Getting Started with HFSS: Coax Tee

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

Release 19.0
December 2017

Conventions Used in this Guide

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

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means the word copy must be **typed**, then a space must be typed, and then **file1** must be typed.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, "click **HFSS>Excitations>Assign>Wave Port**."
  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *file name*" the word **copy must be typed, then** a space must be typed, and then name of the file must be typed.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press Shift+F1" means to press the Shift key and the F1 key at the same time.

Ribbons, menu bars, and short-cut menus are three methods that can be used to see what commands are available in the application.

- *Ribbons* are the rectangular area on top of the application window and contain multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "On the **Draw** ribbon tab, click the **Box** primitive" means you can click the **Box** icon on the **Draw** tab and execute the **Box** command to draw a box.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *short-cut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

"Right-click and select **Assign Excitation>Wave Port**" means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the short-cut menu (and the corresponding sub-menus).

Getting Help: ANSYS Technical Support

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

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

Help Menu

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

- **HFSS Contents** - click here to open the contents of the online help.
- **HFSS Search** - click here to open the search function of the online help.

Context-Sensitive Help

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

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

Table of Contents

1 - Introduction

This document is intended as supplementary material to HFSS for beginners and advanced users. It includes instructions to create, solve, and analyze a Coax Tee design.

This chapter contains the following topic:

- Sample Project - The Coax Tee

Sample Project - The Coax Tee

In this project, you will learn how to create the Coax Tee model. HFSS solves for the fields in an arbitrary volume of:

- Coax Dielectric
- Coax Center Pin
- Outer Boundary
- Coax Shield

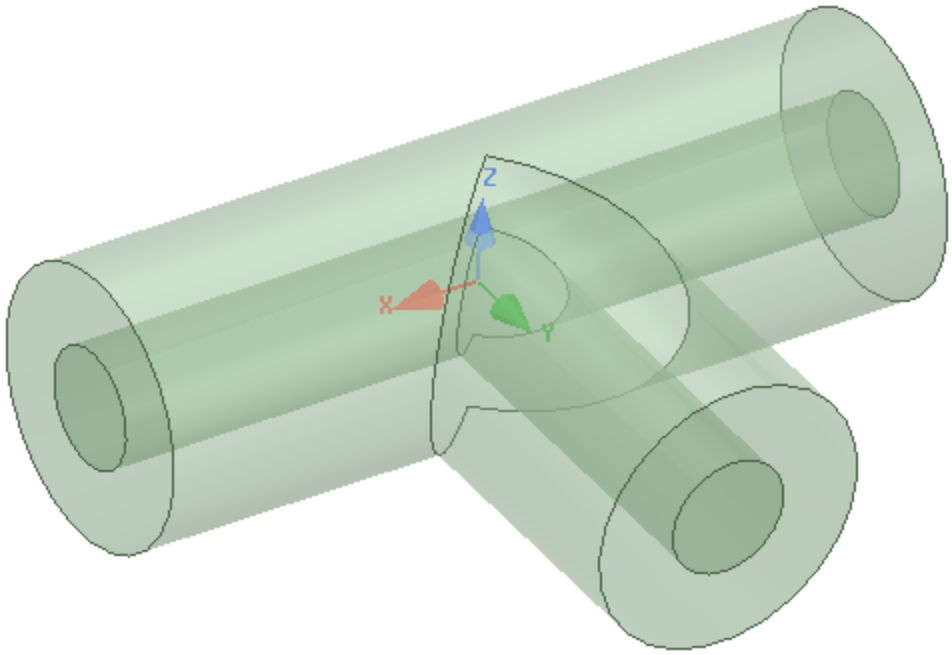


Figure 1-1 Coax Tee

2 - Set Up the Project

This chapter contains the following topics:

- Launch HFSS
- Set Tool Options
- Insert HFSS design
- Set Model Units (mm)
- Set Solution Type (Driven Modal)

Launch HFSS

Store a shortcut of the ANSYS Electronics Desktop application on your desktop.

1. Double-click the **ANSYS Electronics Desktop** icon to launch the application.

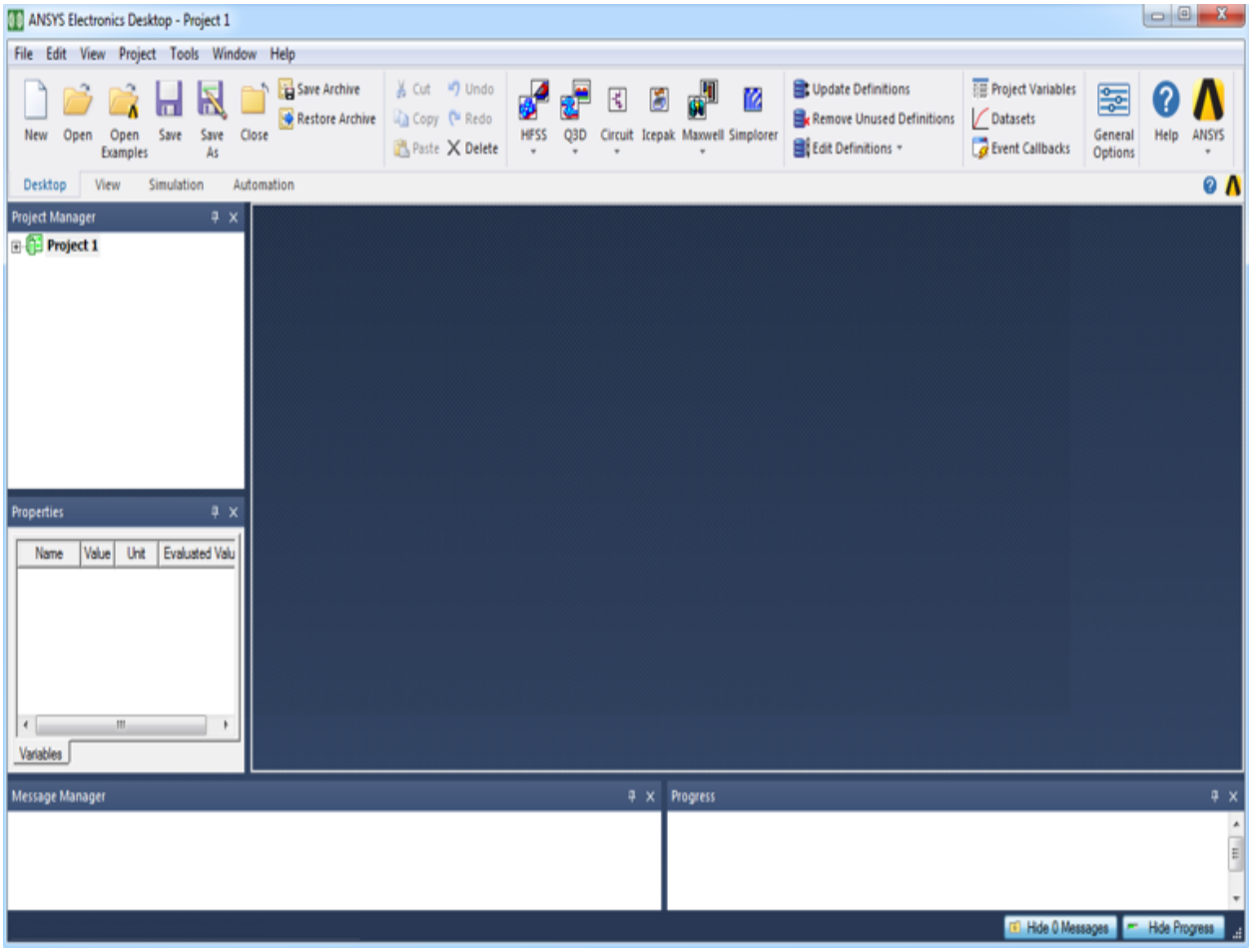


Figure 2-1 Electronics Desktop opens

Note If the application does not list the project folder, go to the **Desktop** tab ribbon and click **New**. If the **Project Manager** window does not appear, go to the **View** tab ribbon and enable it from the **Docking Windows** drop-down option.

Set Tool Options

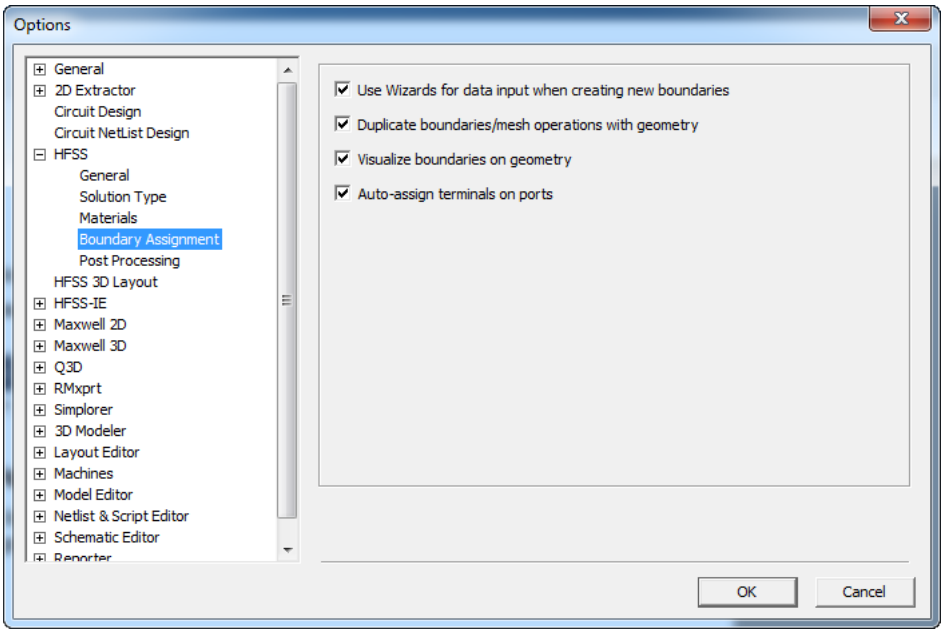
Before you begin creating a design, configure some of the general options of HFSS:

1. Go to the **Desktop** tab and select **General Options**.

   The **Options** dialog box appears.

2. Expand **HFSS** and select the **Boundary Assignment** option.

3. Ensure the boundary assignment options are all selected as shown below.



4. Click **Drawing** under the **3D Modeler Options**.
5. Select the **Automatically cover closed polylines** and the **Edit properties of new primitives** check boxes and click **OK.**

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

Insert HFSS design

The icon highlighted in the image below is the **HFSS** design type.

