Ansys 2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with HFSS™: Probe Feed Patch Antenna

Ansys

Copyright and Trademark Information

© 1986-2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with export-ing laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

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

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys pro-prietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

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, simulate, and analyze a probe feed patch antenna using the Terminal solution type.

This chapter contains the following topic:

- Sample Project - Probe Feed Patch Antenna

Sample Project: Patch Antenna

This example shows you how to create a probe feed patch antenna as an HFSS design type, setup and run a terminal solution analysis, and review the results using the Ansys Electronics Desktop application. The model, shown below, consists of a substrate, infinite ground plane, patch, and a coax and probe feed assembly:
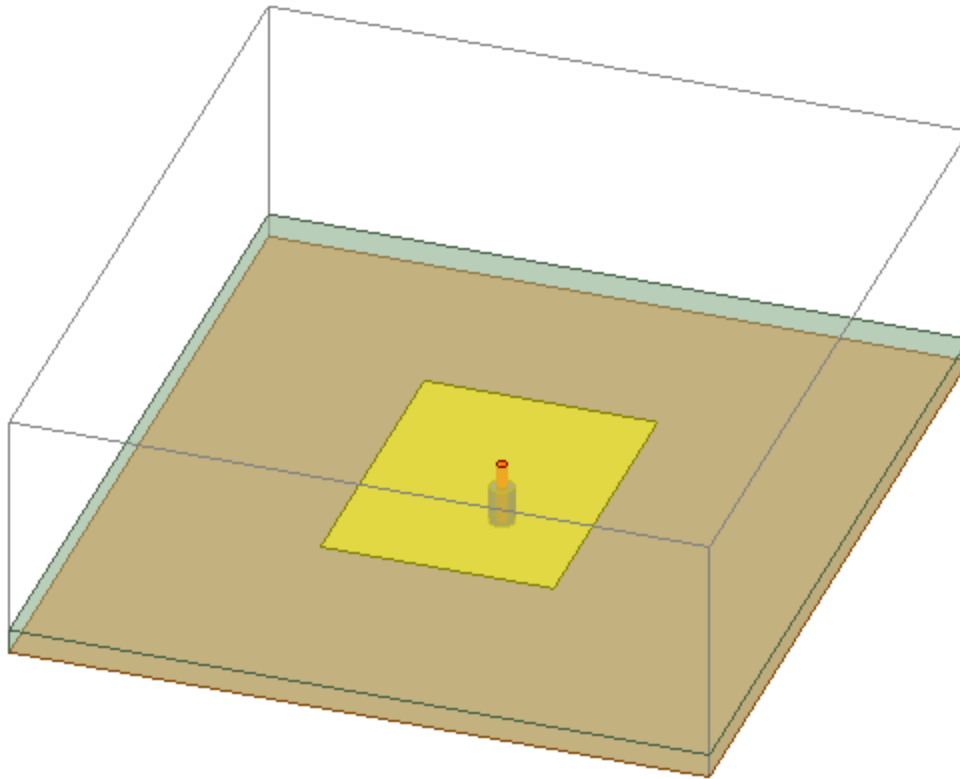

Figure 1-1: Probe Feed Patch Antenna

2 - Set Up the Project

This chapter contains the following topics:

- Launch Ansys Electronics Desktop
- Set General Options
- Insert HFSS design
- Set Model Units (cm)
- Set Solution Type (Terminal)

Launch Ansys Electronics Desktop (EDT)

For convenience, a shortcut to the EDT 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.
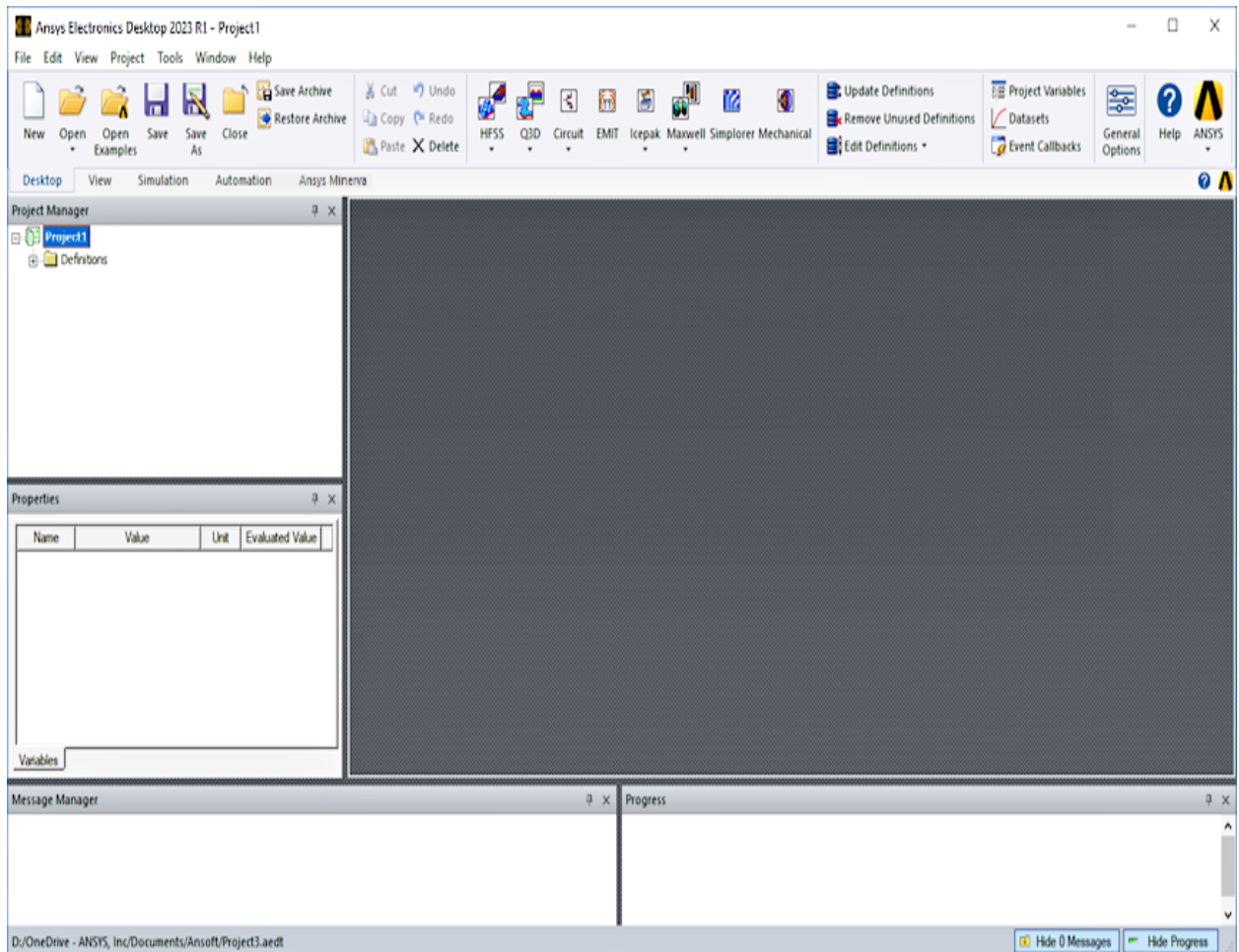
Figure 2-1: *Ansys Electronics Desktop* **Application Launched**

> **Note:**
>
> If a project is not listed at the top of the Project Manager, click [ ] **New** on the **Desktop** ribbon tab to include one. If the *Project Manager* window does not appear after launching the application, go to the **View** menu and select the **Project Manager** option.

Set General Options

Verify the general options that are in effect to make sure they're consistent with the instructions in this guide.

1. On the **Desktop** ribbon tab, click **General Options**.

   The *Options* dialog box appears.

2. Under **HFSS**, select **Boundary Assignment** and ensure that all assignment options are selected:
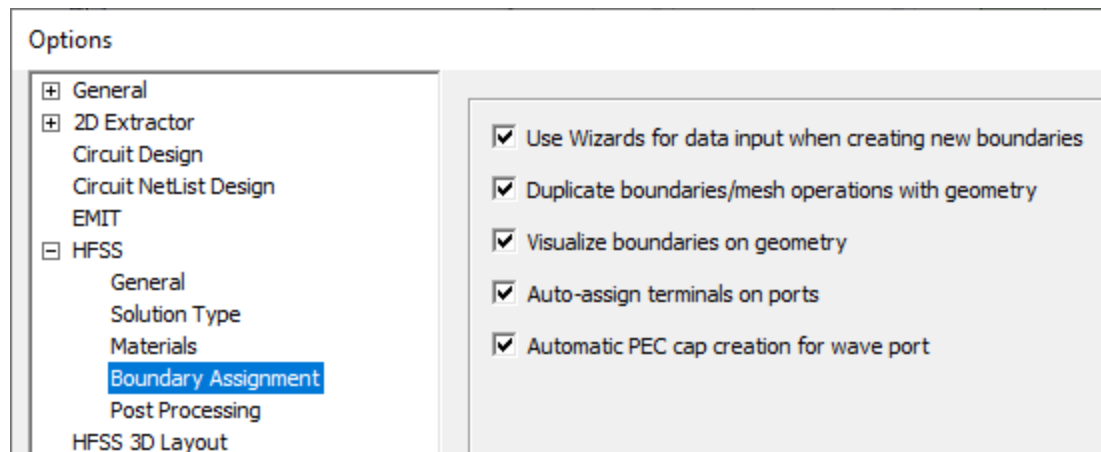

Figure 2-2: HFSS Boundary Assignment Options

3. Under **3D Modeler**, select **Drawing** and ensure that the following two options are both selected:
   - **Automatically cover closed polylines**
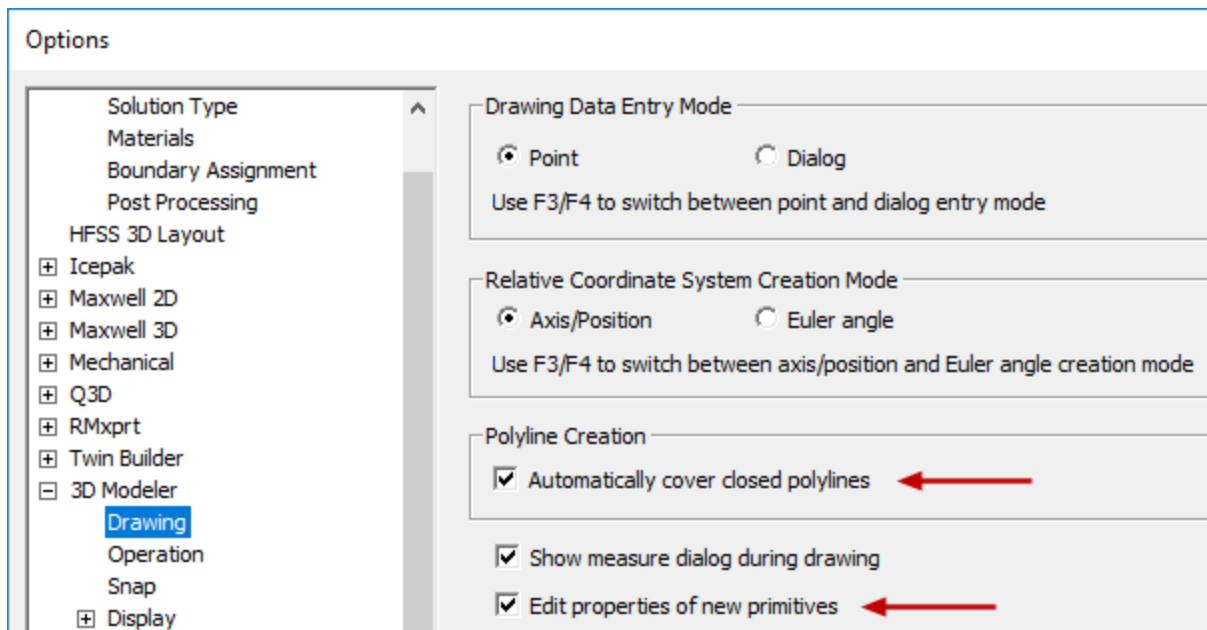   - **Edit properties of new primitives**

