

Getting Started with HFSS™

Probe Feed Patch Antenna



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

March 2015
ANSYS Electromagnetics Suite 16.x

ANSYS, Inc. is
certified to ISO
9001:2008.

Copyright and Trademark Information

© 2015 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, HFSS, and Optimetrics and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

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

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

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


Third-Party Software

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

Published in the U.S.A.

Conventions Used in this Guide

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

- 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 the word copy must be **typed**, then a space must be typed, and then **file1** must be typed.
 - 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 **Enter**” means to press the key labeled **Enter**.
- 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**” the word **copy** must be typed, then a space must be typed, and then name of the file must be typed.
- 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.

Getting Help: ANSYS Technical Support

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

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

Help Menu

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

Contents - click here to open the contents of the online help.

Search - click here to open the search function of the online help.

Index - click here to open the index of the online help.

Context-Sensitive Help

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

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

Table of Contents

1. Introduction

Sample Project: Patch Antenna 1-2

2. Set Up The Project

Launch HFSS 2-2

Set Tool Options 2-2

Insert HFSS design 2-3

 Set Model Units (cm) 2-3

 Set Solution Type (Terminal) 2-4

3. Set Up The Project

Create Substrate 3-2

 Create Infinite Ground 3-4

 Assign Perfect E boundary to the Ground . . . 3-5

 Create Infinite Ground Cut Out 3-6

 Complete the Infinite Ground 3-7

 To complete the ring 3-7

 Create Patch 3-8

 Assign a Perfect E boundary to the Patch . . . 3-9

Create the Coax 3-10

 Create the Coax Pin 3-11

 Assign Excitation 3-13

Create the Probe	3-15
Create Air	3-16
Assign Radiation Boundary	3-17
Create Radiation Setup	3-19

4. Analyze the Model

Add Solution Setup	4-2
Add Frequency Sweep	4-2
Analyze the Model	4-3
Review Solution Data	4-4
Review the Profile Panel	4-4
Review the Convergence Panel	4-5
Review the Matrix Data Panel	4-7
Review the Mesh Statistics Panel	4-7
Create Reports	4-8
Create Terminal Solution Data Report	4-8
Far Field Overlays	4-10
Create Far Field Overlay	4-10

1

Introduction

This document is intended as supplementary material to HFSS for beginners and advanced users. It includes instructions to create, simulate, and analyze a probe feed patch antenna.

This chapter contains the following topics:

- ✓ Sample Project - Probe Feed Patch Antenna

Sample Project: Patch Antenna

This example is intended to show you how to create, simulate, and analyze a probe feed patch antenna using HFSS.

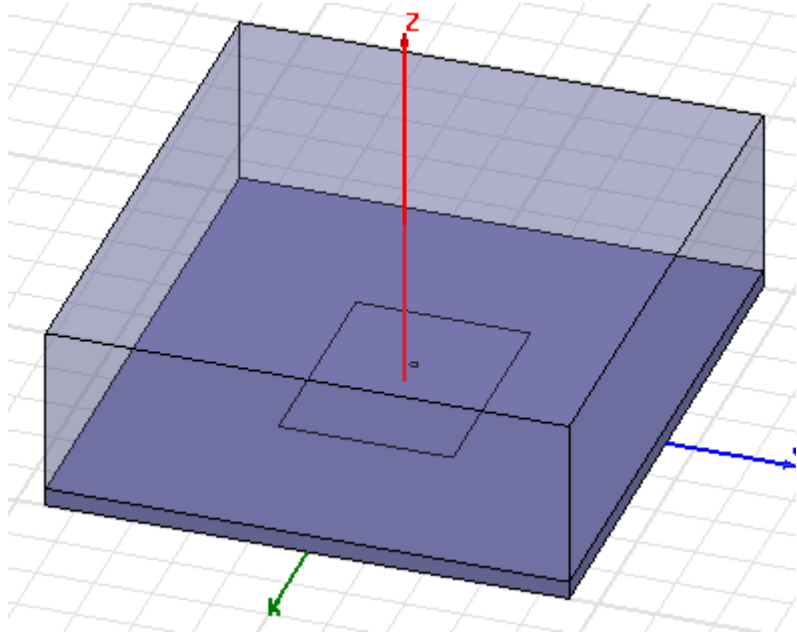


Figure 1. Probe Feed Patch Antenna

1-2 Introduction

2

Set Up The Project

This chapter contains the following topics:

- ✓ Launch HFSS
- ✓ Set Tool Options
- ✓ Insert HFSS design
- ✓ Set Model Units (cm)
- ✓ Set Solution Type (Terminal)

Launch HFSS

- 1 Go to Windows Start > All Programs > ANSYS Electromagnetics > ANSYS Electronics Desktop to launch the application.

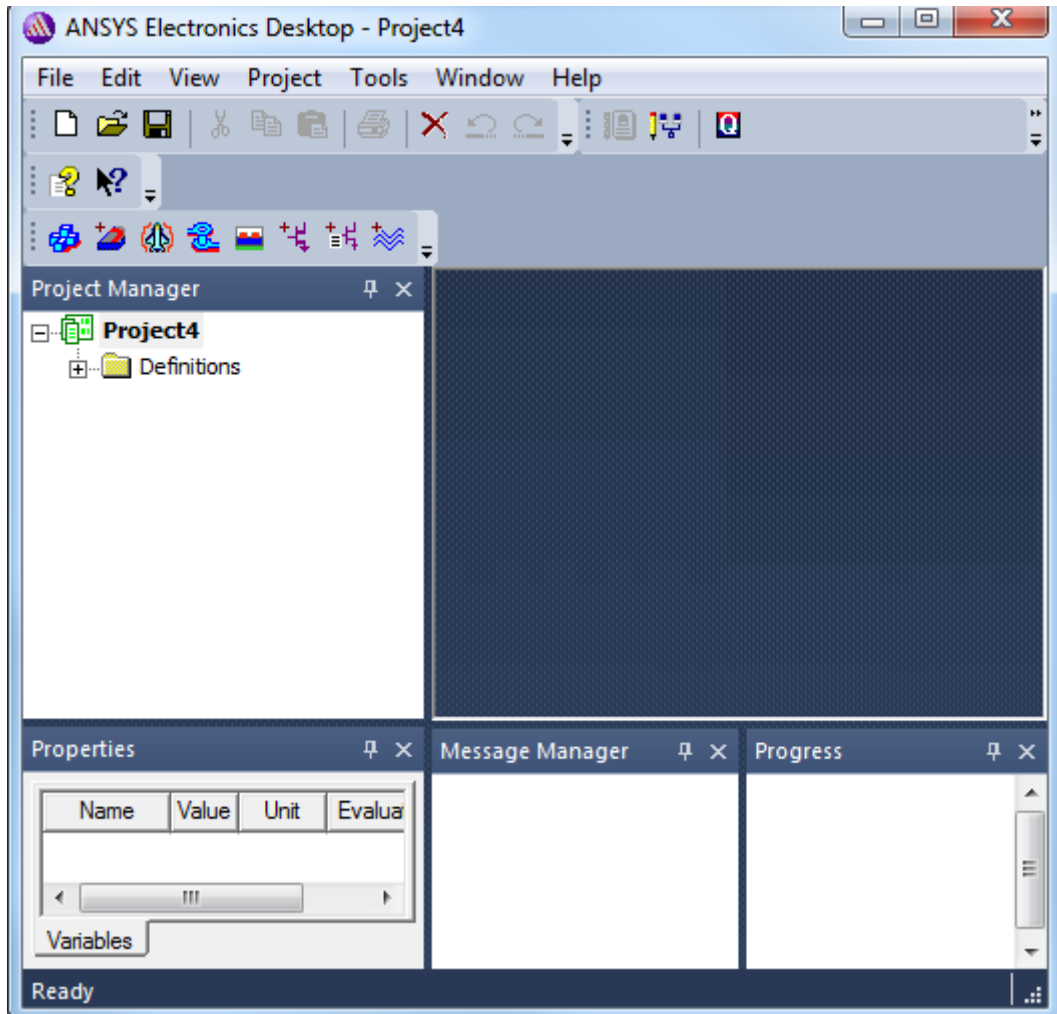


Figure 1. ANSYS Electronics Desktop appears

Note If the application does not list the folder, go to **File** and click **New**. If the **Project Manager** window does not appear, go to **View** and enable it.

2-2 Set Up The Project

Set Tool Options

Verify the options under the **Tools** menu as follows:

- 1 Click **Tools>Options>HFSS Options**.

The **HFSS Options** dialog box appears.

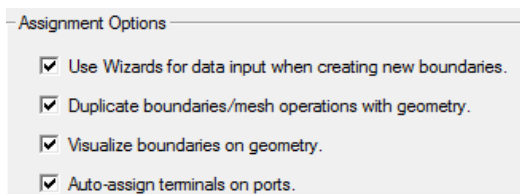


Figure 2. Assignment Options

- 2 On the **General** tab ensure all **Assignment Options** are checked and click **OK** to close the dialog box.
- 3 Click **Tools>Options>Modeler Options**.
The **Modeler Options** dialog box appears.
- 4 On the **Operation** tab check **Automatically cover closed polylines**.
- 5 On the **Drawing** tab check **Edit properties of new primitives** and click **OK**.

Note This option causes a **Properties** dialog box to appear whenever you create a new object.

Insert HFSS design

The icon below represents the **Insert HFSS design (IHd)** option.



Figure 3. IHd

- 1 Expand the project tree.
- 2 Click **Insert HFSS Design** on the toolbar to include it in the project or go to the **Project** menu and select **Insert HFSS Design**.

Note Inclusion of **IHd** modifies the project and hence the asterisk appears on **ProjectN**.