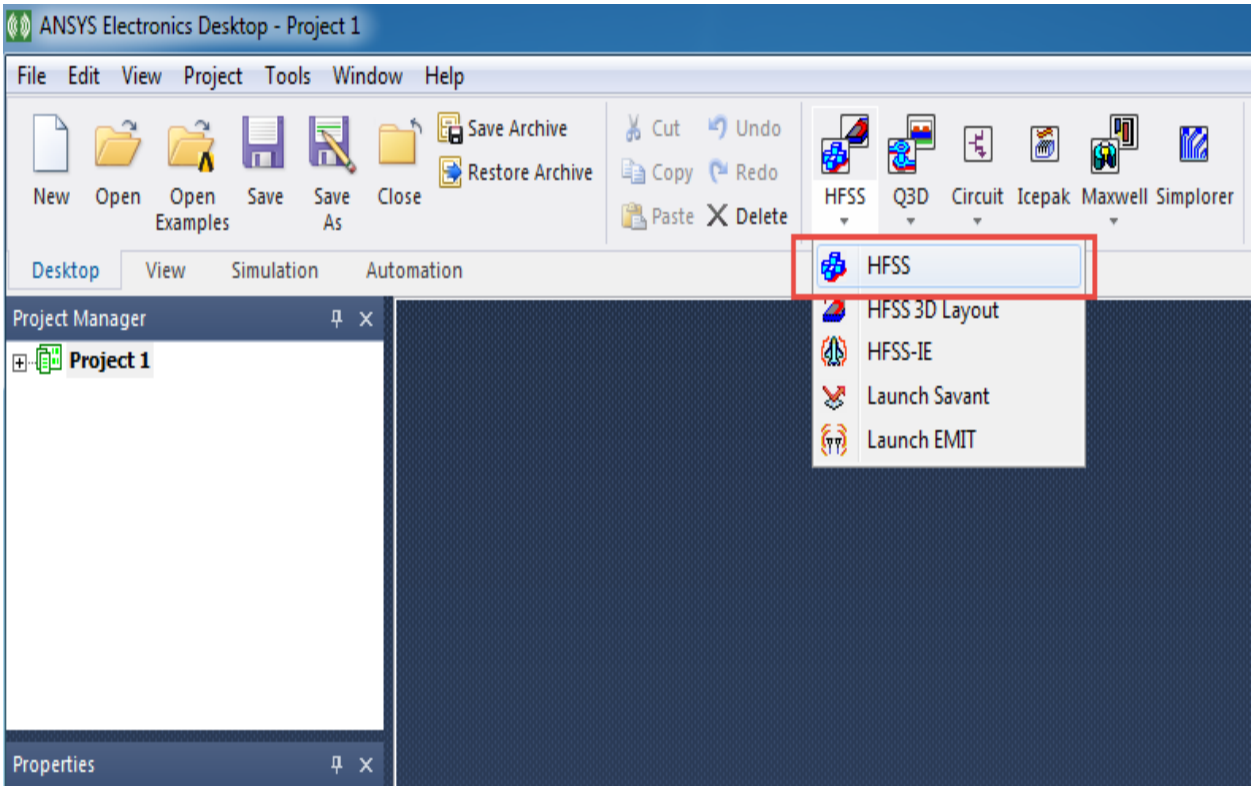


Figure 2-2 HFSS Design Type

1. Click the **HFSS** design type to include it in your project.

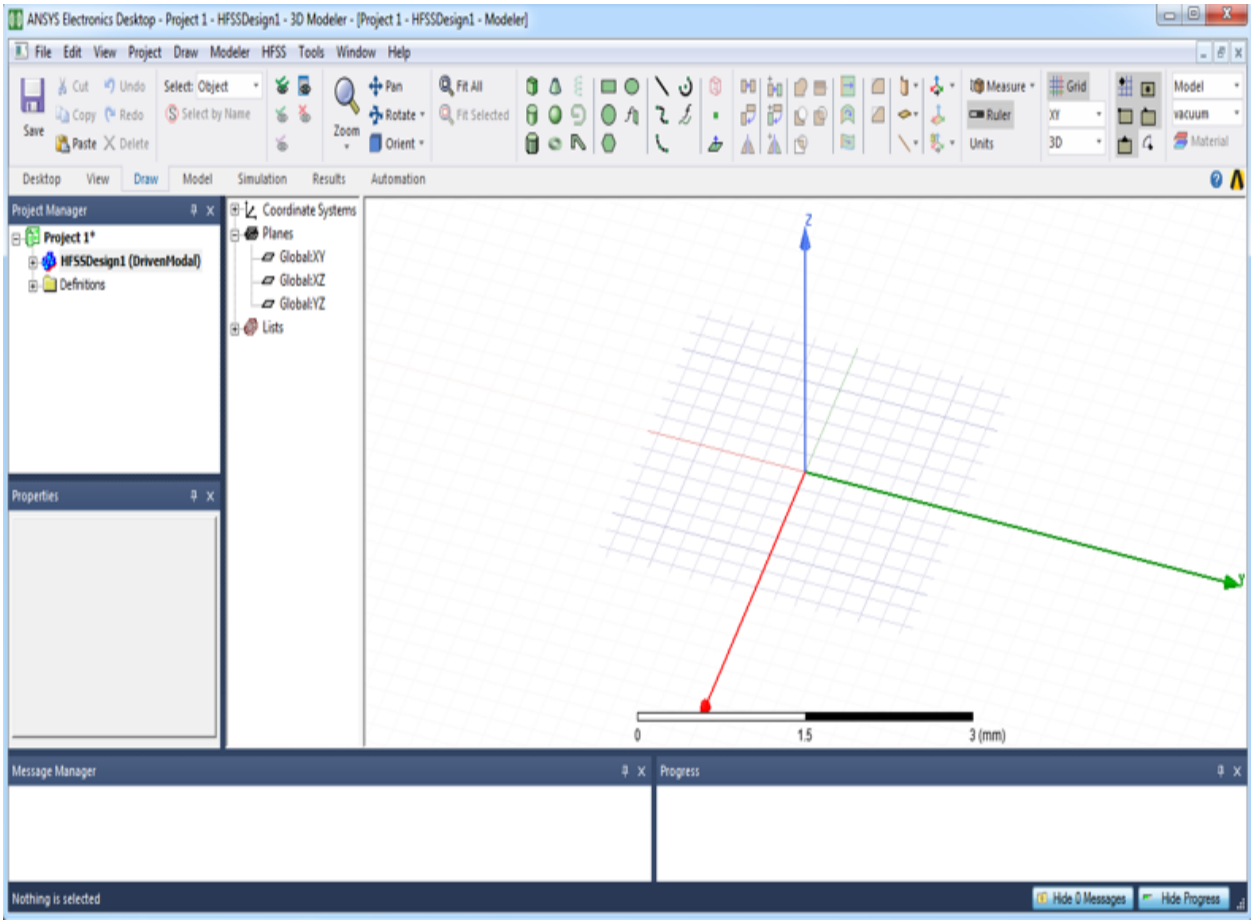


Figure 2-3 IHd included

3. Click **Project*n***, hit **F2**, rename the project and save it.

   Note: Go to the **View** menu item on the toolbar and select the **Coordinate System** option to adjust its size. Similarly, you can adjust grid visibility and its style from **Grid Settings** under the **View** menu item on the toolbar.

Set Model Units (mm)

Define the model units as follows:

1. On the **Draw** tab ribbon, click **Units**.

   The **Set Model Units** dialog box appears.
2. Select **mm** (milimeters) from the **Select units** drop-down menu, and click **OK**.

Figure 2-4 Set Model Units dialog box

Set Solution Type (Modal)

Specify the design's solution type as follows:

1. On the toolbar, click **HFSS>Solution Type**.

   The **Solution Type** dialog box appears.

2. Select **Driven Modal** and click **OK**.

   Note This command ensures that HFSS calculates the modal-based S-parameters using Driven Modal. The S-matrix solutions are expressed in terms of the incident and reflected powers of waveguide modes.

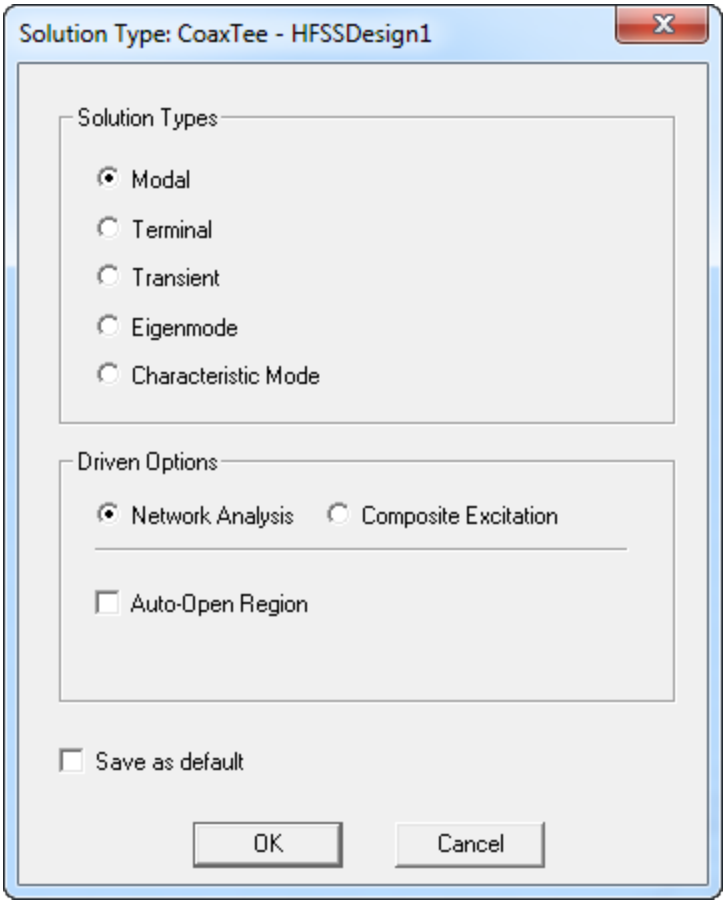


Figure 2-5 Solution Type

3 - Create the Model

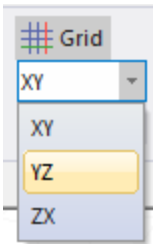
The outline of the process to create the model is as follows:

- Set Grid Plane
- Create the Coax Pin
- Create the Coaxial Connector
- Assign Excitation
- Duplicate to create the Tee
- Unite the Conductors
- Unite the Coaxial Cylinders

