability tensor for an anisotropic material is described by

$$[\mu] = \begin{bmatrix} \mu_1\mu_0 & 0 & 0 \\ 0 & \mu_2\mu_0 & 0 \\ 0 & 0 & \mu_3\mu_0 \end{bmatrix}$$

where

- μ_1 is the relative permeability along one axis of the material's permeability tensor.
- μ_2 is the relative permeability along the second axis.
- μ_3 is the relative permeability along the third axis.
- μ_0 is the permeability of free space.

The relationship between *B* and *H* is:

$$\begin{bmatrix} B_x \\ B_y \\ B_z \end{bmatrix} = [\mu] \begin{bmatrix} H_x \\ H_y \\ H_z \end{bmatrix}$$

To specify the relative permeability for an anisotropic material, enter the μ_1, μ_2, and μ_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. If the relative permeability is the same in all directions, use the same value for μ_1, μ_2, and μ_3. These values can also be entered as variables.

Anisotropic Relative Permittivity Tensors

The relative permittivity tensor for an anisotropic material is described by

$$\varepsilon = \begin{bmatrix} \varepsilon_1\varepsilon_0 & 0 & 0 \\ 0 & \varepsilon_2\varepsilon_0 & 0 \\ 0 & 0 & \varepsilon_3\varepsilon_0 \end{bmatrix}$$

where

- ε_1 is the relative permittivity of the material along one tensor axis.
- ε_2 is the relative permittivity along the second axis.
- ε_3 is the relative permittivity along the third axis.
- ε_0 is the permittivity of free space.

The relationship between *E* and *D* is then

$$\begin{bmatrix} D_x \\ D_y \\ D_z \end{bmatrix} = [\varepsilon] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the relative permittivity for an anisotropic material, enter the ε_1, ε_2, and ε_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as variables.

Anisotropic Conductivity Tensors

The conductivity tensor for an anisotropic material is described by

$$[\sigma] = \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix}$$

where

- σ_1 is the relative conductivity along one axis of the material's conductivity tensor.
- σ_2 is the relative conductivity along the second axis.
- σ_3 is the relative conductivity along the third axis.

The relationship between *J* and *E* is then:

$$\begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} = [\sigma] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the conductivity for an anisotropic material, enter the σ_1 , σ_2, and σ_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. The values of σ_1 and σ_2 apply to axes that lie in the xy cross-section being modeled. The values of σ_3 apply to the z-component. These values affect current flowing in dielectrics between the conductors. These values can also be entered as variables.

Anisotropic Dielectric Loss Tangent Tensors

The dielectric loss tangent tensor for an anisotropic material is described by

$$[\bar{\varepsilon}] = \begin{bmatrix} \varepsilon'_1(1 - j\tan\delta_1) & 0 & 0 \\ 0 & \varepsilon'_2(1 - j\tan\delta_2) & 0 \\ 0 & 0 & \varepsilon'_3(1 - j\tan\delta_3) \end{bmatrix}$$

where

- $\tan\delta_1$ is the ratio of the imaginary relative permittivity to the real relative permittivity in one direction.

$$\tan\delta_1 = \frac{\varepsilon''_1}{\varepsilon'_1}$$

- $\tan\delta_2$ is the ratio of the imaginary relative permittivity to the real relative permittivity in the second direction.

$$\tan\delta_2 = \frac{\varepsilon''_2}{\varepsilon'_2}$$

- $\tan\delta_3$ is the ratio of the imaginary relative permittivity to the real relative permittivity in the third orthogonal direction.

$$\tan\delta_3 = \frac{\varepsilon''_3}{\varepsilon'_3}$$

- ε'_1, ε'_2, and ε'_3 are the real relative permittivities specified earlier.ε
- *j* is the imaginary unit, $\sqrt{-1}$.

The relationship between *D* and *E* will then be

$$\begin{bmatrix} D_x \\ D_y \\ D_z \end{bmatrix} = [\bar{\varepsilon}] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the electric loss tangent for an anisotropic material, enter the $\tan\delta_1$, $\tan\delta_2$, and $\tan\delta_3$ values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as variables.

Anisotropic Magnetic Loss Tangent Tensors

The magnetic loss tangent tensor for an anisotropic material is described by

$$[\bar{\mu}] = \begin{bmatrix} \mu'_1(1 - j\tan\delta_{M1}) & 0 & 0 \\ 0 & \mu'_2(1 - j\tan\delta_{M2}) & 0 \\ 0 & 0 & \mu'_3(1 - j\tan\delta_{M3}) \end{bmatrix}$$

where

- $\tan\delta_{M1}$ is the ratio of the imaginary relative permeability to the real relative permeability in one direction.

$$\tan\delta_{M1} = \frac{\mu''_1}{\mu'_1}$$

- $\tan\delta_{M2}$ is the ratio of the imaginary relative permeability to the real relative permeability in the second direction.

$$\tan\delta_{M2} = \frac{\mu''_2}{\mu'_2}$$

- $\tan\delta_{M3}$ is the ratio of the imaginary relative permeability to the real relative permeability in the third orthogonal direction.

$$\tan\delta_{M3} = \frac{\mu''_3}{\mu'_3}$$

- μ'_1, μ'_2, and μ'_3 are the real relative permeabilities specified earlier.
- *j* is the imaginary unit, $\sqrt{-1}$.

The relationship between *B* and *H* will then be

$$\begin{bmatrix} B_x \\ B_y \\ B_z \end{bmatrix} = [\bar{\mu}] \begin{bmatrix} H_x \\ H_y \\ H_z \end{bmatrix}$$

To specify the magnetic loss tangent for an anisotropic material, enter the $\tan\delta_{M1}$, $\tan\delta_{M2}$, and $\tan\delta_{M3}$ values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as functions.

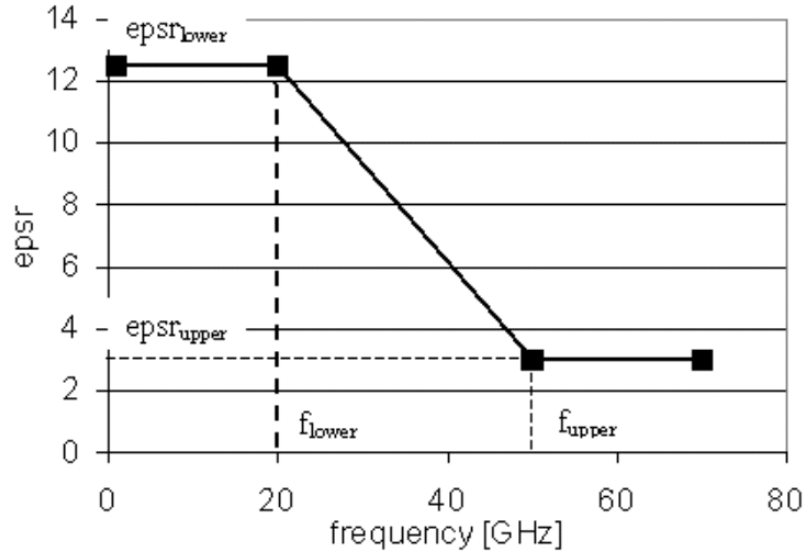
Anisotropic Materials and Ports

An anisotropic material can be in contact with a port under the following conditions:

- There is no loss on the port: a lossy material or boundary condition (finite conductivity or impedance) cannot be in contact with the port. Although a radiation boundary is lossy, it can be in contact with a port in this case because it is generally not modeled as lossy where it touches the port. Note that a radiation boundary can be modeled as lossy if the environment variable `ZERO_ORDER_ABC_ON_PORT` is set.
- One principal axis of the anisotropic material is aligned normal to the port.

Frequency-Dependent Material Properties

The properties of some materials vary with the frequency of the field excitation. This frequency dependence is often linear within a certain frequency range and constant outside of the frequency range, as shown below,



where

- $epsr_{lower}$ is the relative permittivity of a material below the frequency range 20 - 50 GHz.
- $epsr_{upper}$ is the relative permittivity of a material above the frequency range 20 - 50 GHz.
- f_{lower} is the lower frequency, below which the material property is constant.
- f_{upper} is the upper frequency, above which the material property is constant.

In general, to account for this variance within a given frequency range, use the **Piecewise Linear Material Input** window dialog box to specify a property's values at frequencies below and above the frequency range. Based on these values, HFSS automatically creates a linear dataset that specifies the property's values at the desired frequencies during solution generation. This dataset can be modified with additional points if desired.

If the material is a lossy dielectric with a lower frequency near DC, use the **Loss Model Material Input** dialog box to specify the material's conductivity at DC or, if you prefer, its loss tangent value at the lower frequency. HFSS also enables you to specify the lossy dielectric material's high frequency/optical permittivity.

In most materials, up to a couple with a 10-GHz limit, ion and dipole polarization dominate. These polarization types can be described by Debeye's relaxation polarization model:

$$\varepsilon_{rcomplex} = \varepsilon_{roptical} + \frac{(\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + j\omega\tau}$$

where

τ = the relaxation time.

$\varepsilon_{rstatic}$ = the static permittivity.

$\varepsilon_{roptical}$ = the high frequency/optical permittivity.

HFSS uses the values you specify in the **Loss Model Material Input** dialog box in Debeye's equation above to determine the relative permittivity at any frequency.

Debeye's model is valid for most microwave applications. If the frequency exceeds the limit of Debeye's model, other models that take atomic and electron polarization into account are available.

Frequency-dependent materials are appropriate for problems solved using a discrete or interpolating sweep.

Related Topics

Defining Frequency-Dependent Material Properties

Defining Frequency-Dependent Material Properties for Lossy Dielectrics

Frequency Dependent Material Loss Model in HFSS

Frequency Dependent Material Loss Model in HFSS

In the simulation of high speed connectors or PCB boards, it is important to take the losses into account. Especially, in the case of transient analysis, where the improper specification of the frequency dependency of the materials would lead to unphysical results. This section discusses the loss mechanism of dielectric materials. Magnetic losses could also be taken into account, but it is not the aim of the current discussion.

A lossy dielectric material is characterized by two measured values at a certain frequency:

dielectric constant ε_r and loss tangent $tg\delta$. There are two problems at the specification of the frequency dependency of dielectric materials:

- The frequency range, in which the solution has to be calculated, is much wider than the range where measured material data are available. It is very usual, to have just two measured points. A low frequency point, which can be taken as DC value and a higher frequency point, which serves to describe the behavior of the material at high frequency. The question arises: what is to do between and outside the measured points.
- A measurement always suffers from errors. If the measured data are not consistent, it could cause unphysical effects during the time domain analysis. A question arises again: how to

check whether the measured data are consistent, and how to adjust them if they are not.

To answer the questions raised we need to discusses the polarization loss mechanism of dielectric materials.

Figure 1 shows the different polarization mechanisms occur in the frequency domain. It can be seen from the figure that up to a couple of 10 GHz limits, ion and dipole polarization dominate. These polarization types can be described by Debye's relaxation polarization model:

$$\varepsilon_{rcomplex} = \varepsilon_{roptical} + \frac{(\varepsilon_{rsrtatic} - \varepsilon_{roptical})}{1 + j\omega\tau}$$

(1)

where

τ is the relaxation time, $\varepsilon_{rstatic}$ and $\varepsilon_{roptical}$ are the static and high frequency permittivity, respectively. Debye's model is valid for the most of microwave applications. If the frequency exceeds the limit of Debye's model, there exists other models which take the atomic and electron polarization into account [1].

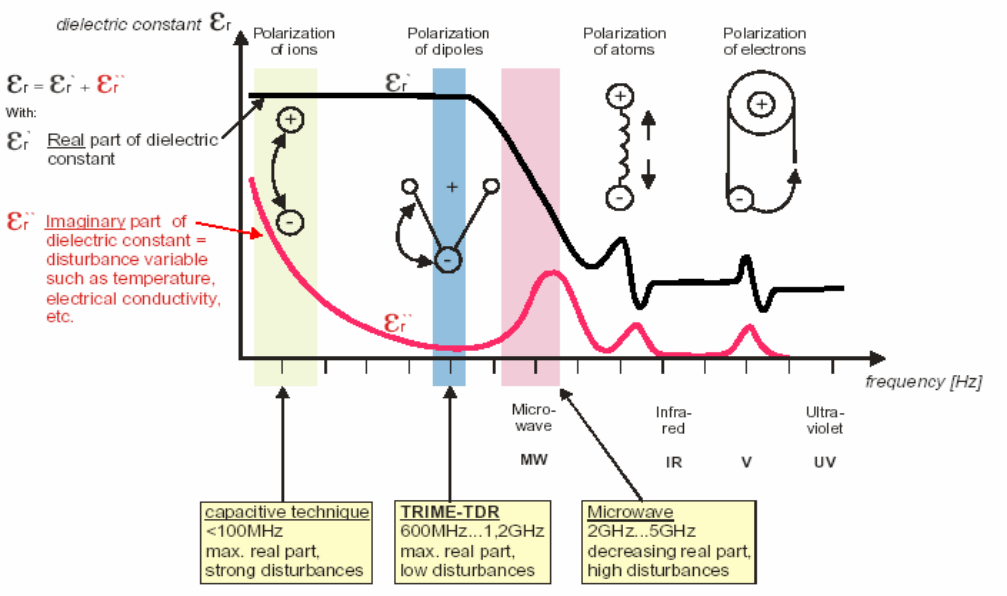


Figure 1 : Polarization mechanisms

The question is now how to fit Debye's model to the actual material characteristics. In order to do this, let us express the real part of the dielectric constant and the conductivity from Eq. (1):

$$\varepsilon_r = \varepsilon_{roptical} + \frac{(\varepsilon_{rsrtatic} - \varepsilon_{roptical})}{1 + (\omega\tau)^2} \quad (2)$$

$$\sigma = \sigma_o + \frac{\omega^2 \varepsilon_o \tau (\varepsilon_{rsrtatic} - \varepsilon_{roptical})}{1 + (\omega\tau)^2} \quad (3)$$

where

σ_o is the DC conductivity.

