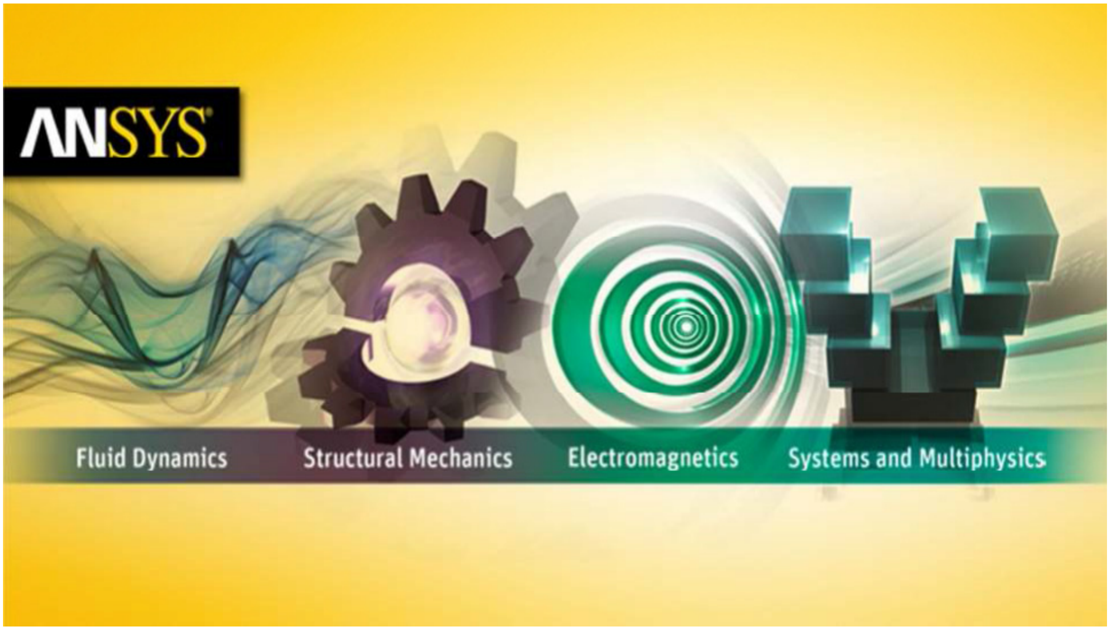


Getting Started with HFSS™
A Dielectric Resonator Antenna

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

November 2013
ANSYS Electromagnetics Suite 15.0

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

Copyright and Trademark Information

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIAR IES, 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 EM 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

6. Analyzing the Solution

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 dielectric resonator antenna.

What You Will Learn

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.

Sample Project - Dielectric Resonator Antenna

In this project you will learn to setup, solve, and analyze a dielectric resonator antenna. The antenna is cavity-backed with an annular-slot-fed hemispherical dielectric resonator. The antenna feed is achieved by coaxial excitation across one side of an annular slot between the cavity and the DRA dielectric. For this project, the engineering focus is on the behavior of the antenna itself, not its feed. Therefore, the model will feed with a lumped port across an annular slot. The design's operating frequency will be 3.5 GHz.

Note This project is also described and analyzed in the following IEEE publication: Leung, K.W., So, K.K., "Annular-slot-Excited Dielectric Resonator Antenna with a Backing Cavity," *IEEE Transactions on Antennas and Propagation*, August 2002.

A Dielectric Resonator Antenna designed on HFSS is illustrated in Figure 1.

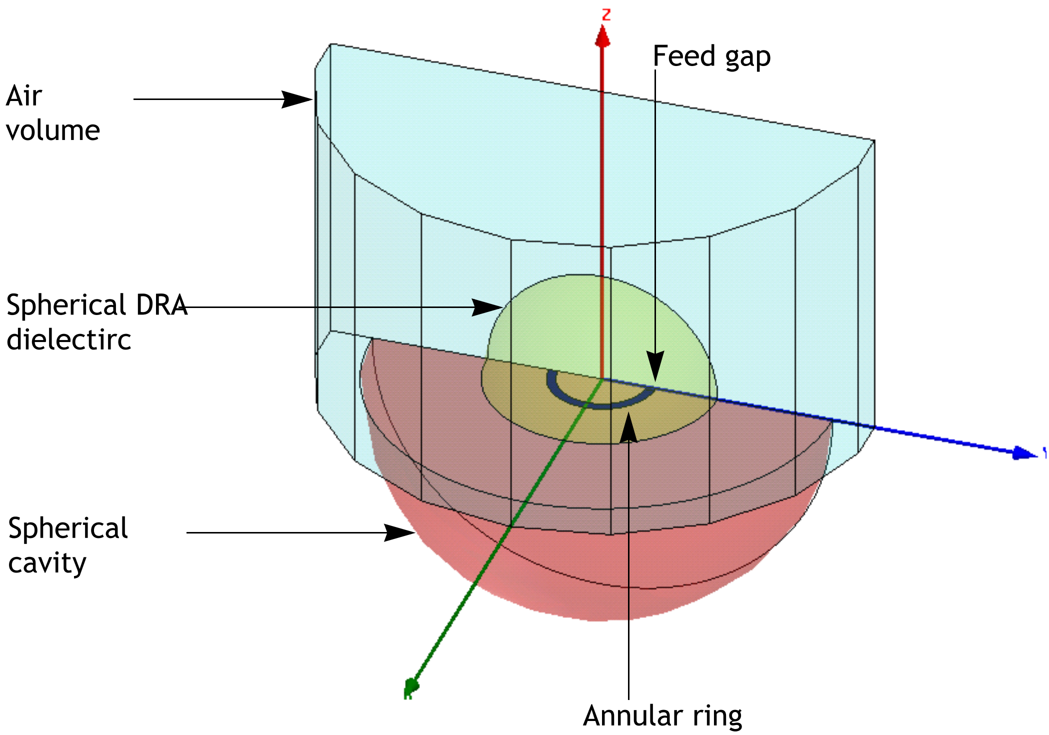


Figure 1. Sample Project - Dielectric Resonator Antenna

Finite Element Method

The geometry for the dielectric resonator antenna problem (DRA) described in this document appears below. The antenna is cavity-backed with an annular-slot-fed hemispherical dielectric resonator. The antenna feed is achieved by coaxial excitation across one side of an annular slot between the cavity and the DRA dielectric.

In HFSS, the model automatically gets 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*. Figure 2 shows the mesh created for the dielectric resonator antenna.

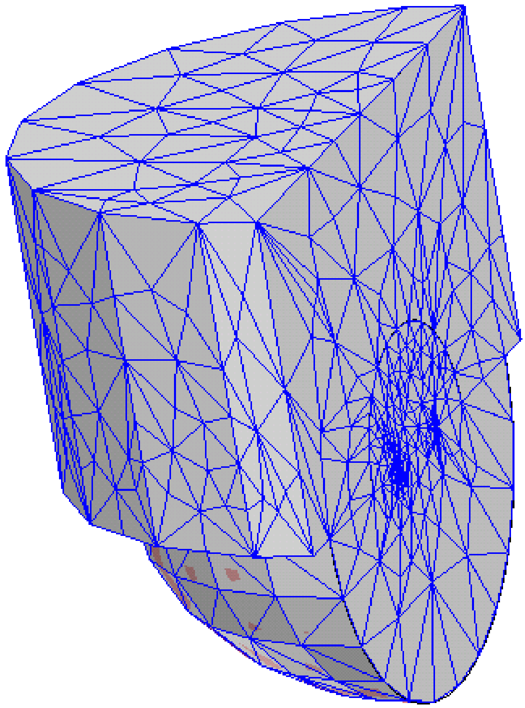


Figure 2. The Mesh that Constitutes the Antenna

Dividing a structure into thousands of smaller regions (elements) allows the system to compute the field solution separately in each element. The smaller the system makes the elements, the more accurate the final solution will be.

2 Create the New Project

This chapter contains the following topics:

- Overview of the Interface
- Launch HFSS and Set-up the Project
- Set the Units Of Measurement
- Verify HFSS and Modeler Options
- Select the Solution Type
- Set the Co-ordinate System

Overview of the Interface

An overview of the major components of HFSS14's interface is in the Figure below.

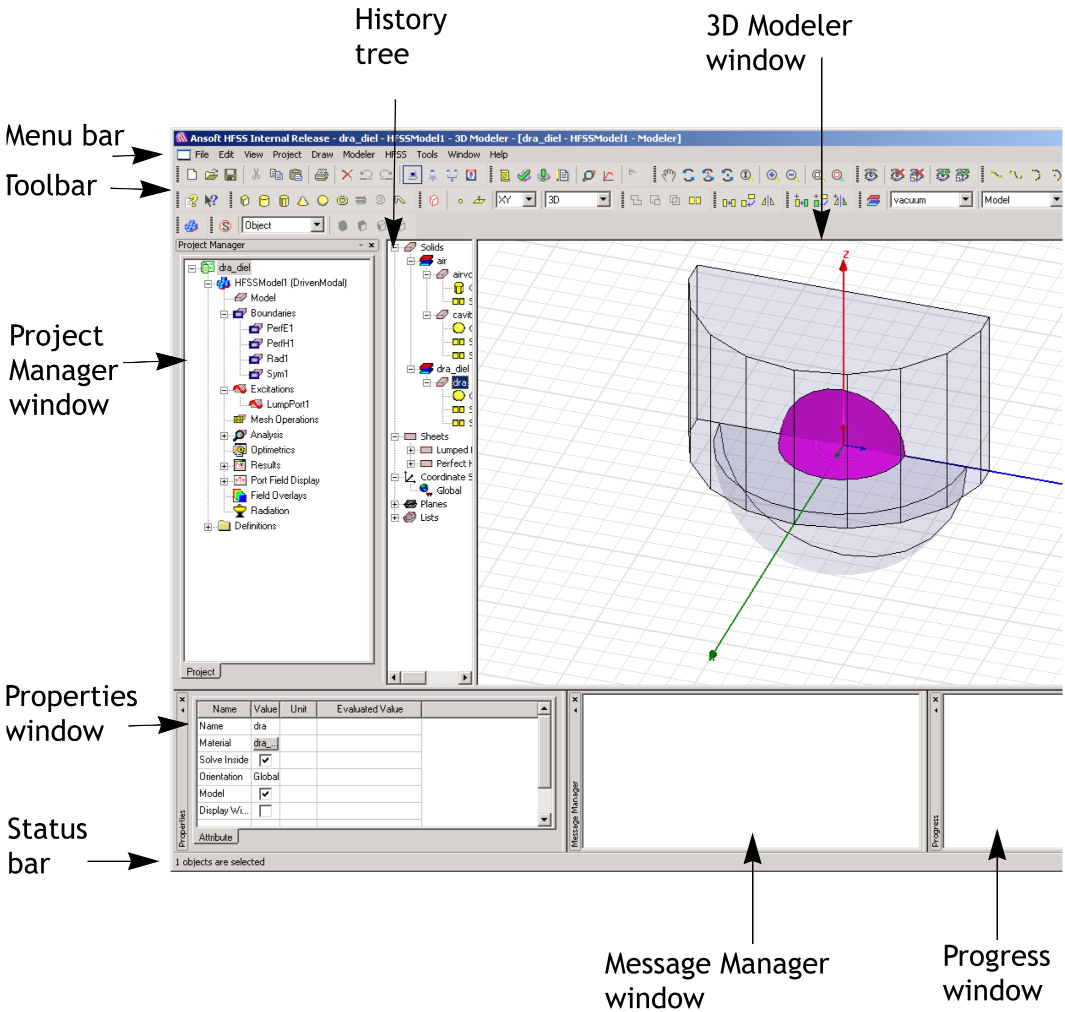


Figure 3. Overview of HFSS GUI

The different options on the GUI are described as follows:

Project Manager window	Displays details about all open HFSS projects. Each project has its own project tree, which ultimately includes a geometric model and its boundaries and excitations, material assignments, analysis setups, and analysis results.
Message Manager window	Displays error, informational, and warning messages for the active project.
Progress window	Displays solution progress information.
Properties window	Displays the attributes of a selected object in the active model, such as the object's name, material assignment, orientation, color, and transparency. Also displays information about a selected command that has been carried out. For example, if a circle was drawn, its command information would include the command's name, the type of coordinate system in which it was drawn, the circle's center position coordinates, the axis about which the circle was drawn, and the size of its radius.
3D Modeler window	Displays the drawing area of the active model, along with the history tree.
History tree	Displays all operations and commands carried out on the active model, such as information about the model's objects and all actions associated with each object, and coordinate system information.

Menu bar	Provides various menus that enable you to perform all of the HFSS tasks, such as managing project files, customizing the desktop components, drawing objects, and setting and modifying all project parameters.
Toolbars	Provides buttons that act as shortcuts for executing various commands.
Status bar	Shows current actions and provides instructions. Also, depending on the command being carried out, the status bar can display the X, Y, and Z coordinate boxes, the **Absolute/Relative** pull-down list to enter a point's absolute or relative coordinates, a pull-down list to specify a point in cartesian, cylindrical, or spherical coordinates, and the active model's unit setting.

Launch HFSS and Set-up the Project

You must launch HFSS and set up a project to design a Dielectric Resonator Antenna. As part of setting up the project you must perform the following tasks:

- Launch HFSS
- Open and Save a New Project
- Set the Units of Measurement
- Verify HFSS Options
- Select Solution Type

This section shows you how to accomplish the above tasks.

Note We recommend you store a shortcut of the HFSS application on your desktop.

1 Double-click the **HFSS 14** icon on your desktop.

HFSS opens with a new project listed on its Project Manager window. See Figure 2.

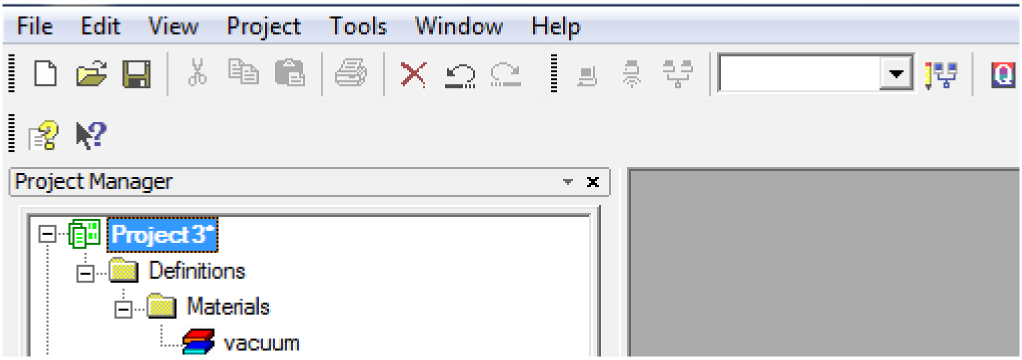


Figure 4. Project Manager Window

Note If a project is not listed in the project tree, go to **File** and click **New** to include one; and if the **Project Manager** window does not appear after launching the application, go to **View** and enable it.

2 Expand the project tree to see if the "**Insert HFSS Design**" option is present and do one of the following steps:

a. If the option is present, go to step 3.

b. If absent, on the toolbar click Project and select "Insert HFSS Design."

HFSSDesign*n* appears in the **Project tree**. See Figure 3.

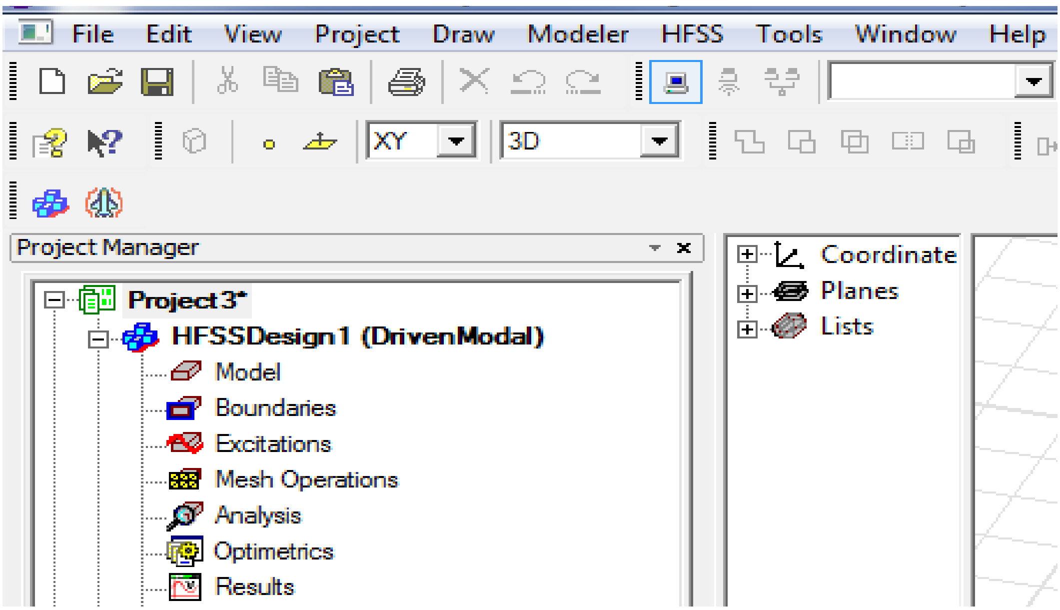


Figure 5. HFSS Design*n* included

Note A new project is listed in the project tree in the Project Manager window with the default label Project*n*. **Definitions** and **Materials** are stored under the project name.

3 Right click **HFSSDesign*n***, click **Rename** on the shortcut menu, type Dielectric Resonator Antenna and hit **Enter**.

The design is renamed as *DielectricResonatorAntenna*.

4 Click **File>Save As**.

The **Save As** dialog box appears.

5 Browse for a location to store the file (such as C:\Program Files\Ansoft\HFSS14.0\Projects) and then, double-click the folder's name.

6 Type ***dr_antenna*** in the **File name** text box, and then, click **Save**.

The project is saved in the folder you selected to the file name *dr_antenna.hfss*.

