Simulation Électromagnétique en 3D

Utilisation du logiciel HFSS/Ansys

Mohamed LATRACH

Ingénieur 3me Année (EOC) **2016-2017**

ESEO 10 Boulevard Jean Jeanneteau – CS 90717 - 49107 Angers Cedex 2 – Tél : 02 41 86 67 6

Simulation Électromagnétique en 3D
Utilisation du logiciel HFSS/Ansys

HFSS : High Frequency Structure Simulator (an. 1990)

Ansoft : Founded 1984

2008 : ANSYS acquires ANSOFT

HFSS: Finite Element Method (FEM) software

•Geometry can be arbitrary, 3-dimensional.

•Boundary Conditions within and on the problem space boundary can be varied to account for different characteristics, symmetry planes, etc.

•Size constraints are predominantly set by available memory and disk space for storage and solution of the problem matrix.

•Solution created is in the frequency domain, assuming steady-state harmonic behavior.

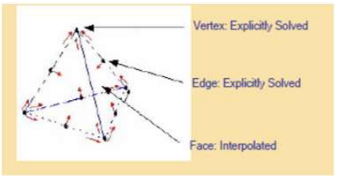
HFSS Applications

• **Antenna**
–Planar Antennas-Patches, Dipoles, Horns, Conformal Cell Phone Antennas, Spirals
–Waveguide–Circular/Square, Horns
–Wire–Dipole, Helix
–Arrays-Infinite Arrays, Frequency Selective Surfaces (FSS) & Photonic Band Gaps (PBG)
–Radar Cross Section (RCS)
• **Microwave**
–Filters–Cavity Filters, Microstrip, Dielectric, LTCC
–EMC/EMI –Shield Enclosures, Coupling, Near-or Far-Field Radiation
–Connectors–Coax, SFP/XFP, Backplane, Transitions
–Waveguide–Filters, Resonators, Transitions, Couplers
•**RFIC/MMIC**
–Silicon-On chip components, Spiral Inductors, Transformers, capacitorsetc.
–GaAs/InP–On chip components, vias, bondwires, modules etc
•**Signal Integrity/High-Speed Digital**
–Package Modeling–BGA, QFP, Flip-Chip
–PCB Board Modeling–Power/Ground planes, Mesh Grid Grounds, Backplanes
–Connectors–SFP/XFP, VHDM, GBX, NexLev, Coax, USB
–Transitions–Differential/Single-ended Vias

What is the Technology Behind the HFSS Field Solver?

–Volumetric Field Solver
•Type: **Full-Wave**
•Solution Method: **3D Finite Element Method (FEM)**

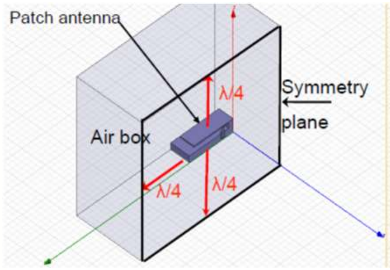
–**Accuracy**: *If there were no limits on the size of the matrix and on the number of digits for computation, there would be no limit to the accuracy of the Finite Element Method!*
•Mesh Type: **Conformal**
•Mesh Element: **Tetrahedron**
•Mesh Process: **Adaptive**
–Convergence: **Complex Magnitude Change in S-Parameters (Delta S)**

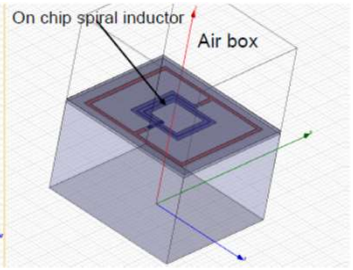


–Excitations -Port Solver
•Solution Method: **2D Finite Element Method**
•Mesh Process:**Adaptive**

–Frequency Sweeps
•**Discrete Sweep**
–Generates field solutions at specific frequency points in a frequency range.
•**Interpolating Sweep**
–Adaptive Discrete Sweep with curve fitting
»Up to 10000 data points, number of discrete solution points varies with response.
•**Fast Frequency Sweep**: **ALPS (Adaptive Lanczos-PadéSweep**)
–Matrix Data and Fields at every frequency in sweep, up to 10000 data points.

Pre-Processing: Geometry definition





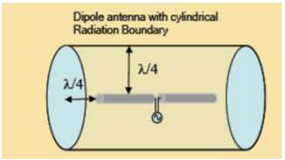
•Symmetry should be used to reduce solve time and improve accuracy.

• For radiating structure: antenna…airbox should be drawn (lambda/4 to lambda/2 in all directions) and radiation boundary or PML used.

• Airbox needs to be drawn far enough here to remove the outer boundary influence.

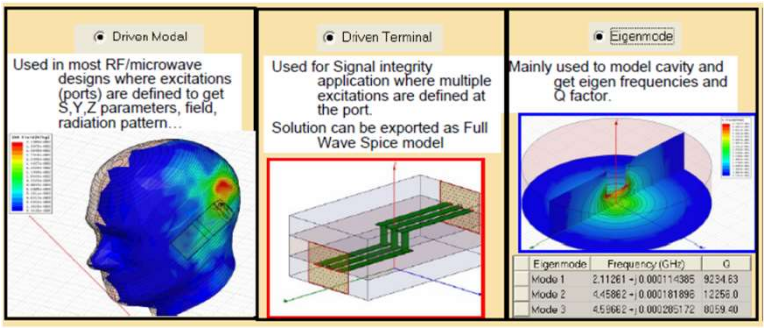
•For non radiating structure (several times smaller than wavelength…a smaller box -approx lambda/10 -is more efficient.

•Accuracy depends on :The distance between the boundary and the radiating object.



The radiation boundary should be located *at least* one-quarter of a wavelength (at lowest frequency of interest) from a radiating structure. If you are simulating a structure that does not radiate, the boundary can be located less then one-quarter of a wave length (The validity of this assumption will require your engineering judgment).

HFSS solution type



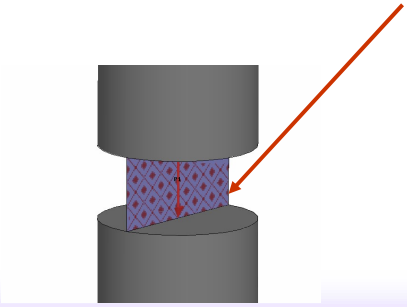
Note: in project where one single excitation mode is used at the port, both modal and terminal based model should provide similar answer

Excitations

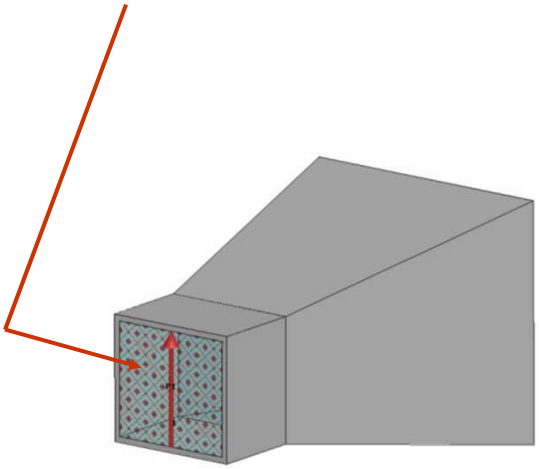
Five different types of excitations: Wave and Lumped Ports, Incident Wave, Voltage and Current Sources. All Excitations allow calculation of fields but only Wave and Lumped Ports yield S parameters.

➥Antennes: Lumped ports ou Wave ports

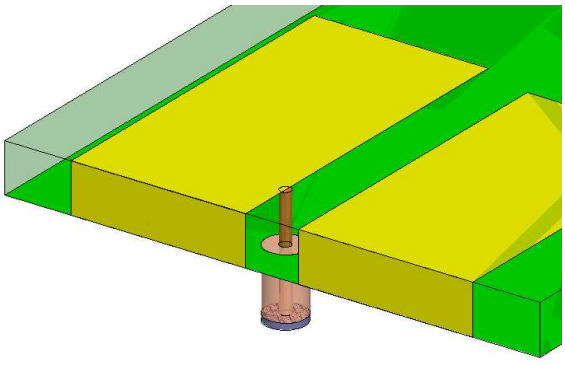
1) Wire antenna such as Dipoles, Monopoles should use Lumped ports.

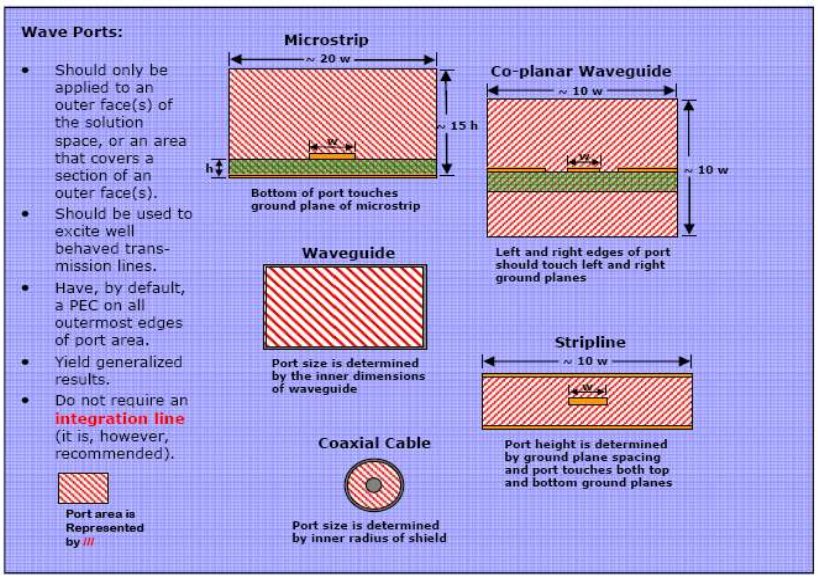


2) Horn antennas should use wave ports



3) Planar antennas fed with coax feeds, microstrip, stripline or other transmission line feeds can use either Wave or Lumped ports





– E-Field can be forced to align with Integration line by selecting Polarize E Field

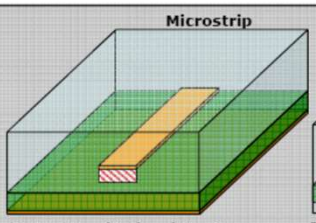
–Appropriate for structures with degenerate modes e.g. square or circular WG.