Eqs (2) and (3) have 4 parameters: $\sigma_o, \varepsilon_{rstatic}, \varepsilon_{roptical}$ and τ .

Usually, measured data are available at two frequencies for a regular material. The measured data are the dielectric constant and the loss tangent. One set of the measured data is at low frequency ($f_1 \approx 1MHz$, $\varepsilon_{r1}, \sigma_1$) and the other set is at higher frequency ($f_2 \approx 1 \sim 2\,GHz$, $\varepsilon_{r2}, \sigma_2$). The static or dc values can be considered as the low frequency measured data. So,

$$\varepsilon_{rstatic} = \varepsilon_{r1} \quad (4)$$

and

$$\sigma_o = \omega_1 \varepsilon_o \varepsilon_{r1} \tan\delta_1 \quad (5)$$

The critical need is to predict the high frequency behavior of the material.

If we know $\varepsilon_{roptical}$ from measurement, the high frequency behavior of Deby's model is set. We just need to calculate the relaxation time as:

$$\tau = \frac{b - \sqrt{b^2 - 4}}{2\omega_2} \quad (6)$$

where

$$b = \frac{\omega_2 \varepsilon_o (\varepsilon_{rstatic} - \varepsilon_{roptical})}{\sigma_2} \quad (7)$$

and

$$\sigma_2 = \omega_2 \varepsilon_o \varepsilon_{r2} \tan\delta_2 - \sigma_o \quad (8)$$

Knowing τ , Eqs. (2) and (3) prescribe the material characteristic in the whole frequency region.

If we do not know $\varepsilon_{roptical}$, it can be calculated by simultaneously solving the following two equations to get $\varepsilon_{roptical}$ and τ:

$$\varepsilon_{r2} = \varepsilon_{roptical} + \frac{(\varepsilon_{rsrtatic} - \varepsilon_{roptical})}{1+(\omega_2\tau)^2} \quad (9)$$

$$\sigma_2 = \sigma_o + \frac{\omega_2^2 \varepsilon_o \tau(\varepsilon_{rsrtatic} - \varepsilon_{roptical})}{1+(\omega_2\tau)^2} \quad (10)$$

where ε_{r2} is measured and σ_2 id determined by Eq.(8).

Solving equations (9) and (10), we get:

$$\tau = \frac{\varepsilon_o \left|\varepsilon_{rstatic} - \varepsilon_{r2}\right|}{\left|\sigma_2 - \sigma_o\right|} \quad (11)$$

$$\varepsilon_{roptical} = \varepsilon_{rstatic} - \left|\varepsilon_{rstatic} - \varepsilon_{r2}\right|\left(\frac{1+(\omega_2\tau)^2}{(\omega_2\tau)^2}\right) \quad (12)$$

This method also yields the parameters of the Debye's model, but the accuracy of the method, especially at high frequencies, depends on the accuracy and the consistency of the measurement at frequency f_2

Typical Debye's model material characteristics can be seen in Fig. 2, where

$$\varepsilon' = \varepsilon_r$$

(11)

and $\varepsilon'' = \frac{\omega\tau(\varepsilon_{rsrtatic} - \varepsilon_{roptical})}{1+(\omega_2\tau)^2}$ (12)

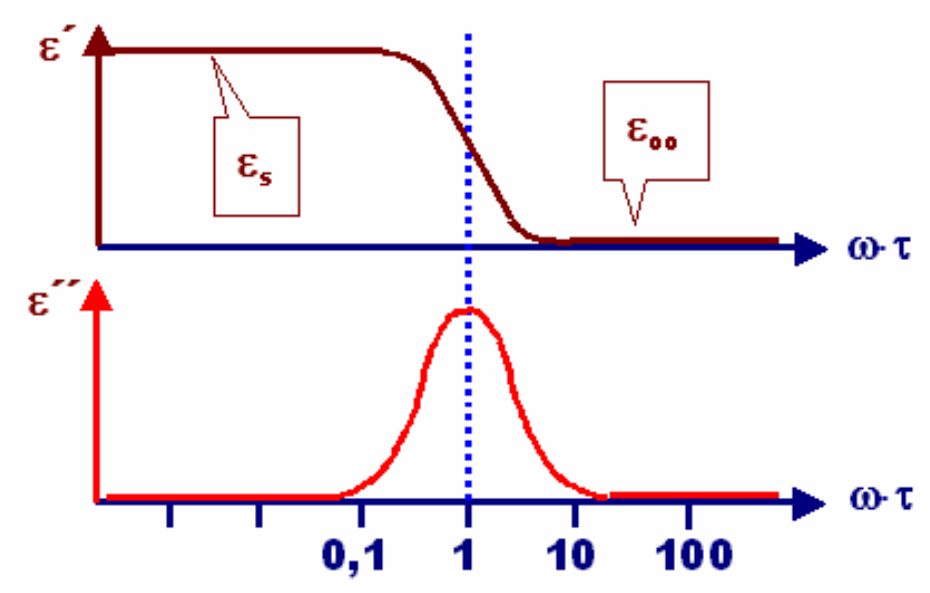


Figure 2 : Material characteristics of a Debye model

The model was verified by using a double strip line filled by FR4 material. A 10 mils section of the line was modeled by HFSS and de-embedded into a 34 inch long line. The structure of HFSS model can be seen in Fig. 3.

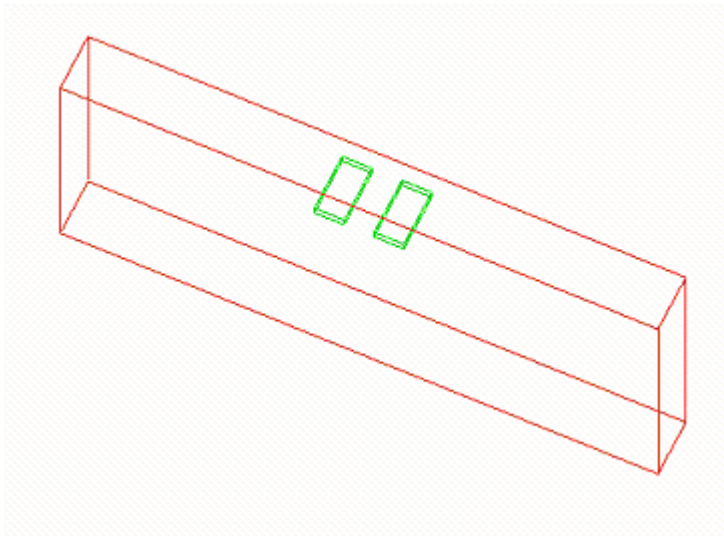


Figure 3 : A 10 mils long structure of a double strip line

The input data panel of HFSS for FR4 can be seen in Fig. 4:

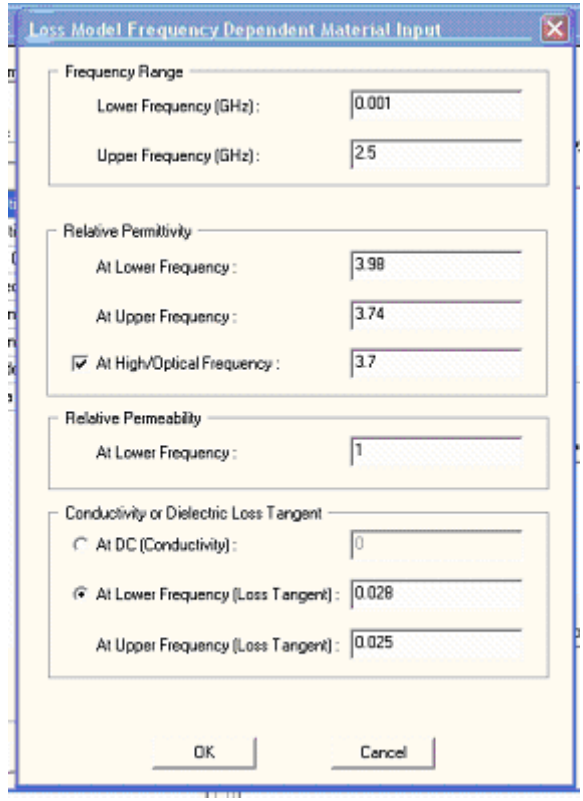


Figure 4 : Input data panel for FR4 material in HFSS

It can be seen from the panel that:

$$\varepsilon_{rstatic} = 3.98; \quad \varepsilon_{roptical} = 3.7; \quad \tan\delta_2 = 0.025; \quad \tan\delta_1 = 0.028$$

The calculated and measured frequency response of $\mathbf{S}_{12}$ can be seen in Fig. 5. The agreement between the calculated and measured values is good.

The transient response of the line to an input pulse also has been calculated. The schematic arrangement of the model in Maxwell Spice is shown in Fig. 6. The output signal is plotted in Fig. 7. The green curve is the transient response of an ideal reference line (lossless) of the same length.

