Ansys 2025 R1

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with HFSS™: Coax Tee

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

Release 2025 R1
January 2025

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Conventions Used in this Guide

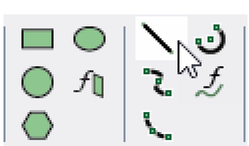
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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 you must type the word **copy**, then type a space, and then type **file1**.
    - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). 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** *filename*” means you must type the word **copy**, then type a space, and then type the name of the file.
- 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, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

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

  "Click **Draw** > **Line**"

This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- 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 *shortcut 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 shortcut 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 help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides -** opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

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.

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

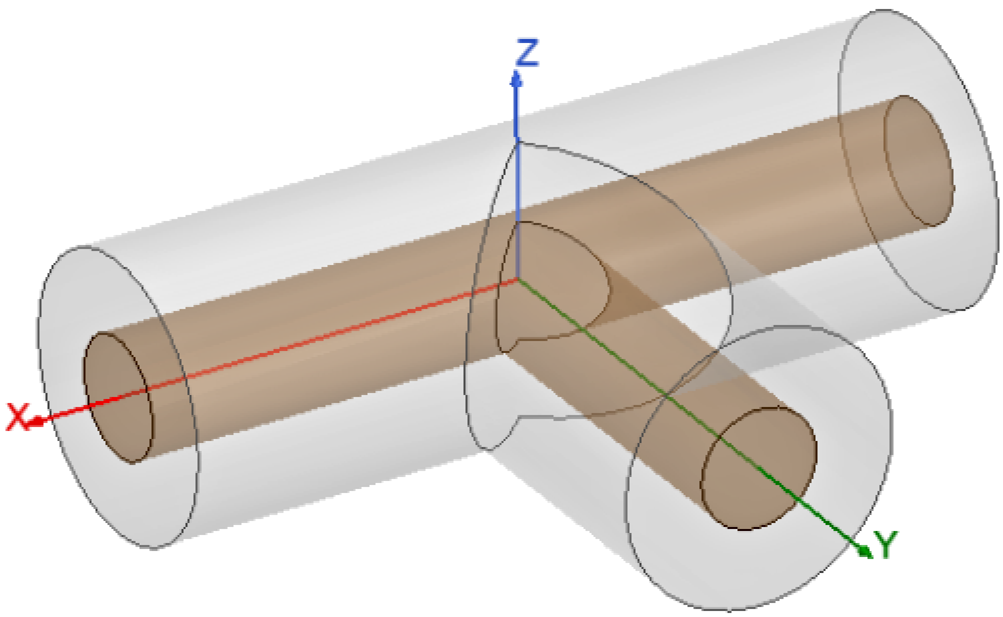


Figure 1-1: Coax Tee Model

2 - Set Up the Project

This chapter contains the following topics:

- Launch Ansys Electronics Desktop
- Set General Options
- Insert HFSS design
- Enable legacy view orientations
- Set Model Units (mm)
- Set Solution Type (Driven Modal)

Launch Ansys Electronics Desktop:

For convenience, a shortcut to the *Ansys Electronics Desktop* application is placed on your desktop during program installation. Optionally, you may want to pin the shortcut to your Windows Start Menu too.

1. Double-click **Ansys Electronics Desktop** (or click the same shortcut on your Start Menu) to launch the application.

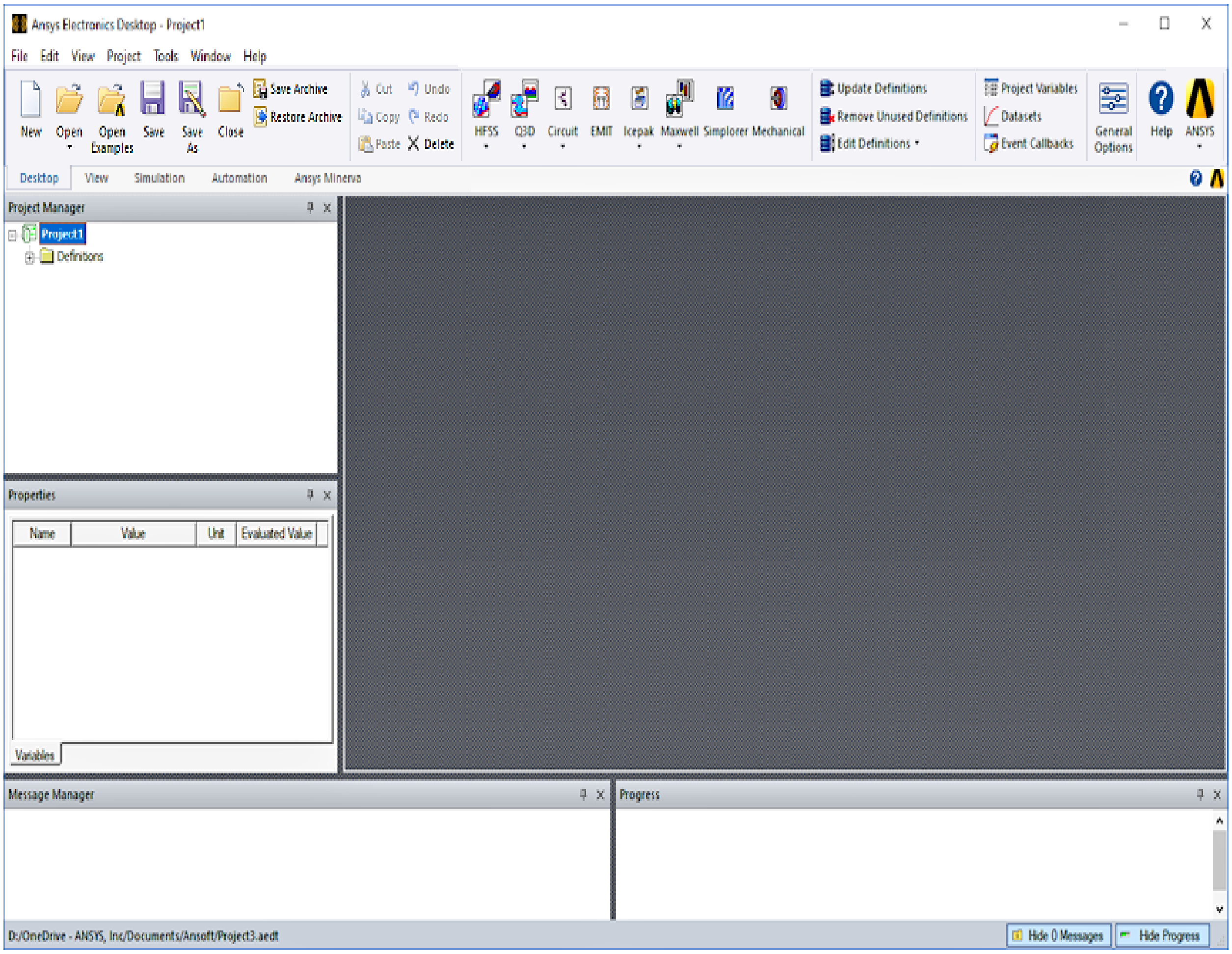


Figure 2-1: Ansys Electronics Desktop

> **Note:**
>
> Refer to the preceding figure. If the **Project*x*** folder is missing, click **New** on the **Desktop** ribbon tab to start a new project. This folder will be missing if you already had Electronics Desktop open to work on another project and closed that project without exiting the application. Also, if the **Project Manager** window does not appear, go to the **View** menu and enable it.

Set General 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** group.
3. Ensure all boundary assignment options are selected, as shown below:

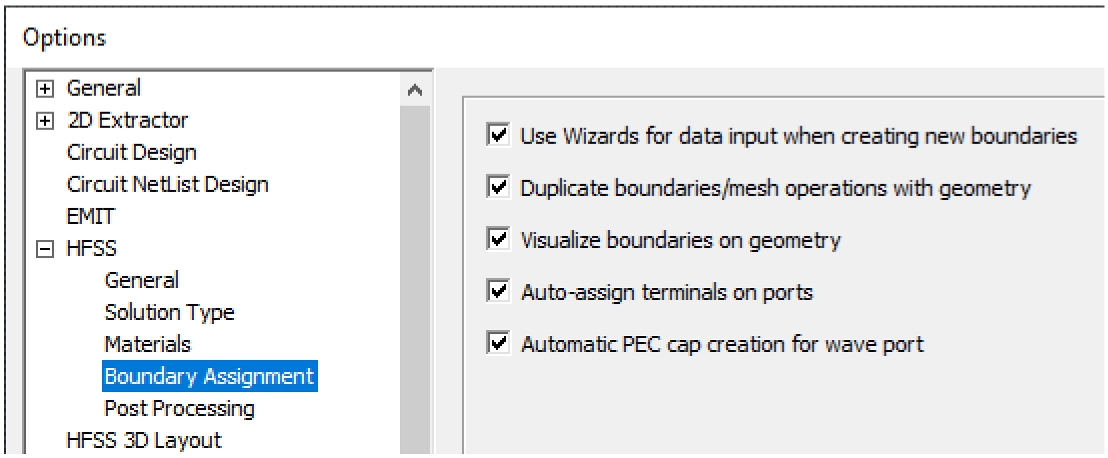
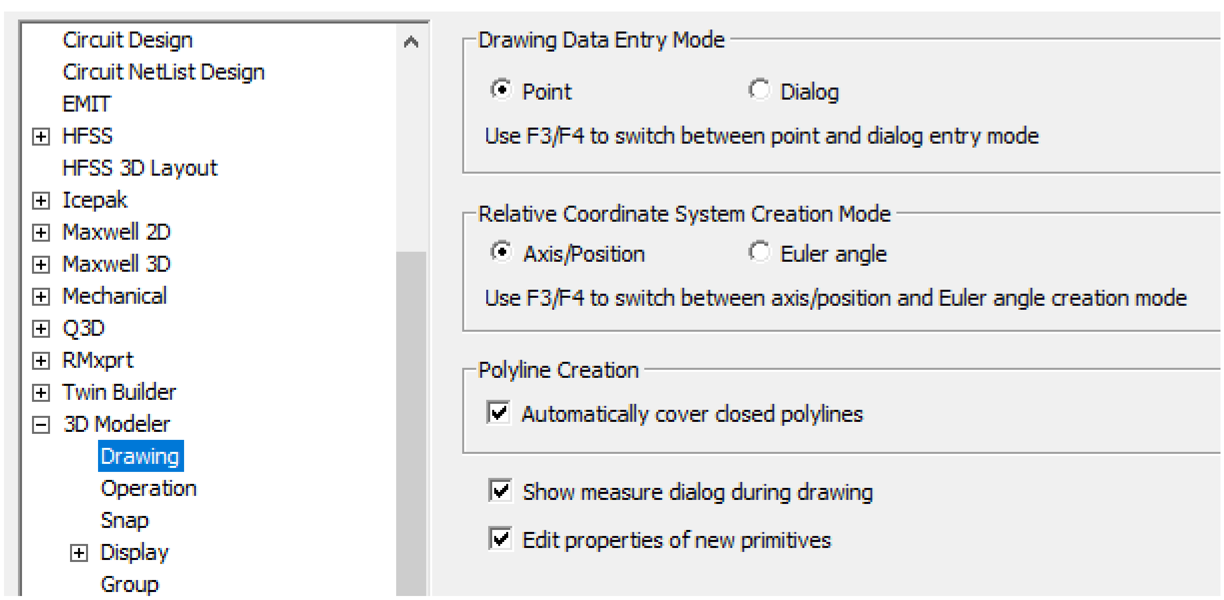


Figure 2-2: *Options* Dialog Box – *HFSS* > *Boundary Assignment* Group

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



> **Note:**
>
> The *Edit properties of new primitives* option causes a *Properties* dialog box to appear whenever you create a new object, which is convenient for immediately editing the object name, location, size, coordinate system, material assignment, or appearance.

6. Click **OK**.

Insert HFSS Design

1. On the **Desktop** ribbon tab, click **HFSS**:

   Either of the two red-boxed icons in the following figure insert an **HFSS** design into the project. The default action for the **HFSS** drop-down menu is to insert a regular HFSS design. Therefore, clicking the first icon has the same effect as accessing the drop-down menu and clicking the second icon.

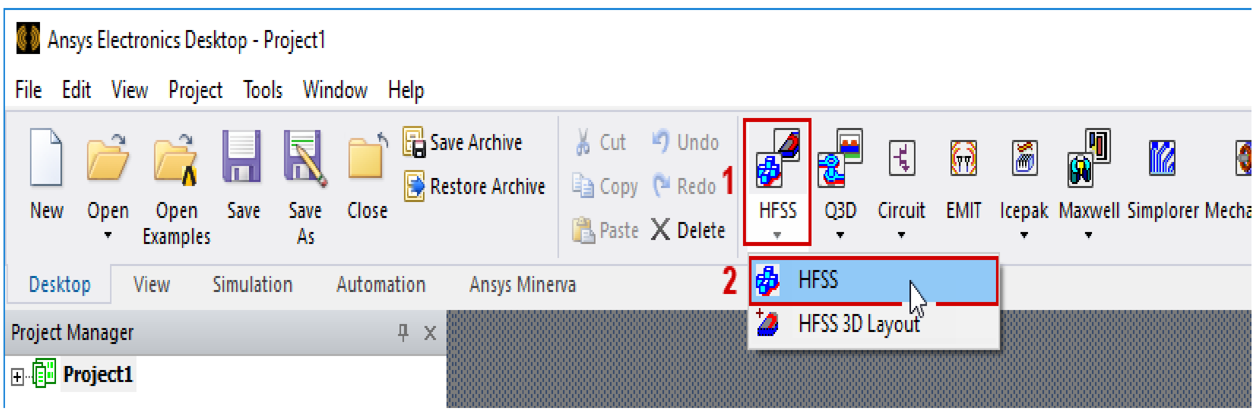


Figure 2-3: Insert HFSS Design

> **Note:**
>
> Adding an HFSS design type modifies the project. In the Project Manager, an asterisk appears after the project name whenever there are unsaved changes.

