

# Getting Started with HFSS: A Dielectric Resonator Antenna Problem



February 2009

The information contained in this document is subject to change without notice. Ansoft makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Ansoft shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

© 2009 Ansoft LLC. All rights reserved.

Ansoft LLC  
225 West Station Square Drive  
Suite 200  
Pittsburgh, PA 15219  
USA  
Phone: 412-261-3200  
Fax: 412-471-9427

HFSS and Optimetrics are registered trademarks or trademarks of Ansoft LLC. All other trademarks are the property of their respective owners.

New editions of this manual incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.


Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

Edition	Date	Software Version
1	Jan 2006	10.0
2	May 2007	11.0
3	February 2009	12.0

## 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 to type the word **copy**, to type a space, and then to type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, click **HFSS>Excitations>Assign>Wave Port**.
  - Labeled keys on the computer keyboard. For example, “Press **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**” means to type the word **copy**, to type a space, and then to type a file name.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press **Shift+F1**” means to press the **Shift** key and the **F1** key at the same time.
- Toolbar buttons serve as shortcuts for executing commands. Toolbar buttons are displayed after the command they execute. For example,
 

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

*Alternate methods or tips are listed in the left margin in blue italic text.*

## Getting Help

### Ansoft Technical Support

To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, <http://www.ansoft.com>, click the **Contact** button, and then click **Support**. Phone numbers and e-mail addresses for the technical support staff are listed. You can also contact your Ansoft account manager in order to obtain this information. All Ansoft software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, 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.
- **Seach** - 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

## Table of Contents

### 1. Introduction

The Sample Problem .....	1-2
General Procedure .....	1-3
Finite Element Method .....	1-4
Results for Analysis .....	1-5

### 2. Creating the New Project

Overview of the Interface .....	2-2
Create the New Project .....	2-5
Optionally, Add the New Project .....	2-5
Optionally, Insert an HFSS Design .....	2-5
Add Project Notes .....	2-6
Save the Project .....	2-6

### 3. Creating the Model

Select the Solution Type .....	3-2
Set Up the Drawing Region .....	3-3
Overview of the 3D Modeler Window .....	3-3
Coordinate System Settings .....	3-4
Units Settings .....	3-4
Grid Settings .....	3-4
Create the Geometry .....	3-6

---

Draw the Cavity . . . . .	3-6
Draw the Sphere . . . . .	3-6
Rename the Sphere . . . . .	3-7
Split the Cavity . . . . .	3-7
Modify the Cavity's Attributes . . . . .	3-8
Draw the Sphere . . . . .	3-11
Rename the Sphere . . . . .	3-11
Split the DRA . . . . .	3-12
Modify DRA's Attributes . . . . .	3-12
Create the Annular Feed Ring . . . . .	3-16
Draw Circle1 . . . . .	3-17
Draw Circle2 . . . . .	3-17
Subtract Circle1 from Circle2 . . . . .	3-18
Rename Circle2 . . . . .	3-19
Modify the Annular Feed Ring's Attributes . . . . .	3-19
Draw the Feed Gap . . . . .	3-21
Draw the Rectangle . . . . .	3-21
Intersect the Rectangle and the Annular Feed Ring . . . . .	3-22
Modify the Feed Gap's Attributes . . . . .	3-23
Draw the Air Volume . . . . .	3-25
Draw the Polyhedron . . . . .	3-25
Rename the Polyhedron . . . . .	3-26
Modify the Air Volume's Attributes . . . . .	3-26
Split the Model for Symmetry . . . . .	3-27

## 4. Setting Up the Problem

Set Up Boundaries and Excitations . . . . .	4-2
Boundary Conditions . . . . .	4-2
Excitation Conditions . . . . .	4-3
Assigning Boundaries . . . . .	4-3
Assign a Radiation Boundary to the Air Volume . . . . .	4-3
Assign a Perfect E Boundary to the Air Volume . . . . .	4-4
Assign a Perfect H Boundary to the Annular Feed Ring . . . . .	4-7
Assign a Symmetry Boundary to the Model . . . . .	4-8

---

Assigning Excitations .....	4-11
Assign a Lumped Port Across the Gap .....	4-11
Modify the Impedance Multiplier .....	4-13
Verify All Boundary and Excitation Assignments .....	4-15

## 5. Generating a Solution

Specify Solution Options .....	5-2
Add a Solution Setup .....	5-2
Add a Frequency Sweep to the Solution Setup .....	5-6
Define Mesh Operations .....	5-7
Validate the Project Setup .....	5-10
Generate the Solution .....	5-11
View the Solution Data .....	5-12
View the Profile Data .....	5-12
View Convergence Data .....	5-14
View Matrix Data .....	5-16

## 6. Analyzing the Solution

Create Modal S-Parameters Reports .....	6-2
Create an S-Parameters Report of S11 .....	6-2
Create an S-Parameters Report of Z11 .....	6-4
Create Field Overlay Plots .....	6-6
Create a Mag E Field Overlay Plot .....	6-6
Modify the Mag E Plot's Attributes .....	6-8
Create a Phase Animation of the Mag E Plot .....	6-10





# 1

## Introduction

This *Getting Started* guide is written for HFSS beginners as well as experienced users who are using version 12 for the first time. This guide leads you step-by-step through creating, solving, and analyzing the results of a dielectric resonator antenna problem.

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.

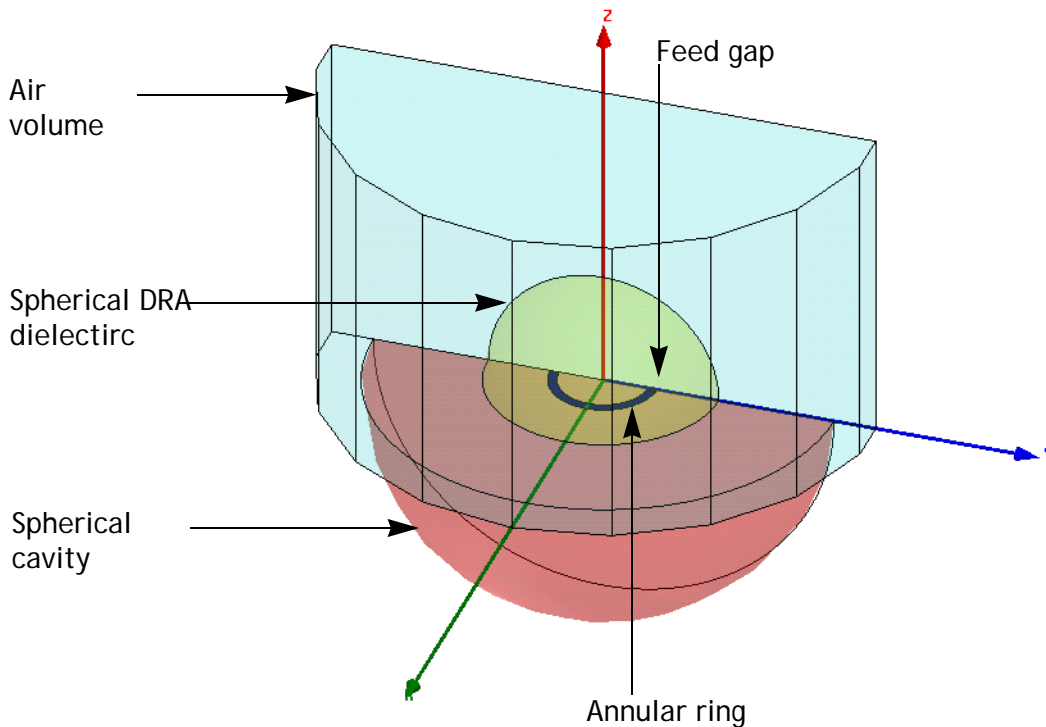
## The Sample Problem

This manual describes how to get started with HFSS by guiding you through the setup, solution, and analysis of 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 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 an annular slot. The design's operating frequency will be 3.5 GHz.

This problem is also described and analyzed in the following:

Leung, K.W., So, K.K., "Annular-slot-Excited Dielectric Resonator Antenna with a Backing Cavity," *IEEE Transactions on Antennas and Propagation*, August 2002.

The geometry for this antenna model is shown in the figure below:



## General Procedure

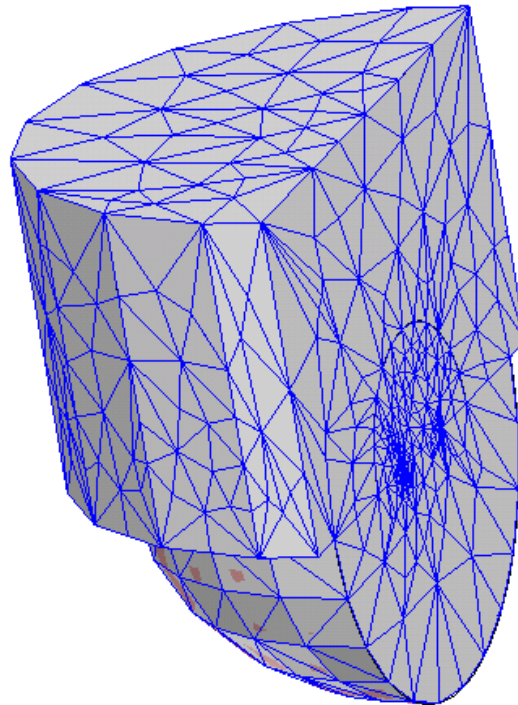
The general procedure for creating and analyzing a project is summarized in the following list:

- 1** Create a project for HFSS.
  - a. Open a new project.
  - b. Add an HFSS design into the new project.
- 2** Draw the geometric model; in this case, a cavity-backed, dielectric resonator antenna (DRA).
  - a. Set up the drawing region.
  - b. Create the objects that make up the antenna model.
  - c. Assign materials to the objects.
- 3** Set up the problem:
  - a. Set up boundary conditions.
  - b. Set up excitatons, which are the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces.
- 4** Generate a solution:
  - a. Set up the solution criteria and (optionally) refine the mesh.
  - b. Generate the solution.
  - c. View the results.
- 5** Analyze the solution results.

## Finite Element Method

The geometry for the dielectric resonator antenna problem (DRA) described in this manual 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. For more information on this model, see “The Sample Problem” on page 1-2.

In HFSS, the geometric model is automatically divided into a large number of tetrahedra, where a single tetrahedron is basically a four-sided pyramid. This collection of tetrahedra is referred to as the *finite element mesh*. The figure below shows the mesh that was created for the dielectric resonator 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.

## Results for Analysis

After setting up the antenna problem and generating a solution, you will:

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

**Time** It should take you approximately 3 hours to work through this manual.



# 2

## Creating the New Project

This guide assumes that HFSS has already been installed as described in the *Installation Guide*.

**Note** If you have not installed the software or you are not yet set up to run the software, STOP! Follow the instructions in the *Installation Guide*.

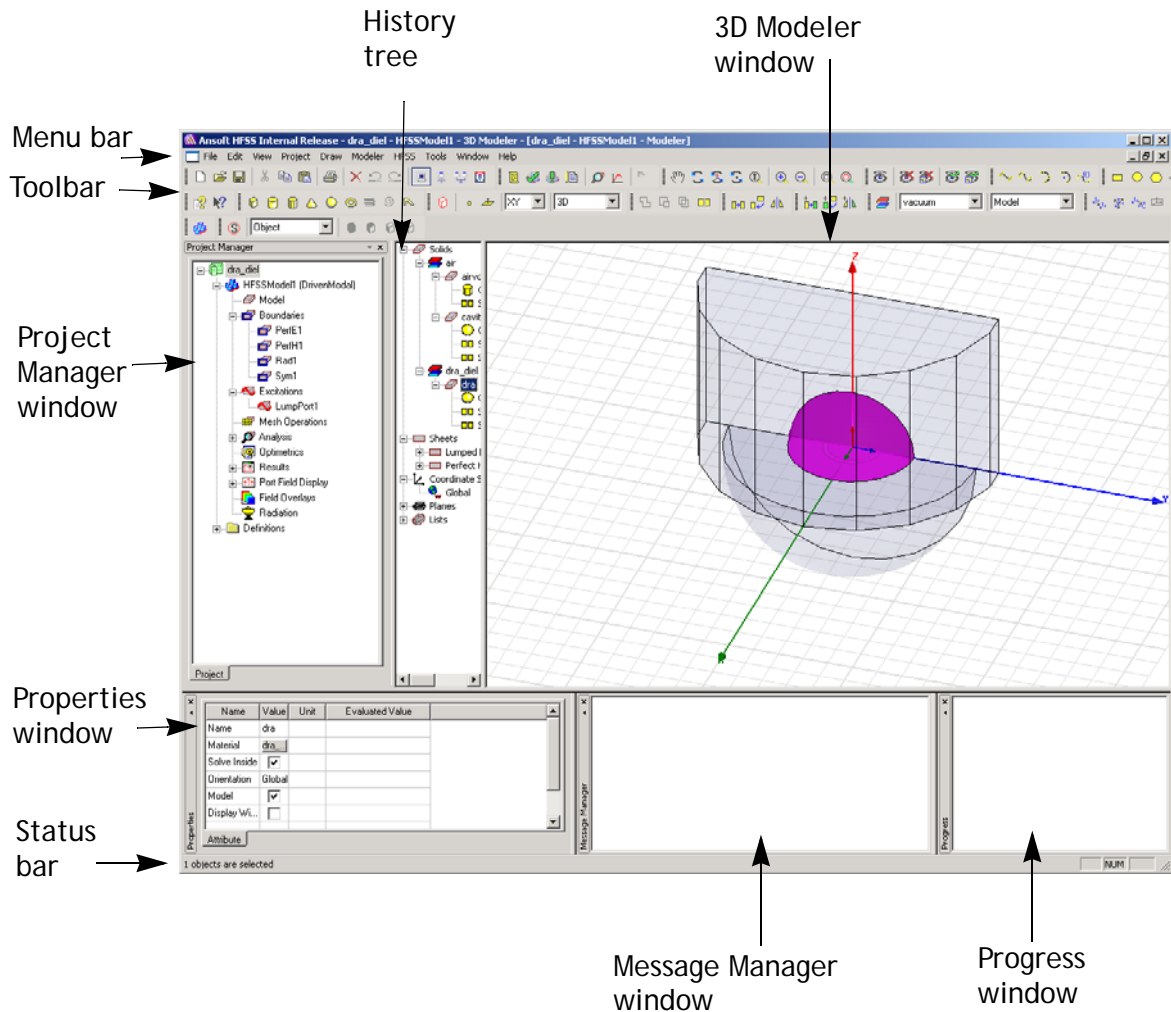
Your goals in this chapter are as follows:

- ✓ Create a new project.
- ✓ Add an HFSS design to the project.

**Time** It should take you approximately 15 minutes to work through this chapter.

## Overview of the Interface

Below is an overview of the major components of the HFSS version 12. interface:





<b>Project Manager window</b>	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.
<b>Message Manager window</b>	Displays error, informational, and warning messages for the active project.
<b>Progress window</b>	Displays solution progress information.
<b>Properties window</b>	<p>Displays the attributes of a selected object in the active model, such as the object's name, material assignment, orientation, color, and transparency.</p> <p>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.</p>
<b>3D Modeler window</b>	Displays the drawing area of the active model, along with the history tree.
<b>History tree</b>	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.

<b>Menu bar</b>	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.
<b>Toolbars</b>	Provides buttons that act as shortcuts for executing various commands.
<b>Status bar</b>	<p>Shows current actions and provides instructions.</p> <p>Also, depending on the command being carried out, the status bar can display the X, Y, and Z coordinate boxes, the <b>Absolute/Relative</b> 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.</p>

## Create the New Project

The first step in using HFSS to solve a problem, is to create a project in which the data associated with the problem can be saved. By default, when you run HFSS 12, it creates a new project named *project $n$*  and inserts a design named *HFSSDesign $n$* .

If you already have an HFSS window open, you can add a new project manually.

### Optionally, Add the New Project

To add a new project HFSS:

- Click **File>New**.

A new project is listed in the project tree in the **Project Manager** window. It is named *project $n$*  by default, where  $n$  is the order in which the project was added to the current session. Project definitions, such as boundaries and material assignments, are stored under the project name in the project tree.