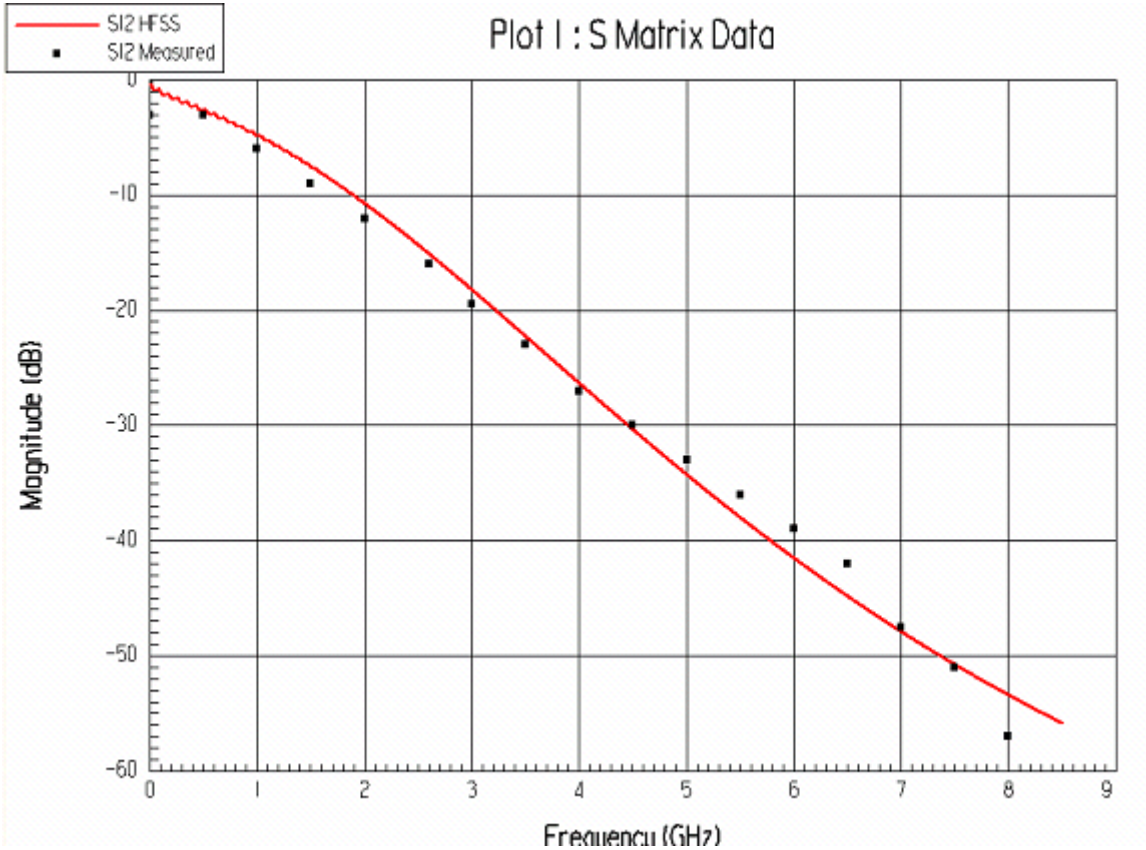


Figure 5 : Calculated and measured frequency response of S_{12}

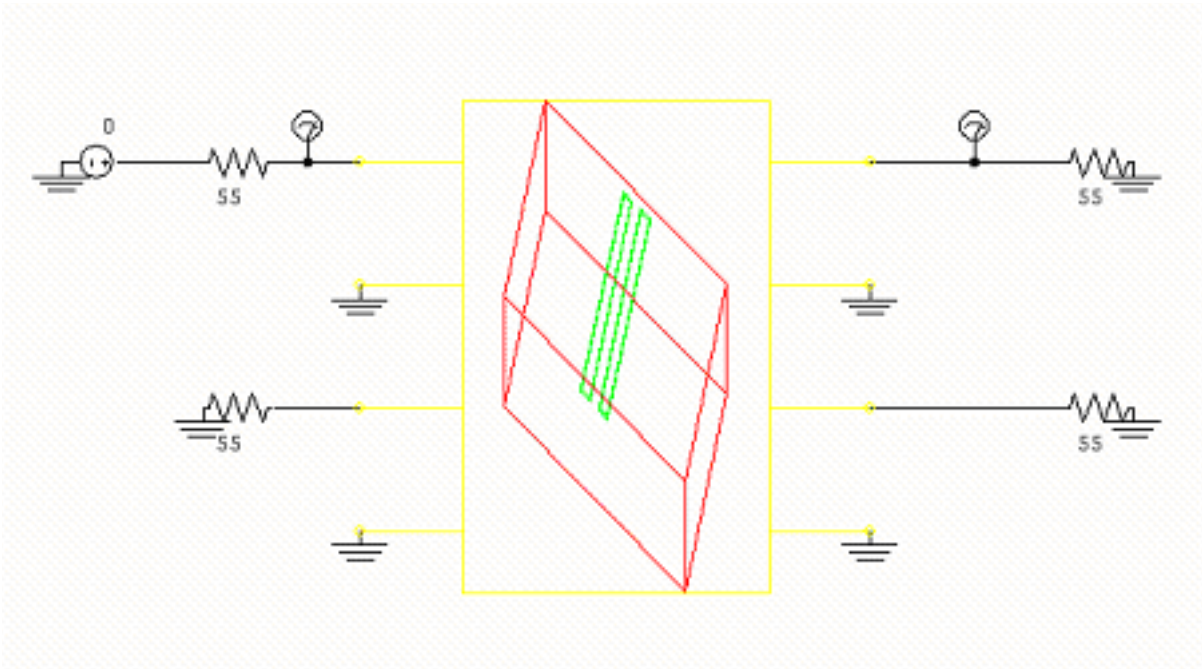


Figure 6 : Schematic arrangement in Maxwell Spice

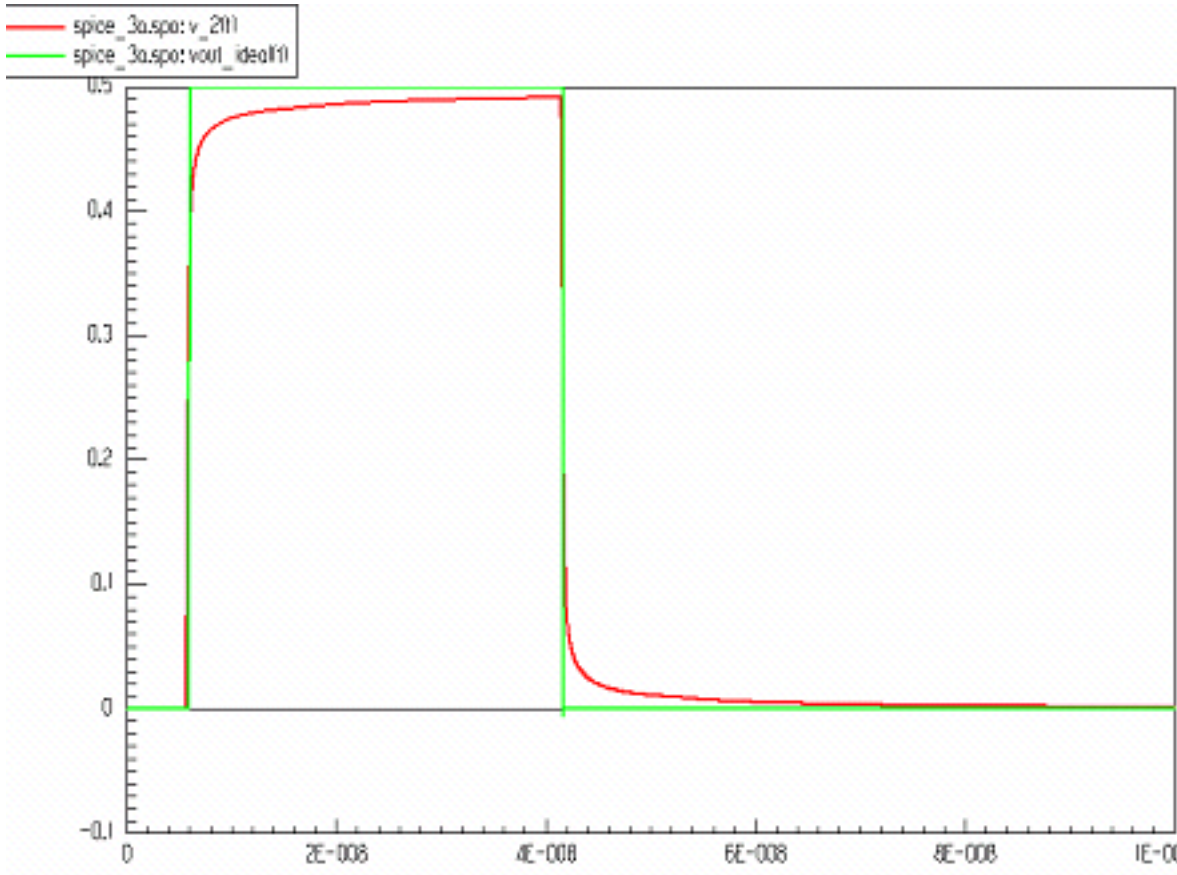


Figure 7 : Transient response by Maxwell Spice

Conclusions

Debys's material loss model predicts the dielectric losses in the whole frequency range within an acceptable accuracy. The model can be set up by using low frequency and higher frequency measured dielectric constants and loss tangents and the optical dielectric constant. If the latter is not available, the model might be less accurate at high frequencies. Using Debye's model, no unphysical phenomena can be observed at the transient response.

References

[9] E.U. Condon and Hugh Odishaw, *Handbook of Physics*, McGraw Hill Book Company, Inc., New York Toronto London, 1958. Pp. 4-113 – 4-119.

[10] G.R. Strobl, *The Physics of Polymers*, Chapter 5, 2nd Ed. Springer, NY, 1977.

Parametric Overview

Running a parametric analysis enables you to simulate several design variations using a single model. You define a series of variable values within a range, or a *variable sweep definition*, and HFSS generates a solution for each design variation. You can then compare the results to determine how each design variation affects the performance of the design.

You can vary design parameters that are assigned a quantity, such as geometry dimensions, material properties, and boundary and excitation properties. (See the online help topic for the specific parameter you want to vary.) The number of variations that can be defined in a parametric sweep setup is limited only by your computing resources.

To perform a parametric analysis, you first create a nominal design. A nominal design is created like any other design, except that variables are assigned to those aspects of the model you want to change. All variables must be defined before you start the parametric analysis. Although you are not required to solve the nominal design before performing a parametric analysis, doing so helps ensure that the model is set up and operates as intended. Alternatively, you can perform a validation check on the nominal design before performing a parametric analysis.

Parametric analyses are often used as precursors to optimization analyses because they enable you to determine a reasonable range of variable values for an optimization analysis.

Related Topics

Setting up a Parametric Analysis

Optimization Overview

Optimization is the process of locating the minimum of a user-defined cost function. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Choosing an Optimizer

When running an optimization analysis, you can choose from four optimizers, though in most cases, the Sequential Non-Linear Programming optimizer is recommended:

- **Quasi Newton**

  If the Sequential Non Linear Programming Optimizer has difficulty, and if the numerical noise is insignificant during the solution process, use the Quasi Newton optimizer to obtain the results. This optimizer uses gradient approximation of a user-defined cost function in its search for the minimum location of the cost function. This gradient approximation is only accurate enough if there is little noise involved in the cost function calculation. The cost function calculation involves FEA, which possesses finite accuracy.

- **Pattern Search**

  If the noise is significant in the nominal project, use the Pattern Search optimizer to obtain the results. It performs a grid-based simplex search, which makes use of simplices: triangles in 2D space or tetrahedra in 3D space. The cost value is calculated at the vertices of the simplex. The optimizer mirrors the simplex across one of its faces based on mathematical guidelines and determines if the new simplex provides better results. If it does not produce a better result, the next face is used for mirroring and the pattern continues. If no improvement occurs, the grid is refined. If improvement occurs, the step is accepted and the new simplex is generated to replace the original one. Pattern Search algorithms are less sensitive to noise.

- **Sequential Non-Linear Programming**

  The main advantage of SNLP over quasi Newton is that it handles the optimization problem in more depth. This optimizer assumes that the optimization variables span a continuous space.

  Like the Quasi Newton, the SNLP optimizer assumes that the noise is not significant. It does reduce the effect of the noise, but the noise filtering is not strong. The SNLP optimzer approximates the FEA characterization with Response Surfaces. With the FEA-approximation and with light evaluation of the cost function, SNLP has a good approximation of the cost function in terms of the optimization variables. This approximation allows the SNLP optimizer to estimate the location of improving points. The overall cost approximations are more accurate. This allows the SNLP optimizer a faster practical convergence speed then that of quasi Newton.

  The **SNLP Optimizer** attempts to solve a series of NLP problems on a series of inexpensive, local surrogates. Direct application of a Nonlinear Programming (NLP) solver is impractical because the cost evaluation involves finite element analysis (FEA), which uses extensive computational resources.

  The SNLP method is similar to the Sequential Quadratic Programming (SQP) method in two ways: Both are sequential, and both use local and inexpensive surrogates. However, in the SNLP case, the surrogate can be of a higher order and is more generally constrained. The inex-

pensive surrogate model is obtained by response surface (RS) techniques. The goal is to achieve a surrogate model that is accurate enough on a winder scale, so that the search procedures are well lead by the surrogate, even for relatively large steps. All functions calculated by the supporting finite element product (for example, Maxwell 3D or HFSS) is assumed to be expensive, while the rest of the cost calculation (for example, an extra user-defined expression) -- which is implemented in Optimetrics -- is assumed to be inexpensive. For this reason, it makes sense to remove inexpensive evaluations from the finite element problem and, instead, implement them in Optimetrics. This optimizer holds several advantages over the Quasi Newton and Pattern Search optimizers.

Most importantly, due to the separation of expensive and inexpensive evaluations in the cost calculation, the SNLP optimizer is more tightly integrated with the supporting FEA tools. This tight integration provides more insight into the optimization problem, resulting in a significantly faster optimization process. A second advantage is that the SNLP optimizer does not require cost-derivatives to be approximated, protecting against uncertainties (noise) in cost evaluations. In addition to derivative-free state of the RS-based SNLP, the RS technique also proves to have noise suppression properties. Finally, this optimizer allows you to use nonlinear constraints, making this approach much more general than either of the other two optimizers.

- **Sequential Mixed Integer Non-Linear Programming**

  To be able to optimize on number of turns or quarter turns, the optimizer must handle discrete optimization variables. This optimizer can mix continuous variables among the integers, or can have only integers, and works if all variables are continuous. The setup resembles that for SNLP, except that you must flag the integer variables.supporting integer variables. You can set up internal variables based on the integer optimization variable.

  For example, consider N to be an integer optimization variable. By definition it can only assume integer values. You can establish another variable, which further depends on this one: K = 2.345 * N, or K = sin(30 * N). This way K has a discrete value, but is not necessarily integer. Or, one can use N directly as a design parameter.

Optimization Variables and the Design Space

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via FEA and assigned a cost value; therefore, the cost function is defined over the design space (cost(*x*): $R^n \rightarrow R$, where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the R^n space. This subset is called the feasible domain and is defined via linear constraints.

Cost Function

Optimetrics manipulates the model's design variable values to find the minimum location of the cost function; therefore, you should define the cost function so that a minimum location is also the optimum location. For example, if you vary a design to find the maximum transmission from *Wave Port 1* to *Wave Port 2* (S_{21}=>1), define the cost function to be -mag(S(WavePort2,WavePort1)).