Set the Units of Measurement

You must set the units of measurement for drawing the geometric model (UHF Probe) as follows:

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

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

Figure 6. Set Model Units dialog box

Select **mm** (millimeters) from the **Select units** drop-down menu, and click **OK**.

Verify HFSS and Modeler Options

Certain options under the **Tools** drop-down menu have to be verified before you create the design model.

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

The **HFSS Options** dialog box opens.

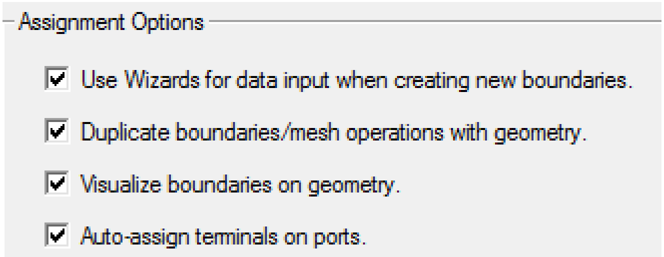


Figure 7. Assignment Options

2 Click the **General** tab.
3 Verify whether the **Assignment Options** are checked. See Figure 5.
4 Click **OK** to close the dialog box.
5 Click **Tools>Options>Modeler Options**.

   The **Modeler Options** dialog box opens.
6 Click **Operation.**
7 Check “**Automatically cover closed polylines.**”
8 Click the **Drawing** tab.
9 Check “**Edit property of new primitives.**”

   This will cause a **Properties** dialog box to open when you create a new primitive object in the Modeler.
10 Click **OK.**

Select the Solution Type

To set the solution type:

1 Click **HFSS>Solution Type** on the toolbar.

   The **Solution Type** dialog box opens.

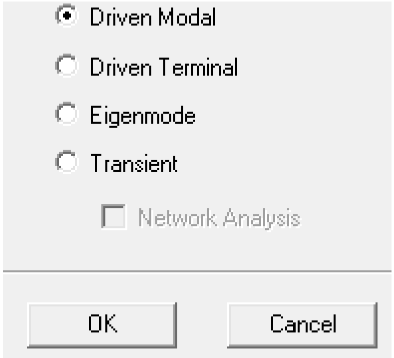


Figure 8. Solution Type dialog box

2 Select **Driven Modal**.

Note This antenna project is a mode-based problem so select the Driven Modal option.

3 Click **OK**.

Now, you are ready to draw the objects for the antenna problem.

Set the Co-ordinate System

For this antenna problem, you will use the fixed, default global coordinate system (CS) as the working CS. HFSS has three types of coordinate systems that let you easily orient new objects. They are described below.

Global CS	The fixed, default CS for each new project.
Relative CS	A user-defined CS. Its origin and orientation can be set relative to the global CS, relative to another **Relative CS**, or relative to a geometric feature. **Relative CSs** enable you to easily draw objects that are located relative to other objects.
Face CS	A user-defined CS. 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.

Grid Settings

The grid displayed in the **3D Modeler** window is a drawing aid to visualize the location of objects. The points on the grid are divided by their local x-, y-, and z-coordinates and grid spacing is set according to the current project’s drawing units.

For this antenna project, it is not necessary to edit any of the grid’s default properties.

Note To edit the grid’s properties, click **Grid Settings** on the **View** menu to control the grid’s type (cartesian or polar), style (dots or lines), density, spacing, or visibility.

3 Draw the Model

This chapter contains the following topics:

- Overview of the Interface
- Launch HFSS and Set-up the Project
- Set the Units Of Measurement
- Verify HFSS and Modeler Options
- Select the Solution Type

Create the DRA Geometry

The geometry for this dielectric resonator antenna (DRA) model consists of five basic objects as follows:

Air volume	30 mm radius and a height of 35 mm
Spherical Cavity	25 mm radius
Spherical DRA	12.5 mm radius
Annular ring	5.8 mm outer radius and a width of 1.0 mm.
Feed gap	1mm thickness

Create the Spherical Cavity

You will first draw the antenna's spherical cavity. To this end, you will draw a sphere first and then, split it to form a hemi-spherical solid.

1 Click **Draw>Sphere**.

2 Draw a sphere with origin as center and radius=25mm.

The **Properties** dialog box appears.

	Name	Value	Unit	Evaluated Value
	Command	CreateSphere		
	Coordinate System	Global		
	Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
	Radius	25	mm	25mm

Figure 9. Sphere Properties

3 Enter the fields in the dialog box so that they are the same as what you see in Figure 1.

The sphere appears in the drawing region.

4 Press **Ctrl+D** to fit the sphere in the drawing region.

The sphere must resemble the one shown in Figure 2.

5 Click **Attribute** and type *cavity* in the **Name** field.

6 Click **OK**.

The **Properties** dialog box disappears.

7 Select **cavity** from **History Tree** to highlight the sphere.

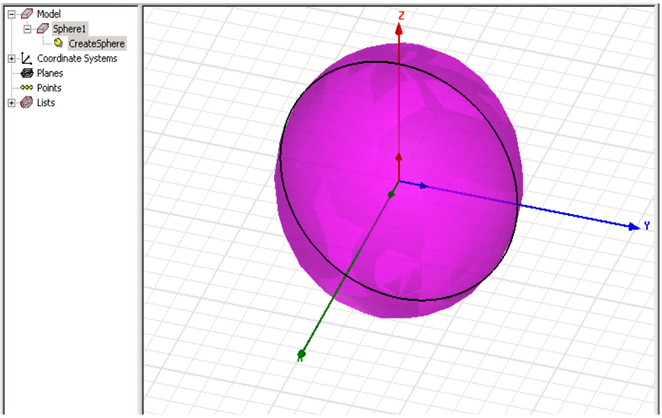


Figure 10. Sphere in the 3-D Modeler window

8 Click **Modeler>Boolean>Split.**

The **Split** dialog box appears.

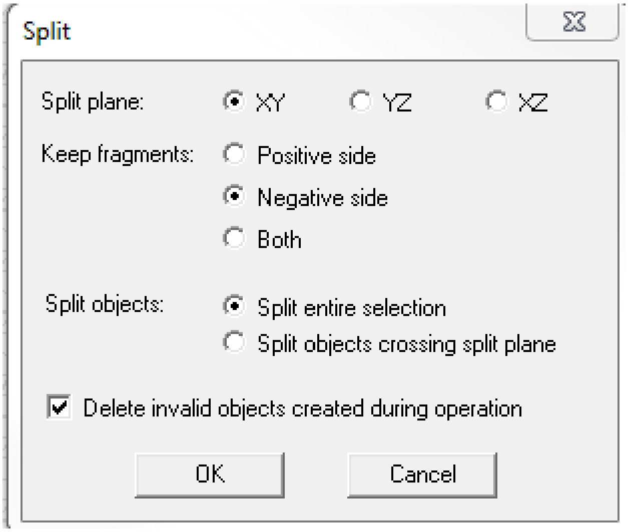


Figure 11. Split dialog box

9 Enter the following settings:

- **Split plane** = *XY*.
- **Keep Fragments** = *Negative side*
- **Split objects** = *Split entire selection*

10 Click **OK.**

The object **cavity** is split into a hemispherical solid.

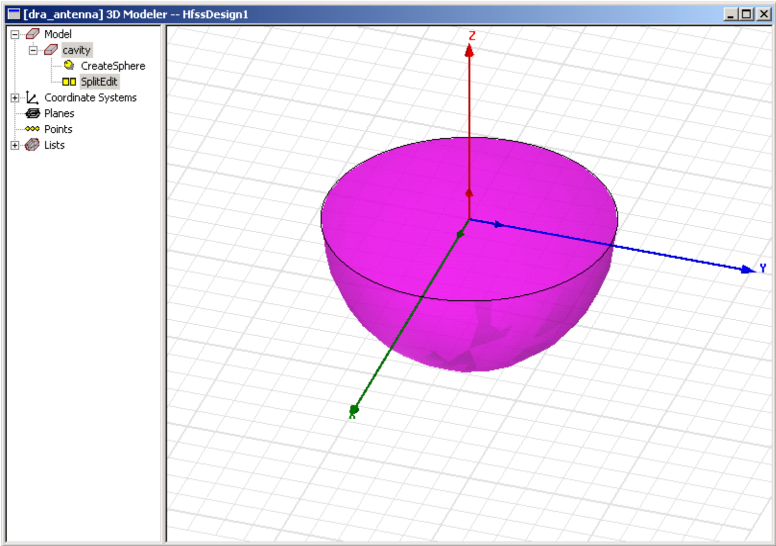


Figure 12. Cavity

Modify Cavity Attributes

1 Doble-click **cavity** on the History Tree.

The **Attribute** tab opens.

Name	Value	Unit	Evaluated Value
Name	cavity		
Material	"vacuum"		"vacuum"
Solve Inside	☑		
Orientation	Global		
Model	☑		
Display Wireframe	☐		
Color	Edit		
Transparent	0.07		

Figure 13. Cavity Attributes

2 Click **Edit** in the **Color** row.

The **Color** palette appears.

Figure 14. Color Pallette

3 Select red.

4 Enter *255, 0, 0* in the **RGB** settings as shown in Figure 6.

5 Click button *0* in the **Transparency** row.

The **Set Transparency** window appears.

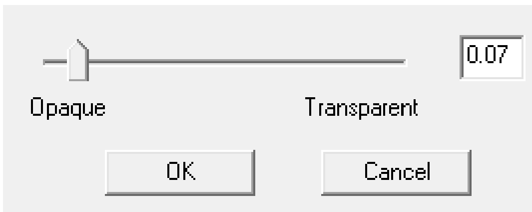


Figure 15. Set Transparency dialog box

6 Enter *.07* for the value of **Transparency**.

The **Transparency** is now set to **.07**.

7 Select "*vacuum*" as the material.

8 Click outside the object, on the grid background, to deselect **cavity.**

The object updates with the attributes that you set.

Verify Lighting Attributes

1 Click **View>Modify Attributes>Lighting.**

The **Lighting Properties** dialog box appears.

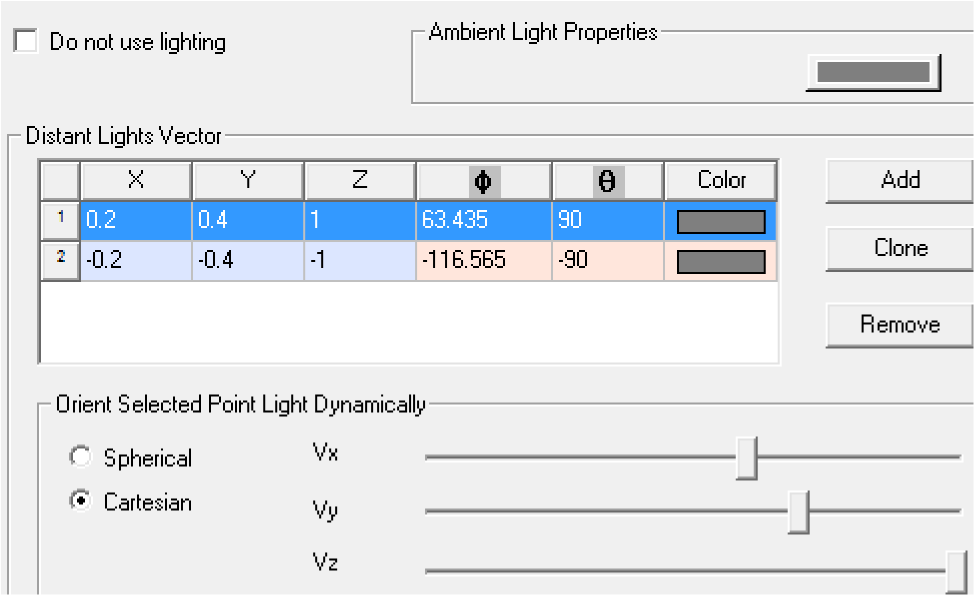


Figure 16. Lighting Properties dialog box

2 Verify that the **Do not use lighting** option is unchecked.

3 Click **OK.**

Your completed object **cavity** should resemble the one shown below:

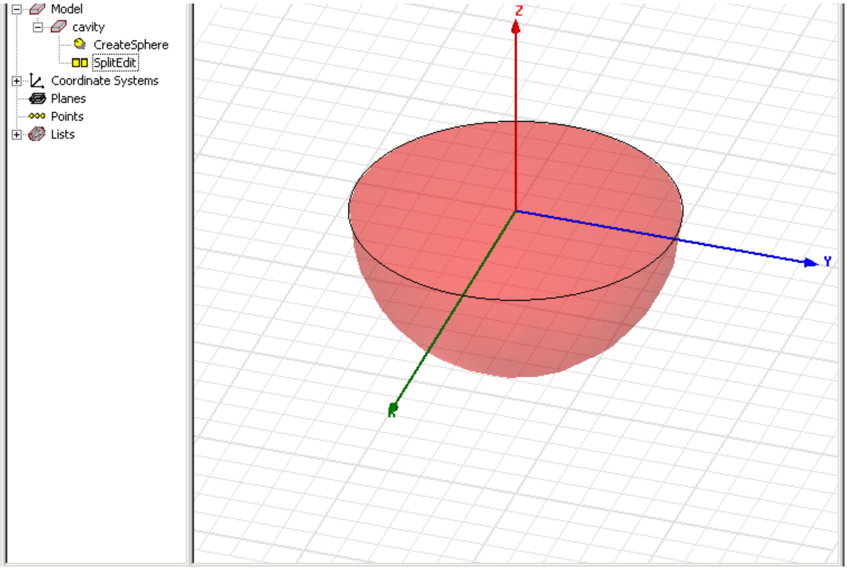


Figure 17. The cavity with the latest attributes applied

Draw the Dielectric Resonator

Now, you will draw another sphere, split it, and then edit its Attributes.

1 Click **Draw>Sphere**.

2 Draw a sphere with center at origin and radius=12.5 mm.

	Name	Value	Unit	Evaluated Value
	Command	CreateSphere		
	Coordinate System	Global		
	Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
	Radius	12.5	mm	12.5mm

Figure 18. Properties for dra

3 Enter the settings as shown in Figure 10.

4 Click **Attribute** and type *dra* in the **Name** field.

5 Click **OK**.

The sphere appears in the 3-D Modeler window.

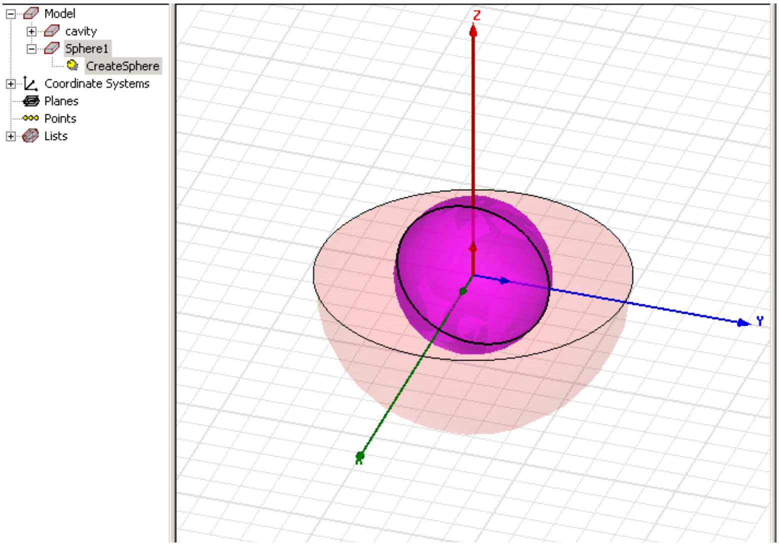


Figure 19. The dra object

6 Select **dra** to highlight it.

7 Click **Modeler>Boolean>Split.**

The **Split** dialog box appears.

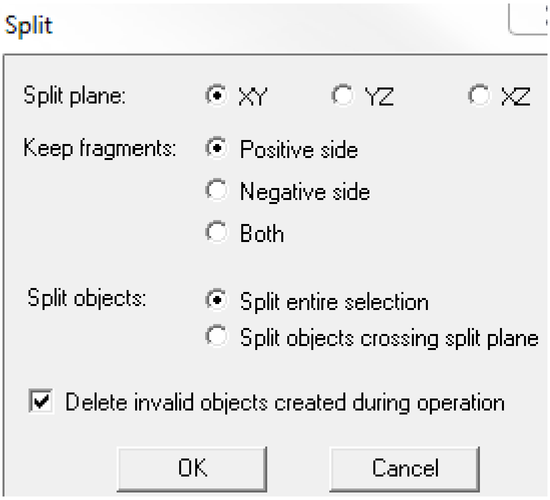


Figure 20. Split dialog box

8 Enter the following settings:

- **Split plane** = *XY*.
- **Keep Fragments** =*Positive side*

- **Split objects** = *Split entire selection*

9 Click **OK.**

The object *dra* is split into a hemispherical solid.

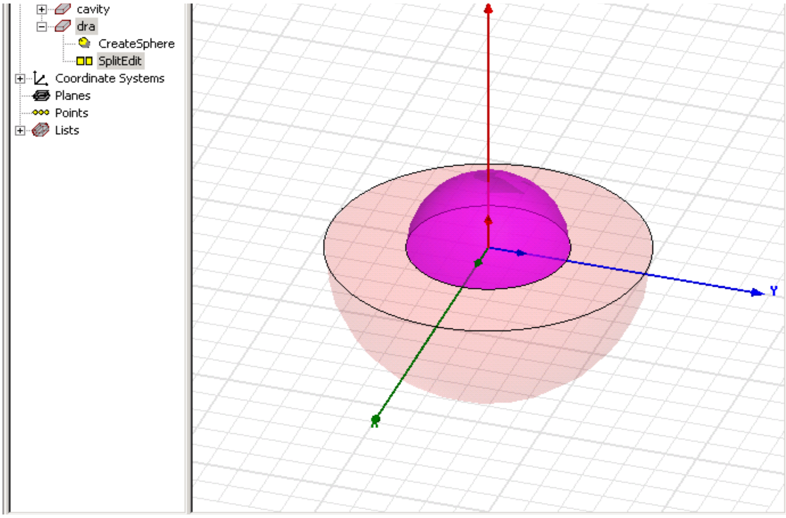


Figure 21. dra split.