Examples:

Lumped Ports:

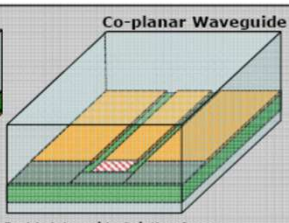
- Should only be applied to 2D sheet objects internal to the solution space, where opposite ends of the 2D object contact conducting objects.
- Should be used to mimic probe locations, test points, component feed points, or active device outputs.
- Yield results that are normalized to the port Source Impedance specified during setup.

Microstrip



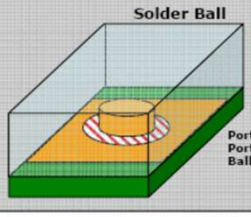
Port is internal to the solution Space. The 2D port rectangle touches the signal trace with one edge and the opposite edge touches the ground plane.

Co-planar Waveguide



Port is internal to Solution Space. The 2D port rectangle touches the signal trace with one edge and the opposite edge touches user drawn PEC objects.

Solder Ball



Port is internal to Solution Space. Port is an annular ring around Ball grid pad.

Summary

Port	applied	Gamma	Yields S,Y,Z	Renormalize	De-embed
Wave	externally	yes	yes	possible	possible
Lumped	internally	no	yes	possible	not possible

Analysis

➢ To ensure accurate HFSS results, several items need to be specified in the analysis setup. They are:

- A **Solution Frequency:** It should be set to the frequency of interest, or a frequency point in the upper quartile of the frequency band of interest.
- A **Maximum number of passes**: Should be at least 3.
- A **Delta S convergence:** Should be greater than 0.005 and less than 0.02.
- A **refinement percentage:** Should be between 15 and 30 percent.

➢ To generate frequency sweep results, HFSS has three types of sweep solutions available:

❖ **Discrete Sweep**: performs an exact solution for every frequency point. Can be time intensive. Generates field solutions at specific frequency points in a frequency range

❖ **Fast Sweep**: Results are obtained quickly but are accurate over a finite (order of magnitude) frequency range only. Recommended for simulations with sharp resonances.

❖ **Interpolating Sweep**: Interpolates response at various frequency points within a frequency range to obtain full sweep results. Accurate for simulations from DC to very high GHz. Recommended for problems that will eventually be used in time domain simulations.

Accuracy

–Accuracy of the model

•All the relevant details needs to be defined.
•Proper material properties.
•Proper port definition and model size.

–Accuracy of the mesh

•The EM field value can strongly vary in some part of the model and less in other: HFSS adaptive process automatically refines the critical area so that a denser mesh will better represent the field.

Exemple d'étude

Antenne microbande à 2,4 GHz

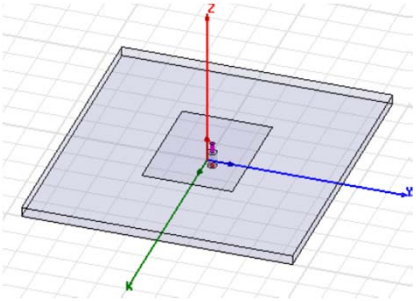


Table of Contents

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.

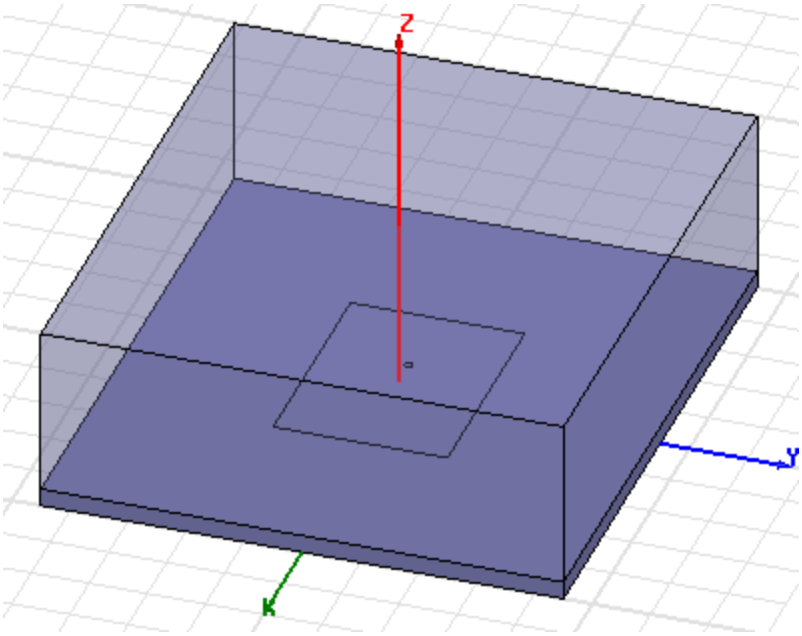


Figure 1-1 Probe Feed Patch Antenna

2 - Set Up the Initial 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.

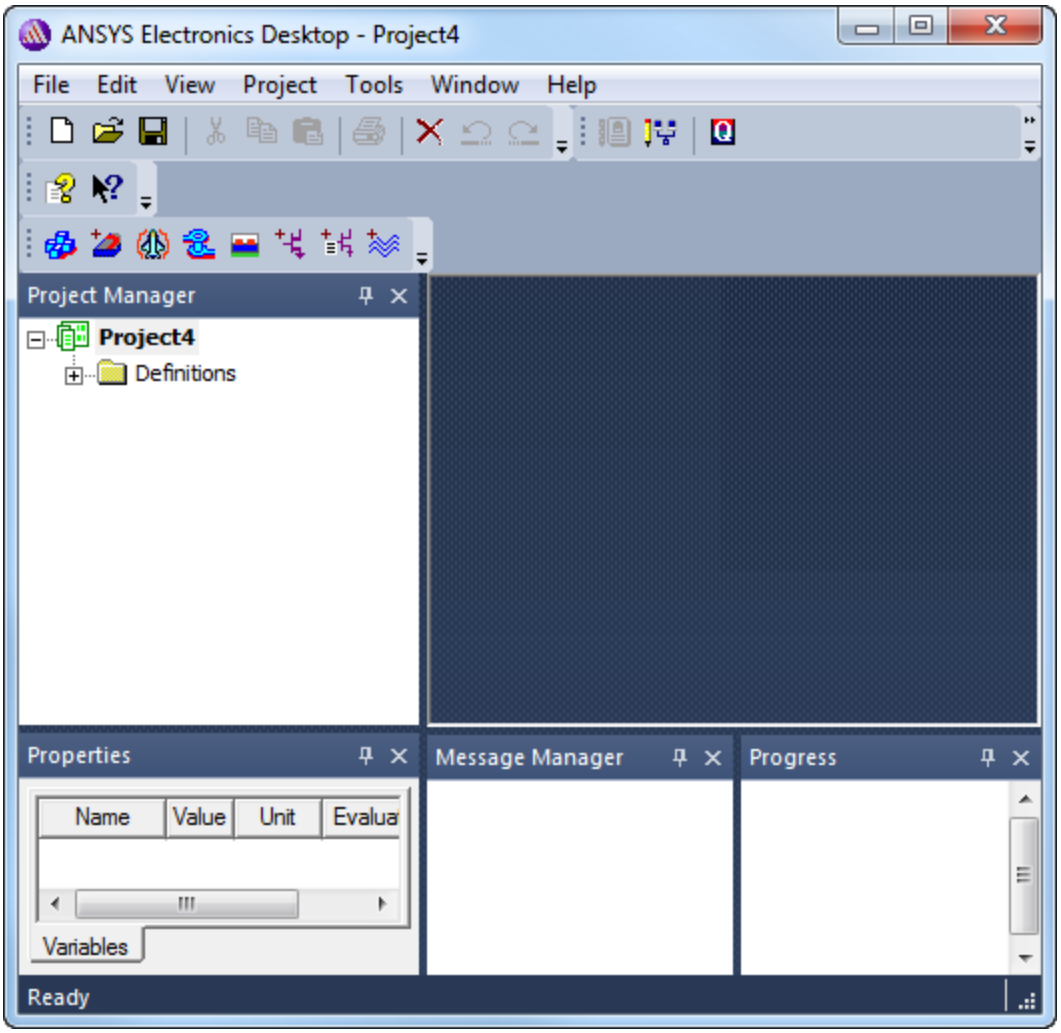


Figure 2-1 ANSYS Electronics Desktop appears

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.

Set Tool Options

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

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

   The **HFSS Options** dialog box appears.

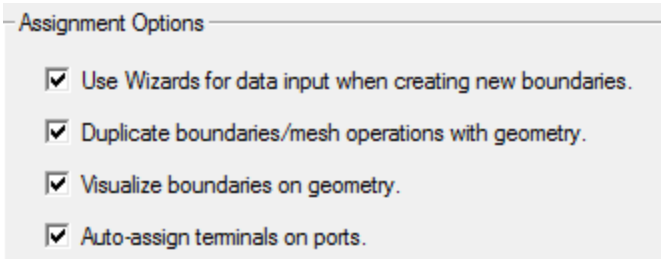


Figure 2-2 Assignment Options

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 represents the **Insert HFSS design** (**IHd**) option.