### Optionally, Insert an HFSS Design

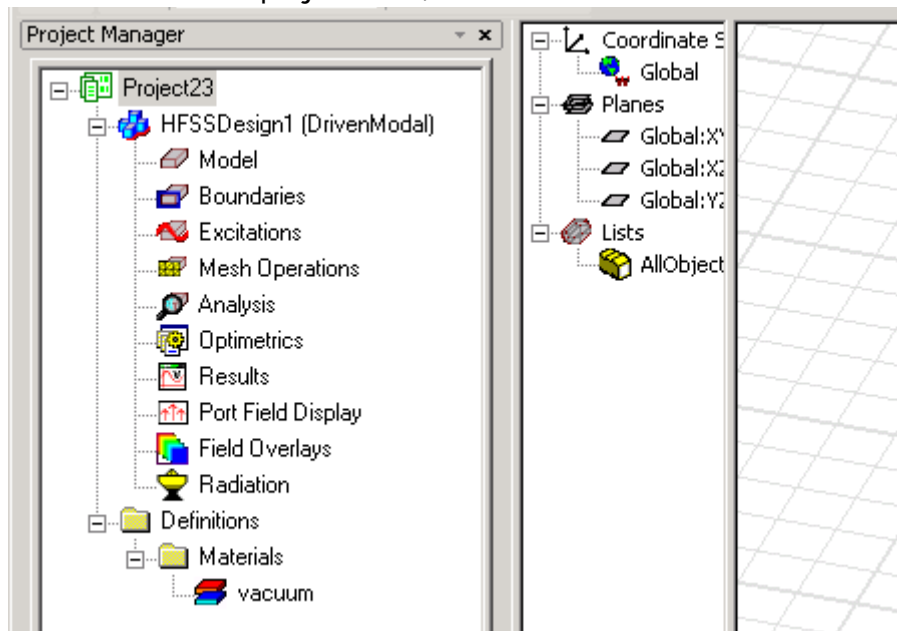
The next step for this antenna problem is to insert an HFSS design into the new project.

To insert an HFSS design into the project, do one of the following:

- Click **Project > Insert HFSS Design**.
- Right-click on the project name in the **Project Manager** window, and then click

Insert >Insert HFSS Design on the shortcut menu.

A 3D Modeler window appears on the desktop and an HFSS Design icon is added to the project tree, as shown below:



### Add Project Notes

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

To add notes to the project:

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

The **Design Notes** window appears.

- 2 Click in the window and type your notes, such as a description of the model and the version of HFSS in which it is being created.
- 3 Click **OK** to save the notes with the current project.

To edit existing project notes, double-click on **Notes** in the project tree. The **Design Notes** window appears, in which you can edit the project's notes.

### Save the Project

Next, save and name the new project.

It is important to save your project frequently. You can set an option to autosave at specified intervals, by using Tools>Options to open the General Options Dialog. The autosave option is controlled on the Project Options tab. Saving frequently helps prevent the loss of your work if a problem occurs.

To save the new project:

- 1** On the File menu, click **Save As**.
- 2** Use the **Save As** window to find the directory where you want to save the file.
- 3** Type the name **dra\_antenna** in the **File name** text box.
- 4** In the **Save as type** list, click **.hfss** as the correct file extension for the file type.

When you create an HFSS project, it is given an **.adsn** file extension by default and placed in the Project directory. Any files related to that project are stored in that directory.

Some common HFSS file extensions are listed below:

<b>.hfss</b>	HFSS version 10-12 projects
<b>.pjt</b>	Pre-version 9.0 HFSS projects
<b>.anfp</b>	Ansoft PCB neutral file

- 5** Click **Save**.  
HFSS saves the project to the location you specified.

**Note** For further information on any topic in HFSS, such as coordinate systems and grids or 3D Modeler commands or windows, you can view the context-sensitive help:

- Click the **Help** button in a pop-up window.
- Press **F1**. The cursor changes to a **?**. Click on the item on which you need help.

Use the commands from the **Help** menu.

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



# 3

## Creating the Model

This chapter shows you how to create the geometry for the antenna problem described earlier. Your goals are as follows:

- ✓ Select the solution type.
- ✓ Set up the drawing region.
- ✓ Create the objects that make up the antenna model, which includes:
  - Drawing the objects.
  - Assigning color and transparency to the objects.
  - Assigning materials to the objects.

You are now ready to start drawing the geometry.

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

## Select the Solution Type

Before you draw the antenna model, first you must specify a solution type. As you set up your model, available options will depend on the design's solution type.

To specify the solution type:

- 1 Click **HFSS>Solution Type**.

The **Solution Type** window appears.

- 2 This antenna project is a mode-based problem; therefore, select the **Driven Modal** solution type.

The possible solution types are described below.

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

- 3 Click **OK** to apply the **Driven Modal** solution type to your design.



## Set Up the Drawing Region

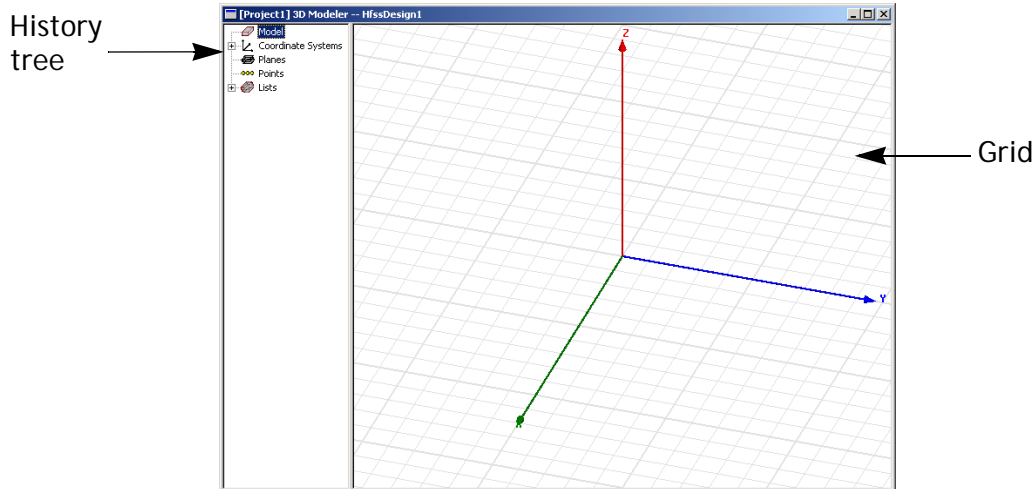
The next step is to set up the drawing region. For this antenna problem, you decide the coordinate system, and specify the units and grid settings.

### Overview of the 3D Modeler Window

The area containing the model is called the *drawing region*. Models are drawn in the 3D Modeler window, which appears on the desktop when you insert a design into the project.

As shown below, the 3D Modeler window consists of a grid and a history tree. The grid is an aid to help visualize the location of objects. For more information about the grid, see "Grid Settings" on page 3-4.

The history tree displays all operations and commands carried out on the active model. For more information about the history tree, see "History tree" on page 2-3.



## Coordinate System Settings

For this antenna problem, you will use the fixed, default global coordinate system (CS) as the working CS. This is the current CS with which objects being drawn are associated.

HFSS has three types of coordinate systems that let you easily orient new objects: a *global* coordinate system, a *relative* coordinate system, and a *face* coordinate system. Every CS has an x-axis that lies at a right angle to a y-axis, and a z-axis that is perpendicular to the xy plane. The origin (0,0,0) of every CS is located at the intersection of the x-, y-, and z-axes.

<b>Global CS</b>	The fixed, default CS for each new project. It cannot be edited or deleted.
<b>Relative CS</b>	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.
<b>Face CS</b>	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.

## Units Settings

Now, specify the drawing units for your model. For this antenna problem, set the drawing units to millimeters.

To set the units:

- 1 Click **Modeler>Units**.

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

- 2 Select **mm** from the **Select units** menu. Make sure **Rescale to new units** is cleared.

If selected, the **Rescale to new units** option automatically rescales the grid spacing to units entered that are different than the set drawing units.

- 3 Click **OK** to accept millimeters as the units for this model.

## Grid Settings

The grid displayed in the **3D Modeler** window is a drawing aid that helps to visualize the location of objects. The points on the grid are divided by

their local x-, y-, and z-coordinates 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.

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.

## Create the Geometry

The geometry for this dielectric resonator antenna (DRA) model consists of the five basic objects listed below with their dimensions:


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

## Draw the Cavity

The first object you will draw is the antenna's cavity, which is created by first drawing a sphere and then splitting it into a hemispherical solid.

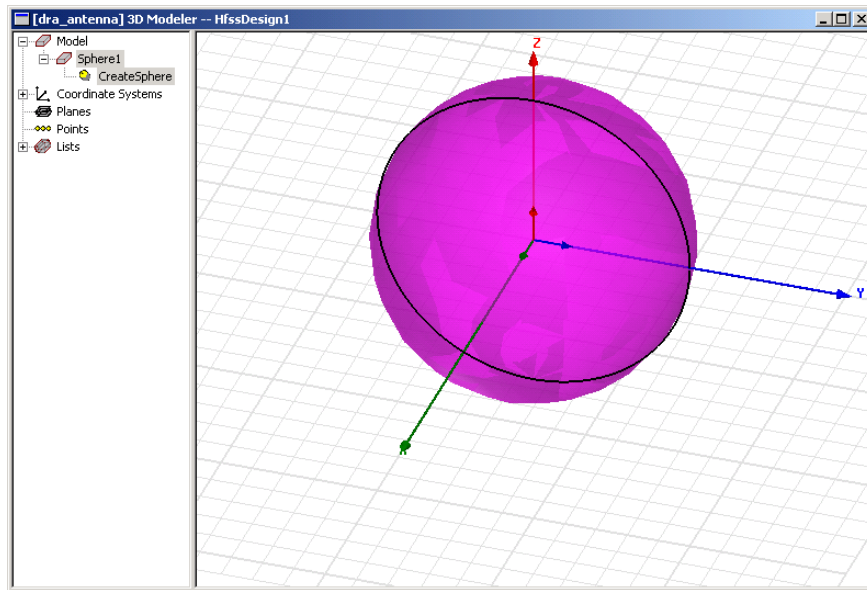
### Draw the Sphere

To draw the sphere:

- 1 Click **Draw>Sphere**, or click the **Draw sphere** button  on the tool-bar.
- 2 Select the center point of the sphere by entering the following values in the coordinate boxes:  

X coordinate	0
Y coordinate	0
Z coordinate	0
- 3 Press the **Tab** key to move to the next coordinate text box.  
To delete the selected point and start over, press **Esc**.  
The status bar now prompts you to enter a radius for the sphere.
- 4 Press **Enter** to accept the centre point.
- 5 Tab into the X box and enter 25.
- 6 Press **Enter** to accept the radius value.
- 7 The **Properties** window appears. Click **OK**.  
The sphere appears in the drawing region.
- 8 Press **Ctrl+D** to fit the sphere in the drawing region.

Your sphere should resemble the one shown below:



**Note** In HFSS 12, objects are automatically selected immediately after being drawn so that you can instantly view the selected object's default attributes in the **Properties** window.

### Rename the Sphere

Next, change the default name of the sphere to specify that it is the antenna's cavity.

To modify the sphere's name:

- 1** Under the **Attribute** tab of the **Properties** window, click the default name (Sphere1) in the **Name** row.
- 2** Type **cavity** to rename the sphere, and then press **Enter** to accept the new name.
- 3** Click **OK** to close the window.

### Split the Cavity

Next, split the cavity into a hemispherical solid.

To split **cavity**:

- 1** Select the object **cavity** by either clicking on it in the **3D Modeler**

window or clicking its name in the history tree.

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

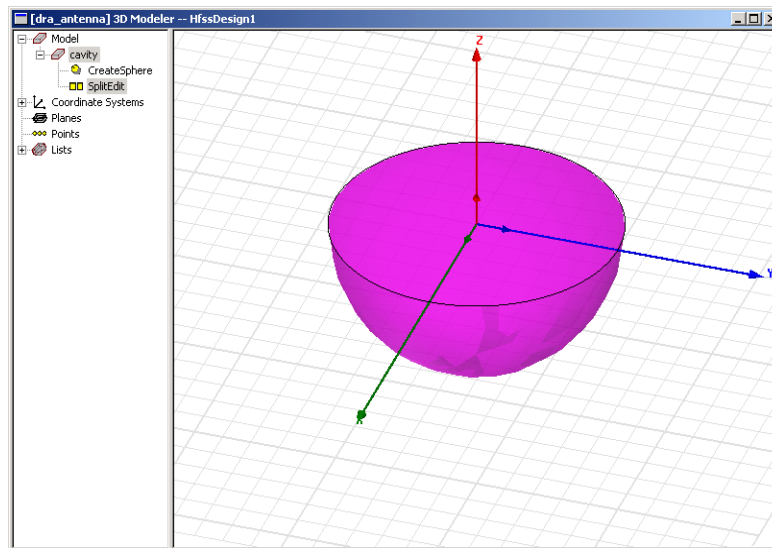
The **Split** dialog box appears.

**3** Select **XY** as the **split plane** that will be used to split the object **cavity**.

**4** Select **Negative side** as the object fragments you want to keep. This keeps the selected object fragments on the *negative* side of the xy plane.

**5** Click **OK**.

The object **cavity** is split into a hemispherical solid, as shown below:



**Note** In the above image that actions are added to the model history tree as they are accomplished, such as the **SplitEdit** entry, which corresponds to the split operations you performed.

### Modify the Cavity's Attributes

The next step in drawing the cavity is to modify its default attributes that are displayed in the

**Properties** window, which include assigning a color and transparency, and verifying the current material assignment.

### Assign a Color to the Cavity

To assign a color to the cavity:

- 1** Select the object **cavity**, if still not selected.
- 2** Under the **Attribute** tab of the **Properties** window, click **Edit** in the **Color** row.
- 3** The **Color** palette appears.
- 4** Select the basic color red (RGB settings 255, 0, 0) from the **Color** palette, and then click **OK** to assign the color to **cavity**.

### Assign a Transparency to the Cavity

To assign a transparency level to the cavity:

- 1** Select the object **cavity**, if it is not already selected.
- 2** Under the **Attribute** tab of the **Properties** window, click **0** in the **Transparency** row.  
The **Set Transparency** window appears.
- 3** Move the slider to the right to increase the transparency level, stopping on the 8th mark.  
The **Transparency** is now set to **.07**.
- 4** Click outside the object, on the grid background, to deselect **cavity** and view the resulting color and transparency assignments.

### Verify Lighting Attributes are Disabled

To verify if the lighting attributes are disabled:

- 1** Click **View>Modify Attributes>Lighting**.  
The **Lighting Properties** dialog box appears.
- 2** Verify that the **Do not use lighting** option is disabled. Clear this option if it is selected.  
If you want, you can change the default ambient and distant light source properties at this time, though it is unnecessary for this antenna problem.
- 3** Click **OK**.

### Verify the Cavity's Material

By default, all new objects created in HFSS 11 are assigned the material **vacuum**.

For this antenna problem, the object **cavity** will keep the default material assignment. Therefore, the next step is to simply verify that the cavity's material assignment is **vacuum**.

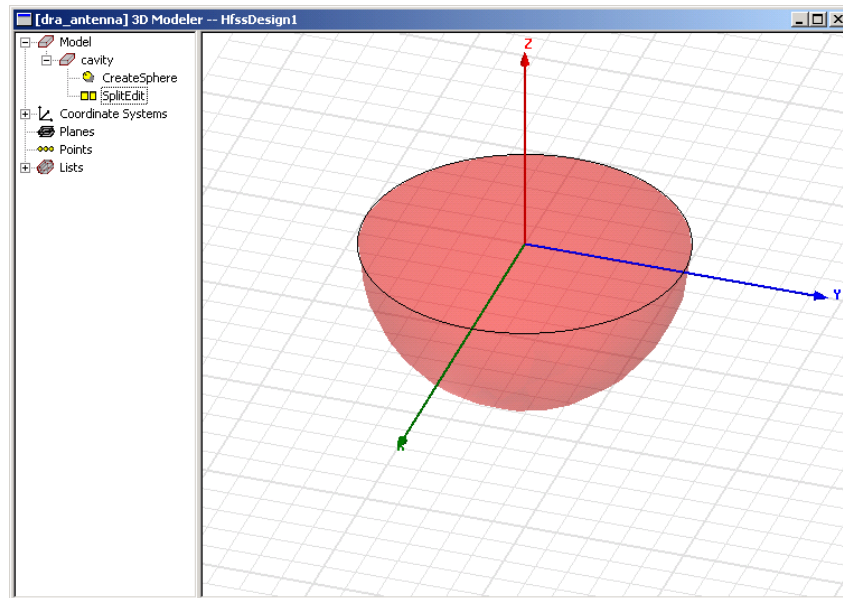
To verify the cavity's material assignment:

- 1 Select the object **cavity** if it is not already selected.
- 2 Click the **Attribute** tab of the **Properties** window.
- 3 Verify that **vacuum** is the current material assignment, which is displayed in the **Material** row.

