Ansoft High Frequency Structure Simulator

Getting Started: An Eigenmode Problem

January 2001

Notice

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.

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

Printing History

New editions of this manual will 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 which 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. Note that pages which are rearranged due to changes on a previous page are not considered to be revised.

Edition	Date	Software Revision
1.0	October 1999	7.0
2.0	January 2001	8.0

Welcome!

This manual is a tutorial guide for setting up a simulation problem using version 8.0 of Ansoft HFSS, a software package for calculating the high-frequency electromagnetic behavior of a structure.

Ansoft HFSS Guides

There are several companion guides for Ansoft HFSS:

- *Getting Started: An Antenna Problem*
- *Getting Started: A Full-Wave Spice Problem*
- *Getting Started: An HFSS Optimetrics Problem*
- *Introduction to the Ansoft Macro Language*

For information on all of the Ansoft HFSS commands, refer to the Ansoft HFSS online documentation.

Installation Guide

Before you use Ansoft HFSS, you must:

1. Set up your system's graphical windowing system.
2. Install the Maxwell software, using the directions in the installation guide.

If you have not yet done these steps, refer to the installation guides and the documentation that came with your computer system, or ask your system administrator for help.

Other References

For detailed information on the Ansoft HFSS commands, refer to the Ansoft HFSS online documentation.

For detailed information on the Control Panel commands, refer to the Maxwell Control Panel online documentation.

Typeface Conventions

Computer	Computer type is used for on-screen prompts and messages, for field names, and for keyboard entries that must be typed in their entirety exactly as shown. For example, the instruction "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.
Menu/Command	Computer type is also used to display the commands that are needed to perform a specific task. Menu levels are separated by forward slashes (/). For example, the instruction "**Choose File/Open**" means to choose the **Open** command under the **File** menu.
Italics	Italic type is used for emphasis and for the titles of manuals and other publications. Italic type is also used for keyboard entries when a name or a variable must be typed in place of the words in italics. For example, the instruction "**copy** *filename*" means to type the word **copy**, to type a space, and then to type the name of a file, such as **file1**.
Keys	Helvetica type is used for labeled keys on the computer keyboard. For example, the instruction "Press **Return**" means to press the key on the keyboard that is labeled **Return**.

Using a Graphical User Interface

If you are familiar with the concepts of using a hand-held mouse, menus, and other graphical user interface (GUI) tools, skip to Chapter 1, “Introduction.”

If you have not used a GUI before, this section will help you understand some of the terminology used in this guide. Since GUIs are basically visual, the best way to learn to use them is by practicing on your system.

Most GUI systems use a “mouse” as a pointing device, with which you can select areas on the screen for command execution and moving from one program to another. Your mouse may have 2 or 3 buttons; Ansoft HFSS ignores the middle button on 3-button models, since Ansoft products do not use this button. You can program mouse buttons to work in non-standard ways, as you might want to if you are left-handed. For simplicity, the left-hand button (under your forefinger if you are right-handed) is referred to as the left mouse button, and the one on far right as the right mouse button. You will probably find the terms intuitive once you use these buttons a few times.

Point and Click; Right Click

To choose an item with the mouse, first move it on your desk until the *arrow* cursor is on that item; you are now “pointing” at the item. Next, press and release the left mouse button; this is called “clicking.” Point-and-click is the most common action you will make with your mouse. Generally, “click” refers to a *left* mouse button click.

Sometimes, you can use your *right* mouse button to access commands. In the 3D Modeler, a right mouse button click causes a menu of commands to appear. Generally, “right click” refers to a right mouse button click.

Double-Click

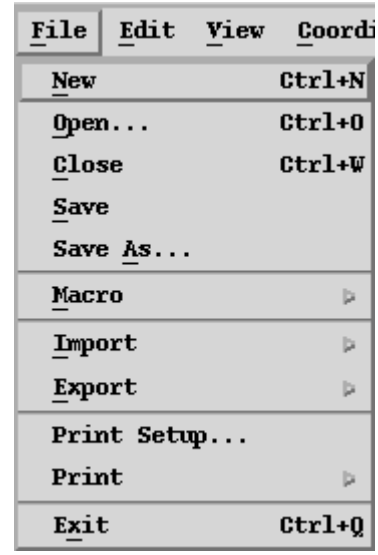
Occasionally you may want to select all of the text in a box, or perform a special task (say, indicating the end of drawing a line). You can do this by quickly clicking twice with your left mouse button — a “double-click.”

Dragging Objects; Click and Hold

When you are drawing in the 3D Modeler, you can often use your mouse to enter objects and move around the screen. Frequently, you will click the mouse button and *hold it down* until the next part of the command is reached (the object is moved, the next point is entered, and so forth). If you click and hold on the edge of a window, you can position, or *drag*, the window on your screen. You can often drag objects in the 3D Modeler; experiment to see what will move.

Menus

Within some screens of Ansoft HFSS are areas which list subsets, or “menus,” of commands. You can access a menu by clicking your mouse on the word or button that indicates the menu. The menu is “pulled down”, and lists the commands available on that menu. Usually, the menu will remain displayed until you choose a command, or click on the desktop to exit. If the menu does not remain displayed, click and *hold* the mouse button, then release the button to make your choice.



An arrow on the right side of a command indicates that there is a submenu for that command. An ellipsis (. . .) indicates that a pop-up window appears after choosing this command.

When you are asked to use a menu command, each level is separated by a “ / ”. For example, to zoom in on a drawing, you would choose the **View/Zoom In** menu command.

There are also pop-up menus, which appear when you right-click on a Maxwell modeler window. Choose commands from these menus in the same way as from menu bars.

For more information on using GUIs, refer to the online help for the Maxwell Control Panel.

Tool Bars

Tool bars are shortcut methods for entering commands. There are tool bars in many of the Ansoft HFSS modules for several commands. To use a tool bar, click the mouse cursor on the button you want to use. Here is an example of a tool bar:

- To execute a command, click on the appropriate button.
- To display a brief description of the command in the message bar, move the cursor to the tool bar icon and hold down the mouse button.

Note:

If a tool bar icon appears to do nothing when you click on it, the command may not be available at the time.

Table of Contents

1

Introduction

Ansoft HFSS is an interactive software package for calculating the electromagnetic behavior of a structure. The simulator also includes post-processing commands for analyzing the electromagnetic behavior of a structure in more detail. Using Ansoft HFSS, you can compute:

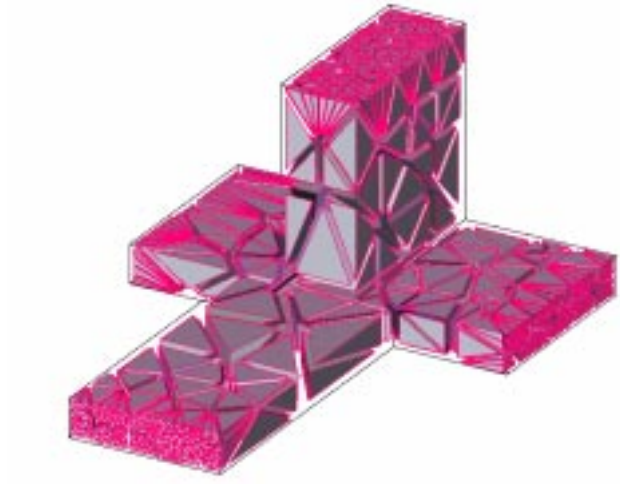
- Basic electromagnetic field quantities and, for open boundary problems, radiated near and far fields.
- Characteristic port impedances and propagation constants.
- Generalized S-parameters and S-parameters renormalized to specific port impedances.
- The eigenmodes, or resonances, of a structure.

You are expected to draw the structure, specify material characteristics for each object, and identify ports, sources, or special surface characteristics. The system then generates the necessary field solutions. As you set up the problem, Ansoft HFSS allows you to specify whether to solve the problem at one specific frequency or at several frequencies within a range.

Ansoft HFSS is available on UNIX workstations running X Windows and personal computers running Windows NT and 2000.

Finite Element Method

In Ansoft 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 a hybrid junction.

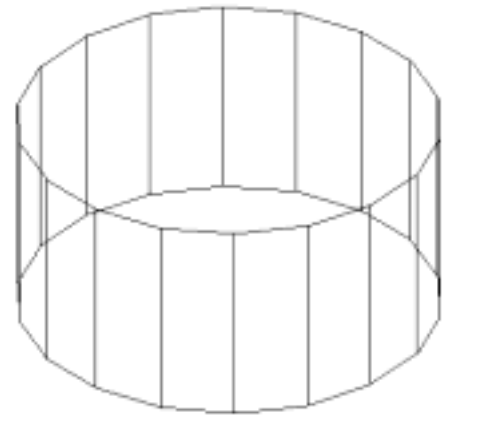


Dividing a structure into thousands of smaller regions (elements) allows the system to compute the field solution separately in each element. The smaller the system makes the elements, the more accurate the final solution will be.

The Sample Problem

This guide examines a cylindrical cavity surrounded by a material with a slight loss (copper). Use the Eigenmode solver to find the natural resonant frequencies.

The general geometry for this problem is shown below; detailed dimensions are given in Chapter 4, “Drawing the Geometric Model.”



Results to be Examined

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

- View the five eigenmodes and their associated Q factor.
- Plot the E field for two of the modes.
- Compare the vector plots of a mode and its degenerate mode.
- Create an animated vector plot.

Results to be Examined

2

Creating the Eigenmode Project

This guide assumes that Ansoft 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 project directory in which to save sample problems.
- Create a new project in that directory in which to save the eigenmode problem.

Time:

The total time needed to complete this chapter is approximately 10 minutes.