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

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

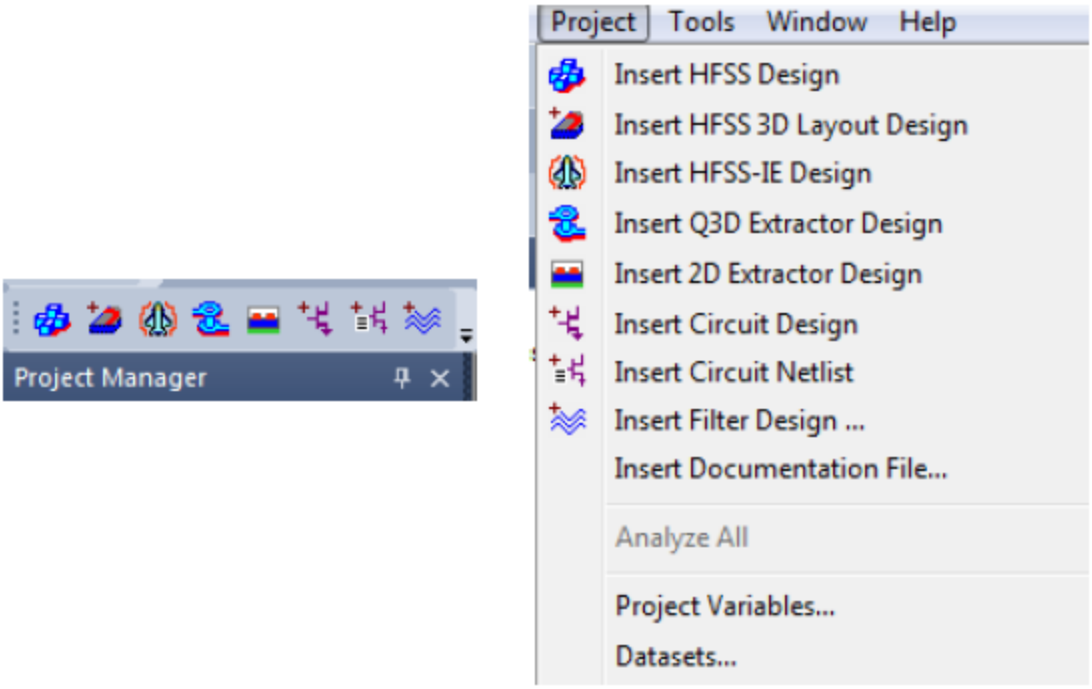


Figure 2-3 Include HFSS Design type

3. Click **Project*N****, 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 2-4 Set Model Units dialog box

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.

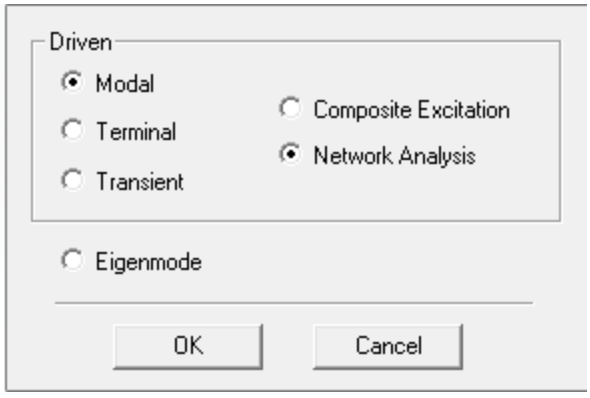


Figure 2-5 Solution Type dialog box

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 axeand 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 3-1 Command dialog box

4. On the **Command** tab edit the fields as shown in "Command dialog box" above. 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 "Select Definition window" below. .
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 Materia**l 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**.

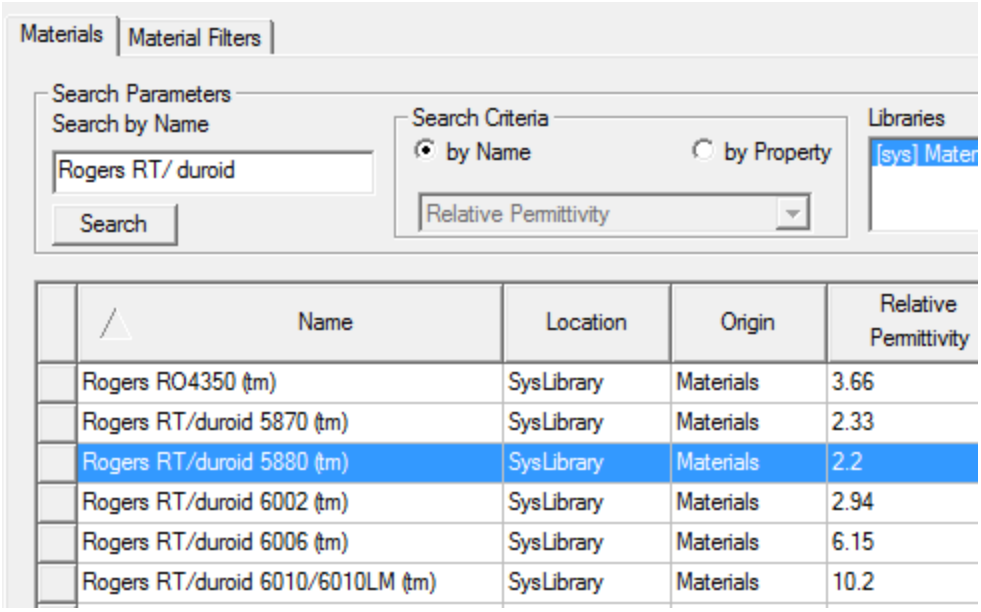


Figure 3-2 Select Definition window

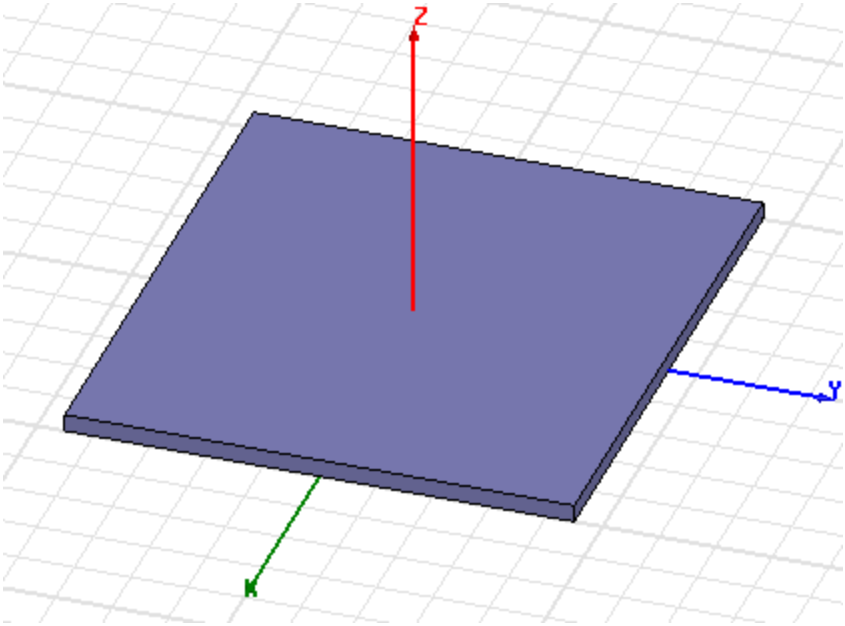


Figure 3-3 Substrate

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 "Infinite Ground Command dialog" below. .
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 3-4 Infinite Ground Command dialog

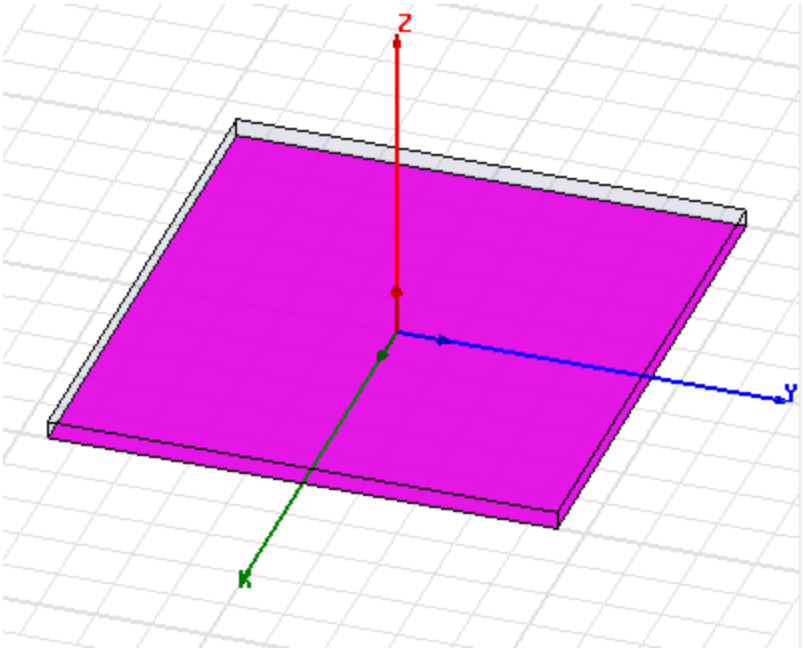


Figure 3-5 Infinite Ground plane

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.**

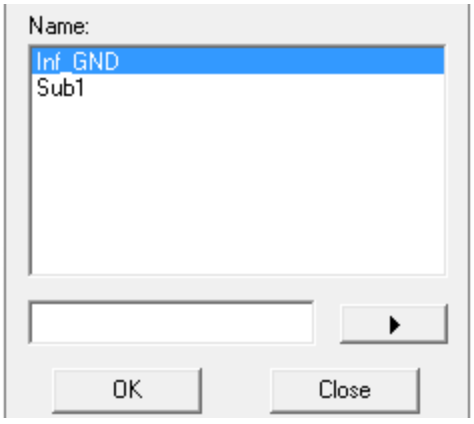


Figure 3-6 Select Object dialog box

3. Right click **Assign Boundary>Perfect E**
   The **Perfect E Boundary** dialog box appears.
4. Edit the fields as shown in "Perfect E Boundary dialog box" below. and click **OK**.