This will be the permanent material assignment for **cavity**.

**Note** HFSS 11 lets you assign materials to objects at any time.

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





## Draw the DRA

Now, draw the dielectric resonator (DRA) object.

### Draw the Sphere

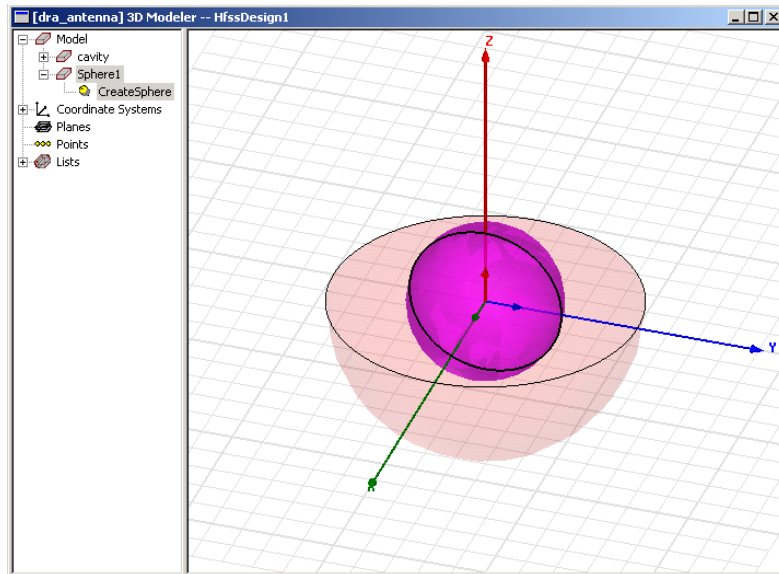
To draw the sphere:

- 1** Click **Draw>Sphere**, or click the **Draw sphere** button on the toolbar.
- 2** Select the center point (0, 0, 0) of the sphere by typing the values in the X, Y, and Z coordinate boxes.
- 3** Press **Enter** to accept the center point.
- 4** Tab into the dX box and enter 12.5 as the radius for the sphere.
- 5** Press **Enter** to accept the radius value.

The sphere appears in the drawing region.

Alternatively, after clicking Draw Sphere, you can click any arbitrary center point, and drag the mouse to an arbitrary radius and click again to draw a sphere. You can then edit the properties for Center Position to 0, 0, 0 and Radius to 12.5.

The sphere should appear in your model as shown below:



### Rename the Sphere

Next, change the default name of the sphere to specify that it is the dielectric resonator antenna (DRA) object.

To modify the sphere's name:

- 1 Under the **Attribute** tab of the **Properties** window, click the default name (**Sphere1**) in the **Name** row.
- 2 Type **dra** to rename the sphere, and then press **Enter** to accept the new name.

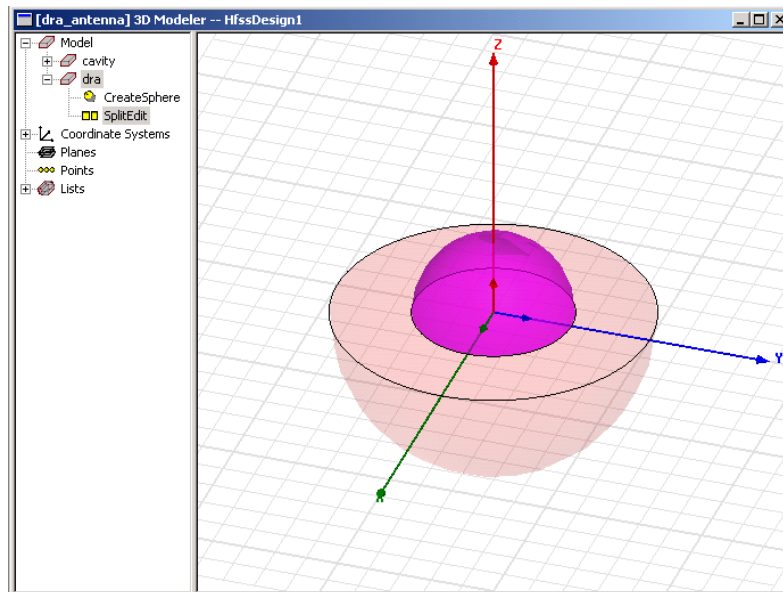
### Split the DRA

Next, split dra into a hemispherical solid.

To split dra:

- 1 Select **dra**, if not already selected.
- 2 Click **Modeler>Boolean>Split**.
- 3 Select **XY** as the **split plane** and **Positive side** as the **keep fragments**. This keeps the selected object fragments on the *positive* side of the xy plane.
- 4 Click **OK**.

The object **dra** is split into a hemispherical solid, as shown below:



### Modify DRA's Attributes

The next step in drawing the dra is to modify its color, transparency, and material default attributes that are displayed in the **Property** window.

### Assign a Color to the DRA

To assign a color to dra:

- 1** Select **dra**, if not already selected.
- 2** Under the **Attribute** tab of the **Properties** window, click **Edit** in the **Color** row.
- 3** Select the basic color yellow (RGB settings 255, 255, 0) from the **Color** palette, and then click **OK**.

### Assign a Transparency to the DRA

To assign a transparency level to the dra:

- 1** Under the **Attribute** tab of the **Properties** window, click the default value **0** in the **Transparency** row.
- 2** Move the slider to the right in the **Set Transparency** window, stopping to set the level to **.07**.
- 3** Click **OK**.
- 4** Deselect **dra** to view the resulting color and transparency assignments.

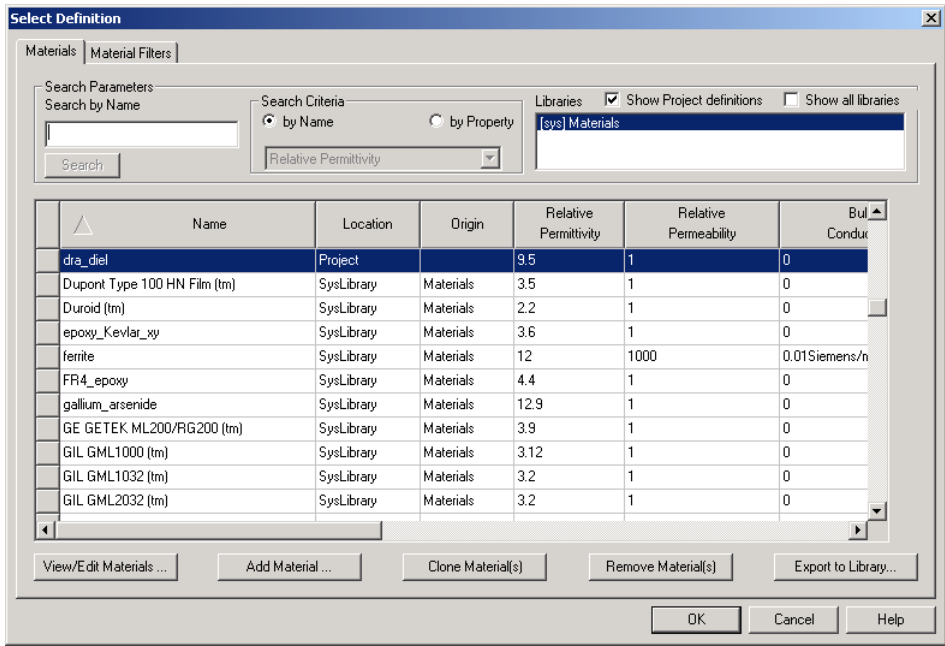
### Create and Assign a New Material to the DRA

The current default material assignment for the object **dra** is **vacuum**. Next, you will create a new material and assign it to **dra**.

To create and assign a new material to the dra:

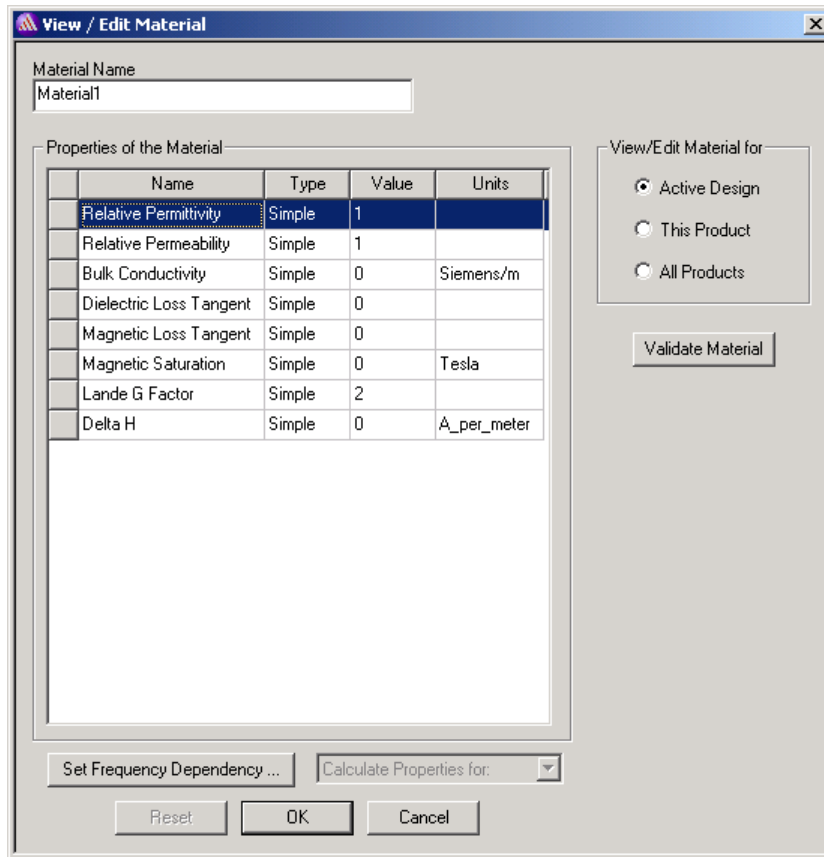
- 1** Select **dra**, if not already selected.
- 2** Under the **Attribute** tab of the **Properties** window, click the material name in the **Material** row.  
The **Select Definition** window appears, which lists all of the materials in Ansoft's global material library and the project's local material

library.



**3** Click Add Material.

The View/Edit Material dialog box appears:

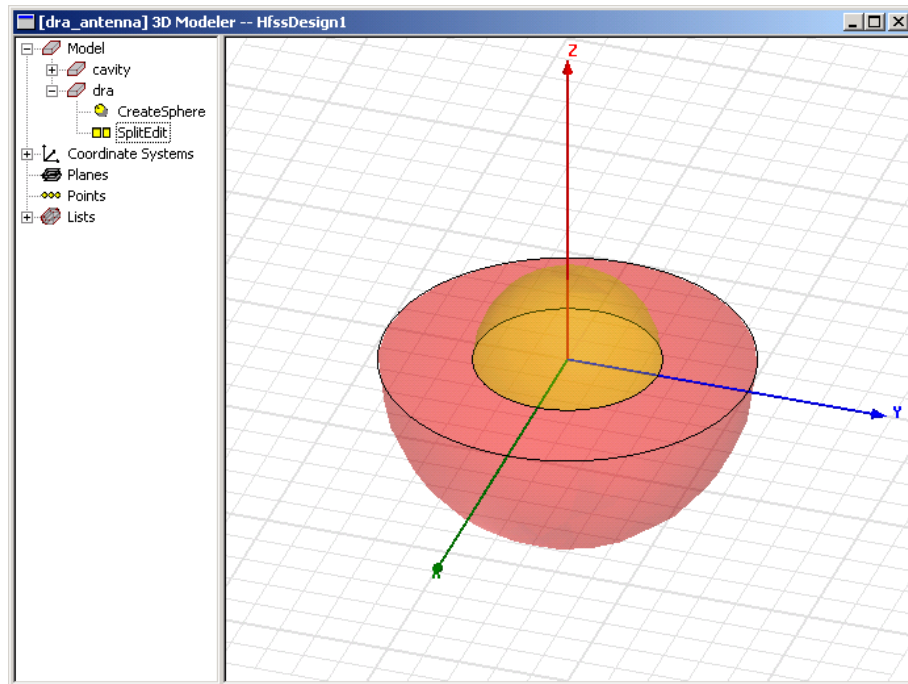


- 4** Type `dra_diel` in the Material Name text box to name the new material.
- 5** In the Relative Permittivity row, type `9.5` in its corresponding Value box, then press `Enter` to accept the value.
- 6** Click `OK`.
- 7** The material `dra_diel` now appears in the material browser.
- 8** To return to the 3D Modeler window, click `OK`.
- 9** The new material `dra_diel` is now assigned to the object `dra`.
- 10** Click `File>Save`, or click the `Save a project` button on the toolbar,

to save the geometry.

**Warning** Be sure to save geometric models periodically; HFSS does *not* automatically save models. Saving frequently helps prevent the loss of your work if a problem occurs.

The completed object **dra** should appear in your antenna model as shown below:




## 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 Chapter 4, “Setting Up the Problem”, you will assign a perfect H boundary to the annular feed ring to allow the E-fields to radiate through it.

Next, you will create the antenna’s annular feed ring, which is the result of subtracting one circle from another.


### Draw Circle1

To draw Circle1:

- 1** Click **Draw>Circle**, or click the **Draw circle** button  on the toolbar.
- 2** Select the center point (0, 0, 0) of the circle by typing the values in the coordinate boxes or by clicking the point on the grid.
- 3** Press **Enter**.
- 4** Tab into the X box and enter 4.8 as the radius.
- 5** Press **Enter** to accept the value.  
The **Properties** window appears.  
Circle1 now appears in the model.

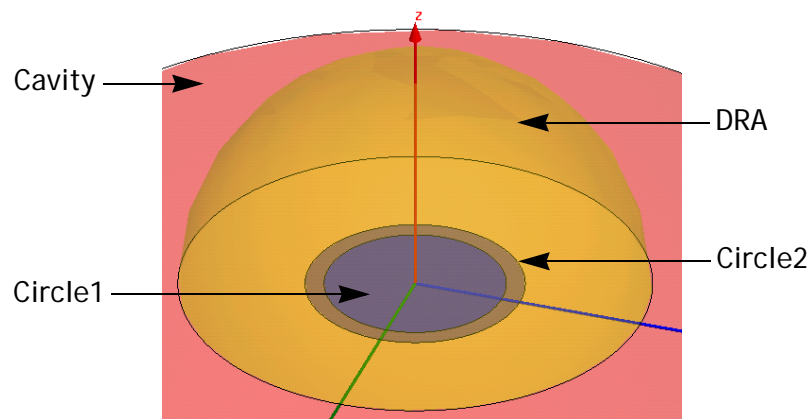
### Draw Circle2

To draw Circle2:

- 1** Click **Draw>Circle**, or click the **Draw circle** button  on the toolbar.
- 2** Select the center point (0, 0, 0) of the circle by typing the values in the coordinate boxes or by clicking the point on the grid.
- 3** Tab into the dX box and enter 5.8 as the radius.
- 4** Press **Enter** to accept the radius value.

**Note** You can also draw both circles arbitrarily and edit their Center Position and Radius properties on the Command tab.


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



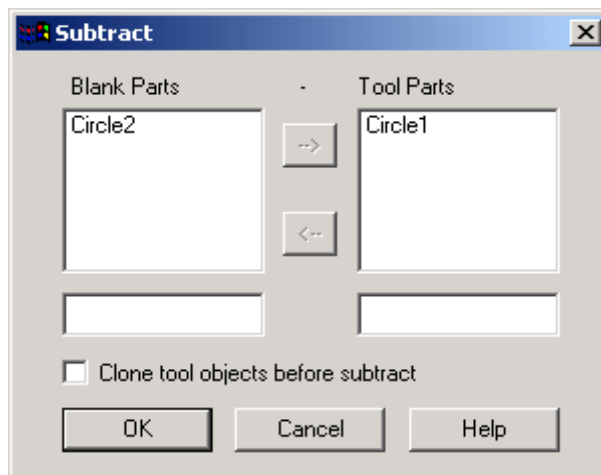
### Subtract Circle1 from Circle2

Next, subtract **Circle1** from **Circle2**, which will result in the 1 mm wide annular feed ring.