Modify DRA Attributes

1 Double-click **dra** on the History Tree.

The **Attribute** dialog box appears.

2 Click **Edit** in the **Color** row.

3 Select yellow.

4 Set *RGB* to *255, 255, 0* from the **Color** palette, and then click **OK**. See Figure 14.

5 Change **Transparency** to *0.07*.

6 Click *Edit* from the **Material** pull-down menu (Figure 15).

The **Select Definitions** dialog box appears. See Figure 16.

7 Click **Add Material**.

The **View/Edit Material** dialog box appears. See Figure 17.

8 Enter **Material Name** as *dra_diel*.

9 Enter **Value** as *9.5*.

10 Click **OK.**

View Edit/Material dialog box closes and Select Definitions dialog box gets updated with the new entry.

11 Click **OK** on the **Select Definitions** dialog box.

The **Attributes** dialog box gets updated with *dra_diel* in the **Material** field. See Figure 19.

12 Click **OK**.

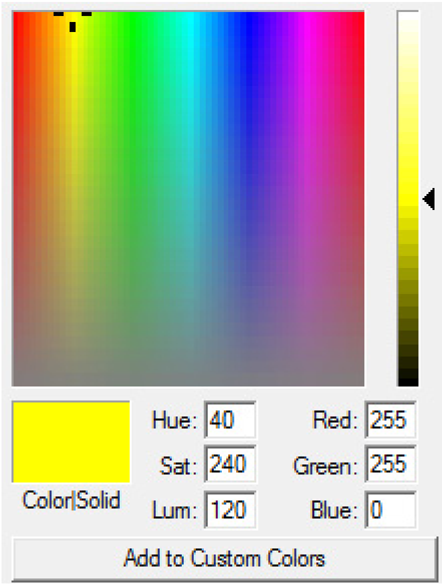


Figure 22. Color Palette

Name	Value	Unit	Evaluated...
Name	dra		
Material	Edit...		"vacuum"
Solve Inside	✓		
Orientation	Global		
Model	✓		
Display Wireframe			
Color	Edit		
Transparent	0.07		

Figure 23. dra Attributes

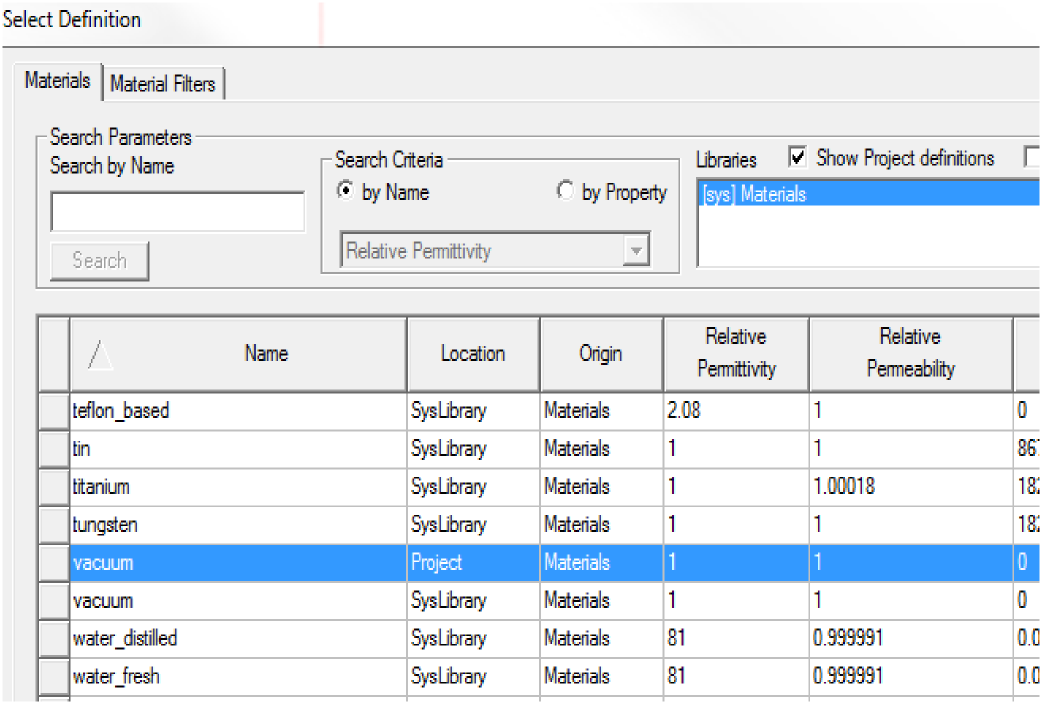


Figure 24. Select Definition dialog box

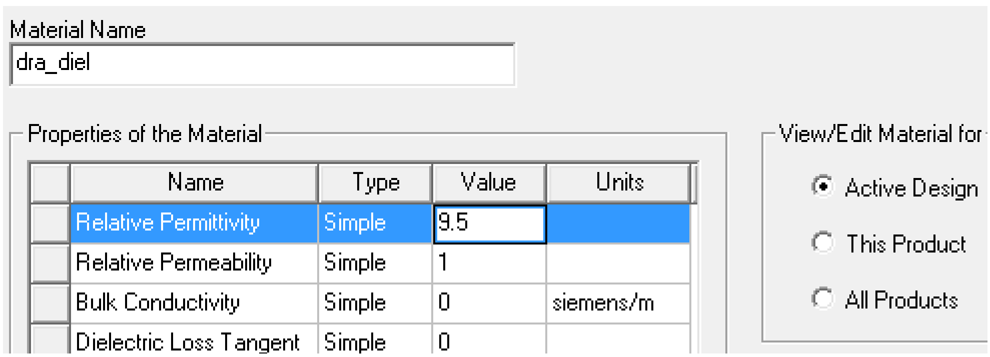


Figure 25. View/Edit Material dialog box

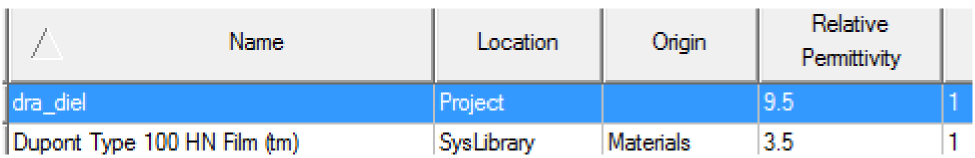
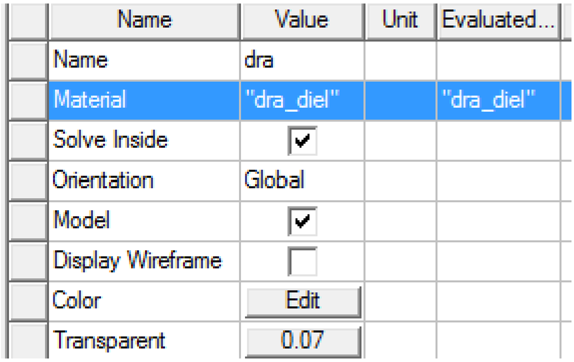


Figure 26. Updated Select Definitions



Name	Value	Unit	Evaluated...
Name	dra		
Material	"dra_diel"		"dra_diel"
Solve Inside	☑		
Orientation	Global		
Model	☑		
Display Wireframe	☐		
Color	Edit		
Transparent	0.07		

Figure 27. Updated Attributes tab

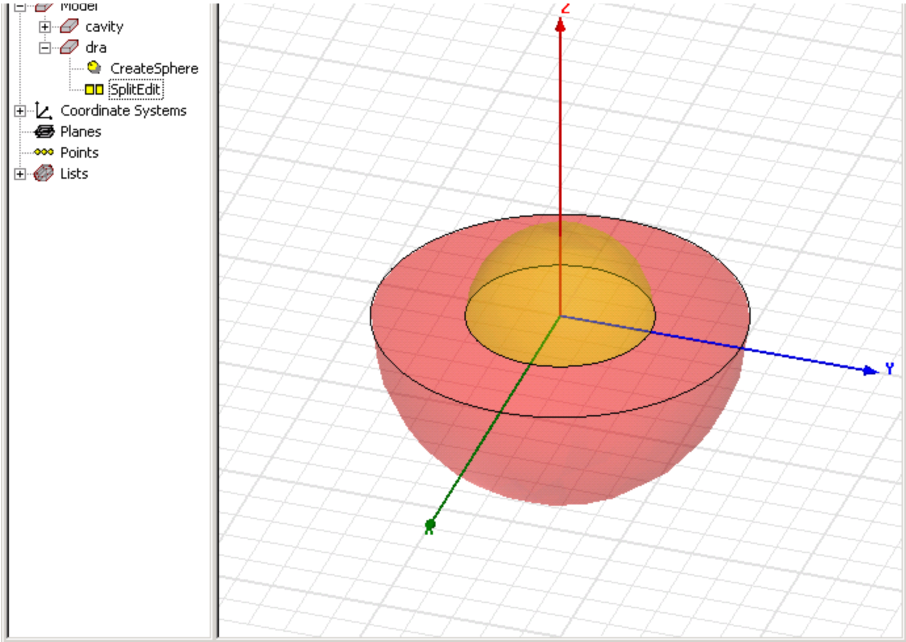


Figure 28. Current Structure

Create the Annular Feed Ring

In this antenna model, the annular feed ring is the controlled aperture through which the E-fields will radiate. Later on in the project, you will assign a perfect H boundary to the annular feed ring to allow the E-fields to radiate through it. In this section, you will create the antenna's annular feed ring, which is the result of subtracting one circle from another.

Draw Circle1

1 Click **Draw>Circle**.

2 Enter (0, 0, 0) in the status bar fields for the position of the circle.

3 Press **Enter**.

4 Enter **dX**=*4.8* as the radius.

5 Press **Enter**.

The **Properties** dialog box appears and **Circle1** appears in the model.

Note You can even draw the circle free hand, click the corresponding CreateCircle option from the history tree and enter the position as (0,0,0) and radius as 4.8mm in the Properties dialog box that will show up. However, this is a roundabout way of drawing the circle, which can be accomplished by simply entering the dimensions in the status bar as shown above.

Draw Circle2

In this section, we will discuss drawing the circle freehand.

1 Click **Draw>Circle**.

2 Draw the circle at any arbitrary location.

The **Properties** dialog box appears with the fields suppressed.

	Name	Value	Unit	Evaluated Value
	Command	CoverLines		
	Suppress Command	☐		

Figure 29. Properties dialog box

The circle appears in the 3-D Modeler field.

3 Double-click **CreateCircle** under **Circle2** in the History Tree.

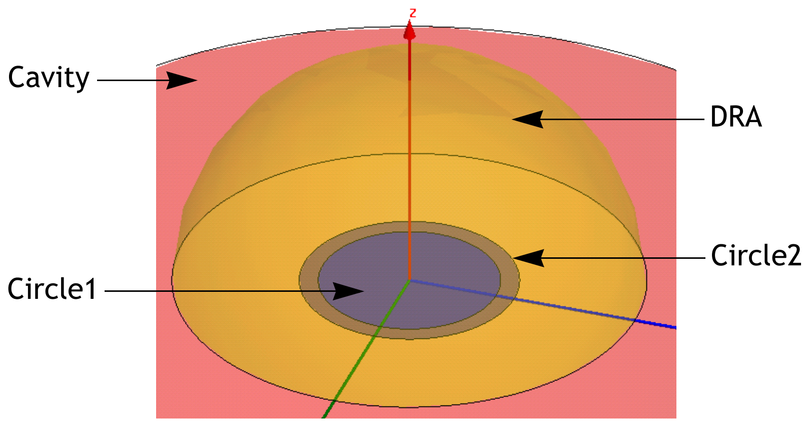
The **Properties** dialog box appears.

4 Change the values for the **Center Position** and **Radius** to (*0, 0,0*) and *5.8mm* as shown in Figure 22.

Name	Value	Unit	Evaluated Value
Command	CreateCircle		
Coordinate System	Global		
Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
Axis	Z		
Radius	5.8	mm	5.8mm
Number of Segments	0		0

Figure 30. Circle2 Properties

Circle1 and **Circle2** should now both appear in your model, as shown below:



Subtract Circle1 from Circle2

Subtracting **Circle1** from **Circle2** will result in an annular feed ring of 1 mm width.

1. Select **Circle2** either by clicking it in the **3D Modeler** window or clicking its name in the history tree.

   Note **Circle2** will be the *blank* object — the object from which subtraction will occur.

2. Press and hold down the **Ctrl** key to also select **Circle1**.

   Note **Circle1** is the *tool* object—the object to be subtracted from the blank object.

3. Verify whether the status bar indicates number of objects selected is two.

4 Click **Modeler>Boolean>Subtract.**

The **Subtract** window appears:

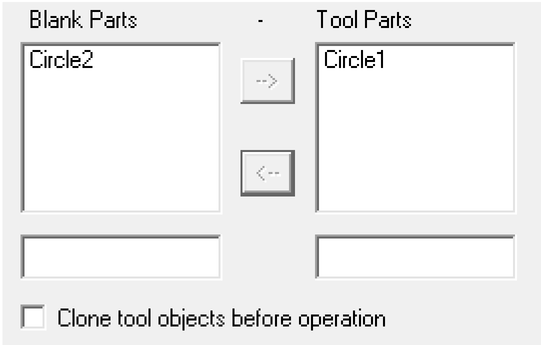


Figure 31. Subtract dialog box

5 Verify that **Circle1** is in the **Tool Parts** list and **Circle2** is in the **Blank Parts** list.

6 Click **OK.**

Circle1 is subtracted from **Circle2**, resulting in a 1 mm wide flat annulus or ring.

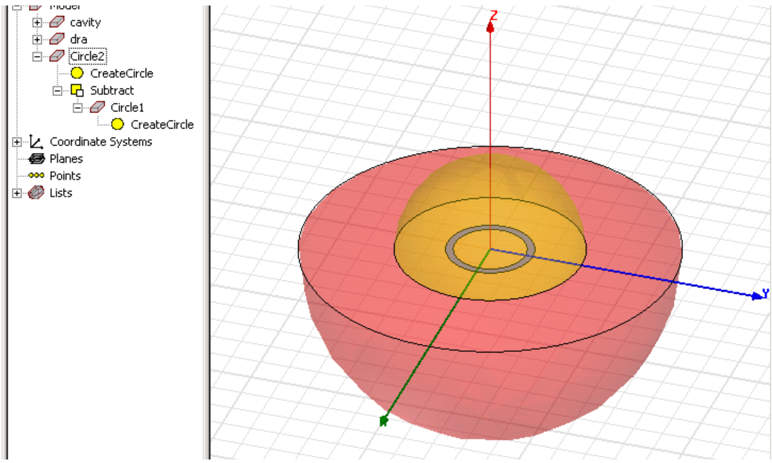


Figure 32. Structure after subtraction

7 Double-click **Circle2** from the history tree.

The **Attribute** dialog box appears.

8 Type **annular_rng** to rename, and then press **Enter** to accept the new name.

9 Click **Edit** in the **Color** row.

10 Enter *0, 0, 128* as the RGB settings from the **Color** palette, and then click **OK**.

11 Set **Transparency** to *.09*.

12 Click **OK**.

The structure gets updated with the latest settings.

13 Deselect **annular_rng** to view the resulting color and transparency assignments.

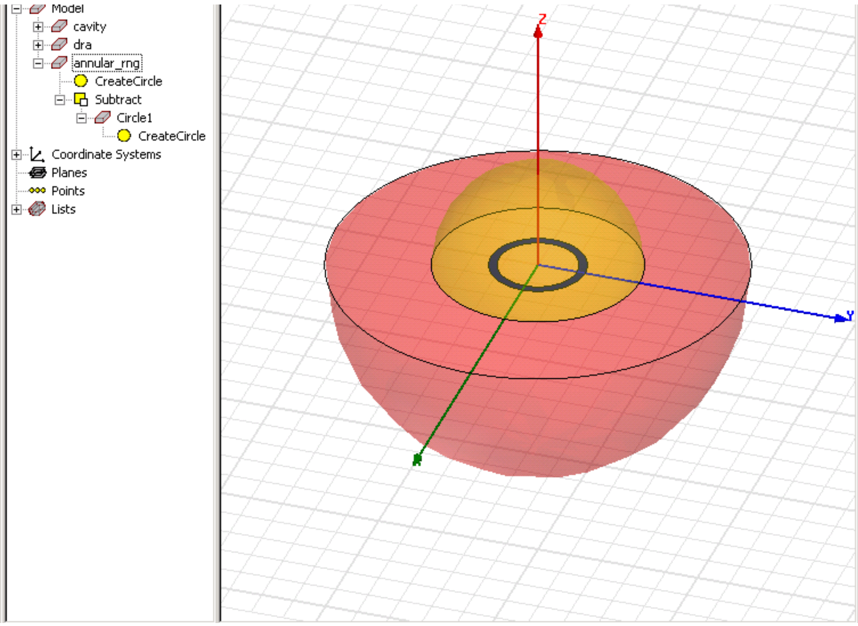


Figure 33. annular_rng

Draw the Feed Gap

Feed gap is the object through which the excitation is fed.

1 Draw a **Rectangle**.

The rectangle is drawn and the **Properties** dialog box appears with suppressed command.

2 Click **OK**.

3 Double-click **CreateRectangle** from the History Tree.

The **Command** tab of the **Properties** dialog box appears.