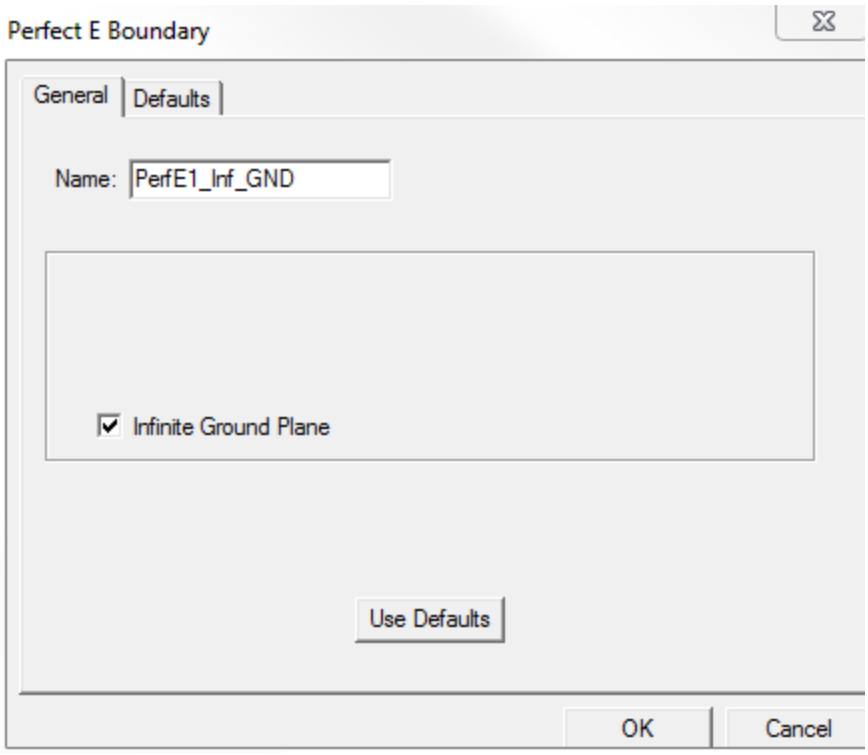


Figure 3-7 Perfect E Boundary dialog box

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 "Ground Cut Out Command dialog" on the next page. and click **OK**.
4. Double-click **Circle** from the history tree and enter *Cut_Out* in the **Name** field and click **OK**.

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 3-8 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.**

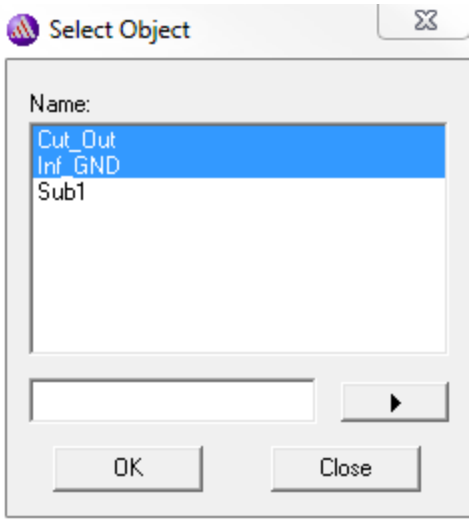


Figure 3-9 Select Object dialog box

Complete the Ring

1. Click **Modeler>Boolean>Subtract**
   The **Subtract** dialog box appears.
2. Set the fields as in "Subtract dialog box" on the facing page. and click **OK**.

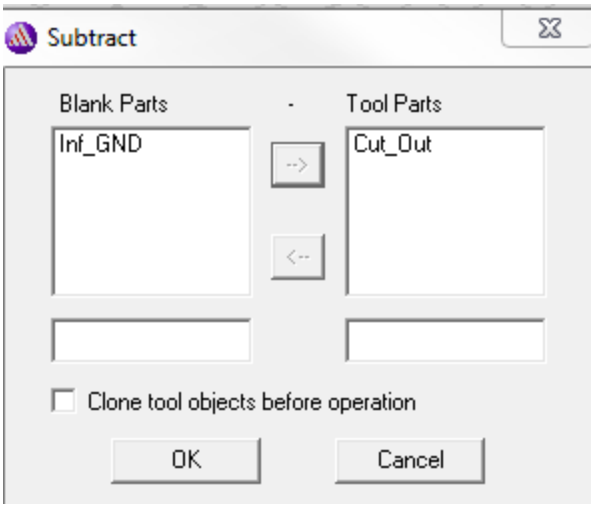


Figure 3-10 Subtract dialog box

Create the 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**.

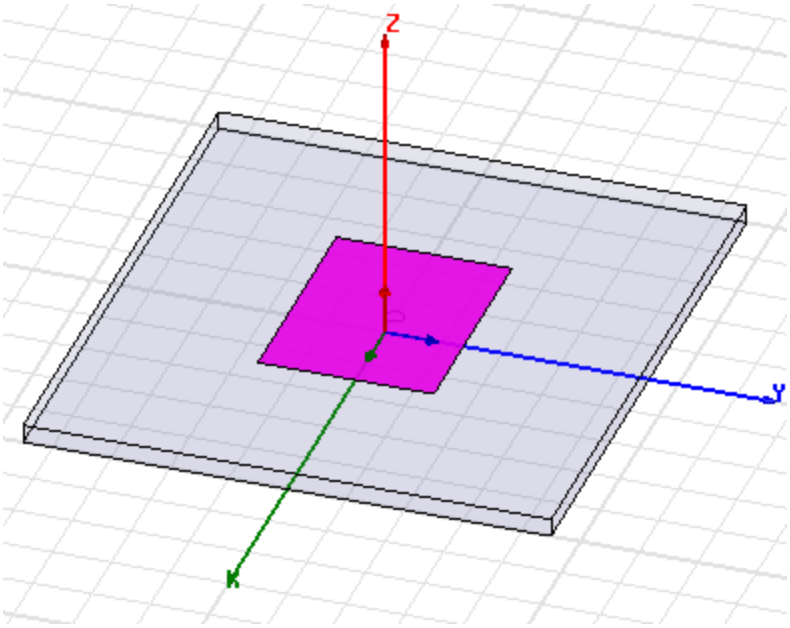


Figure 3-11 The Patch drawn

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**.

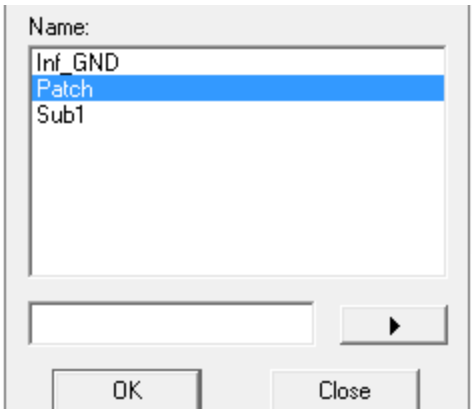


Figure 3-12 Select Objects dialog box

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**.

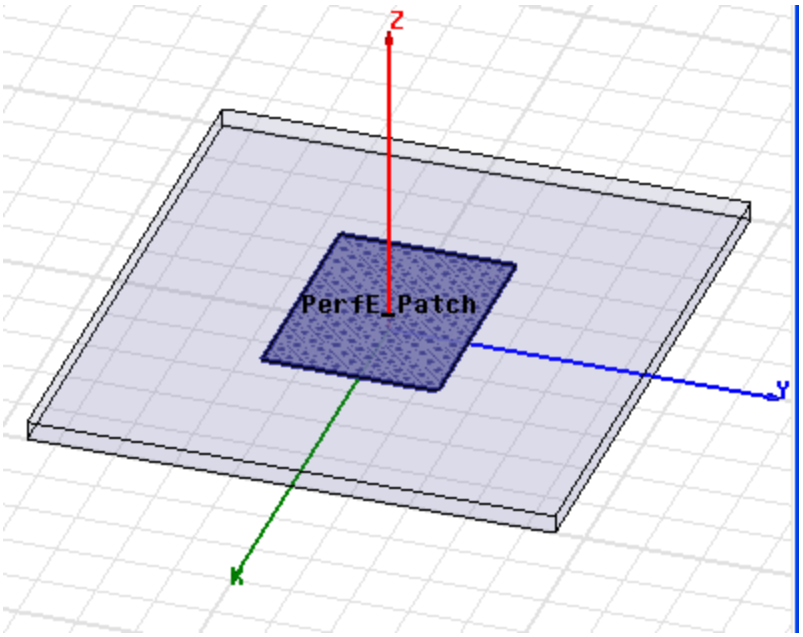


Figure 3-13 PerfE_Patch applied

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.

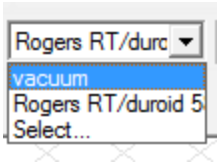


Figure 3-14 Material drop down

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

	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 3-15 Command dialog for Coax

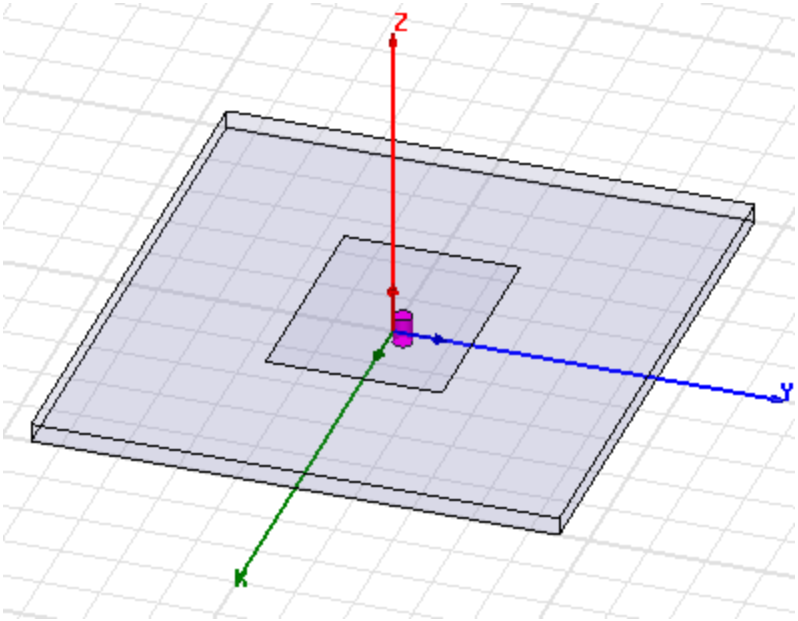


Figure 3-16 Coax drawn

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 "Command dialog for Coax Pin" below. .
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 3-17 Command dialog for Coax Pin