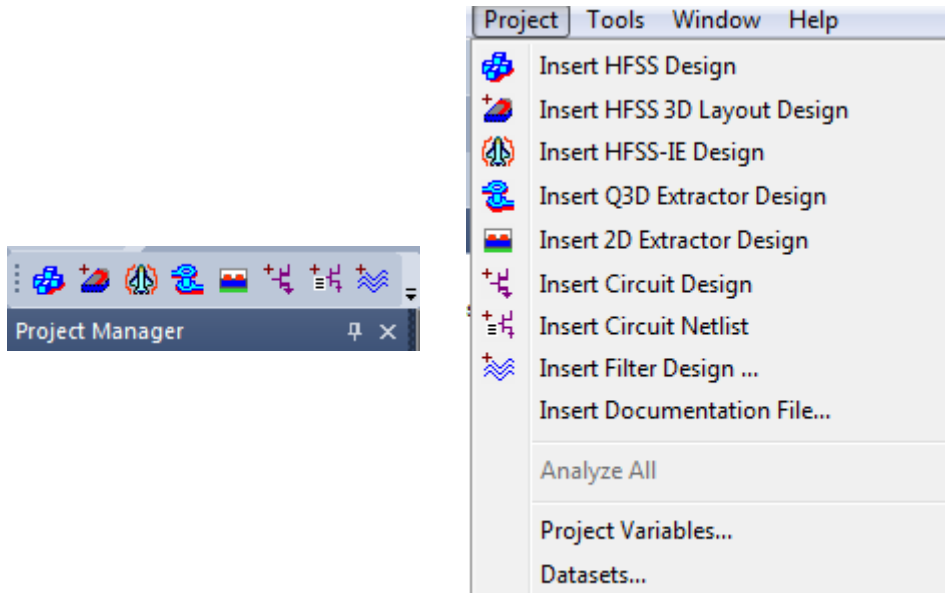


Figure 4. Include HFSS Design type

- 3 Click **ProjectN***, hit **F2**, rename the project and save it.

Set Model Units (cm)

Define the model units as follows:

- 1 On the toolbar, click **Modeler>Units**.

The **Set Model Units** dialog box appears.

- 2 Select **cm** (centimeters) from the **Select units** drop-down menu, and click **OK**.

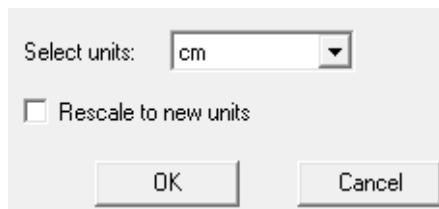


Figure 5. Set Model Units dialog box

2-4 Set Up The Project

Set Solution Type (Terminal)

Specify the design's solution type as follows:

- 1 Right click **HFSSDesign1 (DrivenModal)** and select **Solution Type** from the shortcut menu.

The **Solution Type** dialog box appears.

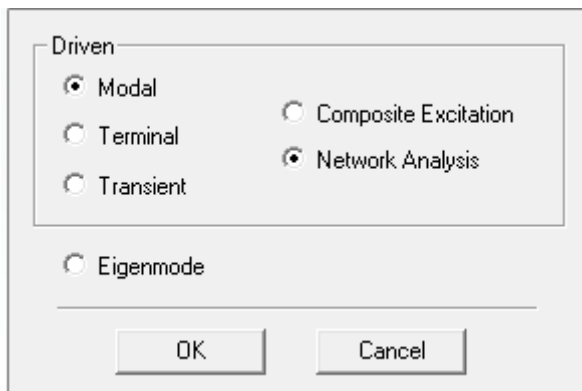


Figure 6. Solution Type dialog box

- 2 Select **Driven Terminal** and click **OK**.

Note Driven Terminal calculates 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.

3

Set Up The Project

This chapter contains the following topics:

- ✓ Create Substrate
- ✓ Create Infinite Ground
- ✓ Assign Perfect E Boundary to the Ground
- ✓ Create Infinite Ground Cut-Out
- ✓ Complete the Infinite Ground
- ✓ Complete the Ring
- ✓ Create Patch
- ✓ Assign Perfect E Boundary to the Patch
- ✓ Create the Coax
- ✓ Create the Coax Pin
- ✓ Assign Excitation
- ✓ Create the Probe
- ✓ Create Air Body
- ✓ Assign Radiation Boundary
- ✓ Create Radiation Setup

Create Substrate

To create the substrate, you will draw a box freehand as follows.

1 Click **Draw>Box**.

The cursor is accompanied by a black square box.

2 Click inside the **Modeler** window to establish the x,y axes and drag the mouse to draw the rectangle.

3 Click the mouse to establish the z axis and drag the mouse along the z-axis to draw the height, and click the mouse to complete the box.

The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-5 , -4.5 , 0	cm	-5cm , -4.5cm , 0cm
	XSize	10	cm	10cm
	YSize	9	cm	9cm
	ZSize	0.32	cm	0.32cm

Figure 7. Command dialog box

4 On the **Command** tab edit the fields as shown in Figure 7 and on the **Attribute** tab rename the box to *Sub1* and select **Edit** from the **Materials** drop down menu.

The **Select Definition** dialog box appears See Figure 8.

5 Type *Rogers RT/duroid 5880 (tm)* in the **Search By Name** field and select the option when it appears in the list.

6 Click **OK** to close the **View/Edit Material** dialog box and repeat the same on the other dialog boxes to exit.

7 Click **View>Fit All>Active View**.

Note As you continue to build the model, whenever you want to fit the view do **Ctrl+D**.

3-2 Set Up The Project

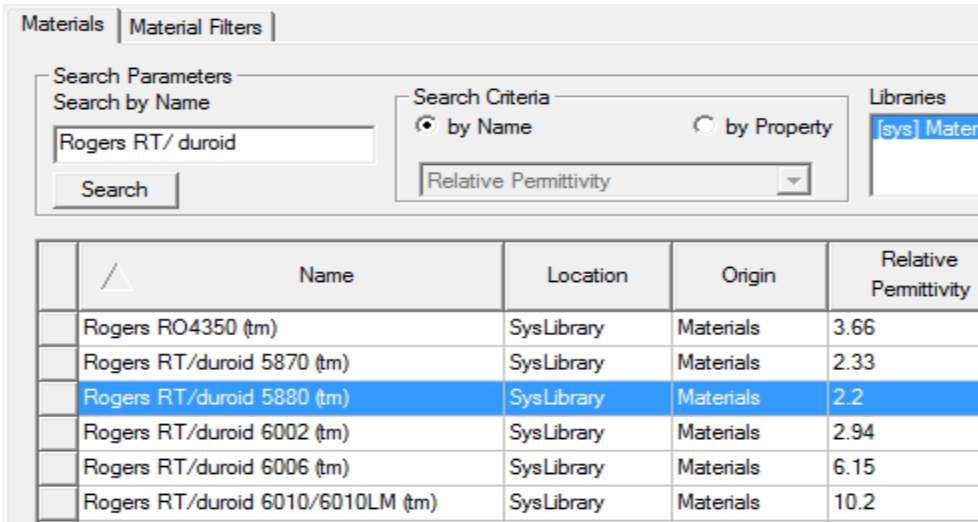


Figure 8. Select Definition window

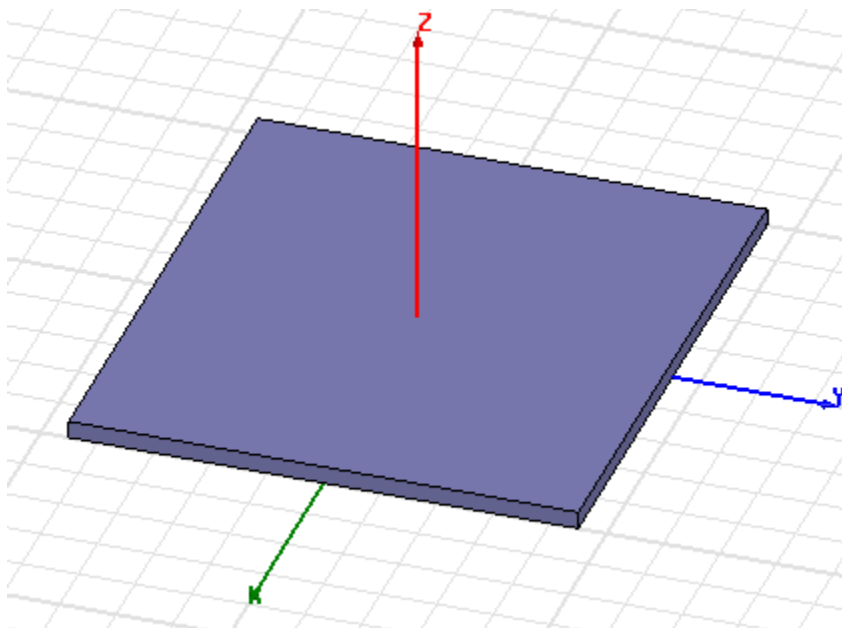


Figure 9. Substrate

Create Infinite Ground

To create the infinite ground, you will draw a rectangle as follows.

- 1 Click **Draw>Rectangle**
- 2 Click inside the modeler window to establish the x,y axes.
- 3 Drag along the x, y axes to draw the rectangle.
The **Properties** dialog box appears.
- 4 Click **OK** to accept the values in the **Properties** dialog box.
- 5 Double-click **CreateRectangle** from the history tree and edit the fields on the **Command** dialog box as in Figure 10.
- 6 Double-click **Rectangle** from the history tree and enter *Inf_GND* in the **Name** field and click **OK**.