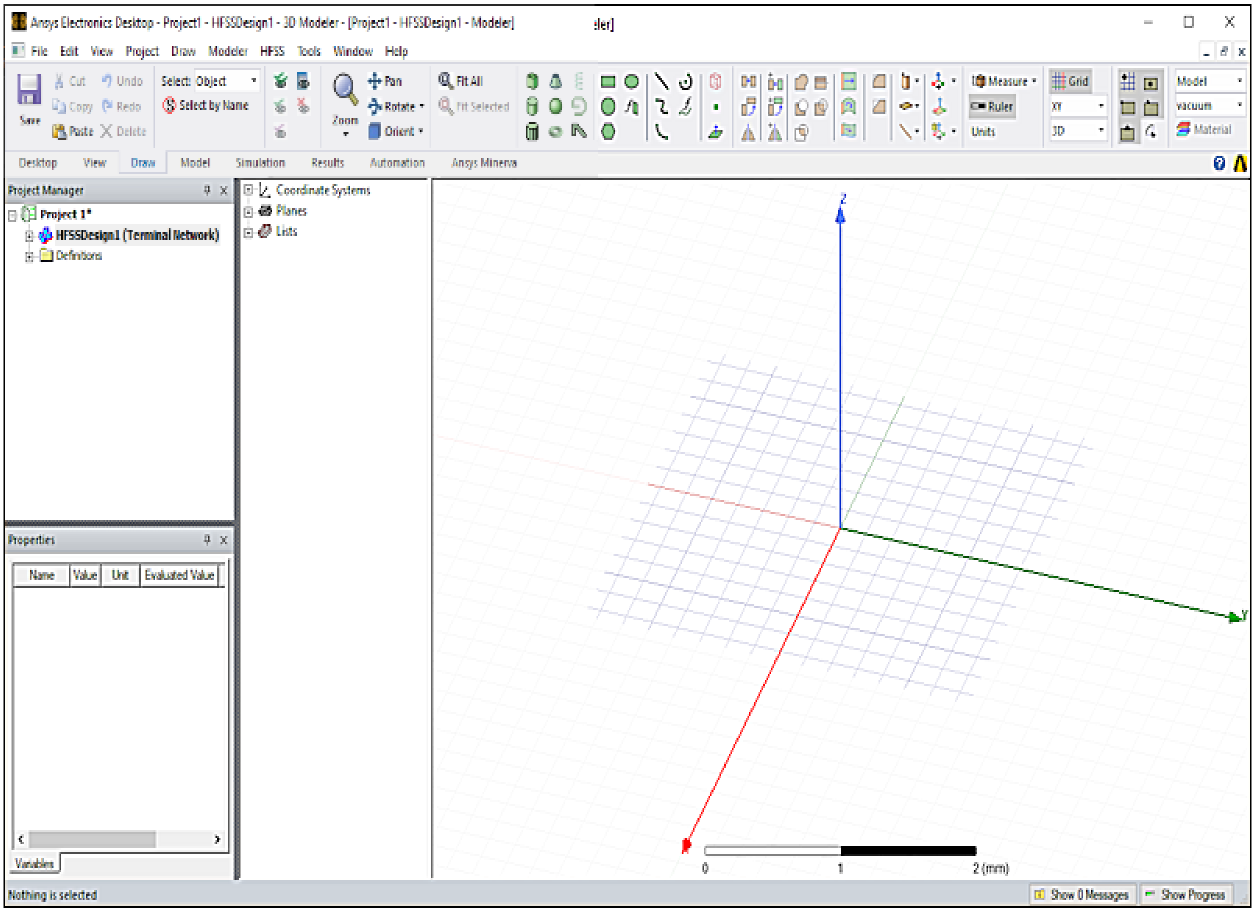


Figure 2-4: *HFSSDesign1* Added

2. Click ***Projectx****, press **F2**, and rename the project **Coax Tee**.

3.  **Save** your model.

> **Note:**
>
> Optionally, you can customize the appearance of the drawing area of the Modeler window in the following ways:
>
> - **Adjust the size of the axes:** From the menu bar, click **View** > **Coordinate System** and choose one of the available options.
> - **Modify the grid visibility or options:** From the menu bar, click **View** > **Grid Settings**. In the dialog box that appears, you can choose the grid type, style, density, and visibility.
> - **Toggle the Ruler visibility:** From the menu bar, click **View** > **Visibility** > **Ruler**.

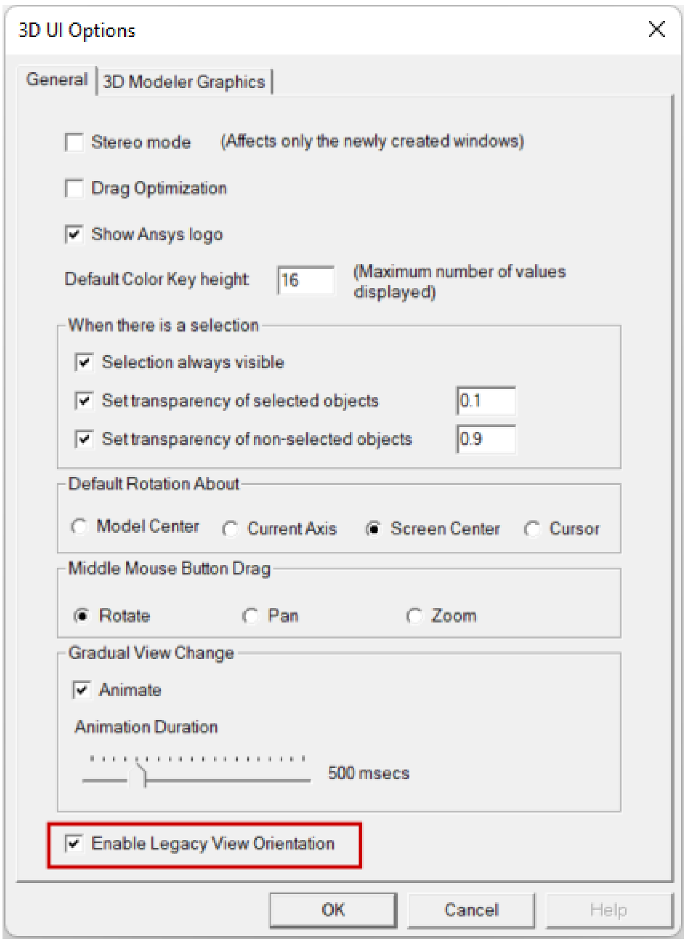
Enable Legacy View Orientations

This getting started guide was created based on standard view orientations that were in effect for version 2023 R2 and earlier of the Ansys Electronics Desktop application. For consistency between your experience and the views and instructions contained in this guide, select the *Enable Legacy View Orientation* option in the 3D UI Options dialog box, as follows:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Select **Enable Legacy View Orientation**:

3. Click **OK**.

   Changing the view orientation option does not change the model viewpoint that was in effect at the time.

4. On the **Draw** ribbon tab, click  **Orient** to change to the *Trimetric* view, which is the default legacy view orientation.

You do not have to select *Trimetric* from the *Orient* drop-down menu. The default view appears when you click *Orient*.

Although this option can only be accessed once a design is added to a project, it is a global option. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

At the end of this guide, you will be prompted to clear the *Enable Legacy View Orientation* option, if you prefer to use the view orientation scheme implemented for 2024 R1 and newer versions going forward.

For a comparison of the legacy and current view orientations, search for *"View Options: 3D UI Options"* in the HFSS help. Additionally, views associated with **Alt + double-click** zones have been redefined. The current orientations are shown in the help topic, *"Changing the Model View with Alt+Double-Click Areas."*

Set Model Units (mm)

Define the model units as follows:

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

   The **Set Model Units and Max Extent** dialog box appears.

2. Select **mm** (milimeters) from the **Select units** drop-down menu if it is not already selected.

   Keep the **Rescale to new units** and **Advanced** options cleared.

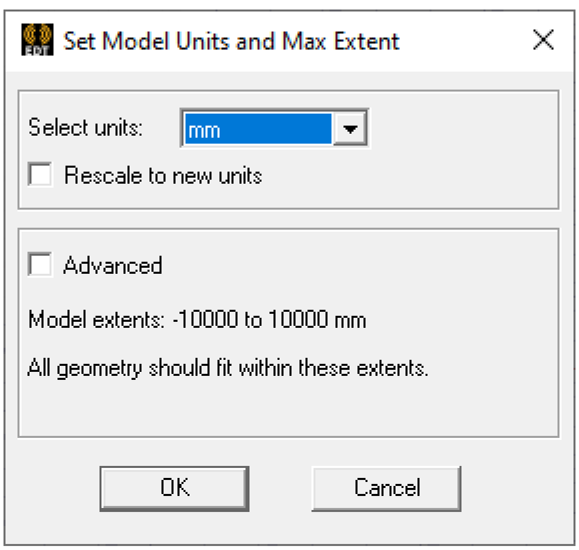


Figure 2-5: ***Set Model Units and Max Extent*** **Dialog Box**

3. Click **OK**.

Set Solution Type (Modal)

Specify the design's solution type as follows:

1. Using the menu bar, click **HFSS** > **Solution Type**.

   The *Solution Type* dialog box appears.

2. Under *Options* select **Modal** and also verify that the remaining settings are as shown in the following image:

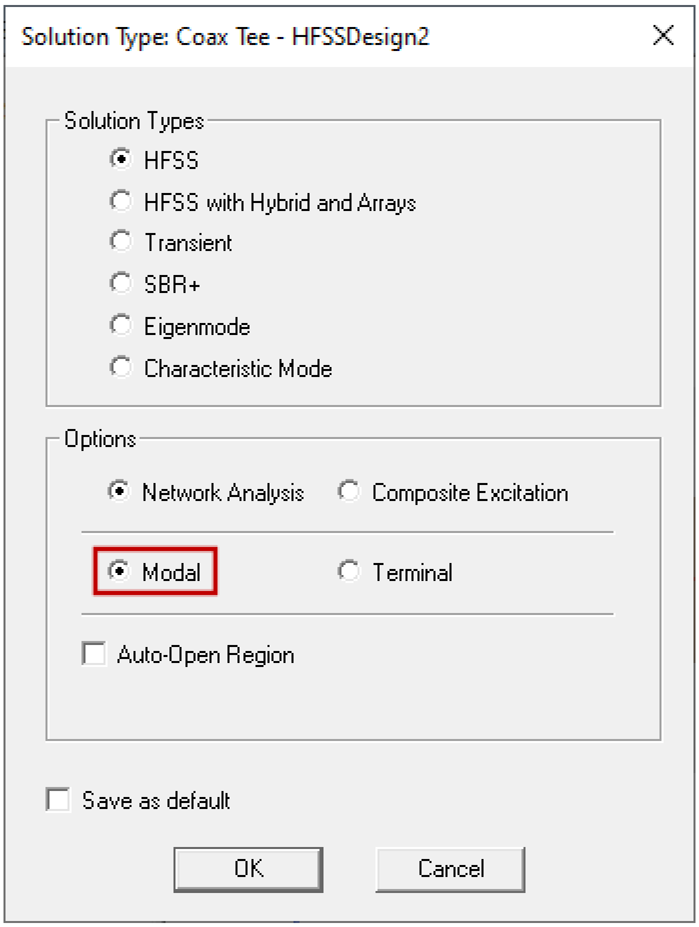


Figure 2-6: *Solution Type* Dialog Box

3. Click **OK**.

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

3 - Create the Model

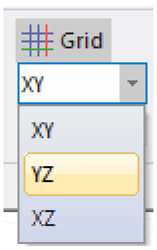
The outline of the process to create the model is as follows:

- 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

Before creating any objects, choose a different grid plane. You will create the Coax Tee along the YZ plane, instead of the default XY plane. Specify the drawing plane on which you want to display the grid, as follows:

1. On the **Draw** tab ribbon, choose **YZ** from the **Drawing Plane** drop-down menu (immediately below *Grid*).



The grid in the Modeler window is now displayed on the YZ plane. When you draw objects by clicking in the Modeler window, the points will snap to a grid point along the YZ plane.

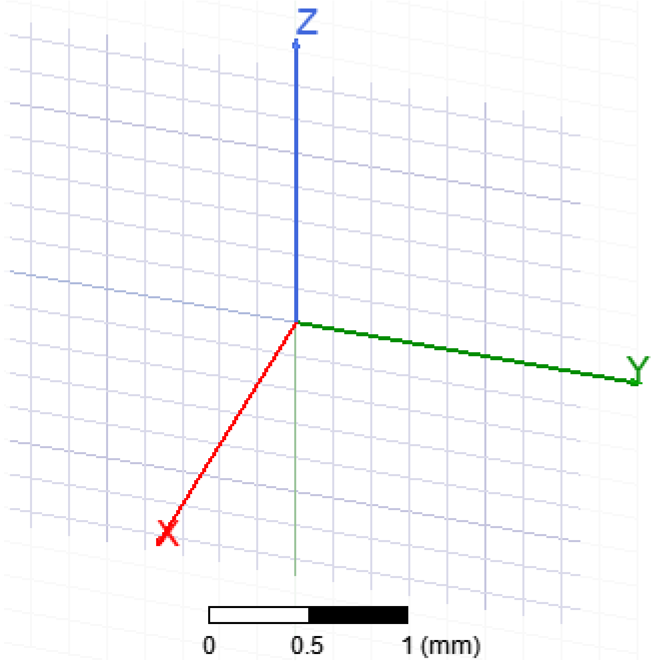


Figure 3-1: YZ Grid Plane

Create the Coax Pin

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

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

The cursor changes to a snapping point indicator.

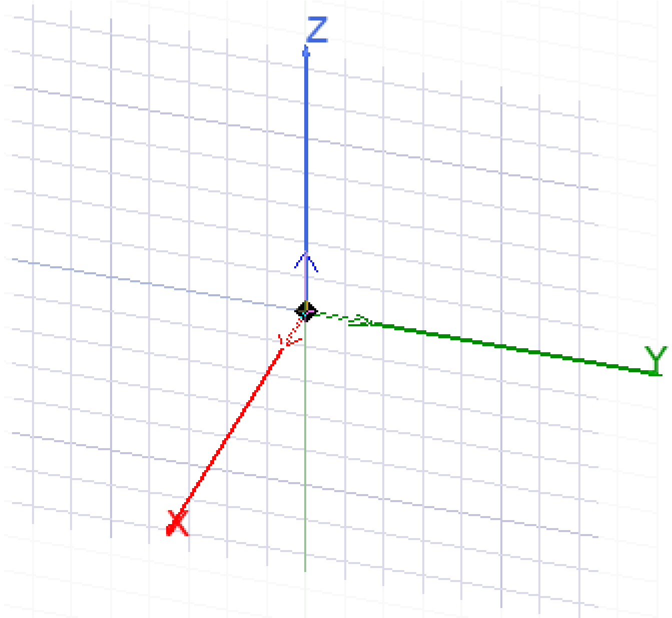


Figure 3-2: Grid Snapping Point Indicator

2. Click on the global origin to define the center of a base circle.

   The **YZ** mini axes appear.

> **Tip:**
>
> Since the completed cylindrical part's base will be centered at the origin, snapping to the origin saves you the trouble of redefining the center position later.

3. Move the cursor to form a circle with an arbitrary radius, and click the mouse again.

   The mini **X** axis appear for the height definition.