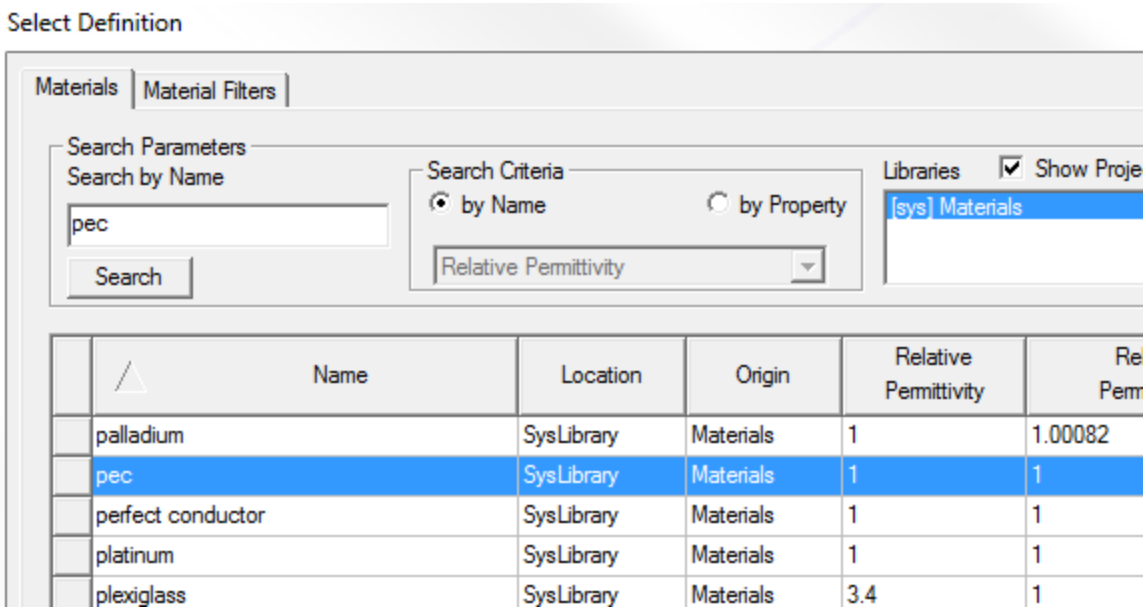


Figure 3-18 Select Definition window

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.

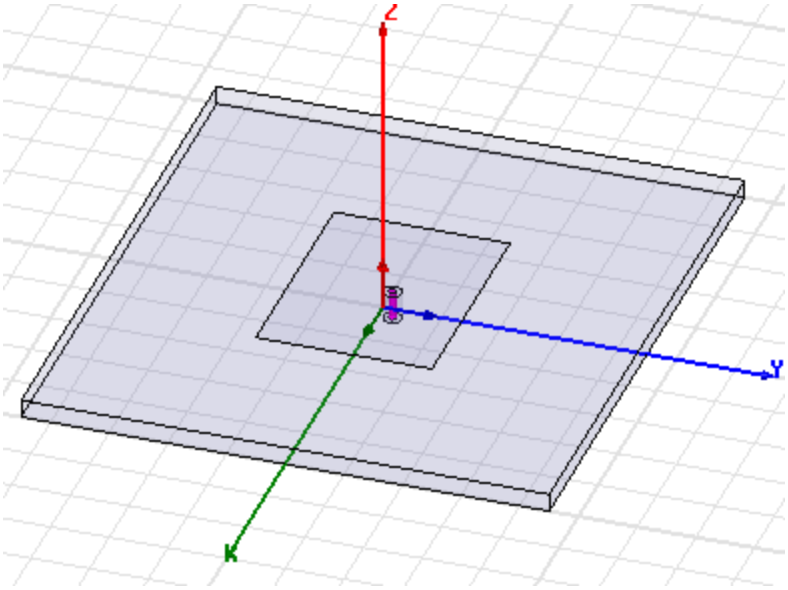


Figure 3-19 coax pin

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.

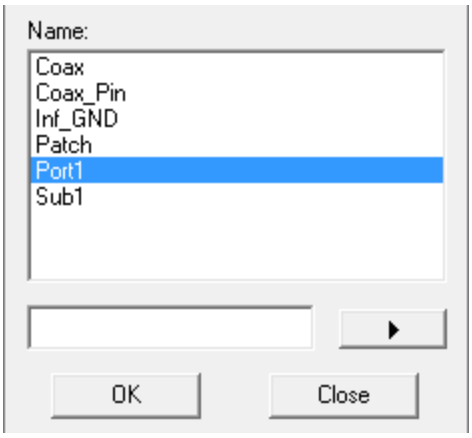


Figure 3-20 Select Object dialog box

8. Edit the fields as shown in "Reference Conductors for Terminals" on the facing page. and click **OK**.

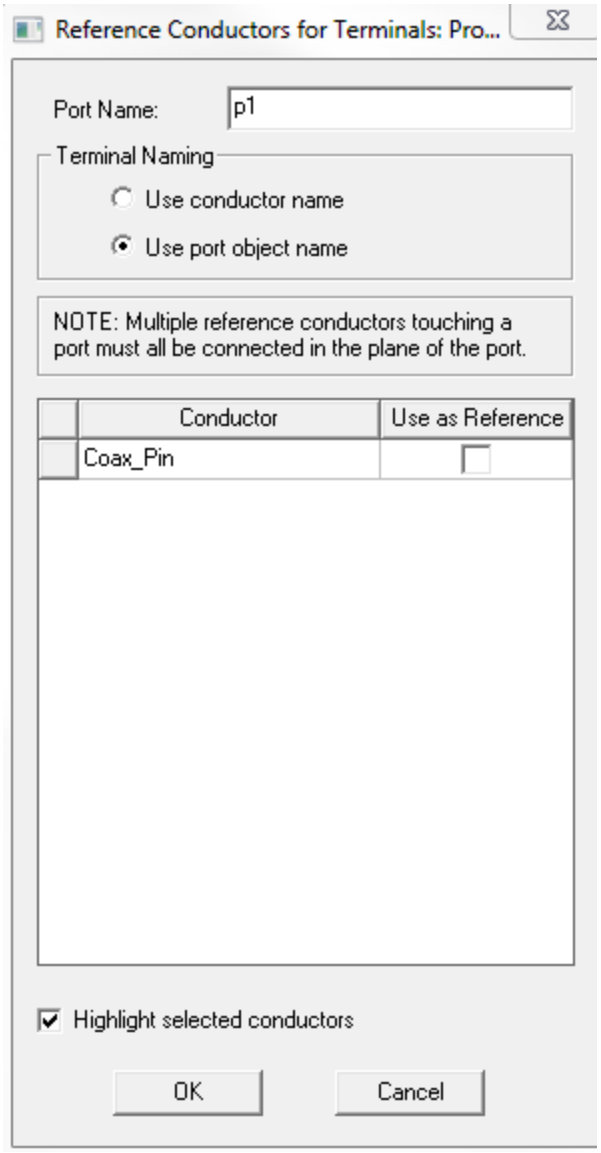


Figure 3-21 Reference Conductors for Terminals

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

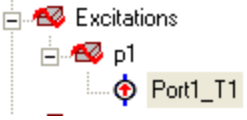


Figure 3-22 Project Tree

Create the Probe

To create the probe:

1. Draw a cylinder freehand and edit the fields on the Command tab as shown in "Command dialog box" below. .
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 3-23 Command dialog box

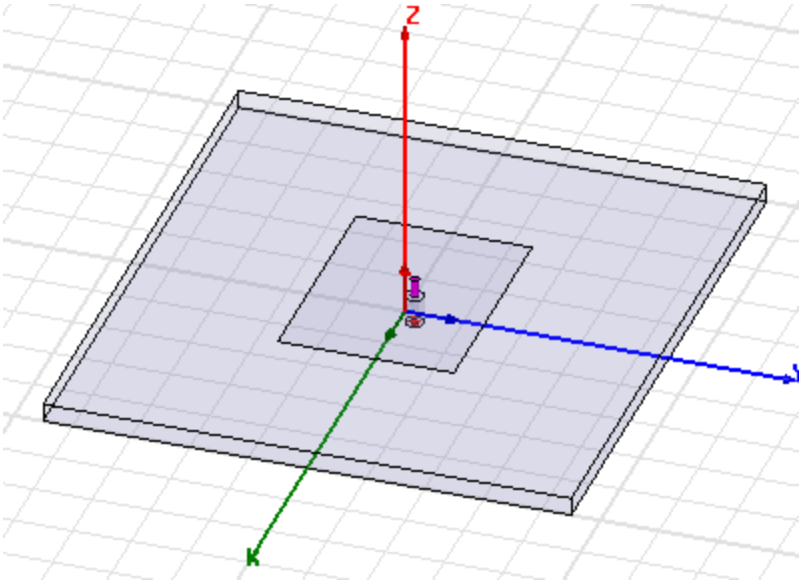


Figure 3-24 Probe drawn

Create Air

1. Draw a box freehand and edit the fields on the Command tab as shown in "Command dialog box" on the facing page. .

	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 3-25 Command dialog box

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

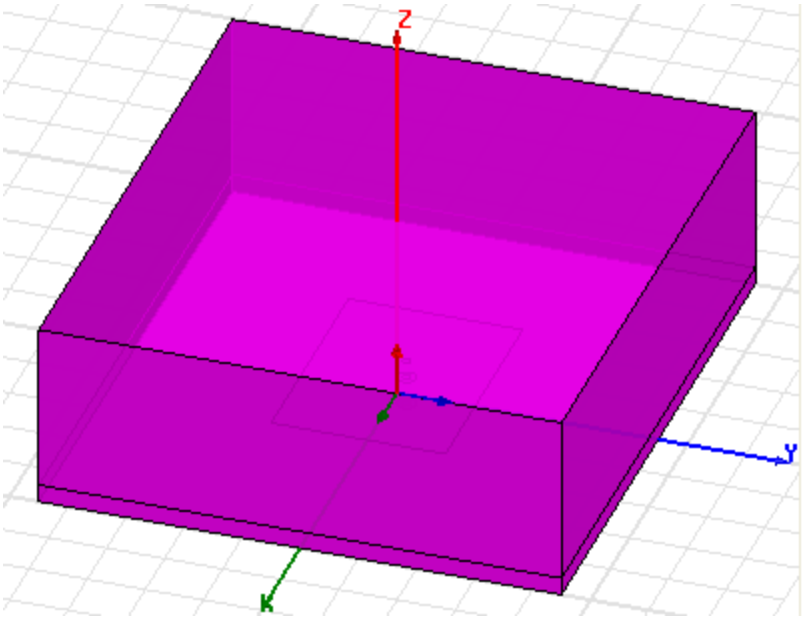


Figure 3-26 Air Body drawn

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.