To subtract **Circle1** from **Circle2**:

- 1 Select **Circle2** either by clicking it in the **3D Modeler** window or clicking its name in the history tree. **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**. **Circle1** is the *tool* object—the object to be subtracted from the blank object. **Circle1** and **Circle2** should now both be selected. To verify, both objects should be highlighted in the model history tree, and the status bar should indicate that the number of objects selected is two.
- 3 Deselect **Clone before intersect**.
- 4 Click **Modeler>Boolean>Subtract** or click the **Subtract** button  from the toolbar.

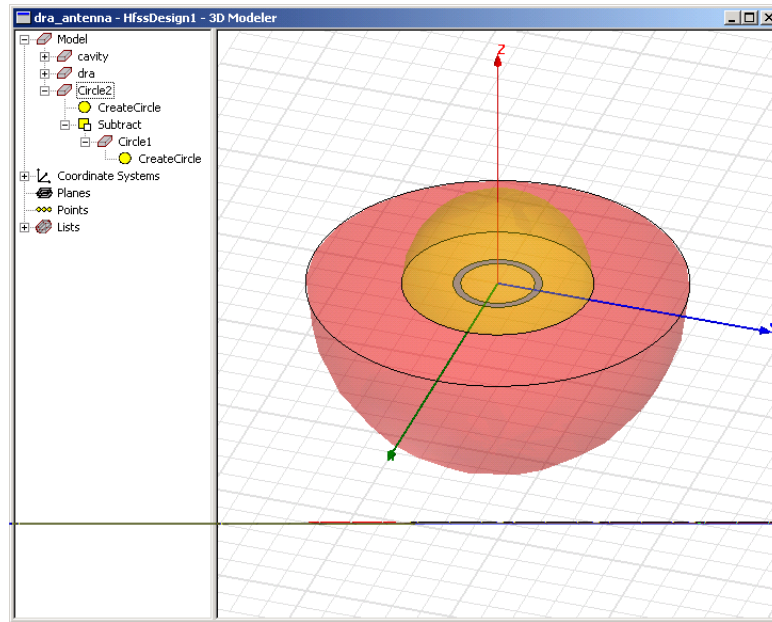
The **Subtract** window appears:



- 5 Verify that **Circle1** is in the **Tool Parts** list and **Circle2** is in the **Blank Parts** list.
- 6 Click **OK** to perform the subtraction. **Circle1** is subtracted from **Circle2**, resulting in a 1 mm wide flat



annulus or ring, as shown below:



**Note** In the above image that the model history tree has automatically re-sorted to reflect the subtraction performed. **Circle1** is now a sub-entry of **Circle2**.

### Rename Circle2

Next, change the name of **Circle2** to specify that it is the antenna's annular feed ring.

To modify the name of **Circle2**:

- 1** Under the **Attribute** tab in the **Properties** window, click **Circle2** in the **Name** row.
- 2** Type **annular\_rng** to rename the circle, and then press **Enter** to accept the new name.

### Modify the Annular Feed Ring's Attributes

The next step to drawing the annular feed ring is to modify its color and transparency.

The annular feed ring is a sheet object, which has surface area but no volume. Since the material parameter is a volumetric assignment, the annular feed ring will not have a material assignment.

### **Assign a Color to the Annular Feed Ring**

To assign a color to the annular feed ring:

- 1** Under the **Attribute** tab in the **Properties** window, click **Edit** in the **Color** row.
- 2** Select the basic color dark blue (RGB settings 0, 0, 128) from the **Color** palette, and then click **OK**.

### **Verify Annular Feed Ring's Transparency**

The annular feed ring object will keep the default transparency assignment. Therefore, you simply have to verify its default transparency assignment.

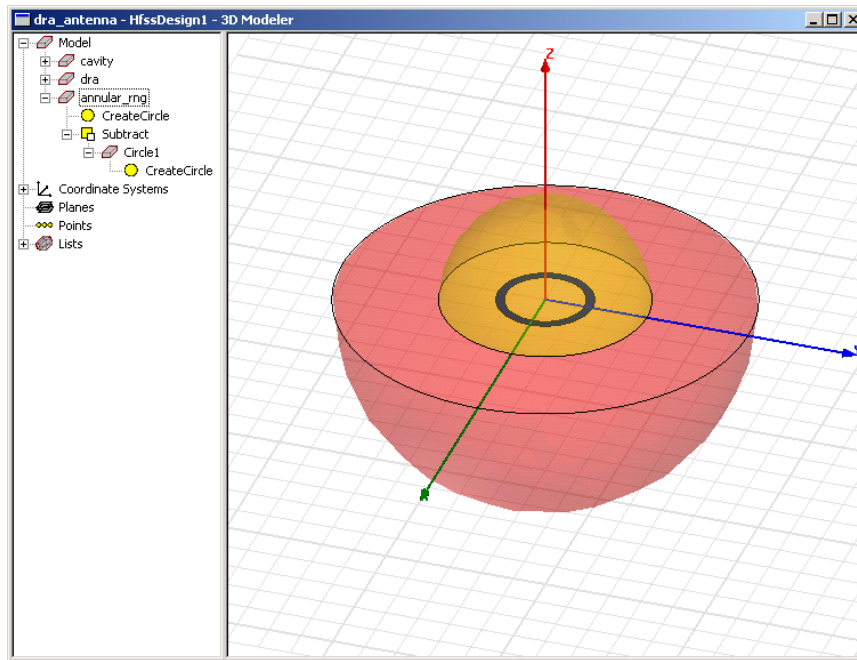
To assign a transparency to the annular feed ring:

- 1** Under the **Attribute** tab in the **Properties** window, click **0** in the **Transparency** row.
- 2** Move the slider to the right in the **Set Transparency** window and stop at the 2nd mark to set the level at **.09**.
- 3** Click **OK**.

Deselect **annular\_rng** to view the resulting color and transparency assignments.

The completed **annular\_rng** object should appear in your model as

shown below:



## Draw the Feed Gap

Next, draw the feed gap object, which is the object through which the excitation is fed.

### Draw the Rectangle

To draw the rectangle:

- 1** Click **Draw>Rectangle**, or click the **Draw rectangle** button on the toolbar.
- 2** Tab into the coordinate boxes and enter the following values to specify the rectangle's origin:
 

X coordinate	-0.5
Y coordinate	0
Z coordinate	0
- 3** Press **Enter** to accept the values.
- 4** Press **Tab** to return to the X box, and then enter the following values

to specify the end point:

dX coordinate    1

dY coordinate    10

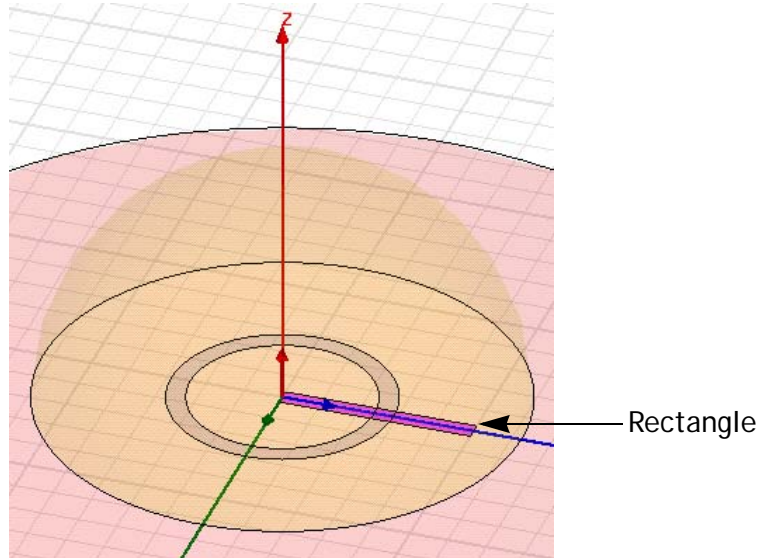
dZ coordinate    0

**5** Press **Enter** to accept the values.

The **Properties** window appears.

**Note** You can also draw an arbitrary rectangle and edit the Command tab properties for the Position as -0.5, 0, 0, the XSize as 1 and the Ysize as 10.

The rectangle appears in the model as shown below:



### Intersect the Rectangle and the Annular Feed Ring

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

To intersect the rectangle and the annular feed ring:

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

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

**2** Click the **Operation** tab.

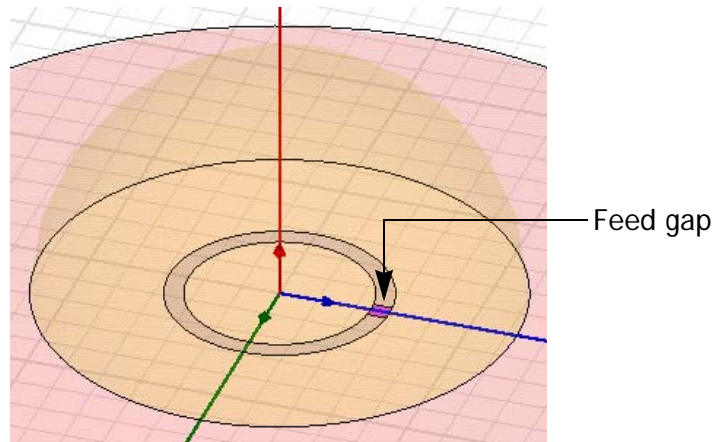
**3** Under **Clone**, select **Clone tool objects before intersecting**, and

then click **OK** to activate.

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

- 4** Select the object **Rectangle1**, if not already selected.
- 5** Press and hold down **Ctrl** to also select the object **annular\_rng**.  
The objects **Rectangle1** and **annular\_rng** should now both be selected.
- 6** Click **Modeler > Boolean > Intersect**, or click the **Intersect** button from the toolbar, to perform the intersection.

As a result of the intersection, the feed gap is produced without deleting the annular slot it is intended to feed, as shown below:



### Rename the Rectangle

Next, change the name of **Rectangle1** to specify that it is the antenna's feed gap.

To modify the name of the rectangle:

- 1** Under the **Attribute** tab in the **Properties** window, click **Rectangle1** in the **Name** row.
- 2** Type **gap** to rename the rectangle, and then press **Enter** to accept the new name.

### Modify the Feed Gap's Attributes

The next step to drawing the feed gap is to modify its color and transparency, and verify its default material assignment.

### **Assign a Color to the Feed Gap**

To assign a color to the feed gap:

- 1** Under the **Attribute** tab in the **Properties** window, click **Edit** in the **Color** row.
- 2** Select the basic color bright green (RGB settings 0, 255, 0) from the **Color** palette, and then click **OK**.

### **Assign a Transparency to the Feed Gap**

To assign a transparency level to the feed gap:

- 1** Under the **Attribute** tab in the **Properties** window, click **0** in the **Transparency** row.
- 2** Deselect **gap** to view the resulting color and transparency assignments.

## Draw 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. Then, in Chapter 4, Setting Up the Problem, you will assign a radiation boundary to this object.

### Draw the Polyhedron

To draw the polyhedron:

- 1** Click **Draw>Regular Polyhedron**, or click the **Draw regular polyhedron** button on the toolbar.
- 2** Select the center point (0, 0, 0) of the circle by typing the values in the coordinate boxes or by clicking the point on the grid.
- 3** Tab into the dX box and enter a radius value of 30, and then press **Enter** to accept the value.  
The status bar now prompts you to enter a height for the polyhedron.
- 4** Tab into the dZ coordinate box and enter a height value of 35, and then press **Enter** to accept the value.  
The **Segment Number** window appears.

- 5 Toggle the **Up** arrow to set the number of segments to **18**, and then click **OK** to accept the value.

The **Properties** window appears.

The polyhedron is drawn.

### **Rename the Polyhedron**

Next, change the name of **Polyhedron1** to specify that it is the antenna's air volume.

To modify the name of the polyhedron:

- 1 Under the **Attribute** tab in the **Properties** window, click the name **Regular Polyhedron1** in the **Name** row.
- 2 Type **airvol** to rename the polyhedron, and then press **Enter** to accept the new name.

### **Modify the Air Volume's Attributes**

The next step to drawing the air volume is to modify its color and transparency, and verify its default material assignment.

#### **Assign a Color to the Air Volume**

To assign a color to the air volume:

- 1 Under the **Attribute** tab in the **Properties** window, click **Edit** in the **Color** row.
- 2 Select the basic color light blue (RGB settings 0, 255, 255) from the **Color** palette, and then click **OK**.

#### **Assign a Transparency to the Air Volume**

To assign a transparency level to the air volume:

- 1 Under the **Attribute** tab in the **Properties** window, click the default value **.04** in the **Transparency** row.
- 2 Move the slider to the right in the **Set Transparency** window and stop at the 2nd mark to set the level at **.01**.

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

#### **Verify Air Volume's Material**

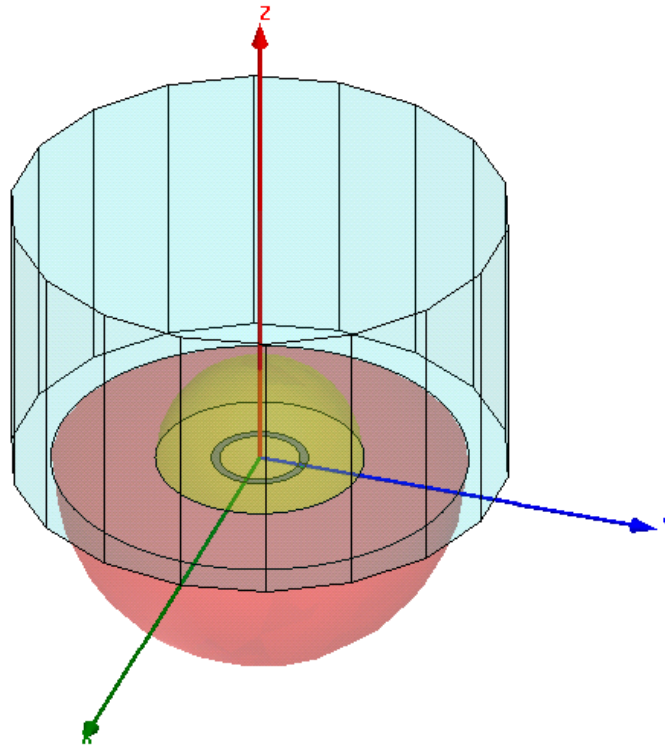
The object **airvol** will keep the default material assignment **vacuum**.



To verify the air volume's material assignment:

- 1** Select **airvol**.
- 2** Under the **Attribute** tab of the **Properties** window, verify that **vacuum** is the current material assignment, which is displayed in the **Material** row.
- 3** Click **File>Save**, or click the **Save a project** button on the toolbar, to save the geometry.

The completed **airvol** object should appear in your model as shown below:



## Split the Model for Symmetry

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

To split the model and create a cut plane:

- 1** Click **Edit > Select All** to select all the objects of the model.
- 2** Click **3D Modeler > Boolean > Split**, or click the **Split** button on the

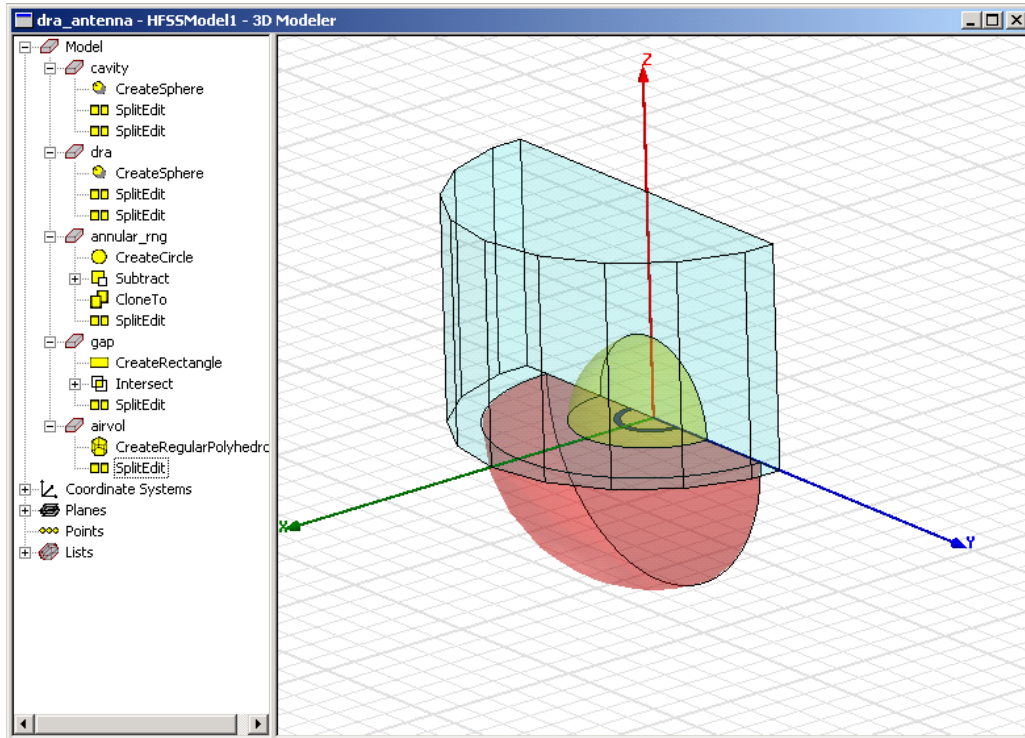
toolbar.