	Name	Value	Unit	Evaluated Value
	Command	CreateRectangle		
	Coordinate Sys...	Global		
	Position	-5,-4.5,0	cm	-5cm,-4.5cm,0cm
	Axis	Z		
	XSize	10	cm	10cm
	YSize	9	cm	9cm

Figure 10. Infinite Ground Command dialog

3-4 Set Up The Project

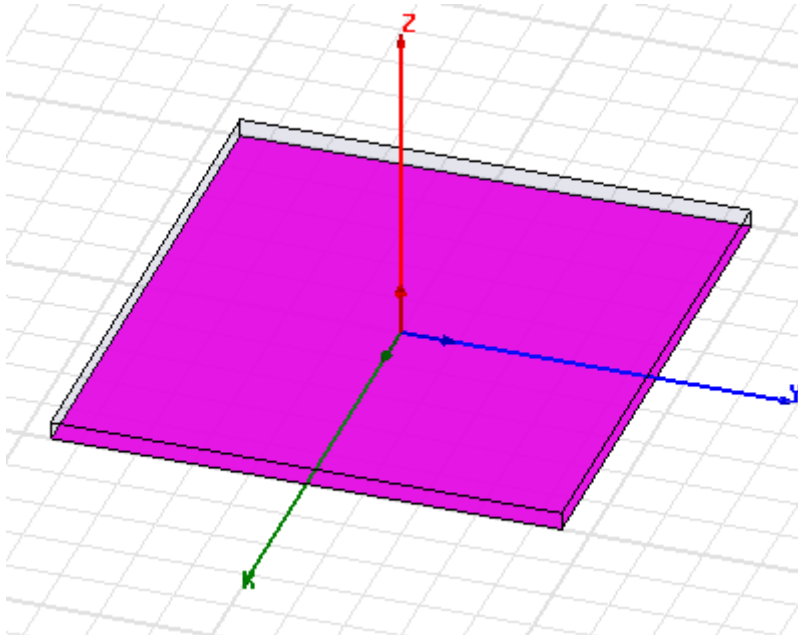


Figure 11. Infinite Ground plane

Assign Perfect E boundary to the Ground

- 1 Click **Edit>Select>By Name**

The **Select Object** dialog box appears.

- 2 Select the object **Inf_GND** and click **OK**.

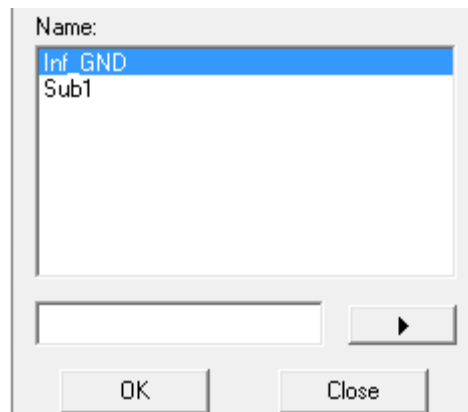


Figure 12. Select Object dialog box

- 3 Right click **Assign Boundary>Perfect E**

The **Perfect E Boundary** dialog box appears.

- 4 Edit the fields as shown in Figure 13 and click **OK**.

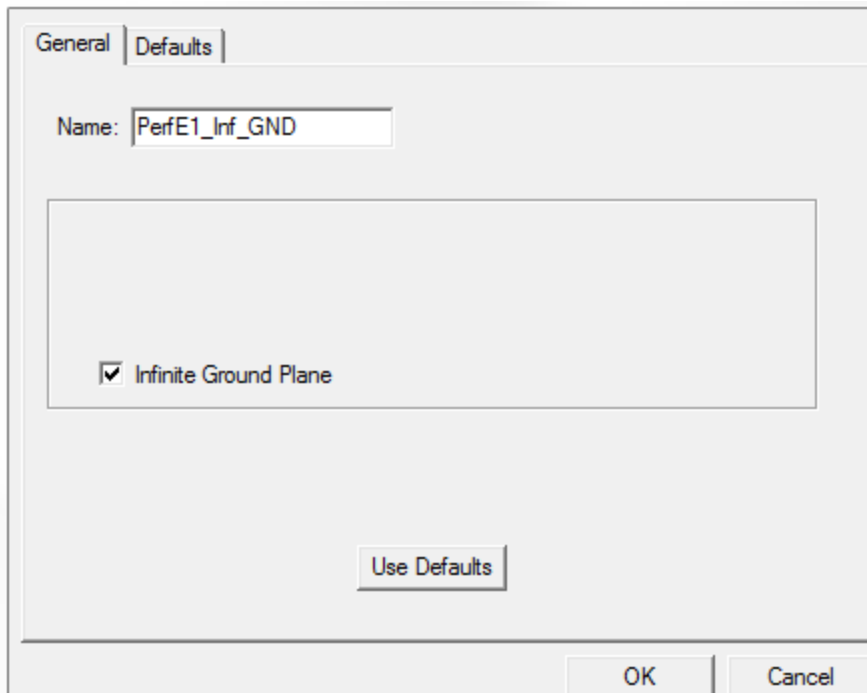


Figure 13. Perfect E Boundary dialog box

Create Infinite Ground Cut Out

To create the cut out perform the following steps.

- 1 Click **Draw>Circle**
- 2 Click in the modeler window to establish the x,y axes and drag the cursor to draw the circle and click **OK** on the **Properties** dialog box when it appears.
- 3 Double-click **CreateCircle** to open the **Command** dialog box and edit the fields as shown in Figure 14 and click **OK**.
- 4 Double-click **Circle** from the history tree and enter **Cut_Out** in the **Name** field and click **OK**.

3-6 Set Up The Project

	Name	Value	Unit	Evaluated Value
	Command	CreateCircle		
	Coordinate Sys...	Global		
	Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
	Axis	Z		
	Radius	0.16	cm	0.16cm
	Number of Seg...	0		0

Figure 14. Ground Cut Out Command dialog

Complete the Infinite Ground

To select the objects `Inf_GND` and `Cut_Out`:

- 1 Click **Edit>Select>By Name**

The **Select Object** dialog box appears.

- 2 Select `Inf_GND` and `Cut_Out` and click **OK**.

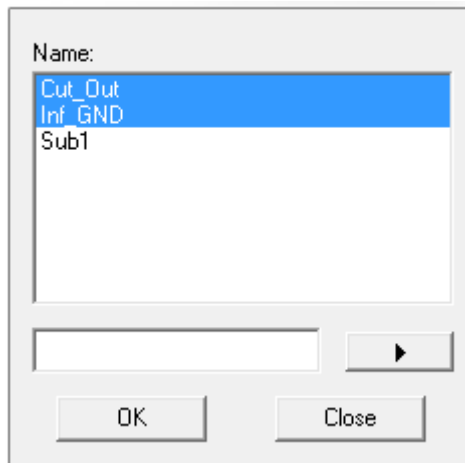


Figure 15. Select Object dialog box

To complete the ring

- 1 Click **Modeler>Boolean>Subtract**

The **Subtract** dialog box appears.

- 2 Set the fields as in Figure 16 and click **OK**.

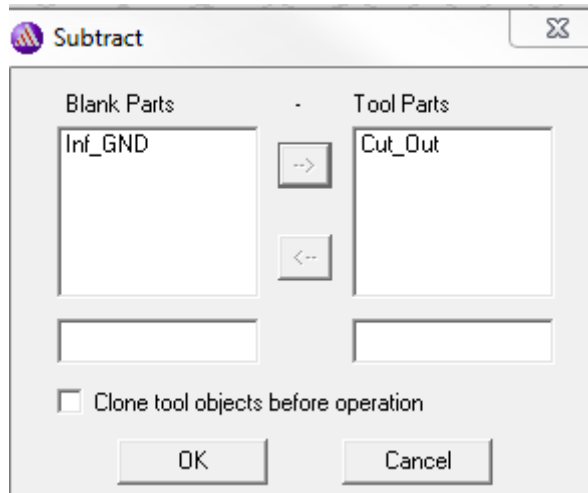


Figure 16. Subtract dialog box

Create Patch

We will describe another method to draw a rectangle here.

- 1 Click **Draw>Rectangle**
- 2 Use the fields on the status bar, and enter the co-ordinates of the center as follows:
X: -2.0, Y: -1.5, Z: 0.32 and press **Enter**.
- 3 Use the fields and enter the opposite corner of the rectangle as follows:
dX: 4.0, dY: 3.0, dZ: 0.0 and press **Enter**.
The **Properties** dialog box appears.
- 4 Click **Attribute** and enter *Patch* in the **Name** field and click **OK**.

3-8 Set Up The Project

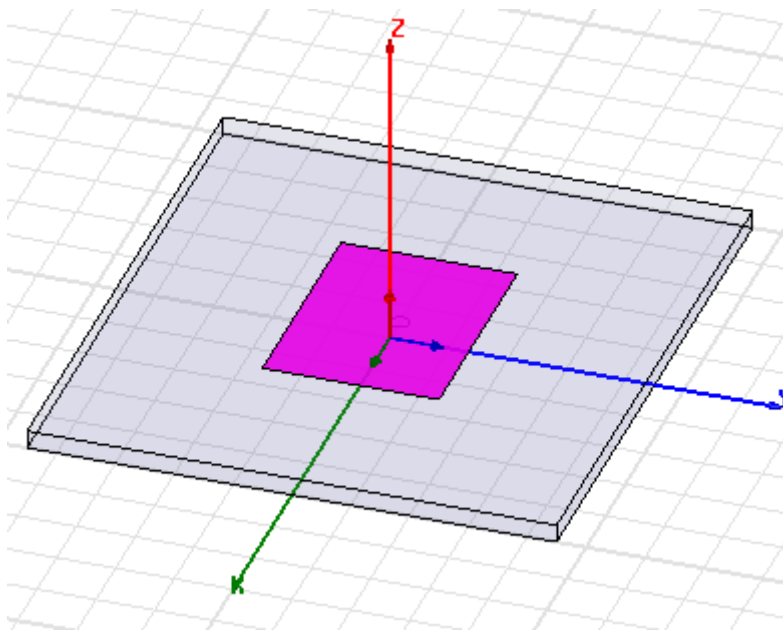


Figure 17. The Patch drawn

Assign a Perfect E boundary to the Patch

- 1 Click **Edit>Select>By Name**

The **Select Object** dialog box appears.