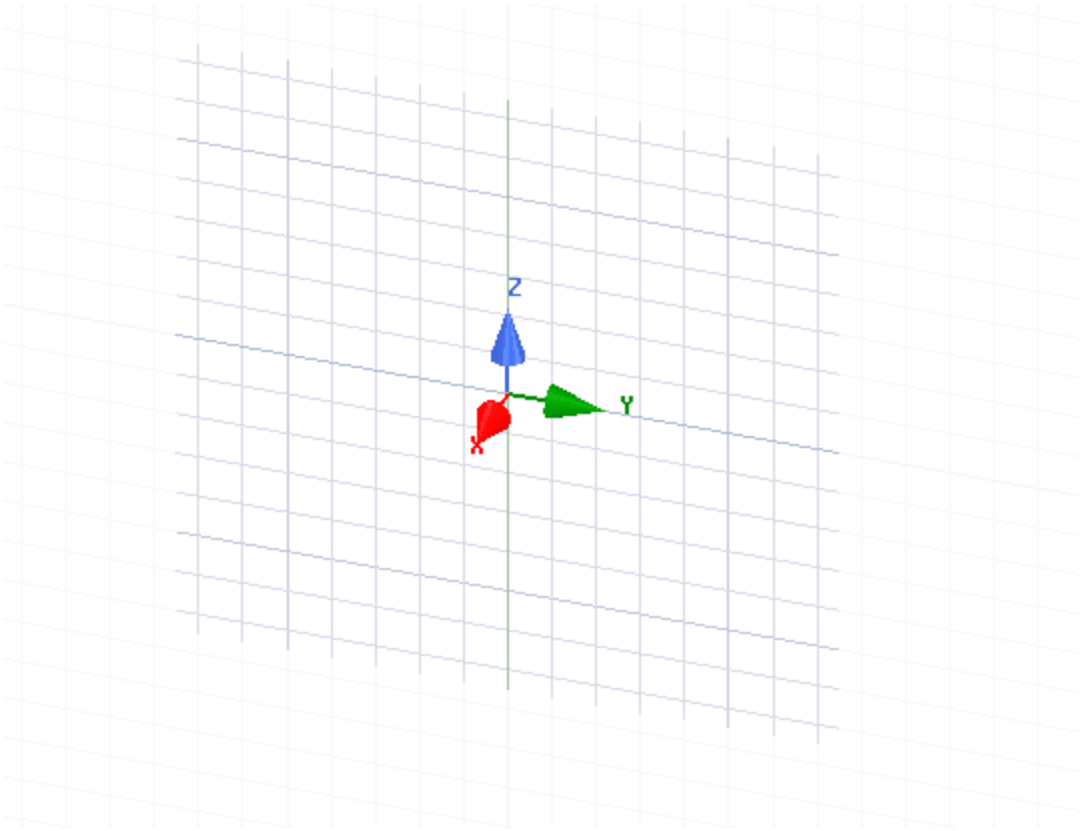
Set Grid Plane

Specify the plane on which you want to display the grid as follows:

1. On the **Draw** tab ribbon, set the drawing plane to YZ.



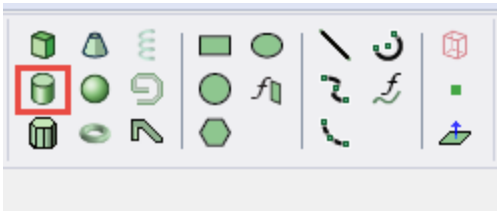
The grid plane in the modeler is set to YZ.



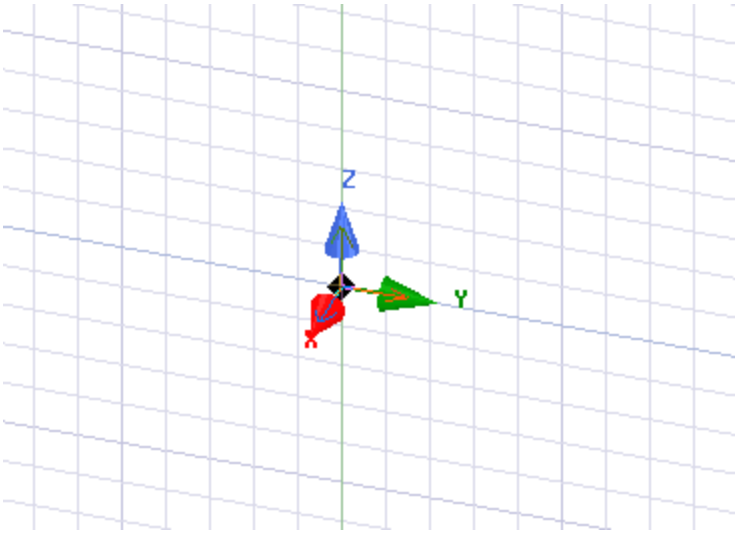
Create the Coax Pin

To create the Coax Pin draw the cylinder freehand as follows:

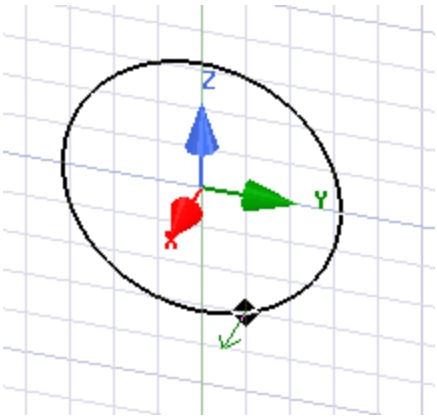
1. On the **Draw** tab ribbon click the **Cylinder** primitive.



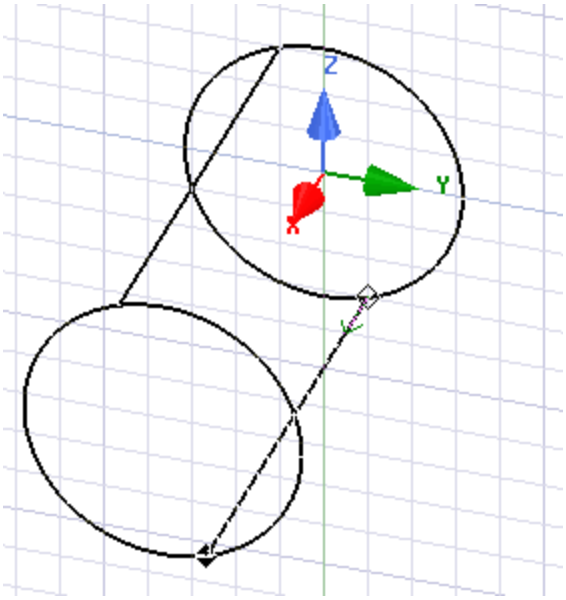
The cursor changes to a snapping point indicator.

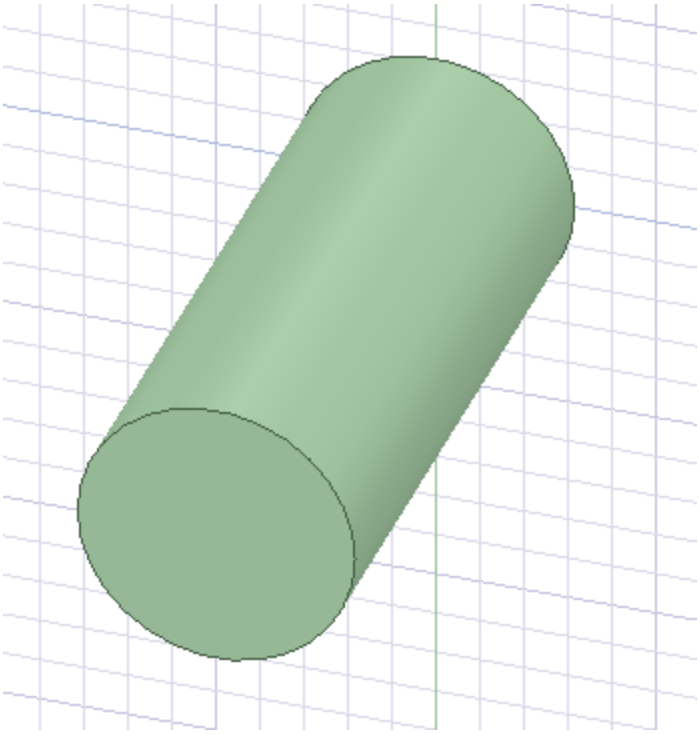


2. Click inside the **3-D Modeler** active view window.
   The **YZ** mini axes is established.
3. Drag the cursor to draw the circle, and click the mouse.
   The mini **X** axis is established for the height.



4. Drag the cursor and to complete drawing the cylinder just click the mouse.





This operation generates a random cylinder and also opens the **Properties** window.

5. On the **Command** tab of the Properties window edit the fields as shown in "Commands window for Coax Pin" below. In field for **Height** type *H* and set the **Add Variable** dialog box as shown in "Add Variable dialog box" below. and click **OK**.

Command | Attribute

Name	Value	Unit	Evaluated Value
Command	CreateCylinder		
Coordinate Sys...	Global		
Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
Axis	X		
Radius	0.86	mm	0.86mm
Height	6	mm	6mm
Number of Seg...	0		0

Figure 3-1 Commands window for Coax Pin

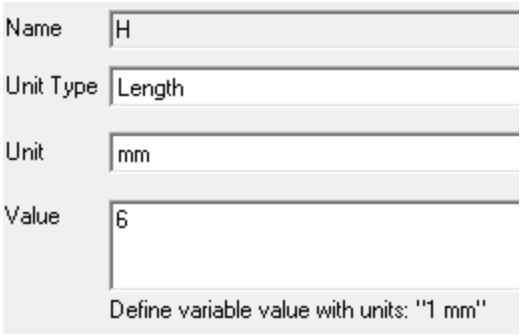


Figure 3-2 Add Variable dialog box

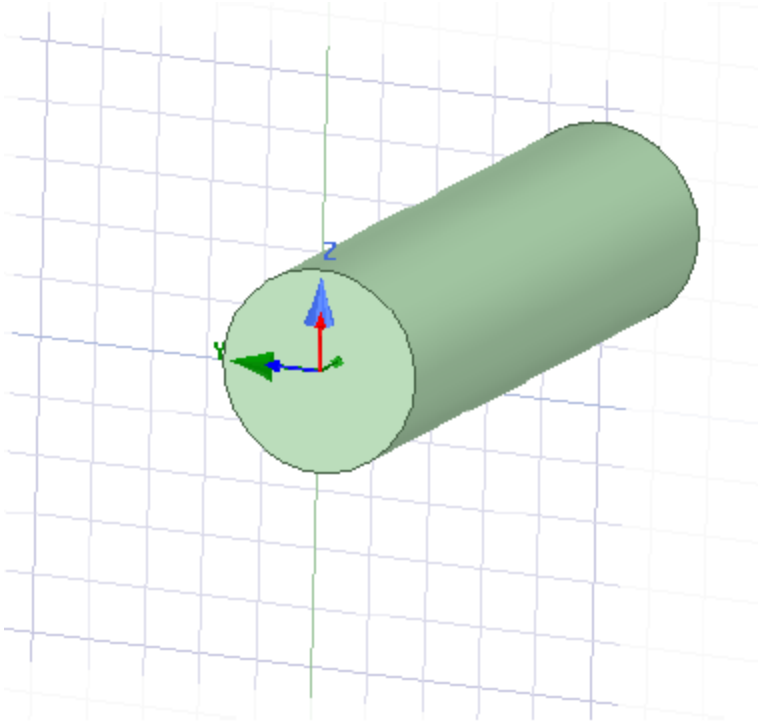


Figure 3-3

Figure 3-4 Cylinder in the 3-D Modeler Window

6. On the **Attribute** tab rename the object as *Coax_Pin* and select **Edit** from the **Materials** drop down menu.

   The **Select Definitions** window appears.