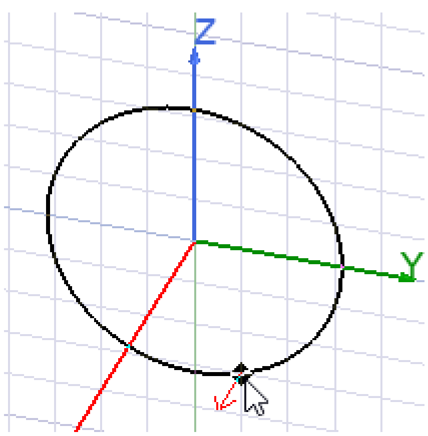


Figure 3-3: Arbitrary Base Circle (with Height Mini Axis Displayed)

4. Move the cursor to form a cylinder with an arbitrary height.

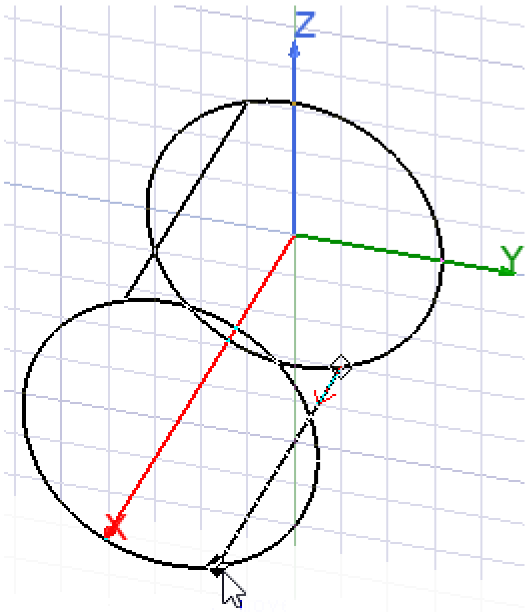


Figure 3-4: Arbitrary Cylinder Height Defined

Click once more to complete the cylinder. This operation generates an initial cylinder that you will modify in the next step.

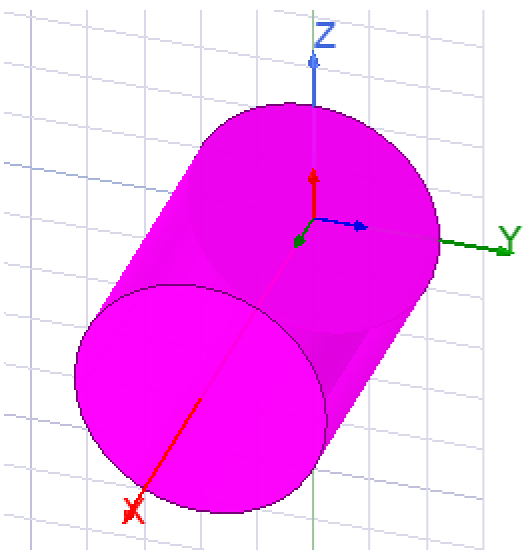


Figure 3-5: Initial Cylinder Completed

As soon as you click for the third time, the *Properties* dialog box appears.

5. On the **Command** tab of the *Properties* dialog box, edit the fields as shown in the following figure:

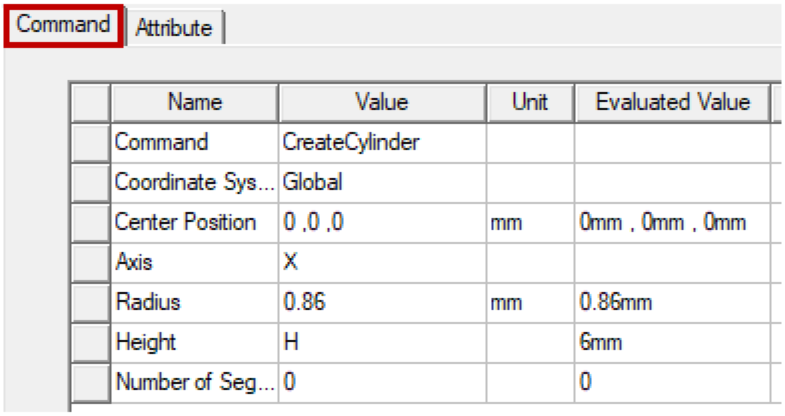


Figure 3-6: Coax Pin Properties – *Command* Tab

After typing **H** for the **Height** value and pressing **Enter**, the *Add Variable* dialog box appears.

6. Define **H** as shown in the following figure and click **OK**. Do not close the *Properties* dialog box yet.

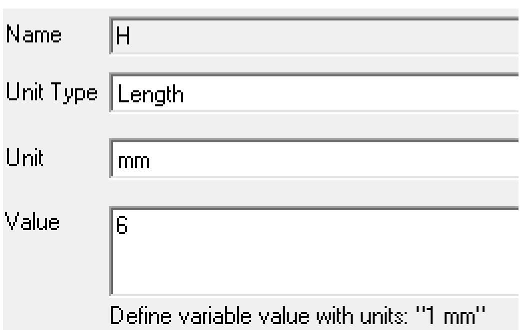


Figure 3-7: Defining H – *Add Variable* Dialog Box

7. On the **Attribute** tab of the *Properties* dialog box, change the object **Name** to **Coax_Pin**.
8. From the **Materials** drop-down menu, select **Edit**.

   The *Select Definitions* window appears:

   a. Type **pec** in the **Search by Name** text box. The material, *pec* (perfect electrical conductor) is highlighted in the materials list.

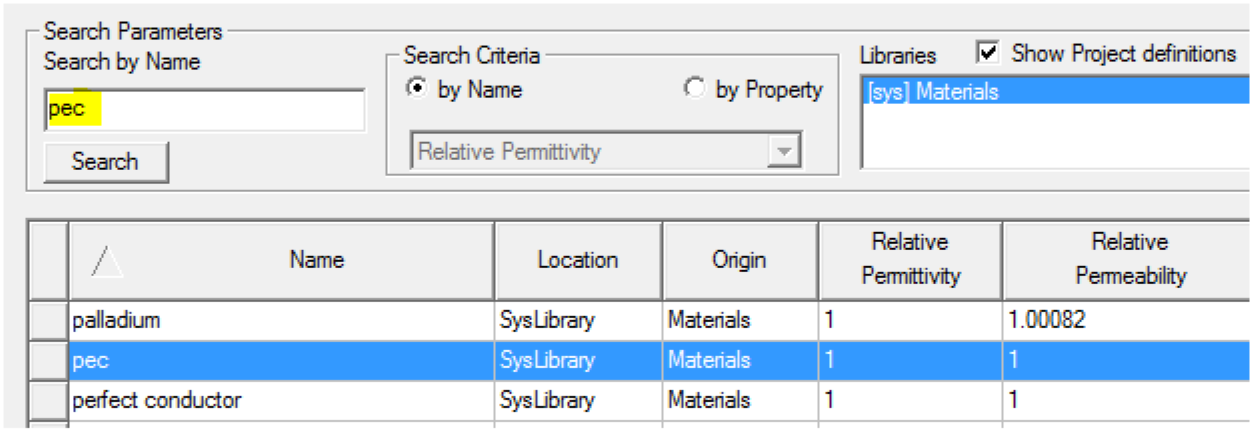


Figure 3-8: *Select Definitions* Window

b. Click **OK** to complete the selection.

9. Clear the **Material Appearance** option if it is selected.
10. Specify a desired **Color** for the electrical conductor. The images in this guide are based on a *brown* color (**Red: 128, Green: 64, Blue: 0**).
11. Set the **Transparent** value to **0.6**.
12. Click **OK** to close the *Properties* dialog box.

> **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.

13. Press **Ctrl+D** to fit the view and click in the Modeler window's background area to deselect the *Coax_Pin*.
14. If you want to hide the grid, click **View** > **Grid Settings** from the menu bar. Select the **Hide** option under *Grid Visibility* and click **OK**.

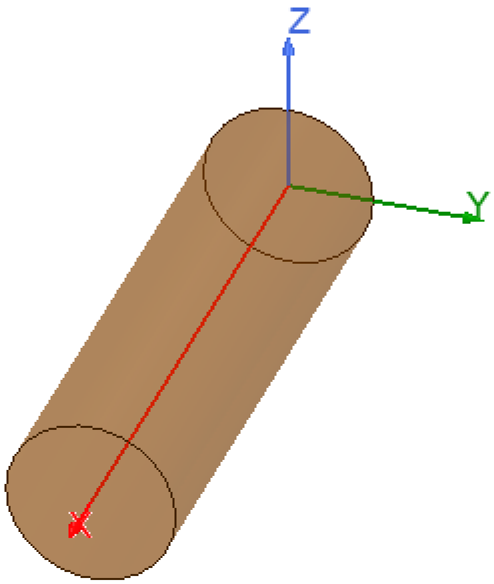


Figure 3-9: Completed *Coax_Pin* Cylinder

Create the Coaxial Cylinder

To create the coaxial cylinder, which encloses the coaxial pin, draw a cylinder freehand and then edit its properties.

1. Draw a cylinder of arbitrary size.

> **Tip:**
>
> You can snap to the center of the *Coax_Pin* cylinder's base circle for the first click to set the location correctly, minimizing the properties that you have to edit.

2. Edit the fields in the **Command** tab of the *Properties* dialog box as shown in the following figure:

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	H		6mm
Number of Seg...	0		0

Figure 3-10: Coax Properties – *Command* Tab

3. On the **Attribute** tab, make the following changes:
   a. Change the cylinder's **Name** to **Coax**.
   b. Select **vacuum** from the **Material** drop-down menu.
   c. Specify a *light gray* color (**Red: 192, Green: 192, Blue: 192**).
   d. Set the **Transparent** value to **0.75**.
4. Click **OK** to close the *Properties* dialog box and click in the background area to deselect the *Coax* part.

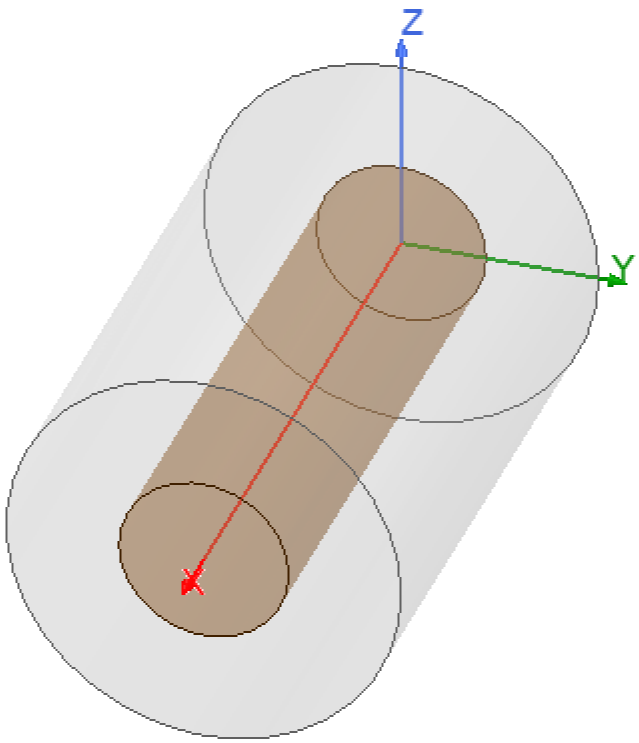


Figure 3-11: *Coax* Cylinder Completed

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.

Assigning the excitation before the model is complete is advantageous because, when you duplicate the two cylindrical parts to produce the complete coax tee, the excitation will be duplicated too.

1. Press **F** to switch to the *face* selection mode.
2. Click the *Coax* positive-X end face, as shown below:

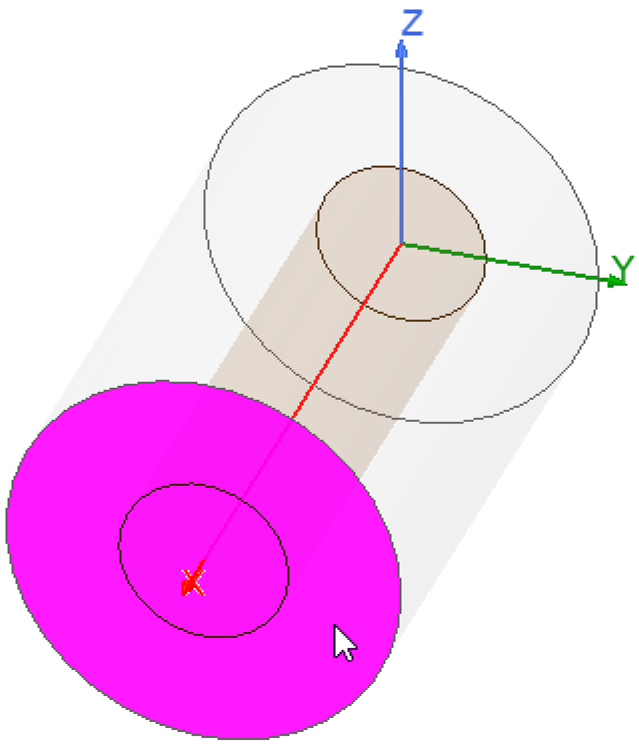


Figure 3-12: Excitation Face Selected

3. Right-click and select **Assign Excitation** > **Port** > **Wave Port** from the shortcut menu.

   The *Wave Port : General* dialog box appears.

4. Type **P1** in the **Name** text box and click **Next**.

   The *Wave Port : Post Processing* dialog box appears.