- 2 Select the **Patch** and click **OK**.

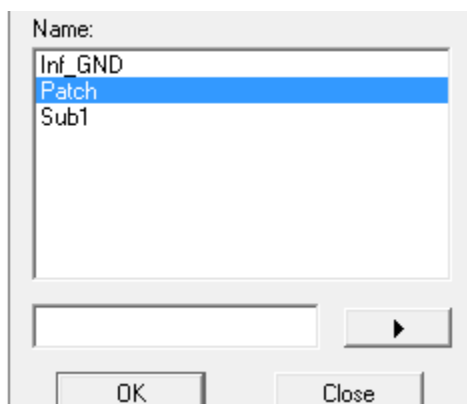


Figure 18. Select Objects dialog box

- 3 Click **HFSS>Boundaries>Assign>Perfect E**

The **Perfect E Boundary** dialog box appears.

- 4 Enter *PerfE_Patch* in the Name field and click OK.

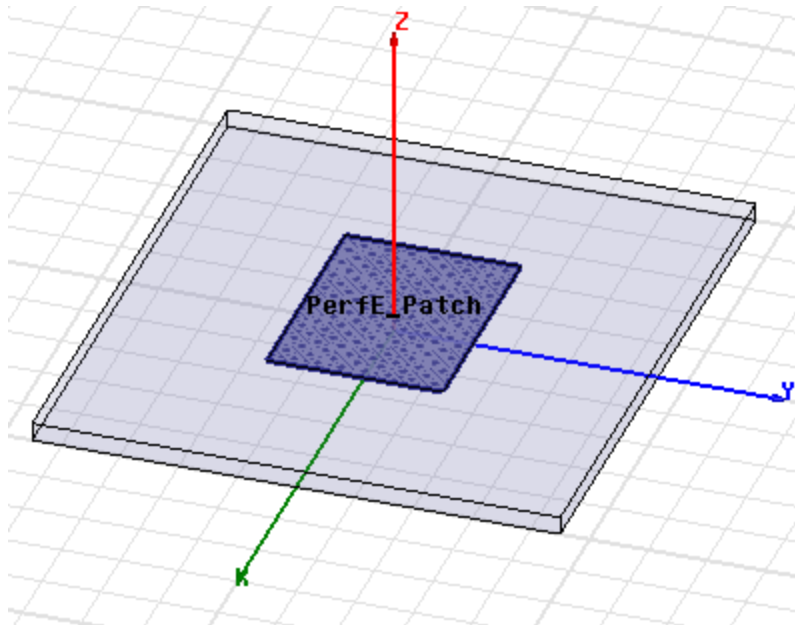


Figure 19. PerfE_Patch applied

Create the Coax

In this section you will create the Coax. You can follow the conventional method of setting the material for the Coax. or set the material before drawing the coax.

- 1 Using the 3D Modeler Materials toolbar, choose vacuum.

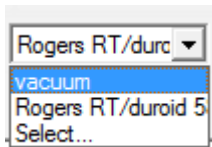


Figure 20. Material drop down

- 2 Click **Draw>Cylinder**
- 3 Draw a cylinder freehand and on the **Command** tab edit the fields as shown in Figure 21
- 4 Click **Attribute**, rename the object to Coax, and click OK.

3-10 Set Up The Project

	Name	Value	U..	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
	Axis	Z		
	Radius	0.16	cm	0.16cm
	Height	-0.5	cm	-0.5cm
	Number of Seg...	0		0

Figure 21. Command dialog for Coax

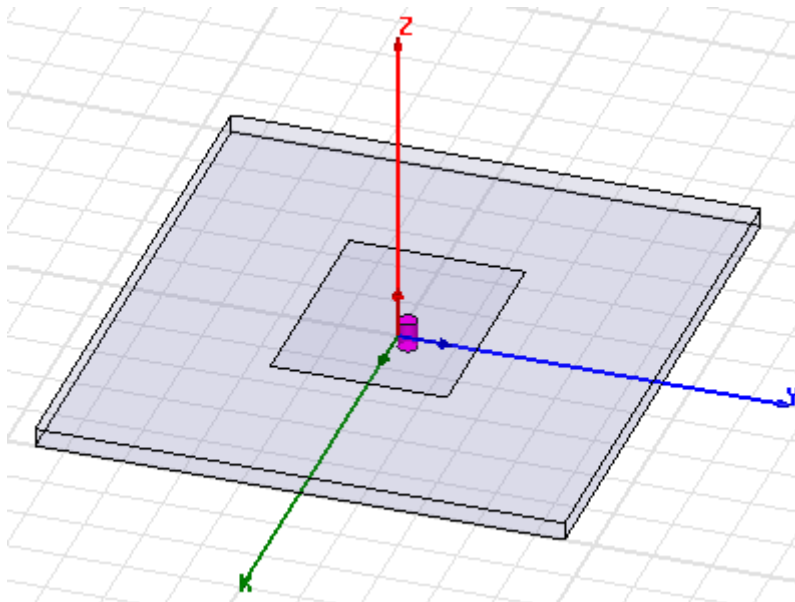


Figure 22. Coax drawn

Create the Coax Pin

To create the coax pin:

- 1 Draw a cylinder free hand and on the the **Command** tab edit the fields as shown in Figure 23.
- 2 On the **Attribute** tab, rename the cylinder to *coaxpin* and select **Edit** from the **Materials** drop-down menu. The **Select Definition** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
	Axis	Z		
	Radius	0.07	cm	0.07cm
	Height	-0.5	cm	-0.5cm
	Number of Seg...	0		0

Figure 23. Command dialog for Coax Pin

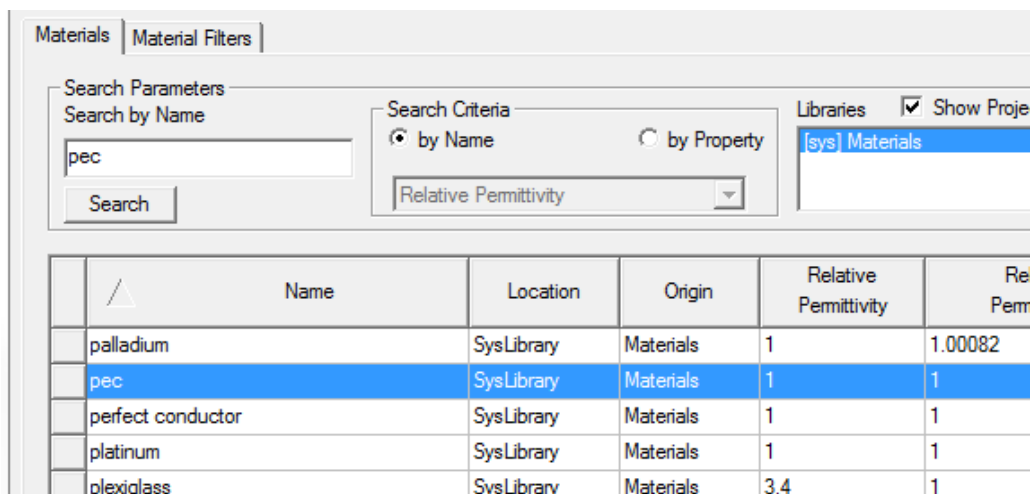


Figure 24. Select Definition window

- 3 Enter **pec** in the **Search By Name** field.
The option **pec** is highlighted in the list.
- 4 Click **OK** to close the **Select Definition** dialog box and repeat the same on the other dialog box to exit.

Note The **Message Manager** shows this message: *Solve inside for object coax pin is unset, due to material assignment change.* You do not solve inside a perfect conductor since the field is zero.

3-12 Set Up The Project

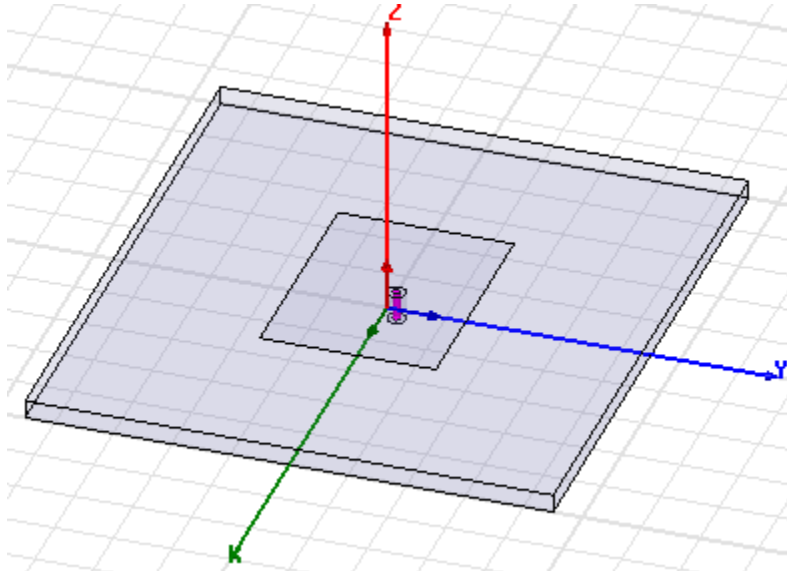


Figure 25. coax pin

Assign Excitation

We can create a circle to access the face where we assign the wave port.

- 1** Click **Draw>Circle**
- 2** Using the coordinate entry fields, enter the center position:
X: -0.5, Y: 0.0, Z: -0.5 and press the **Enter** key.
- 3** Using the coordinate entry fields, enter the radius of the circle:
dX: 0.16, dY: 0.0, dZ: 0.0 and press the **Enter** key.
The Properties dialog box appears.
- 4** On the **Attribute** tab rename Circle1 to **Port1** and click **OK** to close the Properties dialog box.
- 5** Click **Edit>Select>By Name**
This **Select Object** dialog box appears.
- 6** Select **Port1** and click **OK**.
- 7** Right click and select **Assign Excitation>Wave Port**.
The **Reference Conductors for Terminals** dialog box

appears.