When using the Quasi Newton optimizer, which is appropriate for designs that are not sensitive to noise, the best cost function is a smooth, second-order function that can be approximated well by quadratics in the vicinity of the minimum; the slope of the cost function should decrease as Optimetrics approaches the optimum value. The preferred cost function takes values between 0 and 1. In practice, most functions that are smooth around the minimum are acceptable as cost functions. Most importantly, the cost function should not have a sharp dip or pole at the minimum. A well designed cost function can significantly reduce the optimization process time.

The cost function is defined in the **Setup Optimization** dialog box when you set up an optimization analysis. If you know the exact syntax of the solution quantity on which you want to base the cost function, you can type it directly in the **Calculation** text box. Otherwise, you can create an output variable that represents the solution quantity in the **Output Variables** dialog box.

Related Topics

Adding a Cost Function

Acceptable Cost

The acceptable cost is the value of the cost function at which the optimization process stops; the cost function value must be equal to or below the acceptable cost value for the optimization analysis to stop. The acceptable cost may be a negative value.

Cost Function Noise

The numerical calculation of the electromagnetic field introduces various sources of noise to the cost function, particularly because of changes in the finite element mesh. You must provide the optimizer with an estimate of the noise. The noise indicates whether a change during the solution process is significant enough to support achievement of the cost function.

For example, if the cost function, c, is

$$c = 10000 \cdot |S_{11}|^2$$

where $|S_{11}|$ is the magnitude of the reflection coefficient, at the minimum, $|S_{11}|$ is expected to be very small, $|S_{11}| \approx 0$.

From the solution setup, the error in $|S_{11}|$ is expected to be $E_{S11} \approx 0.01$. The perturbed cost function is therefore

$$c_{perturbed} = 10000 \cdot (|S_{11}|_{min} + E_{S11})^2$$

Near the minimum, the error in the cost function E_c is given by

$$E_c = c_{perturbed} - c_{min} = 10000 \cdot (0.0 + 0.01)^2 - (10000 \cdot 0.0) = 1.0$$

Therefore, the cost function noise would be 1.0.

Linear Constraints

You may constrain the feasible domain of a design variable by defining linear constraints for the optimization process. The feasible domain is defined as the domain of all design variables that sat-

isfy all upper and lower bounds and constraints. Linear constraints are defined by the following inequalities:

$$\sum_i \alpha_{ij} x_i < c_j \forall j$$

where

- α_{ij} are coefficients.
- c_j is a comparison value for the j^{th} linear constraint.
- x_i is the i^{th} designer parameter.

Related Topics

Setting a Linear Constraint

Goal Weight

If an optimization setup has a cost function made up of multiple goals, you can assign a different weight to each goal. The goal with the greater weight is given more importance during the cost calculation.

The error function value is a weighted sum of the sub-goal errors. Each sub-goal, at each frequency at which it is evaluated, gives rise to a (positive) error value that represents the discrepancy between the simulated response and the goal value limit. If the response satisfies the goal value limit, then the error value is 0. Otherwise, the error value depends on the differences between the simulated response and the respective goal limit. The error function may be defined as follows:

$$\sum_i^G \frac{W_j}{N_j} \sum_i^{N_j} e_i$$

where

- G is the number of sub-goals.
- W_j is the weight factor associated with the j^{th} sub-goal.
- N_j is the number of frequencies for the j^{th} sub-goal.
- e_i is the error contribution from the j^{th} sub-goal at the i^{th} frequency.

The value of e_i is determined by the band characteristics, target value, and the simulated response value. The choices for band characteristics are <=, =, and >=.

Band Characteristics (Condition)	e_i evaluation where s_i is the simulated response and g_i is the desired limit.
<=	$e_i = \begin{cases} 0 & s_i \le g_i \\ s_i - g_i & s_i > g_i \end{cases}$
=	$e_i = \lvert s_i - g_i \rvert$
>=	$e_i = \begin{cases} 0 & s_i \ge g_i \\ g_i - s_i & s_i < g_i \end{cases}$

If the total error value is within the acceptable cost, the optimization stops.

Related Topics

Adding a Cost Function

Step Size

In order to make the search for the minimum cost value reasonable, the search algorithm is limited in two ways. First, you do not want the optimizer to continue the search if the step size becomes irrelevant or small. This limitation impacts the accuracy of the final optimum. Second, in some cases you do not want the optimizer to take large steps either. In case the cost function is suspected to possess large variations in a relatively small vicinity of the design space, large steps may result in too many trial steps, which do not improve the cost value. In these cases, it is safer to proceed with limited size steps and have more frequent improvements.

For these two limitations, the optimizer uses two independent distance measures. Both are based on user-defined quantities: the minimum and maximum step limits for individual optimization variables. Since the particular step is in a general direction, these measures are combined together in order to derive the limitation for that particular direction.

The step vector between the i^{th} and $(i+1)^{th}$ iterate is as follows: $s_i = x_{i+1} - x_i$.

The natural distance measure is $\|s_i\| = \sqrt{s_i^T s_i}$, which is the Euclidean norm.

A more general distance measure incorporates some “stretching” of the design space: $\|s_i\|_D = \sqrt{s_i^T D^T D s_i}$, where the matrix D incorporates the linear operation of the

stretching of design space. The simplest case is when the *D* matrix is diagonal, meaning that the design space is stretched along the orthogonal direction of the base vectors.

The optimizer stops the search if $\left\| s_i \right\|_{D_{min}} < 1$, where D_{min} consists of diagonal elements equal to the inverse of the **Min. Step** value assigned to the corresponding optimization variable. Similarly the optimizer truncates steps for which $\left\| s_i \right\|_{D_{max}} > 1$

where D_{max} has diagonal elements equal to the inverse of **Max. Step** values of the corresponding optimization variables.

Related Topics

Setting the Min. and Max. Step Sizes

Sensitivity Analysis Overview

During a sensitivity analysis, Optimetrics explores the vicinity of the design point to determine the sensitivity of the design to small changes in variables. The variables and their attributes define the design point, the problem around which the sensitivity analysis is performed.

When Optimetrics performs a sensitivity analysis, its goal is to calculate the second-order regression polynomials for all of the design's output parameters. The algorithm first determines an appropriate interval for each variable. The intervals are further sub-divided according to the available number of iterations and variables. If the master output is not used, the specified initial displacement values define those intervals.

When all of the design calculations are complete, the second-order polynomials are fitted for all the output parameters. Optimetrics then reports the following quantities:

- regression value at the current variable value.
- first derivative of the regression.
- second derivative of the regression.

Related Topics

Setting up a Sensitivity Analysis

Selecting a Master Output

During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. Therefore, if you have defined more than one output parameter, be sure to select **Master Output** for the output variable on which you want the selection of design variations to be based.

Related Topics

Setting up an Output Parameter

Setting up a Sensitivity Analysis

Tuning Analysis Overview

Tuning a variable is useful when you want to manually modify its value and immediately perform an analysis of the design. For example, it is useful after performing an optimization analysis, in which Optimetrics has determined an optimal variable value, and you want to fine tune the value to see how the design results are affected.

A design can be updated after a tuning analysis to reflect a design variation solved during a tuning analysis and the results, including field solutions if **Save Fields** was selected, of each solved design variation are saved for post processing.

Related Topics

Tuning a Variable

17

Scripting

For information on how to create, edit and use scripts in HFSS see:

. in html format.

. in pdf format.

You can also access help for the scripting commands via the menu bar:

- Click **Help>Scripting Contents**
- Click **Help>Scripting Index**
- Click **Help>Search Scripting**

Recording a Script

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

1. On the **Tools** menu, click **Record Script**.

   The **Save As** dialog box appears.
2. Use the file browser to locate the folder in which you want to save the script, such as:

   `C:\Ansoft\HFSS9\Scripts`

   Then double-click the folder's name.
3. Type the name of the script in the **File name** text box, and then click **Save**.

   The script is saved in the folder you selected by the file name *filename.vbs*.
4. Perform the steps that you want to record.
5. When you have finished recording the script, click **Stop Script Recording** on the **Tools** menu.

Stopping Script Recording

- On the **Tools** menu, click **Stop Script Recording**.

HFSS stops recording to the script.

Running a Script

To run a script from HFSS:

1. Click**Tools>Run Script**.

   The **Open** dialog box appears.

2. Use the file browser to locate the folder in which you saved the script, and then double-click the folder's name.

3. Type the name of the script in the **File name** text box, or click its name, and then click **Open**.

   HFSS executes the script.

   To supply script arguments when running from **Tools>Run Script**, use the edit field at the bottom of the file selection dialog. You can access the script arguments using the AnsoftScriptHost.arguments collection from vbscript. This is a standard COM collection.

To run a script from a command line, use:

-runscriptandexit or **-runscript**

You can give **-scriptargs** parameter to the script and specify arguments.

If you run the script from DOS prompt as a .vbs file (that is, you don't launch HFSS, but just launch vbs directly, or use wscript.exe or cscript.exe), the arguments will be in the WSH.arguments collection, not the AnsoftScriptHost.arguments collection. To handle this, you can write this:

```
on error resume next
dim args
Set args = AnsoftScript.arguments
if(IsEmpty(args)) then
Set args = WSH.arguments
End if
on error goto 0
'At this point, args has the arguments no matter if you are
running
'under windows script host or Ansoft script host
msgbox "Count is " & args.Count
for i = 0 to args.Count - 1
  msgbox args(i)
next
```

Pausing and Resuming a Script

To pause a script during its execution:

- Click **Tools>Pause Script**.

To resume a script after pausing it:

- Click **Tools**>**Resume Script**.

Stopping a Script

Click**Tools>Stop Script**.

HFSS stops executing the script that has been paused.

18

Glossaries

Glossary of Terms

cost function	In an optimization setup, a cost function is based on goal values specified for at least one solution quantity. Optimetrics changes the design parameter values to fulfill the cost function. The cost function can be based on any solution quantity that HFSS can compute, such as field values, S-parameters, and eigenmode data.
design variation	A single combination of variable values that is solved during a parametric or optimization setup.
goal	In an optimization setup, a goal is the value of a solution quantity that you want to be achieved during the optimization. A goal is represented as one row in the cost function table. Each cost function defined in an optimization setup must include at least one goal.
nominal design	The original model on which Optimetrics analyses are based.
sweep definition	See *variable sweep definition.*
variable sweep definition	A set of variable values within a range that Optimetrics drives HFSS to solve when a parametric setup is analyzed. A parametric setup can include one or more sweep definitions.

Index

Numerics

1D objects 5-2
2D model files
    exporting 3-17
    importing 3-20
2D objects 5-2
3D model files
    exporting 3-17
3D Modeler
    Chamfer command 5-54
    opening new windows 5-1
3D Modeler commands
    Delete Last Operation 5-37
    Purge History 5-54
3D Modeler window 1-17
3D movement mode 5-69
3D objects 5-2
3D UI Options
    View Options command 10-6

A

aborting analyses 13-16
absolute coordinates, entering 5-66
absorbing boundary
    for ports 12-14
acceptable cost 16-100
accepted power 16-49
adaptive analysis
    and frequency sweeps 12-15
    bypassing 12-2
    maximum refinement per pass 12-11
    setting convergence on real frequency only 12-9
    setting lambda refinement 12-11
    setting matrix convergence 12-12
    setting maximum delta Energy 12-8
    setting maximum delta S 12-8
    setting maximum number of passes 12-7
    setting minimum number of converged passes 12-12
    setting minimum number of passes 12-12
    setting up 12-7
Adaptive Lanczos-Padé Sweep (ALPS) 16-18
adaptive passes
    completed and remaining 15-3
adaptive solution
    single frequency 16-16
admittance matrix
    for differential pairs 16-73
    method of calculation 16-32
    plotting parameters 15-96
analyses
    changing priority of 13-15
    monitoring 13-14
    Optimetrics 14-1

re-solving 13-17
starting 13-1
stopping 13-16
Analysis
distribute analysis command 13-2
Analyze All command
using 13-1
animations
controlling the display 15-26
exporting 15-27
frequency 15-24
geometry 15-25
overview 15-24
phase 15-24
anisotropic materials
about 16-80
and ports 16-84
assigning 8-5
conductivity tensor 16-82
electric loss tangent tensor 16-82
magnetic loss tangent tensor 16-83
permeability tensor 16-81
permittivity tensor 16-81
anisotropy
conductivity 16-82
electric loss tangent 16-82
magnetic loss tangent 16-83
permeability 16-81
permittivity 16-81
anisotropy tensors
defining conductivity 8-6
defining dielectric loss tangent 8-7
defining magnetic loss tangent 8-7
defining relative permeability 8-6
defining relative permittivity 8-6
antenna arrays
defining custom 15-71
defining regular uniform 15-70
antenna properties
accepted power 16-49
axial ratio 16-45
computing 15-71
incident power 16-50
max far-field data 16-37
maximum radiation intensity 16-47
overview 16-43
peak directivity 16-47
peak gain 16-47
peak realized gain 16-48
polarization 16-43
polarization ratio 16-46
radiated power 16-48
radiation efficiency 16-50
arc lines
center-point arcs 5-4
three-point arcs 5-3
arcs
center-point 5-4
three-point 5-3
area
measuring 5-77
array factor
and power normalizations 16-42
calculation 16-39
scan angle 16-38
arrays
defining custom 15-71
defining regular 15-70
arrows
modifying in plots 15-35
types in plots 15-35
attenuation constant 16-32
auto-save file 3-10
average SAR
calculation of 16-26
axial ratio 16-45