Name	Value	Unit	Evaluated Value
Command	CreateRect...		
Coordinate System	Global		
Position	-0.5 ,0 ,0	mm	-0.5mm , 0mm , 0mm
Axis	Z		
XSize	1	mm	1mm
YSize	10	mm	10mm

Figure 34. Rectangle properties

4 Click **OK**.

The **Properties** dialog box appears.

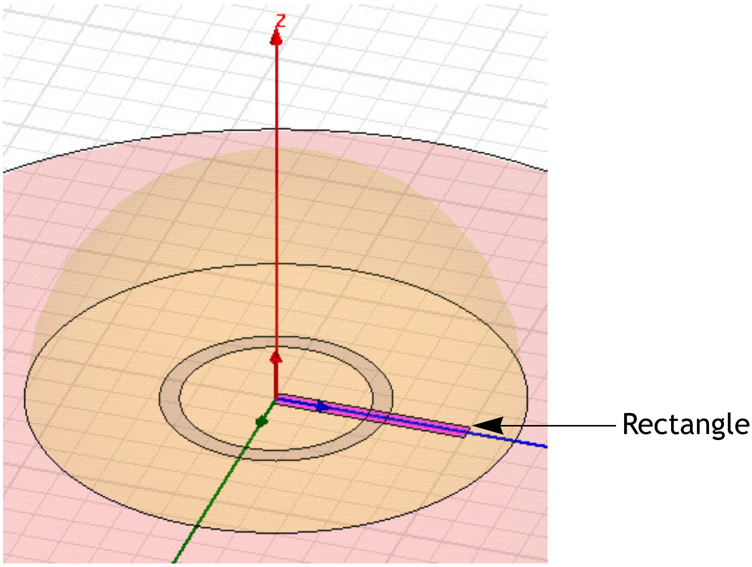


Figure 35. The Rectangle drawn

Intersect Rectangle and Annular Feed Ring

Next, you will intersect the rectangle and the annular feed ring to produce the antenna's feed gap.

1 Click **Tools > Options > Modeler Options.**

The **3D Modeler Options** dialog box appears.

2 Click the **Operation** tab.

3 Under **Clone**, select **Clone tool objects before intersecting**, and then click **OK** to activate.

Note This option instructs HFSS to always keep a copy of the original objects that intersect the first object selected.

4 Select **Rectangle1**.

5 Press and hold down **Ctrl** and select **annular_rng**.

6 Click **Modeler>Boolean>Intersect**.

The feed gap is produced without deleting the annular slot it is intended to feed.

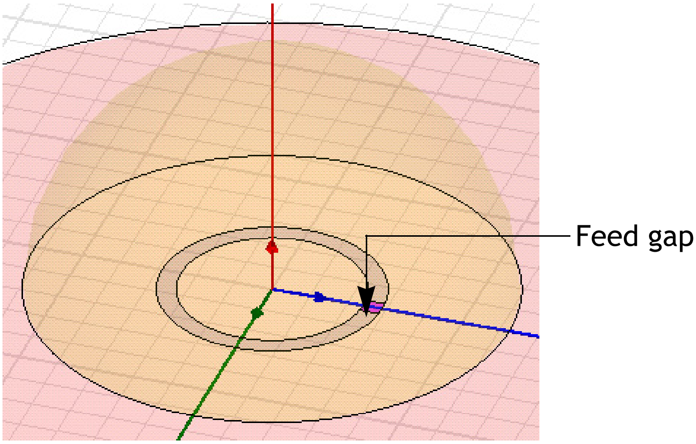


Figure 36. Feed gap drawn

7 Double-click **Rectangle1.**

The **Attribute** dialog box appears.

8 Type **gap** to rename the rectangle.

9 Press **Enter**.

10 Click **Edit** in the **Color** row.

The **Color Palette** appears.

11 Select green and set RGB as *0, 255, 0*.

12 Set **Transparency** to 0.

13 Click **OK**.

14 Deselect **gap** to view the resulting color and transparency assignments.

Create the Air Volume

To analyze radiation effects, you must create a virtual object that represents the radiation boundary. For this antenna model, you will create a radiation-transparent air volume surface sufficiently far from the model.

Next, you will draw a regular polyhedron with 18 segments to represent this virtual object. Finally, you will assign a radiation boundary before you analyze.

Draw the Polyhedron

1 Click **Draw>Regular Polyhedron**.

2 Draw the polyhedron freehand on the modeler window.

The **Segment number** dialog box appears.

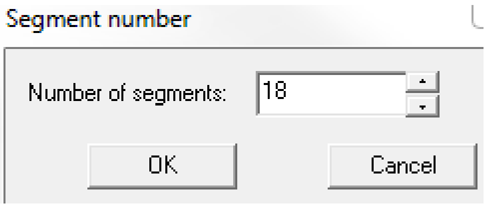


Figure 37. Segment number dialog box

3 Set **Number of Segments** as *18*.

The **Properties** dialog box appears.

Name	Value	Unit	Evaluated Value
Command	CreateRegularPolyhedron		
Coordinate System	Global		
Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
Start Position	30 ,0 ,0	mm	30mm , 0mm , 0mm
Axis	Z		
Height	35	mm	35mm
Number of Segments	18		18

Figure 38. Command tab for the Polyhedron

4 Enter the values as shown in Figure 30.

5 Click **Attribute** and rename Polyhedron1 to *airvol*.

6 Press **Enter**.

7 Click **Edit** in the **Color** row and set RGB to *0, 255, 255*.

8 Set **Transparency** to *0.01*.

9 Move the slider to the right in the **Set Transparency** window and stop at the 2nd mark to set the level at **.01**.

10 Set “vacuum” in the **Material** row.

11 Deselect **airvol** to view the resulting color and transparency assignments.

12 **Save** the project.

The completed **airvol** object appears .

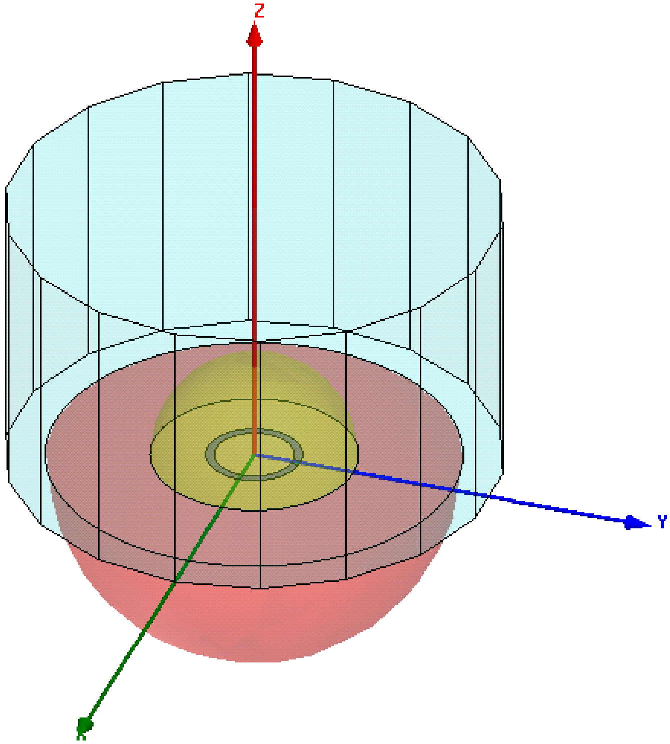


Figure 39. Airvol

Split the Model for Symmetry

This model as constructed is symmetrical about the yz plane. Now,you will split the model along the yz plane for symmetry.

1 Click **Edit>Select All** to select all the objects of the model.

2 Click **Modeler>Boolean>Split**.

The **Split** window appears.

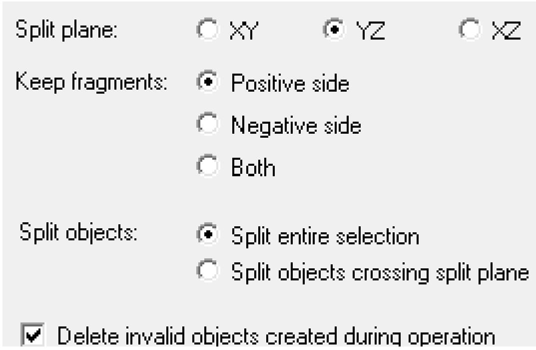


Figure 40. Split dialog box

3 Select **YZ** as the split plane and **Positive side** as the **keep fragments.**

4 Click **OK** to split the entire model.

Your final model should appear similar to the one shown below:

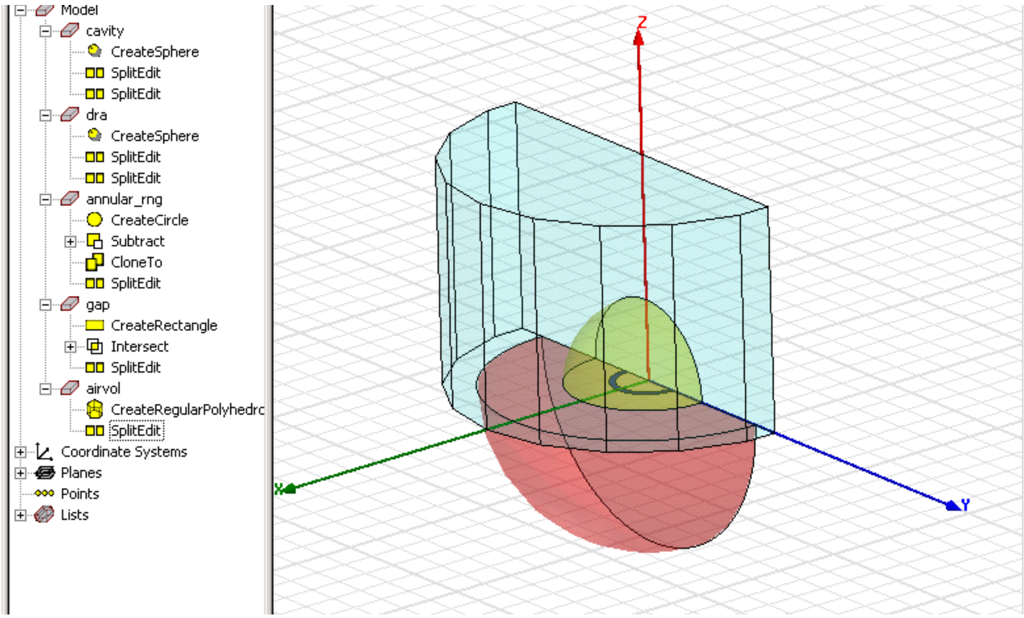


Figure 41.

5 Save the project.

You are now ready to assign ports and boundaries to your antenna model.

4 Setting Up the Problem

Now that you created the geometry and assigned all materials for the antenna problem, you are ready to define its ports and boundaries.

This chapter contains the following topics:

- Overview of the Interface
- Launch HFSS and Set-up the Project
- Set the Units Of Measurement
- Verify HFSS and Modeler Options
- Select the Solution Type

This chapter contains the following topics:

- Define the boundary conditions, such as the location of a radiation boundary and the symmetry plane.
- Define the lumped port through which the signal (voltage) enters the antenna.
- Verify that you correctly assigned the boundaries and excitations to the model.

Now you are ready to set up the problem.

Set Up Boundaries and Excitations

You must define the boundary and excitation conditions that include the excitation signals entering the structure, the behavior of electric and magnetic fields at various surfaces in the model, and any special surface characteristics.

Boundary Conditions

Boundaries specify the behavior of magnetic and electric fields at various surfaces. They can also be used to identify special surfaces — such as resistors — whose characteristics differ from the default.

The following four types of boundary conditions will be used for this antenna problem:

Radiation This type of boundary simulates 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. In this antenna model, the air volume object is defined as a radiation boundary.

Perfect E This type of boundary models a perfectly conducting surface in a structure, which forces the electric field to be normal to the surface. In this antenna model, the *bottom* face of the air volume object is defined as a perfect E boundary.

Perfect H This type of boundary forces the tangential component of the H-field to be the same on both sides of the boundary. In this antenna model, the annular feed ring is the aperture that is assigned this boundary. Because the aperture is defined as a perfect H boundary, the E-fields will radiate through it. If it was not defined as a perfect H boundary, the E-field would not radiate through and the signal would terminate at the aperture.

Symmetry In structures that have an electromagnetic plane of symmetry, such as this antenna model, the problem can be simplified by modeling only one-half of the model and identifying the exposed surface as a perfect H or perfect E boundary. For this antenna problem, a perfect H symmetry boundary is used.

Excitation Conditions

Ports define surfaces exposed to non-existent materials (generally the background or materials defined to be perfect conductors) through which excitation signals enter and leave the structure. One lumped port will be defined for this antenna problem. 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. A lumped port can be defined as a rectangle from the edge of the trace to the ground, as in this antenna problem, or as a traditional wave port. The default boundary is perfect H on all edges that do not come in contact with the metal.

Assigning Boundaries

First, you will assign all boundary conditions to the model.

For information on the types of boundaries you will assign, see “Boundary Conditions” on page 4-2.

Assign a Radiation Boundary to the Air Volume

The first boundary you will assign is a radiation boundary to the air volume object.

Radiation boundary model surfaces represent open space. Energy is allowed to radiate from these boundaries instead of being contained within them. 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.

To assign a radiation boundary to the air volume object:

1. Select **airvol**from the history tree.
2. Click **HFSS**>**Boundaries>Assign>Radiation**.

   The **Radiation Boundary** dialog box appears.
3. Verify that **Name** = *Rad1*.
4. Click **OK.**

   The radiation boundary gets applied.

   Note For this antenna problem, it is not necessary to edit any boundary's visualization default settings.

Hint To edit a boundary's visualization settings:

1. Click **HFSS>Boundaries>Visualization** if you want to show or hide boundaries. The **Boundary Visualization Options** window appears.
2. Clear the **View Geometry**, **View Name**, or **View Vector** check boxes of boundaries that you want to hide from view. Select the options you want to show in the **3D Modeler** window.
3. Click **OK** to apply the new settings.

The resulting radiation boundary is applied to the object **airvol** and now appears as a subentry of **Boundaries** in the Project Tree.

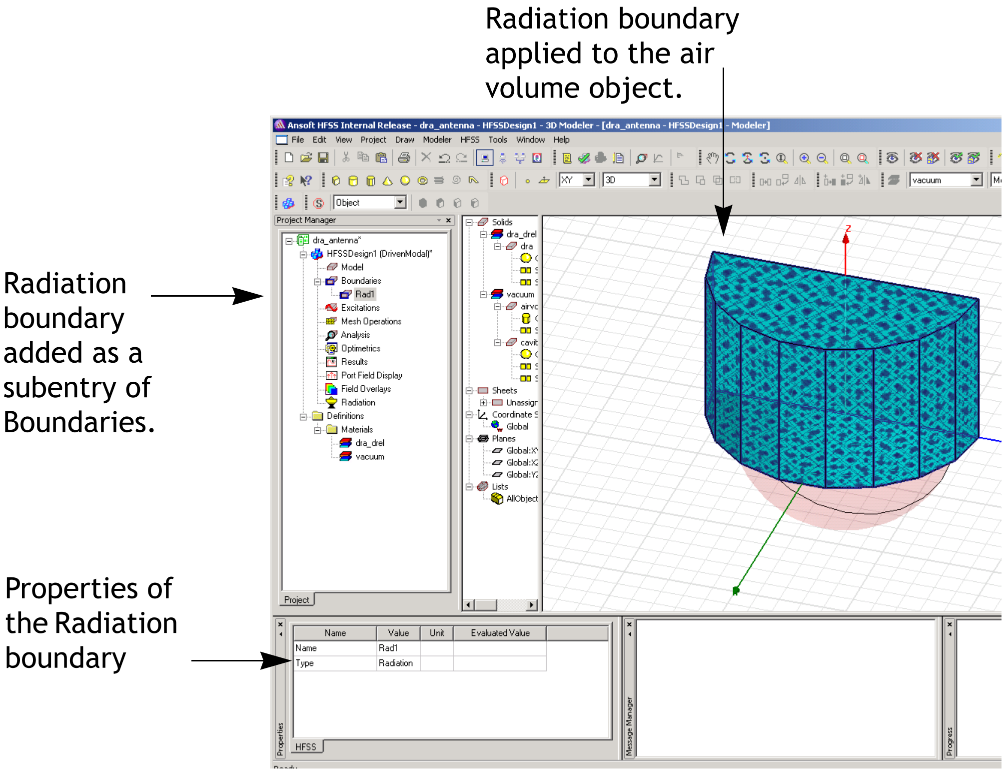


Figure 42.

Assign a Perfect E Boundary to the Air Volume

Next, define the intersection between the cavity and the air volume as a perfect E boundary condition. Therefore, you will assign a perfect E boundary to the *bottom* face of the air volume object, which will be the ground plane of the antenna.

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.

To assign a perfect E boundary to the bottom face of the air volume object:

1 Deselect the radiation boundary you just assigned.

2 Hit **F** on the keyboard to enter Face Selection mode.

3 Select the *bottom* face of the object **airvol** as follows:

- Press and hold down **Alt** and drag the mouse to rotate the model to a position where you can click the bottom face of the object **airvol**.
- Select the back face where the polyhedron was split.
- Hit **B** and the bottom face will get selected.

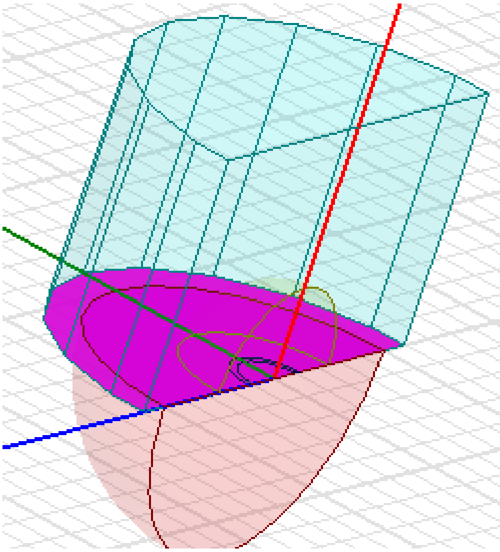


Figure 43. The bottom face selected

4 Click **HFSS**>**Boundaries>Assign>Perfect E**.

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

5 Uncheck **Infinite Ground Plane**.

Note If selected, the option simulates the effects of an infinite ground plane. It 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.