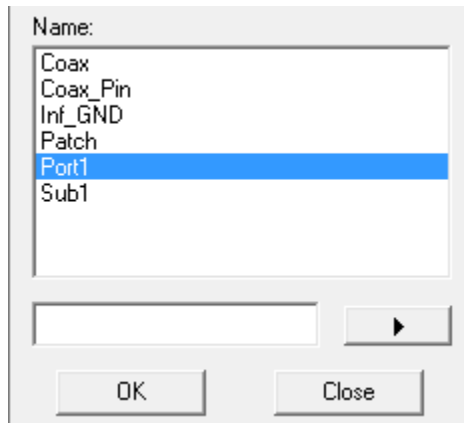


Figure 26. Select Object dialog box

8 Edit the fields as shown in Figure 27 and click OK.

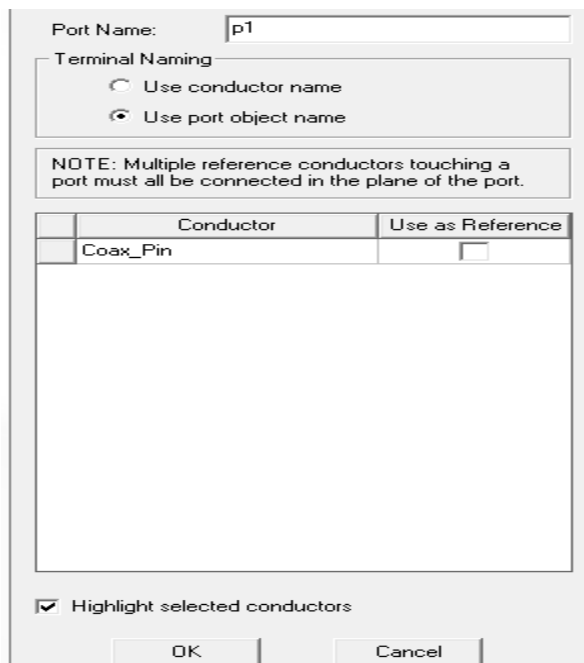


Figure 27. Reference Conductors for Terminals

The terminal is created under **Excitations** for **p1** under the the Project tree.

3-14 Set Up The Project

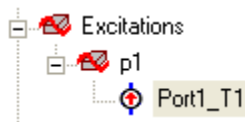


Figure 28. Project Tree

Create the Probe

To create the probe:

- 1 Draw a cylinder freehand and edit the fields on the Command tab as shown in Figure 29.
- 2 On the **Attribute** tab, rename the cylinder to **probe** and set **Material** as *pec*.

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
	Axis	Z		
	Radius	0.07	cm	0.07cm
	Height	0.32	cm	0.32cm
	Number of Seg...	0		0

Figure 29. Command dialog box

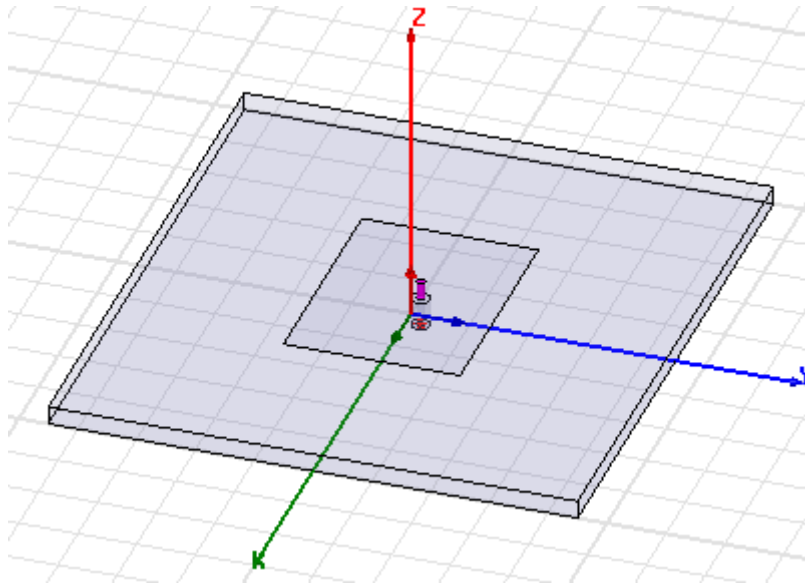


Figure 30. Probe drawn

Create Air

- 1 Draw a box freehand and edit the fields on the Command tab as shown in Figure 31.

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-5 , -4.5 , 0	cm	-5cm , -4.5cm , 0cm
	XSize	10	cm	10cm
	YSize	9	cm	9cm
	ZSize	3.32	cm	3.32cm

Figure 31. Command dialog box

- 2 Click **Attribute** and rename the box to *Air* and click **OK**.

3-16 Set Up The Project

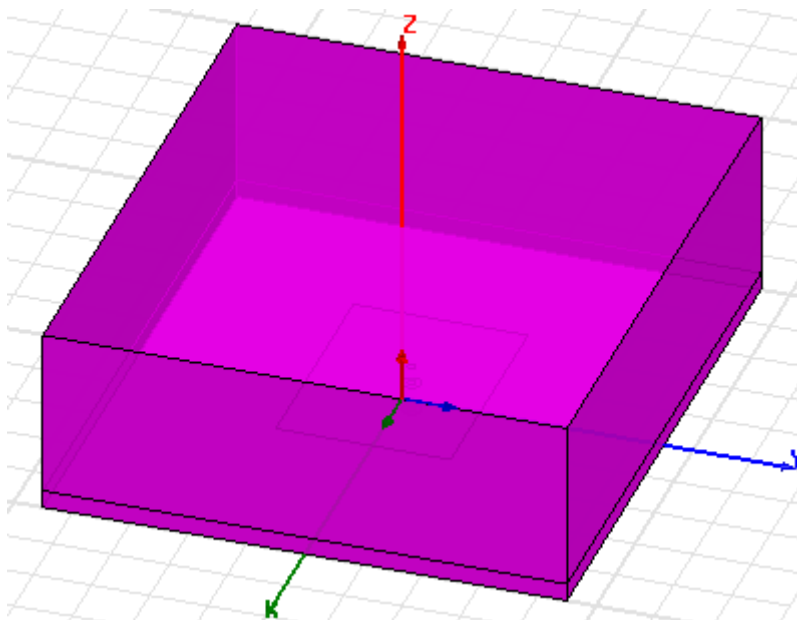


Figure 32. Air Body drawn

Assign Radiation Boundary

In this section you will assign the radiation boundary on all the faces of the air body except the bottom face.

- 1 Hit F to enter Face Selection mode and click **Edit>Select By Name**

The **Select Face** dialog box appears.

- 2 Click **Air**, hold down the Ctrl key and then select all the Face IDs except the one for the bottom face i.e z=0.

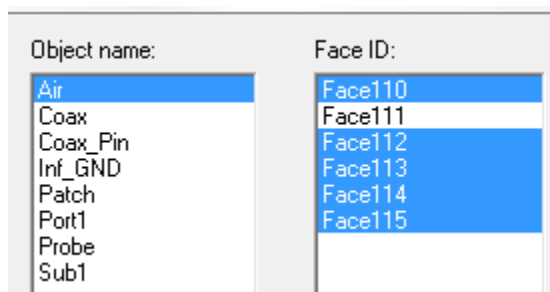


Figure 33. Select Face dialog box

- 3 Click **HFSS>Boundaries>Assign>Radiation**
The Radiation Boundary dialog box appears.
- 4 Edit the fields as shown in Figure 34 and click **OK**.