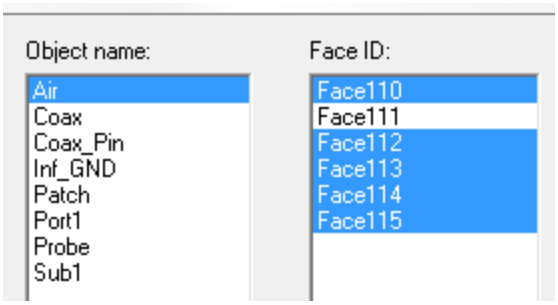


Figure 3-27 Select Face dialog box

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

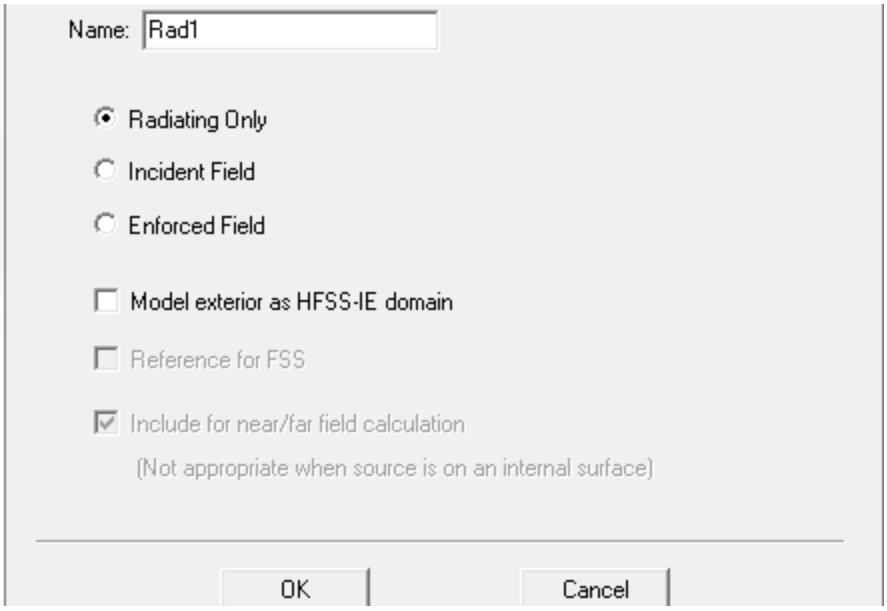


Figure 3-28 Radiation Boundary dialog box

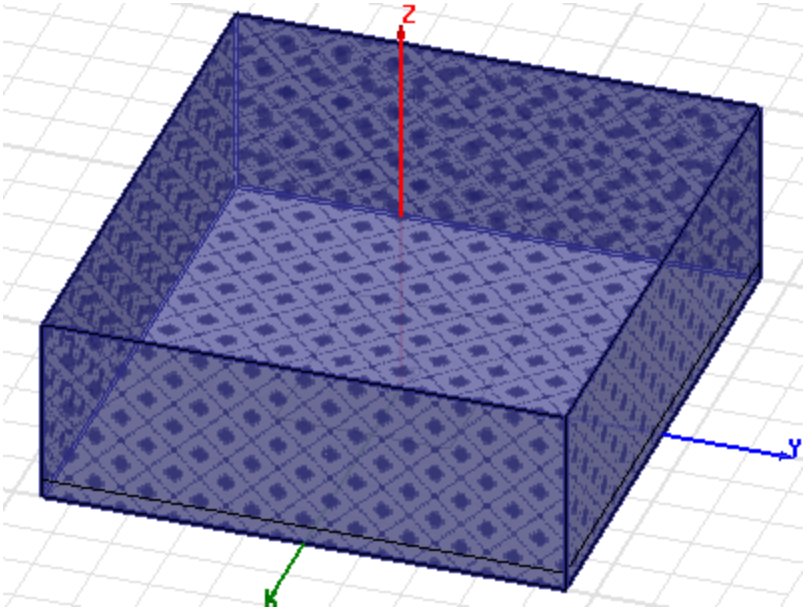


Figure 3-29 Radiation Boundary applied

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 "Far Field Radiation Sphere Setup dialog box" on the next page. and click **OK**.

   An option **ff_2d** appears under **Radiation** in the **Project Manager** window.

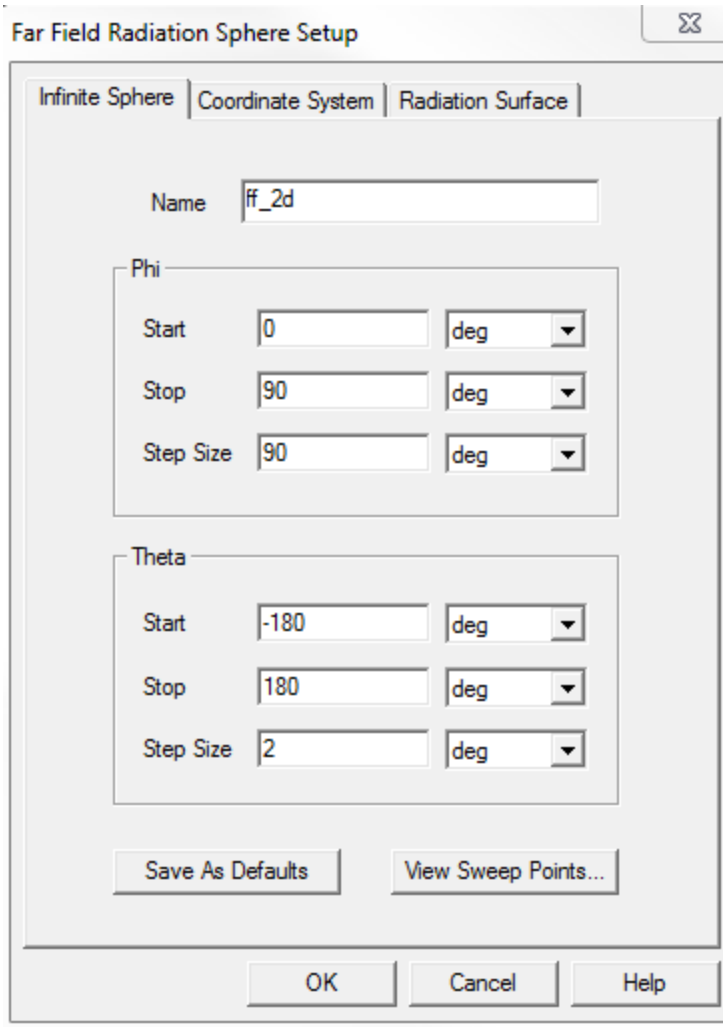


Figure 3-30 Far Field Radiation Sphere Setup dialog box

4 - Analyzing 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 "Add Solution Setup dialog box" on the next page. and click **OK**.

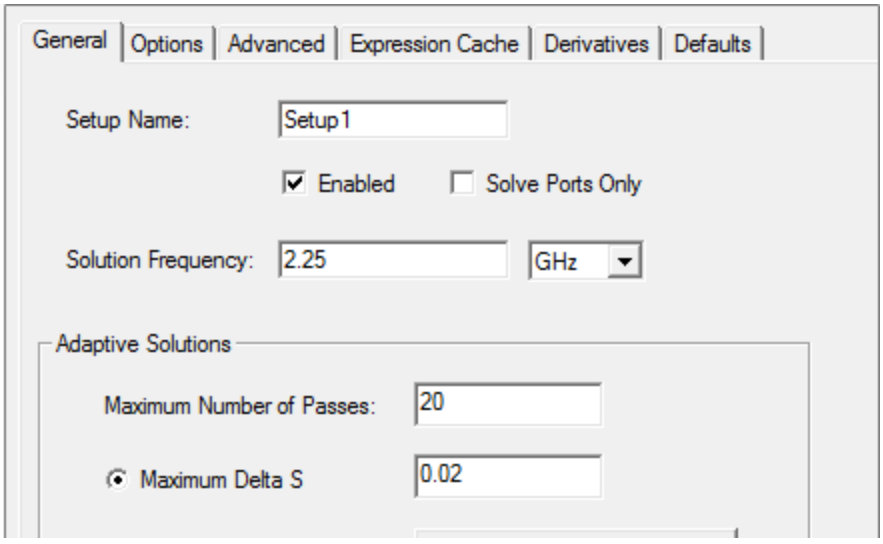


Figure 4-1 Add Solution Setup dialog box

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 Frquency Sweep** dialog box appears.
2. Edit the fields as shown in "Edit Frequency Sweep dialog box" below. and click **OK**.
3. Save the project.