Access the Project Manager

To access Ansoft HFSS, you must first access the Maxwell Control Panel, which allows you to create and open projects for all Ansoft products.

➤ Access the **Project Manager**:

1. Do one of the following to access the Maxwell Control Panel:
   - On a UNIX workstation, enter the following command at the UNIX prompt:

     ```
     maxwell &
     ```

   - On the PC, click the left mouse button twice on the **Maxwell** icon.

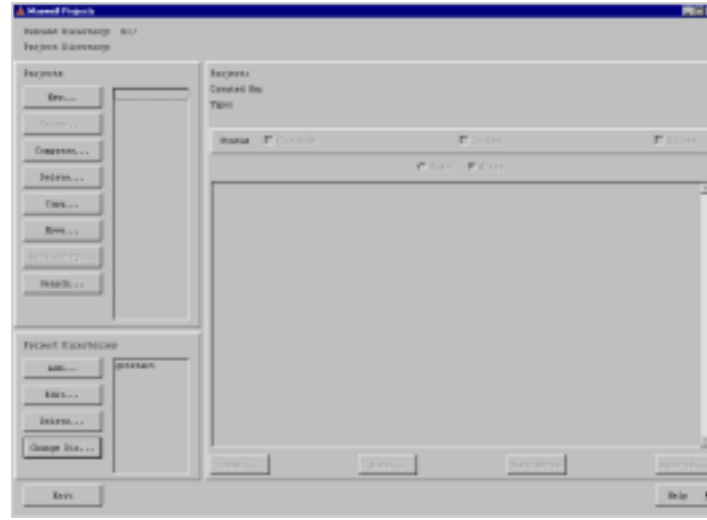
   The Maxwell Control Panel appears as shown below:

   If the Maxwell Control Panel doesn't appear, refer to the installation guide for possible reasons.
2. Click the left mouse button on the **Projects** button in the Maxwell Control Panel to access the **Project Manager**.

Note:

From now on, when you are asked to "choose" a button or a command, click the left mouse button on it.

The **Project Manager** appears as shown below:



Create a Project Directory

The first step in using Ansoft HFSS to solve a problem is to create a directory and a project in which to save all the data associated with the problem.

A project directory is a directory that contains a specific set of projects created with the Ansoft software. You can use project directories to categorize projects any number of ways. For example, you might want to store all projects related to a particular facility or application in one project directory. You will now create a project directory in that default directory.

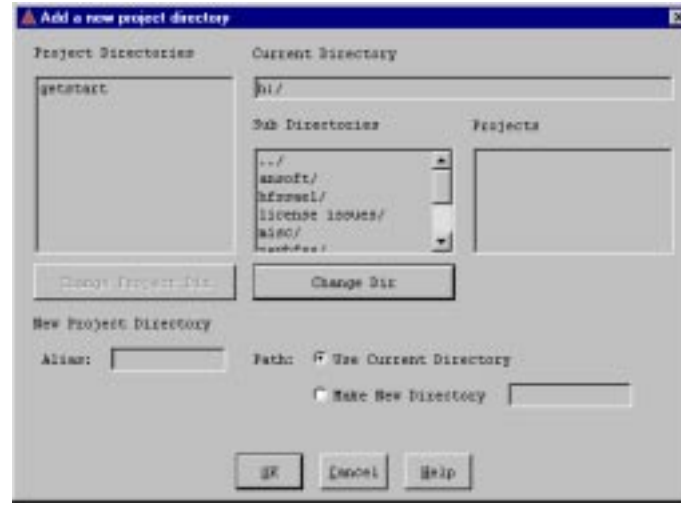
The **Project Manager** should still be on the screen. You will add the **getstart** directory for the Ansoft HFSS project you create using this *Getting Started* guide.

Note:

If you've already created a project directory while working through another *Getting Started* guide, skip to the "Create a Project" section.

➤ Add a project directory for the sample problem:

1. Choose **Add** in the **Project Directories** box at the bottom left of the window. The following window appears, listing directories and subdirectories:



2. Type the following in the **Alias** field:

   ```
   getstart
   ```

3. Choose **Make New Directory**.
4. Choose **OK**.

The directory **getstart** is created under the current default project directory, and **getstart** now appears in the **Projects** box. You return to the **Project Manager** window.

Note:

Ansoft HFSS projects are generally created in directories which have aliases — that is, directories that have been identified as project directories using the **Add** command.

➤ To change directories to look at another's contents:

1. Choose **Change Dir** from the **Project Directories** box.
2. Double-click the left mouse button on the desired directory. (Choose **../** to move up a level in the directory structure.)
3. After you are done, choose **OK**.

Refer to the Maxwell Control Panel online documentation for more information on changing directories and other **Project Manager** functions.

Create a Project

Now you are ready to create a new project named **eigen** in the project directory **getstart**.

Access the Project Directory

Before you create the new project, access the project directory **getstart**.

➤ Access the project directory:

- Choose **getstart** in the **Project Directories** box at the bottom left.

The current directory displayed at the top of the **Project Manager** menu changes to show the path of the directory associated with the alias **getstart**. If you have previously created a model, it will be listed in the **Projects** box. Otherwise, the **Projects** box is empty — no projects have been created yet in this project directory.

Add the New Project

➤ Add a new project:

1. Choose **New** in the **Projects** box at the top left of the menu. A window like the one shown below appears:

2. Type **eigen** in the **Name** field. Use the **Back Space** and **Delete** keys to correct typos.
3. To select the type of project to be created, click the left mouse button on the software package listed in the **Type** field. A menu appears, listing all of the Ansoft software packages you purchased. Click on **Ansoft High Frequency Structure Simulator 8** to select it as the project type.
4. Optionally, enter your name in the **Created By** field.

Note:

If you are running the software on a workstation, or on a PC using Microsoft Windows NT, the name of the person who logged onto the system appears by default.

5. Deselect **Open project upon creation**. When selected, this option opens the project after you choose **OK**. However, don't open the project quite yet.
6. Choose **OK** to create the project.

The information you just entered is now displayed in the corresponding fields in the **Selected Project** box. **Writable** is selected, showing that you have access to the project.

Save Project Notes

It is a good idea to save the notes about your new project so that the next time you use Ansoft HFSS, you can view information about a project without opening it.

➤ Enter notes for the **eigen** problem:

1. Leave **Notes** selected by default.
2. Click the left mouse button in the area under the **Notes** option. A cursor appears, indicating that you can now enter text.

Note:

The **Model** option displays a picture of the selected model in the **Notes** area. It is disabled now because you are creating a new project. After you create the **eigen** problem, its geometry will appear in this area by default when the **eigen** project is selected.

3. Enter your notes on the project, such as the following:

```
This sample eigenmode problem was created
using Ansoft HFSS and the HFSS Getting
Started guide.
```

   When you start entering project notes, the text of the **Save Notes** button (which is located below the **Notes** box) becomes enabled. Before you began typing in the **Notes** box, **Save Notes** was grayed out, or disabled.
4. When you are done entering the description, choose **Save Notes** to save it. After you do, the **Save Notes** option is grayed out again.

Note:

Grayed out text on commands or buttons means that the command or button is temporarily disabled.

You are now ready to open the new project and run Ansoft HFSS.

3

Run Ansoft HFSS

In the last chapter, you created the directory **getstart**, and created the project **eigen** within that directory.

This chapter describes:

- How to open the project you just created and run Ansoft HFSS.
- The Ansoft HFSS Executive Commands window.
- The general procedure for creating an eigenmode problem in Ansoft HFSS.
- The sample problem and the procedures you will use to calculate the field solution.

Time:

The total time needed to complete this chapter is approximately 10 minutes.

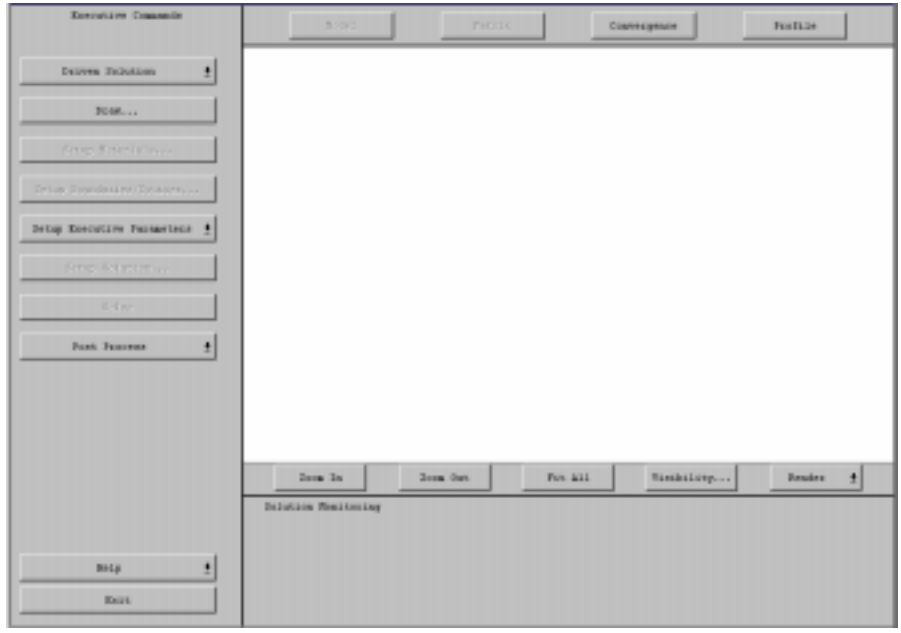
Open the New Project and Run the Simulator

The newly created project **eigen** should still be selected in the **Projects** box. (If it is not, move the cursor on it and click the left mouse button.)