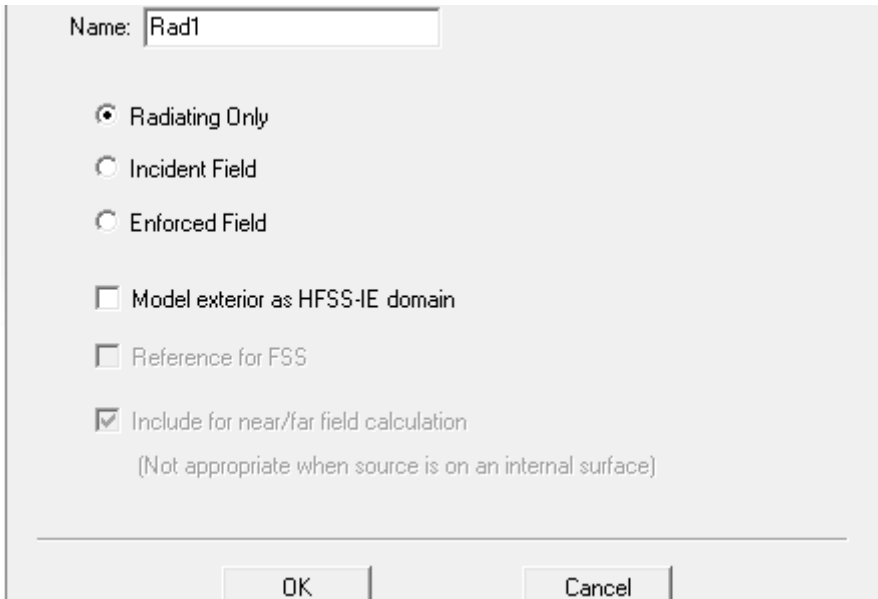


Figure 34. Radiation Boundary dialog box

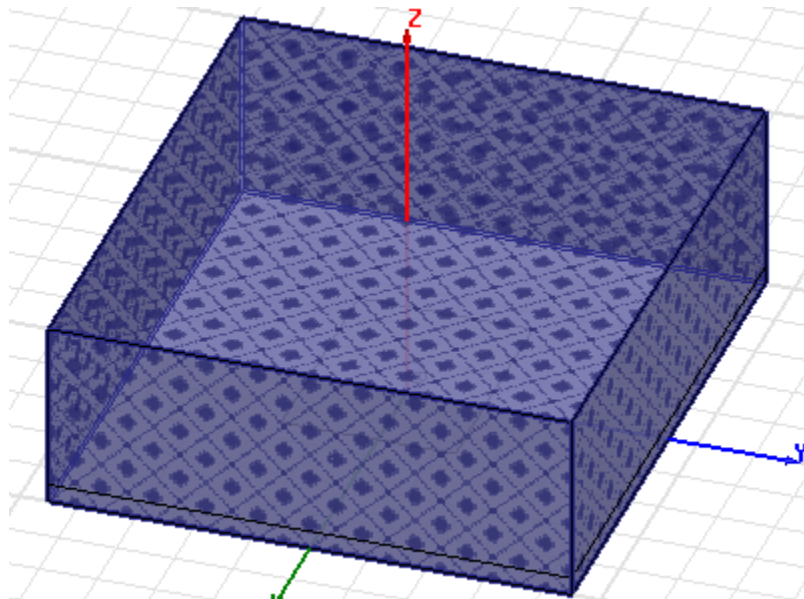


Figure 35. Radiation Boundary applied

3-18 Set Up The Project

Create Radiation Setup

- 1 Click HFSS>Radiation>Insert Far Field Setup>InfiniteSphere

The Far Field Radiation Sphere Setup dialog box appears.

- 2 Edit the fields as shown in Figure 36 and click OK.
An option ff_2d appears under Radiation in the Project Manager window.

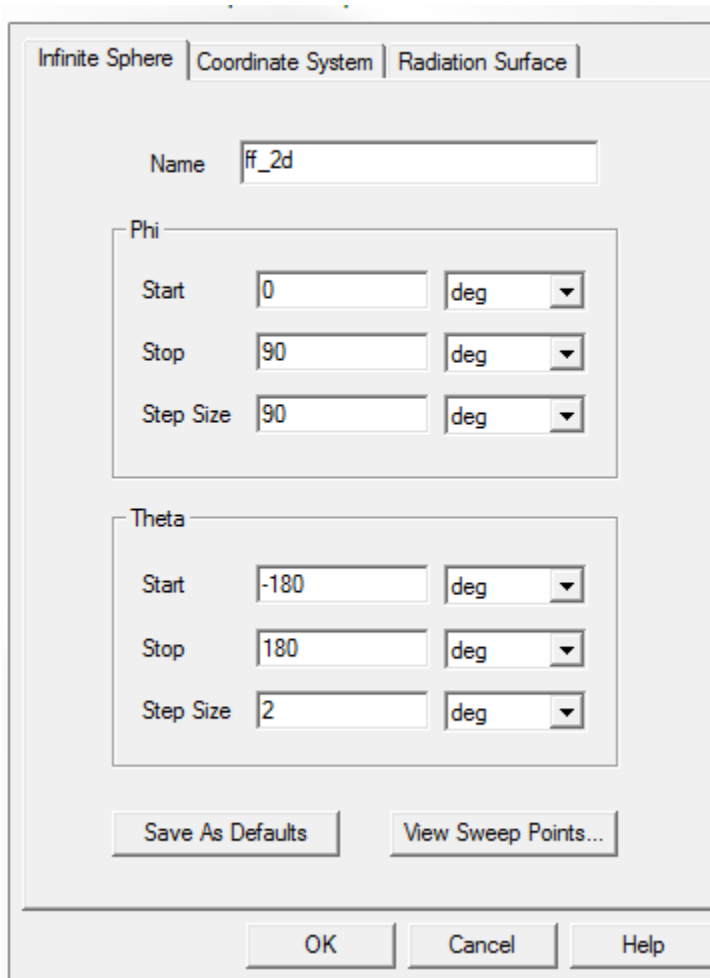


Figure 36. Far Field Radiation Sphere Setup dialog box

4

Analyze the Model

This chapter contains the following topics:

- ✓ Add Solution Setup
- ✓ Add Frequency Sweep
- ✓ Model Validation
- ✓ Analyze All
- ✓ Review Solution Data
- ✓ Review the Profile Panel
- ✓ Review the Convergence Panel
- ✓ Review the Matrix Data Panel
- ✓ Review the Mesh Statistics Panel
- ✓ Create Reports
- ✓ Create Terminal Solution Data Report
- ✓ Create Far Field Overlays

Add Solution Setup

- 1 Right click **Analysis** in the Project Manager window and select **Add Solution Setup** from the shortcut menu.
The **Solution Setup** dialog box appears.
- 2 On the **General** tab edit the fields as shown in Figure 1 and click **OK**.

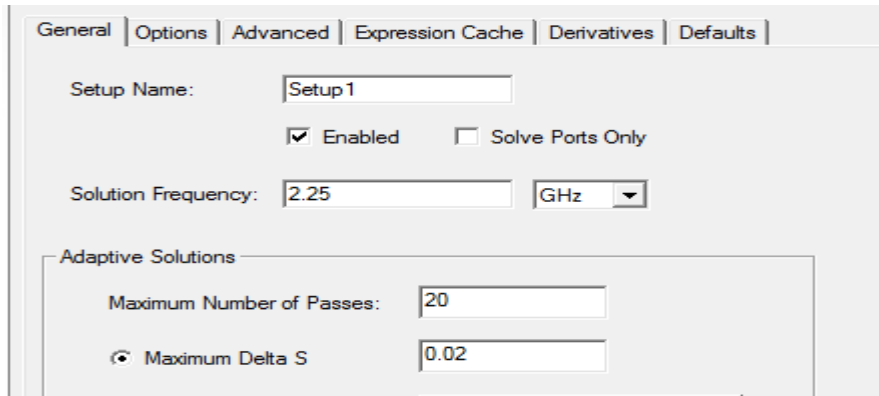


Figure 1. Add Solution Setup dialog box

Note This dialog box defines how HFSS will automatically generate an accurate mesh and the stopping criteria for the mesh adaption process.

Add Frequency Sweep

- 1 Right click **Setup1** under **Analysis** in the **Project Manager** window and select **Add Frequency Sweep**.
The **Edit Frequency Sweep** dialog box appears.
- 2 Edit the fields as shown in Figure 2 and click **OK**.
- 3 Save the project.

4-2 Analyze the Model

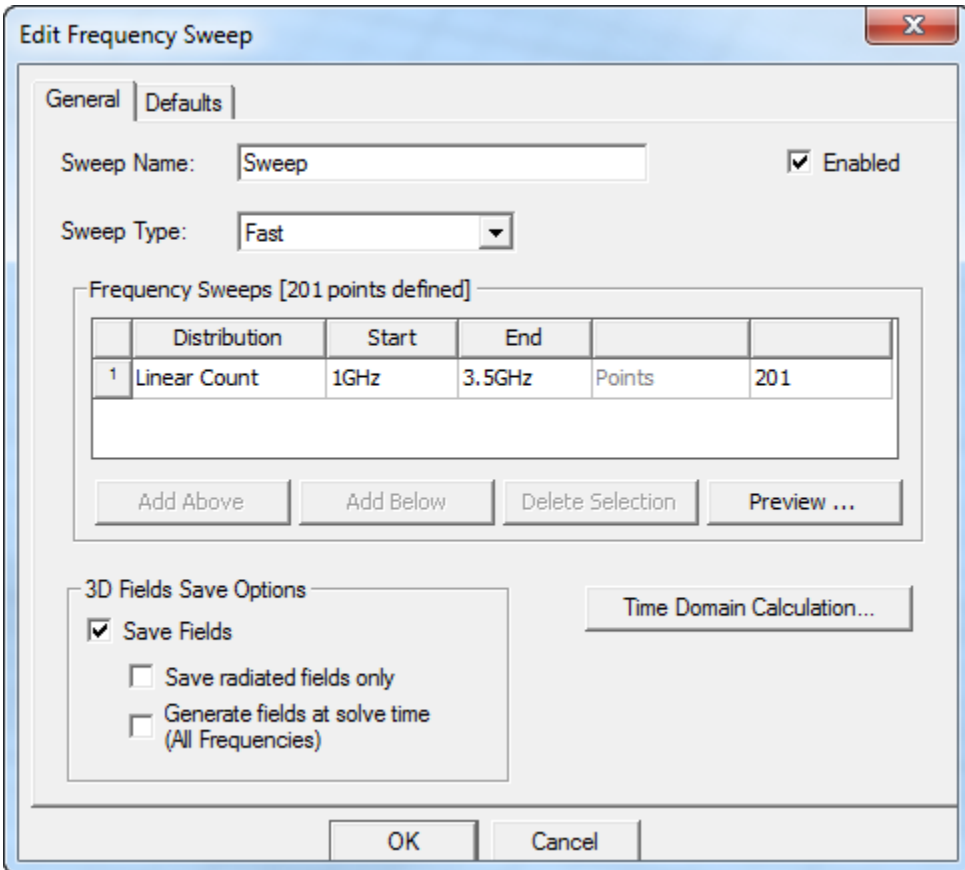


Figure 2. Edit Frequency Sweep dialog box

Analyze the Model

You are all set to validate the project.

- 1 Click HFSS>Validation Check.

The Validation Check dialog box appears.

- 2 Click Close.

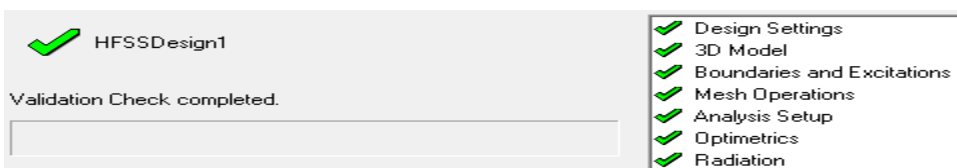


Figure 3. Validation Check dialog box

You are all set to analyze the project i.e. run the simulation.