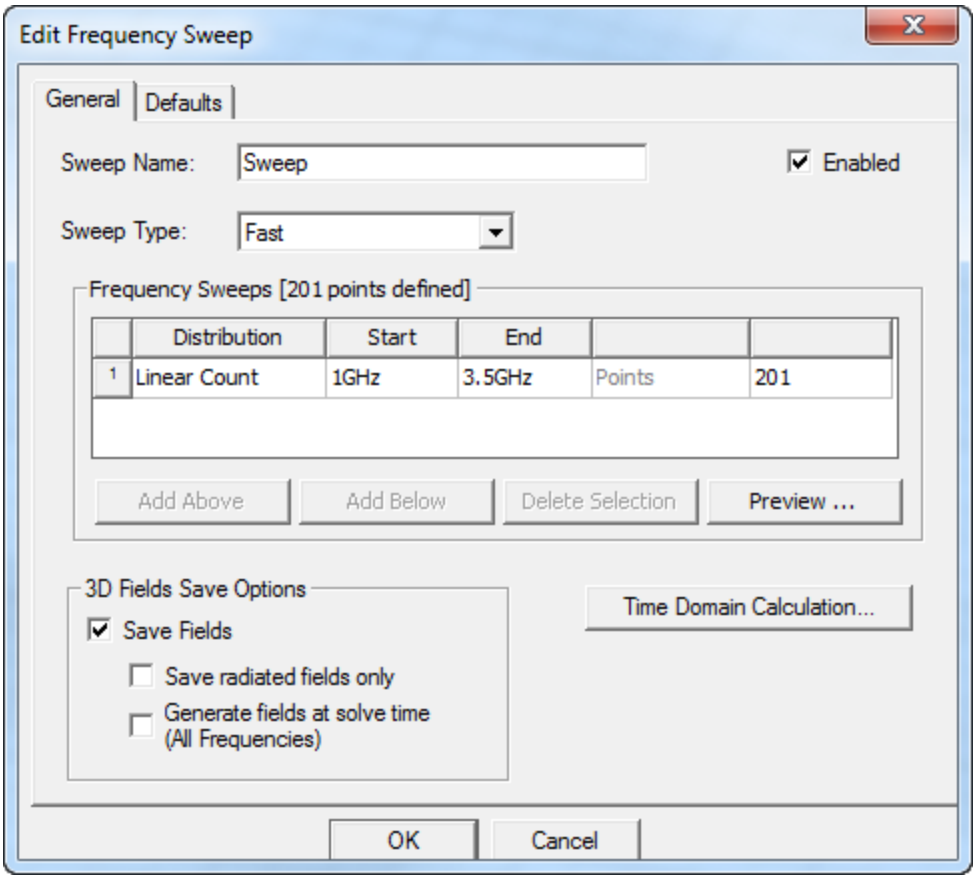


Figure 4-2 Edit Frequency Sweep dialog box

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 4-3 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, the 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.

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 bandwic
Field Recovery	00:00:00	00:00:00	60.8 M	Disk = 237 KBytes, 1 excitations
				Adaptive Passes converged

Figure 4-4 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 "X, Y axes settings" on the facing page. .

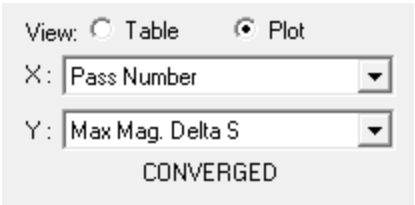


Figure 4-5 X, Y axes settings

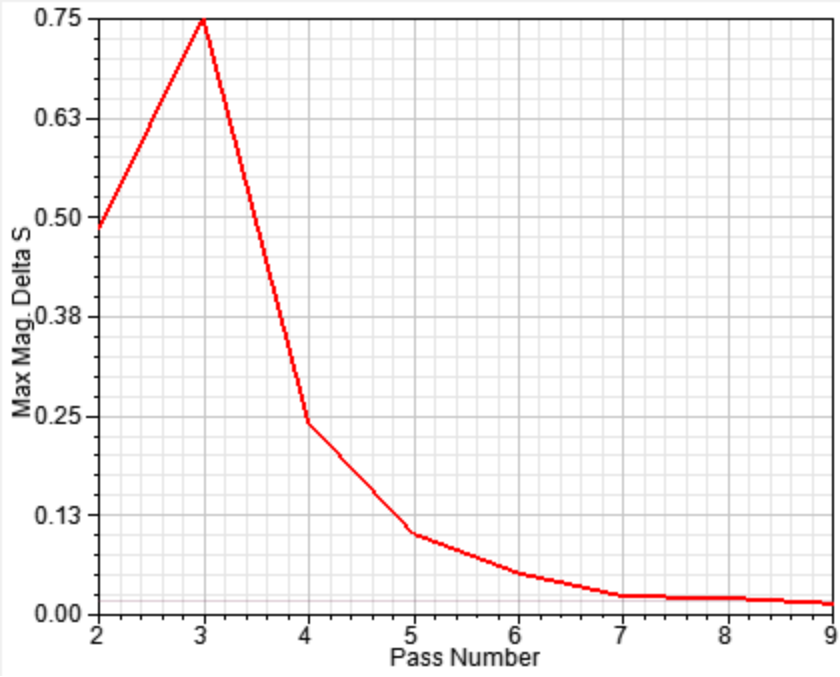


Figure 4-6 Convergence occurs in the 9th pass

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 4-7 Convergence table

Review the Matrix Data Panel

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

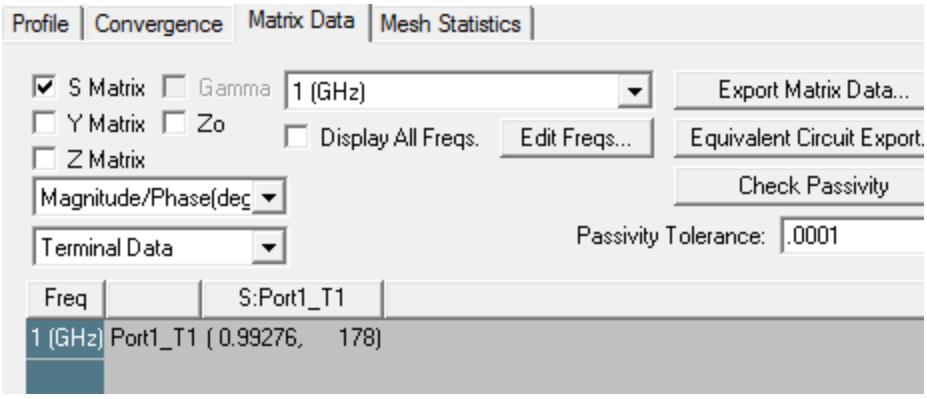


Figure 4-8 Matrix Data panel

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	Max tet vol
Air	3604	0.0955082	3.91333	0.988566	6.59465e-00...	2.84148
Coax	511	0.0389137	0.312081	0.154142	2.75814e-00...	0.000286628...
Coax_pin	238	0.0535757	0.312081	0.135343	2.03037e-00...	0.000223362...
probe	126	0.0823585	0.231698	0.140641	2.43681e-00...	0.000220048...
Sub1	4983	0.0773074	2.91018	0.500523	8.17262e-00...	0.229248

Figure 4-9 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 the following figure.