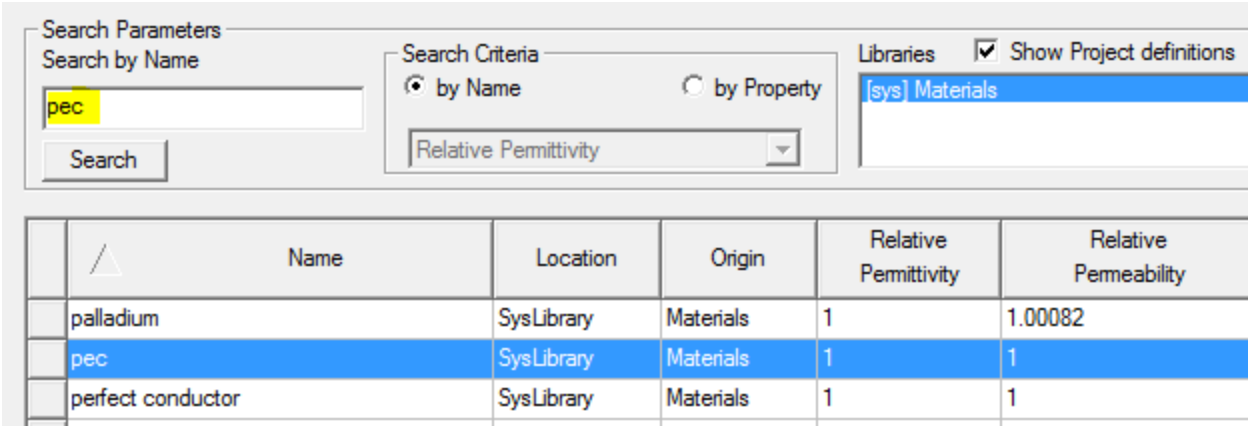


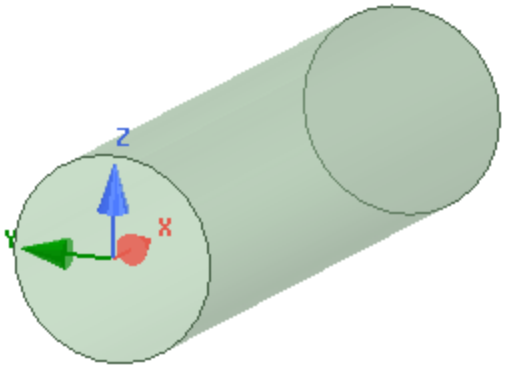
Figure 3-5 Select Definitions window

7. Type *pec* in the **Search by Name** field and press **Enter**.

Note Ignore the following informational message that appears in the **Message Manager** window: Solve Inside for object 'Coax_Pin' is unset, due to material assignment change.

8. Set the desired color and set **Transparency** to 0.6 and click **OK** to close the **Properties** dialog box.
9. Press **Ctrl+D** to fit the view.

   3-D Modeler accommodates the cylinder in its window.
10. If you want to hide the grid, go to **View > Grid Settings** from the toolbar and click the **Hide** option under **Grid Visibility**.



Create the Coaxial Cylinder

To create the coaxial cylinder which encloses the coaxial pin, you will draw a cylinder freehand.

1. Draw a cylinder freehand.
2. Edit the fields in the **Command** dialog box as shown in "Command window for the Coax Cylinder" on the next page. Make sure you enter *H* in the **Height** field.
3. On the **Attribute** rename the cylinder as *Coax* and select vacuum from the **Materials** drop-down menu.

Command | Attribute

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate Sys...	Global		
	Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
	Axis	X		
	Radius	2	mm	2mm
	Height	6	mm	6mm
	Number of Seg...	0		0

Figure 3-6 Command window for the Coax Cylinder

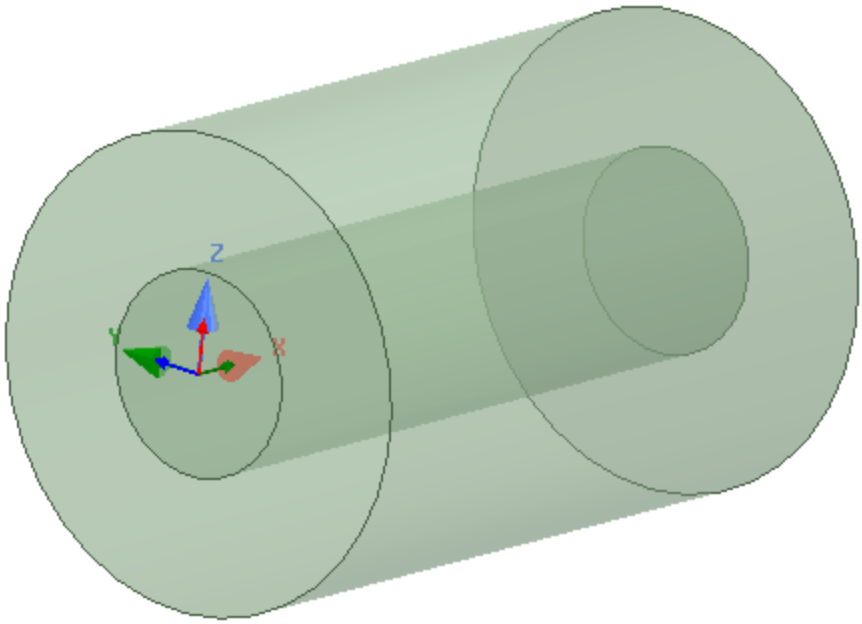


Figure 3-7 Structure accommodated

Assign Excitation

For this model use wave ports to excite both ends of the coax tee. This section describes how to assign the first excitation.

1. Press the hot key **F** to enter face selection mode.
2. Click the left-most face (on the positive X-axis) of Coax. See "The left-most surface selected" below.
3. Right-click and select **Assign Excitations>Wave Port**.
   The **Wave Port : General** dialog box appears.

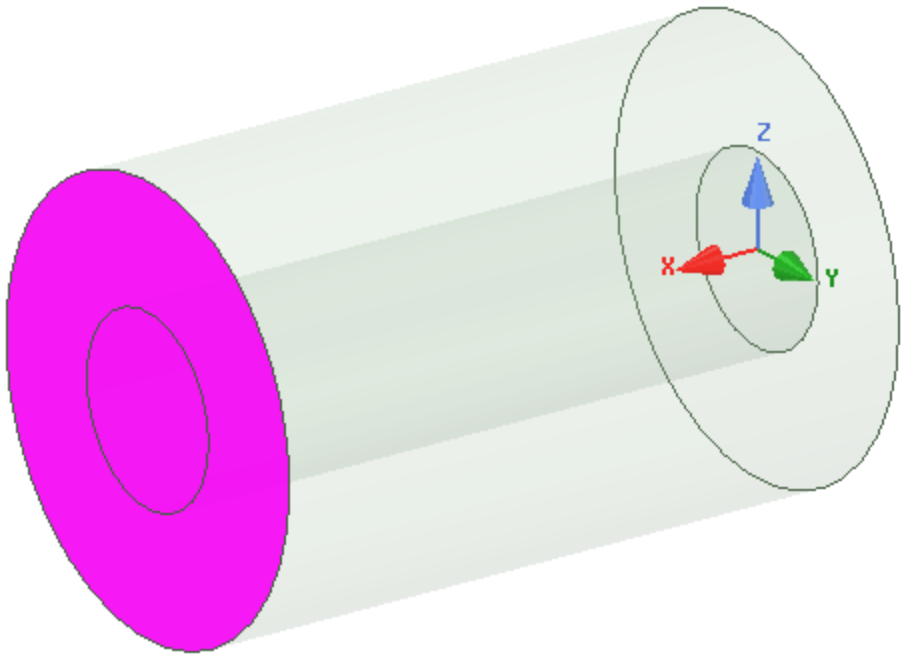


Figure 3-8 The left-most surface selected

4. Type *p1* in the **Name** field and click **Next**.
   The **Wave Port : Modes** dialog box appears.
5. Click **Next** and set the fields as in "Post Processing dialog box" below. Click **Finish**.

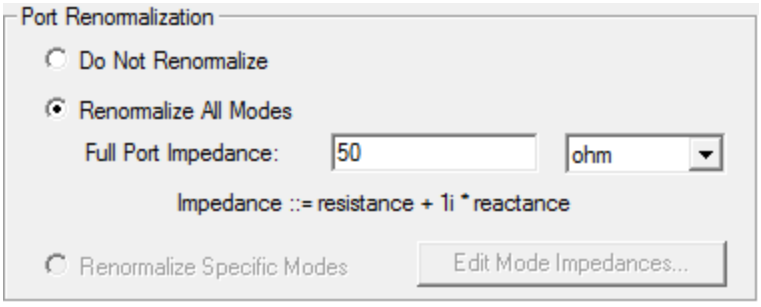


Figure 3-9 Post Processing dialog box

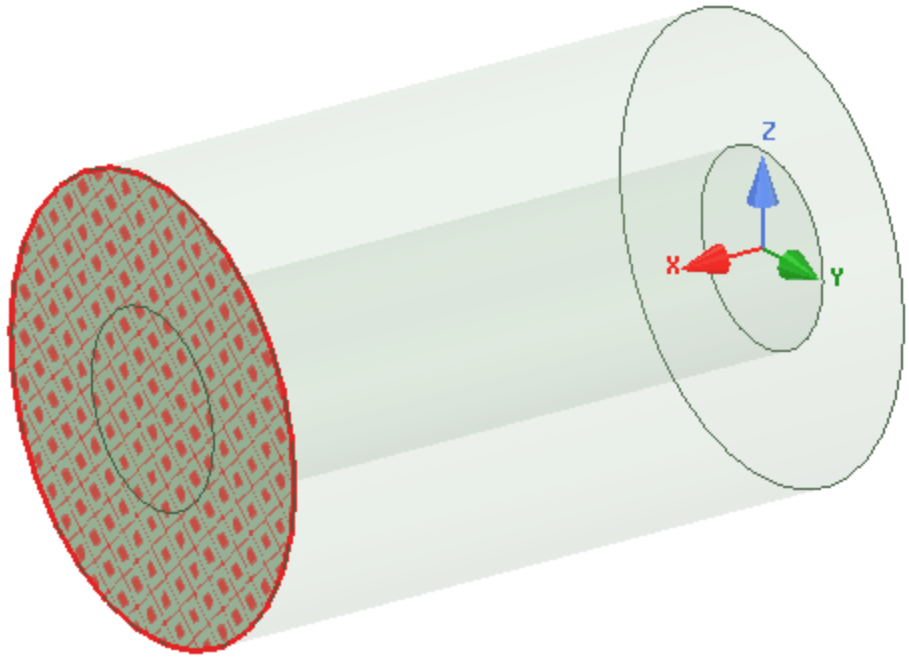


Figure 3-10

Duplicate to Create the Tee

You will select the Coax cylinder (that encloses the Coax-Pin) and duplicate it. Perform the following steps to design the Coax-Tee.