The **Split** window appears.

**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:



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

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

# 4

## Setting Up the Problem

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

Your goals for this chapter are to:

- ✓ 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.

**Time** It should take you approximately 30 minutes to work through this chapter.

## Set Up Boundaries and Excitations

Now that you have created all the objects of the antenna model and defined their properties, you must define the boundary and excitation conditions. These conditions specify 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:

- |                  |   |
|------------------|---|
| <b>Radiation</b> | 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.  |
| <b>Perfect E</b> | 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 <i>bottom</i> face of the air volume object is defined as a perfect E boundary.   |
| <b>Perfect H</b> | 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. |
| <b>Symmetry</b>  | 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.

As discussed in “Boundary Conditions” on page 4-2, radiation boundaries model surfaces that 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 the object **airvol** by either clicking on it in the **3D Modeler** window or clicking its name in the history tree.
- 2** On the **HFSS** menu, click **Boundaries>Assign>Radiation**.  
The **Radiation Boundary** window appears.
- 3** Click **OK** to accept the default name **Rad1** and apply the radiation boundary.

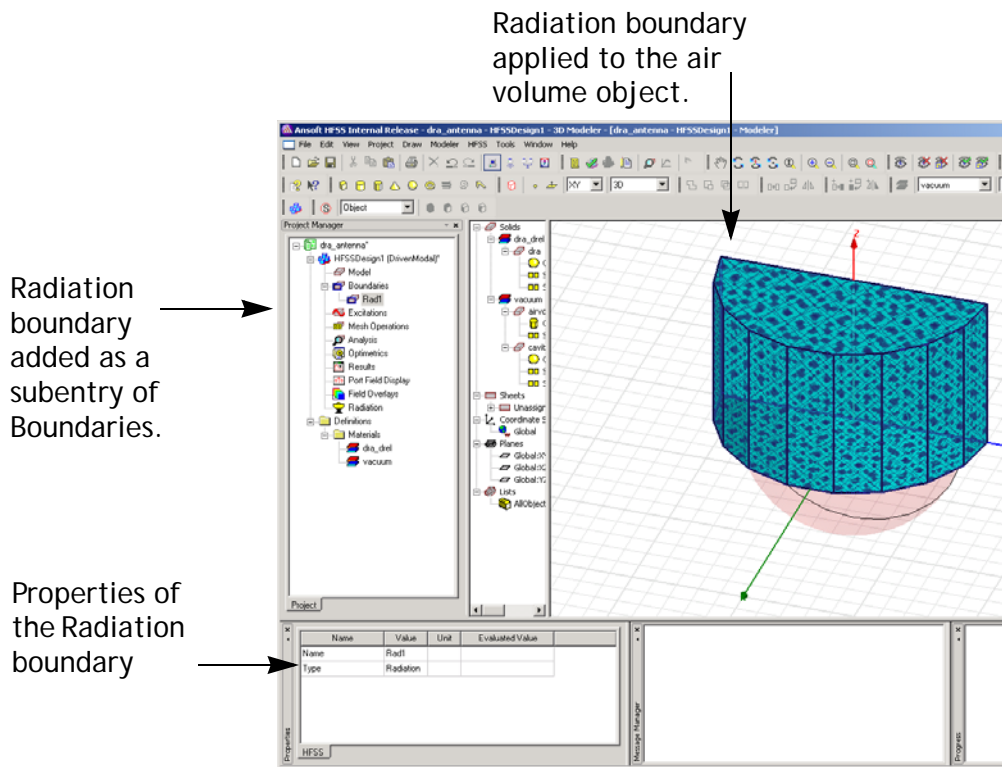
By default, the boundary's boundary, name, and vectors are all shown

in the 3D Modeler window. 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, as shown below:



### 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, if it is still selected.
- 2** Right-click in the **3D Modeler** window, then click **Select Faces** on the shortcut menu.

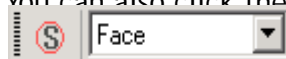
In this mode you can select an object's faces instead of the entire object.

When the mouse hovers over a face in the **3D Modeler** window, that face is highlighted, which indicates that it will be selected when you click.

- 3** Select the *bottom* face of the object **airvol** by doing the following:
  - Press and hold down **Alt** and drag the mouse to rotate the model to a position where you can then click the bottom face of the object **airvol**.

If you are having difficulty selecting this interior bottom face, right-click in the **3D Modeler** window and click **Next Behind** from the shortcut menu. This option selects the face or object behind a selected face or object.

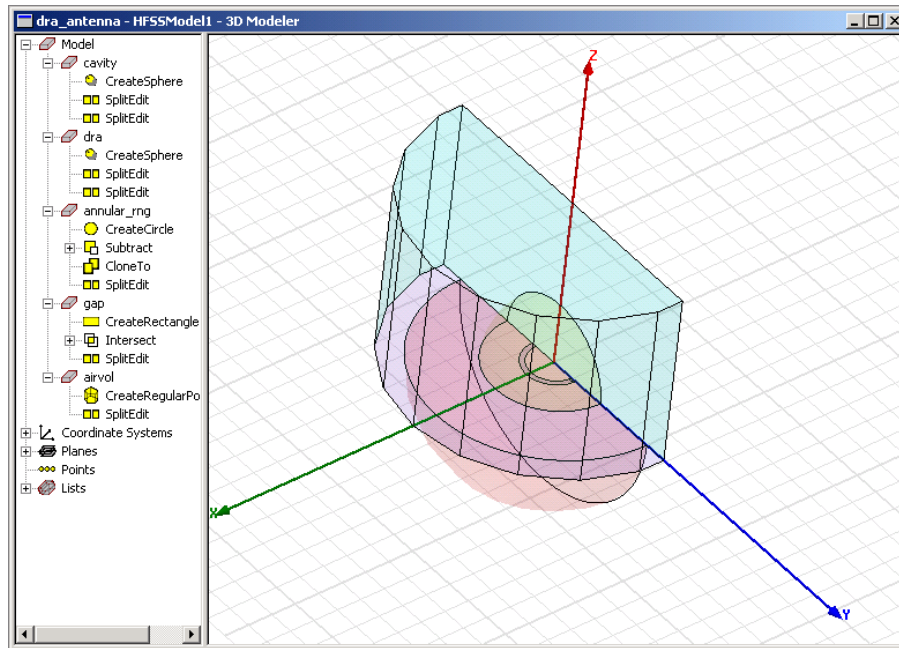
You can also click the **Select** icon with **Face** selected



to display the **Select Face** dialog. In the dialog, select **airvol** as the **Object**, and try selecting different faces in the list to see when the interior bottom face is highlighted.

In the figure below, the bottom face of **airvol** is selected and high-

lighted:



- 4** On the HFSS menu, click **Boundaries>Assign>Perfect E**.  
The Perfect E Boundary window appears.

**Hint** You can also assign boundaries by selecting the object or object face to which you want to assign the boundary, and then doing one of the following:

- Right-click in the **3D Modeler** window, point to **Assign Boundary**, and then click the boundary type you want to assign.
- Right-click on **Boundaries** in the project tree, point to **Assign**, and then click the boundary type you want to assign.

- 5** Clear **Infinite Ground Plane** if it is selected.

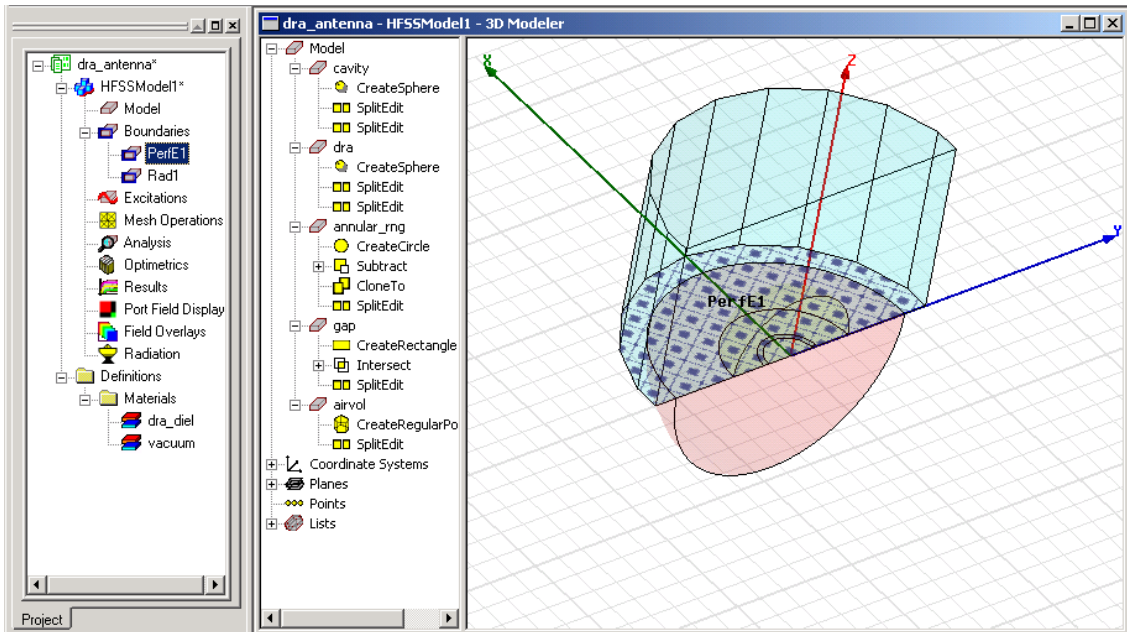
If selected, the **Infinite Ground Plane** option simulates the effects of an infinite ground plane. This option only affects the calculation of near- and far-field radiation during post processing. The 3D Post Processor models the boundary as a finite portion of an infinite, perfectly conducting plane.

- 6** Click **OK** to accept the default name **PerfE1** and apply the perfect E



boundary.

The resulting perfect E boundary condition is assigned to the bottom face of the object `airvol`, as shown below:



In the above image that boundaries are listed alphabetically in the project tree and re-ordered as new ones are added.

### Assign a Perfect H Boundary to the Annular Feed Ring

The next boundary you will assign is a perfect H condition on the annular ring portion of the perfect E boundary 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.

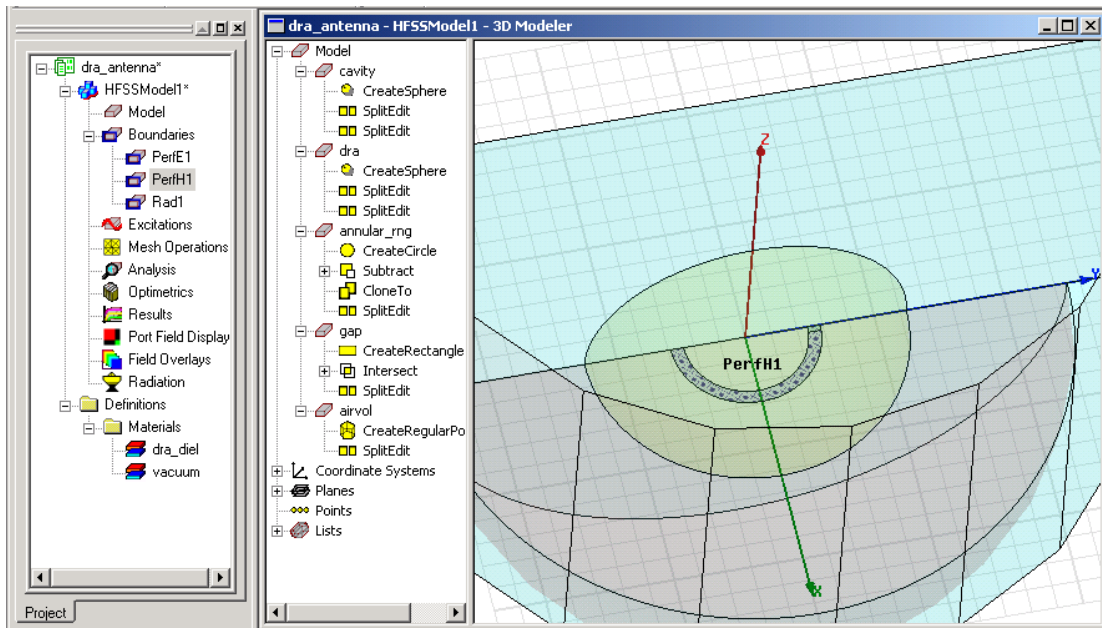
A perfect H boundary represents a surface on which the tangential component of the H-field is the same on both sides. For internal planes, such as the annular ring in this antenna model, this results in a natural boundary through which the field propagates. For planes on the outer surface of the model, 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 In Select Faces mode, click the face of the object `annular_rng`, or use the History tree to find `annular_rng` listed under Sheets.
- 3 On the HFSS menu, click **Boundaries>Assign>Perfect H**.  
The Perfect H Boundary window appears.
- 4 Click OK to accept the default name `PerfH1` and apply the perfect H boundary.

The resulting perfect H boundary condition is assigned to the face of the object `annular_rng`, as shown below:



### 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:

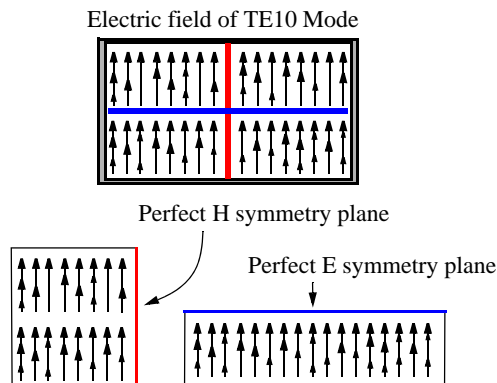
- If the symmetry is such that the E-field is normal to the symmetry

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.

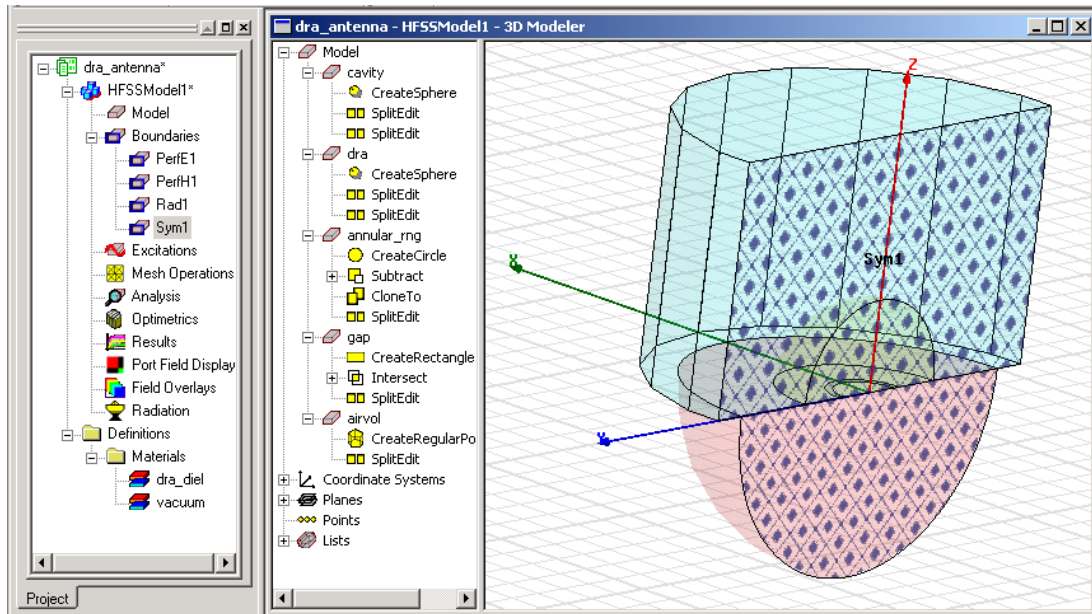


Since the antenna model in this guide has a vertical plane of symmetry and the E-field is tangential to the surface, use a perfect H boundary for the symmetry plane.