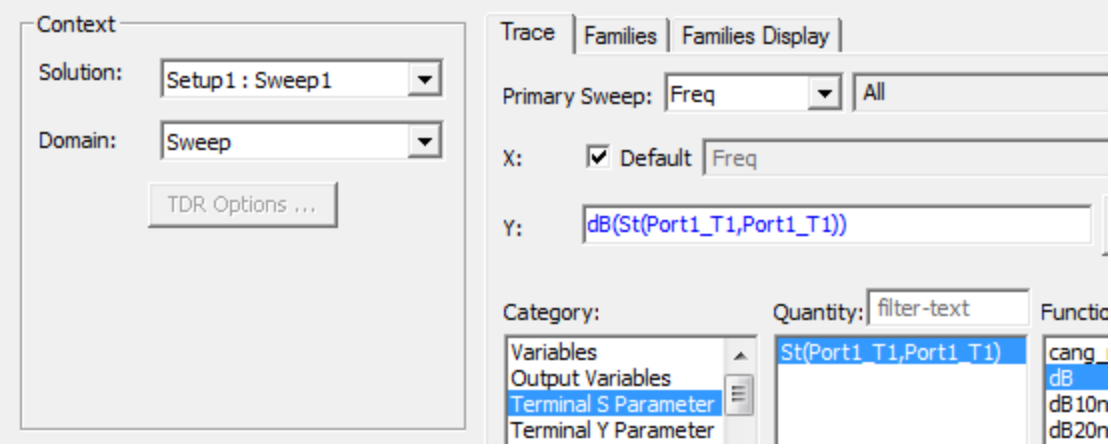


Figure 4-10 Report dialog box

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

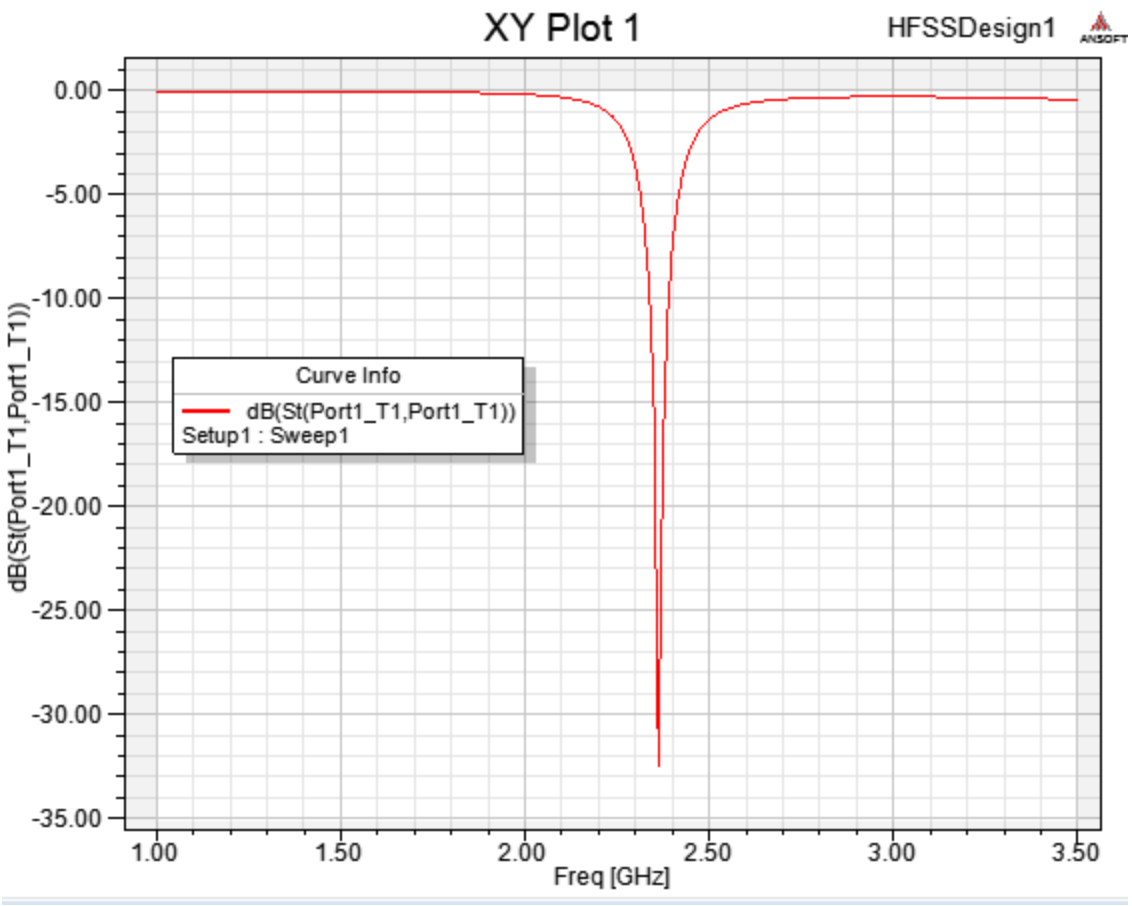


Figure 4-11 Select the trace on the plot

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

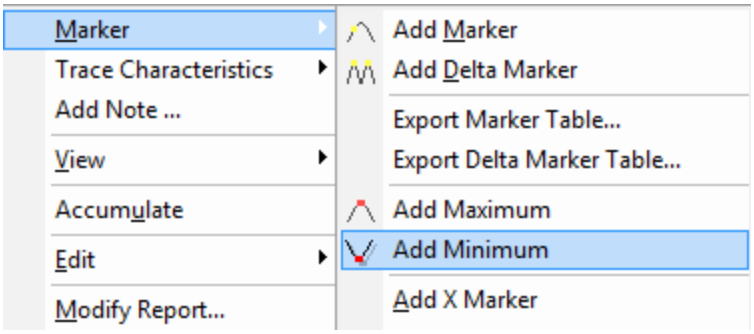


Figure 4-12 Short-cut menu

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

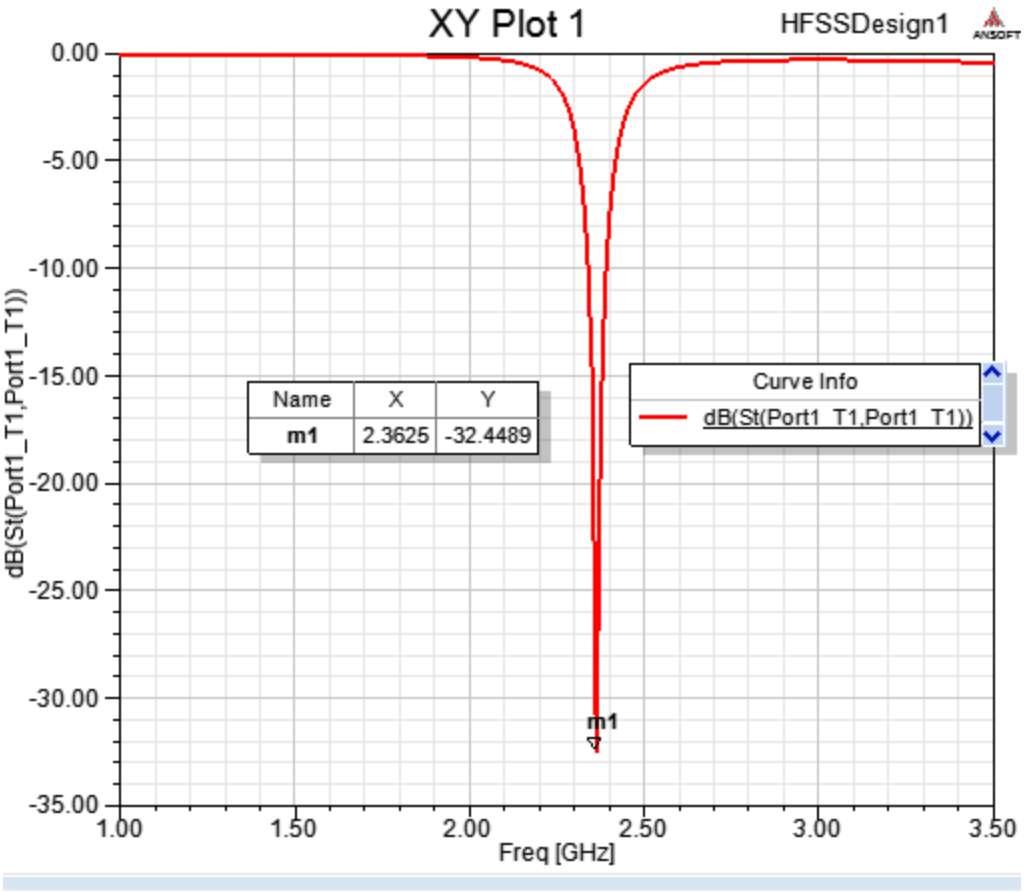


Figure 4-13 S parameter versus Frequency

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**.

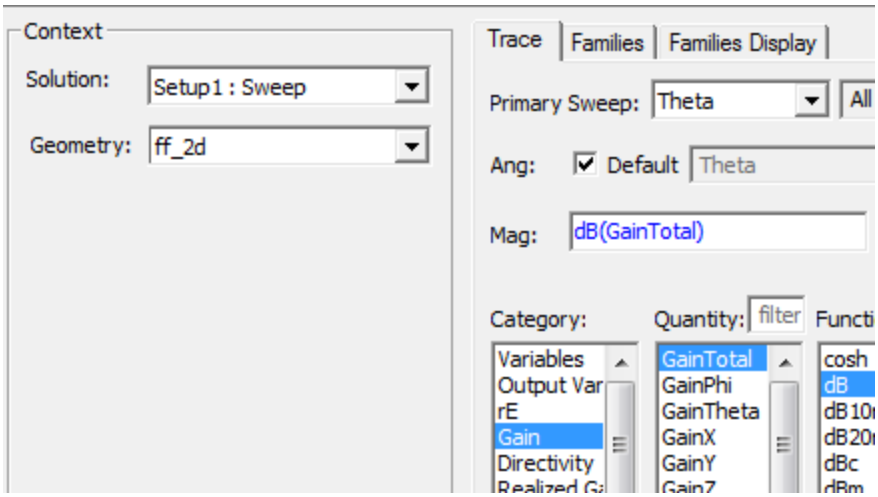


Figure 4-14 Report dialog box

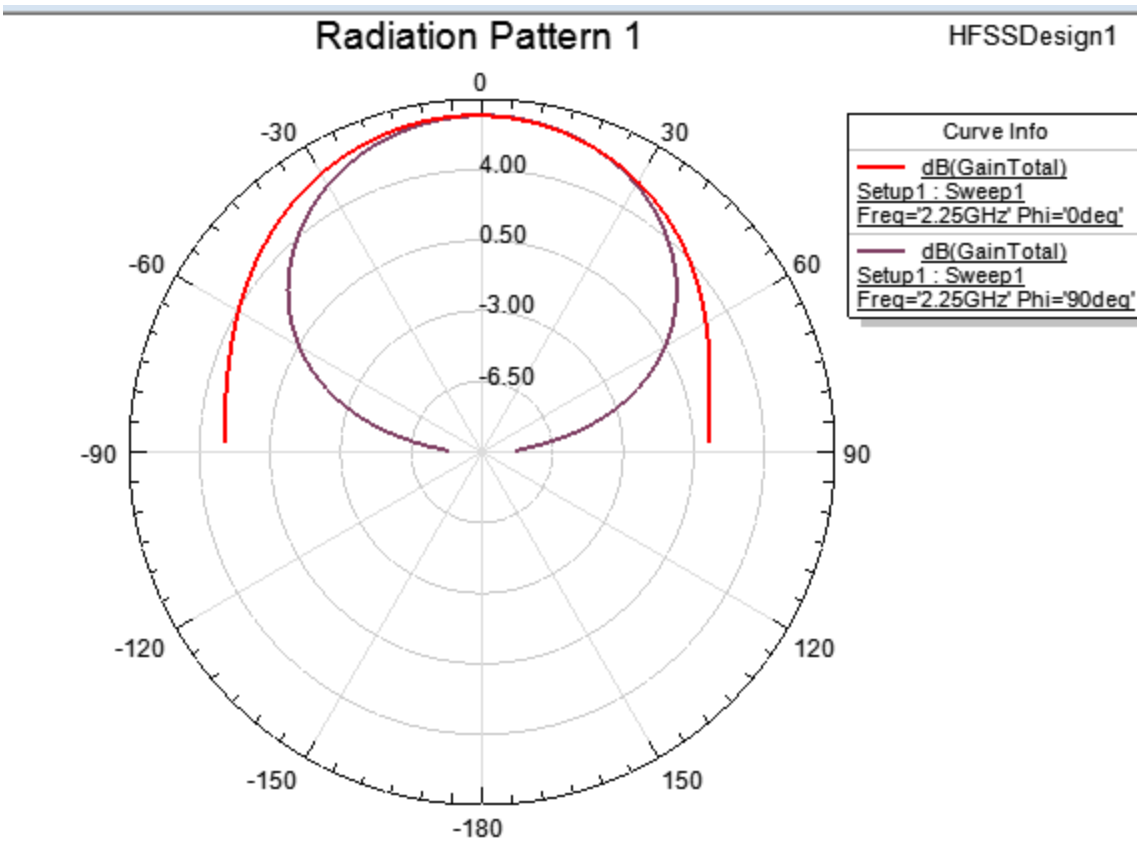


Figure 4-15 Radiation Pattern