1. Press "O" on your keyboard to enter object selection mode.
2. Press **CTRL+A**.
   The model objects are selected.
3. Right-click and on the short-cut menu go to **Edit>Duplicate>Around Axis** .
   The **Duplicate Around Axis** dialog box appears.
4. Edit the fields as shown in "Duplicate around Axis" on the facing page. . Click **OK**.

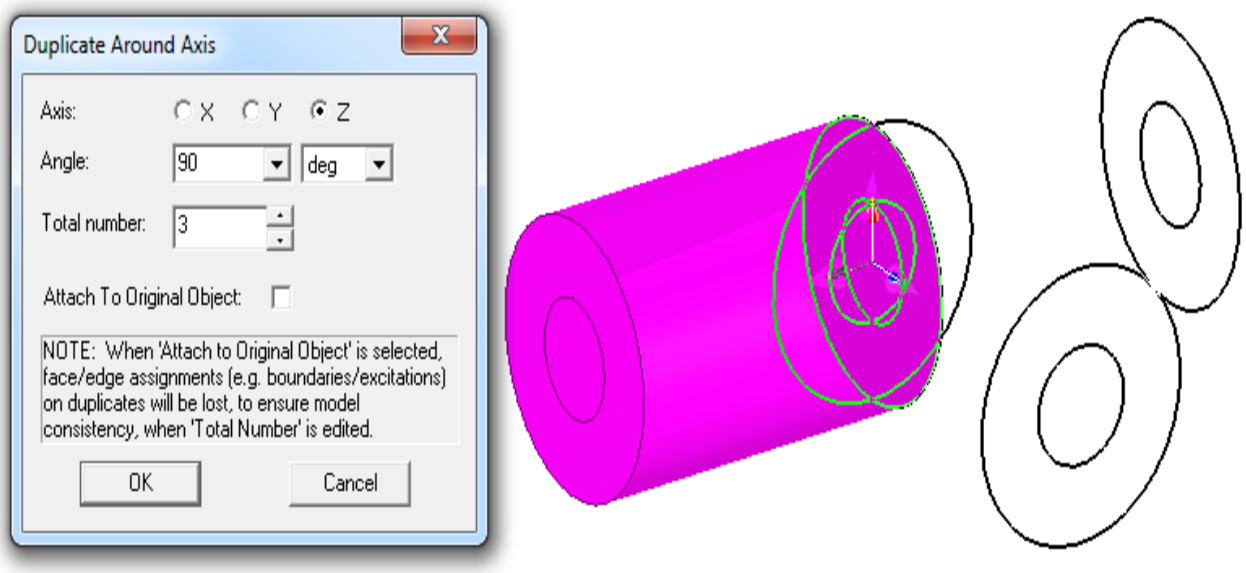


Figure 3-11 Duplicate around Axis

5. Click **OK** on the **Properties** dialog box to close it.

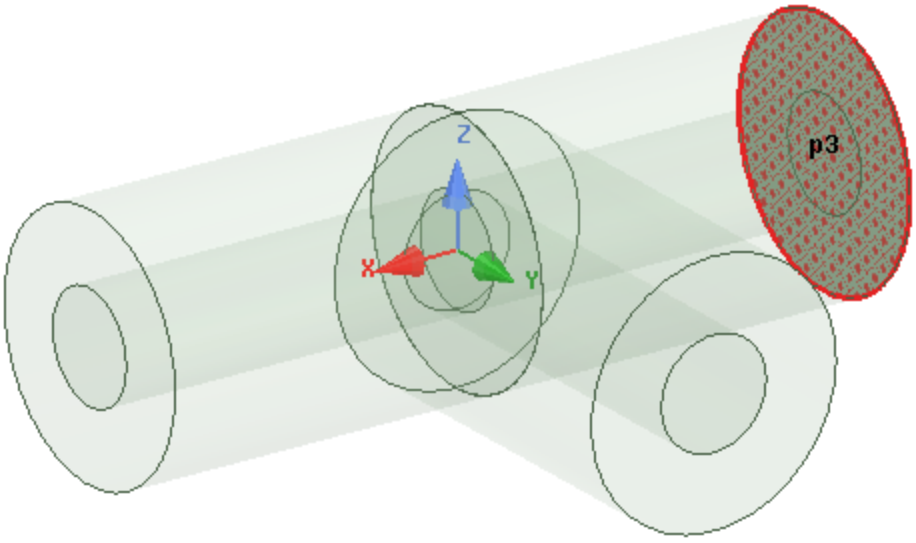


Figure 3-12 The Structure Nearly Complete

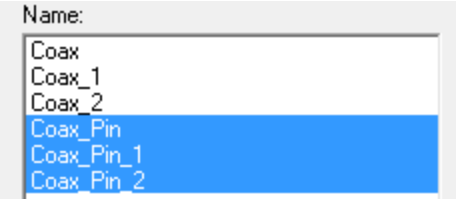
Unite the Conductors

Ensure you are in the object selection mode. So far you created different parts of the structure namely, the coax-pins and the enclosures. Now unite the conductors as follows:

1. Click the **Select by Name** option on the **Draw** tab ribbon.

   The **Select Object** dialog box appears.
2. Select the following objects in the order mentioned and click **OK**: *Coax_Pin*, *Coax_Pin_1*, and then, *Coax_Pin_2*.

   Note See "Unite Objects" below.



3. On the **Draw** tab ribbon, click the **Unite** option.

Figure 3-13 Unite Objects

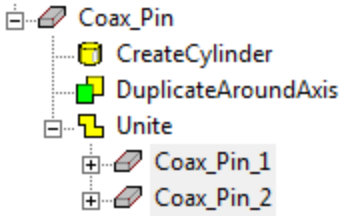


Figure 3-14 History Tree

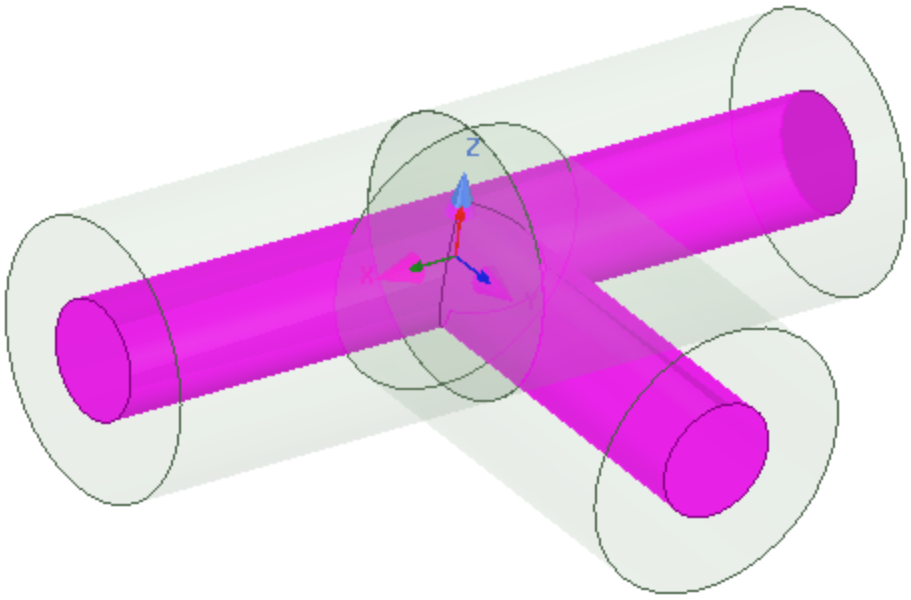


Figure 3-15 The United Conductors

Unite the Coaxial Cylinders

After uniting the co-axial cylindrical pins (the conductors) to form a Tee-shaped structure, the next step will be to unite the enclosure of these cylinders as follows:

1. Click the **Select by Name** option on the **Draw** tab ribbon.

   The **Select Object** dialog box appears.
2. Select the objects in the mentioned order as follows: *Coax*, *Coax_1*, and then, *Coax_2* and click **OK**.

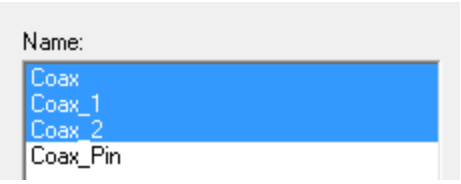


Figure 3-16 Select Object dialog box

Note Leave out Coax_Pin, which is an internal part of the whole structure separate from the enclosure.

3. On the **Draw** tab ribbon, click the **Unite** option.

The enclosures are united.