➤ Run Ansoft HFSS:

- Choose **Open** from the **Projects** box.

The Executive Commands window of Ansoft HFSS appears as shown below:



Overview of the Executive Commands Window

The Executive Commands window acts as a doorway to each step of creating and solving the model problem. You select each module through the Executive Commands menu and the software brings you back to this window when you are finished using that module. You also view the solution process through this window. The Executive Commands window is divided into two sections: the commands area and the display area.

Commands Area

The commands area, located on the left side of the screen, contains the menu that lets you define the type of problem you are solving and then call up the various modules you will need to create and solve the problem. The function of each command will be explained in the section, "General Procedure for Drawing and Solving."

Display Area

The display area shows the project's geometric model. Since you haven't created the model's geometry yet, this area is blank.

The buttons along the top of the window do the following:

Model	Displays the geometric model of the 3D structure. The **Zoom In**, **Zoom Out**, **Fit All**, **Visibility**, and **Render** buttons are only available when **Model** is selected.
Matrix	Allows you to view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive solution.
Convergence	Displays the convergence information.
Profile	Displays the computing resources used in the solution process.

The buttons along the bottom of the window allow you to change your view of the model as follows:

Zoom In	Zooms in toward the object, expanding the view of the model.
Zoom Out	Zooms away from the object, shrinking the view of the model.
Fit All	Fits the entire model in the view window.
Visibility	Displays parts of the model in the view window.
Render	Shows the model as wireframe, flat shaded, or smooth shaded.

This area also displays solution profile and convergence information while the problem is solving, as described in Chapter 6, "Generating a Solution."

General Procedure for Drawing and Solving

➤ Use this general procedure to define and solve a problem:

1. Select the solver. You may select either the **Driven Solution** solver, or the **Eigenmode Solution** solver.
2. Use the **Draw** command to access the 3D Modeler and draw the objects that make up the geometric model.
3. Use the **Setup Materials** command to assign material characteristics to each 3D object in the geometric model.
4. Use the **Setup Boundaries/Sources** command to define the location of boundaries and sources.
5. Use the **Setup Solution** command to specify how the solution to the problem is to be computed. For eigenmode problems, use this command to:
   - Specify the minimum frequency at which the solver begins looking for eigenmodes and the number of eigenmodes to find.
   - Specify solution criteria, such as the number of passes, the percentage of tetrahedra refinement, and so forth.
   - Select which finite element mesh is used for the solution.
6. Use the **Solve** command to solve for the fields associated with the eigenmode problem.
7. Use the **Matrix** (**Eigen Modes** for eigenmode problems), **Convergence**, and **Profile** buttons to view the following information:
   - The parameters computed during each adaptive solution.
   - The difference in parameters between each adaptive solution.
   - Statistics on CPU use, memory size, and so forth.
8. Select one of the following Post Processors from the **Post Process** menu:

   Fields — Accesses the 3D Post Processor which allows you to display contour, shaded, vector, value versus distance, and animated plots of fields.

   Matrix Data — Accesses the Matrix Data Post Processor which allows you to view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive solution. This is not available for eigenmode problems.

   Matrix Plot — Accesses the Matrix Plot Post Processor which allows you to plot a variety of parameters versus time or frequency. This is not available for eigenmode problems.

These commands must be chosen in the sequence in which they appear. For example, you must first create a geometric model with **Draw** before you specify material characteristics for objects with **Setup Materials**. A checkmark appears on the menu next to each step as it is completed.

4

Drawing the Geometric Model

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

- Set up the problem region.
- Create the object that makes up the eigenmode model.
- Save the model to a set of disk files.

You are now ready to start using the simulator. The Executive Commands window should still be on the screen.

Time:

The total time needed to complete this chapter is approximately 10 minutes.

Select the Solver

For this problem, select the Eigenmode solver.

➤ Select the solver:

- Select **Eigenmode Solution** from the Solver menu. (Initially, **Driven Solution** appears as the selected solver.)

The name of the menu changes to reflect the currently selected solver.

Driven Solution

Select **Driven Solution** to use the finite element-based solver to generate a solution for a structure that is "driven" by a source. Ansoft HFSS calculates the S-parameters of passive, high-frequency structures such as microstrips and transmission lines. The software includes post-processing commands for analyzing the electromagnetic behavior of a structure in more detail.

Using the **Driven Solution**, you can compute:

- Basic electromagnetic field quantities and, for open boundary problems, radiated near and far fields.
- Characteristic port impedances and propagation constants.
- Generalized S-parameters and S-parameters renormalized to specific port impedances.

Eigenmode Solution

Select **Eigenmode Solution** to calculate the eigenmodes, or resonances, of a structure. The eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

The Ansoft HFSS eigenmode solver can find the eigenmodes of lossy as well as lossless structures, and can calculate the unloaded Q of a cavity. Q is the quality factor, and is a measure of how much energy is lost in the system. Unloaded Q is the energy lost due to lossy materials. Because ports and other sources are unavailable for eigenmode problems, the calculated Q does not include losses due to those sources.

The following restrictions apply to Eigenmode solutions:

- Emissions may not be calculated.
- The following boundary conditions may not be defined:
  - Port
  - Incident Wave
  - Voltage Drop
  - Current
  - Magnetic Bias
  - Radiation
- Fast frequency, interpolating, and discrete frequency sweeps are not available.

- Nonlinear materials may not be used.
- **Matrix Data** and **Matrix Plot** are not available, and the **Matrix** button on the Executive Commands window changes to **Eigen Modes**.

Start the 3D Modeler

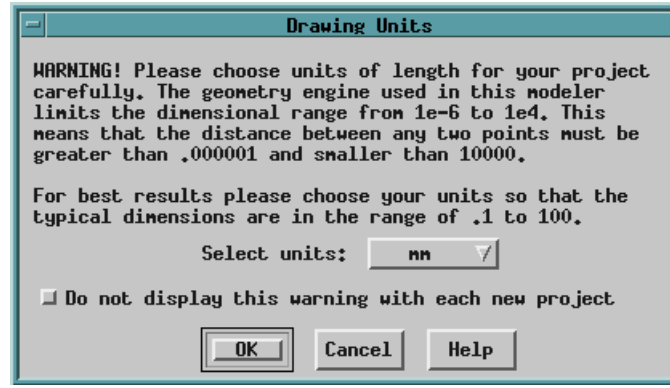
To draw the geometric model, use the 3D Modeler, which is the portion of Ansoft HFSS that allows you to create objects.

When you start the 3D Modeler, four distinct windows — known as view windows — appear. Three of these windows show 2D views of the model you are creating, while the fourth window displays a full 3D view.

A dot in each window shows the position in the model; this dot is blinking in the active window. To draw in a window, activate it by clicking in that window. You can tell which coordinate system a window uses by moving the cursor over that window and looking at the letters next to the cursor.

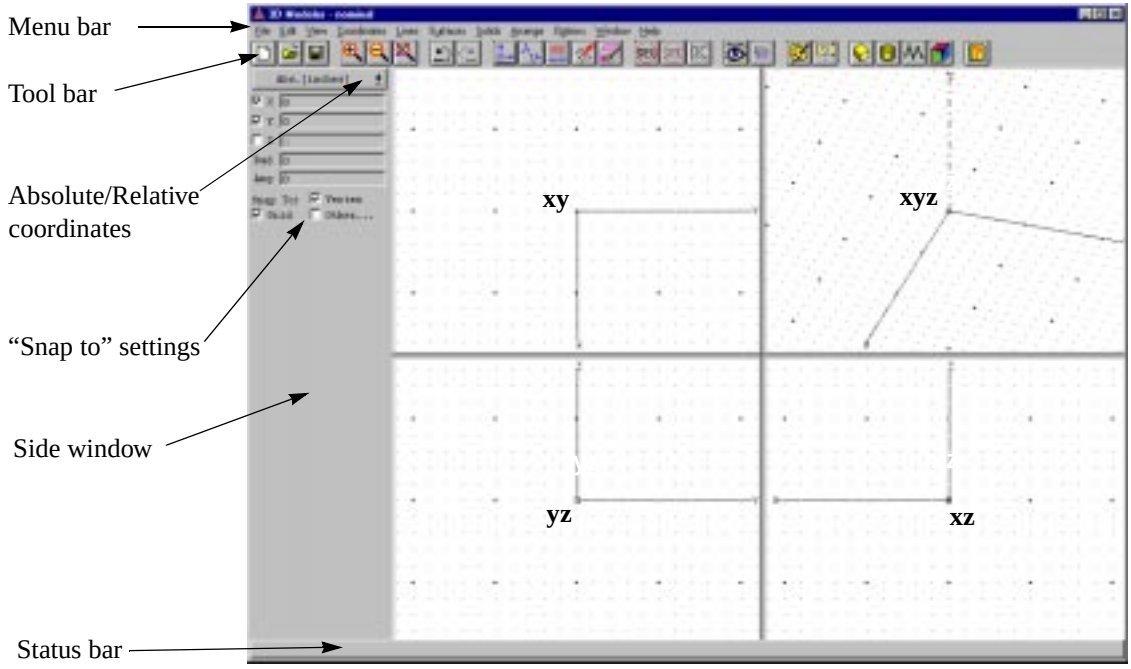
➤ Start the 3D Modeler:

1. Choose **Draw** from the top of the Executive Commands menu. The 3D Modeler window appears.
2. You are prompted to choose the units of length you want to use:



   Select **cm** from the menu next to **Select units**.
3. Choose **OK** to accept centimeters as the units for this problem. The units chosen are shown at the **Absolute/Relative** coordinates menu.