- 1 Right click **Analysis** from the Project Manager window and select **Analyze All** from the shortcut menu.

Note Normal completion of simulation is notified in the Message Manager window.

Review Solution Data

You can view the solution data to better understand the results and the simulation.

- 1 Right click **Results** from the Project Manager window and select **Solution Data** from the shortcut menu.

The **Solution Data** dialog box appears.

- 2 You can select the various options **Profile**, **Convergence**, **Matrix Data**, **Mesh Statistics** to view the results displayed on the corresponding panels.

Review the Profile Panel

- 1 On the **Solutions** dialog box click **Profile**.

The Profile window lets you view a synopsis of the simulation results ranging from mesh creation and refinement to information about the different adaptive passes, the matrix assembly and solve along with extraction of electromagnetic field and SYZ parameter data. The more highly refined the mesh, i.e. higher the number of tetrahedra, more accurate is HFSS' solution of the design generating optimum results. However, there is a trade-off in the number of tetrahedra used and the computational resources required. Higher the number of tetrahedra the more accurate the solutions. Keep in mind that increased accuracy requires more computational resources and more time.

4-4 Analyze the Model

Mesh Refinement				Lambda Based
Mesh (lambda based)	00:00:00	00:00:00	31.1 M	1493 tetrahedra
Mesh Refinement				Port Adapt
Simulation Setup	00:00:00	00:00:00	25.2 M	Disk = 0 KBytes
Port Adaptation	00:00:00	00:00:00	35.4 M	Disk = 5 KBytes, 1134 tetrahedra
Mesh (port based)	00:00:00	00:00:00	31.1 M	1586 tetrahedra
Adaptive Pass 1				Frequency: 2.25 GHz
Simulation Setup	00:00:00	00:00:00	25.3 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	33.3 M	Disk = 36 KBytes, 1227 tetrahedra , p1: 104 triangl
Solver MCS1	00:00:00	00:00:00	41.1 M	Disk = 0 KBytes, matrix size 6801 , matrix bandwidt
Field Recovery	00:00:00	00:00:00	41.1 M	Disk = 490 KBytes, 1 excitations
Adaptive Pass 2				Frequency: 2.25 GHz
Mesh (volume, adapti...	00:00:00	00:00:00	31.4 M	1933 tetrahedra
Simulation Setup	00:00:00	00:00:00	26 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	34.7 M	Disk = 0 KBytes, 1572 tetrahedra , p1: 104 triangle
Solver MCS1	00:00:00	00:00:00	50 M	Disk = 0 KBytes, matrix size 8863 , matrix bandwidt
Field Recovery	00:00:00	00:00:00	50 M	Disk = 187 KBytes, 1 excitations
Adaptive Pass 3				Frequency: 2.25 GHz
Mesh (volume, adapti...	00:00:00	00:00:00	31.9 M	2409 tetrahedra
Simulation Setup	00:00:00	00:00:00	26.4 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	37.2 M	Disk = 0 KBytes, 2039 tetrahedra , p1: 104 triangle
Solver MCS1	00:00:00	00:00:00	60.8 M	Disk = 0 KBytes, matrix size 11713 , matrix bandwid
Field Recovery	00:00:00	00:00:00	60.8 M	Disk = 237 KBytes, 1 excitations
				Adaptive Passes converged

Figure 4. Profile

Review the Convergence Panel

The Convergence panel lets you view the plot of **Max Mag Delta S** versus **Pass Number**.

- 1 On the **Solutions Data** dialog box click **Convergence**.
- 2 Set the fields as shown in Figure 5 .

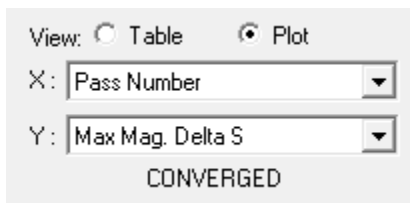


Figure 5. X, Y axes settings

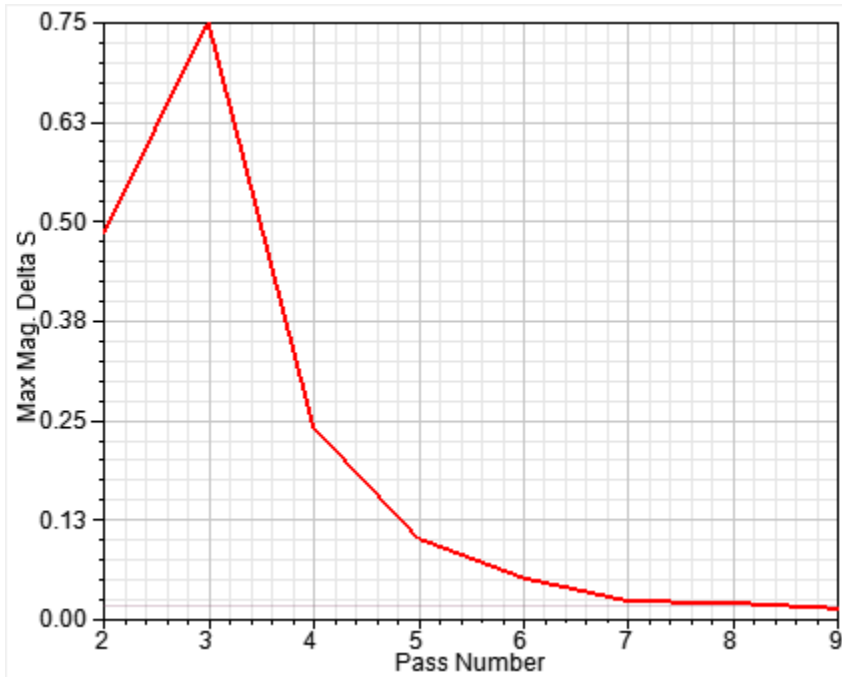


Figure 6. Convergence occurs in the 9th pass

3 Click **Table** to view the results listed in a table.

Pass Number	Total Tetrahedra	Max Mag. Delta S
1	1586	N/A
2	1959	0.48503
3	2441	0.74945
4	3068	0.24298
5	3884	0.10294
6	4943	0.053602
7	6323	0.024739
8	8115	0.02249
9	9462	0.015536

Figure 7. Convergence table

4-6 Analyze the Model

Review the Matrix Data Panel

To view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution, click the **Matrix Data** tab.

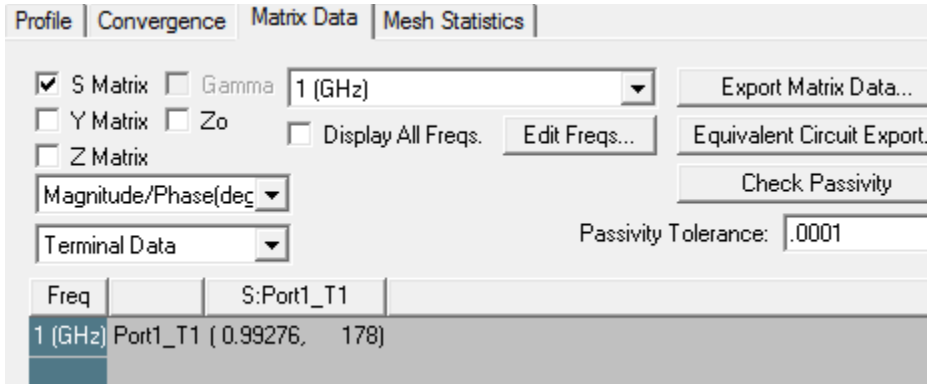


Figure 8. Matrix Data panel

- 1 On the **Solution Data** dialog box, click **Matrix Data**.

Note To view a real-time update of the Matrix Data, set the Simulation to Setup1, Last Adaptive.

Review the Mesh Statistics Panel

This panel shows statistics of the mesh, more specifically, it gives break-ups of the tets used to solve the different components of the model and their size and data.

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	M
Air	3604	0.0955082	3.91333	0.988566	6.59465e-00...	2.84
Coax	511	0.0389137	0.312081	0.154142	2.75814e-00...	0.00
Coax_pin	238	0.0535757	0.312081	0.135343	2.03037e-00...	0.00
probe	126	0.0823585	0.231698	0.140641	2.43681e-00...	0.00
Sub1	4983	0.0773074	2.91018	0.500523	8.17262e-00...	0.22

Figure 9. Mesh Statistics panel

Note After you have viewed all of the results in the different panels on the Solution Data dialog box, click **Close** and proceed to the next section.

Create Reports

This section deals with report creation. HFSS offers the report dialog box where you can enter the desired settings and generate reports.

Create Terminal Solution Data Report

- 1 Right click Results and select **Create Terminal Solution Data Report>Rectangular Plot**.

The **Report** dialog box window appears.

- 2 Ensure that the fields are as shown in Figure 45.

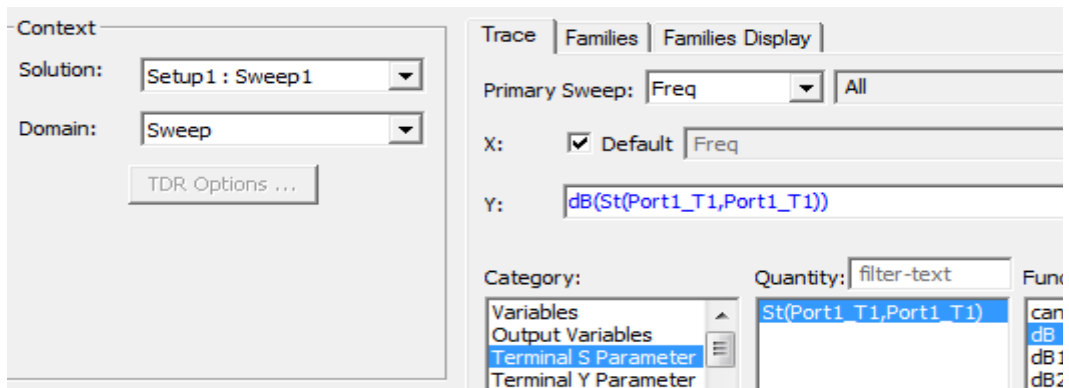


Figure 10. Report dialog box

- 3 Click **New Report**. and then, **Close**.