5. Under *Port Renormalization*, set the fields as shown below and then click **Finish**.

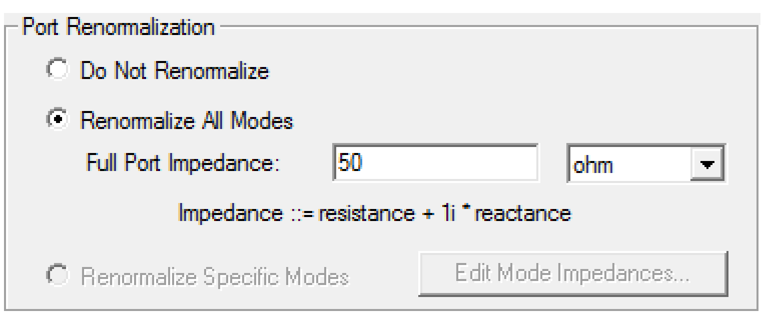


Figure 3-13: Post Processing dialog box

6. Clear the current selection. Then, under *Excitations* in the Project Manager, click **P1** to display the wave port excitation on the model:

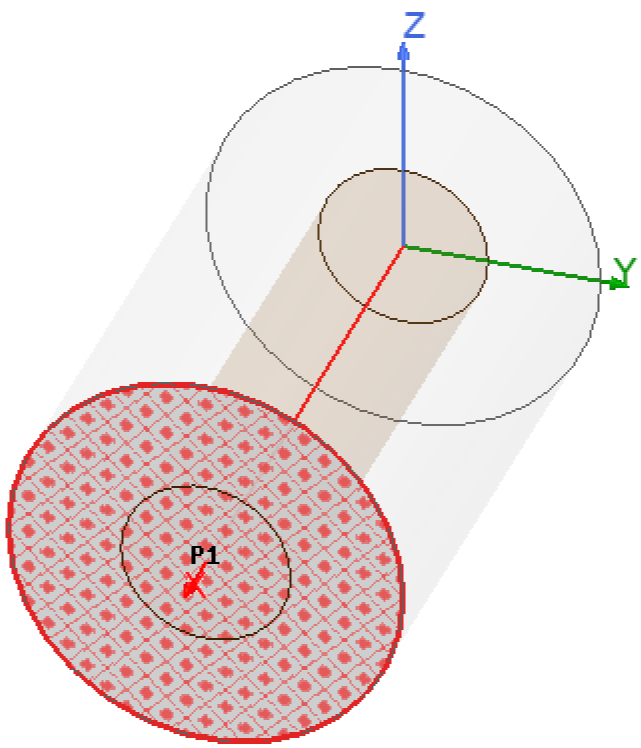


Figure 3-14: Visualization of Wave Port Excitation, *P1*

Duplicate to Create the Tee

You will select the two cylinders (*Coax_Pin* and *Coax*) and duplicate them around the Z axis to form a tee. Along with the geometry, the wave port excitation you assigned will also be duplicated

1. Press **O** to switch to the *object* selection mode.
2. Press **CTRL+A** to select all model objects.
3. Right-click and choose **Edit** > **Duplicate** > **Around Axis**.

   The *Duplicate Around Axis* dialog box appears.
4. Edit the settings as shown below and then click **OK**.

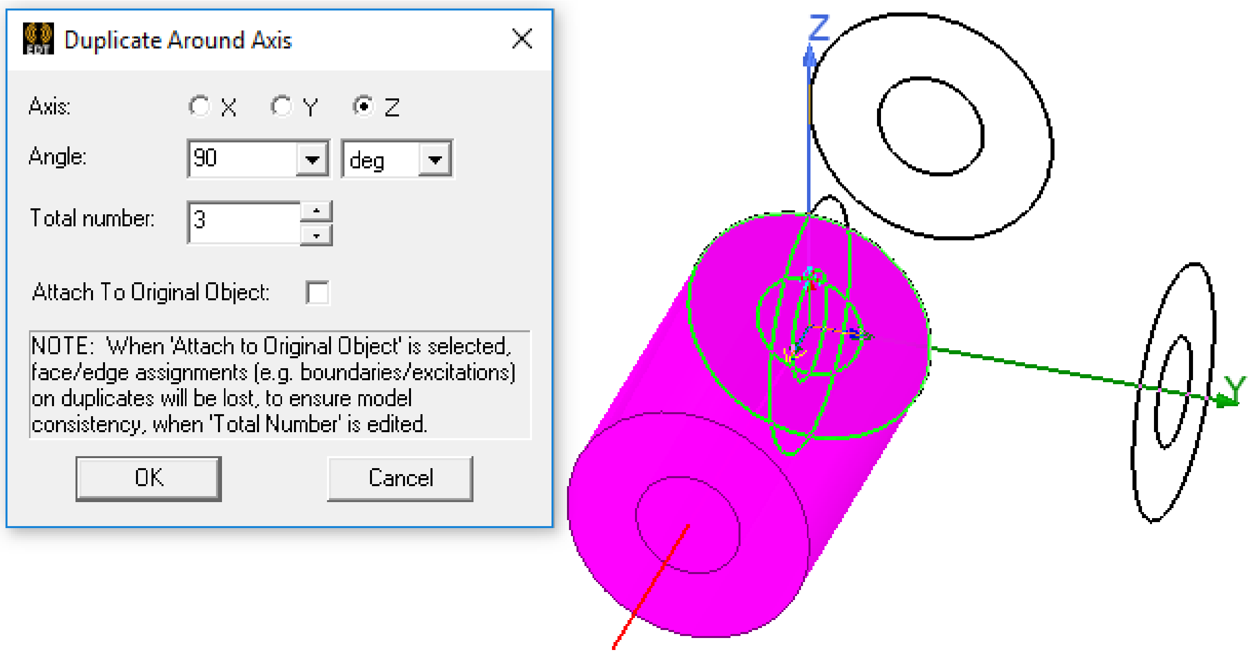


Figure 3-15: *Duplicate Around Axis* Settings

5. Click **OK** to close the *Properties* dialog box and click in the background area to clear the current selection.

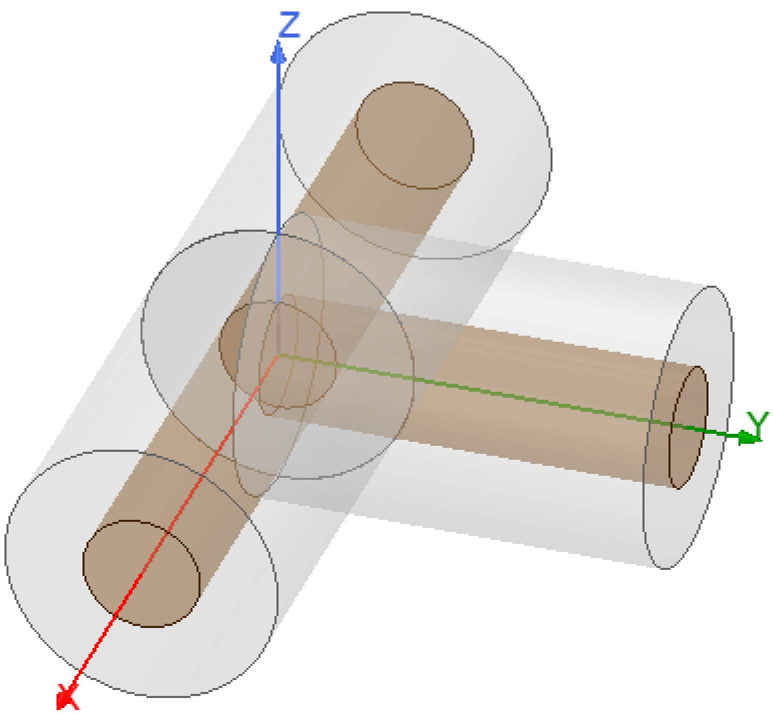


Figure 3-16: The Nearly Completed Tee

The numbers in the wave port names (for the two duplicate ports) have been incremented automatically. *P1, P2,* and *P3* now appear under *Excitations* in the Project Manager:

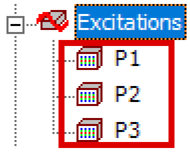


Figure 3-17: Original and Duplicate Wave Ports in Project Manager

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. On the **Draw** ribbon tab, ensure that the **Select** option is **Object**. Then, click **Select by Name**.

   The *Select Object* dialog box appears.

2. Select the following objects, in the order specified, and click **OK**:
   - **Coax_Pin**
   - **Coax_Pin_1**
   - **Coax_Pin_2**.

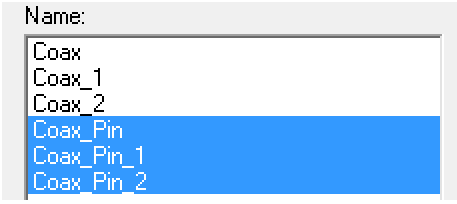


Figure 3-18: Select Conductors to Unite

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

   There is now a single object, *Coax_Pin*, listed under the material *pec* in the History Tree. Expand the *Unite* subfolder to see the original objects that were merged into it:

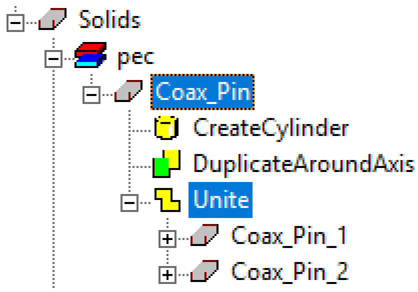


Figure 3-19: History Tree - United *Coax_Pin* Conductor

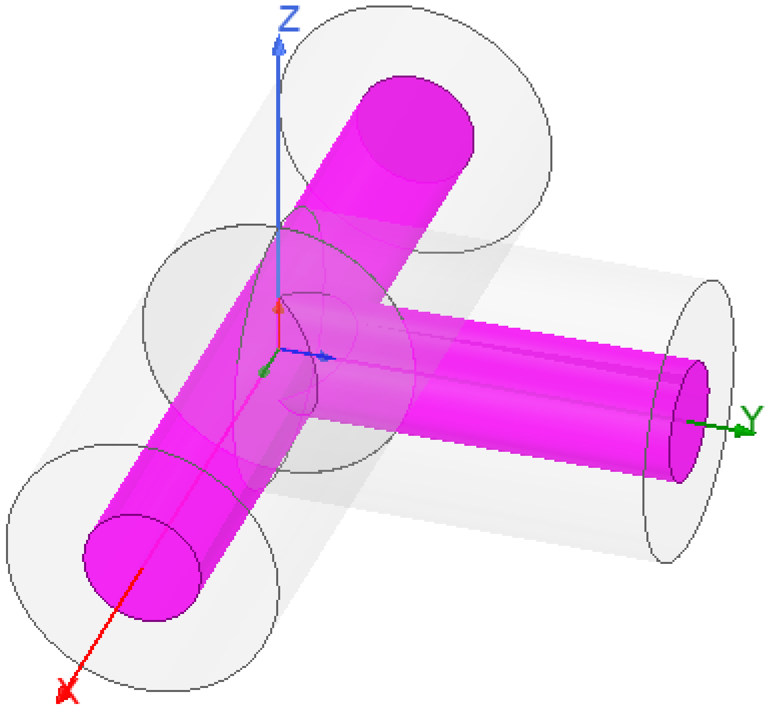


Figure 3-20: The United Conductors

Unite the Coaxial Cylinders

After uniting the co-axial cylindrical pins (the conductors) to form a tee-shaped structure, the next step is to unite the enclosure cylinders, as follows:

1. On the **Draw** ribbon tab, click **Select by Name**.
2. Select the following objects, in the specified order, and click **OK**:
    - **Coax**
    - **Coax_1**
    - **Coax_2**

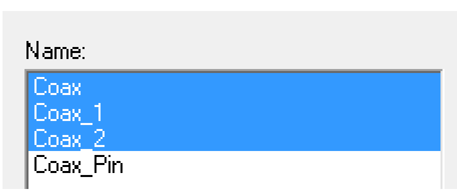


Figure 3-21: Select Coax Objects to Unite

> **Note:**
>
> Leave out Coax_Pin, which is the internal conducting part of the tee, separate from the enclosure.

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

   The enclosure objects are united:

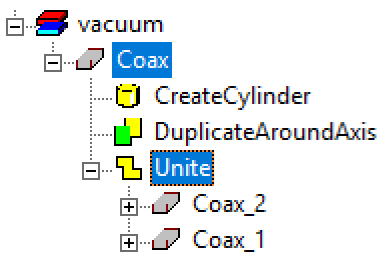


Figure 3-22: History Tree – United *Coax* Enclosure

4. Clear the current selection. Rotate your model for a good view of the tee intersection. Your model should look like the following figure:

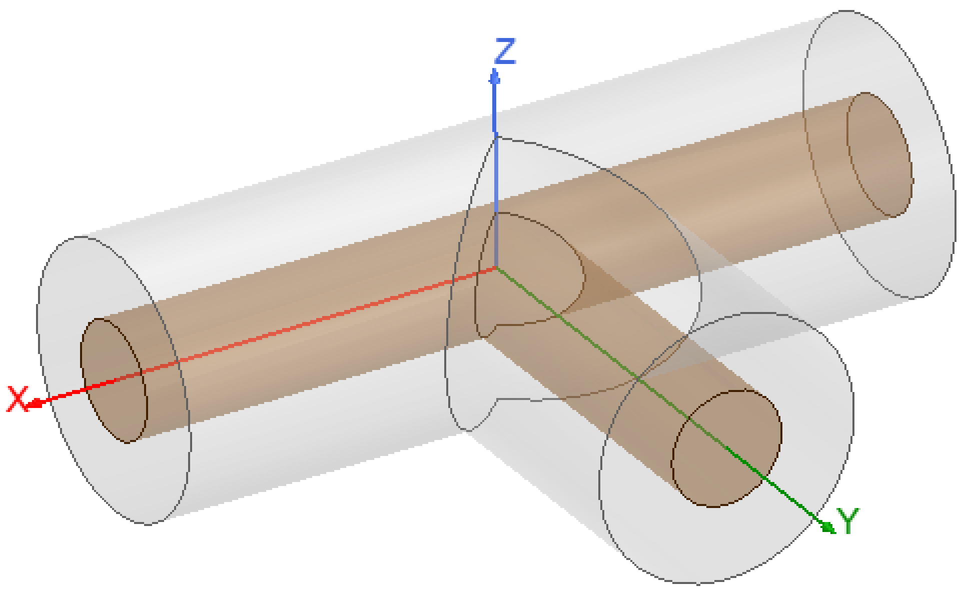


Figure 3-23: The Completed *Coax Tee* Geometry

4 - Analyze the Model

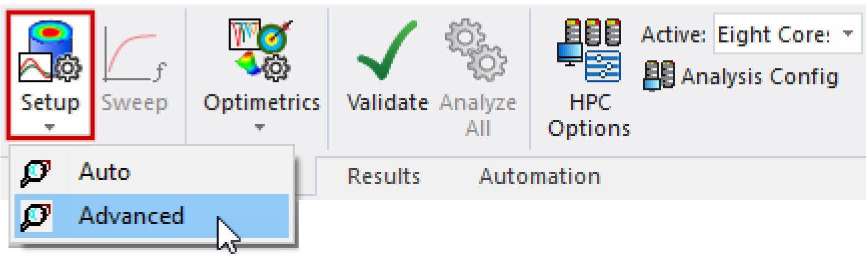
This chapter contains the following topics:

- Add Solution Setup
- Add Frequency Sweep
- Validate the Design
- Analyze the Design
- Review Solution Data:
    - Review the Profile Tab
    - Review the Convergence Tab
    - Review the Matrix Data Tab
    - Review the Mesh Statistics Tab
- Create the Reports
- Create Field Overlays:
    - Edit Field Overlay Sources
    - Modify Attributes of a Field Plot
    - Animate the E-Field Overlay

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. On the **Simulation** ribbon tab, select **Setup** > **Advanced**:



   The *Driven Solution Setup* dialog box appears.