B

background color, setting 10-15
basis functions
low-order solution basis 12-13
overview 16-2
bondwires
drawing 5-18
overview 16-53
boundaries
and frequency dependence 16-68

assigning 6-1
assigning at face intersections 5-51
default assignments 16-68
deleting 6-23
duplicating with geometry 6-27
editing properties of 6-22
hiding 6-28
moving to different surfaces 6-24
reassigning 6-24
reprioritizing 6-25
setting default values 6-31
types of 6-1
Boundary
Zoom to selected 6-2
boxes, drawing 5-12

C

calculated expressions
plotting 15-30
calculation range
setting for a cost function 14-12
setting in a parametric setup 14-7
calculator
entering values and geometries 15-46
exporting 15-61
general operations 15-50
output commands 15-59
registers 15-44
scalar operations 15-52
stack commands 15-45
stacks 15-44
units assumed as SI 15-45
vector operations 15-56
capacitors
modeling lumped 6-18
Cartesian
coordinates on grid 10-17
entering coordinates 5-64
setting as grid type 10-17
category
traces 15-95
center-point arcs 5-4
Chamfer command
3D modeler 5-54
characteristic impedance 16-28
and renormalized S-parameters 16-27
plotting 15-96
specifying impedance lines 7-10
Zpi impedance 16-29
Zpv impedance 16-29
Zvi impedance 16-29
circles, drawing 5-7
circuit types, modeling 6-18
circular polarization 16-45
clean stop 13-16
cleaning up solutions 15-14
clear linked data 12-11
clearing selections 5-64
cloning objects
before intersecting 5-50
before subtracting 5-49
before uniting 5-48
closed objects 5-2
cloud plots 15-36
color key
moving 15-34
setting visibility 15-33
colors
assigning to objects 5-34
default for objects 5-34
default for outlines 5-34
of field overlays 15-33
of highlighted objects 5-59
of selected objects 5-58
setting background color 10-15
command properties 1-16
command-line options 1-22
complex numbers
in calculator registers 15-44
complex propagation constant 16-10
complex weight 16-38
conductivity 16-78
anisotropic 16-82
defining anisotropy tensors 8-6
cones, drawing 5-13
configuring Distributed COM 13-3, 13-8
connecting objects 5-45

constraints, setting linear 14-39
context-sensitive help 2-1
convergence
in solution process 16-3
on real frequency only 12-9
output variable 12-9
viewing output variable 15-4
convergence criteria
matrix 12-12
maximum number of passes 12-7
minimum converged passes 12-12
minimum number of passes 12-12
setting delta Energy 12-8
setting maximum delta S 12-8
convergence data
for design variations 15-15
mag margin 16-15
maximum delta Energy results 15-4
maximum delta S results 15-3
number of passes completed 15-3
output variable 15-4
phase margin 16-15
plotting 15-6
viewing 15-2
viewing the mag margin 15-5
viewing the max delta (Mag S) 15-5
viewing the max delta (phase S) 15-6
viewing the maximum delta frequency 15-6
viewing the phase margin 15-5
coordinate systems
creating face 5-81
creating relative 5-79
default planes 5-81
deleting 5-82
enlarging axes 10-16
hiding axes 10-16
modifying 5-82
modifying view of axes 10-16
operations affecting 5-82
overview 5-78
setting the working CS 5-78
showing axes 10-16
shrinking axes 10-16
specifying for matching boundaries 6-15
coordinates
entering absolute 5-66
entering Cartesian 5-64
entering cylindrical 5-65
entering relative 5-66
entering spherical 5-65
Copying 5-35
copying and pasting objects 5-35
copying materials 8-14
copying to the clipboard 5-35
copyright notices 1-26
corners
removing rounded 5-46
cost function
adding 14-10
and acceptable cost 16-100
guidelines for defining 16-99
overview 16-99
plotting results vs. iteration 15-17
setting a goal 14-10
setting the calculation range 14-12
specifying solution quantity for 14-11
viewing results vs. iteration 15-17
cost function noise 16-100
count
setting for sweep definitions 14-4
covering faces 5-44
covering lines 5-44
CPU time
viewing for solution tasks 15-7
vs. real time 15-7
creating a User Defined Primitive 5-30
cross-sections
creating 5-45
plotting spherical 16-35
CTRL+A shortcut keys 5-57
current coordinate system
about 5-78
current flow direction 7-28
current flow lines 6-18
current sources
assigning 7-28
editing 7-28
scaling the magnitude 15-21

setting the phase 15-21
custom arrays
geometry file setup 16-41
setting up 15-71
cylinders, drawing 5-11
cylindrical coordinates, entering 5-65

D

data tables
exporting 3-19
importing 3-22
data tables, creating 15-86
data types for convergence
sweep setup 12-17
dataset expressions
adding 3-40
using 3-40
datasets
adding 3-40
and frequency-dependent materials 8-8
modifying 3-41
DC thickness
assigning 2-1
Debeye's model 16-85, 16-86
decade count sweep definitions 14-4
decay of higher order modes 16-12
de-embedding
S-matrices 7-37
theory 16-27, 16-33
default impedance
changing 15-89
default variable value
overriding for a parametric setup 14-5
defaults 5-34, 5-35
assigned boundaries 16-68
auto cover closed polylines 5-44
auto cover polylines 5-44
background color 10-15
basic field quantities 16-24
clone objects before intersecting 5-50
clone objects before subtracting 5-49
clone objects before uniting 5-48
color of highlighted objects 5-59
color of selected objects 5-58
field plot attributes 15-32
lighting 10-13
mesh plot attributes 15-74
open Properties window after drawing objects 1-15
rendering mode 10-10
SAR settings 15-31
setting face CS 5-81
setting for boundary values 6-31
setting for excitation values 6-31
snap settings 5-72
view orientation 10-11
Delete Last Operation command
3D Modeler 5-37
deleting
boundaries 6-23
excitations 7-31
field overlay plots 15-40
materials 8-15
polyline segments 5-36
solution data 15-14
start points and end points 5-36
deleting objects 5-36
delta between markers
in reports 15-88
delta E, See maximum delta Energy 12-8
delta frequency, maximum 15-6
delta H 16-80
delta S, See maximum delta S 12-8
density of grid 10-17
derived field quantities
plotting 15-30
design variables
deleting 3-37
See local variables 3-36
design variations
manually modifying points 14-5
viewing all in a parametric setup 14-3
viewing solution data 15-15
designs
in project tree 1-12
inserting in project 4-2
setting up 4-1

desktop
menu bar 1-6
overview 1-6
status bar 1-11
toolbars 1-8
detaching faces 5-44
dielectric loss tangent 16-78
defining anisotropy tensors 8-7
differential pairs
admittance and impedance matrices 16-73
computing the S-matrix 16-73
setting up 7-13
directivity, peak 16-47
Discrete frequency sweep 16-19
Discrete sweeps
saving fields 12-20
saving fields at single frequencies 12-21
specifying single frequencies 12-21
display types, of reports 15-81
distributed solve
license issues 13-12
distribution criteria
setting for statistical setups 14-26
donuts, See toruses 5-14
draft angles
and sweeping objects 5-42
draft types
and sweeping objects 5-42
drawing a model
overview 5-1
drawing objects
bondwires 5-18
boxes 5-12
center-point arcs 5-4
circles 5-7
cones 5-13
cross-sections 5-45
cylinders 5-11
ellipses 5-8
equation based curve 5-7
for post processing 5-20
helices 5-15
non model 5-20
overview 5-2
planes 5-19
points 5-19
polylines 5-5
rectangles 5-9
regular polygons 5-10
regular polyhedrons 5-13
spheres 5-11
spiral using UDP 5-17
spirals 5-16
splines 5-4
straight line segments 5-2
three-point arcs 5-3
toruses 5-14
Driven Modal solutions
overview 16-21
setting 4-3
Driven Terminal solutions
overview 16-21
setting 4-3
duplicates and parent objects 5-39
duplicating
boundaries 6-27
excitions 6-27
integration lines 7-11
terminal lines 7-13
duplicating objects
along a line 5-39
and mirroring 5-40
around an axis 5-40
overview 5-39

E

edges
creating objects from 5-51
rounding 5-54
edges, selecting 5-62
effective wavelength 16-32
E-field
aligning with integration lines 16-69
specifying direction of 7-27
e-field
specifying direction of 7-10
E-fields

calculating on slave boundary 16-64
mapping to other surfaces 16-63
relating master and slave boundary 6-16
Eigenmode
setting Maximum delta Frequency 12-8
Eigenmode solutions
overview 16-21
setting 4-3
viewing solution data 15-13
eigenmodes
minimum frequency to search for 12-5
specifying number of 12-6
ellipses, drawing 5-8
Emission test
selecting quantity to plot 15-103
energy error 12-8
epsilon 16-32
equation based surface
drawing 5-10
equations based curve, drawing 5-7
equivalent circuit export options 15-11
error tolerance
for interpolating sweeps 12-18
Eval command
fields calculator 15-59
Excitation
Zoom to selected 7-2
excitation fields 16-10
excitations
and frequency dependence 16-68
deleting 7-31
duplicating with geometry 6-27
editing properties of 7-30
hiding 6-28
invalidated 7-32
moving to another surface 7-32
scaling the magnitude 15-21
setting default values 6-31
setting the phase 15-21
types of 7-1
exponential count sweep definitions 14-4
export
from the calculator 15-61
Export command
fields calculator
fields calculator
Export command 15-61
exporting
2D model files 3-17
3D model files 3-17
animations 15-27
data tables 3-19
equivalent circuit data files 15-11
field overlay plots 15-39
materials to libraries 8-16
matrix data 15-9
W-element data 15-13
expressions
dataset 3-40
defining 3-37
including in functions 3-41
intrinsic functions in 3-38
piecewise linear functions in 3-40
using as cost function goal 14-13
valid operators 3-38
extruding faces 5-46

F

face coordinate system
about 5-78
creating 5-81
creating automatically 5-81
operations affecting 5-81
faces
copying 5-50
covering 5-44
creating coordinate system on 5-81
creating face lists 5-62
creating objects from 5-50
detaching 5-44
extruding 5-46
intersections 5-51
moving along a vector 5-46
moving along the normal 5-46
removing 5-44
selecting 5-60
selecting all object 5-60

selecting behind 5-64
selecting by name 5-61
uncovering 5-44
far field
plotting on spherical cross-sections 16-35
far fields
and radiation boundaries 15-67
calculation of 16-34
computing antenna parameters 15-71
computing max data 15-71
defining antenna arrays 15-70
horizontal cross-sections 15-101
infinite spheres 15-68
plotting on spherical cross-sections 16-35
plotting quantities 15-97
reports 15-80
vertical cross-sections 15-101
far-field pattern 16-38
Fast frequency sweep 16-18
Fast sweeps
modifying matrix data 15-8
saving fields 12-20
ferrite materials
adding to libraries 8-5
properties of 8-5
ferrites
about 16-79
and delta H 16-80
and lande G 16-80
and magnetic saturation 16-79
assigning bias field 7-29
assigning magnetic bias sources 7-29
field overlay plots 15-29, 15-30
default settings 15-40
hiding color key 15-33
modifying attributes 15-32
modifying colors of 15-33
modifying field quantities 15-31
modifying phase 15-29
modifying plot scale 15-34
moving color key 15-34
on lines 15-38
SAR 16-26
SAR settings 15-31
scalar plots 15-30
vector plots 15-31
field pattern direction, specifying 16-70
field quantities
defaults 16-24
phase angle 15-52
plotting basic 15-29
plotting derived 15-30
within tetrahedra 16-2
field reports 15-80
field solutions
linking from simulations 7-15
saving at all frequency points 12-20
saving for a parametric setup 14-31
saving for a sensitivity setup 14-32
saving for a statistical setup 14-31, 14-33
saving for a tuning analysis 14-31, 14-32
saving for all Optimetrics setups 14-31
saving for an optimization setup 14-31
fields
saving at specific frequency points 12-21
fields calculator
Eval command 15-59
Value command 15-59
Write command 15-60
Fields Reporter options 3-30
file formats
animated GIF 15-27
.asol 3-2
.avi 15-27
data table 15-10
.dsp 15-39
Ensemble ver. 6+ 15-10
.gds 3-20
.hfss 3-2
HFSS ver. 6+ 15-10
.hfssresults 3-2
HSpice 15-11
.lib 15-11
Libra 15-10
.m files 15-10
Maxwell Spice 15-11
Neutral Model Format 15-10
nmf 15-10