Figure 2-3: 3D Modeler Drawing Options

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

Insert HFSS Design

1. On the **Desktop** ribbon tab, click [icon] **Insert HFSS design**. (You do not have to access the drop-down menu; the default menu action is to insert a conventional HFSS design.)
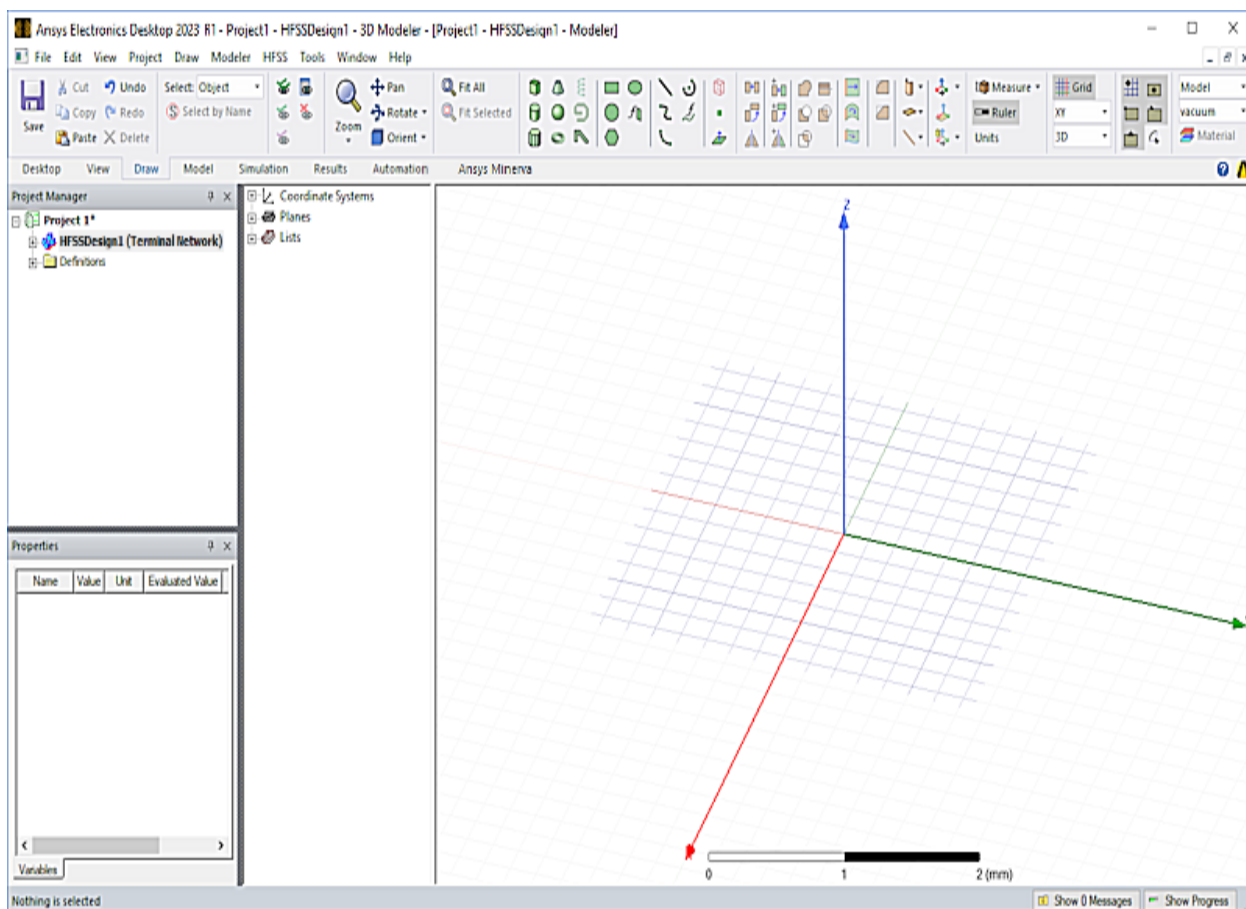
Figure 2-4: HFSS Design Added to Project

2. At the top of the Project Manager, select **Project*x*** , press **F2**, change the name to **PF_ PatchAntenna**, and press **Enter**.

3. In the Project Manager, click the plus sign (**+**) to expand the **HFSSDesign1** branch.

4. Optionally, you can customize the display of the coordinate system axes. Using the menu bar, click **View** > **Coordinate System**.

    The images in this guide are based on the default, *Large*, option.

5. Optionally, you can customize the style, spacing/density, and visibility of the grid . Using the menu bar, click **Grid Settings**.

    The images in this guide are based on default grid settings.

6. To hide the ruler that appears at the bottom of the Modeler window, click ▭ **Ruler** on the **Draw** ribbon tab.

The *Ruler* is hidden for most of the images in this guide. The command on the ribbon tab acts as a toggle. When the ruler is hidden, click it again to once again display the ruler.

7. **Save** your project. This command is available from any ribbon tab.

> **Note:**
>
> Even though Ansys Electronics Desktop automatically saves your projects at defined intervals, it's a good idea to save your changes frequently.
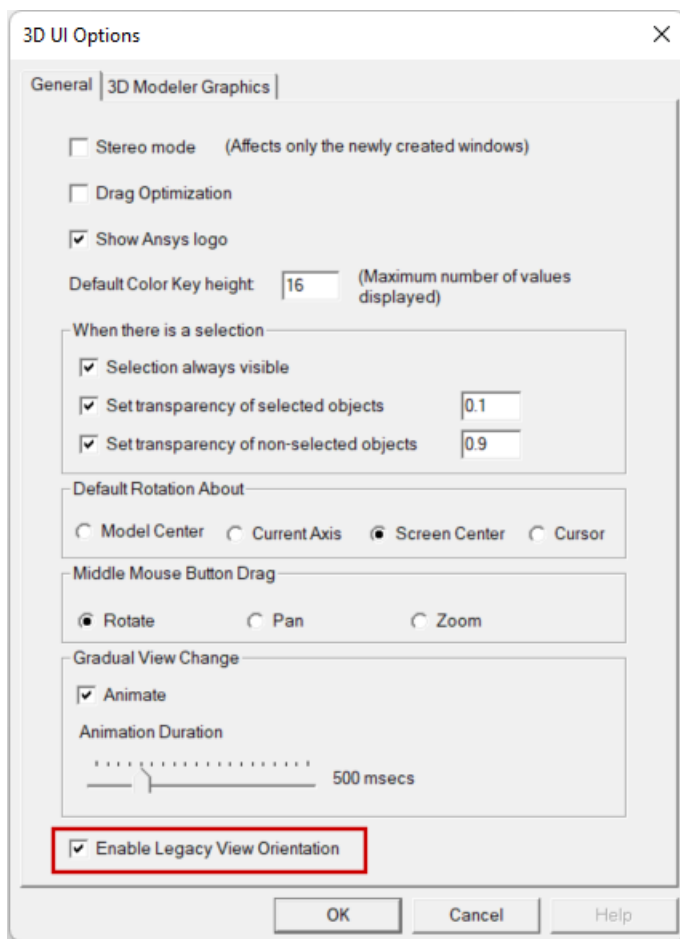
Enable Legacy View Orientations

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 (cm)

Define the model units as follows:

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

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

2. Select **cm** (centimeters) from the **Select units** drop-down menu.

    Keep the **Rescale to new units** and **Advanced** options cleared.
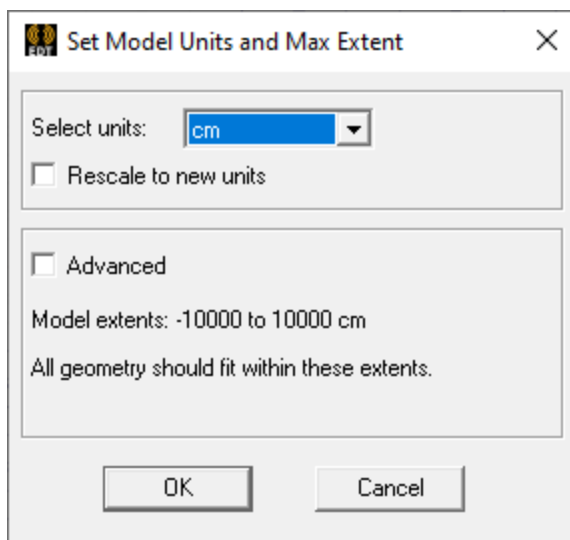
Figure 2-5: *Set Model Units and Max Extent* **Dialog Box**

3. Click **OK**.

Verify Solution Type (Terminal)

Specify the design's solution type as follows:

1. In the Project Manager, right-click **HFSSDesign1** and select **Solution Type** from the shortcut menu.

    The *Solution Type* dialog box appears.

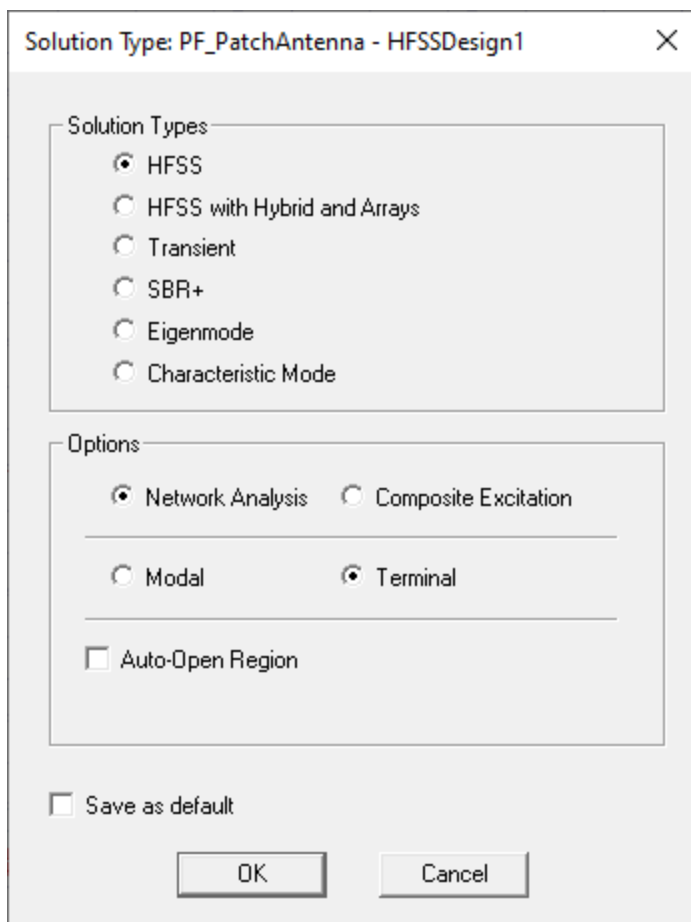2. Verify that the settings are as shown in the following image:

Figure 2-6: *Solution Type* **Dialog Box**

The HFSS solution type and the Network Analysis and Terminal options are the default settings for a clean program installation, but users can save new default settings if wanted.

The design type is labeled as *HFSSDesign1 (Terminal Network)**. The asterisk indicates that there are unsaved changes.