2. Under the *General* tab, edit the settings as shown below:

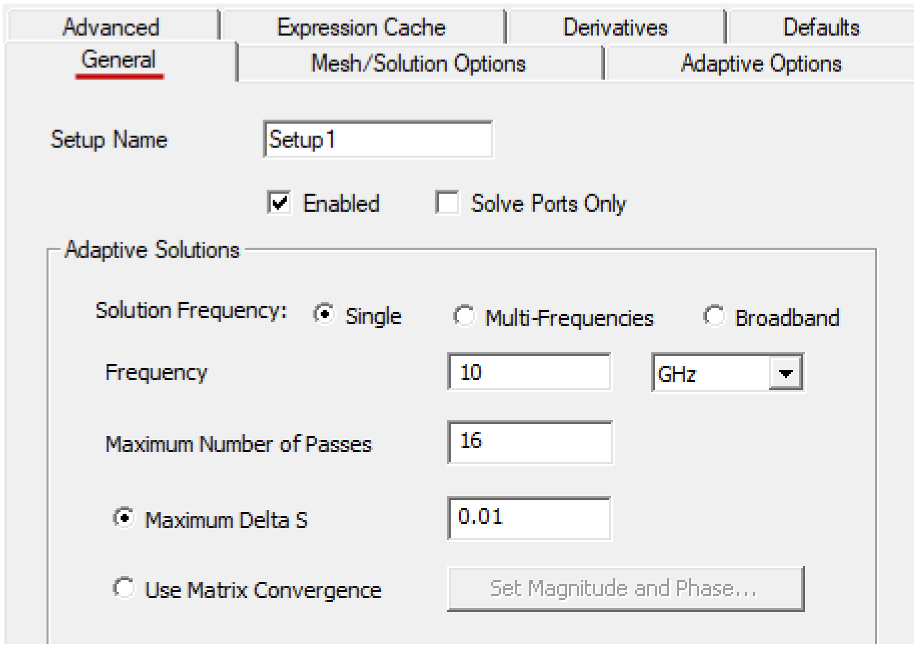


Figure 4-1: *Driven Solution Setup* Dialog Box – *General* Tab

3. Click **OK**.

   Because you have already assigned at least one port, the *Edit Frequency Sweep* dialog box opens automatically as soon as you finish adding a solution setup. You will define the sweep in the next procedure.

Add Frequency Sweep

The *Edit Frequency Sweep* dialog box should already be open. Define the sweep as follows:

1. Specify the following settings:
   a. *Sweep Type:* **Fast**
   b. *Distribution Type:* **Linear Step**
   c. *Start:* **1GHz**
   d. *End:* **10GHz**
   e. *Step Size:* **0.1GHz**
   f. *Save Fields:* **Selected**

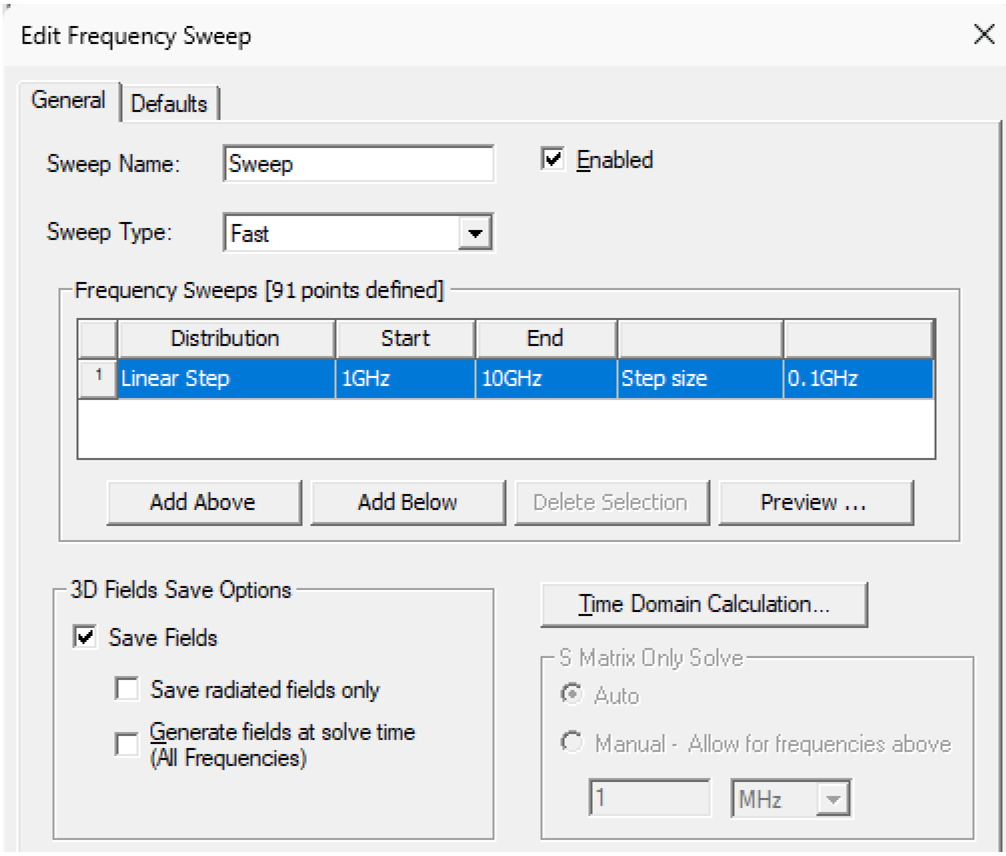


Figure 4-2: *Edit Frequency Sweep* Dialog Box

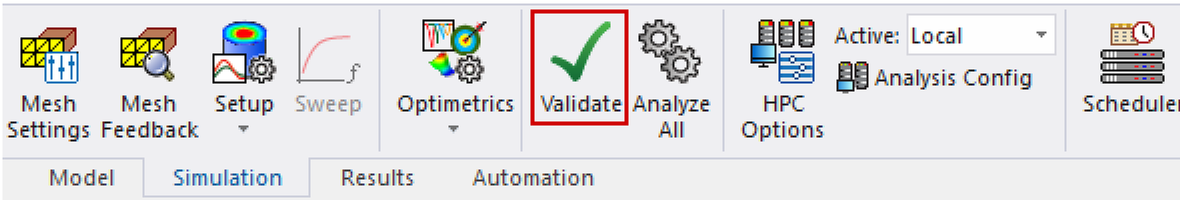
> **Note:**
>
> For the purpose of this exercise, the **Sweep Type** is **Fast**, even though the bandwidth of this solution is relatively wide. A fast sweep requires the least solution time and also has the advantage of providing field information (via the *Save Fields* option).

2. Click **OK**.

Validate and Analyze the Design

Before you run an analysis, the model has to pass the validation check to confirm your design is set up properly.

1. On the **Simulation** ribbon tab, click **Validate**.



The *Validation Check* window appears. A green check mark next to each item indicates that the model is ready to be analyzed.

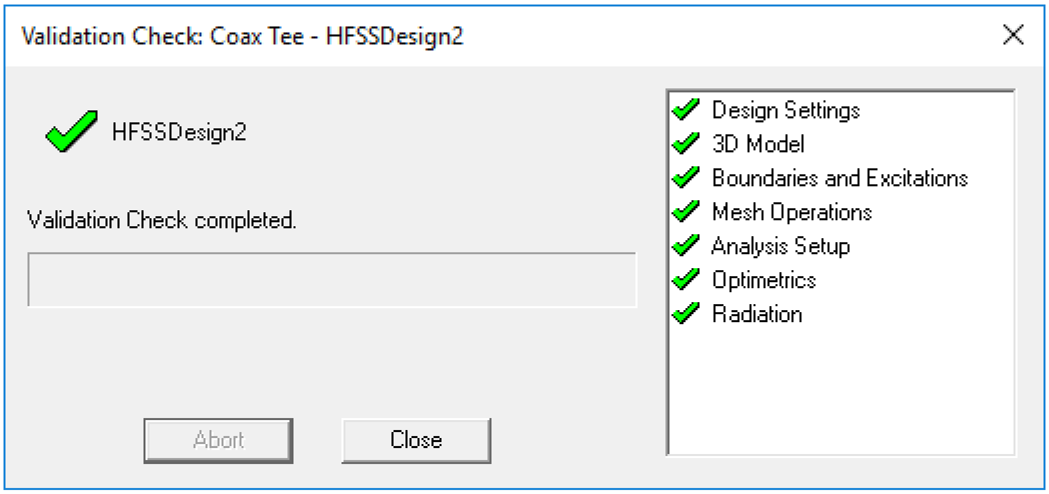
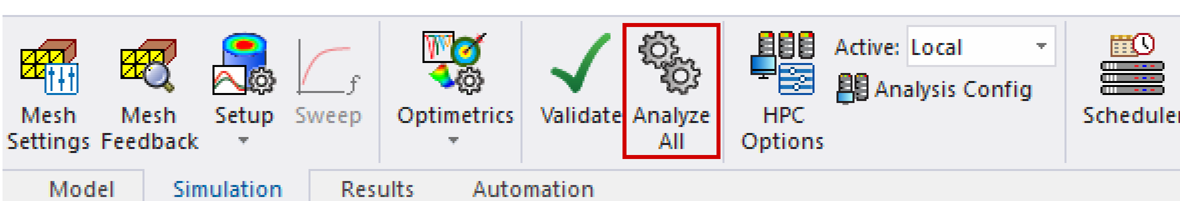


Figure 4-3: *Validation Check* Window

2. Click **Close**.

> **Note:**
>
> For HFSS projects, warnings may appear in the *Message Manager* window. Some of these messages warn you of potential problems, but you are not always required to take any action.

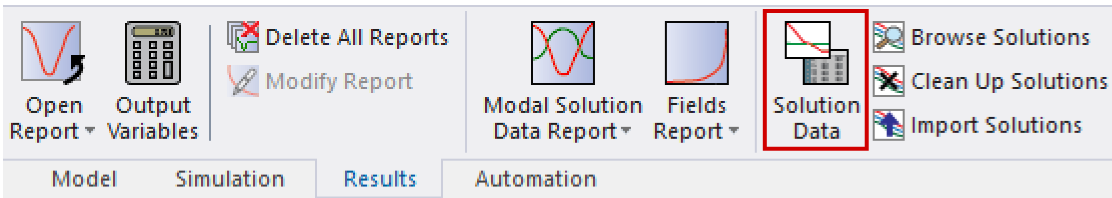
3.  **Save** your project.
4. On the **Simulation** ribbon tab, click **Analyze All**.



> **Note:**
>
> This model is simple and will solve in a relatively short time. If the analysis is successful, the *Message Manager* will notify you of a normal completion of the simulation.

Review Solution Data

1. On the **Results** tab ribbon, click **Solution Data**.



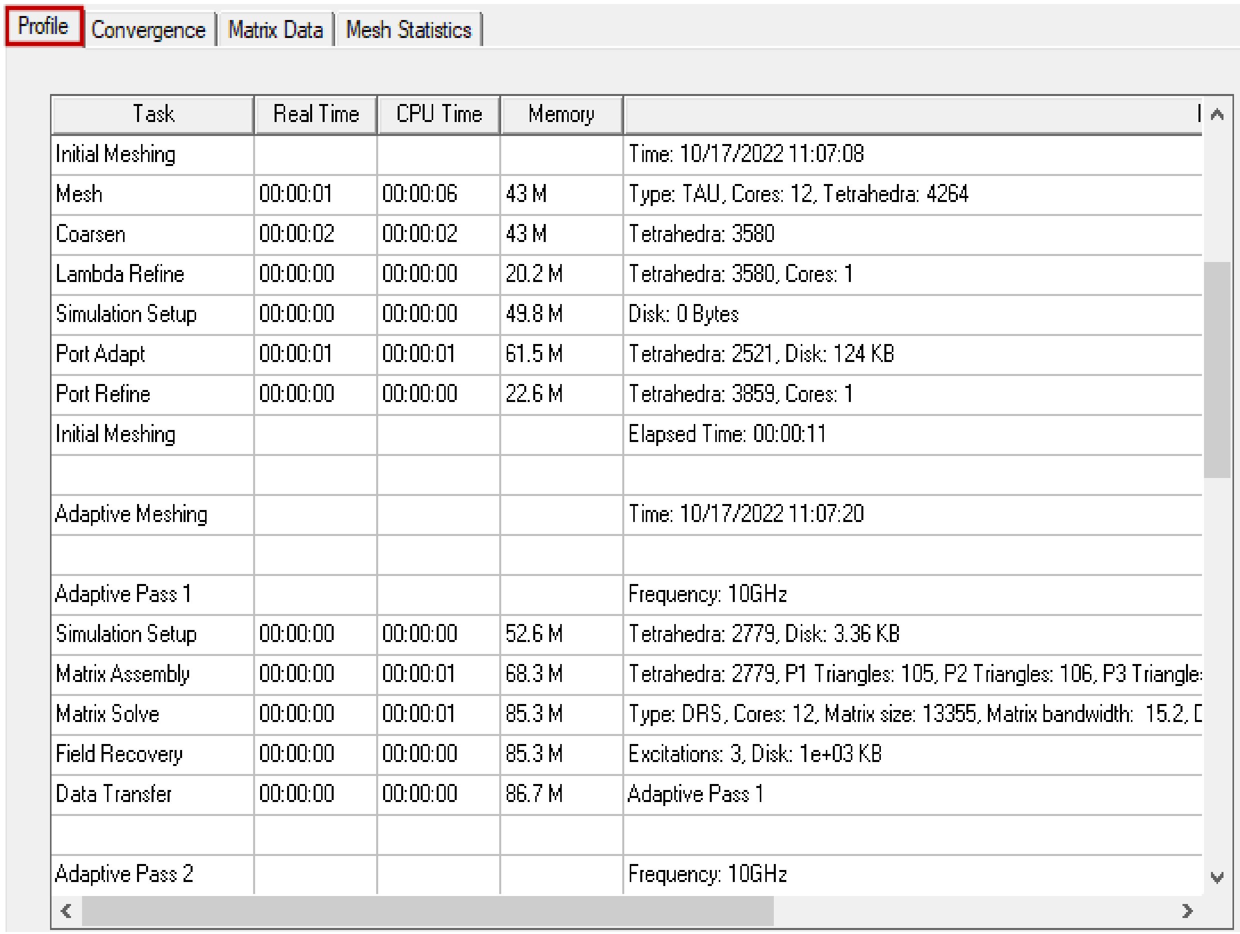
The *Solutions* window appears.

> **Note:**
>
> Subsequent sections describe the tabs that constitute the *Solution* window.
>
> - Profile
> - Convergence
> - Matrix Data
> - Mesh Statistics

Review the Profile Tab

1. In the *Solutions* window, select the **Profile** tab.

   After reviewing the information in this tab, keep the *Solutions* window open for reviewing the remaining tabs.