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, if it is still selected.
- 2 In **Select Faces** mode, select the symmetry cut faces of the objects **airvol** and **cavity**.  
To select multiple faces, press and hold down **Ctrl** while selecting the faces.
- 3 Press and hold down **Alt** and drag the mouse to rotate the model to a position where you can then click the symmetry cut faces, just as you did when you selected the bottom face of the air volume object.
- 4 On the **HFSS** menu, click **Boundaries>Assign>Symmetry**.  
The **Symmetry Boundary** dialog box appears.
- 5 Select **Perfect H** as the symmetry type, and then click **OK** to accept the default name **Sym1** and apply the symmetry boundary.  
The resulting perfect H symmetry boundary condition is assigned to the faces of the objects **airvol** and **cavity**, as shown below:



- 6 Click **File>Save**, or click the **Save a project** button on the toolbar, to save the project.

## Assigning Excitations

Now you will assign all excitation conditions to the model.

For information on the types of excitations you will assign, see “Excitation Conditions” on page 4-3.

### 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 you just assigned, if it is still selected.
- 2** Click **View > Zoom In** to zoom in on the area where the gap object is located.
- 3** In **Select Faces** mode, select the face of **gap**, or use the History tree and select gap under Sheets-Unassigned.
- 4** On the HFSS menu, click **Excitations>Assign>Lumped Port**.  
The **Lumped Port** wizard appears.  
The first time you assign a lumped port, HFSS walks you through the process with a step-by-step wizard.
- 5** In the **Lumped Port:General** step, enter the values listed below, and

then click **Next**.

<b>Name</b>	<b>LumpPort1</b>	The name of the port.
<b>Resistance</b>	<b>100 Ohms</b>	The resistance, or real port impedance.
<b>Reactance</b>	<b>0 Ohms</b>	The reactance, or imaginary port impedance.

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

**7** The **Lumped Port** wizard disappears while you draw the vector.

**8** Define the integration line:

- Select the start point by clicking the point where the *outside* of the gap and the y axis intersect (0,5.8, 0).
- Select the end point by clicking the point where the *inside* of the gap and the y axis intersect (0,4.8, 0).

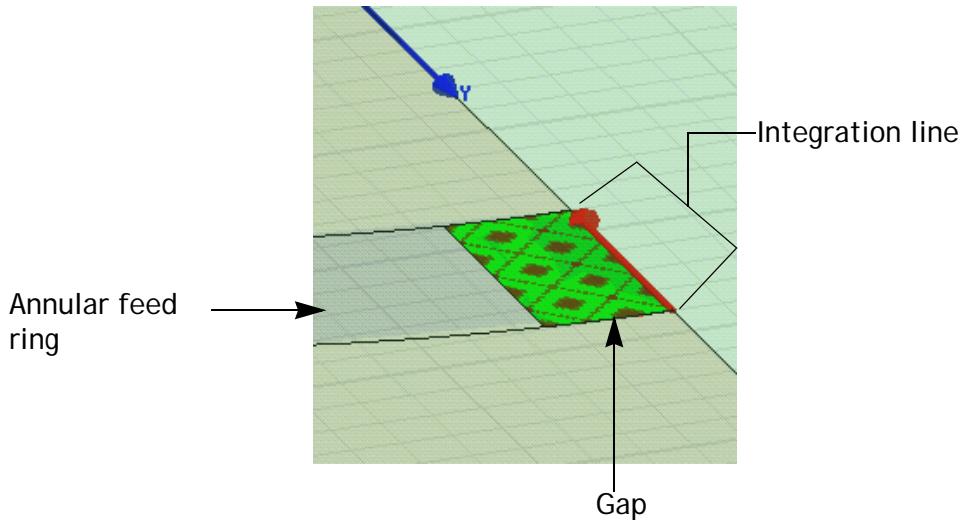
The endpoint defines the direction and length of the integration line.

The **Lumped Port** wizard reappears.

**9** Accept **Zpi** as the method with which to calculate the characteristic impedance, and then click **Finish** to complete the lumped port assignment.

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

below:



**Note** To edit a lumped port assignment:

1. Double-click on **Lumped Port** in the **Project Manager** window. The **Lumped Port** dialog box appears.
2. Click on the appropriate tabs (General, Modes, Defaults) to edit any port assignment information.
3. Click **OK** to apply the assignment revisions.

### Modify the Impedance Multiplier

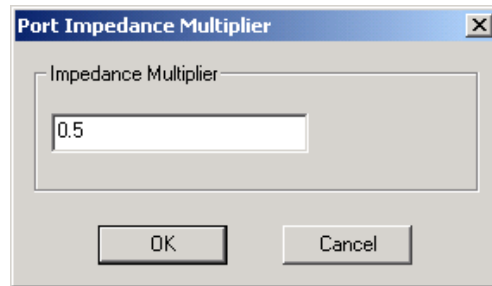
Because you defined a symmetry plane (allowing the model of a structure to be cut in half), the impedance computations must be adjusted by specifying an impedance multiplier.

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 Impedance Mult.**

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



- 2 Enter the value 0.5 in the **Impedance Multiplier** box, and then 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.

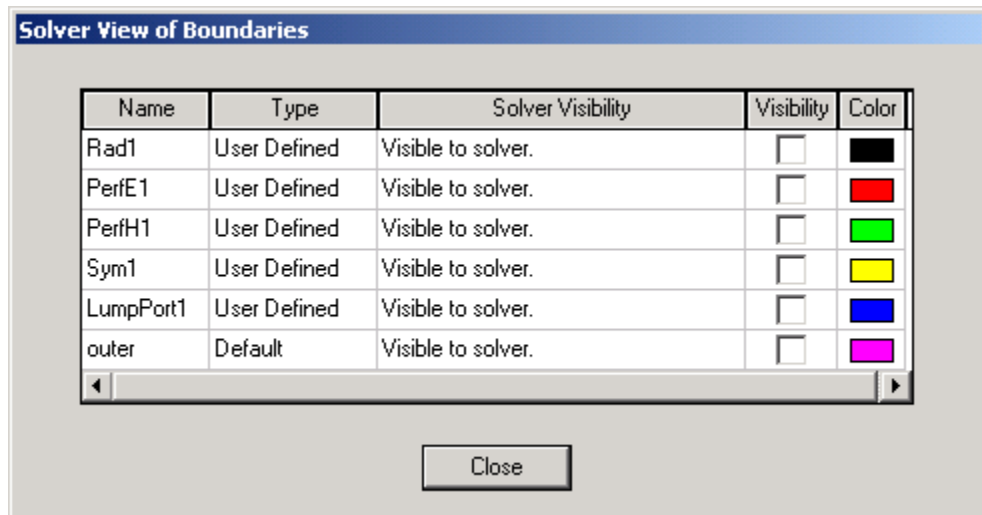
Then, you can select a boundary or excitation from the Boundaries list in the **Project** window to view its highlighted area in the model.

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.



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

The selected boundary or excitation 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.

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

**Warning** Be sure to save geometric models periodically; HFSS does *not* automatically save models. Saving frequently helps prevent the loss of your work if a problem occurs.

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.

## Specify Solution Options

Before you can generate a solution, you need to specify the solution parameters. This controls how HFSS computes the requested solution.

Each solution setup includes the following information:

- General data about the solution's generation.
- Adaptive mesh refinement parameters, if you want the mesh to be refined iteratively in areas of highest error.
- Frequency sweep parameters, if you want to solve over a range of frequencies.

You can define more than one solution setup per design; however, you will define only *one* solution setup for this antenna problem.

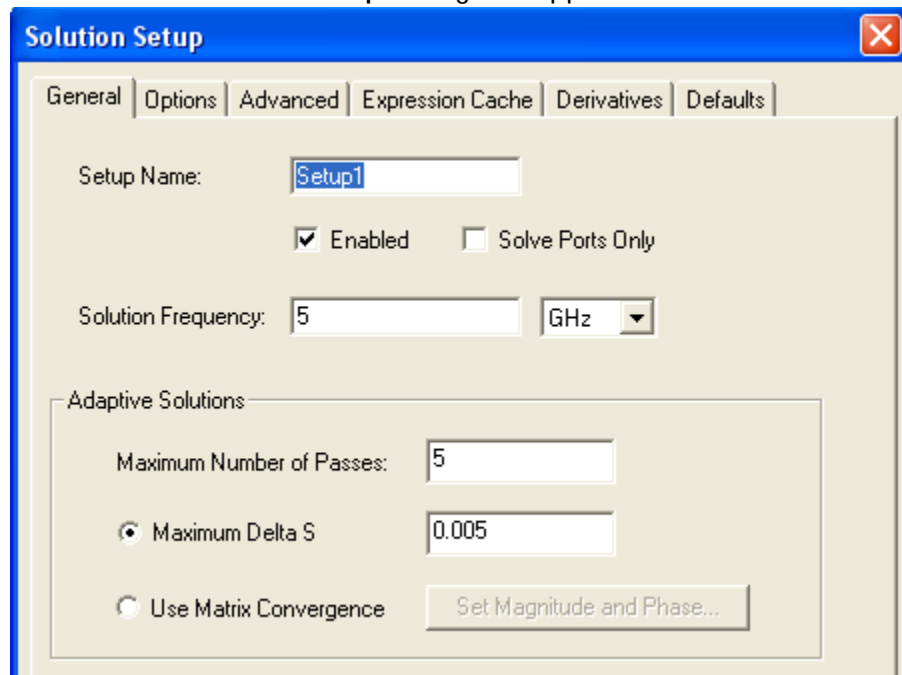
## Add a Solution Setup

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:



It is divided into the following tabs:

<b>General</b>	Includes general solution settings.
<b>Options</b>	Includes advanced settings for initial mesh generation, adaptive analysis, and solution options.
<b>Advanced</b>	Includes initial mesh options, port options, and two-way coupling options.
<b>Expression Cache</b>	Includes variables defined for output variable convergence, as well as adaptive options.
<b>Derivatives</b>	If your design includes variables, you can have HFSS produce derivatives for them.
<b>Defaults</b>	Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

**2** Click the **General** tab, and specify the following:

- a. Enter these values:

<b>Solution Frequency</b>	3.75 GHz For every modal driven solution setup, you must specify the frequency at which to generate the solution. For this antenna model, you will solve over a range of frequencies, which will require you to define a frequency sweep in “Add a Frequency Sweep to the Solution Setup” on page 5-6. If a frequency sweep is solved, an adaptive analysis is performed only at the solution frequency.
<b>Max. Number of Passes</b>	15 The <b>Maximum Number of Passes</b> value is the maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis will continue unless the convergence criteria are reached.
<b>Max Delta S Per Pass</b>	0.005 The delta S is the change in the magnitude of the S-parameters between two consecutive passes. The value you set for <b>Maximum Delta S Per Pass</b> is a stopping criterion for the adaptive solution. If the magnitude and phase of all S-parameters change by an amount less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.

- b. Accept all remaining current default settings.

**3** Click the **Options** tab, and specify the following:

- a. Select **Do Lambda Refinement**.

Lambda refinement is the process of refining the initial mesh based on the material-dependent wavelength. It is recommended and selected by default.

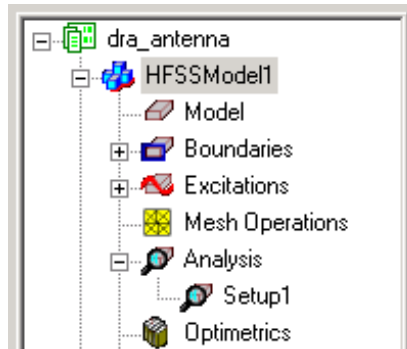
- b. Uncheck the Use Default Value box, and enter 0.25 in the **Lambda Target** text box.

This value specifies the size of wavelength by which HFSS will refine the mesh.

- c. Enter 2 in the **Minimum Converged Passes** text box.  
This value specifies the least number of passes for which the convergence criteria must be met before the adaptive analysis will stop.
- d. Accept all remaining current default settings.

**4** Click **OK**.

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

**Edit Sweep**

Sweep Name:  ☒ Enabled

Sweep Type:

**Frequency Setup**

Type:

Start:

Stop:

Step Size:

☐ Save Fields (All Frequencies)

**Interpolating Sweep Options**

Max Solutions:

Min Solutions:

Error Tolerance:  %

**DC Extrapolation Options**

☐ Extrapolate to DC

Minimum Solved Frequency:



- 3** Under the **Sweep Type** section, select **Fast** as the frequency sweep type you want to add.
- 4** Verify **Linear Step** is selected as the **Type**.
- 5** Under the **Frequency Setup** section, enter these values to define the sweep:
 

Start	2.5 GHz
Stop	5 GHz
Step Size	0.01 GHz
- 6** Select **Save Fields**, which saves the 3D field solutions associated with all port modes at each frequency.

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

- 7** 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.
- 8** Click **OK**.  
**Sweep1** now appears as a frequency sweep under **Setup1** in the project tree.

## 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:

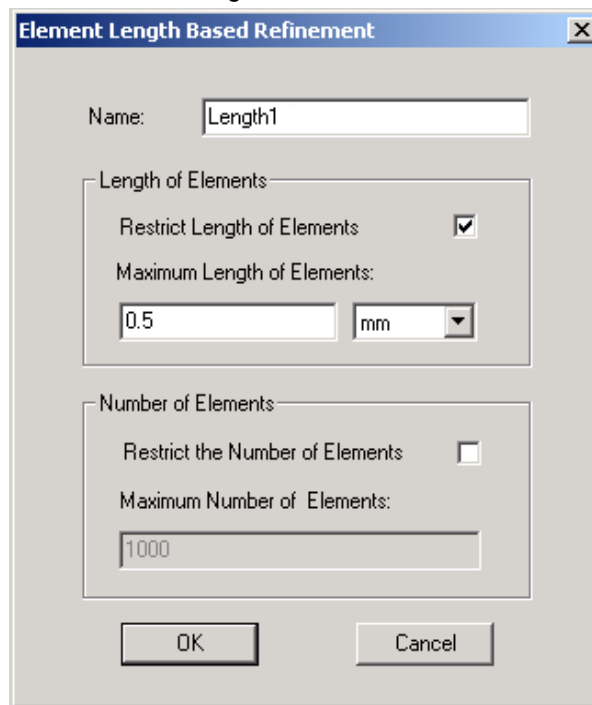
- 1 Use the History tree to select the object **annular\_rng**.

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**.

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

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



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

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.

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

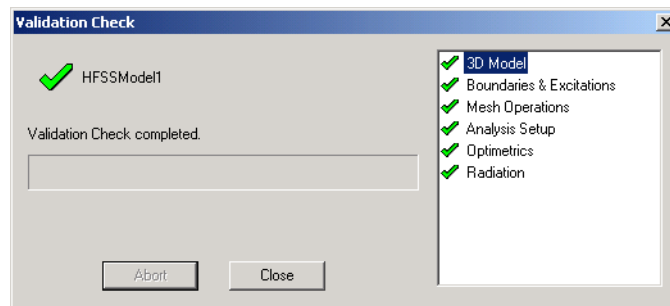
Before you run an analysis on the antenna model, it is important to first perform a validation check on the project. HFSS runs a check on all the setup details of the active project to verify that all the necessary steps have been completed and their parameters are reasonable.

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.

The following icons can appear next to an item:



Indicates the step is complete.



Indicates the step is incomplete.



Indicates the step may require your attention.

- 3 If the validation check indicates that a step in your antenna project is incomplete or incorrect, go back to the step in HFSS and carefully review its setup along with its instructions in this manual.
- 4 Click **Close**.
- 5 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.

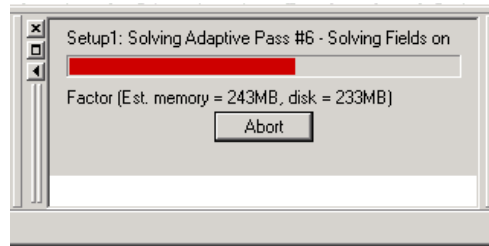
The following is the general process carried out during an adaptive analysis:

- 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.