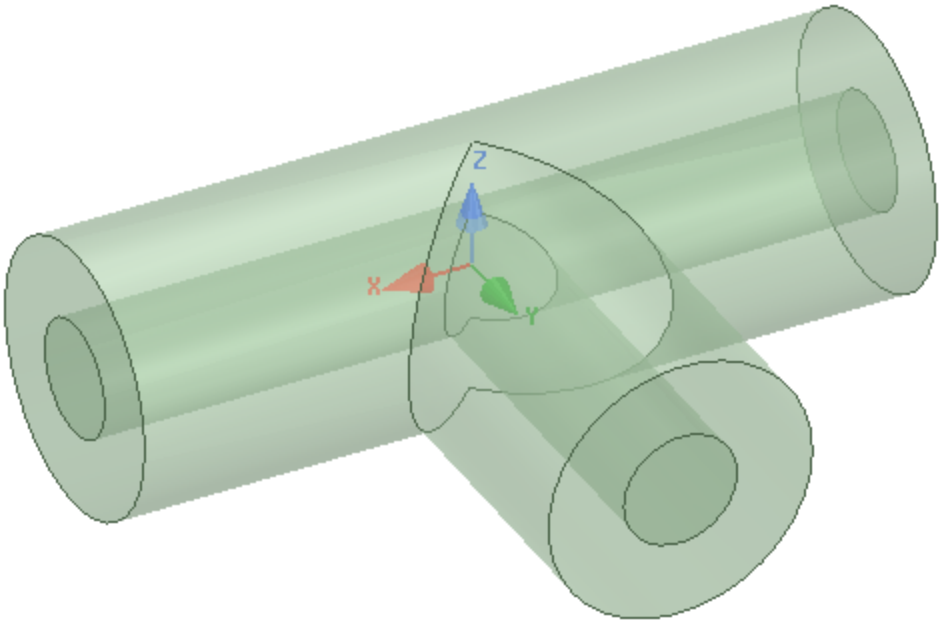


Figure 3-17 The Final Complete Coax-Tee Geometry

4 - Analyze the Model

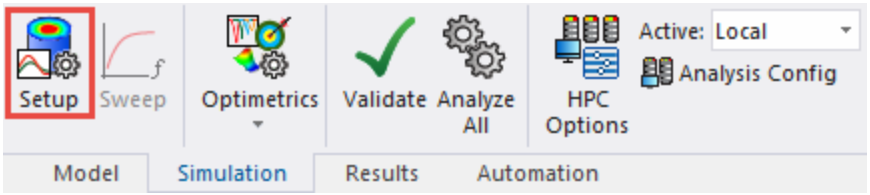
This chapter contains the following topics:

- Add Solution Setup
- Add Frequency Sweep
- Validate the Design
- Analyze the Design
- Review Solution Data
- Review the Profile Panel
- Review the Convergence Panel
- Review the Matrix Data Panel
- Review the Mesh Statistics Panel
- Create the Reports
- Create Field Overlays
- Modify Attributes of a Field Plot
- Plot Quality

Add Solution Setup

To solve the Coax Tee, define an analysis set-up and assign a frequency sweep. After specifying the adaptive frequency and the sweep, you can simulate the design. Define the adaptive frequency as shown below.

1. Select the **Setup** icon on the **Simulation** tab ribbon.



   The **Solution Set-Up** dialog box appears.
2. Click **General**, and edit the fields as shown in "Solution Setup Dialog Box" on the next page.

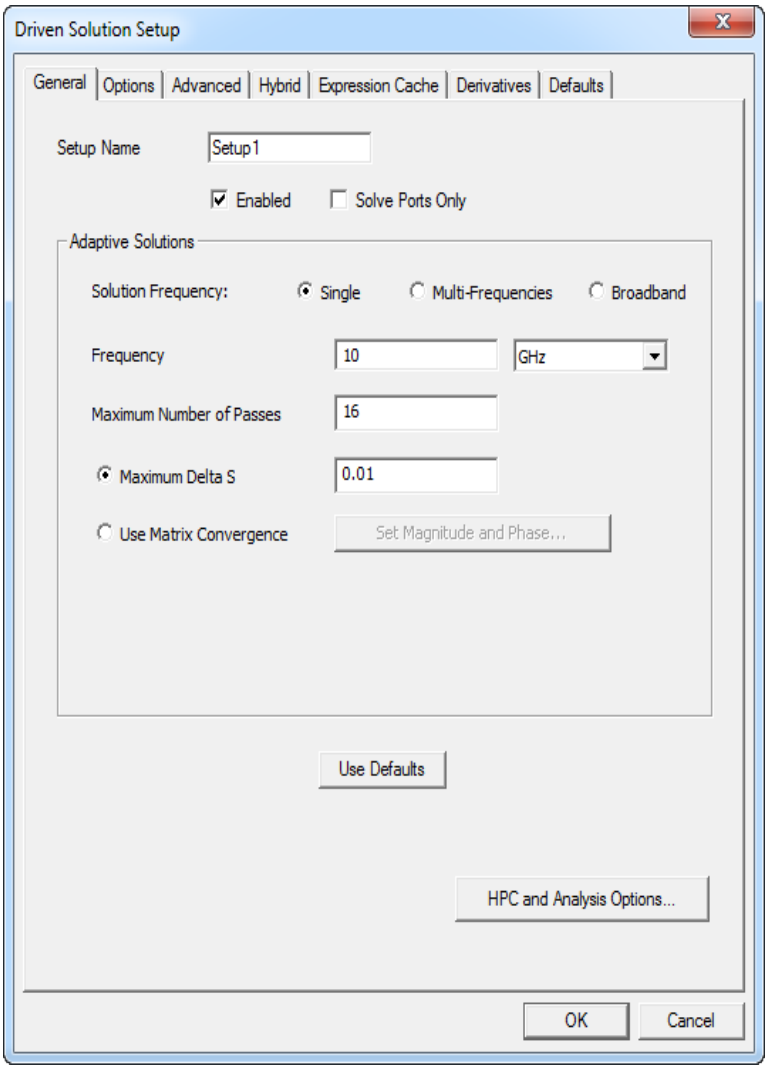
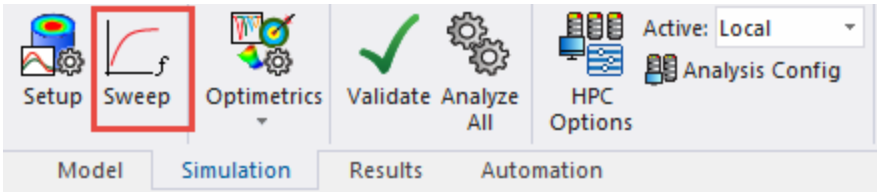


Figure 4-1 Solution Setup Dialog Box

Add Frequency Sweep

1. Under **Analysis**, click **Setup1** and select the **Sweep** option on the **Simulation** tab.



The **Edit Frequency Sweep** window appears.

2. Edit the fields as in "Edit Frequency Sweep box" below. Click **OK**.

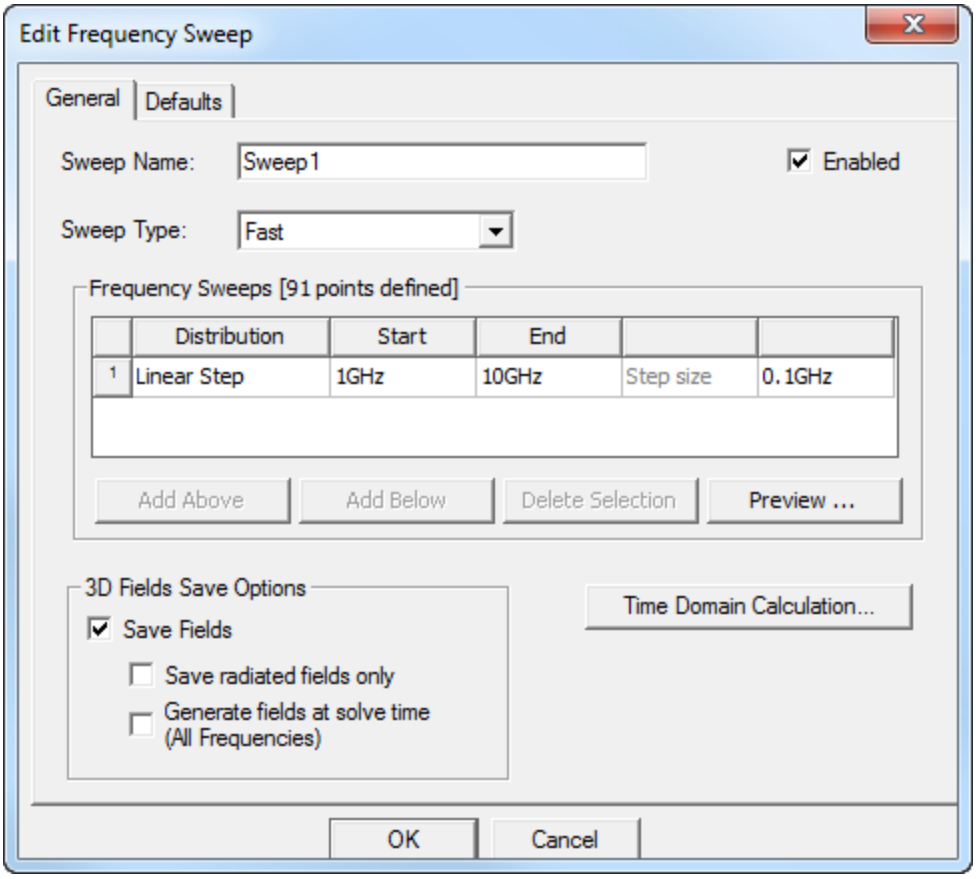
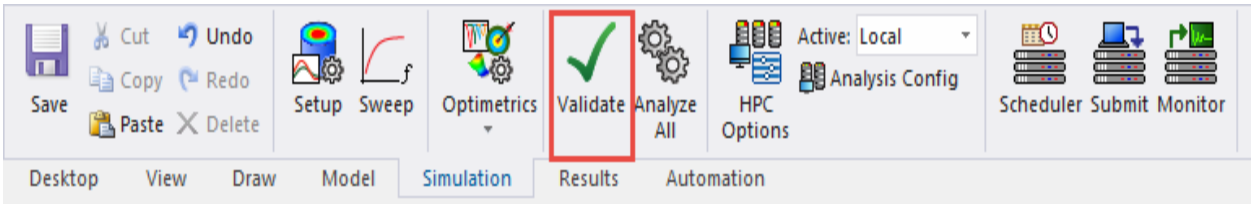


Figure 4-2 Edit Frequency Sweep box

Validate the Design

The model has to pass the validation check to confirm your design is accurate before you run an analysis.

1. Click the **Validate** option on the **Simulation** tab ribbon.



The Validation Check window appears.

The **Validation Check** dialog box must show a tick near each of the listed options, in which case, you can analyze the design.