Task	Real Time	CPU Time	Memory	
Initial Meshing				Time: 10/17/2022 11:07:08
Mesh	00:00:01	00:00:06	43 M	Type: TAU, Cores: 12, Tetrahedra: 4264
Coarsen	00:00:02	00:00:02	43 M	Tetrahedra: 3580
Lambda Refine	00:00:00	00:00:00	20.2 M	Tetrahedra: 3580, Cores: 1
Simulation Setup	00:00:00	00:00:00	49.8 M	Disk: 0 Bytes
Port Adapt	00:00:01	00:00:01	61.5 M	Tetrahedra: 2521, Disk: 124 KB
Port Refine	00:00:00	00:00:00	22.6 M	Tetrahedra: 3859, Cores: 1
Initial Meshing				Elapsed Time: 00:00:11
Adaptive Meshing				Time: 10/17/2022 11:07:20
Adaptive Pass 1				Frequency: 10GHz
Simulation Setup	00:00:00	00:00:00	52.6 M	Tetrahedra: 2779, Disk: 3.36 KB
Matrix Assembly	00:00:00	00:00:01	68.3 M	Tetrahedra: 2779, P1 Triangles: 105, P2 Triangles: 106, P3 Triangle:
Matrix Solve	00:00:00	00:00:01	85.3 M	Type: DRS, Cores: 12, Matrix size: 13355, Matrix bandwidth: 15.2, D
Field Recovery	00:00:00	00:00:00	85.3 M	Excitations: 3, Disk: 1e+03 KB
Data Transfer	00:00:00	00:00:00	86.7 M	Adaptive Pass 1
Adaptive Pass 2				Frequency: 10GHz

Figure 4-4: *Solutions* Window – *Profile* Tab

The *Profile* tab of the *Solutions* window shows you a synopsis of the simulation process. The information on this tab includes the following:

- Mesh creation and refinement
- Adaptive meshing and adaptive passes
- Simulation setup
- Port adaptation
- Matrix assembly
- Solver data
- Field recovery (extraction of the electromagnetic field)
- Data transfer
- Frequency sweep (generation of S, Y, and Z parameter data)
- Solution process and total solution time (summary)

For each reported phase of the analysis, the Real Time, CPU Time, and additional applicable data are reported.

The more highly refined the mesh (that is, the higher the number of tetrahedral elements), the more accurate the HFSS solution of the design will be. However, a greater number of tetrahedra requires more computational resources (that is, CPU time and memory) to solve. Adaptive mesh refinement is applied locally, only where required, thus balancing accuracy and computing resources.

Review the Convergence Tab

1. Select the **Convergence** tab to view data about the solution convergence history. This tab shows the number of passes completed, number of elements solved, and *Max Mag. Delta S* (see note) for each pass.

> **Note:**
>
> After reviewing the information in this tab, keep the *Solutions* window open for reviewing the remaining tabs.

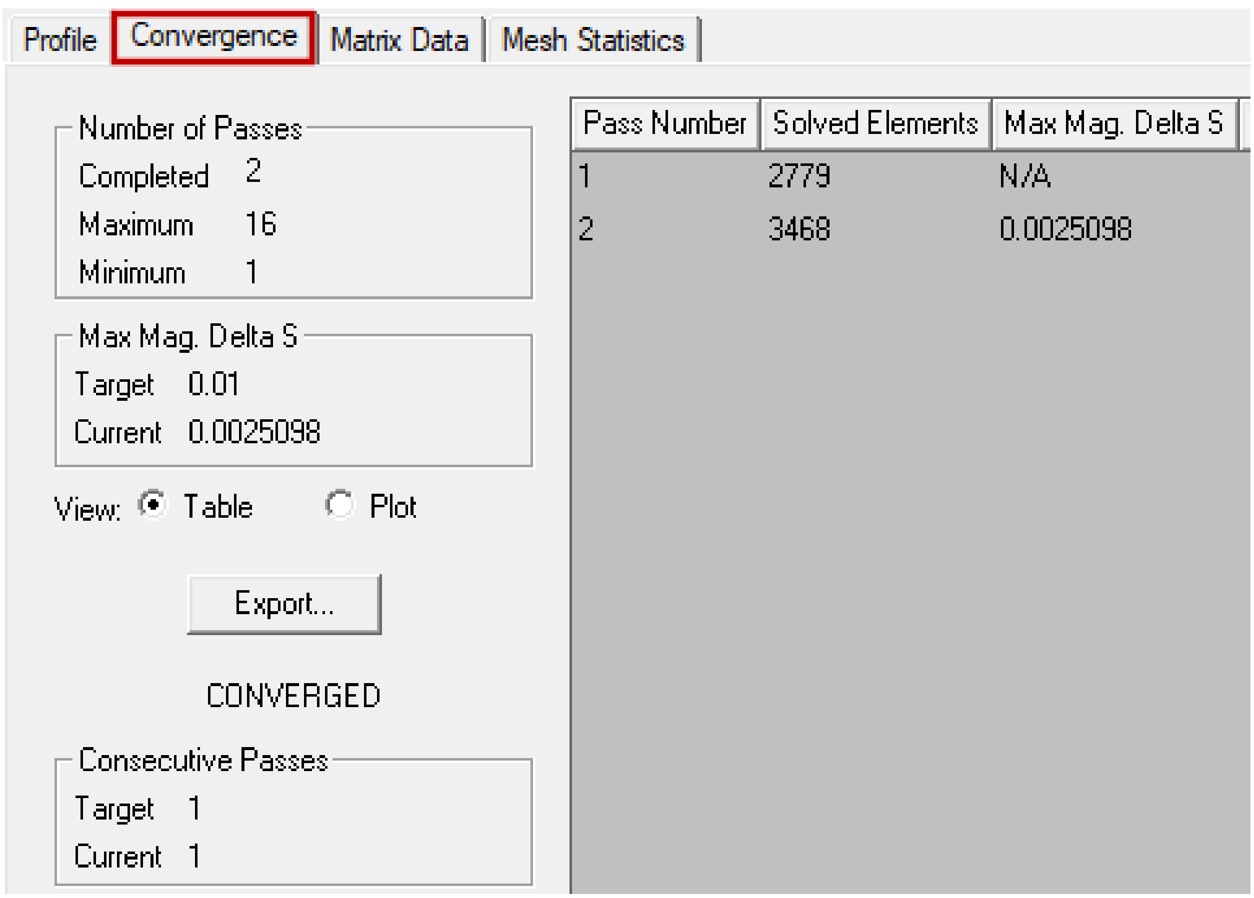


Figure 4-5: *Solutions* Window – *Convergence* Tab

> **Note:**
>
> The element count increases with each adaptive pass (due to mesh refinement). The *Max Mag. Delta S* column shows the maximum change in the S-matrix magnitude at each pass, which is used as the stop criterion. When this value is less than the prescribed tolerance in the solution setup (*Maximum Delta S*) for as many consecutive passes as dictated (*Minimum Converged Passes*), mesh adaptation ceases. The solution is deemed to be sufficiently accurate, and the solution process proceeds to its conclusion.

You can display the convergence history as a **Table** or **Plot**. However, the plot option only displays a curve if, three or more passes were completed.

You can export the data to a text file for inclusion within report documents.

Review the Matrix Data

1. Select the **Matrix Data** tab to view the S Matrix data.

> **Note:**
>
> After reviewing the information in this tab, keep the *Solutions* window open for reviewing the remaining tab.

2. Select **Display All Frequencies**.

   The *P1, P2,* and *P3* S-matrix values are tabulated for each frequency in the specified sweep.

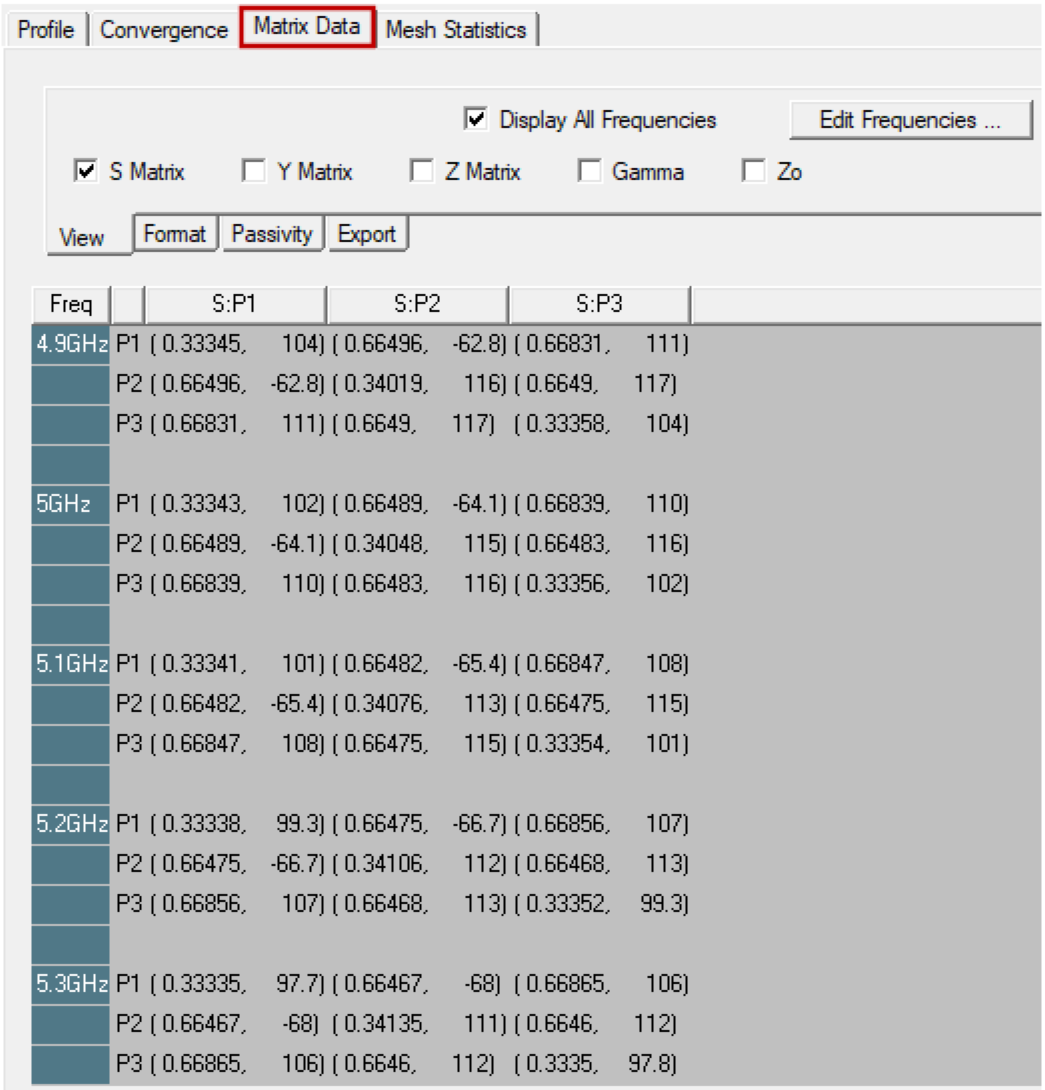
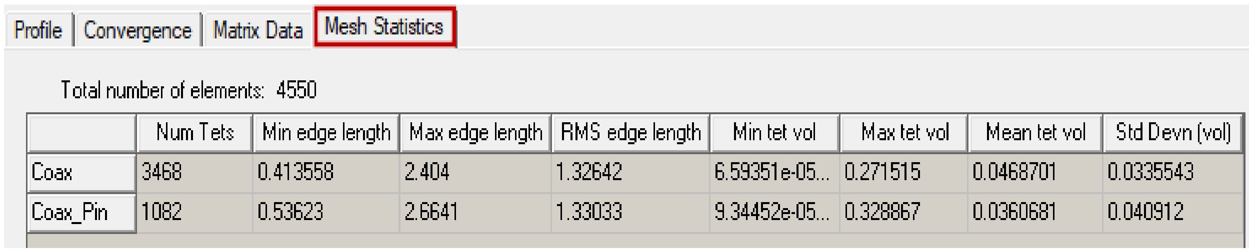


Figure 4-6: *Solutions* Window – *Matrix Data* Tab

> **Note:**
>
> For a real-time update of the Matrix Data while the solution is progressing, choose **Setup1** and **LastAdaptive** from the Simulation drop-down menus.

Review the Mesh Statistics

1. Click **Mesh Statistics** to view data about the tetrahedral 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	3468	0.413558	2.404	1.32642	6.59351e-05...	0.271515	0.0468701	0.0335543
Coax_Pin	1082	0.53623	2.6641	1.33033	9.34452e-05...	0.328867	0.0360681	0.040912

Figure 4-7: *Solutions* Window – *Mesh Statistics* Tab

The mesh statistics are listed separately for each part comprising the model, along with the total number of elements (reported above the table). The following mesh statistics are listed on this tab:

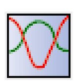
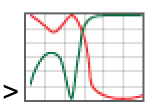
- Minimum, maximum, and RMS tetrahedra edge lengths
- Minimum, maximum, and mean tetrahedra volume
- Standard deviation of the tetrahedra volume.

> **Note:**
>
> You can click and drag the column heading borders to resize any column width. You can also resize the *Solutions* window's overall size.

2. Click **Close**.

Create S-Parameter versus Frequency Plot

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

1. On the **Results** ribbon tab, click  **Modal Solution Data Report** >  **2D**.

   The *Report* dialog box appears.