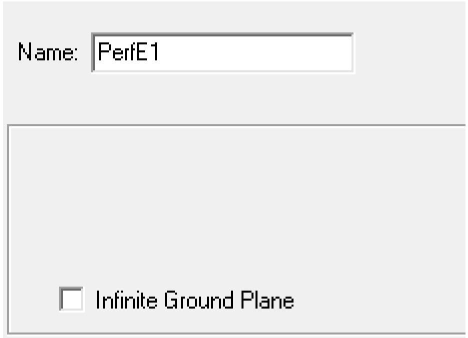


Figure 44. Perfect E Boundary

6 Click **OK.**

The perfect E boundary is applied.

The resulting perfect E boundary condition is assigned to the bottom face of the object **airvol**.

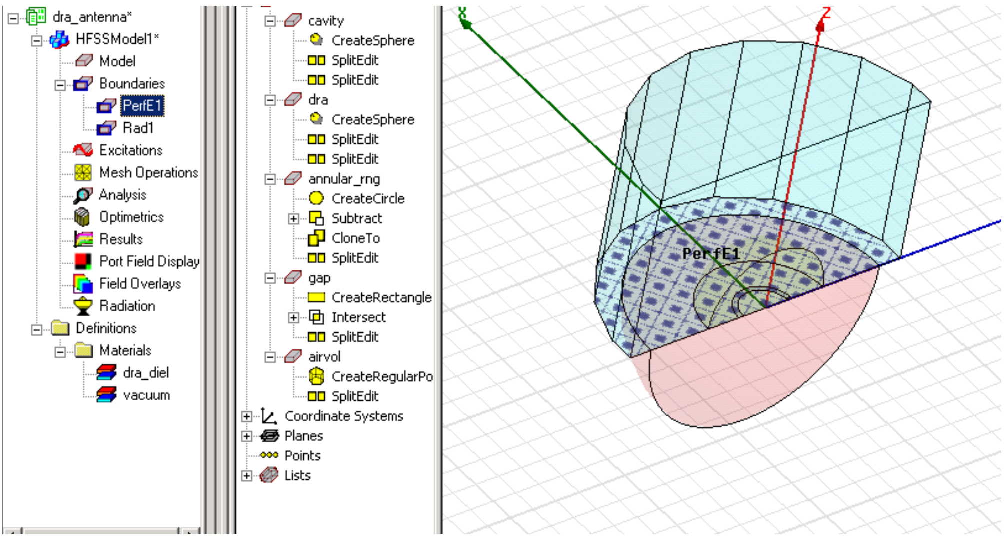


Figure 45. Perfect E-boundary applied

Assign a Perfect H to the Annular Feed Ring

The next boundary you will assign is a perfect H on the annular ring portion of the perfect E you just assigned to the bottom face of the air volume object. This perfect H boundary assignment will supersede the annular ring area from the prior Perfect E definition.

A perfect H boundary represents a surface on which the tangential component of the H-field is the same on both sides. For internal planes, such as the annular ring in this antenna model, this results in a natural boundary through which the field propagates. For planes 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.

To assign a perfect H boundary to the face of annular ring:

1. Deselect the perfect E boundary you just assigned, if it is still selected.
2. Hit **F** to enter Face Selection mode.
3. Click **annular_rng** on the History tree.
4. Click **HFSS**>**Boundaries**>**Assign**>**Perfect H**.

   The **Perfect H Boundary** dialog box appears.
5. Click **OK**.

   The perfect **H** boundary gets assigned.

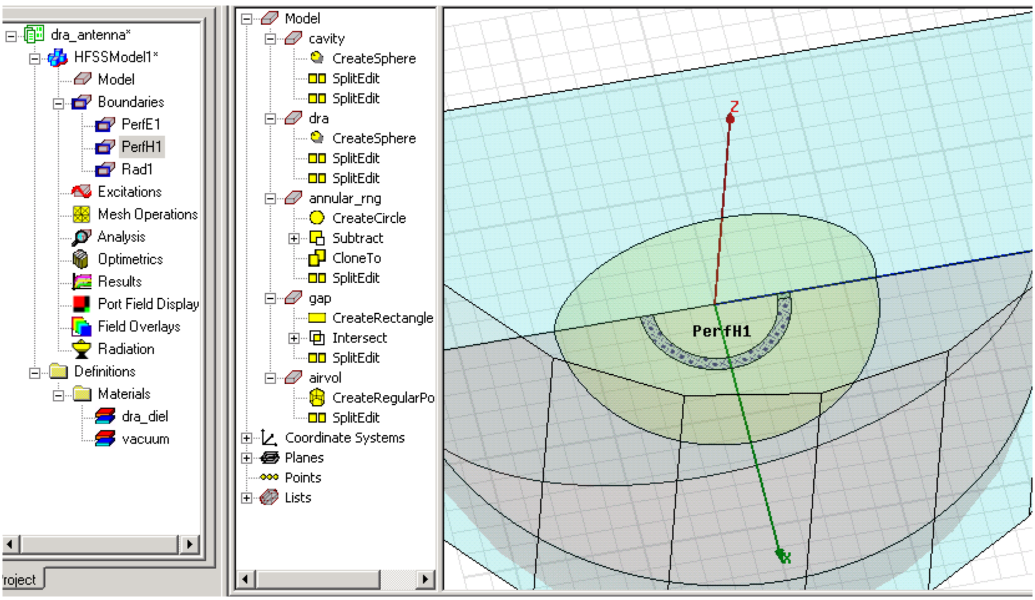


Figure 46. Perfect H Boundary assigned

Assign a Symmetry Boundary to the Model

HFSS has a boundary condition specifically for symmetry planes. Instead of defining a perfect E or perfect H boundary, you define a perfect E or perfect H symmetry plane.

When you are defining a symmetry plane, you must decide which type of symmetry boundary should be used, a perfect E or a perfect H. In general, use the following guidelines to decide which type of symmetry plane to use:

- 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 two-port rectangular waveguide shown below illustrates the differences between the two types of symmetry planes. 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.

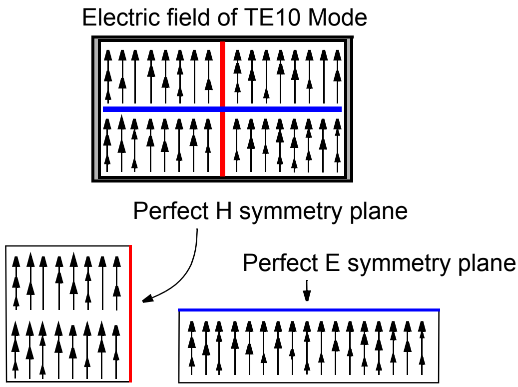


Figure 47. Field and Symmetry planes

Since the antenna model in this guide has a vertical plane of symmetry and the E-field is tangential to the surface, use a perfect H boundary for the symmetry plane.

Next, you will assign a perfect H symmetry boundary to the symmetry cut faces of the objects **airvol** and **cavity** (the model's symmetry plane).

To assign a perfect H symmetry boundary to the model's symmetry plane:

1. Deselect the perfect H boundary you just assigned.
2. Hit **F** to enter face selection mode.
3. Click **airvol** and **cavity** from the History Tree.
4. Rotate the model and select the symmetry cut faces as shown in Figure 7.

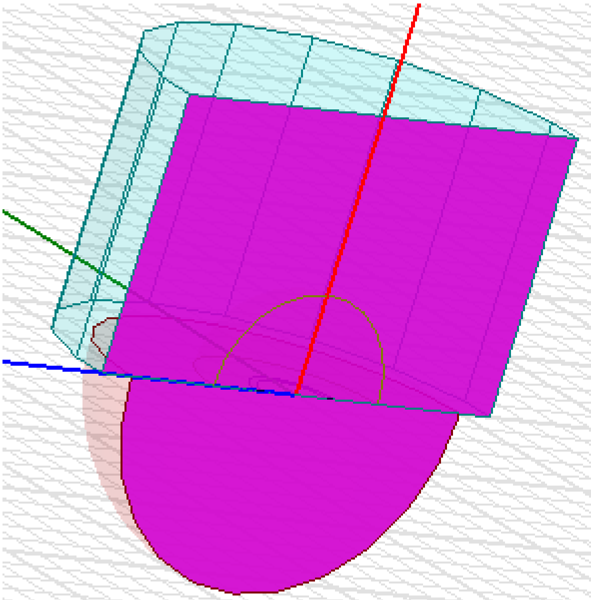


Figure 48. Symmtery cut faces selected.

5. Click **HFSS**>**Boundaries>Assign>Symmetry.**

   The **Symmetry Boundary** dialog box appears.
6. Verify that the default name is **Sym1**.
7. Select **Perfect H** as the symmetry type, and click **OK**.

   The resulting perfect H symmetry boundary condition is assigned to the faces of the objects **airvol** and **cavity**, as shown below:

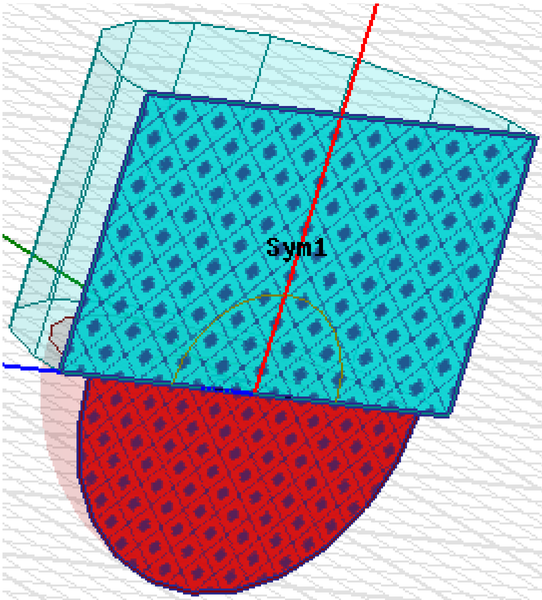


Figure 49. Symmetry boundary applied

Note At this juncture, you have finished assigning the boundaries and it is a good idea to save your project.

Assigning Excitations

You will now assign all excitation conditions to the model.

Assign a Lumped Port Across the Gap

For this antenna problem, the engineering focus is on the behavior of the antenna itself, not its feed. Therefore, the model will feed with a lumped port across the annular slot, or gap object. 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. A lumped port can be defined as a rectangle from the edge of the trace to the ground, as in this antenna problem, or as a traditional wave port. The default boundary is perfect H on all edges that do not come in contact with the metal.

Note The setup of a lumped port varies slightly depending on whether the solution is modal or terminal. As a reminder, the solution type for this antenna problem is modal driven.

To assign a lumped port across the gap object:

1 Deselect the perfect H symmetry boundary.

2 Click **gap** from the History Tree to select it. See Figure 9.

Note Recall **gap** is the rectangle you renamed. You should look under **Unassigned**.

3 Click **View>Interaction>Zoom In** to zoom in on the area where the gap object is located.

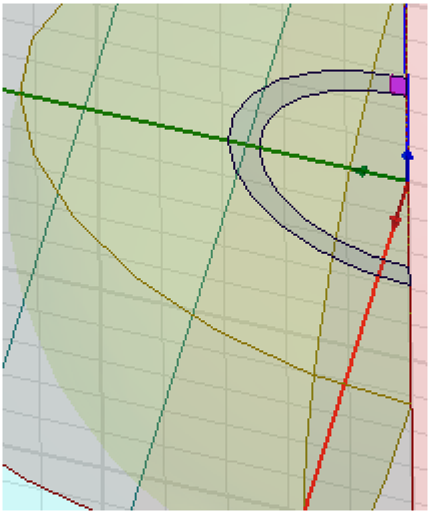


Figure 50. The gap is selected

4 Click **HFSS>Excitations>Assign>Lumped Port**.

The **Lumped Port** dialog box appears.

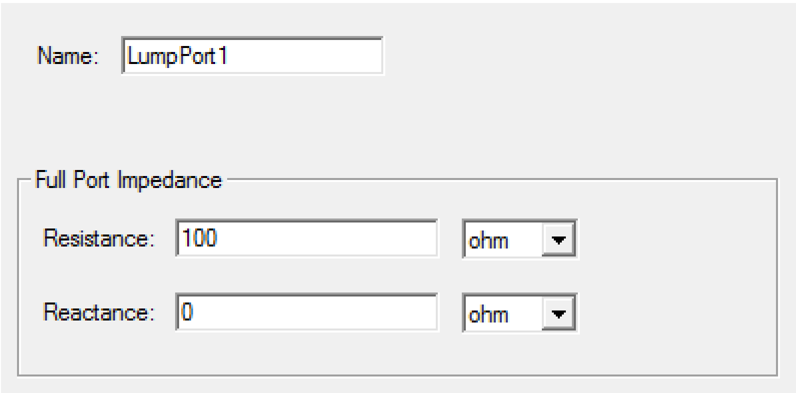


Figure 51. Lumped Port: General dialog box

Note The first time you assign a lumped port, HFSS walks you through the process with a step-by-step wizard.

5 Enter the values as shown in Figure 10.

6 Click **Next.**

7 In the **Lumped Port:Modes** step, click in the **Integration Line** list, and then select **New Line**.

The cursor changes to a dotted line and the Measure Data and Create Line dialog boxes appear.

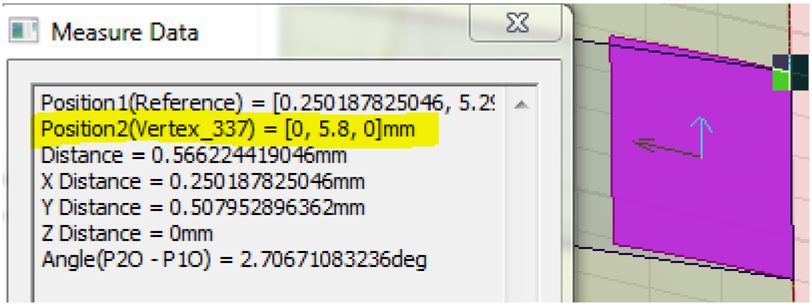


Figure 52. Measure Data for the first point of the Integration Line.

8 Click the start point (*0, 5.8, 0*) as shown in Figure 11.

Note Use the Meassure Data dialog box as reference as you move your cursor to reach *(0, 5.8, 0)*.

9 Click the end point (*0, 4.8, 0*) as shown in Figure 12.

The **Lumped Port** dialog box appears.

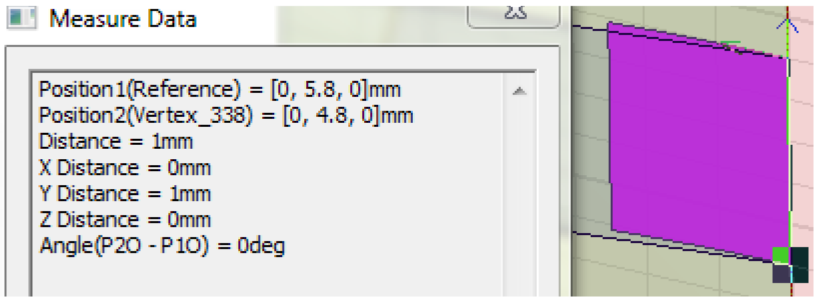


Figure 53.

Note The endpoint defines the direction and length of the integration line.

10 Click **Next.**

The **Post Processing** dialog box appears.

11 Enter *100* ohm for **Full Port Impedance** and click **Finish**.

The lumped port assignment is completed.

The resulting lumped port is assigned across the object **gap**, as shown below:

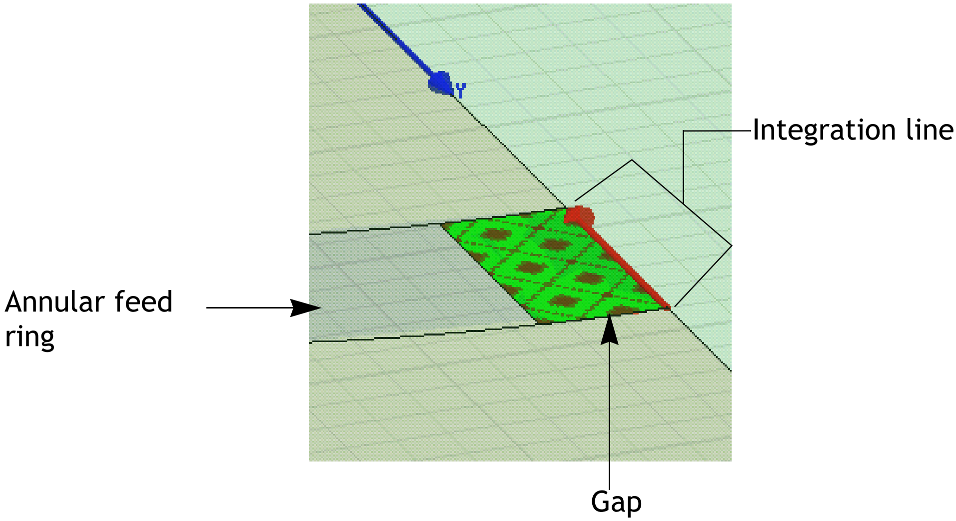


Figure 54. Lumped Port assigned

Modify the Impedance Multiplier

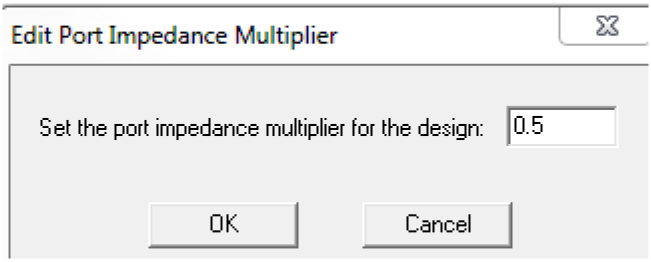
Because you defined a symmetry plane (allowing the model of a structure to be cut in half), the impedance computations must be adjusted by specifying an impedance multiplier.