> **Note:**
>
> The *Terminal* option calculates the terminal-based S-parameters of multi-conductor transmission line ports. The S-matrix solutions are expressed in terms of terminal voltages and currents.

3. Click **OK**.

4. 💾 **Save** the project.

3 - Create the Model

This chapter contains the following topics:

- Create Substrate
- Create Infinite Ground
- Assign Perfect E Boundary to the Ground
- Create Infinite Ground Cutout
- Complete the Infinite Ground
- Create Patch
- Assign Perfect E Boundary to the Patch
- Create the Coax
- Create the Coax Pin
- Create the Probe
- Assign Excitation
- Create Air Body
- Assign Radiation Boundary
- Create Radiation Setup

Create Substrate

To create the substrate, draw a box freehand (that is, of any size and location) and then modify its properties, as follows.

1. On the **Draw** ribbon tab, click 📦 **Draw box**.
2. Press **F3** to ensure that you are in the *Point* data entry mode and not the dialog entry mode.

   A black diamond appears at the tip of the cursor, indicating grid snapping points.

3. Click inside the **Modeler** window to define an arbitrary point for the fist corner of the box's base.

   Small X-Y mini-axes appear indicating the direction to move the cursor to draw the base rectangle.

4. Click the mouse a second time to define the opposite corner of the base rectangle at another arbitrary location.

   A small Z mini-axis appears to indicate the mouse movement direction to define the rectangle's height.

5. Click one more time to define the height of the box, completing the freehand drawing operation.

The *Properties* dialog box appears.

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

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate Sys...	Global		
Position	-5 ,-4.5 ,0	cm	-5cm , -4.5cm , 0cm
XSize	10	cm	10cm
YSize	9	cm	9cm
ZSize	0.32	cm	0.32cm

Figure 3-1: *Substrate* Properties – Command Tab

7. On the **Attribute** tab of the *Properties* dialog box, change the **Name** to **Sub1**.
8. Select **Edit** from the **Materials** drop down menu. Then:
    a. In the *Select Definition* dialog box that appears, Type **Rogers** in the **Search By Name field** to jump to the desired portion of the library materials list.
    b. Scroll down to locate and select **Rogers RT/duroid 5880 (tm)**.
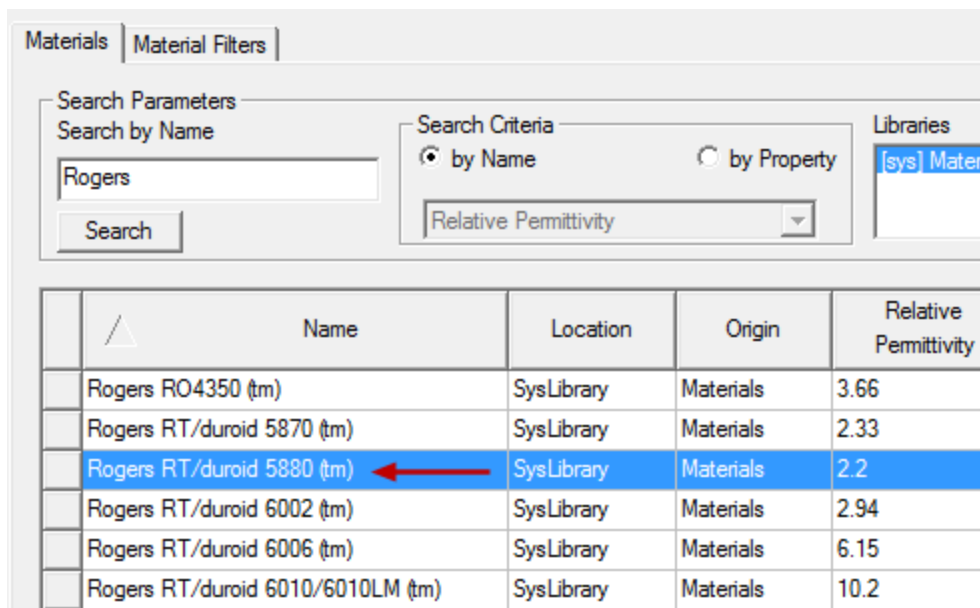

Figure 3-2: Selecting the Substrate Material

    c. Click **OK** to close the *Select Definition* dialog box.
9. Ensure that the **Material Appearance** option is **not** selected.
10. Change the **Transparent** value to **0.6**.
11. Click **OK** to close the *Properties* dialog box.

12. On the Draw ribbon tab, click **Fit All** (or, press **Ctrl+D** to fit the model view to the Modeler window.

13. Click in the Modeler window's background area to clear the current selection.

> **Note:**
>
> As you continue to build the model, whenever you want to fit the view, press **Ctrl+D**.
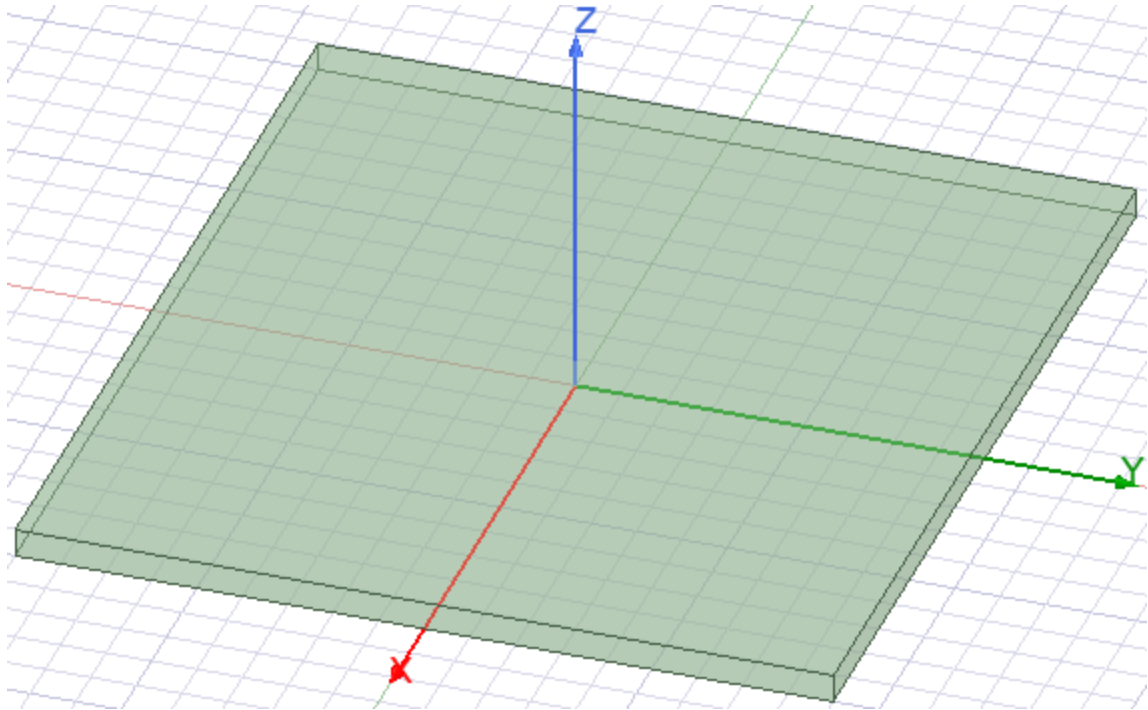

Figure 3-3: Substrate Created

Create Ground Plane

To create the ground plane, create a sheet object from the bottom face of the substrate, as follows:

1. Press **F** to begin the *face selection* mode.

2. Click the top face of the substrate (*not* close to an edge) and then press **B** (*next Behind*) to select the **bottom** face.
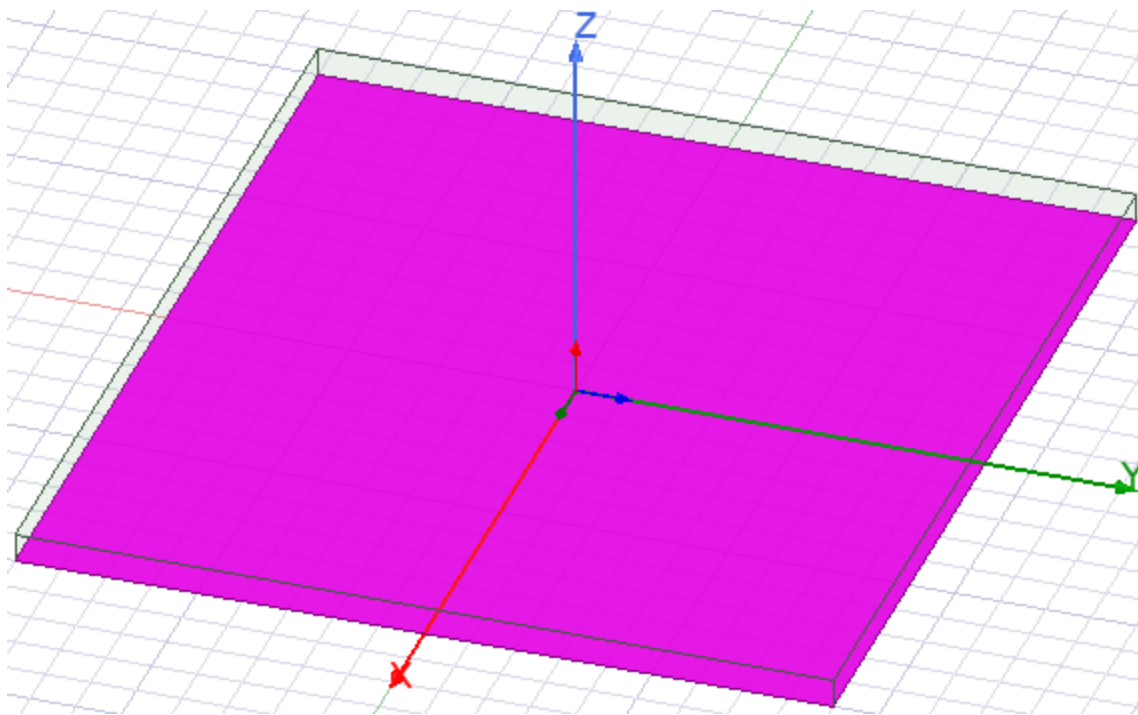
Figure 3-4: Face Selected for Ground Plane

3. On the **Draw** ribbon tab, click 📙 **Surface** > **Create Object From Face**.

   The object, *Sub1_ObjectFromFace*, appears in the History Tree under *Model > Sheets > Unassigned*. Also, the object's details appear in the docked *Properties* window.

4. In the **Attribute** tab of the docked *Properties* window, change the Name to **Inf_GND** and press **Enter**.

   Though the ground plane is a finite rectangle, in the next procedure, you will apply a boundary that declares it an infinite ground plane.