PSpice 15-11
.sat 3-17
.sm2 3-17, 3-20
.sm3 3-17
.sNp 15-10
.sp 15-11
.spc 15-11
Touchstone 15-10
files
auto-save 3-10
HFSS 3-2
importing 3-20
fillet command
rounding edges 5-54
filtering materials 8-18
finite array patterns 16-38
finite conductivity boundaries
and frequency dependence 16-68
assigning 6-13
conditions applied at 16-61
guidelines for assigning 16-61
finite element mesh, overview 16-2
finite element method, overview 16-2
fitting all objects in view 10-7
fitting selections in view 10-7
fixed variables
setting values during analyses 14-38
free space lambda refinement 12-11
free space termination, and PMLs 16-58
free space wave number 16-23
Freq variable 16-68
frequency 16-10
and propagating modes 16-12
frequency animations 15-24
frequency depedent data points
adding for materials 8-10
frequency points
choosing for full-wave SPICE 12-22
deleting from solution 12-21
inserting in solution 12-21
frequency sweep
range for full-wave SPICE 12-22
frequency sweeps
adding to designs 12-15
and adaptive analysis 12-15
and bypassing adaptive analysis 12-2
and lumped RLC boundaries 16-65
error tolerance 12-18
Fast sweep overview 16-18, 16-19
Interpolating sweep overview 16-20
linear step frequencies 12-20
maximum number of solutions 12-18
overview 16-17
selecting type 12-15
setting points to solve 12-20
setting single frequencies 12-21
settings for 12-15
frequency-dependent materials
and lossy dielectrics 16-85, 16-86
defining 8-8
defining properties for lossy dielectrics 8-9
FSS surfaces
assigning reference surface for radiation boundaries 6-6, 6-9, 6-10
full-wave SPICE
choosing frequencies 12-22
guidelines for calculating maximum frequency 12-22
guidelines for calculating step size 12-22
functions
defining 3-41
reserved names in HFSS 3-41
selecting for a quantity 15-93
valid operators 3-38

G

gain, peak 16-47
gain, peak realized 16-48
Generate History command 5-55
geometry animations 15-25
Getting Started guides 1-24
global coordinate system
about 5-78
global materials
Ansoft 8-19
user-defined 8-19
glossary

of terms 17-1
goal
setting a complex value 14-13
setting a real value 14-13
setting a single value 14-13
setting as variable dependent 14-13
setting for cost function 14-12
setting weight of 14-10
using an expression for 14-13
goal weight
overview 16-101
setting 14-10
gotolink face selection toolbar 5-60, 5-61
gradient background colors 10-15
grid settings
choosing 10-17
density 10-17
dots or lines 10-17
grid plane 10-18
spacing 10-17
style 10-17
type 10-17
visibility 10-18
ground planes
lossy 16-67
group delay
plotting 15-96

H

helices, drawing 5-15
help
about conventions used 2-2
on context 2-1
on dialog boxes 2-1
on menu commands 2-1
HFSS
command-line options 1-22
getting started 1-1
introduction 1-1
setting options 3-27
HFSS Options dialog
solving inside threshold 8-3
hiding
boundaries 6-28
color key 15-33
excitations 6-28
field overlay plots 15-39
objects 10-8
selections 10-8
history tree
controlling view of objects 1-20
operations affecting CSs 5-82
operations affecting face CS 5-81
operations affecting relative CS 5-78
Unclassified folder 1-20
holes
moving 5-46
resizing 5-46
horizontal cross-sections 15-101
HSpice files
exporting to 15-11

I

i_<object name> boundary type 16-68
impedance
and frequency 16-12
and symmetry 16-63
changing 15-89
characteristic 16-28
method of calculating 16-28
renormalizing S-matrices 7-36
renormalizing S-parameters 16-27
impedance boundaries
and frequency dependence 16-68
assigning 6-5
conditions applied at 16-56
overview 16-56
impedance boundary
units 16-57
impedance lines
See integration lines 7-10
impedance matrix
for differential pairs 16-73
method of calculation 16-31
plotting parameters 15-96
impedance multiplier

and symmetry planes 16-63
setting 7-35
theory behind 16-30
imperfect conductors, modeling 6-13
importing
data tables 3-22
files 3-20
solution data 3-22
incident power 16-50
incident waves
assigning 7-15
scaling the magnitude 15-21
scaling the phase 15-21
inductors
modeling lumped 6-18
infinite arrays 16-38
defining custom 15-71
defining regular 15-70
infinite sphere, defining 15-68
initial displacement, setting 14-23
initial mesh
source from current or other design 12-10
initial mesh, reverting to 11-7
in-plane movement mode 5-68
input signal, for time domain reports 15-89
input time signal duration 12-22
inserting
line segments 5-6
inserting designs 4-2
integration lines
and multiple modes 7-10
defining 7-10
deleting 7-11
duplicating 7-11
guidelines for defining 7-10
modifying 7-11
reversing direction of 7-11
swapping endpoints 7-11
Interpolating sweep 16-20
Interpolating sweeps
modifying matrix data 15-8
interpolating sweeps
error tolerance 12-18
intersecting faces 5-50
intersecting objects 5-49
intrinsic functions 3-38
introduction to HFSS 1-1
invalid excitations 7-32
isosurface display 15-36

J

JEDEC 4-point bondwires 16-53
JEDEC 5-point bondwires 16-53
joining objects 5-47

L

lambda 16-32
lambda refinement
setting 12-11
lande G factor 16-80
layered impedance boundaries
and frequency dependence 16-68
assigning 6-19
external vs. internal 16-65
impedance calculation 16-65
impedance calculation formula 16-66
surface roughness calculation 16-67
legacy HFSS projects
opening 3-4
translation overview 3-4
length
measuiing 5-75
length of transmission line
adding to ports 7-37
subtracting from ports 7-37
length-based refinement
inside objects 11-3
on faces 11-2
libraries
editing methods for user and system 8-19
lighting 10-13
line segments
inserting 5-6
linear constraints
deleting 14-40
modifying 14-39

overview 16-100
setting 14-39
linear count sweep definitions 14-4
linear materials
adding to libraries 8-5
properties of 8-5
linear polarization 16-43
linear step frequencies 12-20
linear step sweep definitions 14-4
lines
between grid points 10-17
converting to arcs 5-53
converting to splines 5-53
covering 5-44
drawing center-point arc segments 5-4
drawing straight segments 5-2
drawing three-point arc segments 5-3
field plots on 15-38
integration 7-10
terminal voltage 7-12
value vs. distance plots 15-92
linking from simulations 7-15
Linux
system requirements 1-5
local SAR
calculation of 16-26
local variables
adding 3-36
units in definition 3-36
log of solution tasks 15-7
lossy ground planes 16-67
low-order solution basis 12-13
Ludwig-3 16-45
Lumped Port wizard 7-7
lumped ports
for modal solutions 7-7
for terminal solutions 7-8
guidelines for assigning 16-70
overview 16-70
lumped RLC boundaries
and frequency dependence 16-68
and frequency sweeps 16-65
assigning 6-18
conditions applied at 16-65
overview 16-65
lumps, multiple 5-52

M

M3DFS
as source of field information 7-29
MAFET Consortium 15-10
mag margin 16-15
magnetic bias
non-uniform 16-75
uniform 16-74
magnetic bias sources
assigning to ferrites 7-29
magnetic loss tangent 16-79
defining anisotropy tensors 8-7
magnetic saturation 16-79
magnifying objects 10-4
magnitude
setting maximum change for matrix entries 12-13
magnitude margin 15-5
markers
delta between markers 15-88
markers, point plot 15-37
master boundaries
assigning 6-15
guidelines for assigning 16-63
master output
overview 16-104
matching boundaries
assigning 6-15
defining coordinate systems 16-63
guidelines for assigning 16-63
material browser
accessing 8-1
material characteristics
magnetic loss tangent 16-79
permeability 16-77
material properties
anisotropic 8-5
changing units of 8-5
conductivity 16-78
defining frequency dependent 8-8
defining variables for 8-8

delta H 16-80
dielectric loss tangent 16-78
lande G 16-80
magnetic loss tangent 16-79
magnetic saturation 16-79
permeability 16-77
permittivity 16-77
simple 8-5
types of 8-5
using expressions for 8-11
using functions for 8-11
using variables for 8-8
materials
about ferrites 16-79
adding to library 8-5
assigning to objects 8-1
copying, cloning 8-14
deleting 8-15
exporting to libraries 8-16
filtering 8-18
global 8-19
modifying 8-12
removing from libraries 8-15
search by name 8-4
search by property 8-4
solving inside an object 8-3
solving on object surface 8-3
sorting 8-17
user-defined database 8-19
validating 8-13
viewing 8-12
mathematical functions
See functions 3-41
matrices, admittance
computing from S-parameters 16-32
method of calculation 16-32
matrices, impedance
method of calculation 16-31
relationship to S-parameters 16-27
matrix convergence, setting 12-12
matrix data
display format 15-9
exporting 15-9
for design variations 15-15
modifying frequencies 15-8
renaming 15-11
viewing 15-8
matrix entries
selecting convergence criteria 12-12
max delta (Mag S) 15-5
max delta (phase S) 15-6
max U 16-47
maximum delta Energy
setting 12-8
viewing results 15-4
maximum delta frequency 15-6
maximum delta S
setting 12-8
viewing results 15-3, 15-4
maximum far-field data
computing 15-71
overview 16-37
maximum near-field data
computing 15-67
overview 16-36
maximum number of iterations
setting for a sensitivity analysis 14-19
setting for an optimization 14-10
setting for statistical analysis 14-24
maximum number of passes, setting 12-7
maximum number of solutions 12-18
maximum refinement
solution setup options 12-11
Maximum refinement per pass
setting maximum 12-11
maximum step size
how used by Optimetrics 16-102
setting for optimization analysis 14-15
maximum variable
Optimetrics calculation of 14-14, 14-22
maximum variable value
changing for all setups 14-27
overriding for a sensitivity setup 14-22
overriding for all optimization setups 14-15
overriding for all sensitivity setups 14-22
overriding for an optimization setup 14-15
Maxwell Spice files
exporting to 15-11

Measure Mode
distance between two points 5-76
Measure mode
area 5-77
length 5-75
position 5-74
volume 5-77
memory
setting hard limit 12-28
setting soft limit 12-27
used during solution 15-7
menu bar
overview 1-6
menus
shortcut menus 1-8
mesh
color in plots 15-74
for discrete frequency sweep 16-19
for fast frequency sweep 16-18
matching on master/slave boundaries 16-63
plotting 15-74
purpose of 16-2
size vs. accuracy 16-3
mesh generation
and surface approximation settings 16-7
Copy geometric equivalent meshes options 14-34
copy geometrically equivalent meshes 14-34
process 16-4
reverting to initial mesh 11-7
Mesh operations
model resolution 11-6
mesh operations
applying without solving 11-8
defining 11-1
modifying surface approximation 11-5
surface approximation overview 16-7
mesh plots
color of mesh 15-74
creating 15-74
setting attributes 15-74
tetrahedra scale factor 15-74
transparency 15-74
mesh refinement
based on material-dependent wavelength 12-11
defining mesh operations 11-1
maximum refinement per pass 12-11
on ports 16-11
setting lambda refinement 12-11
without solving 11-8
meshing region 16-8
Message window
about 1-17
displaying 1-17
Min and Max focus
SNLP optimizer 14-16
minimum frequency
failure to solve 12-22
minimum frequency, setting 12-5
minimum number of converged passes, setting 12-12
minimum number of passes, setting 12-12
minimum rise time 12-22
minimum step size
how used by Optimetrics 16-102
setting for optimization analysis 14-15
minimum variable value
changing for all setups 14-27
Optimetrics calculation of 14-14, 14-22
overriding for a sensitivity setup 14-22
overriding for all optimization setups 14-15
overriding for all sensitivity setups 14-22
overriding for an optimization setup 14-15
mirroring objects 5-38
modal solutions
assigning lumped ports 7-7
assigning wave ports 7-3
selecting 4-3
modal S-parameter reports 15-80
model analysis
3D modeler menu command 5-22
model resoliution
mesh operations 11-6
modes 16-11
and lumped ports 7-7
field pattern conversion 16-12
multiple at wave ports 7-3
multiple modes and integration lines 7-10
modes, multiple
for multi-conductor ports 16-12