2. Edit the settings in the **Trace** tab of the *Report* dialog box, as shown in the following figure:

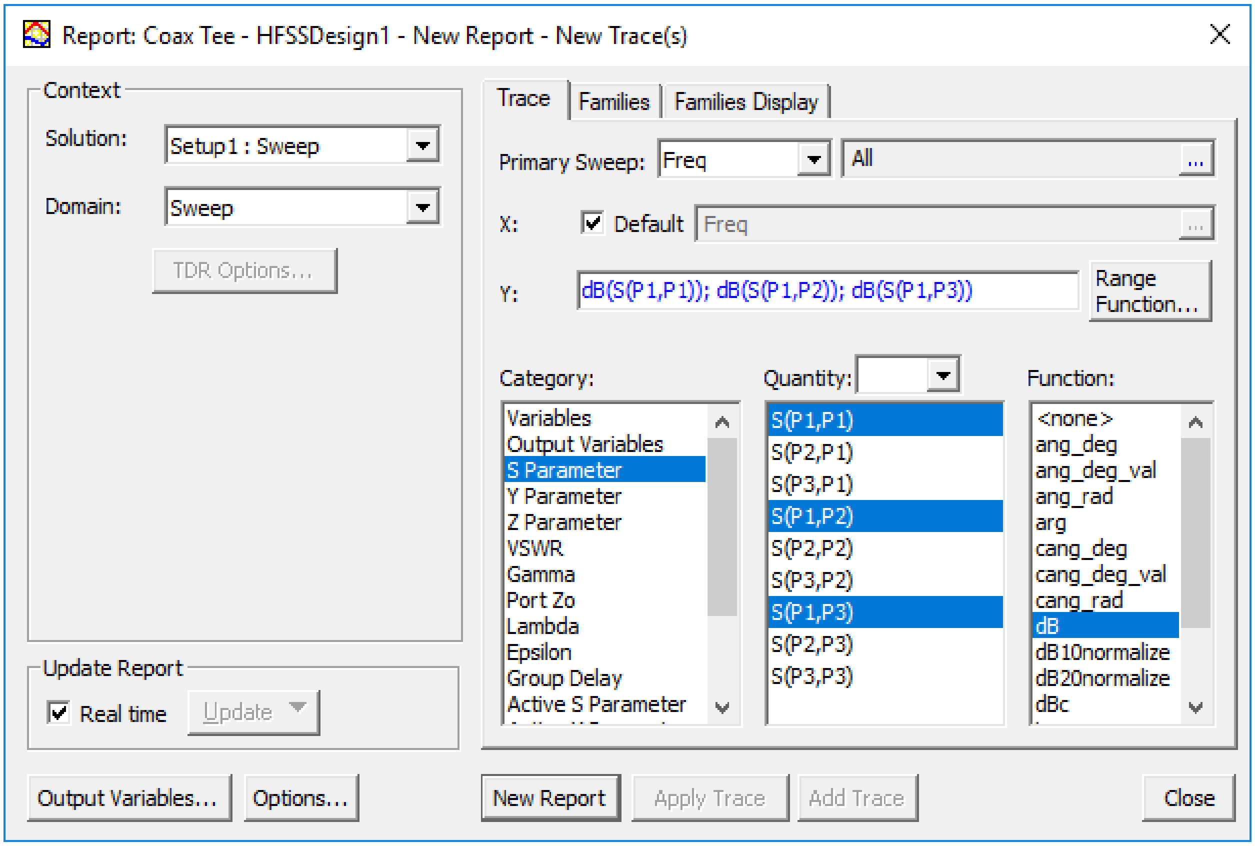


Figure 4-8: *Report* Dialog Box

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

   HFSS generates the report:

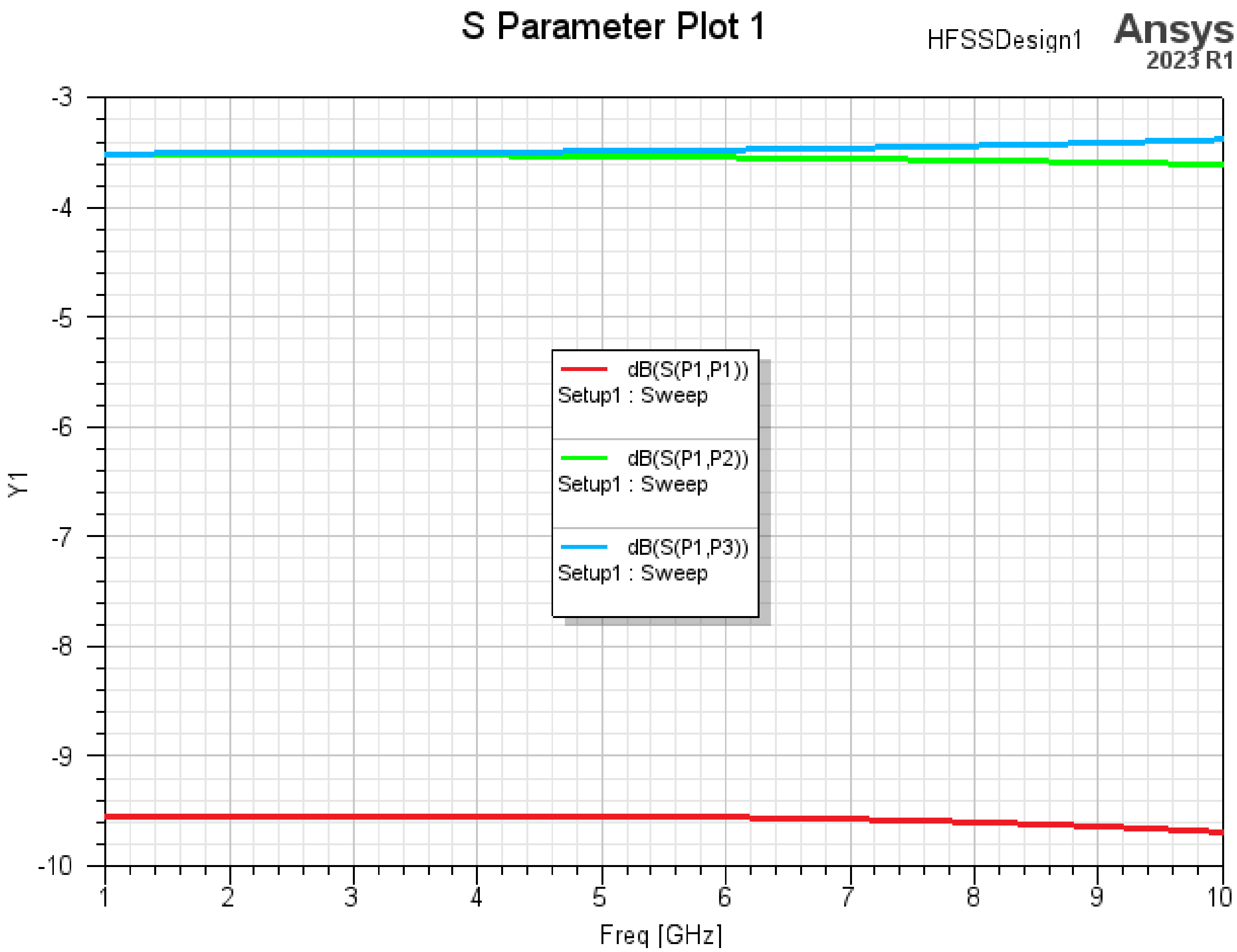


Figure 4-9: S Parameter Vs Frequency

> **Note:**
>
> The red curve represents S(P1, P1), the green curve S(P1, P2), and the blue curve S(P1, P3).

Create Field Overlay

Next, you will select a model object on which to overlay an electric field magnitude plot and add the overlay.

1. Minimize your S-parameter plot and return to the Modeler window.
2. On the **Draw** ribbon tab, choose **Orient** > **Isometric** for a more optimal view of the overlay to be added.
3. Under *vacuum* in the History Tree, select **Coax**.

4. Right-click in the Modeler window and select **Plot Fields** > **E** > **Mag_E** from the shortcut menu.

   The *Create Field Plot* dialog box appears.

5. Edit the settings as shown below and then click **Done**.

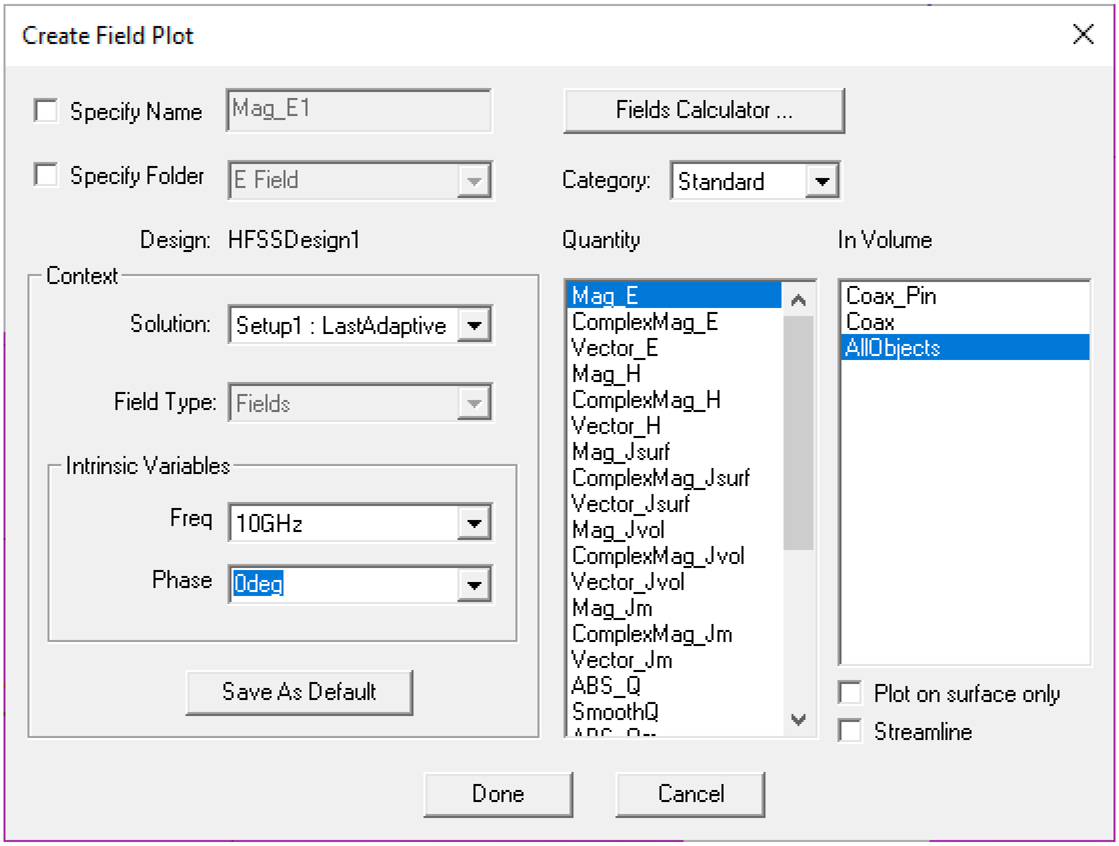


Figure 4-10: *Create Field Plot* Dialog Box

The E-field overlay is applied to the model:

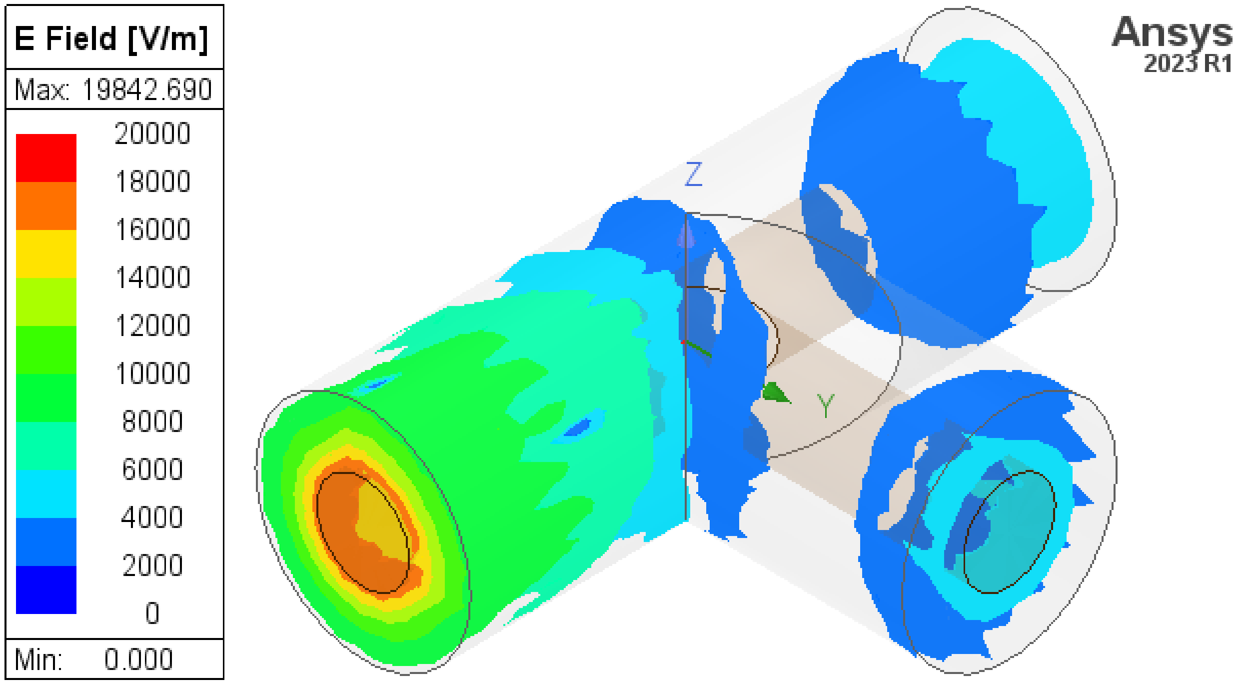


Figure 4-11: Initial E-Field Overlay Plot

In the next two procedures, you will edit the excitation source for this field overlay and modify the attributes of the plot.

Edit Field Overlay Sources

To better reflect the ideal application of a coax tee, you should edit the excitation source on which the E-Field overlay is based. By default, input power is applied to the first wave port (P1). In this case, the excitation is applied to the leg at the +X end of the tee (the first one drawn). Ideally, the power input to a tee would be at the middle leg (in this case, P2), resulting in a symmetrical output to the other two legs (P1 and P3).

Edit the field sources as follows:

1. In the **Project Manager**, right-click **Field Overlays** and select **Edit Sources**.

   The *Edit post process sources* dialog box appears.

2. Under the *Spectral Fields* tab, make the following changes to the values in the *Magnitude* column:
   a. Change the **Magnitude** for **P1:1** from *1* to **0**.
   b. Change the **Magnitude** for **P2:1** from *0* to **1**.

   The dialog box should look like the following figure:

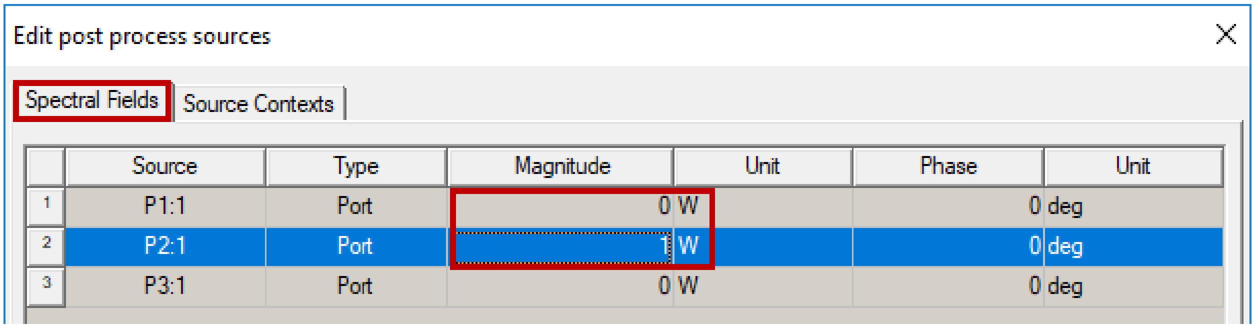


Figure 4-12: *Edit Post Process Sources* Dialog Box

3. Click **OK**.

Modify Attributes of the Field Plot

Change the Scale

Specify a logarithmic scale with user-defined limits, as follows:

1. Double-click within the *E Field* plot legend perimeter.

   The *E Field* dialog box appears.