In cases such as this antenna problem, 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.

To edit the impedance multiplier:

1 Click **HFSS>Excitations>Edit Port Impedance Multiplier.**

The **Edit Port Impedance Multiplier** dialog box appears.



2 Enter *0.5* in the **Impedance Multiplier** box.

3 Click **OK.**

Verify All Boundary and Excitation Assignments

Now that you have assigned all the necessary boundaries and excitations to the model, you should review their specific locations on the model in the solver view.

When you verify boundaries and excitations in the solver view, you review the locations of the boundaries and excitations as you have defined them for generating a solution (solving).

HFSS runs an initial mesh and determines the locations of the boundaries and excitations on the model.

Then, you can select a boundary or excitation from the Boundaries list in the **Project** window to view its highlighted area in the model.

To check the solver's view of boundaries and excitations:

1 Click **HFSS>Boundary Display (Solver View)**.

HFSS runs 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 in which they were assigned.

Name	Type	Solver Visibility	Visibility	Color
Rad1	User Defined	Visible to solver.	☐	black
PerfE1	User Defined	Visible to solver.	☑	red
PerfH1	User Defined	Visible to solver.	☐	green
Sym1	User Defined	Visible to solver.	☑	yellow
LumpPort1	User Defined	Visible to solver.	☐	blue
outer	Default	Visible to solver.	☐	magenta

Figure 55. Solver View of Boundaries dialog box

2 Select a check box in the **Visibility** column that corresponds with the boundary or excitaton for which you want to review its location on the model.

Note For example, Visibility is selected for **PerfE1** and **Sym1** and they will appear in the model in the color it has been assigned, as indicated in the **Color** column.

- **Visible to Solver** will appear in the **Solver Visibility** column for each boundary that is valid.

- **Overridden** will appear in the **Solver Visibility** column for each boundary or excitation that overwrites any existing boundary or excitation with which it overlaps.

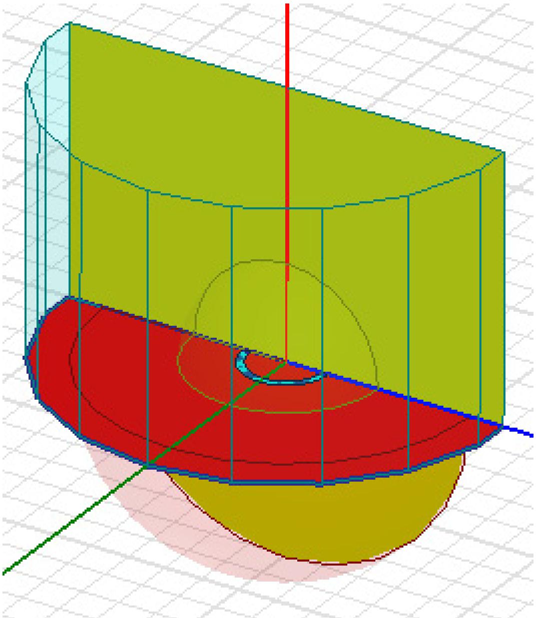


Figure 56. PerfE1 and Sym1 boundaries selected

3 Verify that the boundaries or excitations you assigned to the model are being displayed as you intended for solving purposes. See Figure 15.

4 Modify the parameters for those boundaries or excitations that are not being displayed as you intended.

5 Click **Close**, and then click **File>Save**, or click the **Save a project** button on the toolbar, to save the geometry.

You are now ready to set up the solution parameters for this antenna problem and generate a solution.

5 Generating a Solution

Now that you have created the geometry and set up the model, you are ready to generate a solution.

Your goals for this chapter are to:

- Set up the solution parameters that will be used in calculating the solution.
- Define meshing instructions.
- Validate the project setup.
- Generate a solution.
- View the solution data, such as convergence and matrix data information.

Add Solution Set-up

Now, you will specify how HFSS will compute the solution by adding a solution setup to the antenna project's design.

To add a solution setup to the design:

1 Click **HFSS>Analysis>Add Solution Setup**.

The **Solution Setup** dialog box appears:

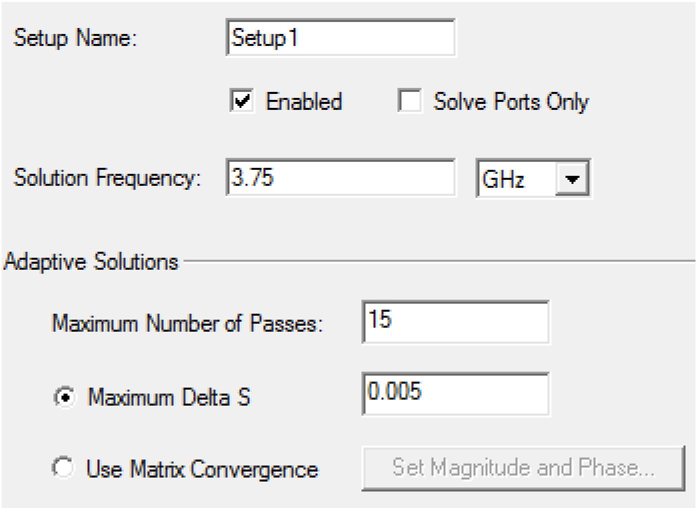


Figure 57. Solution Set-up

2 Click the **General** tab and enter values in the fields as shown in Figure1.

3 Click **Options**.

4 Set the fields as shown in Figure 2.

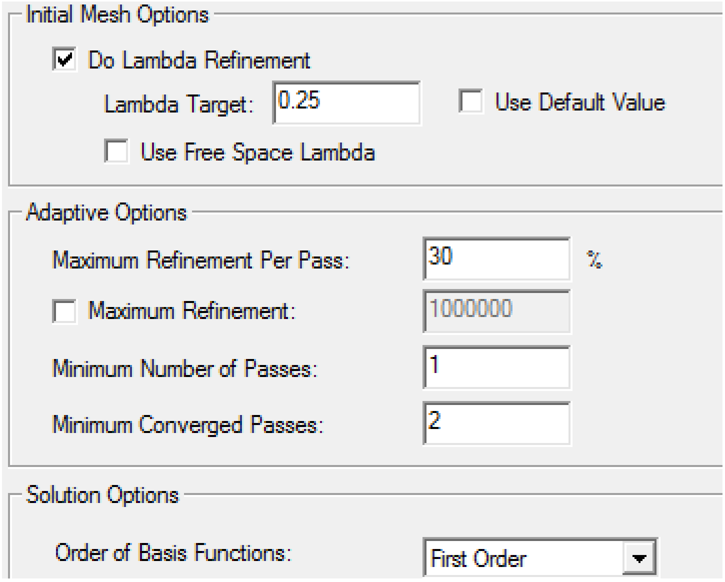


Figure 58. Options dialog box

5 Click **OK.**

Setup1 now appears as a solution setup under **Analysis** in the project tree.

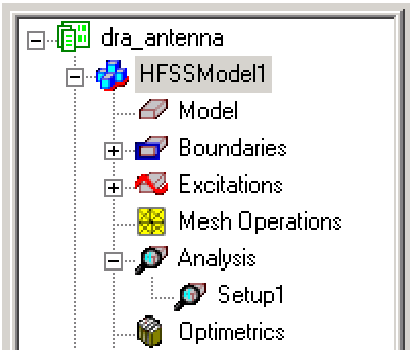


Figure 59. Project Tree

Add a Frequency Sweep to the Solution Setup

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.

For this antenna model, you will add a *Fast* frequency sweep to the solution setup. A Fast sweep generates a unique full-field solution for each division within a frequency range. It is best for models that will abruptly resonate or change operation in the frequency band, and obtains an accurate representation of the behavior near the resonance.

To add a fast frequency sweep:

1 Click **HFSS>Analysis Setup>Add Sweep**.

The **Select** window appears.

2 Select **Setup1** for the solution setup to which the sweep applies, and click **OK**.

The **Edit Sweep** dialog box appears.

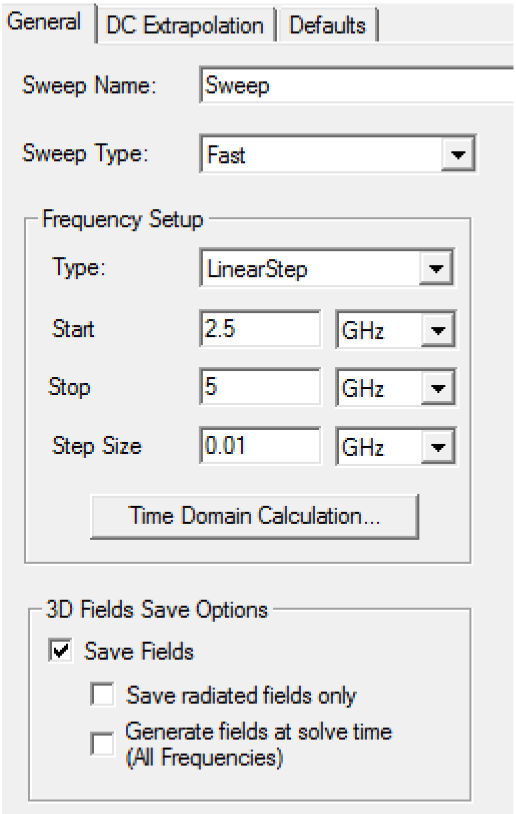


Figure 60. Edit Frequency Sweep

3 Enter the settings as shown in Figure 4.

Note If you do not save the field solution, the associated mode will not be available as a source stimulation during post processing.

4 Click **OK**.

Sweep1 now appears as a frequency sweep under **Setup1** in the project tree.

Note Click **Display** if you want to display each of the sweep values at the 0.01 GHz step size increment within the frequency range you specified.

Define Mesh Operations

In HFSS, mesh operations are optional mesh refinement settings that are specified before a mesh is generated. The technique of providing HFSS with mesh construction guidance is referred to as “seeding” the mesh.

Since the fields in the annular feed ring are very important in this antenna model, you will provide some meshing instructions on the faces of this object.

You will assign a length-based mesh refinement to the faces of the annular feed ring. Requesting length-based mesh refinement instructs 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 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 mesh is generated, the refinement criteria you specified will be used.

To assign a length-based mesh refinement to all the faces of the annular feed ring:

1 Select **annular_rng** from the History Tree.

You may need to expand the hierarchy under Sheets, then again under Perfect H to select the annular_rng.

2 On the **HFSS** menu, point to **Mesh Operations>Assign>On Selection**, and then click **Length Based**.

Note Applying the **On Selection** command refines *every* face on the annular feed ring.

The **Element Length Based Refinement** dialog box appears.

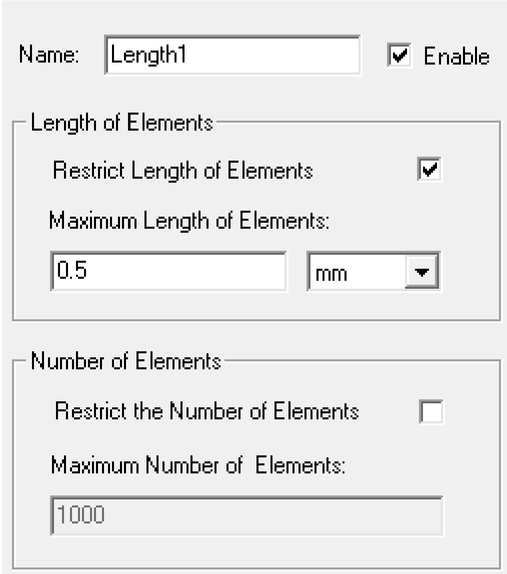


Figure 61. Element Length Based Refinement

3 Restrict the length of tetrahedra edges touching the faces:

- Select the **Restrict Length of Elements** check box.
- Enter **0.5 mm** in the **Maximum Length of Elements** text box as the maximum length of the tetrahedral elements touching the faces.

Note HFSS will refine the element edges touching the selected faces until they are equal to or less than this value.

4 Accept the default name **Length1**.

5 Clear **Restrict the Number of Elements**, if it is selected.

Note If selected, **Restrict the Number of Elements** restricts the number of elements added during refinement on the faces.

6 Click **OK**.

Length1 now appears as a mesh refinement under **Mesh Operation** in the project tree.

Validate the Project Setup

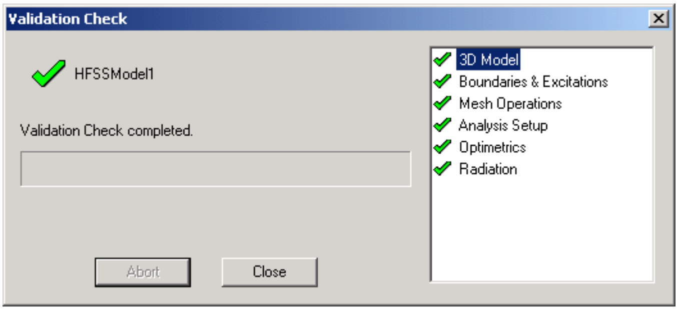
Before you run an analysis on the antenna model, it is important to first perform a validation check on the project. HFSS runs a check on all the setup details of the active project to verify that all the necessary steps have been completed and their parameters are reasonable.

To perform a validation check on the project **dra_antenna**:

1. Click **HFSS>Validation Check**, or click the **Validation Check** button on the toolbar.

   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.



   For this antenna project, a green check mark should appear next to each project step in the list.

3. Click **Close**.
4. Click **File>Save** to save any changes you have made to your project.

Generate the Solution

Now that you have entered all the appropriate solution criteria and defined the mesh operations, the antenna problem is ready to be solved.

When you set up the solution criteria, you specified values for an adaptive analysis. 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.

The following is the general process carried out during an adaptive analysis:

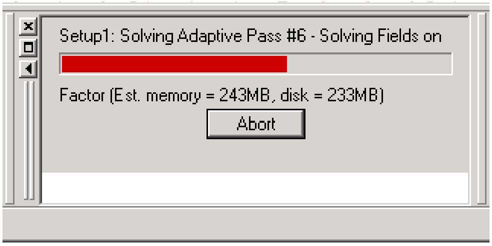
1. HFSS generates an initial mesh.
2. Using the initial mesh, HFSS computes the electromagnetic fields that exist inside the structure when it is excited at the solution frequency. (If you are running a frequency sweep, an adaptive solution is performed only at the specified solution frequency.)
3. Based 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. The software 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 complete.
6. If a frequency sweep is being performed, HFSS then solves the problem at the other frequency points without further refining the mesh.

To begin the solution process:

1. Select the **Setup1** solution setup in the project tree.
2. Click **HFSS>Analyze**. This command solves every solution setup in the design.

   HFSS computes the 3D field solution inside the structure.

The **Progress** window displays the solution progress as it occurs:



Note The results that you obtain should be approximately the same as the ones given in this section. However, there may be a slight variation between platforms.

View the Solution Data

While the analysis is running, you can view a variety of profile, convergence, and matrix data about the solution.

View the Profile Data

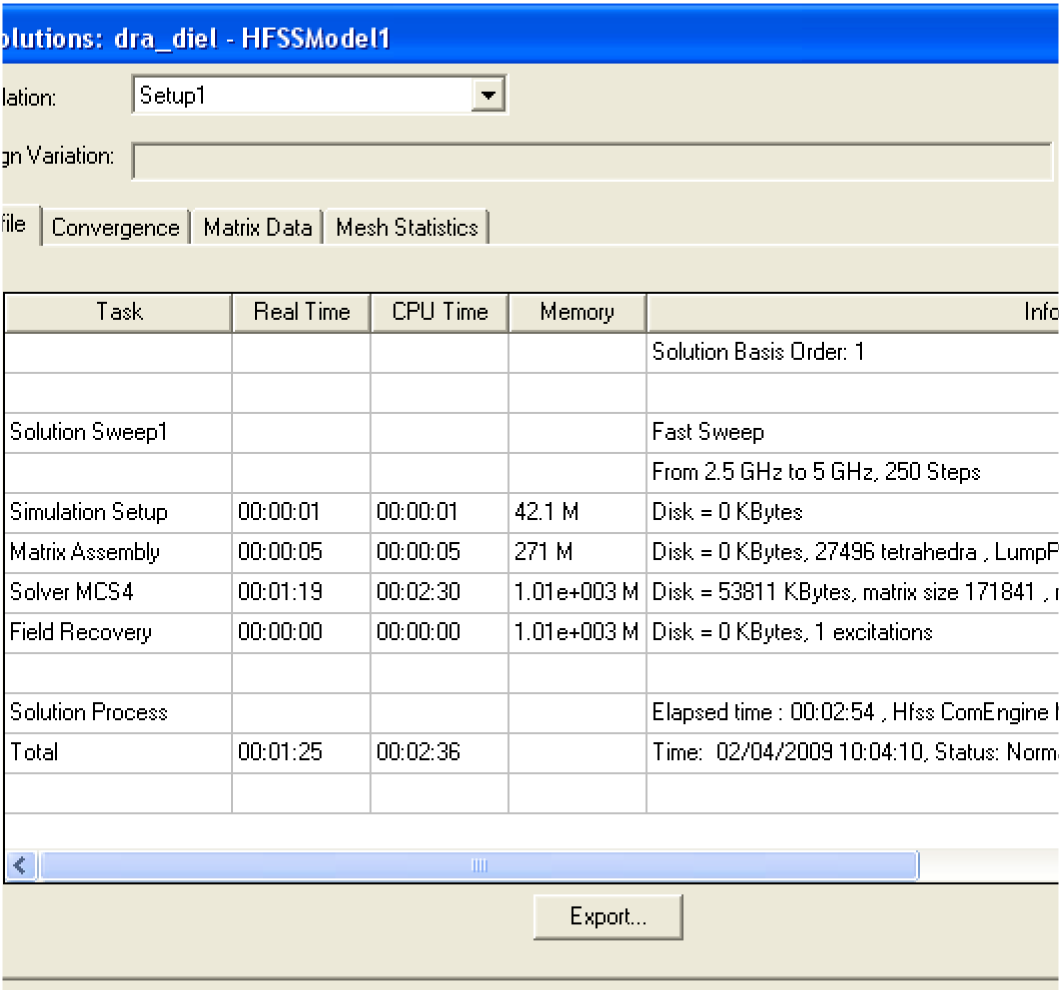
While the solution proceeds, examine the computing resources or profile data, that were used by HFSS during the analysis.