See multiple modes 16-63
modifying in Properties window 1-16
modifying objects 5-33
monitoring solutions 13-14
movement mode
3D 5-69
along x-axis 5-70
along y-axis 5-71
along z-axis 5-71
choosing 5-68
in plane 5-68
out of plane 5-68
moving faces
a specified distance 5-46
along a vector 5-46
along the normal 5-46
moving holes 5-46
moving objects 5-37
moving the cursor
along x-axis 5-70
along y-axis 5-71
along z-axis 5-71
in 3D space 5-69
in plane 5-68
out of plane 5-68
selecting movement modes 5-68
multiple modes
and lumped ports 7-7, 16-70
and symmetry planes 16-63
at wave ports 7-3

N

named expressions
plotting 15-30
near field
plotting on spherical cross-sections 16-35
near fields
and radiation boundaries 15-66
calculation of 16-34
computing max parameters 15-67
line setup 15-67
plotting quantities 15-102
reports 15-80
sphere setup 15-65
new projects, creating 3-3
Next Behind command 5-64
noise, cost function 16-100
nominal design 14-1
non-adaptive solution
single frequency 16-16
non-model objects 5-20
non-uniform magnetic bias 16-75
normalized distance
overview 15-92
notes
saving with project 3-26
number of passes
setting maximum 12-7
setting minimum 12-12
setting minimum converged 12-12
number of processors, setting 12-26
number of time points 12-22

O

objects
associating with faces 5-81
bondwires 5-18
boxes 5-12
center-point arcs 5-4
circles 5-7
cones 5-13
converting polyline segments 5-53
creating from faces 5-50
creating from intersections 5-49
creating object lists 5-59
cylinders 5-11
deleting parts on a plane 5-51
drawing relative to 5-79
duplicates and parents 5-39
ellipses 5-8
equation based curve 5-7
for post processing 5-20
helices 5-15
modifying 5-33
non model 5-20
planes 5-19

points 5-19
polylines 5-5
rectangles 5-9
regular polygons 5-10
regular polyhedrons 5-13
separating 5-52
spheres 5-11
spirals 5-16, 5-17
splines 5-4
straight lines 5-2
three-point arcs 5-3
toruses 5-14
types of 5-2
ways to select 5-57
octave count sweep definitions 14-4
offsetting objects 5-38
ohms per square 16-57
old HFSS projects, opening 3-4
open objects 5-2
opening
existing projects 3-4
field overlay plots 15-40
legacy HFSS projects 3-4
recent projects 3-4
operating systems
HP-UX 1-5
Sun Solaris 1-4
Windows 1-2
Optimetrics
Copy geometric equivalent meshes 14-34
overview 14-1
tuning a variable 14-29
types of analyses 14-1
viewing analysis results 15-15
viewing solution data 15-15
optimization analysis
choosing variables to optimize 3-42
optional settings 14-9
overview 16-98
plotting cost vs. iteration results 15-17
setting up 14-9
viewing cost vs. iteration results 15-17
optimization setups
adding 14-9
adding a cost function 14-10
procedure for defining 14-9
setting a goal 14-10
setting the max. iterations 14-10
solving 13-1
optimizers
choosing between 16-98
Pattern Search 16-98
Quasi Newton 16-98
options
setting in HFSS 3-27
orientation
creating new view directions 10-11
deleting view directions 10-12
setting in view window 10-11
orthographic view 10-14
outer boundary type 16-68
out-of-plane movement mode 5-68
output parameter
adding to sensitivity setup 14-20
plotting results 15-18
setting calculation range 14-21
specifying solution quantity for 14-20
viewing results in table format 15-17
output variable
viewing convergence 15-4
output variable convergence
Setup Context for field quantities 12-9
output variables
deleting 15-106
specifying 15-105

P

panning the view 10-3
parameterizing
See variables 3-35
parameters
assigning variables to 3-42
parametric analysis
overview 16-97
setting up 14-3
solution quantity results 15-16
parametric setup

adding 14-3
overview 14-3
parametric setups
adding sweep definitions 14-3
adding to a design 14-3
plotting solution quantity results 15-16
setting the calculation range 14-7
solution quantity results 15-16
solving 13-1
solving before optimization 14-17
solving before sensitivity analysis 14-23
solving during optimization 14-17
solving during sensitivity analyses 14-23, 14-28
specifying a solution setup 14-6
specifying solution quantities for 14-6
using results for optimization 14-17
using results for sensitivity analysis 14-23
parent objects
and duplicates 5-39
pasting objects 5-35
Pattern Search optimizer 16-98
pausing a script 17-2
peak directivity 16-47
peak gain 16-47
peak phasors
and gap sources 16-25
and incident waves 16-25
calculating 16-24
peak realized gain 16-48
perfect conductors, modeling 6-3
perfect E boundaries
and symmetry planes 16-62
assigning 6-3
impedance multiplier for 16-30
overview 16-56
perfect H boundaries
and symmetry planes 16-62
assigning 6-4
impedance multiplier for 16-30
perfectly matched layers, See PML boundaries 6-8
permeability 16-77
anisotropic 16-81
permittivity 16-77
anisotropic 16-81
perspective view 10-14
phase
modifying for field overlays 15-29
setting for excitations 15-21
setting maximum change for matrix entries 12-13
phase angle
in the calculator 15-52
phase animations 15-24
phase constant 16-32
phase delay
defining for matching boundaries 6-16
phase difference
defining for matching boundaries 6-16
phase margin 15-5, 16-15
phasors
peak 16-24
RMS 16-24
piecewise linear functions
dataset expressions in 3-40
for material properties 8-8
using in expressions 3-40
planes
created with coordinate system 5-81
default 5-19
drawing 5-19
setting the grid plane 10-18
planes of periodicity, modeling 16-63
planes of symmetry, modeling 6-14
play panel 15-26
plots
convergence data 15-6
deleting field overlays 15-40
distribution results for statistical analyses 15-18
hiding 15-39
mesh 15-74
modifying field overlays 15-32
modifying field quantities 15-31
named expressions 15-30
on spherical cross-sections 16-35
opening field overlays 15-40
parametric solution quantity results 15-16
saving field overlays 15-39
plotting
basic quantities 15-29

characteristic port impedances 15-96
derived quantities 15-30
group delay 15-96
on spherical cross-sections 16-35
TDR impedance 15-96
VSWR 15-96
PML boundaries
applications for 16-58
assigning 6-8
creating automatically 6-8
creating manually 6-10
guidelines for assigning 6-11
material tensors at 16-59
modifying 6-12
recalculating materials 6-12
specifying boundaries at 16-61
point of reference 5-68
point plots
markers 15-37
points
measuring distance between 5-76
points, drawing 5-19
polar
coordinates on grid 10-17
setting as grid type 10-17
polar plots
creating 2D 15-83
creating 3D 15-84
creating radiation patterns 15-87
creating Smith charts 15-86
information displayed 15-84
polarization 16-43
circular 16-45
linear 16-43
Ludwig-3 16-45
spherical 16-44
polarization ratio 16-46
polarizing E-fields 16-69
polygons, drawing 5-10
polyhedrons, drawing 5-13
polylines
center-point arcs 5-4
connecting between planes 5-45
connecting with sheet objects 5-45
converting segments 5-53
converting to sheet objects 5-44
covering 5-44
defining sweep paths 5-43
drawing 5-5
plotting value vs. distance 15-92
setting up near-field 15-67
spline segments 5-4
straight line segments 5-2
three-point arcs 5-3
port field accuracy
overview 16-13
port impedance, plotting in time domain 15-89
port solutions
overview 16-10
setting only 12-4
ports
and anisotropic materials 16-84
and mesh refinement 16-11
assigning lumped ports 7-7
assigning wave ports 7-3
internal to model 16-70
multiple 16-12
scaling the magnitude 15-21
setting the phase 15-21
use absorbing boundary 12-14
ports only solutions
setting 12-4
position
measuring 5-74
post processing
overview of options 15-1
viewing convergence data 15-2
viewing matrix data 15-8
viewing profile data 15-7
post processing objects
points 5-19
power
accepted 16-49
incident 16-50
radiated 16-48
power flow
and perfect E symmetry planes 16-30
and perfect H symmetry planes 16-30

Poynting vector
for peak phasors 16-25
for RMS phasors 16-25
primary sweep
modifying the variable 15-91
specifying for 2D rectangular plots 15-81
specifying for 3D polar plots 15-85
specifying for 3D rectangular plot 15-82
specifying for data tables 15-87
specifying for radiation patterns 15-88
priority of boundaries, changing 6-25
priority, changing simulation 13-15
problem region 16-8
profile information
for design variations 15-15
for Optimetrics solutions 15-15
viewing 15-7
Progress window
monitoring solutions 13-14
Project Manager window
overview 1-11
showing 1-11
project tree
auto expanding 1-12
field overlays 15-29
field plot folders 15-29
showing 1-11
project variables
adding 3-35
and material properties 8-8
deleting 3-36
naming conventions 3-35
units in definition 3-35
projection view 10-14
projects
creating new 3-3
default names 3-1
managing 3-1
opening existing 3-4
opening legacy HFSS 3-4
opening recent 3-4
renaming 3-10
saving 3-8
saving active 3-9
saving automatically 3-10
saving copies 3-9
saving new 3-8
saving notes 3-26
propagation constant 16-10
and de-embedding 16-33
and frequency 16-12
plotting 15-96
viewing 16-32
propagation of higher order modes 16-12
properties
report backgrounds 15-78
Properties window
modifying command properties 1-16
set to open after drawing objects 1-15
PSpice files
exporting to 15-11
Purge History command
3D Modeler 5-54

Q

Q factor 16-22
viewing results 15-13
quality factor 16-22
quantities
plotting far field 15-97
plotting field 15-97
plotting near field 15-102
plotting S-parameter 15-95
Quasi Newton optimizer 16-98
queued simulations
removing 13-14
viewing 13-14

R

radiated fields
calculation of 16-34
post-processing capabilities 15-65
radiated power 16-48
radiation boundaries
and far fields 15-67
and near fields 15-66

assigning 6-6
conditions applied at 16-58
FSS surfaces 6-6, 6-9, 6-10
guidelines for assigning 16-58
overview 16-58
radiation efficiency 16-50
radiation intensity, maximum 16-47
radiation patterns, creating 15-87
radius, on polar grid 10-17
RAM
available to HFSS 12-27
used during solution 15-7
RAM requirements
HP-UX 1-5
Sun Solaris 1-4
Windows 1-2
real frequency only convergence 12-9
reassigning boundaries 6-24
reassigning excitations 7-32
recording a script 17-1
rectangles, drawing 5-9
rectangular plots
creating 2D 15-81
creating 3D 15-82
of parametric solution quantity results 15-16
reference point
moving relative to 5-68
selecting 5-68
reflection
of higher order modes 16-12
reflection-free termination, and PMLs 16-58
Refresh Sweeps
Traces dialog, Sweeps tab 15-93
registers 15-44
regular arrays
calculation of 16-40
scan specification 16-40
setting up 15-70
regular polygons, drawing 5-10
regular polyhedrons, drawing 5-13
relative coordinate system
about 5-78
creating 5-79
operations affecting 5-78
relative coordinates, entering 5-66
relative permeability 16-77
defining anisotropy tensors 8-6
relative permittivity 16-77
defining anisotropy tensors 8-6
viewing 16-32
renaming
matrix data 15-11
renaming projects 3-10
rendering objects
as shaded solids 10-10
as wireframes 10-10
setting default for 10-10
renormalizing S-matrices 7-36
report types, selecting 15-80
Reports
background properties 15-78
reports
adding traces 15-90
creating 15-77
creating 2D polar plots 15-83
creating 2D rectangular plots 15-81
creating 3D polar plots 15-84
creating 3D rectangular plots 15-82
creating data tables 15-86
creating radiation patterns 15-87
creating Smith charts 15-86
display types 15-81
finding delta between markers 15-88
modifying data in 15-78
overview 15-77
plotting far-field quantities 15-97
plotting field quantities 15-97
plotting near-field quantities 15-102
selecting a function 15-93
specifying time or frequency domain 15-78
sweeping variables 15-91
types of 15-80
value vs. distance 15-92
reprioritizing
boundaries 6-25
Reset command, in Tuning dialog box 14-30
resistive surfaces
modeling 6-5

modeling layered structures 6-19
resistors
modeling lumped 6-18
resizing holes 5-46
resizing objects 5-41
re-solving a problem 13-17
resonant frequency 16-22
resuming a script 17-2
rise time
for time domain reports 15-89
rise time, minimum 12-22
RMS
and radiated power 16-25
RMS phasors
calculating 16-24
rotating
and sweeping objects 5-43
objects 5-37
the view 10-2
roughness, surface 16-67
rounded corners
removing 5-46
running a script 17-2