5. Change the **Color** to **orange** (column 2, row 4 of the color samples; Red: 255, Green: 128, Blue: 0).

6. Change the **Transparent** value to **0.7**.

7. Clear the selection.
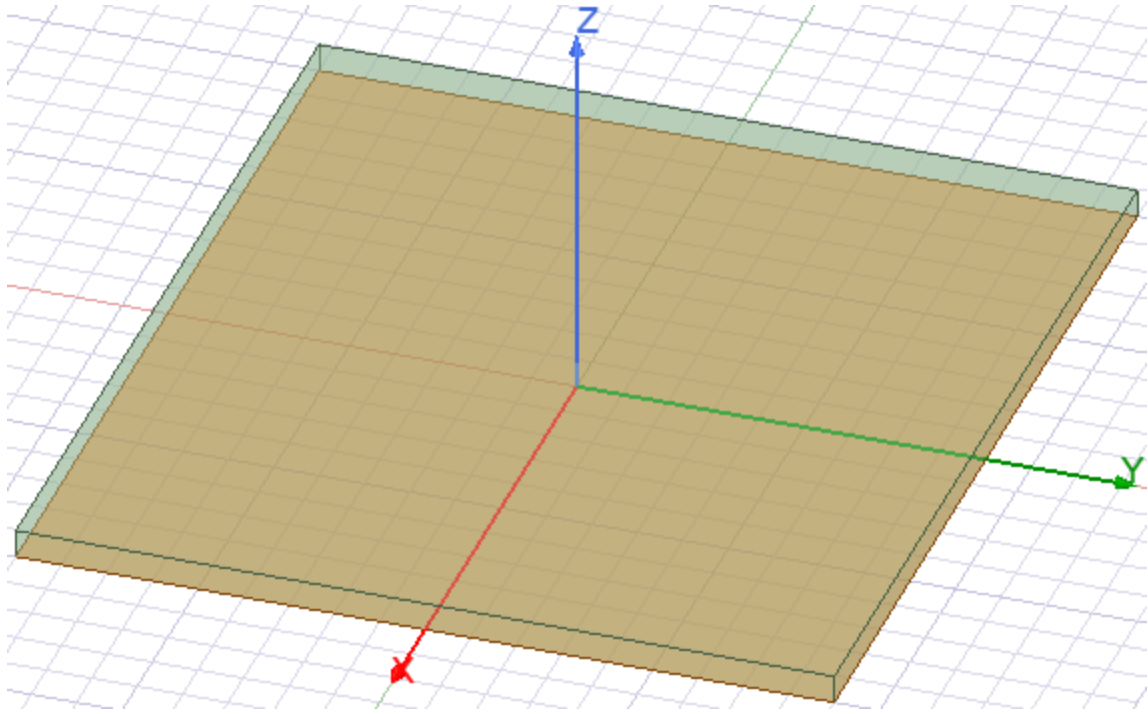
   Your model should look like the following figure:

Figure 3-5: Ground Plane Created

Assign Perfect E Boundary to Ground

1. Press **O** to return to the *object selection* mode.
2. On the **Draw** ribbon tab, click ⓢ **Select by Name**.
3. In the *Select Object* dialog box that appears, select the object, **Inf_GND**, and click **OK**.
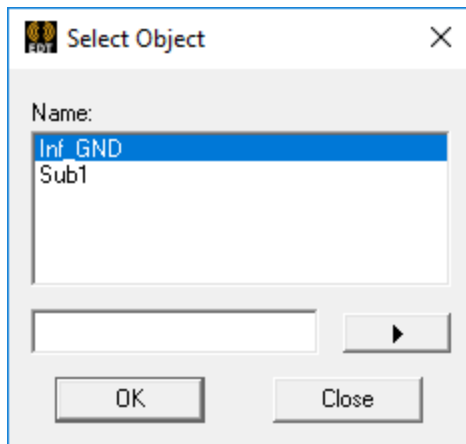

Figure 3-6: *Select Object* Dialog Box

4. Right-click in the Modeler window and choose **Assign Boundary** > **Perfect E**.

    The *Perfect E Boundary* dialog box appears.

5. Change the **Name** to **PerfE_Inf_GND** and select the **Infinite Ground Plane** options, as shown in the following figure:
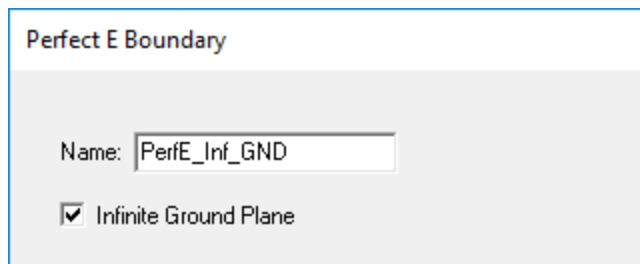
Perfect E Boundary

Name: PerfE_Inf_GND

☑ Infinite Ground Plane

Figure 3-7: *Perfect E Boundary* **Dialog Box**

6. Click **OK**.

Create Infinite Ground Cutout

To create the cut out, perform the following steps:

1. On the **Draw** ribbon tab, click ⬤ **Draw circle**.

2. Click in the Modeler window to define the circle's center point at an arbitrary location, move the mouse in the X and Y directions, and click again to define an arbitrary radius.

    The *Properties* dialog box appears.

3. In the **Command** tab of the *Properties* dialog box, edit the values as shown in the following figure:

Name	Value	Unit	Evaluated Value
Command	CreateCircle		
Coordinate Sys...	Global		
Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
Axis	Z		
Radius	0.16	cm	0.16cm
Number of Seg...	0		0

Figure 3-8: Cutout Properties – *Command* **Tab**

Leave the *Properties* dialog box open.

> **Note:**
>
> If you ever need to change the command properties after dismissing the *Properties* dialog box, select the associated command in the History Tree (in this case, *Sheets > Unassigned > Circle1 >* **Create Circle**) to edit the *Command* tab contents within the docked *Properties* window. Alternatively, you can double-click the command's History Tree entry to display the *Properties* dialog box.

4. In the **Attributes** tab of the *Properties* dialog box, change the **Name** to **Cutout** and then click **OK**.

> **Note:**
>
> If you ever need to change the object attributes after dismissing the *Properties* dialog box, select the associated object in the History Tree (in this case, *Sheets > Unassigned >* **Cutout** or **Circle1** (depending on whether or not you have renamed it yet). Then, edit the *Attribute* tab contents within the docked *Properties* window. Alternatively, you can double-click the object's History Tree entry to display the *Properties* dialog box.

You should see a small circle on the ground plane face. Its center is near the origin, 0.5 cm away in the -X direction.

Complete the Infinite Ground

Next, you will select the infinite ground (*Inf_GND*) and *Cutout* objects, and you will subtract the cutout from the infinite ground:

To select the objects **Inf_GND** and **Cutout**:

1. Select the following objects from the History Tree, in the order specified:
   - **Inf_GND** (under *Sheets > Perfect E*)
   - **Cutout** (under *Sheets > Unassigned*)
2. On the **Draw** ribbon tab, click 🔳 **Subtract**.

   The *Subtract* dialog box appears.

3. Ensure that **Inf_GND** is listed in the *Blank Parts* box, **Cutout** is listed in the *Tool Parts* list, and **Clone tool objects before operation** is **not** selected, as shown below:
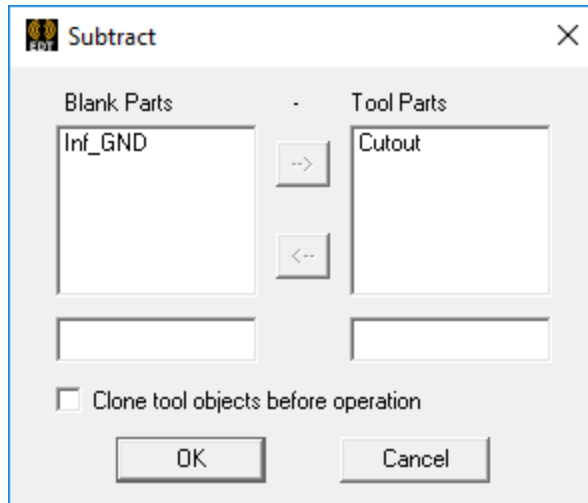
Figure 3-9: *Subtract* **Dialog Box**

> **Note:**
>
> The object selection order determines which part is the blank and which is the tool.

4. Click **OK** to subtract the *Cutout* from the *Inf_GND* object.
5. Clear the current selection.

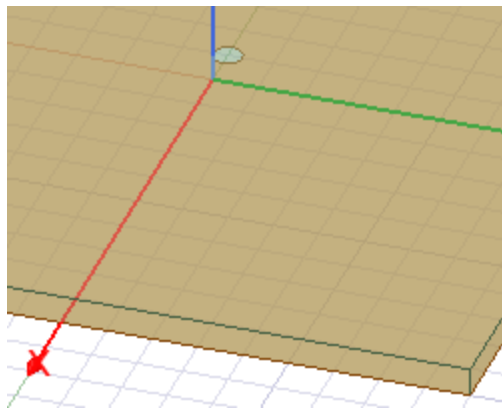   Your model should look like the following figure:

Figure 3-10: *Cutout* **Subtracted from** *Inf_GND*

Create the Patch

This time, for the patch object, you will use a different method of drawing the required object.

1. On the **Draw** ribbon tab, click ▭ **Draw rectangle**.
2. Use the coordinate entry text boxes on the status bar to enter the absolute coordinates of the first corner, as follows:
   - **X: -2**
   - **Y: -1.5**
   - **Z: 0.32**, then press **Enter**.

> **Note:**
>
> Use **Tab** to jump into the first coordinate entry text box and also use **Tab** to navigate between them. Be careful not to move the mouse while entering coordinates numerically, or the cursor location will override your typed entries.

3. Use the coordinate text boxes to enter the following relative coordinates for the opposite corner of the rectangle:
   - **dX: 4**
   - **dY: 3**
   - **dZ: 0**, then press **Enter**.

   The *Properties* dialog box appears.

4. On the **Attribute** tab, change the **Name** to **Patch**.
5. Set the **Color** to **yellow** (column 2, row 2 of the color samples; Red: 255, Green: 255, Blue: 0).
6. Set the **Transparent** value to **0.6**.
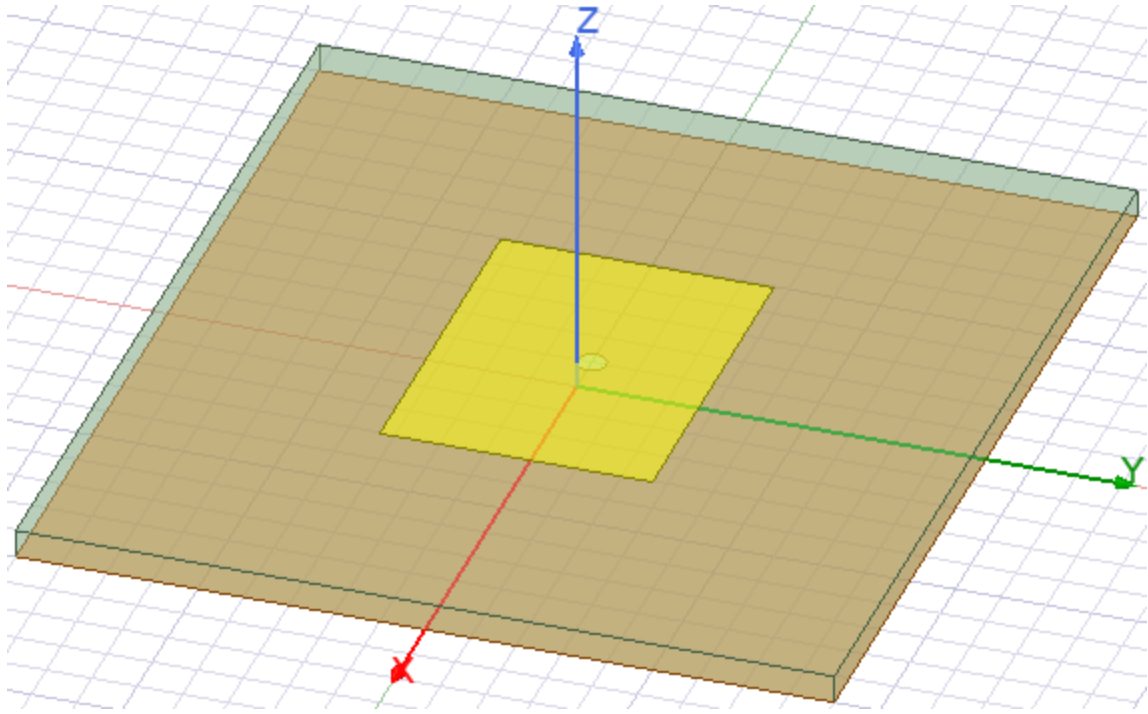7. Click **OK** and clear the current selection.