The 3D Modeler window is divided into several parts:



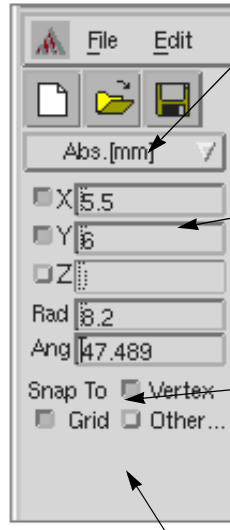
Note:

If you require further information on any topic in Ansoft HFSS, such as the 3D Modeler commands or windows, there are several options for displaying context-sensitive help:

- Choose **Help** in a window.
- Press **F1**, see the cursor change to **?**, then click on the item on which you need help.
- Use the commands from the **Help** menu.

Side Window

The area on the left of the 3D Modeler (or on the right, depending on your preference settings) is referred to as the *side window*. The side window is where you can change the coordinates or set the snaps of the model to something other than the default. This window is also where many command-specific fields appear.



Use this menu to select the type of coordinate system to use. You may select from an absolute coordinate system or a relative coordinate system. The selected coordinate system appears as the name of the menu.

Use these fields to enter the x-, y-, or z-coordinates and the radius, distance, or angle. Notice the checkbox next to the coordinate fields. The checkbox must be selected to enable the coordinate field. These coordinate fields are used to enter the coordinates for a variety of commands.

Use these checkboxes to select the type of "snap-to" you wish to employ when selecting objects or object artifacts (vertices, lines, faces, and so forth). When you select the **Other** checkbox, a window appears allowing you to select from a variety of "snap-to" options.

Use the blank area under the coordinate section for entering information for many commands. Fields that appear in this area allow you to enter information specific to the command you just selected.

Snaps

The **Grid** and **Vertex** snap-to settings are set by default and already active. You do not need to specify the settings for this problem.

➤ To select the snap-to behavior:

1. Choose **Other** from the **Snap to** buttons. A window appears below the coordinates fields.
2. Select the type of **Edge Snap** you prefer. You may select from the following:

   Grid inters. Allows you to set the snap at the point where the grid intersects an axis.

   Edge center Allows you to set the snaps at the central points of the edges.

   Arc center Allows you to set the snap at the center of an arc.

3. Select the type of **Face Snap** that you prefer. You may select from the following:

   Axis inters. Allows you to set the face snap at the point where an axis crosses the face of an object.

   Face center Allows you to set the snap at the center of the face of an object.

4. Choose **OK** to accept the snap-to behavior.

Define the Drawing Region

The area containing the model is called the *drawing region*; the four view windows provide you with different perspectives of this area, which is initially empty except for the coordinate axis.

Note:

Because the window can be customized, it may differ slightly from those shown in this guide. For instance, if someone has used the 3D Modeler before you, they may have changed the default position of the tool bar. Don't worry if this happens; simply use the windows as they appear in the program. This guide reminds you to check when default settings are needed for your project.

In order to draw the model, you must activate a window in which to create it.

➤ Activate the window in which you will draw the model:

- Click in the bottom left (yz) window. The cursor is tagged with the current window's coordinate system. This window is where you will begin to draw your model.

Absolute and Relative Coordinates

In this guide, you will be working in absolute coordinates. If relative coordinates are set, the coordinate system measures from an origin defined by you, which you can change. If absolute coordinates are set, the coordinate system measures from a system-defined origin.

➤ Check the **Absolute/Relative** coordinates menu at the top of the side window.

- If absolute coordinates are *not* selected, select **Absolute** from the **Rel. [cm]** menu in the side window.

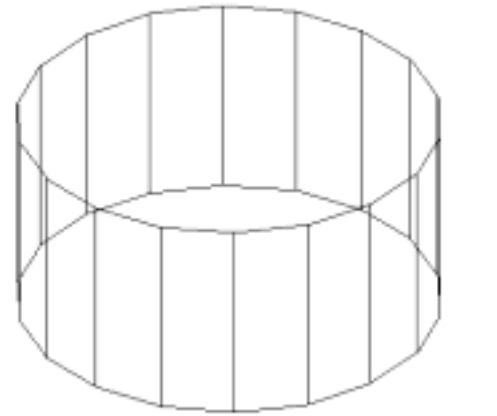
Grids

The 3D Modeler uses grid settings to provide a visual guide as you draw objects. There are no *particular* unit types associated with the grid points. However, you can set up the grid so that each grid point is displayed at a given number of units. The default settings keep the grid points at about 30 pixels apart, even if you zoom in and out, and start with each grid point displayed at 20 cm.

Now you are ready to begin drawing the object that makes up the geometry.

Create the Geometry

The geometry for the eigenmode problem consists of a single object: a cylinder, which represents a circular cavity.

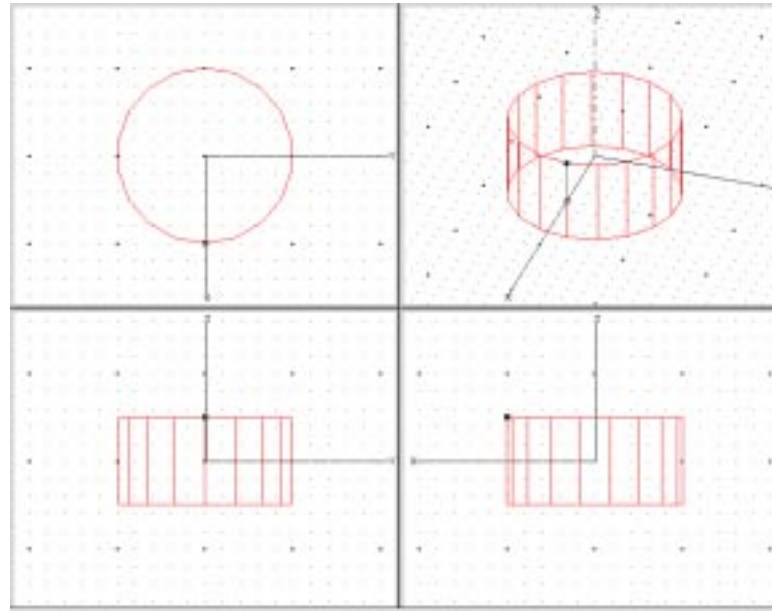


For this problem, draw a cylinder with a radius and height of 10 cm.

➤ Draw the cylinder:

1. Choose **Solids/Cylinder** to create a cylinder. The controls for entering the base center and selecting the axis for the cylinder appear in the side window.
2. Leave the **X** and **Y** fields set to zero. If they are not set to zero, enter **0** in the **X** and **Y** fields.
3. Enable the **Z** field (if it is not enabled) and enter **-5** in it.
4. If it is not already, set the **Cylinder Axis** to the z-axis.
5. Choose **Enter** to set the center of the base, and the axis of the cylinder. The 3D Modeler accepts the settings, and fields for entering the cylinder information appear in the side window.
6. Enter **10** in the **Radius** field under **Radius & Height**.
7. Enter **10** in the **Height** field under **Radius & Height**.
8. Leave **Num segments** selected. This allows you to approximate the shape of a cylinder using line segments (facets). Otherwise, it is treated as a true surface, which requires greater computing resources.
9. Enter **18** for the number of segments.
10. Enter **pillbox** in the **Name** field.
11. Click on the colored box next to **Color**. Select red from the color palette that appears.
12. Choose **Enter**. The cylinder appears in all the windows; however, it is too small to see clearly.
13. Press **f**, which is the hotkey for **View/Fit All/All Views**. The view changes, making the cylinder easier to see.

The completed geometry appears as shown below:



Exit the 3D Modeler

You are now finished drawing the model.

➤ Exit the 3D Modeler:

1. Choose **File/Exit**. A window appears, prompting you to save your geometry and verify the model.
2. Choose **Yes**. A progress bar appears displaying the progress of the system as it verifies that there are no overlapping objects present in the model.

You return to the Executive Commands window and are now ready to assign materials and define boundaries.

5

Setting Up the Problem

Now that you have created the geometry for the eigenmode problem you are ready to define material properties.

Your goals for this chapter are to:

- Assign material properties to the cylinder.
- Assign a Finite Conductivity boundary to the outside of the pillbox to simulate a lossy cavity.

Now you are ready to set up the problem.

Time:

The total time needed to complete this chapter is approximately 5 minutes.

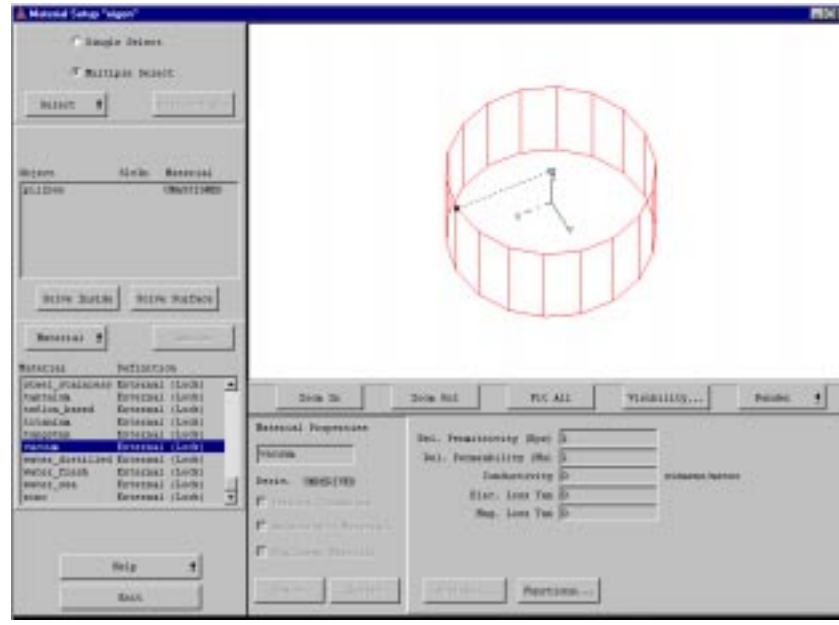
Assign Materials

To completely set up this eigenmode problem, you must assign material characteristics to each 3D object in the geometric model.