**Solutions: dra\_diel - HFSSModel1**

Simulation: Setup1

Design Variation:

Profile | Convergence | Matrix Data | Mesh Statistics

Task	Real Time	CPU Time	Memory	Information
				Solution Basis Order: 1
Solution Sweep1				Fast Sweep
				From 2.5 GHz to 5 GHz, 250 Steps
Simulation Setup	00:00:01	00:00:01	42.1 M	Disk = 0 KBytes
Matrix Assembly	00:00:05	00:00:05	271 M	Disk = 0 KBytes, 27496 tetrahedra , LumpPort1: 25
Solver MCS4	00:01:19	00:02:30	1.01e+003 M	Disk = 53811 KBytes, matrix size 171841 , matrix b
Field Recovery	00:00:00	00:00:00	1.01e+003 M	Disk = 0 KBytes, 1 excitations
Solution Process				Elapsed time : 00:02:54 , Hfss ComEngine Memory
Total	00:01:25	00:02:36		Time: 02/04/2009 10:04:10, Status: Normal Comp

< ||| >

Export...

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:

<b>Task</b>	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.
<b>Real Time</b>	The amount of real time (clock time) required to perform the task.
<b>CPU Time</b>	The amount of CPU (Central Processing Unit) time required to perform the task.
<b>Memory</b>	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.
<b>Information</b>	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.

**Solutions: dra\_diel - HFSSModel1**

Simulation: Setup1

Design Variation:

Profile | **Convergence** | Matrix Data | Mesh Statistics

Number of Passes

Completed 7

Maximum 15

Minimum 1

Max Mag. Delta S

Target 0.005

Current 0.0029869

View: ☒ Table ☐ Plot

Export...

**CONVERGED**

Consecutive Passes

Target 2

Current 2

Pass Number	Total Tetrahedra	Max Mag. Delta S
1	9696	N/A
2	11636	0.019736
3	13965	0.016388
4	16763	0.0081793
5	20123	0.0054562
6	22904	0.0023284
7	27496	0.0029869

Close

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.

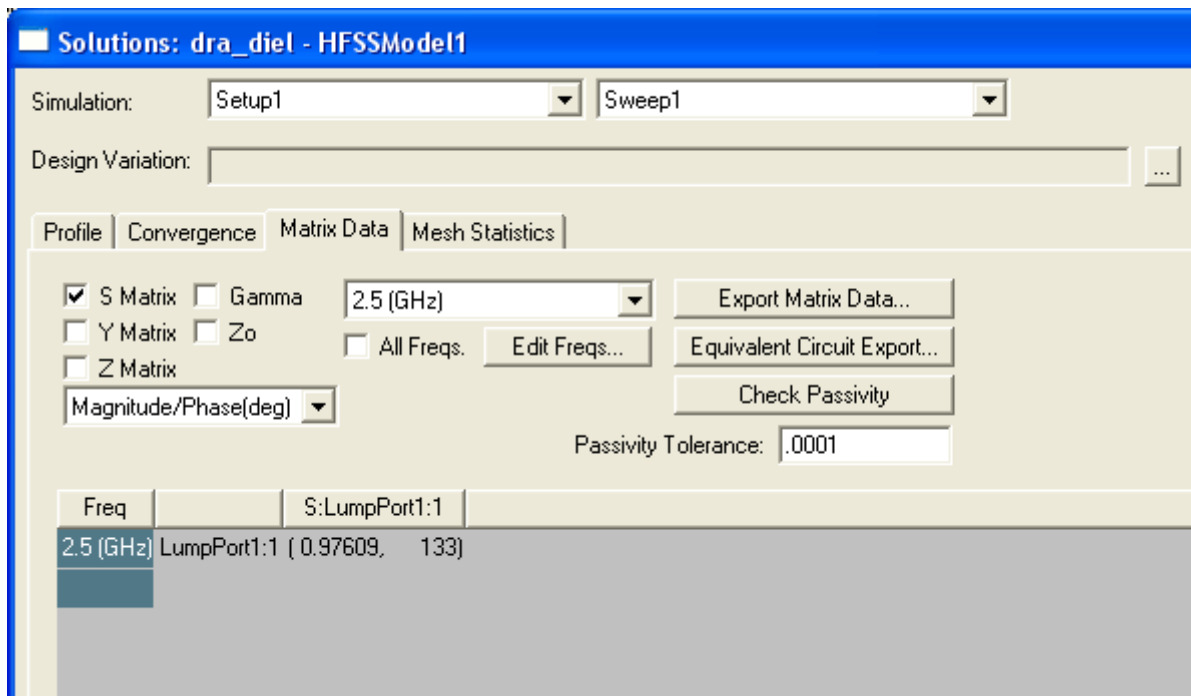
The convergence data can be displayed in table format or on a rectangular (X - Y) plot.

### 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:

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

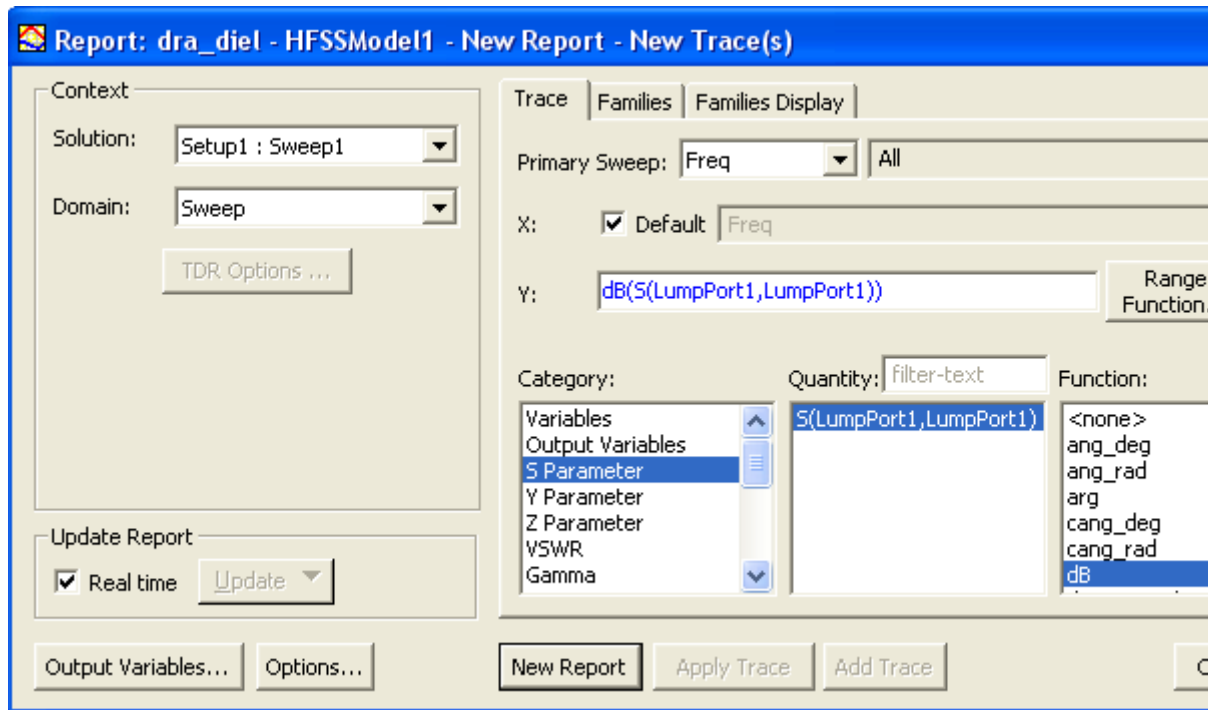
Now you are ready to create the modal S-parameters and Z-parameters reports.

### 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

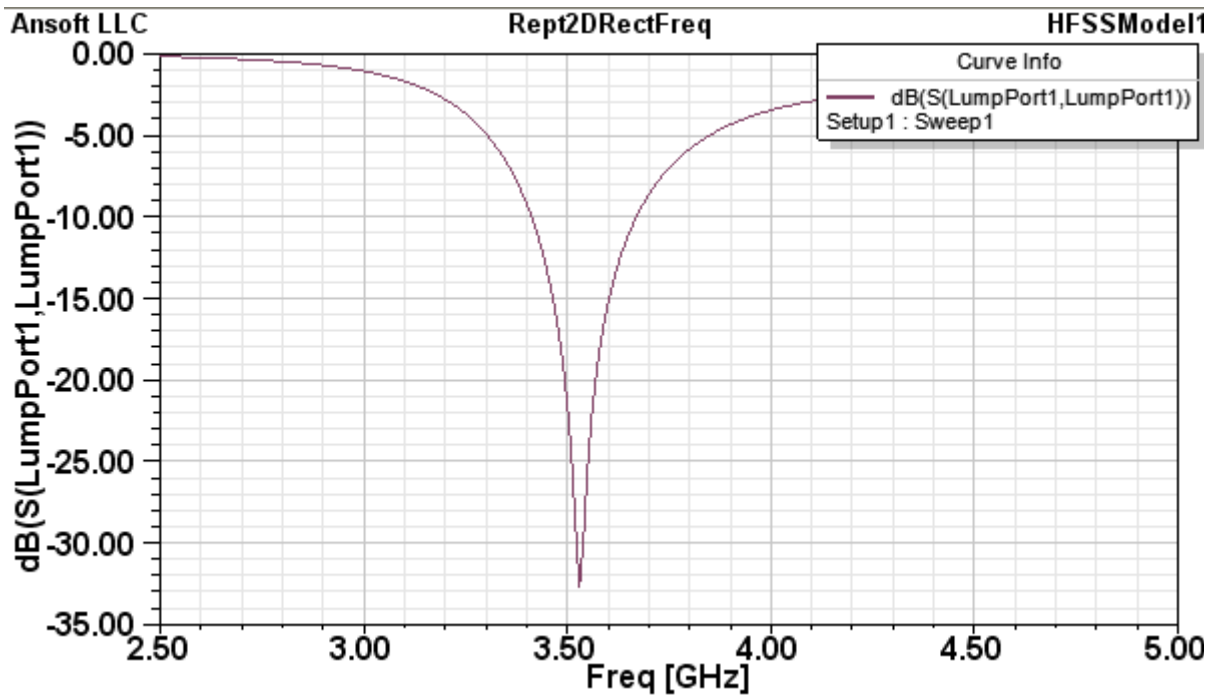
along the y-axis:

Category	<b>S Parameter</b> This is the type of information to plot.
Quantity	<b>S(LumpPort1,LumpPort1)</b> This is the value to plot.
Function	<b>dB</b> 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:

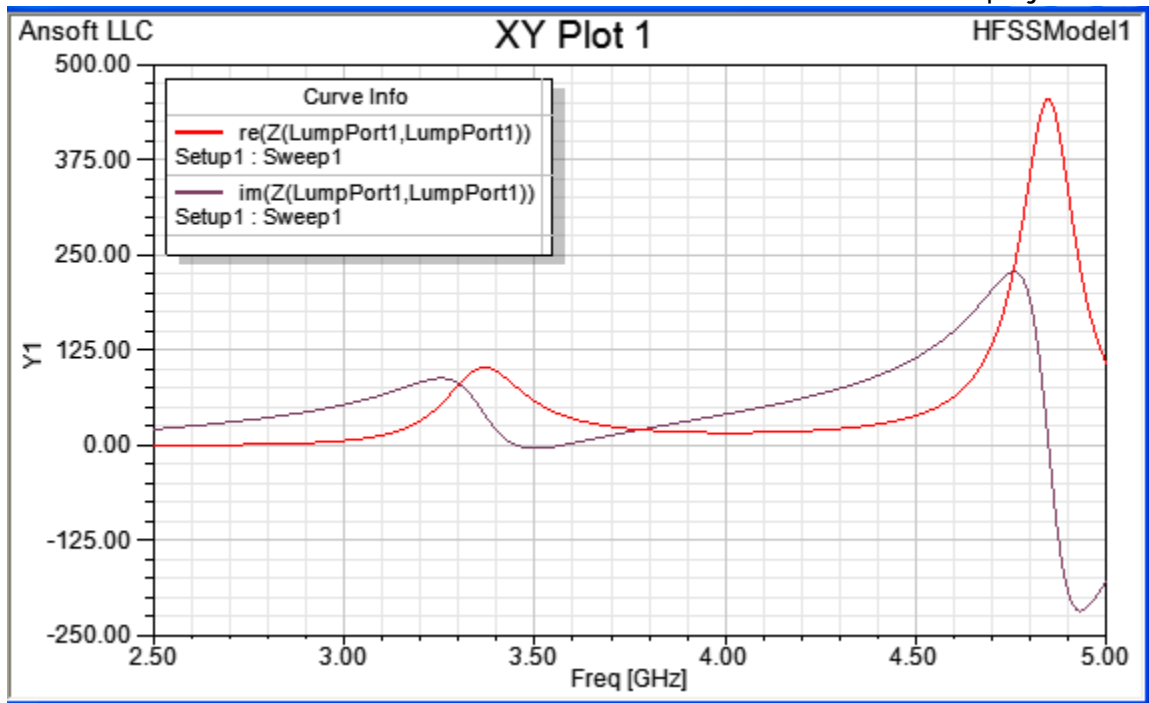
Category	Z Parameter
Quantity	Z(LumpedPort1, LumpedPort1)
Function	im

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

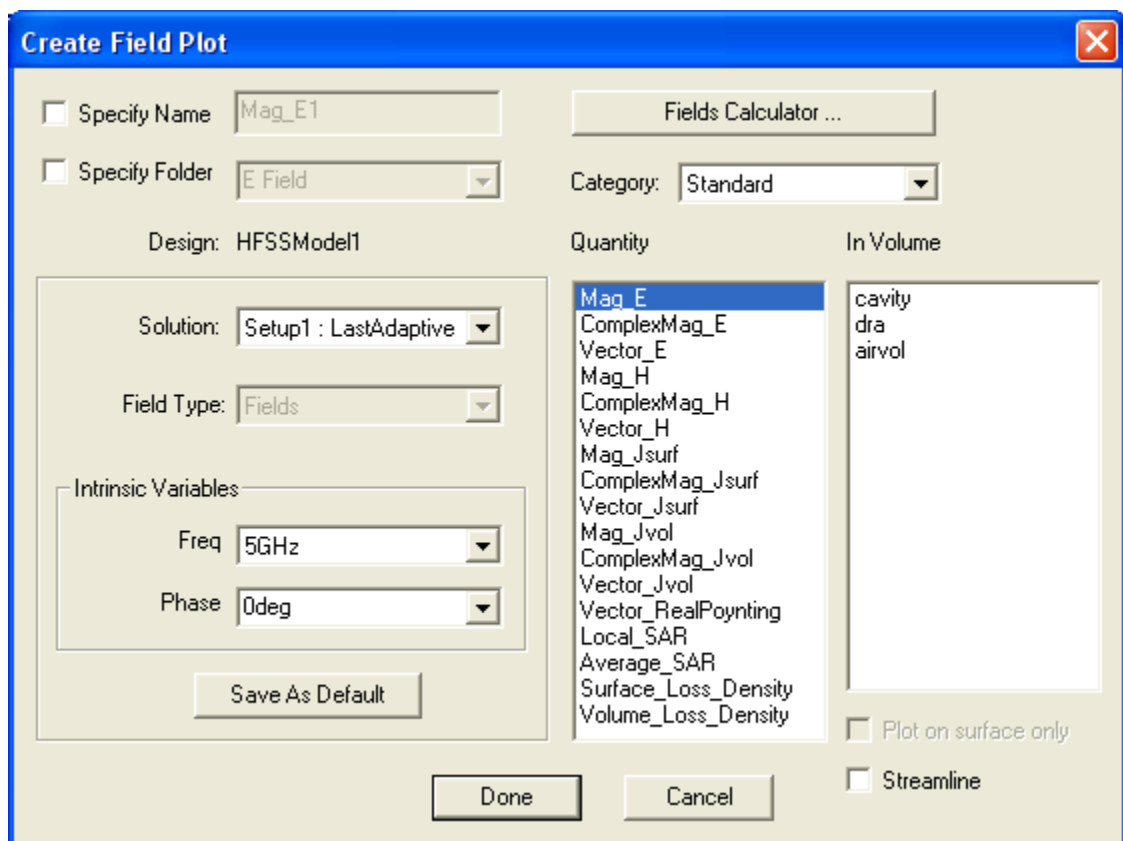
Now, you are ready to create a field overlay plot of the magnitude of E of the bottom face of the air volume object and examine the resulting E-field pattern.

- 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  $|\mathbf{E}|(x,y,z,t)$  as the quantity to plot.