2. Select the **Scale** tab and edit the settings as shown in the following figure:

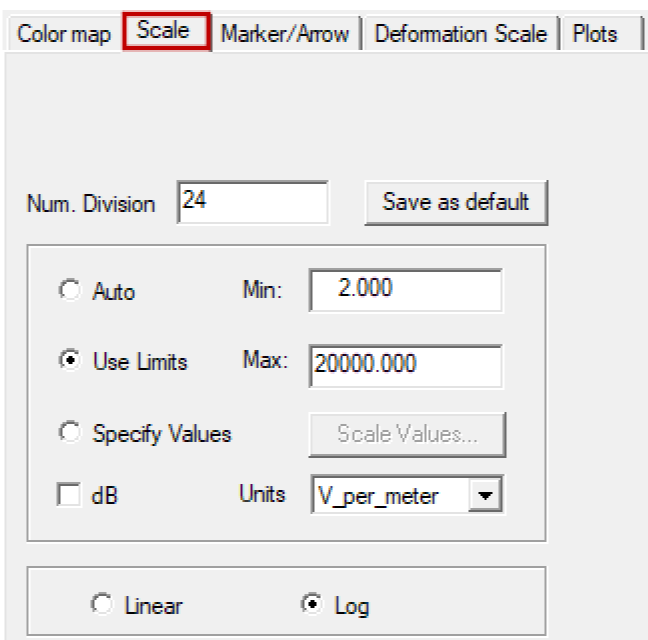


Figure 4-13: *E Field* Dialog Box – *Scale* Tab

Adjust the Plot Quality

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

- Coarse
- Normal
- Fine
- Very Fine

3. Select the **Plots** tab and choose **Very Fine** from the **Plot quality** drop-down menu, as shown in the following figure:

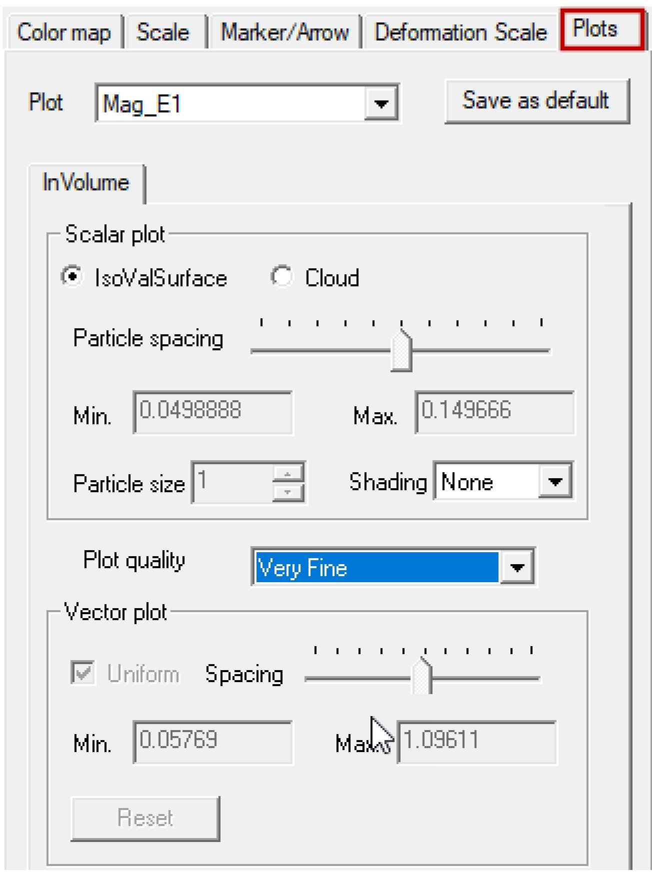


Figure 4-14: *E Field* Dialog Box – *Plots* Tab

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

   The revised plot attributes are applied, and your model should resemble the figure below:

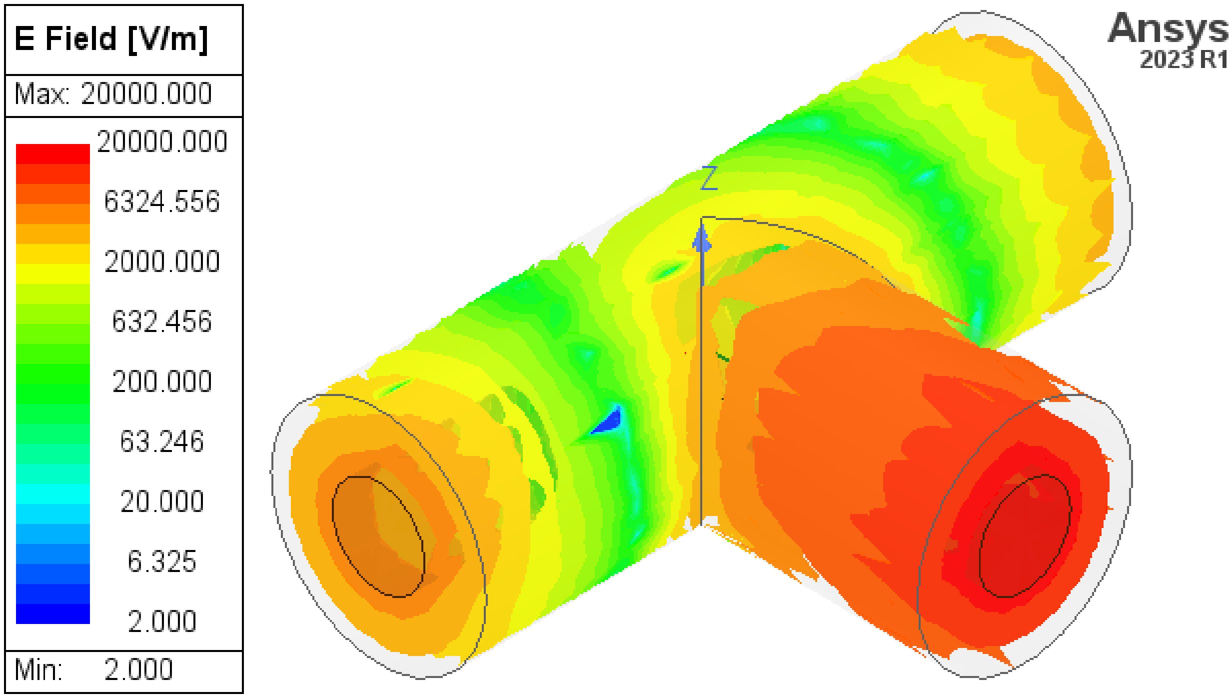


Figure 4-15: The E-Field Overlay on the Coax Tee

Animate the E-Field Overlay

Field animations are an excellent means of visualizing the propagation of electromagnetic waves through the device your model represents.

1. Under *Field Overlays > E Field* in the *Project Manager*, right-click **Mag_E1** and select **Animate**.
2. In the *Create Animation Setup* dialog box that appears, accept the default settings and click **OK**.

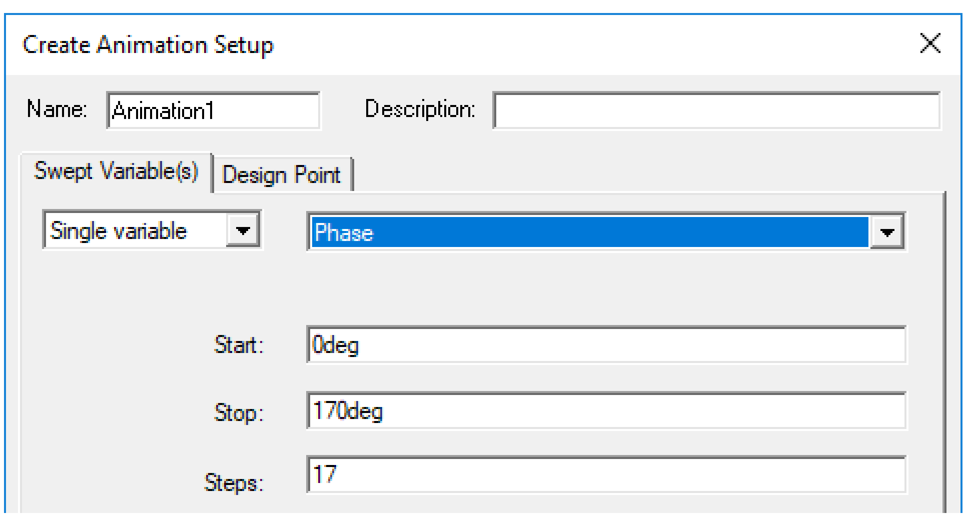


Figure 4-16: *Create Animation Setup* Dialog Box

There may be a brief delay while the animation frames are generated. Then, the *Animation* dialog box appears, and the E field overlay on the model is animated.

3. Use the controls in the *Animation* dialog box to change the animation speed, stop and start the animation, or to reverse the playback direction.

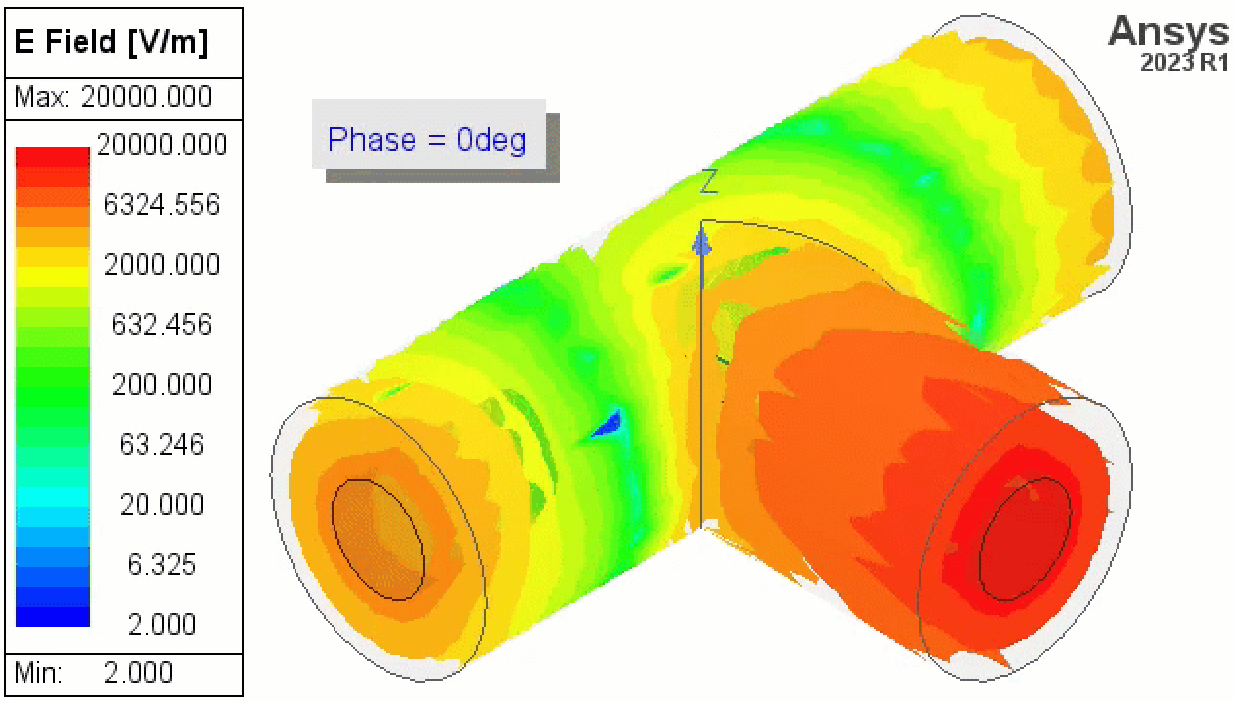


Figure 4-17: E Field Animation

4. Click **Close** when you're done viewing the animation.
5. Under *Field Overlays* > *E Field* in the Project Manager, right-click **Mag_E1** and deselect **Plot Visibility** in the shortcut menu to hide the electric field overlay.
6. **Save** your model.

5 - Optionally, Restore Current View Orientations

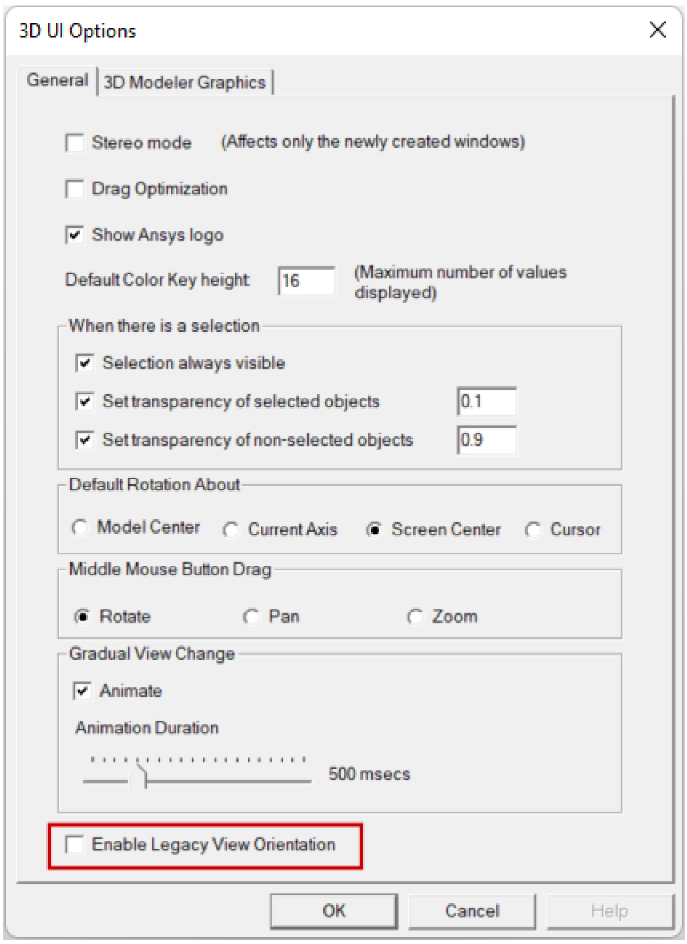
You have completed this getting started guide.

If you prefer to use the new view orientations implemented in version 2024 R1 of the Ansys Electronics Desktop application, clear the *Use Legacy View Orientation* option as follows:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is cleared:



3. Click **OK**.

The settings in the 3D UI Options dialog box are global. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

You can now save and close this project.

Index