Start the Material Manager

➤ Start the Material Manager:

- Choose **Setup Materials**. The **Material Setup** window appears as shown below:



All objects — in this case **pillbox** — are listed in the **Object** box, and the materials in the material database provided with the software are listed in the **Material** box. The characteristics of the materials are listed under **Material Attributes**.

Assign Vacuum to the Pillbox

Assign a material with the properties of a vacuum to the pillbox.

➤ Assign vacuum to **pillbox**:

1. Select **pillbox** from the **Object** list. The object is highlighted to indicate that it is selected.
2. Select **vacuum** from the **Material** list. Use the scroll bars to navigate through the list.
3. Choose **Assign**. The material is assigned to the pillbox and its name appears under **Material** in the **Object** list.

Notice that when you assigned the material, **Solve Inside** (indicated as **SlvIn** in the **Object** list) was set to **Yes.** This indicates that Ansoft HFSS will create a mesh inside the object and generate a solution from the mesh. By default, **SlvIn** is automatically set to **Yes** for all objects with a conductivity less than 10^5 siemens / meter.

Exit Setup Materials

➤ Exit the Material Manager:

1. Choose **Exit**. You are prompted to save the material assignments.
2. Choose **Yes**. The material assignments are saved.

You return to the Executive Commands window, and a checkmark is displayed next to **Setup Materials**, indicating that all objects have been successfully assigned material characteristics.

Set Up Boundaries

After you define the material properties, you must define the boundary conditions. Because this cavity is enclosed in a lossy conductor, the default boundary automatically defined for the portion of the cylinder exposed to the background object must be changed to simulate a lossy material, such as copper.

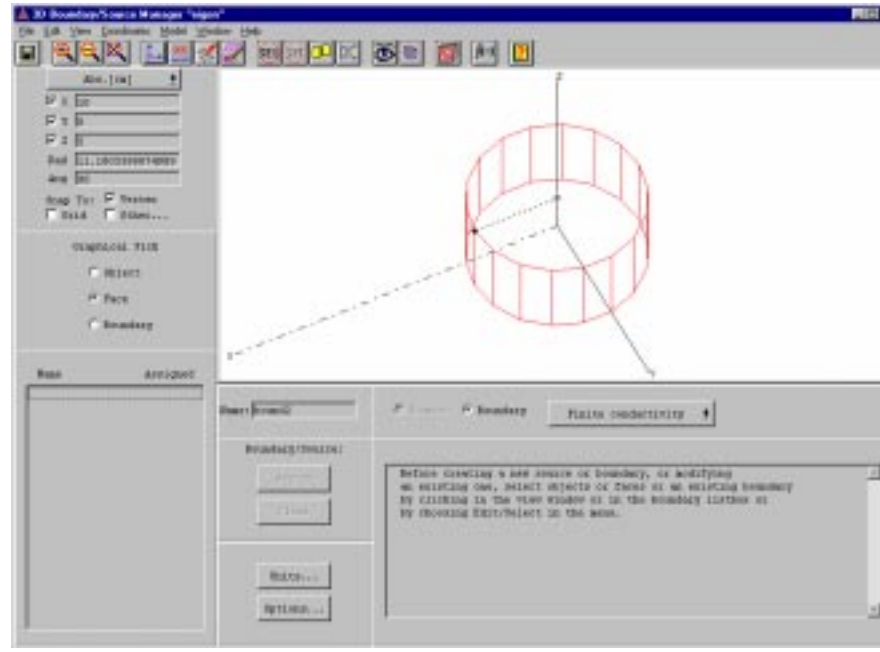
Note:

By default, any object exposed to the background is assigned a perfect **E** boundary. Because this boundary is automatically assigned, a check mark appears next to **Setup Boundaries/Sources** after you finish assigning materials.

Start the 3D Boundary Manager

➤ Start the 3D Boundary Manager:

- Choose **Setup Boundaries/Sources**. The **3D Boundary/Source Manager** window appears:



Define the Finite Conductivity Boundary

Finite conductivity boundaries can be assigned to surfaces that represent imperfect conductors. The E-field has a tangential component at the surface of imperfect conductors, which simulates the case in which the surface is lossy. The amount of loss will be equal to the component of **E**x**H*** that flows into the surface.

The surfaces of any objects defined to be non-perfect conductors are automatically set to finite conductivity boundaries. Note that the system does not attempt to compute the field inside these objects; the finite conductivity boundary approximates the behavior of the field at the surfaces of the objects.

Note:

The finite conductivity boundary condition is valid only if the conductor being modeled is a "good" conductor — generally defined as greater than 10^5 siemens / meter.

➤ Define the surface **pillbox** as a finite conductor:

1. Under **Graphical Pick**, select **Object**. This allows you to select the surface of the entire object, instead of having to select each face.
2. Select the pillbox by clicking on it.
3. Select the button next to the **Boundary** field as the type of condition to apply to the objects.
4. Select **Finite conductivity** from the menu of boundary types. The **Conductivity** and **Permeability** fields appear.
5. Leave the conductivity set to **58000000** and the permeability set to **1**. This is equivalent to surrounding the cavity with copper.
6. Leave the name of the boundary set to its default.
7. Choose **Assign**. The new boundary appears in the boundary list as **bound1**.

Exit Setup Boundaries/Sources

➤ Exit the Boundary Manager:

1. From the menu bar, choose **File/Exit**. You are prompted to save the boundary assignments.
2. Choose **Yes**. The boundary assignments are saved.

You return to the Executive Commands window, and a checkmark is displayed next to **Setup Boundaries/Sources**, indicating that all objects have been successfully assigned boundaries.

6

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 as follows:

- Set up the solution parameters that will be used in calculating the solution.
- Generate a solution.
- View the convergence and eigenmode information.

Time:

This problem solved in approximately 12 minutes on an Pentium II PC with 128 megabytes of RAM. Depending on the computing resources you have available, this solution time may vary greatly.

Specify Solution Options

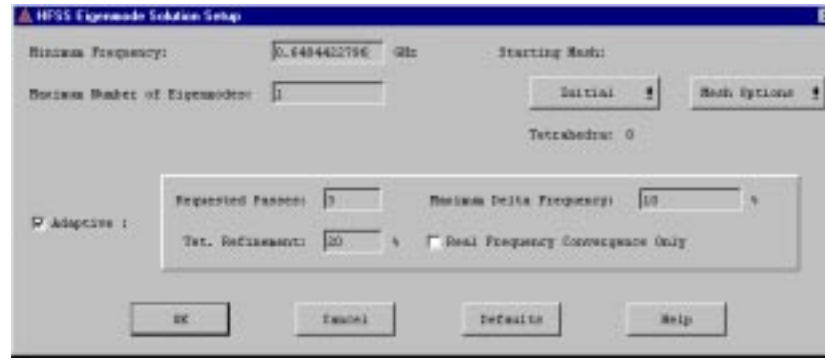
Before you can generate a solution for the eigenmode problem, you need to specify the solution parameters. This controls how the software computes the requested solution.

For the eigenmode problem, specify the following solution criteria:

- Specify that the solver find 5 resonances starting at a minimum frequency of 0.6484422796 GHz. This value was automatically calculated by the software based on the problem geometry.
- Perform 8 adaptive passes.
- Specify that the maximum change in the frequency from one pass to the next be less than or equal to 0.1 %.

➤ Set up the solution criteria for the eigenmode problem:

1. Choose **Setup Solution**. The **HFSS Solution Setup** window appears as shown below:



2. Leave the **Minimum Frequency** field set to **0.6484422796**. This is the frequency at which the eigenmode solver begins the search for eigenmodes. The solver searches for the user-specified number of modes with a higher resonant frequency than the **Minimum Frequency**.

Warning:

Because the **Minimum Frequency** is used to normalize some matrices, if the frequency is set too low, the solver ends up trying to solve a nearly-singular matrix, which may erode the accuracy of the calculations. As a general rule, this may happen when the minimum frequency entered is less than 0.01 times the suggested, or default, value for the **Minimum Frequency**.

3. Enter **5** in the **Maximum Number of Eigenmodes** field. This is the number of eigenmode solutions that the solver finds above the **Minimum Frequency**. The solver can only find up to 20 eigenmode solutions.

4. Leave **Adaptive** selected. This instructs Ansoft HFSS to solve the problem iteratively, refining the regions of the mesh in which the largest error exists. Use the following adaptive solution criteria:
   - Enter **8** in the **Requested Passes** field. In this case, because **Requested Passes** is set to **8**, the simulator generates up to eight successive solutions for the problem, refining the mesh before it generates the next solution.
   - Enter **25** in the **Tet. Refinement** field. This determines the percent increase in the number of tetrahedra after each iteration.
   - Enter **0.1** in the **Maximum Delta Frequency** field. This criterion examines the change in the computed frequencies for each eigenmode from one adaptive pass to the next. For this problem, five resonant frequencies are calculated for one mesh, the mesh is refined, and the five frequencies are calculated for the refined mesh. For each of the five modes, the difference between the frequencies calculated for these two meshes is computed. The largest among these differences is the delta frequency. If the delta frequency is less than the **Maximum Delta Frequency**, the problem has converged.