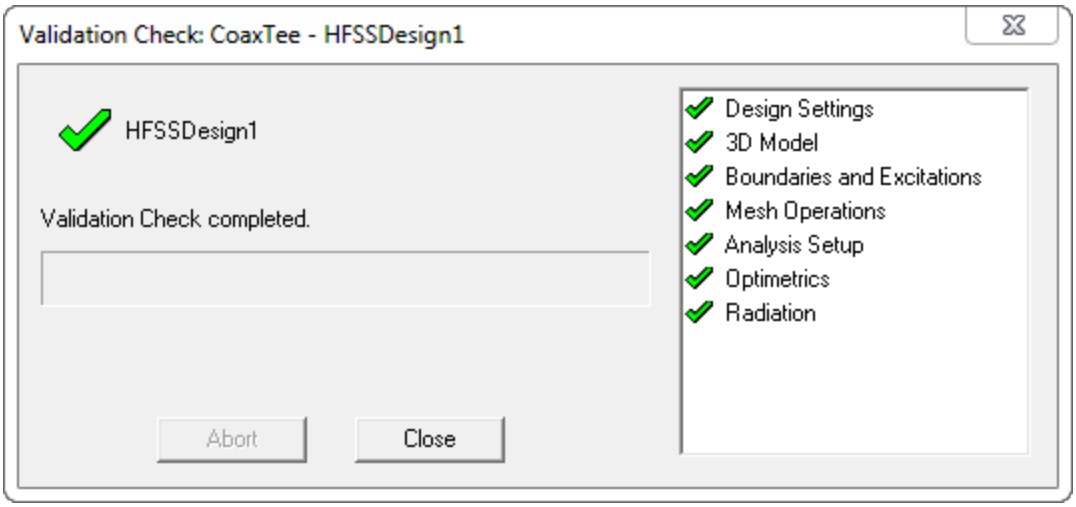


Figure 4-3 Validation Check window

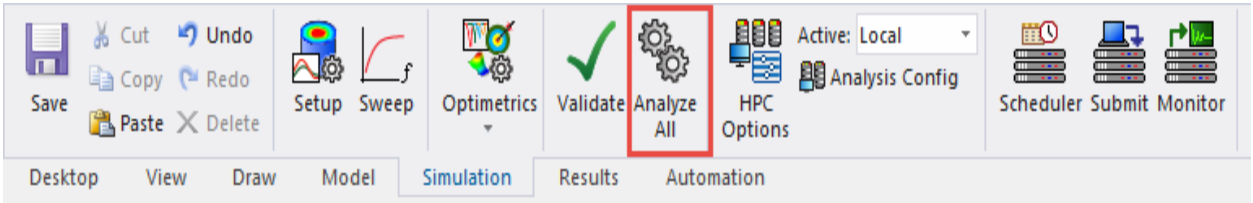
2. Click **Close**.

   Note For most HFSS projects, warnings in the **Message Manager** window may appear. Some of these messages warn you of potential problems. It may not always require you to perform any action to deal with the messages.

Analyze the Design

If your design passed the validation check, you can analyze it. Before that, save the project.

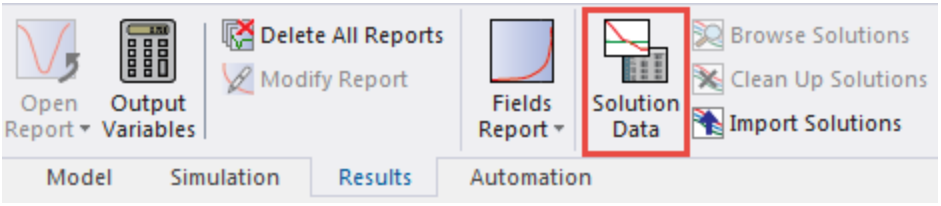
1. Save the project.
2. Click **Analyze All** on the Simulation tab ribbon.



If the analysis was smooth, the **Message Manager** will notify a normal completion of simulation.

Review Solution Data

1. On the **Results** tab ribbon, select the **Solution Data** option.



   The **Solutions** dialog box appears.

Review the Profile Panel

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

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

				Solution Basis Order: 1
Mesh Pre	00:00:00	00:00:00	37.5 M	360 triangles
Mesh TAU (Strict)	00:00:00	00:00:01	43.9 M	5085 tetrahedra
Mesh TAU (Coarseni...	00:00:00	00:00:01	43.9 M	3789 tetrahedra
Mesh Post	00:00:00	00:00:00	45.2 M	3789 tetrahedra
Mesh Refinement				Lambda Based
Mesh (lambda based)	00:00:00	00:00:00	34.3 M	3634 tetrahedra
Mesh Refinement				Port Adapt
Simulation Setup	00:00:00	00:00:00	29 M	Disk = 0 KBytes
Port Adaptation	00:00:01	00:00:01	41.8 M	Disk = 129 KBytes, 2615 tetrahedra
Mesh (port based)	00:00:01	00:00:01	34.4 M	3952 tetrahedra
Adaptive Pass 1				Frequency: 10 GHz
Simulation Setup	00:00:00	00:00:00	29.8 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	40.2 M	Disk = 0 KBytes, 2941 tetrahedra , p1: 122 triangles , p2
Solver MRS1	00:00:00	00:00:00	50.8 M	Disk = 0 KBytes, matrix size 14571 , matrix bandwidth 1
Field Recovery	00:00:00	00:00:00	50.8 M	Disk = 2501 KBytes, 3 excitations
Adaptive Pass 2				Frequency: 10 GHz
Mesh (volume, adapti...	00:00:00	00:00:00	35.1 M	4624 tetrahedra
Simulation Setup	00:00:00	00:00:00	30.7 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	43.1 M	Disk = 0 KBytes, 3600 tetrahedra , p1: 122 triangles , p2
Solver MRS1	00:00:00	00:00:00	58.4 M	Disk = 0 KBytes, matrix size 18653 , matrix bandwidth 1
Field Recovery	00:00:00	00:00:00	58.4 M	Disk = 608 KBytes, 3 excitations
				Adaptive Passes converged

Figure 4-4 Profile

Review the Matrix Data Panel

1. Open **Solutions** and click **Matrix Data** to view the S Matrix data.

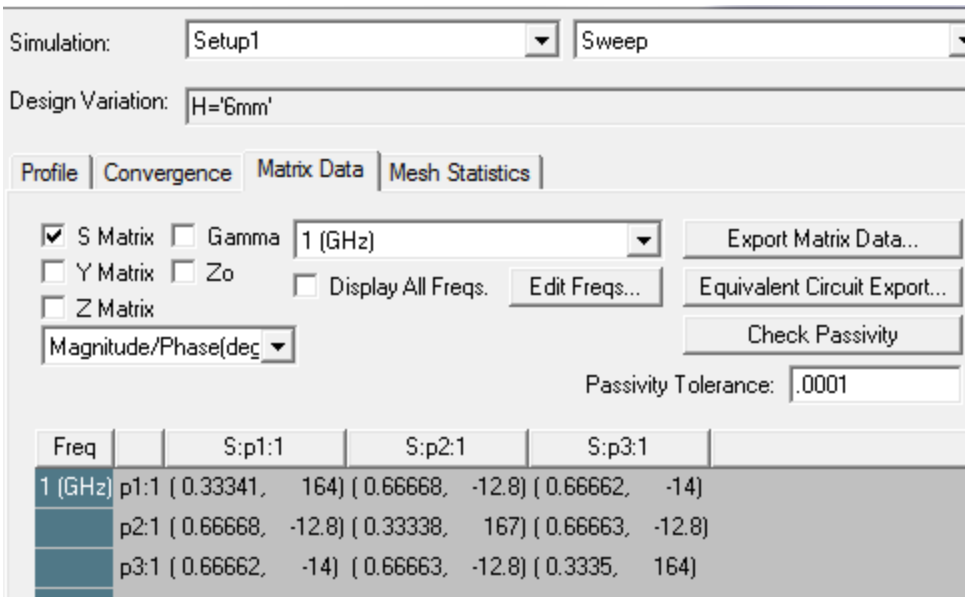
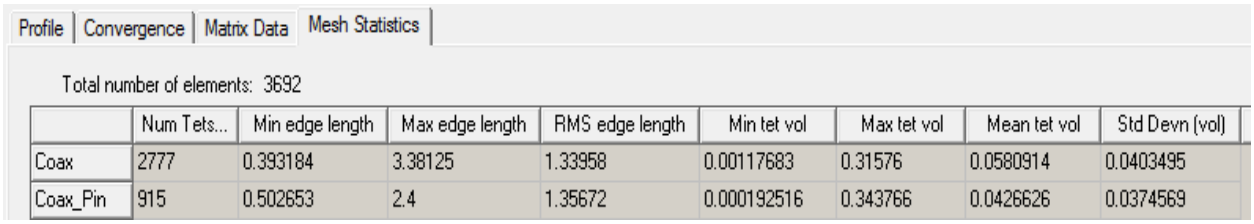


Figure 4-5 Matrix Data

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

Review the Mesh Statistics

1. Click **Mesh Statistics** to view data about mesh elements.



	Num Tets...	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Coax	2777	0.393184	3.38125	1.33958	0.00117683	0.31576	0.0580914	0.0403495
Coax_Pin	915	0.502653	2.4	1.35672	0.000192516	0.343766	0.0426626	0.0374569

Figure 4-6 Mesh Statistics

Create Reports

To create the rectangular plots for the S-parameters versus frequency perform the following steps.

1. Right click **Results** and select **>Create Modal Solution Data Report>Rectangular Plot** from the short-cut menu.

   This opens a **Create Report** window.