Figure 3-11: The Patch Created

Assign a Perfect E Boundary to the Patch

1. Under *Sheets > Unassigned* in the History Tree, click **Patch** to select it.

2. Using the menu bar, click **HFSS** > **Boundaries** > **Assign** > **Perfect E**.

   The *Perfect E Boundary* dialog box appears.

3. Type **PerfE_Patch** in the **Name** text box and click **OK**.

4. Select **PerfE_Patch** under *Boundaries* in the Project Manager to visualize the boundary on the model:
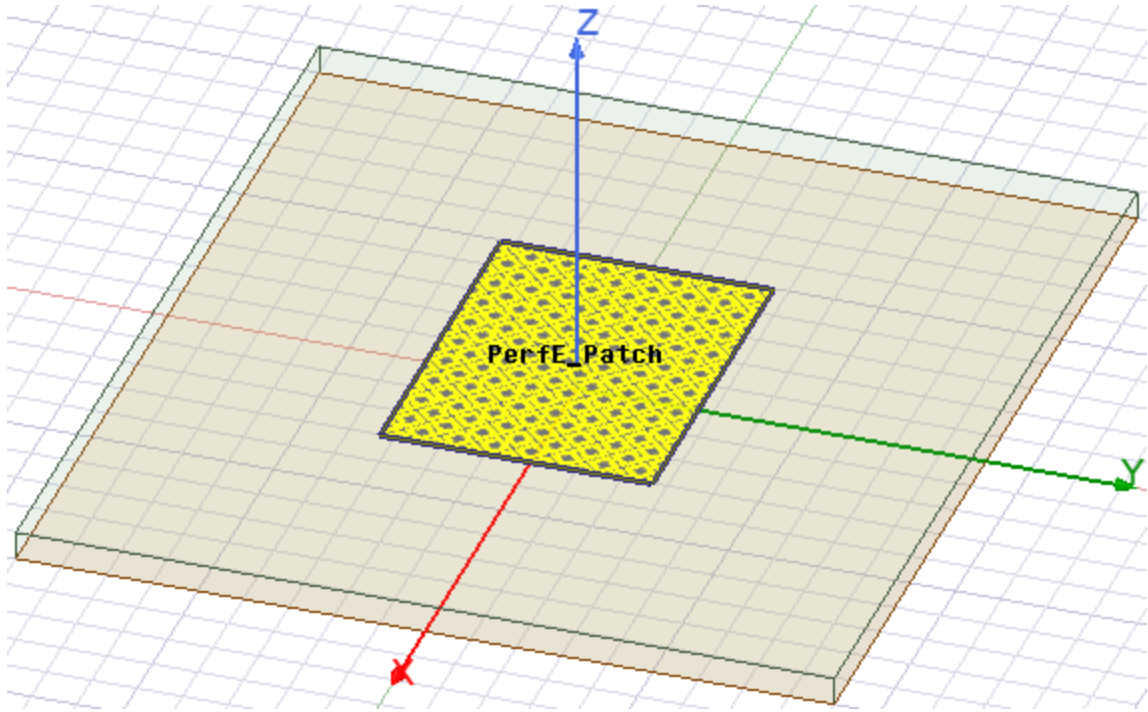
Figure 3-12: PerfE_Patch Applied

5. Clear the selection.

Create the Coax

In this section you will create the Coax (specifically, the dielectric portion around the conductor). You could follow the previously used method of setting the material for the Coax after drawing it. But, this time, set the material before drawing the coax.

1. At the far right end of the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu.
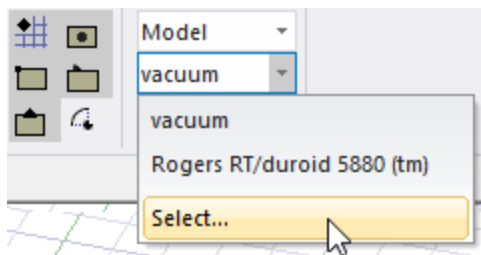

Figure 3-13: Selecting the Default Material

The *Select Definition* dialog box appears.

2. Select **polyethylene** from the list of library materials and click **OK**.

3. On the **Draw** ribbon tab, click 🗄 **Draw cylinder**.

4. Click at three arbitrary points to draw a cylinder freehand.

   After the third click, the *Properties* dialog box appears.

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

	Name	Value	Unit	Evaluated Value
	Command	CreateCylinder		
	Coordinate System	Global		
	Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
	Axis	Z		
	Radius	0.16	cm	0.16cm
	Height	-0.5	cm	-0.5cm
	Number of Segments	0		0

Figure 3-14: Coax Properties – *Command* Tab

6. On the **Attribute** tab:
   a. Change the **Name** to **Coax**.
   b. Set the **Color** to **medium gray** (column 4, row 6 of the color samples; Red: 128, Green: 128, Blue: 128).
   c. Set the **Transparent** value to **0.5**.
7. Click **OK** to close the *Properties* dialog box.
8. Clear the selection.

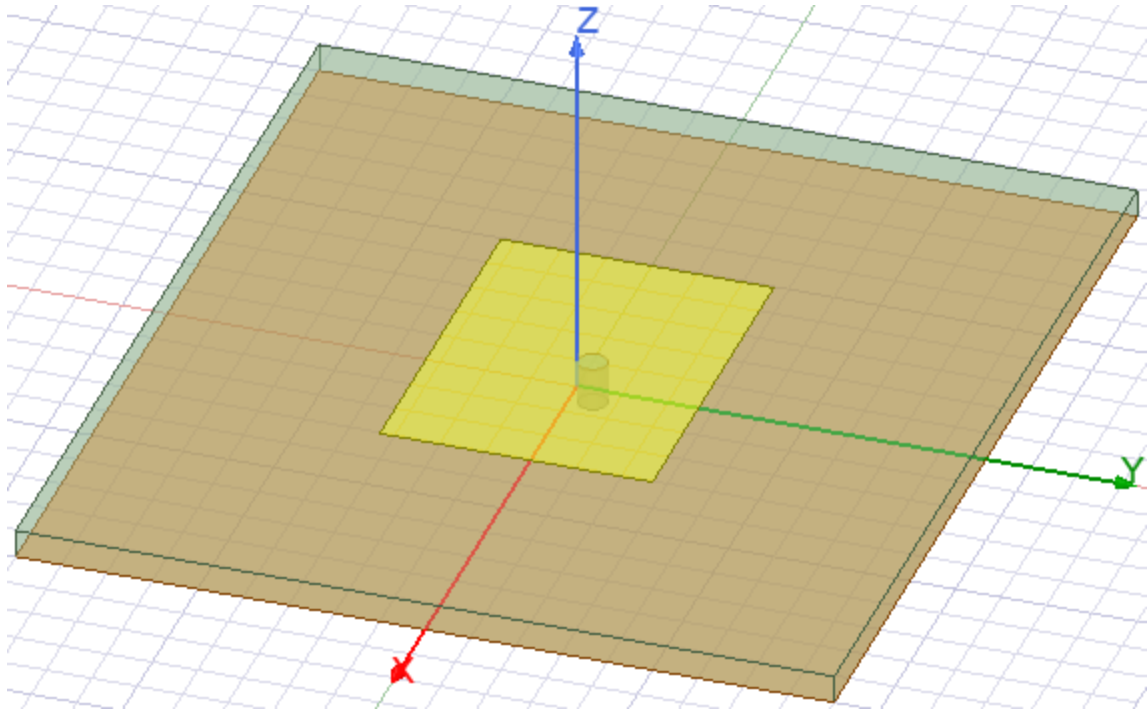   Your model should look like the following figure:

Figure 3-15: Coax Drawn

Create the Coax Pin

To create the coax pin:

1. Of the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu.
2. In the *Select Definition* dialog box, select **pec** (perfect electrical conductor) from the listed library materials and click **OK**.
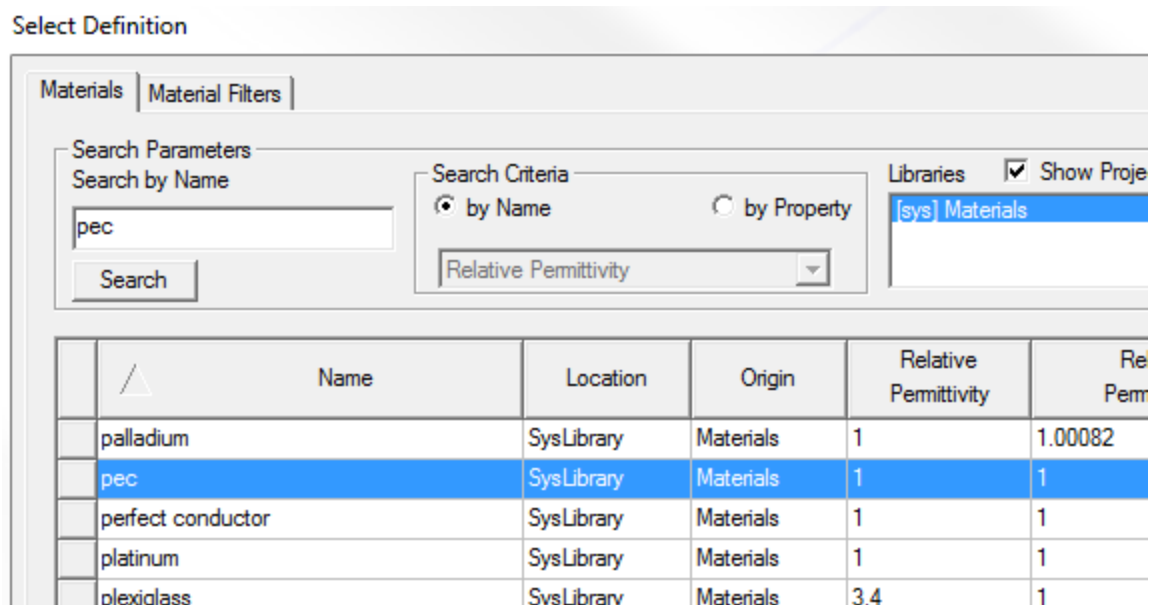
Figure 3-16: Setting Default Material to *pec*

3. Draw another arbitrary cylinder and edit the properties under the **Command** tab of the *Properties* dialog box, as shown in the following figure:

Name	Value	Unit	Evaluated Value
Command	CreateCylinder		
Coordinate Sys...	Global		
Center Position	-0.5 ,0 ,0	cm	-0.5cm , 0cm , 0cm
Axis	Z		
Radius	0.07	cm	0.07cm
Height	-0.5	cm	-0.5cm
Number of Seg...	0		0

Figure 3-17: Command dialog for Coax Pin Properties – *Command* Tab

4. On the **Attribute** tab, specify the following settings:
   a. Change the cylinder **Name** to **CoaxPin**.
   b. Set the **Color** to **red** (column 1, row 2 of the color samples; Red: 255, Green: 0, Blue: 0).
   c. To make the coax pin opaque, ensure that **Transparent** = **0**.
5. Click **OK** to close the *Properties* dialog box but leave the coax pin selected.