S

SAR
calculating 16-26
modifying settings 15-31
saving
field overlay plots 15-39
tuned states 14-30
saving fields
at all frequency points 12-20
at specific frequency points 12-21
for a parametric setup 14-31
for a sensitivity setup 14-32
for a statistical setup 14-31, 14-33
for a tuning analysis 14-31, 14-32
for all Optimetrics setups 14-31
saving projects 3-8
active projects 3-9
automatically 3-10
new projects 3-8
saving copies 3-9
scalar field plots
cloud plots 15-36
creating 15-30
isosurface display 15-36
transparency 15-37
scalar operations 15-52
scale
modifying for field overlays 15-34
scaling
magnitude of excitations 15-21
objects 5-41
scan angles
defining for slave boundaries 6-16
scripts
pausing 17-2
recording 17-1
resuming 17-2
running 17-2
stop recording 17-1
stopping execution of 17-3
secondary sweep
modifying the variable 15-91
specifying for 3D polar plots 15-85
specifying for 3D rectangular plot 15-82
seeding the mesh 16-5
Select All command 5-57
Select All Visible command 5-57
selecting
all object faces 5-60
all visible objects
selecting
all objects in design 5-57
clearing a selection 5-64
edges 5-62
face behind 5-64
faces 5-60
faces by name 5-61
multiple objects 5-57
object behind 5-64
objects 5-57
objects by name 5-58
vertices 5-63
sensitivity analysis

choosing variables to include 3-43
optional settings 14-19
overview 16-104
setting up 14-19
sensitivity setups
adding 14-19
adding an output parameter 14-20
procedure for defining 14-19
setting initial displacement 14-23
setting the max. iterations 14-19
separating bodies 5-52
setting object color 5-34
setting object transparency 5-35
setting outline color 5-34
setting up designs 4-1
setting up projects 4-1
Setup Context dialog
output variable convergence 12-9
setups, dependent solution 12-2
setups, solution 12-1
sheet objects 5-2
shortcut keys
CTRL+A 5-57
shortcut menus
overview 1-8
Show Queued Simulations command
using 13-14
showing
selections 10-9
some objects 10-9
shrinking objects 10-4
signal rise time 12-22
signals
minimum rise time 12-22
simulations
changing priority of 13-15
monitoring 13-14
re-solving 13-17
running Optimetrics 13-1
starting 13-1
stopping 13-16
single frequency points 12-21
single frequency solution 16-16
method for computing 16-16
single frequency solutions
specifying 12-2
skin depth-based refinement
creation of layers 16-6
on faces 11-4
slave boundaries
and frequency dependence 16-68
assigning 6-16
guidelines for assigning 16-63
S-matrices
de-embedding 7-37
renormalizing 7-36
S-matrix
for differential signals 16-73
smetal boundary type 16-68
Smith charts, creating 15-86
snap settings
choosing 5-72
guidelines for 5-73
modes 5-72
setting default 5-72
SNLP optimizer
setting Min and Max focus 14-16
solid objects 5-2
solution data
deleting 15-14
for design variations 15-15
importing 3-22
viewing 15-2
viewing convergence data 15-2
viewing eigenmode 15-13
solution frequency
and Fast sweeps 12-3
setting 12-3
solution process
overview 16-4
tetrahedra used during 15-7
viewing memory used during 15-7
viewing profile data 15-7
viewing tasks performed 15-7
solution quantity
calculation range for optimization 14-12
calculation range for parametric setups 14-7
calculation range for sensitivity 14-21

calculation range for statistical 14-25
plotting parametric setup results 15-16
specifying for cost function 14-11
specifying for output parameter 14-20
specifying for parametric setups 14-6
specifying for statistical setups 14-24
solution settings
adding frequency sweeps 12-15
available memory 12-27
convergence on real frequency 12-9
lambda refinement 12-11
matrix convergence 12-12
maximum delta Energy per pass 12-8
maximum delta S per pass 12-8
maximum number of passes 12-7
maximum refinement per pass 12-11
memory limit 12-28
minimum frequency 12-5
minimum number of converged passes 12-12
minimum number of passes 12-12
number of eigenmodes 12-6
number of processors 12-26
setting up adaptive analyses 12-7
solution frequency 12-3
solving ports only 12-4
solution settings, specifying 12-1
solution settings, specifying dependent 12-2
solution setup
data types for convergence 12-17
waveport adapt options 12-14
solution setups
adding 12-1
adding dependent 12-2
choosing for a parametric analysis 14-6
solution type
overview 16-21
setting 4-3
solutions
after modifying the model 13-17
at single frequency 16-16
changing priority of 13-15
monitoring 13-14
re-solving 13-17
starting 13-1
stopping 13-16
Solutions window
displaying 13-14
solving 13-1
batch solution 1-22
inside an object 8-3
on surface 8-3
parametric setup before optimization 14-17
parametric setup before sensitivity analysis 14-23
parametric setup during optimization 14-17
parametric setup during sensitivity analysis 14-23, 14-28
solving remotely 13-3
sorting materials 8-17
spacing between grid points 10-17
S-parameters
and reflected modes 16-12
de-embedding 16-27, 16-33
plotting quantities 15-95
renormalized 16-27
specific absorption rate
see SAR 15-31
spheres
drawing 5-11
plotting value vs. distance 15-92
setting up infinite 15-68
setting up near-field 15-65
spherical coordinates, entering 5-65
spherical cross-sections 16-35
spherical polar 16-44
spinning the view 10-2
spirals
drawing 5-17
spirals, drawing 5-16
splines
converting to straight lines 5-53
drawing 5-4
splitting objects on a plane 5-51
square (unit of) 16-57
stack 15-44
stack commands
clear 15-45
exch 15-45
loading into 15-46

performing operations on 15-50
pop 15-45
push 15-45
rldn 15-45
rlup 15-45
undo 15-46
starting frequency 12-20
starting variable value
overriding for optimizations 14-14
overriding for sensitivity 14-21
overriding for statistical 14-27
statistical analysis
choosing variables to include 3-44
plotting distribution results 15-18
setting up 14-24
viewing distribution results 15-18
statistical setups
adding 14-24
procedure for defining 14-24
setting the max. iterations 14-24
specifying solution quantities for 14-24
status bar
overview 1-11
step size
determined by Optimetrics 16-102
overview 16-102
setting constraints for optimization 14-15
setting for sweep definitions 14-4
step size between frequencies 12-20
stopping a script 17-3
stopping an analysis 13-16
stopping criteria
matrix convergence 12-12
maximum delta Energy 12-8
maximum delta S 12-8
maximum number of passes 12-7
minimum number of converged passes 12-12
minimum number of passes 12-12
stopping criteria for optimization
acceptable cost 16-100
maximum number of iterations 14-10
minimum step size 16-102
stopping criteria for sensitivity analysis
max. iterations 14-19
stopping frequency 12-20
stopping script recording 17-1
subtracting objects 5-48
surface approximation
guidelines for setting 16-8
modifying settings 11-5
overview 16-7
surface roughness 16-67
surfaces
covering 5-44
creating face lists 5-62
detaching 5-44
uncovering 5-44
sweep definitions
See variable sweep definitions 14-3
sweep variables in reports
modifying values 15-91
normalized distance 15-92
normalized line 15-92
spherical coordinates 15-92
sweeping objects
along a path 5-43
along a vector 5-42
and draft angles 5-42
and twisting 5-43
around an axis 5-41
draft types 5-42
overview 5-41
sweeps
refreshing values 15-93
symmetry boundaries
and multiple modes 16-63
and polarizing E-fields 16-70
assigning 6-14
guidelines for assigning 16-61
overview 16-61
symmetry planes
and impedance 16-63
and multiple modes 16-63
guidelines for modeling 16-61
Perfect E vs. Perfect H 16-62
symmetry planes, modeling 6-14
Sync # column 14-4
synchronizing sweep definitions 14-4

system material libraries 8-19
system requirements
HP-UX 1-5
Sun Solaris 1-4
Windows 1-2

T

TDR Impedance
changing default value 15-89
TDR impedance
plotting 15-96
TDR impedance, plotting 15-89
TDR Options dialog box 15-89
tensor coordinate system 16-74
terminal lines
and differential pairs 7-13
defining 7-12
deleting 7-13
duplicating 7-13
editing 7-13
polarity reference 7-12
reversing direction of 7-13
swapping endpoints 7-13
terminal solutions
assigning lumped ports 7-8
assigning wave ports 7-4
setting 4-3
terminal S-parameter reports 15-80
terminals
scaling the magnitude 15-21
setting a complex impedance 15-21
specifying multiple at ports 7-5
suggested number at ports 7-5
tetrahedra
color in plots 15-74
display options 15-74
field quantities within 16-2
scale factor in plot 15-74
setting number added each pass 12-11
used during solution 15-7
theta, on polar grid 10-17
three-point arcs 5-3
time
real vs. CPU 15-7
viewing for solution tasks 15-7
time domain
specifying for report 15-89
time steps per rise time 12-22
T-junction Getting Started guide 1-24
toolbars
overview 1-8
toruses, drawing 5-14
traces
adding blank 15-91
adding to reports 15-90
categories 15-95
removing 15-90
replacing 15-91
Traces dialog box 15-77
trademark notices 1-26
translating legacy projects 3-4
transparency
assigning to objects 5-34
default for objects 5-35
in scalar field plots 15-37
of mesh plots 15-74
traveling waves
at ports 16-10
tuning
choosing variables to tune 3-43
overview 16-105
tuning analysis
resetting variable values after 14-30
reverting to a state 14-30
saving a state 14-30
setting up 14-29
twisting objects
while sweeping 5-43

U

UDPPrimitiveTypeInfo
data structure 5-31
Unclassified folder
history tree 1-20
uncovering faces 5-44
uniform magnetic bias 16-74

uniting objects 5-47
units
as part of variable definitions 3-35
of impedance boundaries 16-57
setting for design 4-4
User Defined Primitives 5-17, 5-30
user interface
overview 1-6
user material libraries 8-19

V

validating materials 8-13
validation check 3-15
Value command
fields calculator 15-59
value vs. distance plots 15-92
variable
for output convergence 12-9
variable sweep definitions
adding to parametric setups 14-3
manually modifying 14-5
overview 14-3
setting values to solve 14-4
synchronizing 14-4
tracking changes to 14-5
viewing all design variations 14-3
variable-dependent goal 14-13
variables
adding local variables 3-36
adding project variables 3-35
and object parameters 5-2
assigning to material properties 8-8
assigning to parameters 3-42
choosing to optimize 3-42
choosing to tune 3-43
dataset expressions in 3-40
defining sweep definitions 14-3
deleting from design 3-37
deleting from project 3-36
excluding from Optimetrics analyses 14-37
including in functions 3-41
including in sensitivity analysis 3-43
including in statistical analysis 3-44
material properties 8-8
min. and max values for optimization 14-14
min. and max values for sensitivity analysis 14-22
output 15-105
overriding default value for a parametric setup 14-5
overview 3-35
predefined in HFSS 3-41
setting default value 3-36
setting distribution criteria 14-26
setting fixed values 14-38
setting range of values 14-22
setting range of values for optimization 14-14
tuning 14-29
types in HFSS 3-35
updating to optimized values 14-17
.vbs file format 17-1
VBScript
.vbs file format 17-1
vector field plots
creating 15-31
modifying arrows 15-35
vector operations 15-56
vertex, selecting 5-63
vertical cross-sections 15-101
vertices, selecting 5-63
video drivers
recommended 1-2
view direction 10-11
view options
background color 10-15
fit all in view 10-7
lighting 10-13
modifying 10-1
orientation 10-11
pan 10-3
projection 10-14
rotate 10-2
spin 10-2
view direction 10-11
zoom 10-4
zoom on rectangle 10-4
visibility
hiding objects 10-8

of color key 15-33
of field overlay plots 15-39
of mesh on field plots 15-36
showing objects 10-9
visualization options
for boundaries 6-28
for excitations 6-28
voltage differentials
and perfect E symmetry planes 16-30
and perfect H symmetry planes 16-30
voltage sources
assigning 7-27
editing 7-27
setting the phase 15-21
voltage standing wave ratio
plotting 15-96
voltage transform
plotting'plotting
voltage transform 15-96
volume
measuring 5-77

W

Wave Port wizard 7-3
wave ports
for modal solutions 7-3
for terminal solutions 7-4
overview 16-69
specifying multiple modes 7-3
suggested number of terminals 7-5
Waveport adapt options
solution setup 12-14
weight of goals
See goal weight 16-101
W-element data
exporting 15-13
wireframe objects 10-10
working coordinate system
about 5-78
selecting 5-78
Write command
fields calculator 15-60

Y

Y-parameters, plotting 15-96

Z

zero order basis functions 16-2
Zo
changing default impedance 15-89
Zoom
to selected boundary 6-2
to selected excitation 7-2
zooming
in and out 10-4
on rectangular area 10-4
Z-parameters, plotting 15-96
Zpi
definition of 16-29
Zpv
definition of 16-29
Zvi
definition of 16-29