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.

To view the solution's profile data:

- Click **HFSS>Analysis Setup>Profile**.

  The **Solution Data** window appears. The figure shows the **Profile** tab selected.



Notice in the **Simulation** pull-down list that **Setup1** is selected as the solution setup. By default, the most recently solved solution is selected.

For the **Setup1** solution setup, you can view the following profile data:

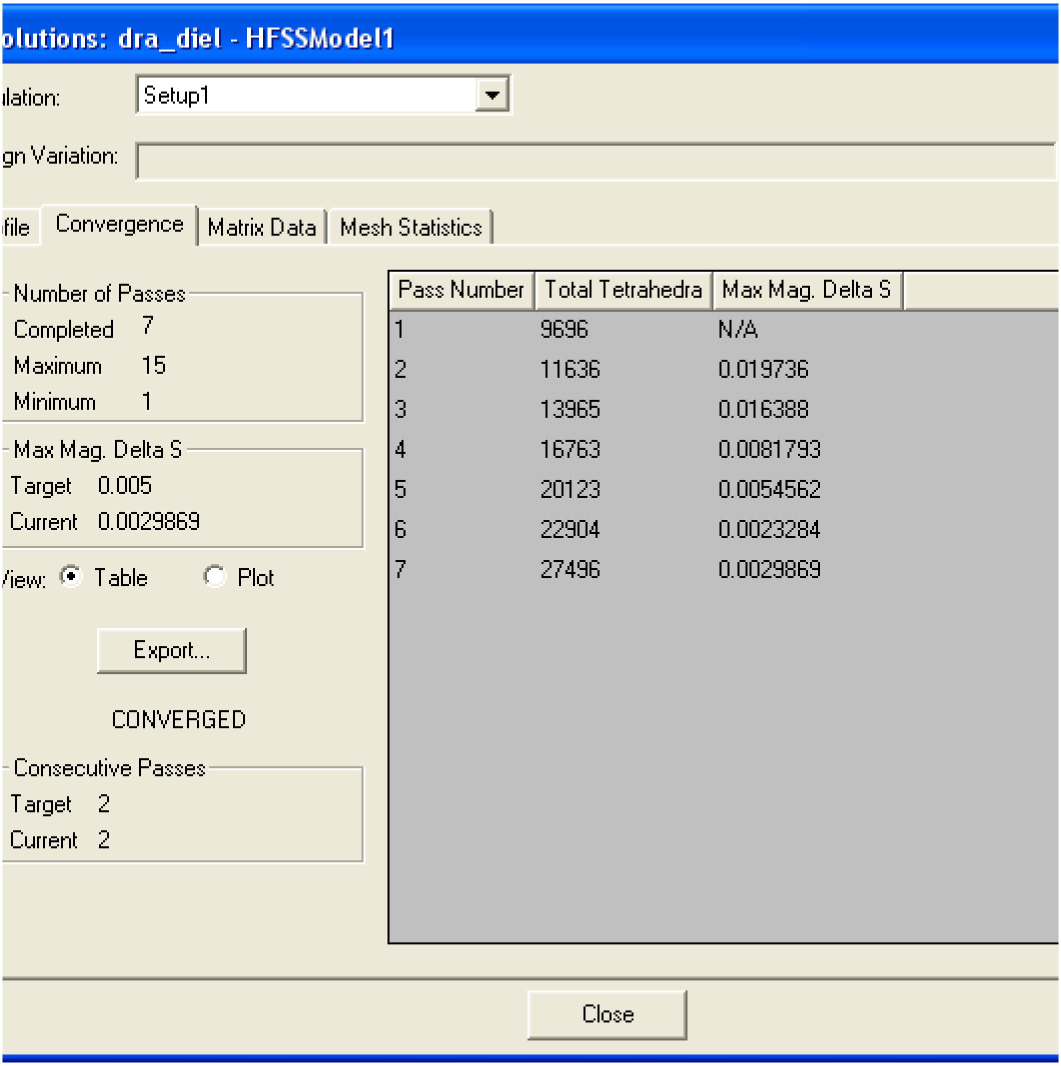
Task	Lists the software module that performed a task during the solution process, and the type of task that was performed. For example, for the task mesh3d_adapt, Mesh3d is the software module that adaptively refined the mesh.
Real Time	The amount of real time (clock time) required to perform the task.
CPU Time	The amount of CPU (Central Processing Unit) time required to perform the task.
Memory	The amount of RAM/virtual memory required of your machine to complete the task. This value includes the memory required of all applications running at the time, not just HFSS.
Information	The number of triangles, tetrahedra and matrices generated.

View Convergence Data

Next, while the solution proceeds, view the convergence data.

To view a solution’s convergence information:

- In the **Solution Data** window, click the **Convergence** tab.



Based on the criteria you specified for **Setup1**, you can view the following convergence data:

- Number of adaptive passes completed and remaining. 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.

- Number of tetrahedra created at each adaptive pass.
- Maximum magnitude of delta S between two passes.

For solutions with ports, as in **Setup1**, 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.

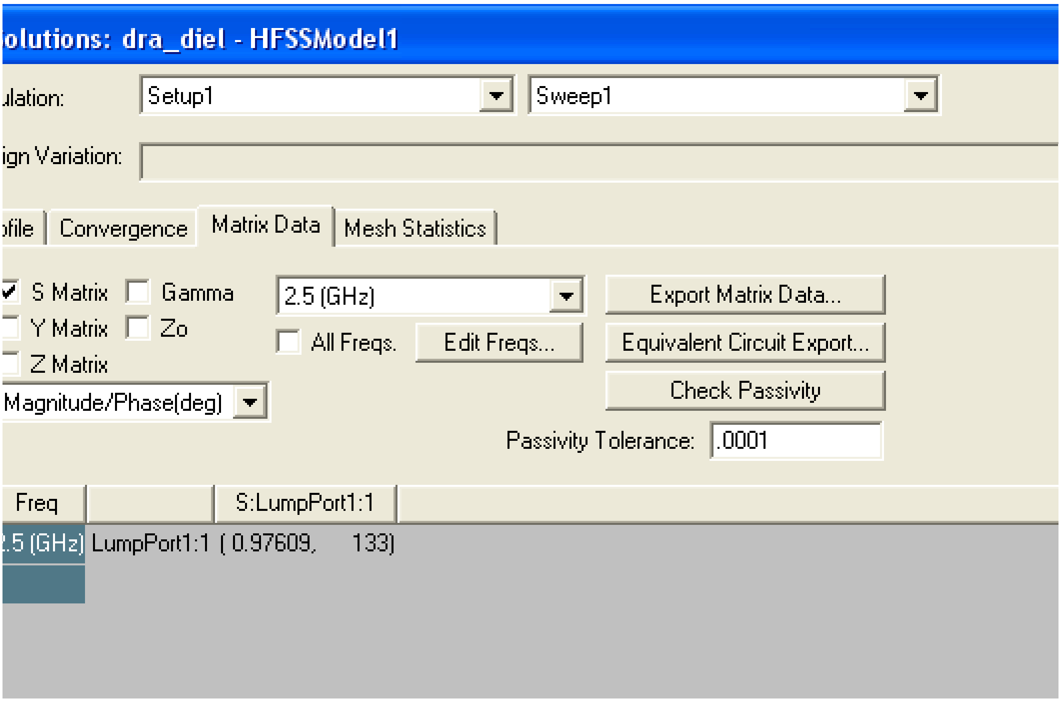
The convergence data can be displayed in table format or on a rectangular (X - Y) plot.

View Matrix Data

Next, view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive and sweep solution.

To view matrices:

1 In the **Solution Data** window, click the **Matrix Data** tab.



2 In the **Simulation** pull-down lists, do the following:

a. Verify that **Setup1** is selected as the solution setup for which you want to view matrices.

b. Verify that **LastAdaptive** is selected. This is the solved pass for which you want to view matrices.

3 Select **S Matrix** as the type of matrix data you want to view.

4 Select **Magnitude/Phase** from the pull-down list as the format in which to display the matrix information.

You can display matrix data in the following formats:

Magnitude/ Phase	Displays the magnitude and phase of the matrix type.
Real/ Imaginary	Displays the real and imaginary parts of the matrix type.
dB/Phase	Displays the magnitude in decibels and phase of the matrix type.
Phase	Displays the phase of the matrix type.
Real	Displays the real parts of the matrix type.
Magnitude	Displays the magnitude of the matrix type.
Imaginary	Displays the imaginary parts of the matrix type.
dB	Displays the magnitude in decibels of the matrix type.

5 Select the solved frequencies to display:

- To display the matrix entries for all solved frequencies, select **All Freqs**. It is selected be default.
- To show the matrix entries for one solved frequency, clear **All Freqs** 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.

The data is displayed in the table.

6 Click **Close** to close this window.

Once the simulation has run successfully, you will be ready to analyze the results, as described in the next chapter, “Analyzing the Solution.”

6 Analyzing the Solution

Now, HFSS has generated a solution for the antenna problem. In general, you can display and analyze the results of a project in many different ways. You can:

- Plot field overlays - representations of basic or derived field quantities - on surfaces or objects.
- Create 2D or 3D rectangular or circular plots and data tables 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.
- Delete solution data that you do not want to store.

For this antenna problem, you will specifically:

- ✔ Create Modal S-parameter reports.
- ✔ Create a field overlay plot of the magnitude of E on the top face of the antenna's cavity.
- ✔ Create an animation of the mag-E plot.

Time It should take you approximately 1 hour to work through this chapter.

Create Modal S-Parameters Reports

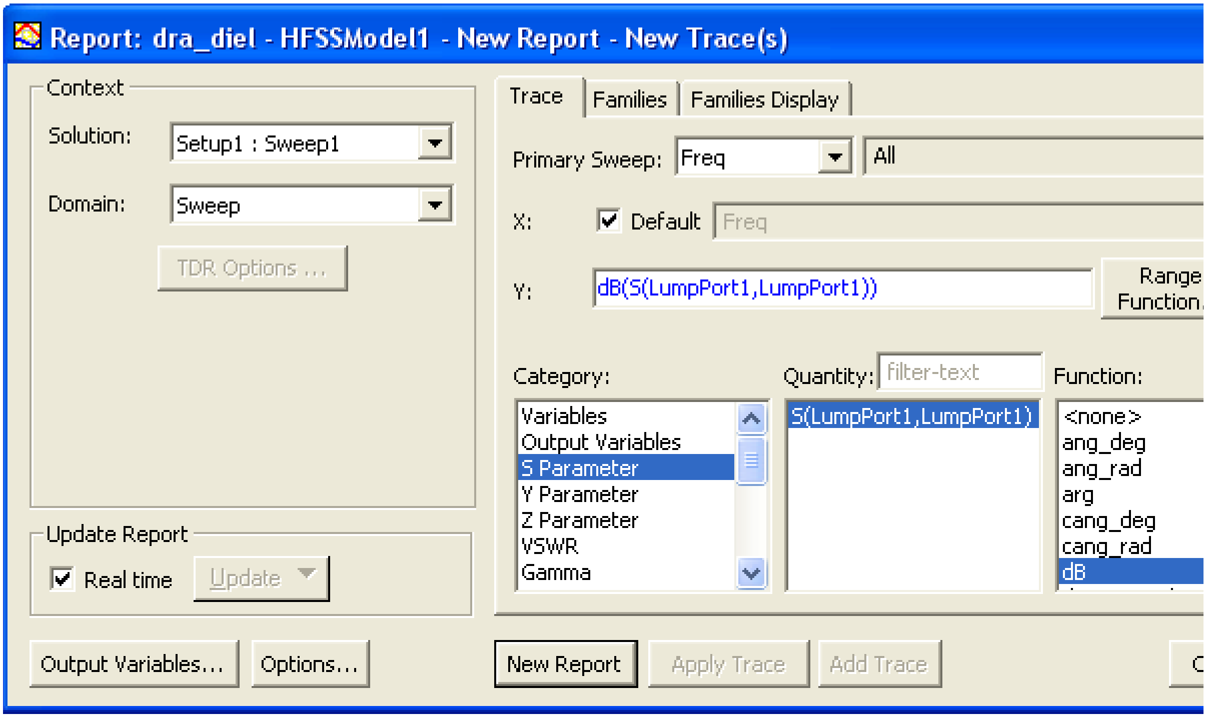
Now you are ready to create the modal S-parameters and Z-parameters reports.

Create an S-Parameters Report of S11

To generate a 2D report of S11:

1 Click **HFSS>Results>Create Modal Solution Data Report>Rectangular Plot**.

The **New Report** dialog box appears.



2 Verify that **Setup1: Sweep1** is selected in the **Solution** pull-down list.

3 Verify that **Sweep** is selected from the **Domain** pull-down list.

4 For the Primary Sweep, select the variable Freq.

This selection is plotted along the x-axis.

5 In the **Y** Component pane, specify the following information to plot

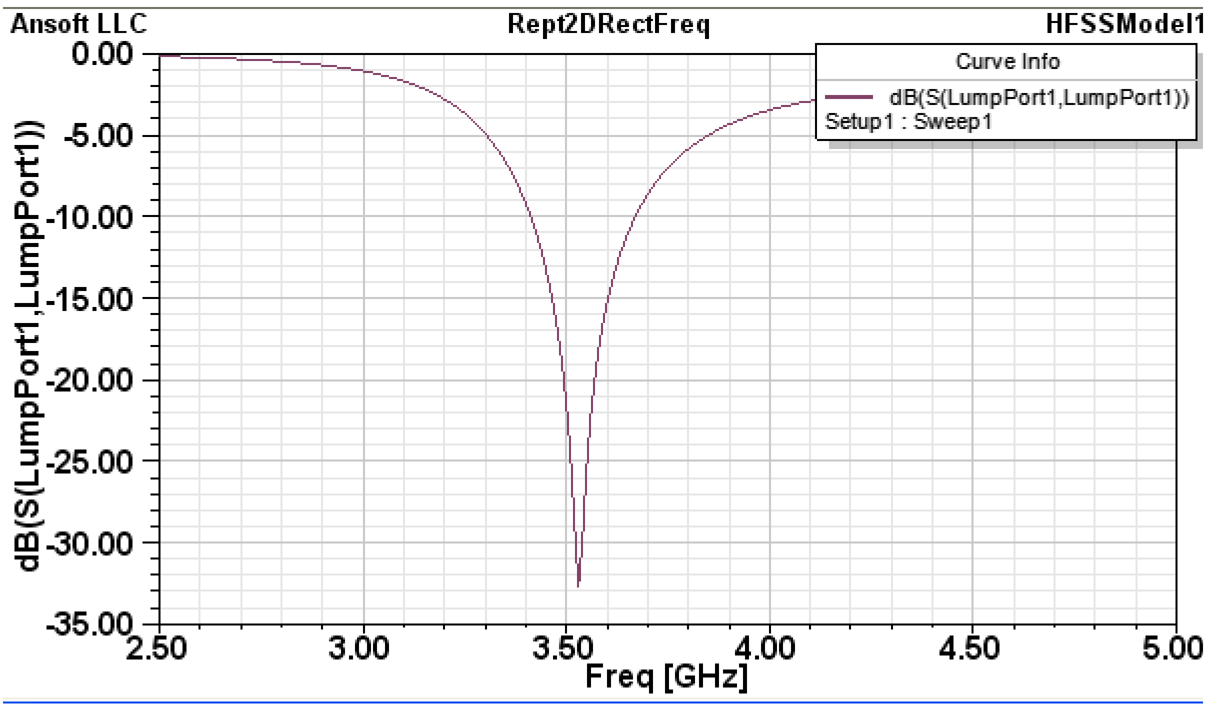
along the y-axis:

Category	**S Parameter** This is the type of information to plot.
Quantity	**S(LumpPort1,LumpPort1)** This is the value to plot.
Function	**dB** This is the mathematical function of the quantity to plot.

6 In the Families tab, the Sweeps radio button is selected.

7 Click **New Report**, and then click **Close**.

The report **XY Plot 1** appears in the **3D Modeler** window and is now listed under **Results** in the project tree. A trace icon is also listed under XY Plot 1. A trace is a single line that connects the data points on the graph.



Create an S-Parameters Report of Z11

To generate a 2D report of Z11 Real and Imaginary traces:

1 Click **HFSS>Results>Create Modal Solution Data Report>Rectangular Plot**.

The **Report** dialog box appears.

2 Verify that **Setup1: Sweep1** is selected from the **Solution** pull-down list.

3 Verify that **Sweep** is selected from the **Domain** pull-down list.

4 For the **X** field, the sweep variable Freq is selected.

This select plots the sweep variable selected along the x-axis.

5 In the **Y** pane, add the following two traces to plot along the y-axis:

a. Specify these values for the *real* trace to plot:

Category	**Z Parameter**
Quantity	**Z(LumpedPort1, LumpedPort1)**
Function	**re** This is the real part of the complex number.

b. Click **New Report**, but do not click Close.