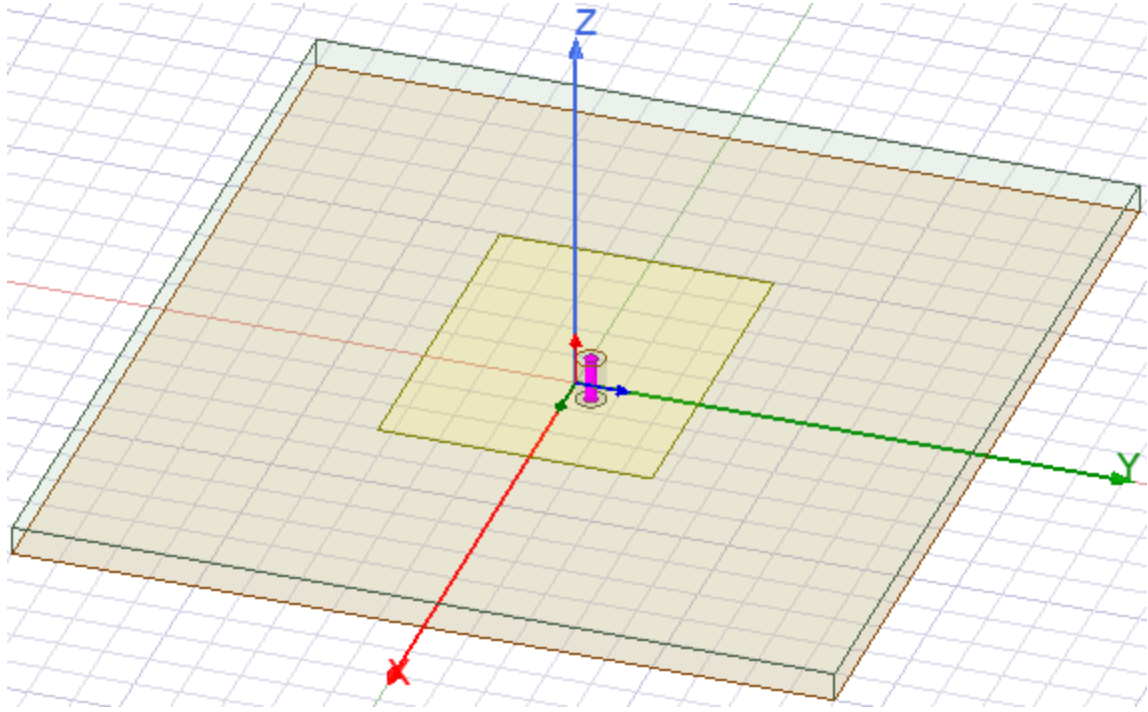   The model appearance is as follows:

Figure 3-18: Coax Pin Created

6. Clear the selection.

Create the Probe

To create the probe:

1. Under *Solids > polyethylene* in the History Tree, select **Coax**.
2. On the **Draw** ribbon tab, click 🔍 **Fit Selected** to zoom into the selected object.
3. On the **Draw** ribbon tab, click ⬤ **Draw cylinder** and then:

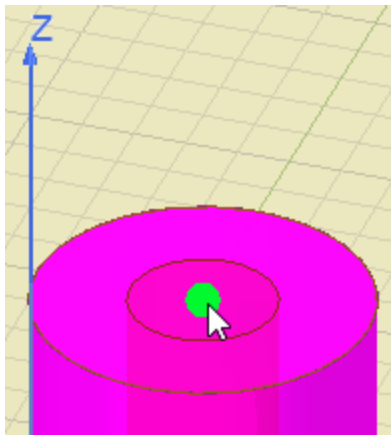   a. Click the center of the coax pin's top face. (A large solid dot indicates the snapping point.)

Figure 3-19: Center Snapping Point for Probe's Base Circle

b. Click on a quadrant point of the coax pin's top face. (A solid quarter-round shape indicates the snapping point.)
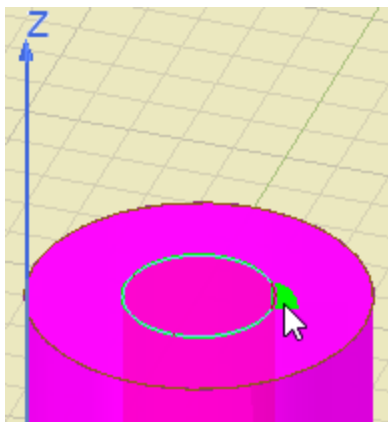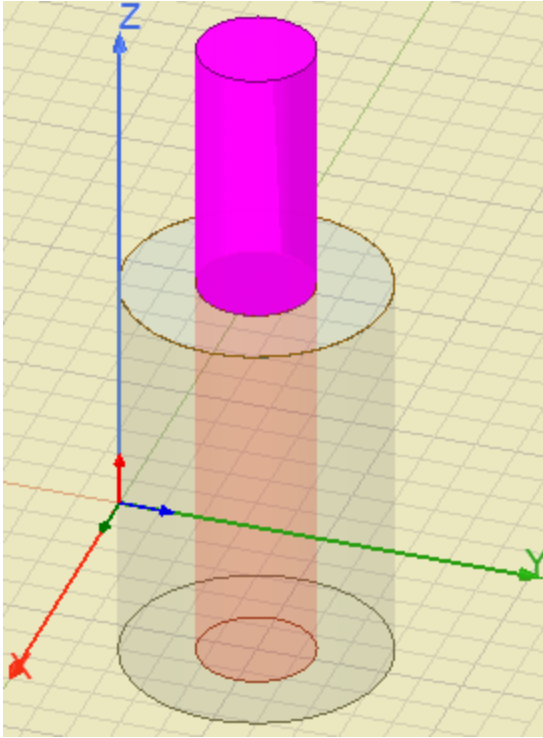

Figure 3-20: Quadrant Snapping Point for Probe's Base Circle Radius

c. Tab into the **dZ** coordinate entry text box, type **0.32** an press **Enter**.

4. On the **Attribute** tab of the *Properties* dialog box that appears, specify the following settings:

a. Change the cylinder **Name** to **Probe**.

b. Verify that the **Material** is **pec**.

c. Ensure that **Material Appearance** is not selected.

d. Change the **Color** to **red** (column 1, row 2 of the color samples; Red: 255, Green: 0, Blue: 0).

e. Verify that **Transparent** = **0**.

5. Click **OK** to close the *Properties* dialog box, but keep the probe selected.

6. Zoom out a little bit or pan the model if needed to see the top of the probe. Keep the *Probe* selected for now, which will make the next procedure easier.

Figure 3-21: Probe Drawn

> **Note:**
>
> It is not necessary for you to unite the conductors into a single object. The solver will treat the two cylinders as a single conductor.

Assign Excitation

Next, you will assign a wave port excitation to the bottom face of the coax.

1. Press **F** to change to the *face selection* mode.
2. Click within the perimeter of the bottom face of the Coax object, between the OD of the face and the OD of the coax pin:
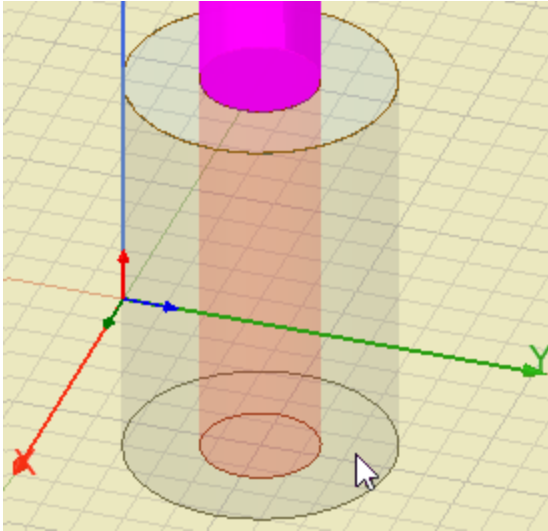
Figure 3-22: Clicking Location for Coax Bottom Face Selection

3. Press **B** (Next Behind) five times to select the bottom face of the coax:
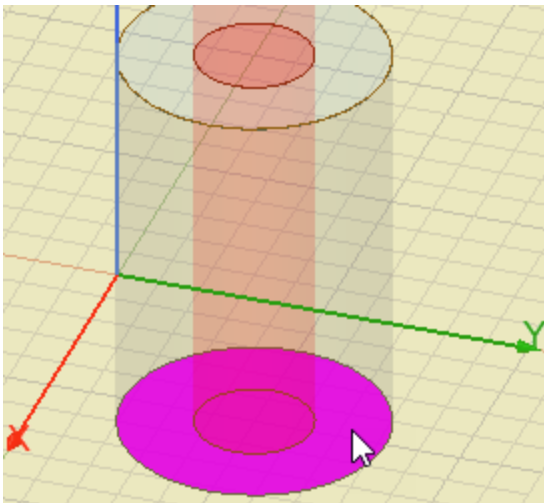

Figure 3-23: Bottom Face of Coax Selected

4. Right-click in the Modeler window and select **Assign Excitation** > **Port** > **Terminal Wave Port**.

    The *Reference Conductors for Terminals* dialog box appears.

5. Edit the settings as shown in the following figure and then click **OK**:
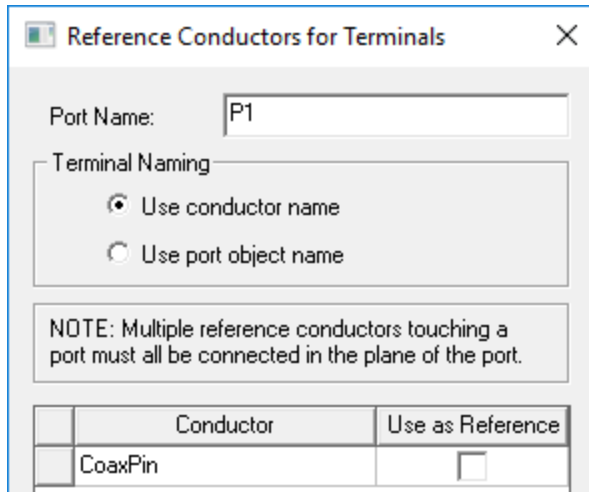
Figure 3-24: *Reference Conductors for Terminals* **Dialog Box**

The wave port and terminal are listed under *Excitations* in the Project Manager:
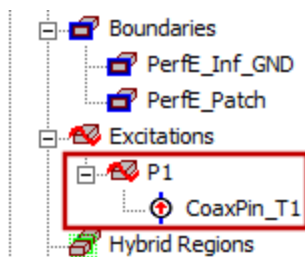

Figure 3-25: Excitations Listed in Project Manager

Create Air Body

Normally, you could take advantage of the *Auto Open Region* solution option or *Create Open Region* command to automate the process of defining an appropriately sized region around the model and assigning the radiation boundary. However, for this exercise you will manually create an air body of a custom size. Since the bottom of the substrate is defined as an infinite ground plane, there's no point in having an open region extend below the bottom of the model. Additionally, you will not provide padding between the front, back, or sides of the model and the radiation boundary. The air body will only extend above the model (that is, you will provide padding only in the +Z direction).

1. On the **Draw** ribbon tab, click ▯ **Draw box**.
2. Click three different places in the Modeler window to define the base rectangle corners and height of an arbitrary box.
3. Edit the values in the **Command** tab of the *Properties* dialog box that appears, as shown in the following figure:

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	-5 ,-4.5 ,0	cm	-5cm , -4.5cm , 0cm
	XSize	10	cm	10cm
	YSize	9	cm	9cm
	ZSize	3.32	cm	3.32cm

Figure 3-26: *Air* **Box Properties –** *Command* **Tab**

4. On the **Attribute** tab of the *Properties* dialog box, make the following changes:

   a. Change the **Name** to **Air**

   b. Choose **Edit** from the **Material** drop-down menu. Then, choose **air** from the *Select Definition* dialog box and click **OK**.

   c. Select the **Display Wireframe** option

   d. Ensure that **Material Appearance** is **not** selected.

   e. Set the **Color** to **medium gray** (column 4, row 6 of the color samples; Red: 128, Green: 128, Blue: 128).