     For example, in a two-mode problem, if the difference between mode 2 in the first pass and mode 2 in the second pass is greater than the **Maximum Delta Frequency**, the problem is not converged, even if the difference between mode 1 in the first pass and second pass is less than the **Maximum Delta Frequency**.

Note:

The solution process stops if either the **Requested Passes** or **Maximum Delta Frequency** is reached.

5. Leave the other parameters set to their defaults. For an explanation of these parameters, refer to the Ansoft HFSS online documentation.
6. Choose **OK**.

The criteria are saved and you return to the Executive Commands window. You are now ready to generate a solution.

Generate the Solution

Now that you have entered the solution criteria and specified the solution frequency, the problem is ready to be solved.

Note:

Depending on how closely you followed the guide, 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.

➤ Generate a solution:

- Choose **Solve** from the Executive Commands window. A sequence of progress bars appears in the **Solution Monitoring** area as the solution is being generated. These allow you to monitor the progress of the solution process. Note the descriptive messages that accompany the progress bars. These describe the various stages of the solution process.

  When the solution is complete, the following message appears:

  ```
  Solution process complete
  ```

- Choose **OK**.

You remain in the Executive Commands window. A checkmark appears next to **Solve**, indicating that the solution was completed successfully.

Note:

After a solution is generated, the system does not allow you to change the geometry, material properties, or boundary conditions unless you first delete the solution. For example, if you change the geometry of the problem after generating a solution, you must generate a new solution after you save the changes to the new geometry.

View Convergence

While the solution is generating, you can view information such as the following:

- Number of passes completed and remaining.
- The number of tetrahedra per pass.
- The maximum change in the frequency per pass.

➤ View the convergence:

- Choose the **Convergence** button from the row of buttons across the top of the Executive Commands window. The view of the model disappears and is replaced by the **Convergence Data** window.

Take a few moments to look over the convergence information. Note that the frequency change is much larger in the earlier passes than in the later ones.

View the Results

Use the **Eigen Modes** button from the top of the Executive Commands window to view the eigenmodes and the quality factor Q.

Eigen Modes

Eigenmodes are the resonances of the structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

For a Driven Solution, Ansoft HFSS solves the following matrix equation (for a lossless case):

$$S\boldsymbol{x} + k_o^2 T\boldsymbol{x} = b$$

where:

- S and T are matrices that depend on the geometry and the mesh.
- $\boldsymbol{x}$ is the electric field solution.
- k_o is the free-space wave number.
- b is the value of the source defined for the problem.

However, in order to find the resonances of the structure, the eigenmode solver sets b to zero, and solves the equation:

$$S\boldsymbol{x} + k_o^2 T\boldsymbol{x} = 0$$

for sets of $(k_o,\boldsymbol{x})$, one k_o for every $\boldsymbol{x}$. The variable $\boldsymbol{x}$ is still the electric field solution, and ko is the free space wave number corresponding to that mode. The wave number is related to the frequency of the resonant modes through the following:

$$f = \frac{k_o c}{2\pi}$$

where c is the speed of light.

Quality Factor

Q is the unloaded quality factor, and is a measure of how much energy is lost in the system due to lossy materials. Because ports and other sources are restricted for eigenmode problems, the calculated Q does not include losses due to those sources.

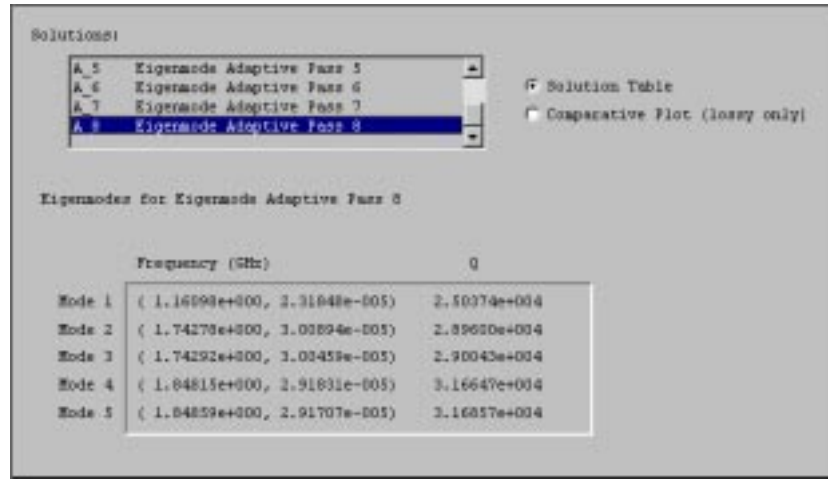
The quality factor calculated by Ansoft HFSS is an approximation. It is accurate for problems with small to moderate losses, when the imaginary part of the frequency is less than 0.1 times the real part. Ansoft HFSS uses the following equation to calculate the approximate quality factor:

$$Q = \left| \frac{Re(freq)}{2 \cdot Im(freq)} \right|$$

View the Eigenmodes

➤ View the eigenmodes and Q factor:

- Choose **Eigen Modes** from the top of the Executive Commands window. The following window appears:



For this problem, the analytical results for a cavity with these dimensions with boundaries having a conductivity sigma = 5.8e+7 siemens / meter are shown below. Notice the degeneracy (the second, imaginary part of frequency) between Mode 2 and 3, and Mode 4 and 5:

Mode	Theoretical Frequency (real part)	Theoretical Q
Mode1	1.1483	25,638
Mode 2	1.7386	29,220
Mode 3	1.7386	29,220
Mode 4	1.8296	32,363
Mode 5	1.8296	32,363

The results calculated by Ansoft HFSS are shown below:

Mode	Frequency (real part)	Q
Mode1	1.16098	25,037
Mode 2	1.74278	28,960
Mode 3	1.74292	29,004
Mode 4	1.84815	31,665
Mode 5	1.84859	31,686

Note:

The calculated values differ slightly from the analytical values. The accuracy, while good, can be improved by using a true-surface cylinder instead of a segmented cylinder. The results for a true-surface cylinder are shown below:

Mode	Frequency (real part)	Q
Mode1	1.15190	25,483
Mode 2	1.73223	29,472
Mode 3	1.73628	29,319
Mode 4	1.82918	32,069
Mode 5	1.83003	32,084

Using a true surface increases the solution time by approximately 50% for this particular case.

7

Analyzing the Solution

Now that you have generated a solution for the eigenmode problem, you can analyze it using Ansoft HFSS's post-processing features, accessed with the **Post Process** menu.

You will:

- Plot the E-field for Mode 1 and Mode 4.
- Compare the vector plots of **E** for Mode 2 and its degenerate mode, Mode 3.
- Create an animated vector plot of **E**.

Time:

The total time needed to complete this chapter is approximately 30 minutes.

The Post Processors

Ansoft HFSS provides three Post Processors for analyzing the results of your solutions. The following Post Processors are available from the **Post Process** menu:

Fields	Accesses the 3D Post Processor which allows you to display contour, shaded, vector, value versus distance, and animated plots of fields.
Matrix Data	Accesses the Matrix Data Post Processor which allows you to view matrices computed for the S-parameters, impedances, and propagation constants during each adaptive solution. This is not available for eigenmode problems.
Matrix Plot	Accesses the Matrix Plot Post Processor which allows you to plot a variety of parameters versus time or frequency. This is not available for eigenmode problems.

In this guide you will use the 3D Post Processor.

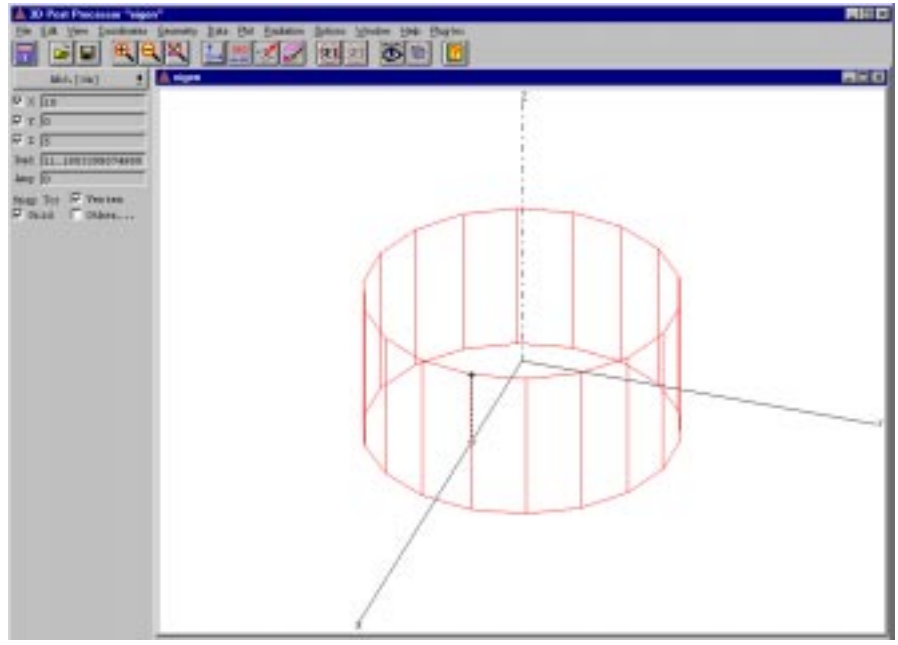
Access the 3D Post Processor

The 3D Post Processor can:

- Display shaded or wire frame views of the solved model.
- Display contour, shaded, vector, value vs. distance, and animated plots of fields.
- Compute derived quantities from the general field solution.