4-8 Analyze the Model

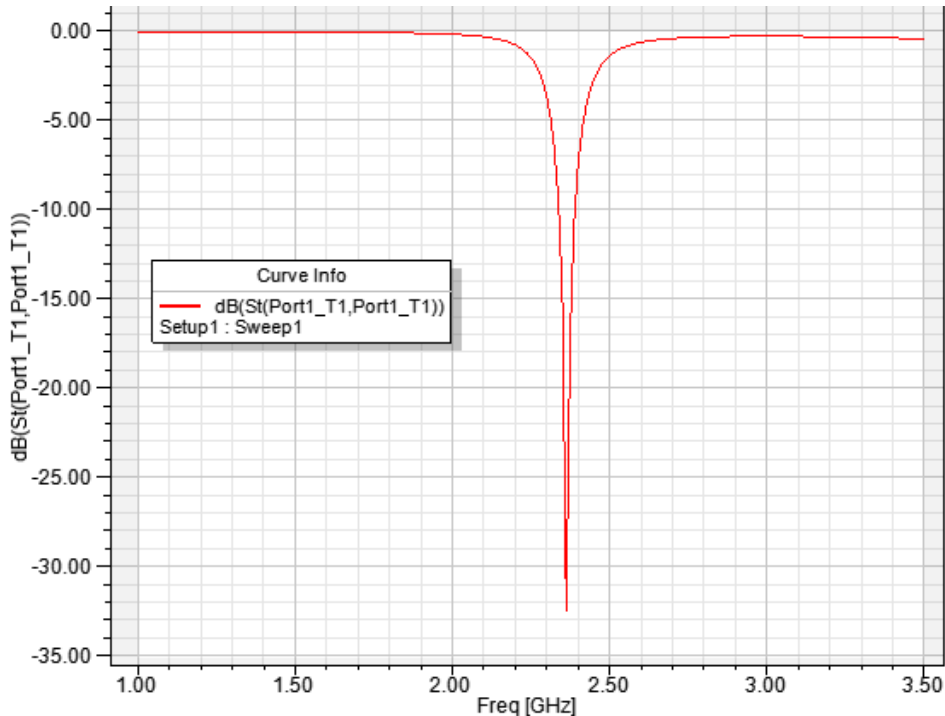


Figure 11. Select the trace on the plot.

- 4 Right click to display the shortcut menu and select-**Marker>Add Minimum**

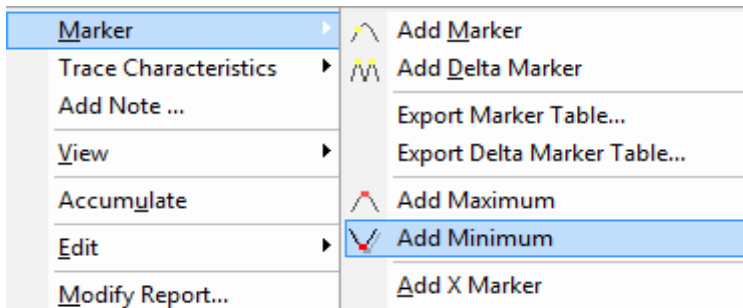


Figure 12. Short-cut menu

A minimum marker (m1) appears on the trace, and a marker table lists the x and y coordinate values.

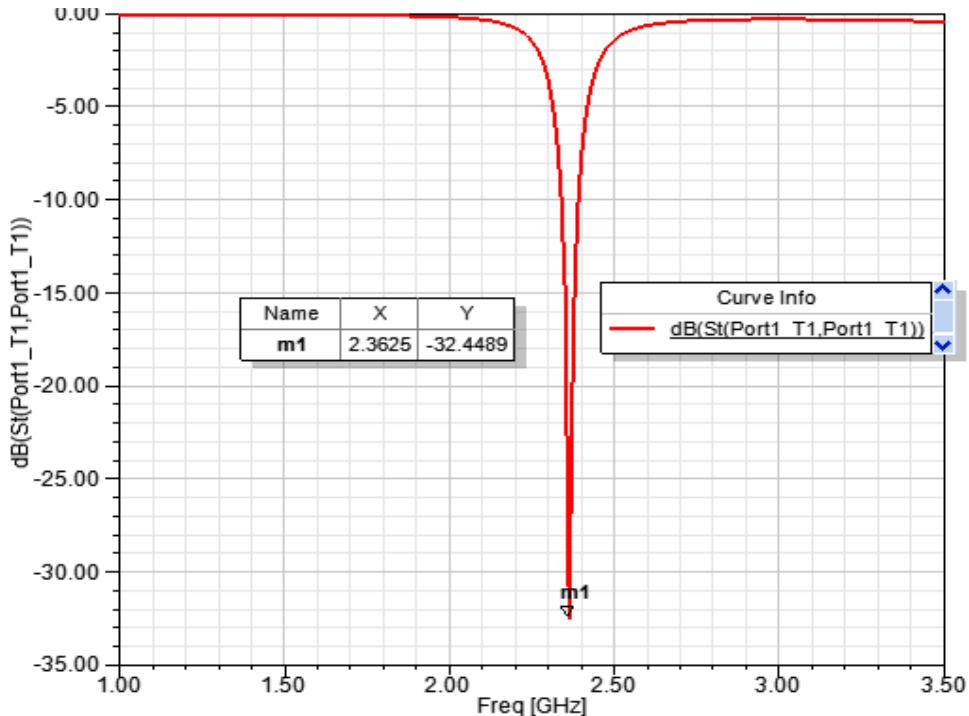


Figure 13. S parameter versus Frequency

Far Field Overlays

We will now plot far-field quantities such as antenna gain.

Create Far Field Overlay

To create a 2D polar far field plot :

- 1 Right click **Results** on the **Project Manager** window and select **Create Far Fields Report>Radiation Pattern**.

The **Reports** dialog box appears.

- 2 Edit the fields as shown in Figure 50:
- 3 Click **New Report** and then, click **Close**.

4-10 Analyze the Model

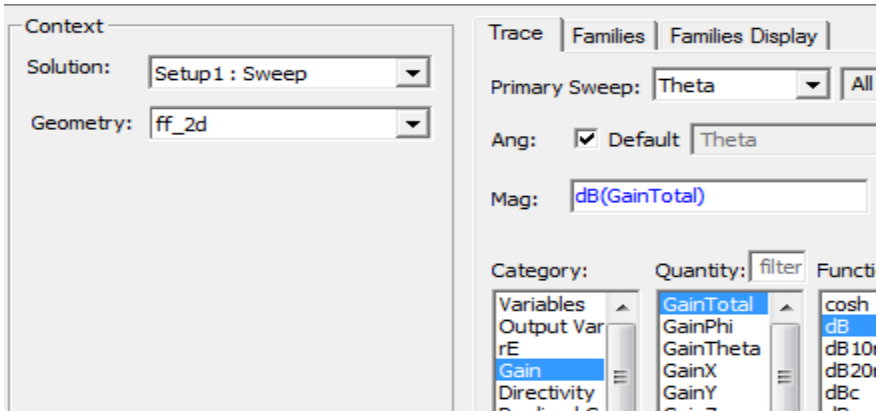


Figure 14. Report dialog box

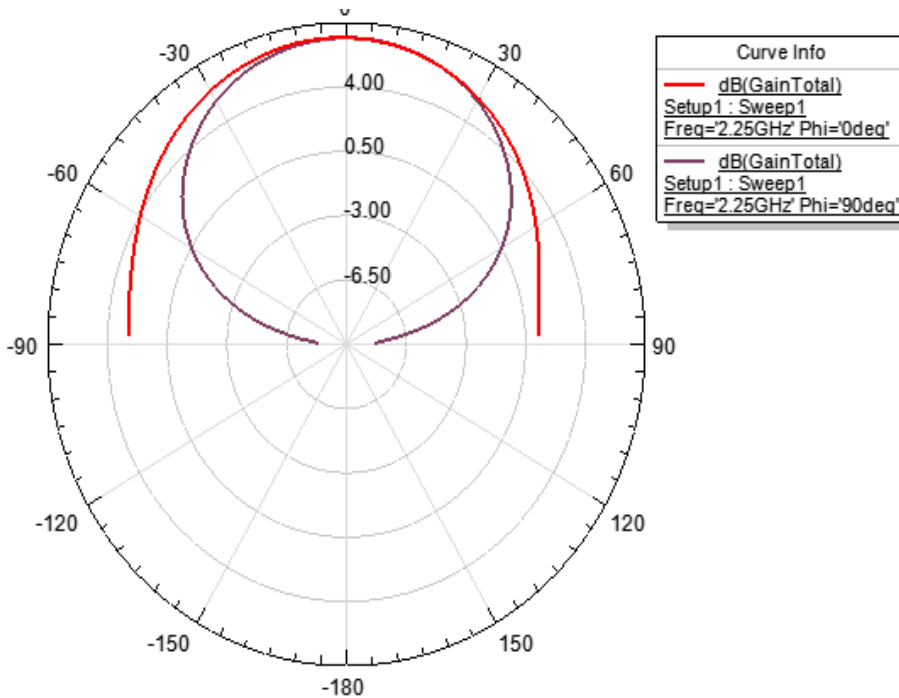


Figure 15. Radiation Pattern