   > **Note:**
   >
   > The transparency setting has no effect on the object appearance when it is displayed as a wireframe only.

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

6. Clear the selection and press **Ctrl+D** to fit the model to the viewing area:
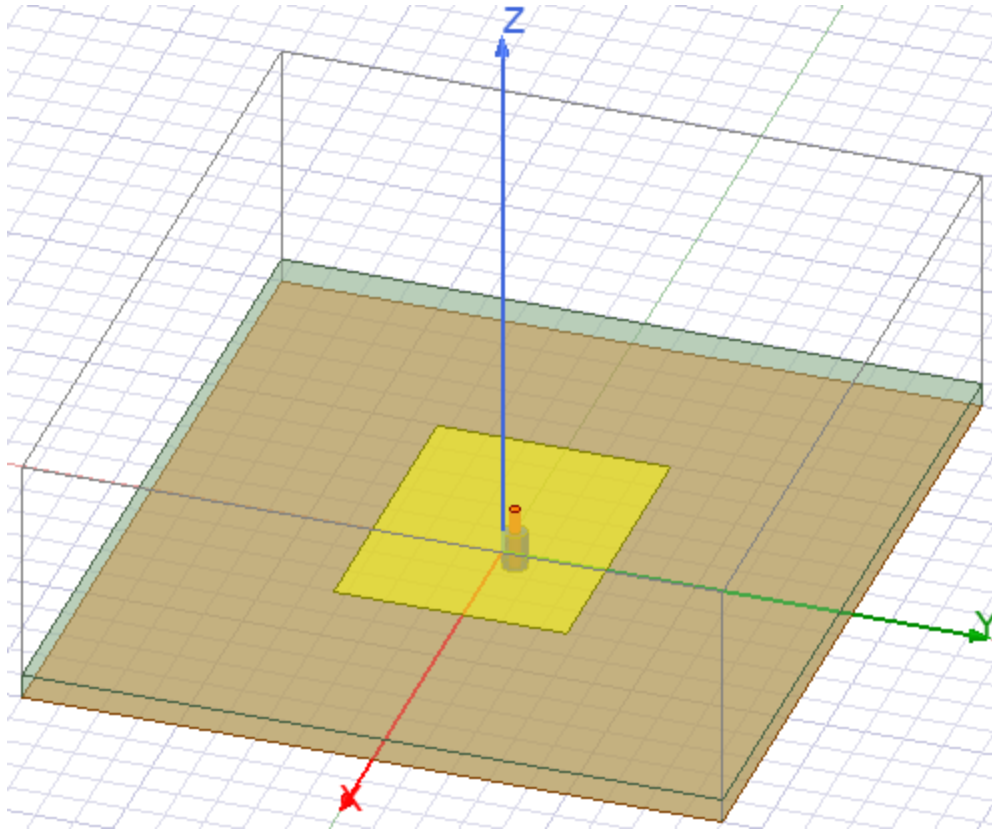
Figure 3-27: Air Body Created

Assign Radiation Boundary

In this section you will assign the radiation boundary to all the faces of the air body except the bottom face.

You should still be if the *face selection* mode.

1. On the Draw ribbon tab, click **Select By Name**.

    The *Select Face* dialog box appears.

2. In the Object name list, select **Air**.

3. Identify the **Face ID** associated with the bottom face (that is, the face at Z=0) by clicking each ID until the bottom face is selected.

4. Holding down the **Ctrl** key, deselect the *bottom* Face ID and select all of the remaining the Face IDs:
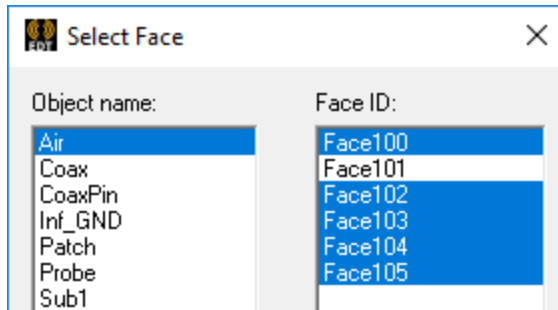
Figure 3-28: Selecting Radiation Boundary Faces

Note:

Your *Face ID* numbers may differ from those shown in the preceding figure. Numbering may vary for a few reasons (for example, if you did not exactly follow the construction sequence specified in this guide, if you deleted and recreated any object, or due to changes between different software versions).

5. Right-click in the Modeler window and choose **Assign Boundary** > **Radiation**.

    The *Radiation Boundary* dialog box appears.

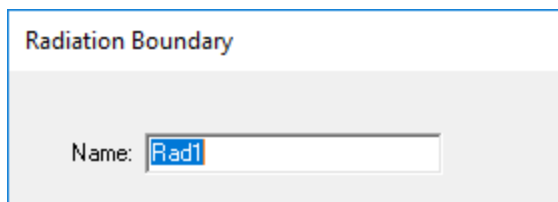6. Keep the default name, and shown in the following figure and click **OK**.

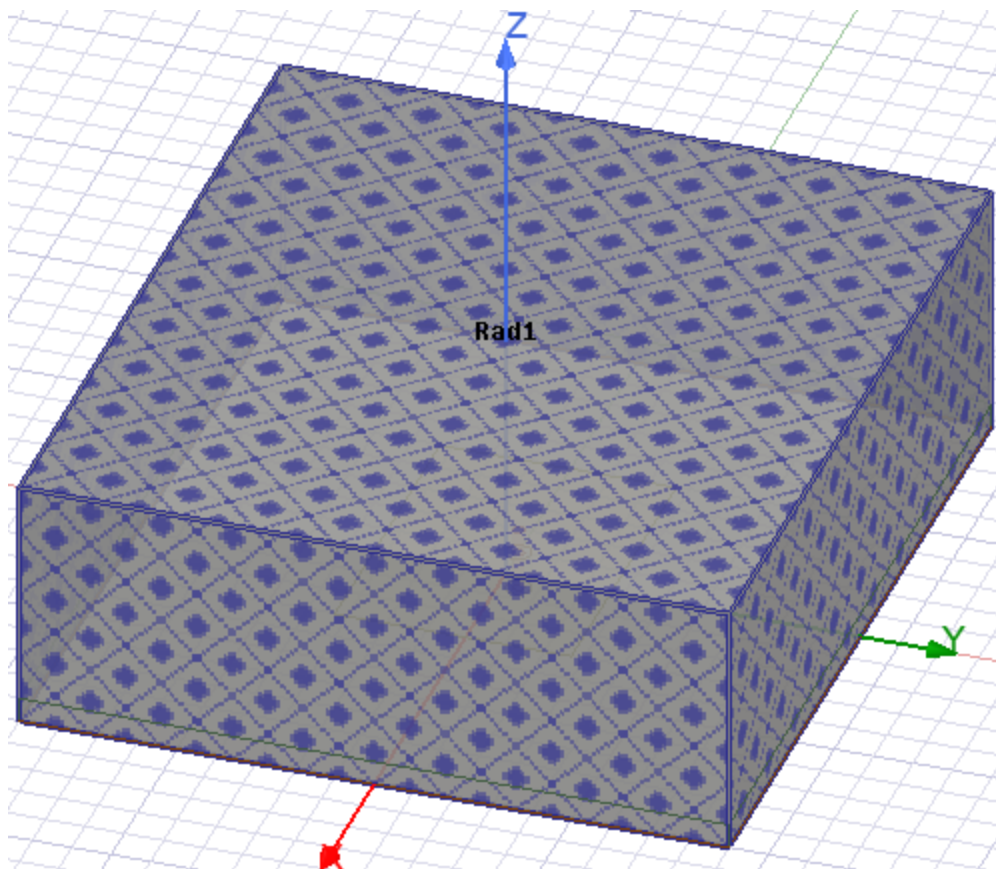**Figure 3-29: Radiation Boundary dialog box**

Figure 3-30: Radiation Boundary Applied

Create Radiation Setup

1. Right-click **Radiation** in the Project Manager and choose **Insert Far Field Setup > Infinite Sphere** from the shortcut menu.

    The *Far Field Radiation Sphere Setup* dialog box appears.

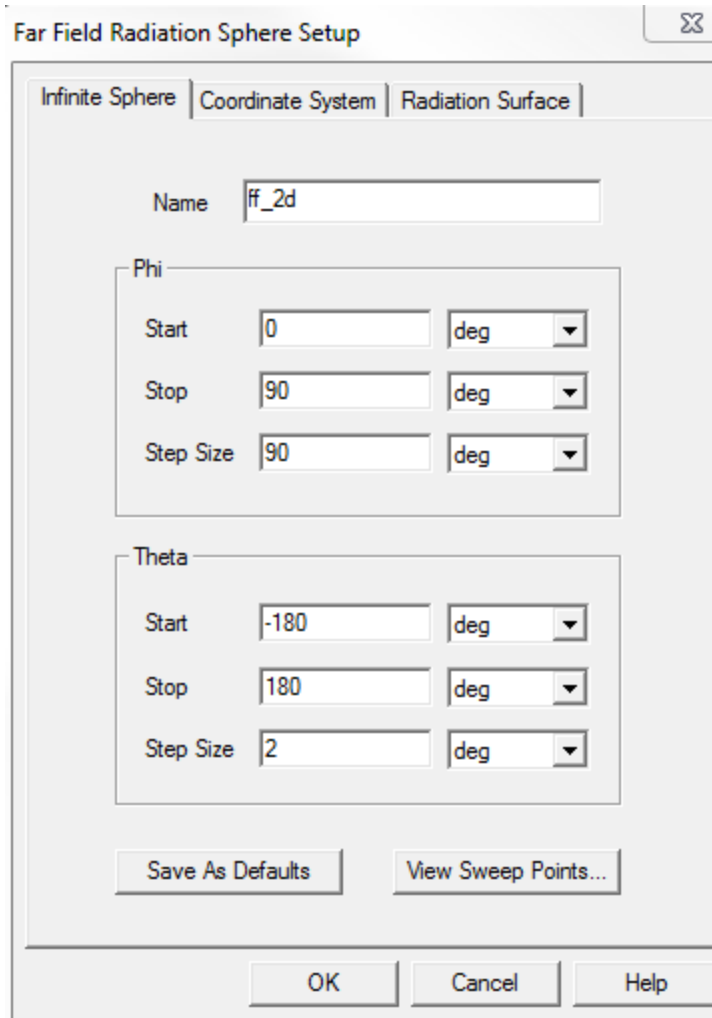2. Edit the **Name** and other settings as shown in the following figure:

Figure 3-31: *Far Field Radiation Sphere Setup* Dialog Box

3. Click **OK**.

An *ff_2d* entry appears under *Radiation*in the Project Manager.

4. 💾 **Save** your model.

This step completes the model construction process.

4 - Add Solution Setup and Frequency Sweep

This chapter contains the following topics:

- Add Solution Setup
- Add Frequency Sweep

Add Solution Setup

1. On the **Simulation** ribbon tab, click **Setup** > **Advanced**.

   The *Driven Solution Setup* dialog box appears.