➤ Access the 3D Post Processor:

- Choose **Post Process/Fields**. The following window appears:



Plot the E-field for Mode 1 and Mode 4

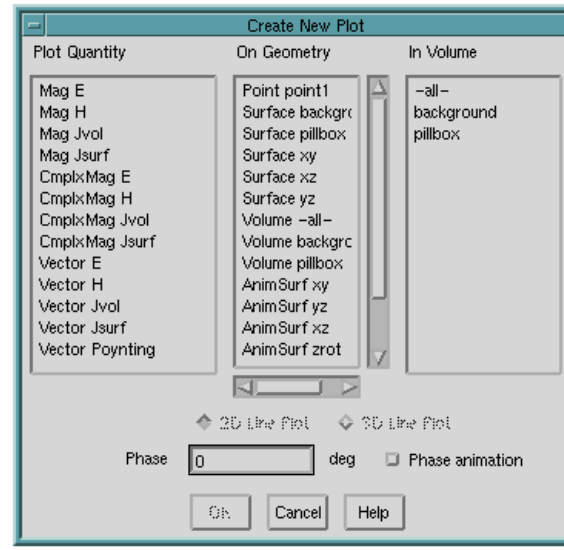
By default, the fundamental mode (Mode 1) is automatically loaded into the 3D Post Processor. This means that when you create a plot, the fields calculated for the fundamental mode are the fields that are plotted. To switch modes, use the **Data/Edit Sources** command.

Plot the E-field for Mode 1

Create a shaded plot in the xy plane of the magnitude of **E** for the fundamental mode.

➤ Plot mag **E** for Mode 1:

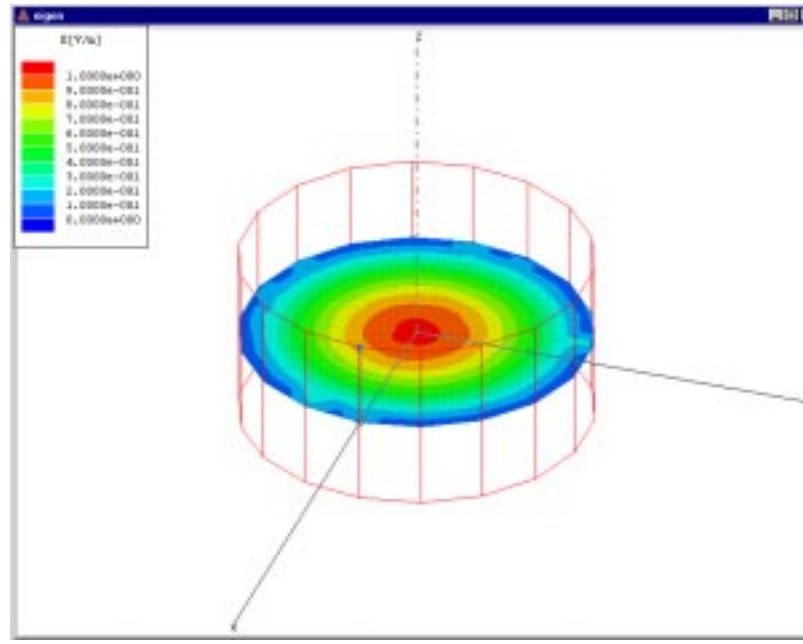
1. Choose **Plot/Field** to generate a new plot. The following window appears:



2. Select **Mag E** from the **Plot Quantity** list. This selects the magnitude of the electric field $|\mathbf{E}|(x,y,z,t)$ as the quantity to plot.
3. Select **Surface xy** from the **On Geometry** list. This selects the geometry on which to plot. For this example, you will be plotting on the xy plane (z=0).
4. Select **pillbox** from the **In Volume** list. Since only the fields inside the cavity are of interest, there is no need to plot over the entire problem extent.
5. Leave the other settings set to their default settings.

6. Choose **OK**. The following window appears, allowing you to specify the attributes of the plot:

7. Make sure **Use Limits** is selected with a range of **1** and **0**, then choose **OK**. A progress bar appears as Ansoft HFSS generates a plot. The following plot appears:



Plot the E-field for Mode 4

Now that you have created a plot of the magnitude of **E** for the fundamental mode, you can display a similar plot for another mode without creating a new plot. Use the **Data/Edit Sources** command to change the mode from which the fields are taken. When you do this, the currently displayed plot can be updated automatically to use the fields from the new mode.

➤ Display the mag **E** plot for Mode 4:

1. Choose **Data/Edit Sources**. The following window appears:

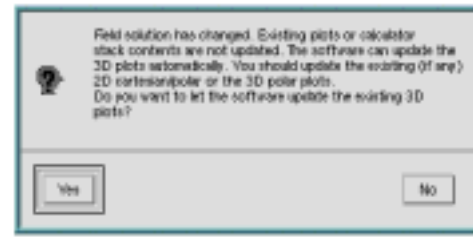
   Notice that **Eigenmode_1** has a **Scale Factor Magnitude** of 1 and all the other modes are set to 0. This indicates that the fields for this mode are loaded into the 3D Post Processor.
2. With **Eigenmode_1** selected, enter **0** in the **Magnitude (peak)** field.
3. Leave **Phase** set to **0**.
4. Choose **Set** to assign the magnitude and phase values to **Eigenmode_1**. With the magnitude set to **0**, the fields for this mode are no longer loaded into the 3D Post Processor.
5. Select **Eigenmode_4**. Now you are going to assign a positive magnitude to this mode, which will load the fields associated with Mode 4 into the 3D Post Processor.

Note:

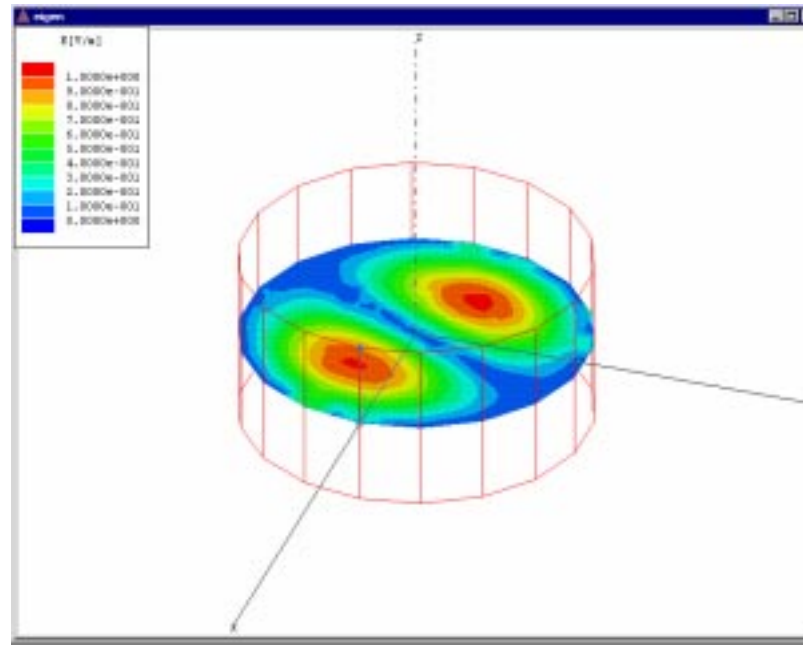
With eigenmodes, you can only choose one mode to plot, not a linear combination of two. This is because each mode can have a different frequency, so if you chose two modes to plot, Ansoft HFSS could figure out **E**, but wouldn't know what frequency to use to calculate **H**.

6. Enter **1** in the **Magnitude (peak)** field and leave **Phase** set to **0**.
7. Choose **Set** to assign these values to **Eigenmode_4**.

8. Choose **OK** to accept the new scale factors for the modes. The following window appears, prompting you to recalculate the plot for the newly loaded fields:



9. Choose **Yes** to recalculate the E-field plot using the fields from Mode 4. A progress bar appears as Ansoft HFSS generates a plot. The following plot appears:



Data/Edit Sources is a convenient way to activate and deactivate eigenmodes for a problem. For Driven Solution problems, you may also activate and deactivate any sources.

Compare Vector Plots

You are now ready to compare the vector plots of **E** for Mode 2 and its degenerate mode, Mode 3. Unlike magnitude plots, vector plots use arrows to indicate the direction and strength of the field being plotted. The length and color of the arrows indicate the strength of the field.

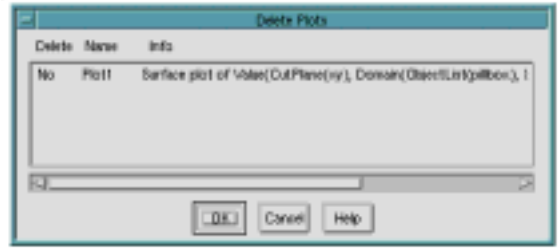
To compare the two fields, create a vector plot for Mode 2, then use **Data/Edit Sources** to display the vector plot for Mode 3. Before creating the plot, clear the current magnitude plot from the display.

Clear the Magnitude Plot

To conserve resources, remove the magnitude plot currently displayed.

➤ Delete the magnitude plot:

1. Choose **Plot/Delete**. The following window appears, prompting you to select the plot you wish to delete:



2. Select **Plot1** from the list. When you select it, **Delete** changes to **Yes**.
3. Choose **OK** to delete the plot. A message appears confirming your decision to delete the plot.
4. Choose **Yes** to delete the plot.

You are now ready to create a vector plot.

Create the Vector Plot for Mode 2

Because the fields for Mode 4 are still loaded into the 3D Post Processor, you must use the **Data/Edit Sources** command to switch to the Mode 2 fields.