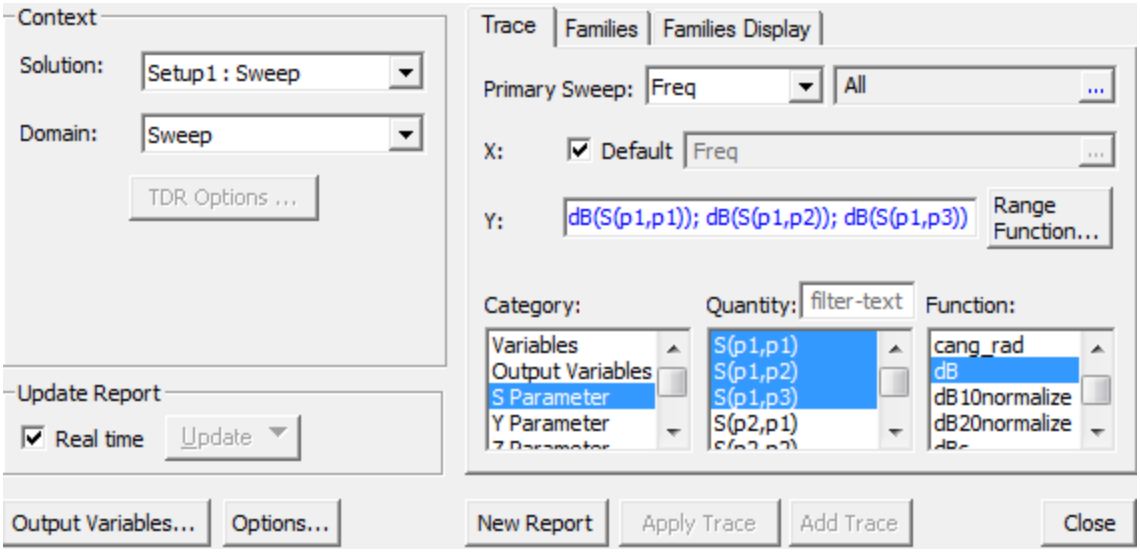


Figure 4-7 Report Window

2. Edit the fields in the **Report** window as shown in "Report Window" above.

   Note Select *dB* from the **Function** menu

3. Click the **New Report** button and then click **Close** to exit the dialog.
   HFSS generates the report.

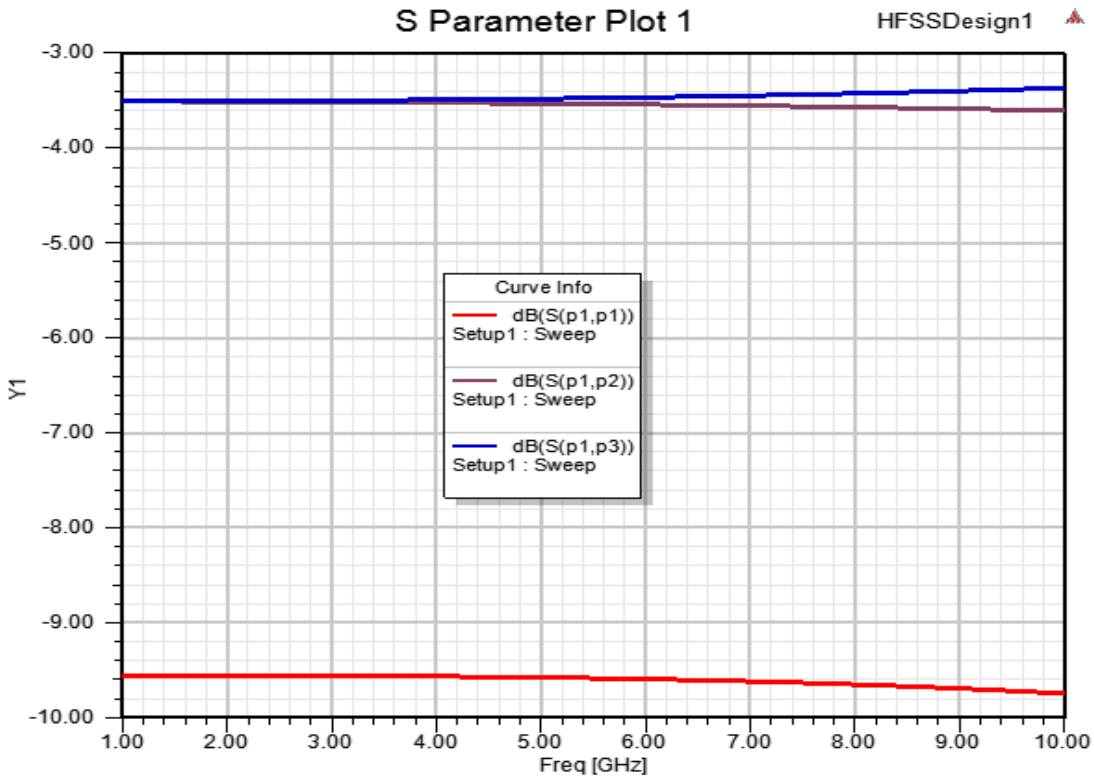


Figure 4-8 S Parameter Vs Frequency

Note The red curve represents S(p1, p1), the brown curve S(p1, p2), and the blue curve S (p1, p3).

Create Field Overlays

Minimize your plot and return to the model design.

1. Select an object to overlay fields.
2. On the Draw tab ribbon, click the **Select by Name** option.
   The **Select Object** dialog box appears.
3. Select the object *Coax* and click **OK**.
4. With the coax selected, right-click and go to **Plot Fields > E > Mag_E**.
   The **Create Field Plot** dialog box appears.

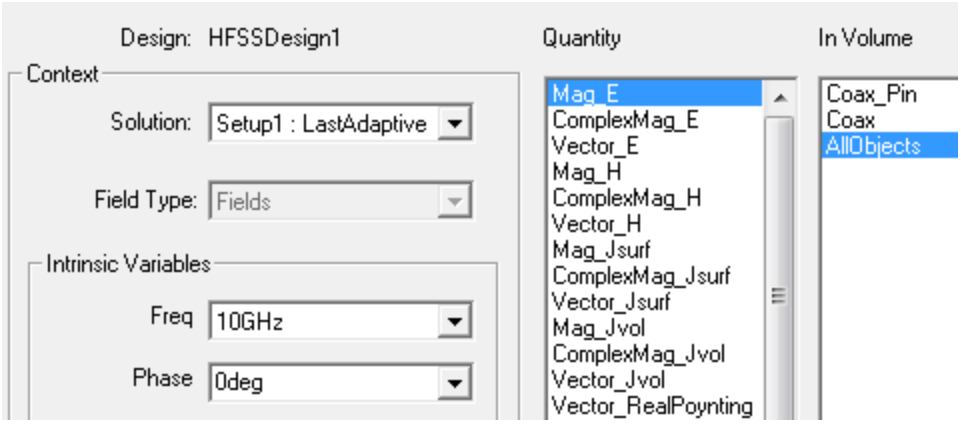


Figure 4-9 Field Plot dialog box

5. Edit the fields as shown in "Field Plot dialog box " above. Click **Done**.
   The fields are applied.

Modify Attributes of a Field Plot

1. On the toolbar click ***HFSS>Fields>Modify Plot Attributes***
   The **Select Plot Folder** dialog box appears.

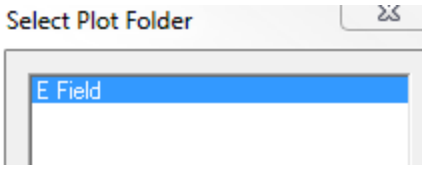


Figure 4-10 Select Plot Folder

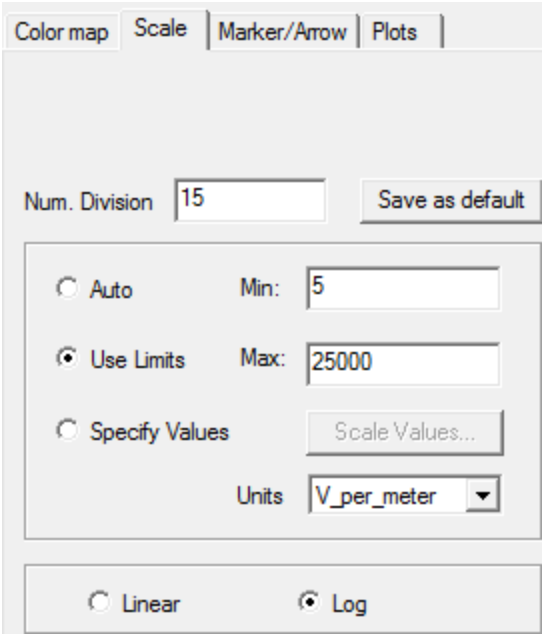


Figure 4-11 E Field dialog box

2. Select **E Field** and click **OK**.
   E Field dialog box appears.
3. Click **Scale** and edit the fields as shown in "E Field dialog box" above.
4. Click **Plot** and set the plot fields as in "The Plot tab" on the facing page.

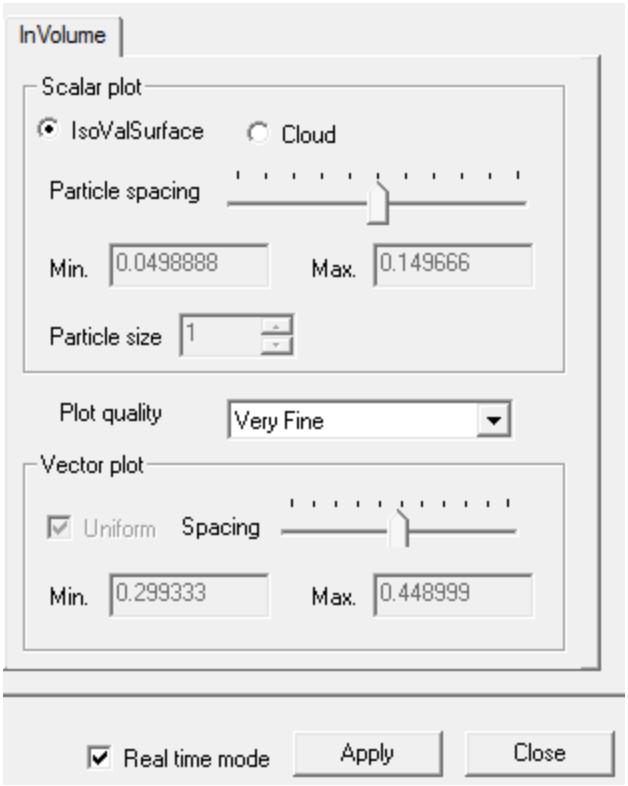


Figure 4-12 The Plot tab

5. Click **Apply** and then, **Close**.

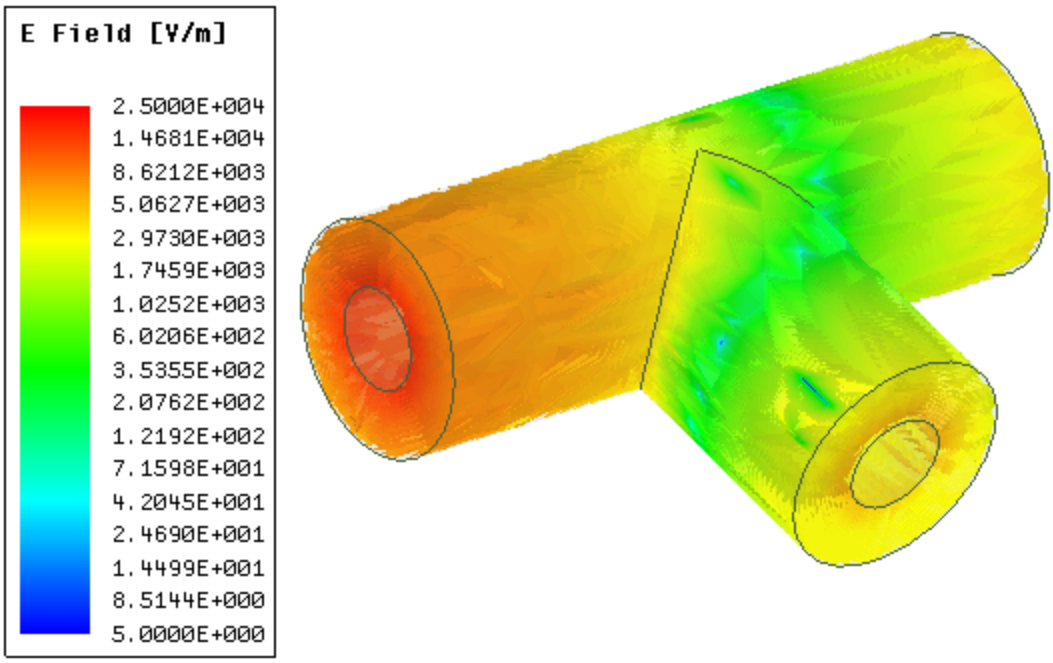


Figure 4-13 The E-Field of Coax-Tee

Plot Quality

Plot quality is a drop-down menu available in the **E-Field** dialog box. You can select from among the following options:

- Normal
- Coarse
- Fine
- Very Fine