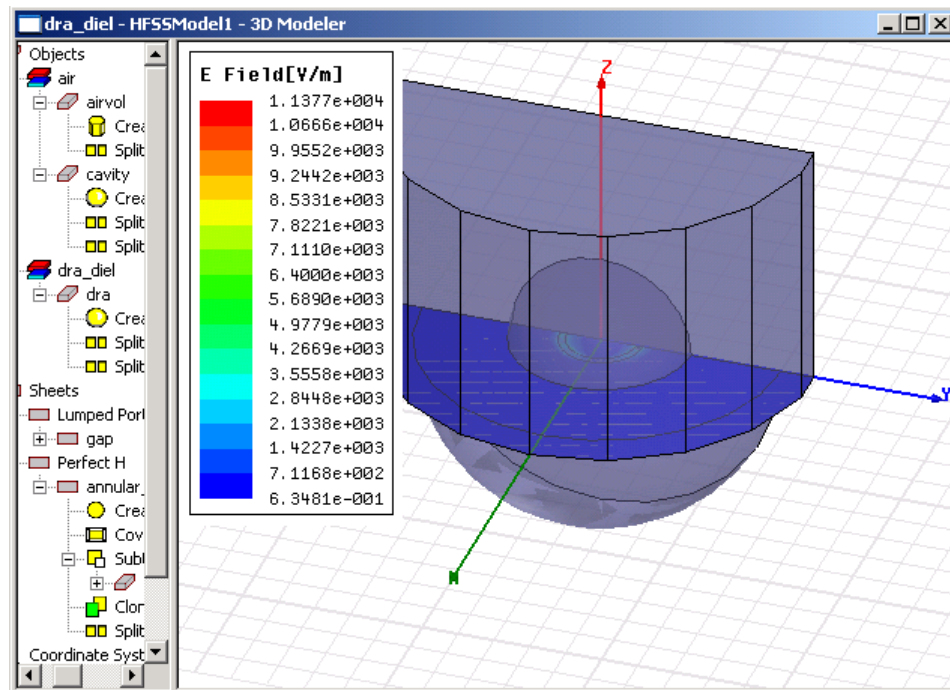
**4** Select **All** from the **In Volume** list to specify that HFSS will plot over the entire volume of the model.**5** Verify that **3.75GHz** is selected from the **Freq** pull-down list.

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

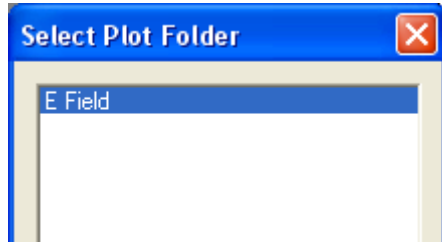
Now, you will modify the attributes of the **Mag\_E1** field overlay plot you just created to prepare it for an effective animation.

- 1 Select **Mag\_E1** in the project tree under Field Overlays-E Field.

This is the folder in which plot **Mag\_E1** is located.

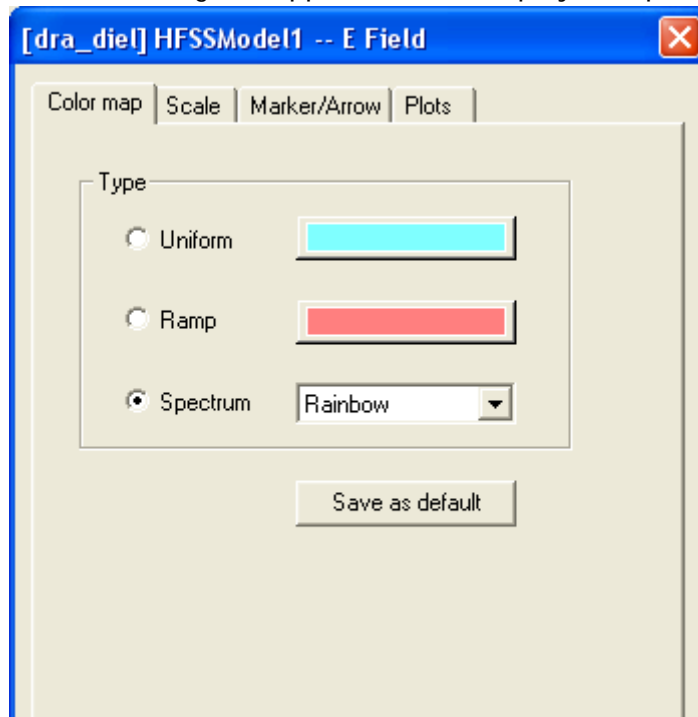
- 2 On the **HFSS** menu, point to **Fields**, and then click **Modify Attributes**.

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:

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

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

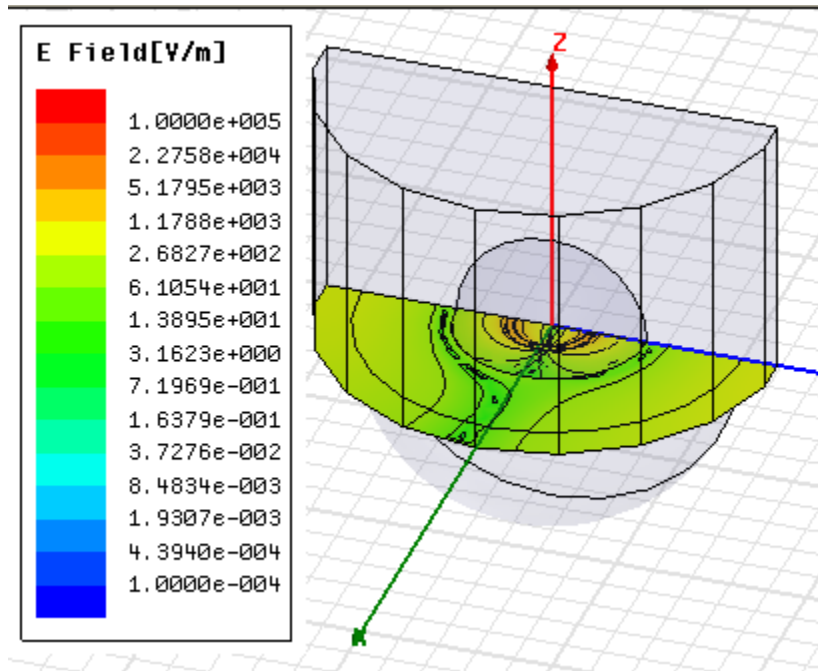
- |                    |  |
|--------------------|--|
| <b>Use Limits</b>  | Select this option.<br>Only the field values between the minimum and maximum values will be plotted. Field values below or above these values will be plotted in the colors assigned to the minimum or maximum limits, respectively.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

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

- |                    |   |
|--------------------|---|
| <b>IsoValType</b>  | Select <b>Tone</b> from the pull-down list.<br>This isosurface display type varies color continuously between isovalues.          |
| <b>Outline</b>     | Select this option.   |
| <b>Map transp.</b> | Clear this option, if it is selected.<br>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:



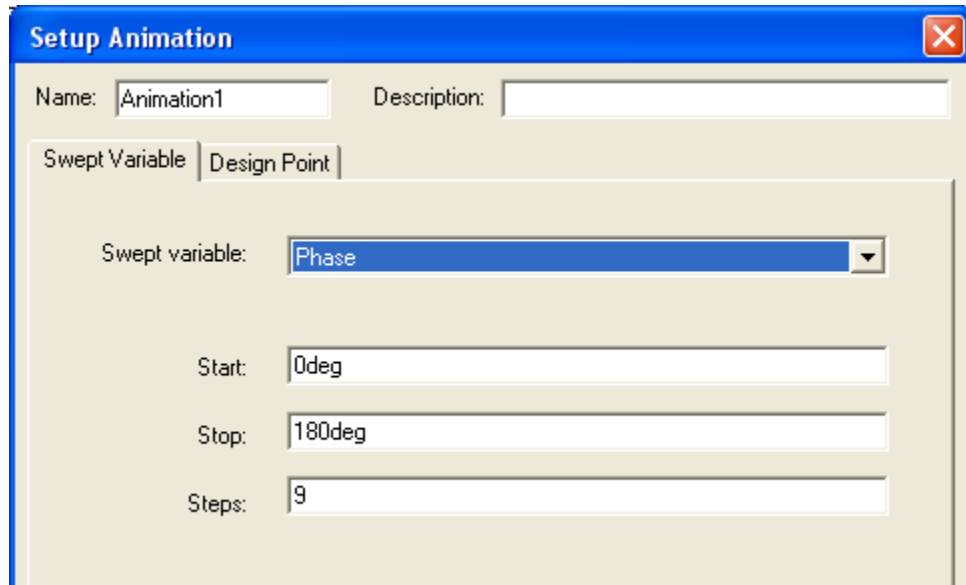
### Create a Phase Animation of the Mag E Plot

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

3D Modeler window  
defined 2-3  
location on desktop 2-2  
overview 3-3

---

## A

adaptive solution 5-2  
air volume 3-25  
assigning a color to 3-26  
assigning a transparency to 3-26  
assigning perfect E boundary to 4-4  
assigning radiation boundary to 4-3  
naming 3-26  
verifying its material 3-26  
analysis  
of solution 6-1  
results 1-5  
animation  
creating a phase animation 6-10  
annular feed ring 3-16  
assigning a color to 3-20  
assigning a transparency to 3-20

assigning perfect H boundary to 4-7  
naming 3-19

---

## B

boundaries  
about 4-2  
assigning 4-3  
assigning a perfect E boundary 4-4  
assigning a perfect H boundary 4-7  
assigning a radiation boundary 4-3  
assigning a symmetry boundary 4-8  
editing visualization settings of 4-4  
model-specific conditions 4-2  
solver view of 4-15  
verifying 4-15

---

## C

cavity  
assigning a color to 3-9  
assigning a transparency to 3-9

- naming 3-7
- splitting 3-7
- verifying its material 3-9
- clone before intersect 3-22
- context-sensitive help 1-iv, 2-7
- conventions used in guide 1-iii
- coordinate system
  - about 3-4
  - face type 3-4
  - global type 3-4
  - relative type 3-4
- copyright notice 1-ii

---

## D

- design, inserting 2-5
- dialog boxes
  - Create Field Plot 6-6
  - Edit Sweep 5-6
  - Element Length based refinement 5-8
  - Lumped Port wizard 4-11
  - Port Impedance Multiplier 4-14
  - Set Model Units 3-4
  - Setup Animation 6-11
  - Solution Setup 5-2
  - Solution Type 3-2
  - Split 3-8
  - Symmetry Boundary 4-10
  - Traces 6-2
  - View/Edit Material 3-15
- Do lambda refinement option 5-4
- DRA 3-11
  - assigning a color to 3-13
  - assigning a material to 3-13
  - assigning a transparency to 3-13
  - naming 3-11
  - splitting 3-12
- drawing
  - circles 3-17
  - polyhedrons 3-25
  - rectangles 3-21
  - setting the drawing units 3-4
  - spheres 3-6, 3-11

- the air volume object 3-25
- the annular feed ring object 3-16
- the DRA object 3-11
- the feed gap object 3-21
- drawing region 3-3

---

## E

- E-field
  - analyzing 6-6
  - animating 6-10
  - plotting 6-6
- elements, meshing 1-4
- excitations
  - about 4-3
  - assigning 4-11
  - impedance multiplier 4-13
  - model-specific conditions 4-3
  - solver view of 4-15
  - verifying 4-15

---

## F

- F1 key 2-7
- feed gap 3-21
  - assigning a color to 3-24
  - assigning a transparency to 3-24
  - assigning lumped port to 4-11
  - naming 3-23
- field overlay plots
  - creating 6-6
  - modifying attributes of 6-8
- finite element method 1-4
- frequency sweep
  - adding 5-6
  - defining the sweep 5-7
  - saving fields 5-7

---

## G

- geometric model
  - assigning symmetry boundary to 4-8

- fitting in drawing region 3-6
- sample problem 1-2
- solution parameters 5-2
- splitting for symmetry 3-27
- graphical user interface 2-2
- grid
  - setting its drawing units 3-4

---

## H

- help
  - accessing 2-7
  - Ansoft technical support 1-iv
  - context-sensitive 1-iv, 2-7
  - on dialog boxes 1-iv
  - on menu commands 1-iv
- HFSS
  - file extensions 2-6
  - inserting an HFSS design 2-5
  - interface overview 2-2
- history tree
  - defined 2-3
  - location on desktop 2-2

---

## I

- impedance multiplier
  - about 4-13
  - modifying 4-13
- infinite ground plane 4-6
- installation 2-1
- integration line, defining 4-12

---

## L

- lighting
  - ambient 3-9
  - disable attributes 3-9
  - distant 3-9
- lumped port 4-3
- lumped ports
  - and defining the integration line 4-12

- assigning 4-11
- editing 4-13
- impedance multiplier 4-13

---

## M

- mag E
  - field overlay plots of 6-6
- Mag E, plotting 6-6
- materials
  - assigning 3-13
  - creating new 3-13
  - verifying 3-9, 3-26
- menu bar
  - defined 2-4
  - location on desktop 2-2
- mesh
  - defining mesh operations 5-7
  - elements 1-4
  - example of 1-4
  - refining 5-7
  - seeding 5-7
- mesh operations 5-7
  - restricting length 5-8
- Message Manager window
  - defined 2-3
  - location on desktop 2-2

---

## O

- objects
  - air volume 3-25
  - annular feed ring 3-16
  - assigning color to 3-9, 3-13, 3-20, 3-24, 3-26
  - assigning lumped ports to 4-11
  - assigning materials to 3-13
  - assigning transparency to 3-9, 3-13, 3-20, 3-24, 3-26
  - DRA 3-11
  - feed gap 3-21
  - intersecting 3-22
  - naming 3-7, 3-11, 3-19, 3-23, 3-26

- selecting faces of 4-5
- sheet object defined 3-19
- splitting 3-8, 3-12
- splitting for symmetry 3-27
- subtracting 3-18

---

## P

- perfect E boundary
  - assigning 4-4
  - defined 4-2
- perfect H boundary
  - assigning 4-7
  - defined 4-2
- plots
  - animating 6-10
  - appearance in the project tree 6-3, 6-5
  - creating a cloud plot 6-6
  - creating a mag E plot 6-6
  - creating field overlays 6-6
  - modifying attributes of 6-8
- port impedance
  - imaginary 4-12
  - real 4-12
- profile data 5-12
- Progress window
  - defined 2-3
  - displaying solution progress 5-11
  - location on desktop 2-2
- project
  - validating 5-10
- Project Manager window
  - defined 2-3
  - location on desktop 2-2
- project tree
  - inserting an HFSS design 2-5
- projects
  - adding 2-5
  - adding project notes 2-6
  - creating 2-5
  - general procedure 1-3
  - saving 2-6
- Properties window

- defined 2-3
- location on desktop 2-2

---

## R

- radiation boundary
  - assigning 4-3
  - defined 4-2
- reports
  - animating 6-10
  - appearance in the project tree 6-3, 6-5
  - creating a cloud plot 6-6
  - creating S11 6-2
  - creating Z11 6-4
  - field overlay plots 6-6
  - generating 2D graphs 6-2
  - modal S-parameters 6-2
- rescale to new units 3-4

---

## S

- S11
  - creating report of 6-2
- select faces mode 4-5
- setting up 3-3
- simulation
  - running 5-11
- solution
  - adding a fast frequency sweep 5-6
  - adding to a design 5-2
  - analyzing 6-1
  - defining mesh operations 5-7
  - generating 5-11
  - specifying frequency 5-4
  - specifying lambda refinement 5-4
  - specifying maximum delta S 5-4
  - specifying maximum number of passes 5-4
  - specifying minimum converged passes 5-5
  - viewing convergence data 5-14
  - viewing matrix data 5-16
  - viewing profile data 5-12
- solution types

- Driven Modal 3-2
- Driven Terminal 3-2
- Eigenmode 3-2
  - selecting 3-2
- solutions
  - adaptive 5-2
  - adaptive analysis 5-11
  - adding a solution setup 5-2
  - and setting parameters 5-2
  - frequency 5-2
  - setting solution parameters 5-2
- status bar
  - defined 2-4
  - location on desktop 2-2
- subtracting
  - and blank objects 3-18
  - and tool objects 3-18
- symmetry boundary
  - assigning 4-8
  - defined 4-2

---

## T

- toolbars
  - defined 2-4
  - location on desktop 2-2
- trademark notice 1-ii

---

## U

- units, selecting 3-4

---

## V

- validation check 5-10

---

## W

- wave ports
  - impedance multiplier 4-13

---

## Z

- Z11
  - creating report of 6-4