Switch Modes

➤ Activate Mode 2:

1. Choose **Data/Edit Sources**. The **Edit Source Values** window appears.
2. Select **Eigenmode_4**.
3. Enter **0** in the **Magnitude (peak)** field.
4. Leave **Phase** set to **0**.
5. Choose **Set**.
6. Select **Eigenmode_2**.
7. Enter **1** in the **Magnitude (peak)** field and leave **Phase** set to **0**.
8. Choose **Set**.
9. Choose **OK** to accept the new scale factors for the modes.

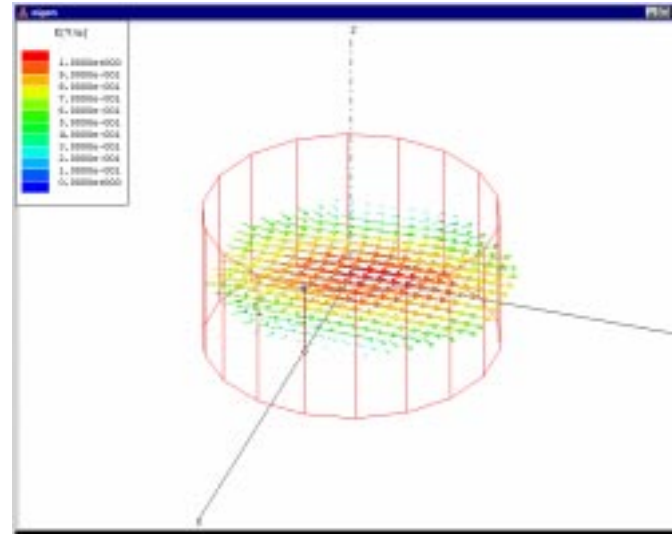
Create the Plot

➤ Create the vector plot:

1. Choose **Plot/Field**. The **Create New Plot** window appears.
2. Select **Vector E** from the **Plot Quantity** list. This selects the electric field |**E**|(x,y,z,t) as the quantity to plot.
3. Select **Surface xy** from the **On Geometry** list.
4. Select **pillbox** from the **In Volume** list.
5. Leave the other settings set to their default settings.
6. Choose **OK**. The following window appears, allowing you to specify the attributes of the plot:

Notice the difference between these plot options and the plot options for a magnitude plot. For vector plots, you specify the size and type of the arrows displayed.

7. Leave the options set to their defaults and choose **OK**. A progress bar appears as Ansoft HFSS generates a plot. The following plot appears:

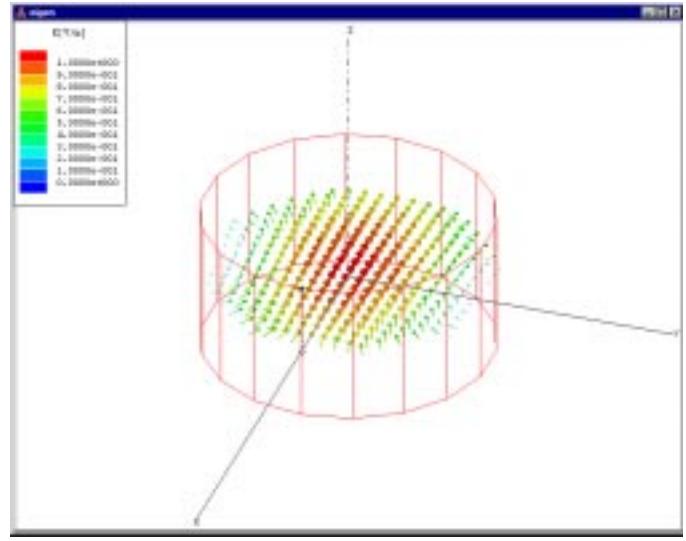


Create the Vector Plot for Mode 3

Now you can use the **Data/Edit Sources** command to change the mode from which the fields are taken.

➤ Display the vector plot for Mode 3:

1. Choose **Data/Edit Sources**. The **Edit Source Values** window appears.
2. Select **Eigenmode_2**.
3. Enter **0** in the **Magnitude (peak)** field.
4. Leave **Phase** set to **0**.
5. Choose **Set**.
6. Select **Eigenmode_3**.
7. Enter **1** in the **Magnitude (peak)** field and leave **Phase** set to **0**.
8. Choose **Set**.
9. Choose **OK** to accept the new scale factors for the modes. You are prompted to recalculate the plot for the newly loaded fields.
10. Choose **Yes** to recalculate the vector plot. A progress bar appears as Ansoft HFSS generates the following plot:



Comparing this plot of Mode 3 with the plot of Mode 2 on the previous page shows that the two modes are basically the same, except Mode 3 is rotated 90 degrees around the z-axis from Mode 2. The Eigenmode solver distinguishes them as two separate modes.

Create an Animated Vector Plot

An animated plot is created frame by frame, in much the same way as an animated cartoon. A series of pictures is taken of a plot, with each picture representing a different stage of the plot. In this example, you will generate a series of pictures of the vectors of the E-field, while varying the phase. Displaying the pictures (frames) in sequence creates the animation.

Before creating the animated vector plot, delete the previous vector plot and switch to the fundamental mode.

Clear the Magnitude Plot

To conserve resources, remove the vector plot currently displayed.

➤ Delete the vector plot:

1. Choose **Plot/Delete**. The **Delete Plots** window appears, prompting you to select the plot you wish to delete.
2. Select **Plot1** from the list. When you select it, **Delete** changes to **Yes**.
3. Choose **OK** to delete the plot. A message appears confirming your decision to delete the plot.
4. Choose **Yes** to delete the plot.

You are now ready to create a switch to the fundamental mode.

Switch Modes

➤ Activate Mode 1:

1. Choose **Data/Edit Sources**. The **Edit Source Values** window appears.
2. Select **Eigenmode_3**.
3. Enter **0** in the **Magnitude (peak)** field.
4. Leave **Phase** set to **0**.
5. Choose **Set**.
6. Select **Eigenmode_1**.
7. Enter **1** in the **Magnitude (peak)** field and leave **Phase** set to **0**.
8. Choose **Set**.
9. Choose **OK** to accept the new scale factors for the modes.

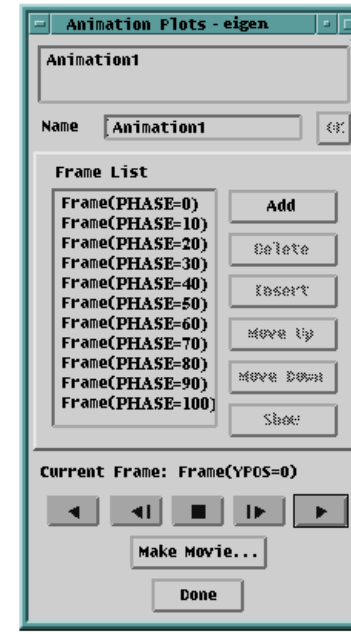
Create the Animated Plot

➤ Create the animated plot:

1. Choose **Plot/Field** to generate a new plot. The **Create New Plot** window appears.
2. Select **Vector E** from the **Plot Quantity** list. This selects the electric field |**E**|(x,y,z,t) as the quantity to plot.
3. Select **Surface xy** from the **On Geometry** list.
4. Select **pillbox** from the **In Volume** list.
5. Select **Phase animation** to make an animated plot by varying the phase of the electric field.
6. Choose **OK** to generate the plot. The following window appears, allowing you to specify how to vary the phase:

7. Do the following to specify how to vary the phase:
   a. Leave **Variable** set to **PHASE**. When you choose **Phase animation**, the variable is limited to the phase. If you had chosen an animated geometry, such as **AnimSurf xy**, from the **On Geometry** list, other variables would be present.
   b. Leave **Start** set to **0** degrees. This is the starting value for the phase.
   c. Enter **360** in the **Stop** field. This is the stopping value for the phase.
   d. Enter **10** in the **Delta** field. This is the amount by which the phase changes between one plot and the next.
   e. Select **Store Frames** to save the each frame of the animation. From these saved frames you can create an animated movie.
   f. Choose **OK**. The **Vector Surface Plot** window appears, allowing you to specify the attributes of the plot.
8. Leave the options set to their defaults and choose **OK**. After a pause, a progress bar appears as Ansoft HFSS generates a plot at each phase. After Ansoft HFSS has finished, the **Animation Plots** window appears allowing you to manipulate the movie. Since you selected **Store Frames**, the **Animation Plots** window appears. Otherwise, the controls for the animation would appear in the side window.

The **Animation Plots** window appears as shown to the right.



Each frame in the **Frame List** is a plot computed at a different phase. The phase of each plot is listed next to each frame. The controls at the bottom of the window allow you to play the animation sequence forward or backward, or step through it one frame at a time.

9. Choose **Make Movie**. This creates a replaying movie from the series of animated plots. Ansoft HFSS generates a series of pixmaps from the frames in the **Frame List**, then it animates these into a movie.
   Once the series of pixmaps have been created, a window appears allowing you to control the movie. You can change the speed, stop the move, or access the **Animation Plots** window again to manipulate the frames.
10. Choose **Done** when you wish to end the movie.

From this animated plot, you can see that the resonance of the closed cavity is a standing wave.

Exit the Field Simulator

To exit Ansoft HFSS, follow the steps below:

- Exit the 3D Post Processor:
  1. Choose **File/Exit**. The following message appears:

     ```
     Exit Post Processor?
     ```

  2. Choose **Yes**.

  You return to the Executive Commands window.

- Exit Ansoft HFSS:
  1. Choose **Exit**. The following message appears:

     ```
     Exit HFSS?
     ```

  2. Choose **Yes**.

  You return to the Maxwell Control Panel.

Index

N

O

P

R

S

T

U

V

W

Z