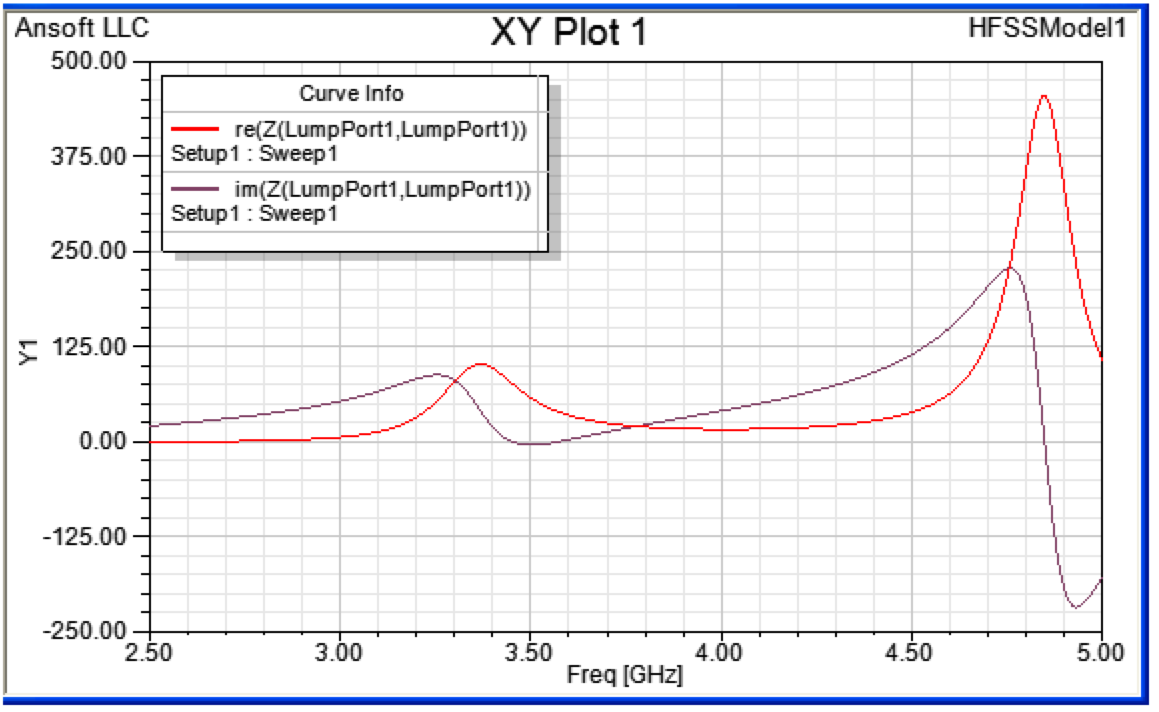
This displays the XY Plot 2 report with one trace and enables the **Update Trace** and **Add Trace** buttons in the **New Report** dialog.

c. In the **New Report** dialog, specify these values for the *imaginary* trace to plot:

Category	**Z Parameter**
Quantity	**Z(LumpedPort1, LumpedPort1)**
Function	**im** This is the imaginary part of the complex number.

6 Click **Add Trace**.

The Z-parameter trace is added to the **XY Plot 2** in the **3D Modeler** window and is now listed under the **Results** icon in the project tree.



Create Field Overlay Plots

Next, you will create a field overlay plot of the magnitude of E of the bottom face of the antenna's air volume object and examine the resulting E- field pattern.

Create a Mag E Field Overlay Plot

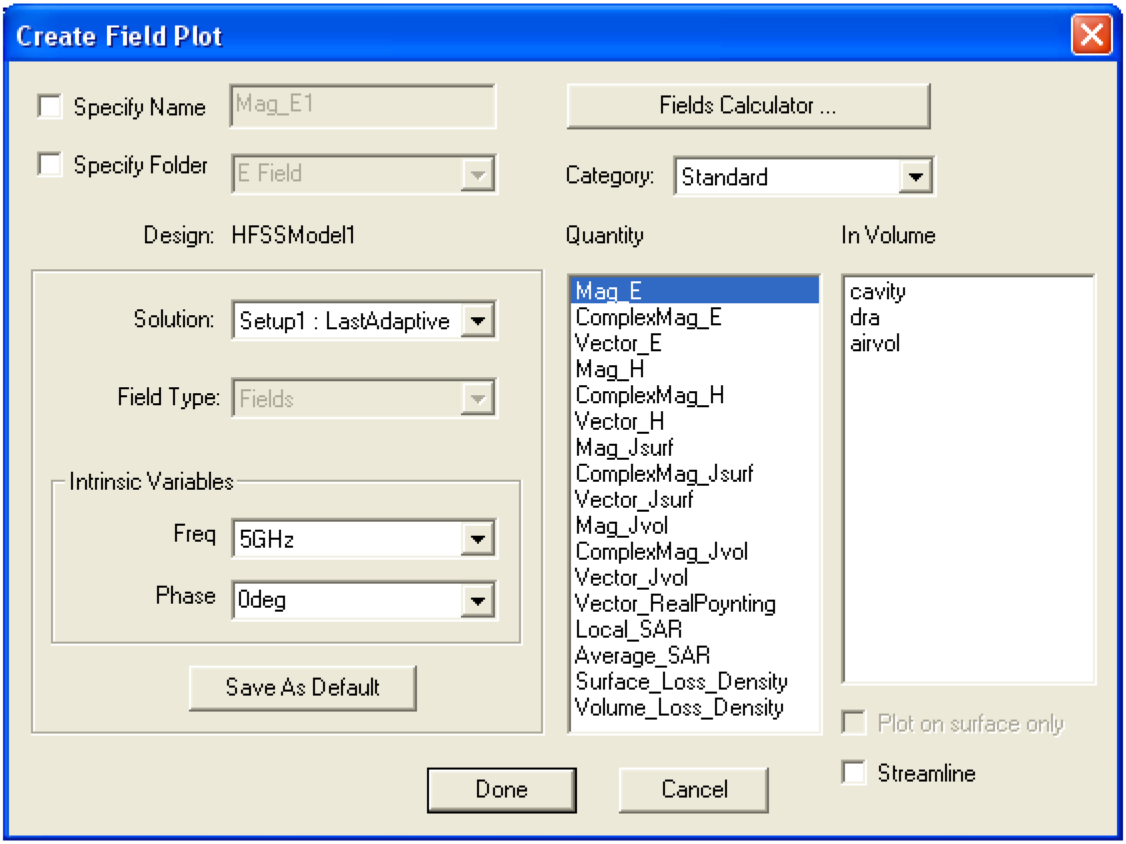
Now, you are ready to create a field overlay plot of the magnitude of E of the bottom face of the air volume object and examine the resulting E- field pattern.

1 In **Select Faces** mode, select the bottom face of the object **airvol**.

One way to do this is to click the cursor near the bottom of one of the rectangular facets, and then type "b" to select a face behind.

2 On the **HFSS** menu, click **Fields>Plot Fields>E>Mag_E**.

The **Create Field Plot** dialog box appears.



3 Select **Mag_E** from the **Quantity** list.

This selects the magnitude of the real part of the electric field |**E**|(x,y,z,t) as the quantity to plot.

4 Select **All** from the **In Volume** list to specify that HFSS will plot over the entire volume of the model.

5 Verify that **3.75GHz** is selected from the **Freq** pull-down list.

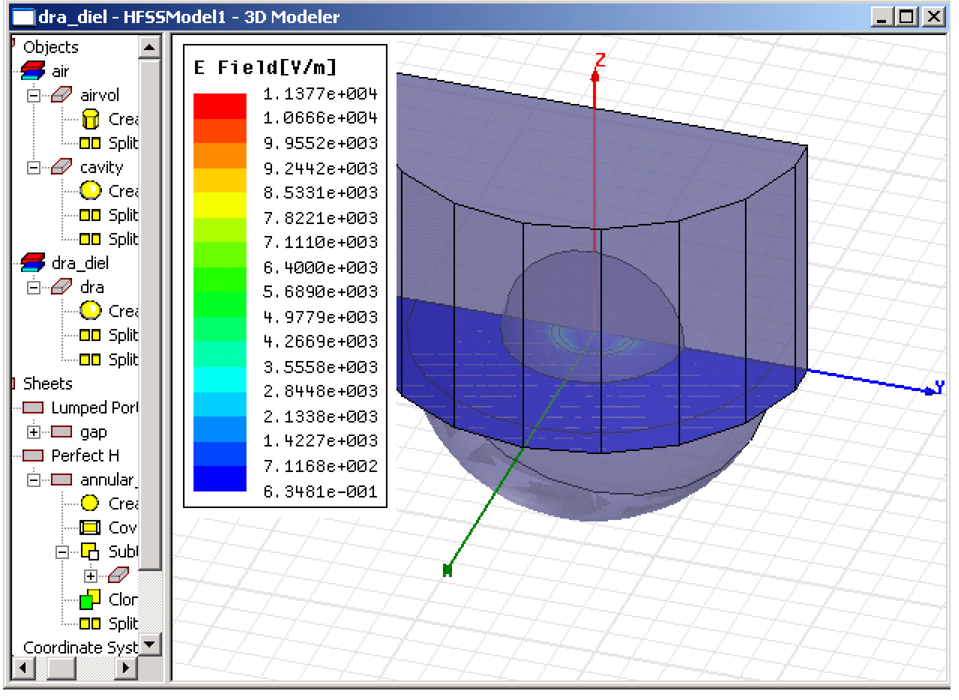
The **Freq** pull-down list includes a list of frequencies for which a field solution is available.

6 Verify that **0deg** is selected from the **Phase** pull-down list.

7 Click **Done**.

The **Mag_E1** field overlay cloud plot appears in the **3D Modeler** window and is now listed under **Field Overlays-E Field-Mag_E1** in the project tree.

Your field overlay plot should resemble the one shown below:



Modify the Mag E Plot's Attributes

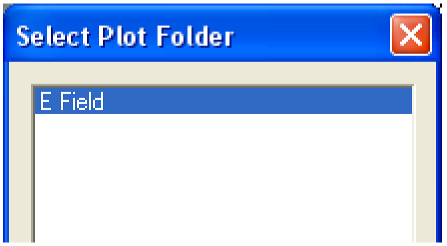
Now, you will modify the attributes of the **Mag_E1** field overlay plot you just created to prepare it for an effective animation.

1 Select **Mag_E1** in the project tree under Field Overlays-E Field.

This is the folder in which plot Mag_E1 is located.

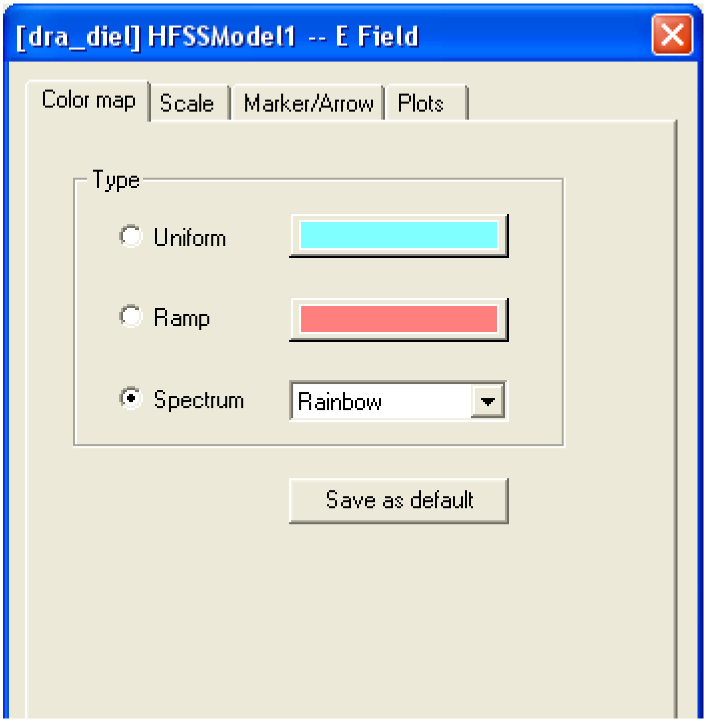
2 On the **HFSS** menu, point to **Fields**, and then click **Modify Attributes**.

The **Select Plot Folder** window appears.



3 Verify that **Mag_E1** is selected, and then click **OK**.

A dialog box appears, which displays the plot's attributes.



4 Click the **Color map** tab, and then specify the following settings:

Type	Select **Spectrum** and **Rainbow** from the pull-down list. Field quantities are plotted in multiple colors. Each field value is assigned a color from the selected spectrum.
Real time mode	Select this option. This option immediately applies changes to the plot's attributes.

5 Click the **Scale** tab, and then specify the following settings:

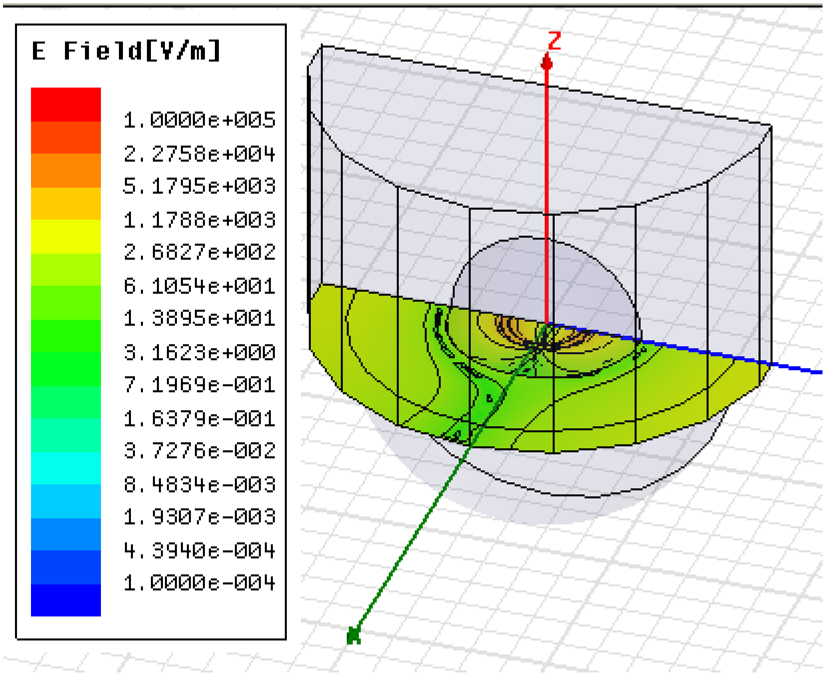
Use Limits	Select this option. Only the field values between the minimum and maximum values will be plotted. Field values below or above these values will be plotted in the colors assigned to the minimum or maximum limits, respectively.
Min	Enter **1e-4** (0.0001).
Max	Enter **1e5** (100000).
Linear/ Log	Select **Log.** Field values will be plotted on a logarithmic scale.

6 Click the **Plots** tab, and then specify the following settings:

IsoValType	Select **Tone** from the pull-down list. This isosurface display type varies color continuously between isovalues.
Outline	Select this option.
Map transp.	Clear this option, if it is selected. If selected, the transparency of field values increases as the solution values decrease.

7 Accept all the remaining default settings in this dialog box. With Real Time Mode selected, **Apply** is grayed out, and the changes apply immediately.

Your modified plot **Mag_E1** should resemble the one shown below:



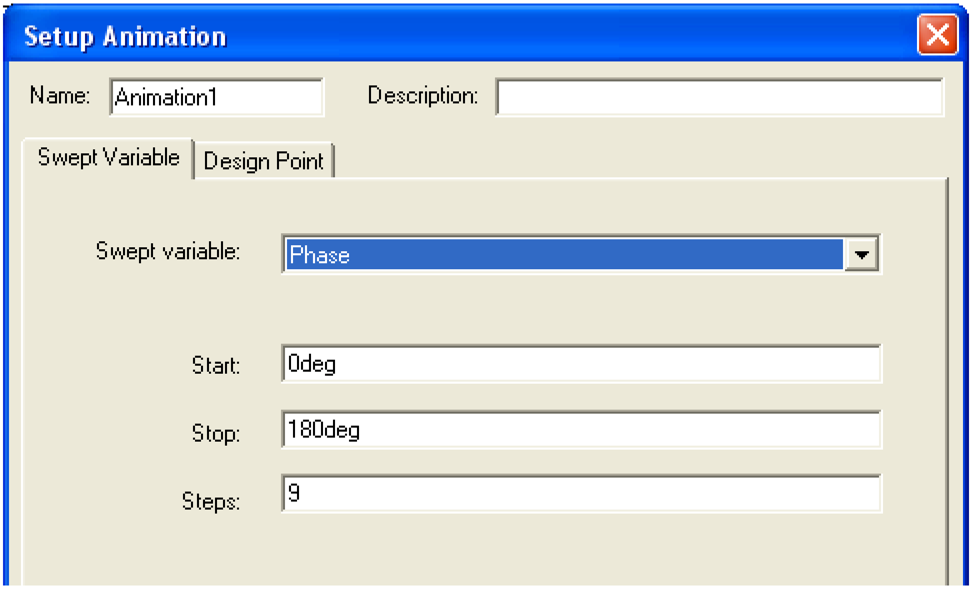
Create a Phase Animation of the Mag E Plot

Next, you will create an animation of the field overlay plot of the magnitude of E to examine a frame-by-frame, animated behavior of the plot.

To create a phase animation of the Mag E plot:

1. Select the **Mag_E1** field overlay plot from the project tree.
2. Click **HFSS>Fields>Animate**.

The **Setup Animation** dialog box appears.



3 Accept the default name **Animation1** in the **Name** text box.

4 Optionally, type a description of the animation in the **Description** text box.

5 Under the **Swept Variable** tab, select **Phase** from the **Swept Variable** list.

6 Accept the remaining default settings in the **Start**, **Stop**, and **Steps** boxes for the phase values of the animation.

If the Start value is 0, the Stop value is 160, and the number of steps is 9, the animation will display the plot at 9 phase values between 0 and 160. The start value will be the first frame displayed, resulting in a total of 10 frames in the animation.

7 Click **OK**.

The animation begins in the **3D Modeler** 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.

Index

Numbers

A

B

C

D

E

F

G

H

I

L

M

O

P

R

S

T

V

W

Z