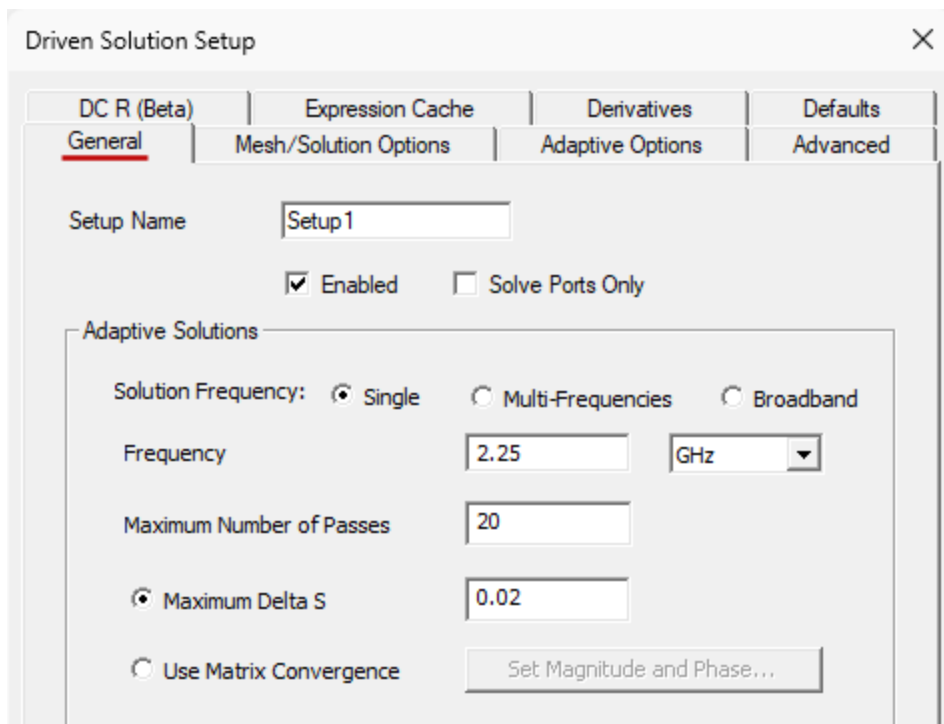2. On the **General** tab, edit the settings as shown in the following figure:

Figure 4-1: *Driven Solution Setup* **Dialog Box**

> **Note:**
>
> This dialog box defines the solution frequency at which automatic mesh adaptation is performed, the maximum number of passes during which HFSS is automatically refining the mesh for accurate results, and the stopping criteria for the mesh adaptation process. In this example, when the maximum change in S matrix values is 0.02 or less, the solution is considered to be converged.

3. Click **OK**.

   The *Edit Frequency Sweep* dialog box opens automatically when you complete an HFSS solution setup and the model already has at least one port assigned.

Add Frequency Sweep

4. Edit the settings as shown in the following figure and then click **OK**.
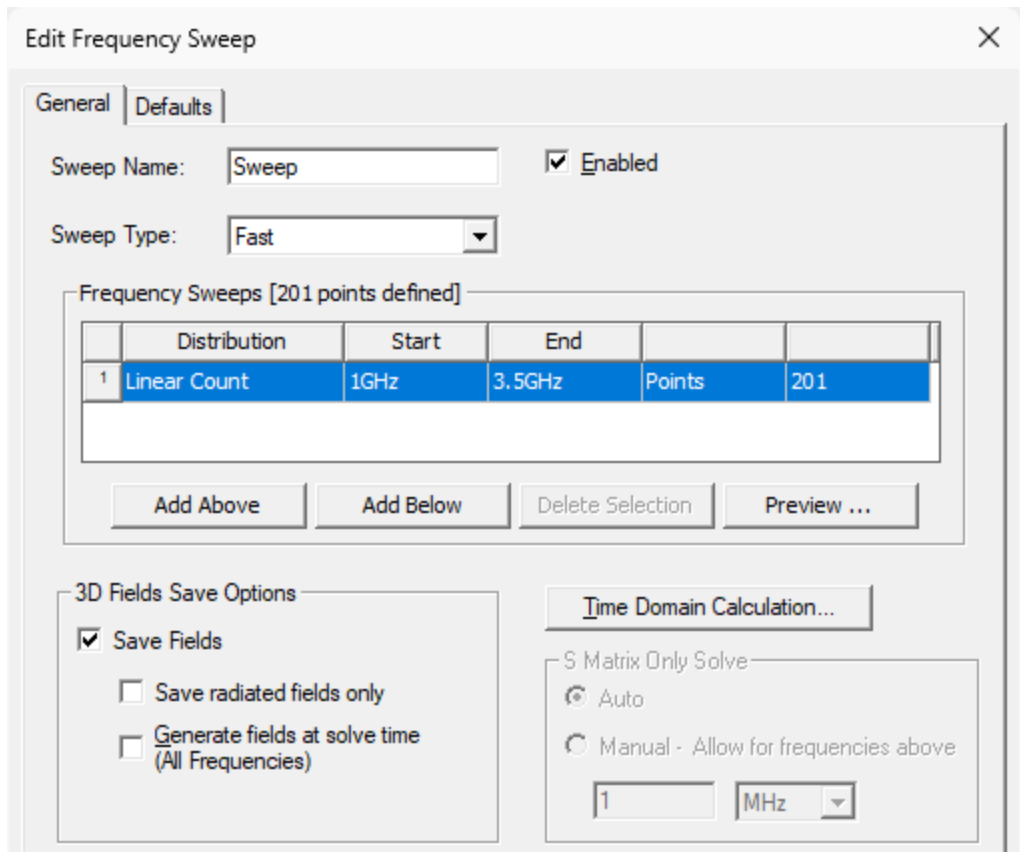

Figure 4-2: *Edit Frequency Sweep* **Dialog Box**

5. 💾 **Save** the project.

---

5 - Analyze and Review Results

This chapter contains the following topics:

- Model Validation
- Analyze All
- Review Solution Data
    - Review the Profile Tab
    - Review the Convergence Tab
    - Review the Matrix Data Tab
    - Review the Mesh Statistics Tab
- Create Terminal Solution Data Report
- Create Gain Plot

Model Validation

You are all set to validate the project.

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

    The *Validation Check* dialog box appears. There should be no errors or warnings:
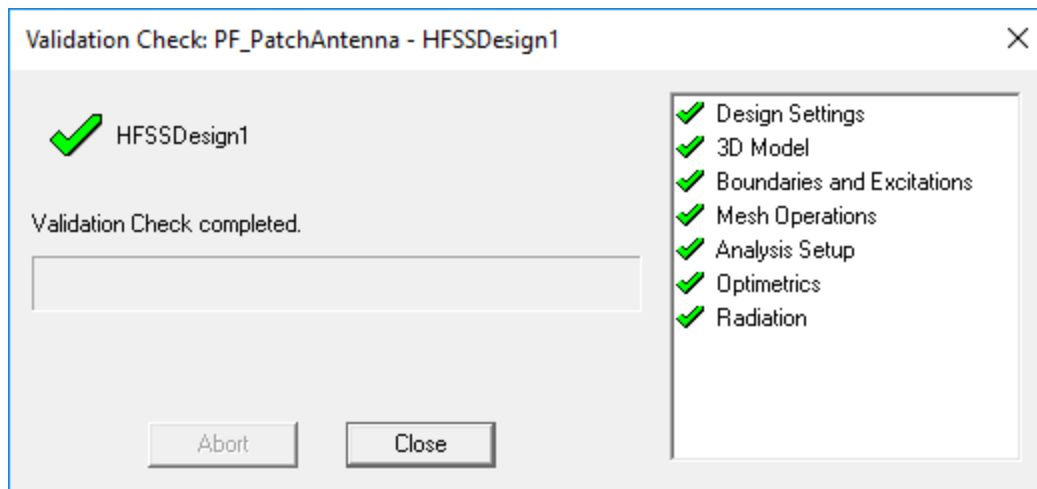

Figure 5-1: Validation Check dialog box

2. Click **Close**.

Analyze All

You are all set to analyze the project (that is, run the simulation).

3. On the **Simulation** ribbon tab, click ⚙ **Analyze All**.

> **Note:**
>
> The Message Manager window notifies you when *Normal completion of sim-ulation on server* has been achieved. With a relatively current workstation, this solution will likely take less than one minute to complete.

Review Solution Data

You can view the solution data to better understand the results and the simulation.

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

    The *Solutions* dialog box appears.

    You can select the various tabs to view the corresponding results. The individual tabs of the *Solutions* dialog box are covered in the four following topics:

    - Review the Profile Tab
    - Review the Convergence Tab
    - Review the Matrix Data Tab
    - Review the Mesh Statistics Tab

Review the Profile Tab

1. On the *Solutions* dialog box select the **Profile** tab.

    In the *Profile* tab, you can view a synopsis of the simulation results ranging from mesh cre-ation and refinement to information about the different adaptive passes, the matrix assembly, solver, field recovery, data transfer, and frequency sweep. The *Real Time, CPU Time,* and *Memory* usage are reported for the various processes.

    The more highly refined the mesh (that is, the higher the number of tetrahedra generated), the more accurate the HFSS solution is. The mesh is refined locally as needed, rather than globally, to produce accurate results with optimal efficiency. However, there is a trade-off in the number of tetrahedra used and the computational resources required. Keep in mind that increased accuracy requires more computational resources and more time.

Task	Real Time	CPU Time	Memory	Inf
Adaptive Pass 9				Frequency: 2.25GHz
Adaptive Refine	00:00:00	00:00:00	27.3 M	Tetrahedra: 10635, Cores: 1
Simulation Setup	00:00:00	00:00:00	64.3 M	Tetrahedra: 10278, Disk: 4.11 KB
Matrix Assembly	00:00:00	00:00:03	153 M	Tetrahedra: 10278, P1 Triangles: 105, Disk: 0 Bytes
Matrix Solve	00:00:00	00:00:06	365 M	Type: DCS, Cores: 12, Matrix size: 60897, Matrix bandwidth: 20.3, Disk
Field Recovery	00:00:00	00:00:01	365 M	Excitations: 1, Disk: 471 KB
Data Transfer	00:00:00	00:00:00	89.2 M	Adaptive Pass 9
				Max Mag. Delta S: 0.01755
Adaptive Meshing				Elapsed Time: 00:00:34
				Adaptive Passes converged
Frequency Sweep				Time: 10/27/2022 13:44:18
				HPC: Enabled
Solution Sweep				Fast Sweep
				From 1 GHz to 3.5 GHz, 200 Steps
Simulation Setup	00:00:00	00:00:00	61.4 M	Disk: 0 Bytes
Matrix Assembly	00:00:01	00:00:03	147 M	Tetrahedra: 10278, P1 Triangles: 105, Disk: 0 Bytes
Matrix Solve	00:00:10	00:00:46	420 M	Type: DCS, Cores: 12, Matrix size: 60897, Matrix bandwidth: 20.3, Redu
Field Recovery	00:00:00	00:00:00	420 M	Excitations: 1, Disk: 2.43 MB
Frequency Sweep				Elapsed Time: 00:00:13
Simulation Summary				
Design Validation				Elapsed Time: 00:00:00, Total Memory: 86 MB
Initial Meshing				Elapsed Time: 00:00:05, Total Memory: 54.7 MB
Adaptive Meshing				Elapsed Time: 00:00:34, Average memory/process: 365 MB, Max memo
Frequency Sweep				Elapsed Time: 00:00:13, Total Memory: 420 MB
				Max solved tets: 10278, Max matrix size: 60897, Matrix bandwidth: 20.3
Solution Process				Elapsed Time: 00:00:53, ComEngine Memory: 89.3 M
				Stop Time: 10/27/2022 13:44:32, Status: Normal Completion

Figure 5-2: Solution Data – *Profile* Tab

Review the Convergence Tab

The Convergence panel lets you view the plot or table of **Max Mag Delta S** versus **Pass Number**.

---

1. On the *Solutions* dialog box, select the **Convergence** tab.
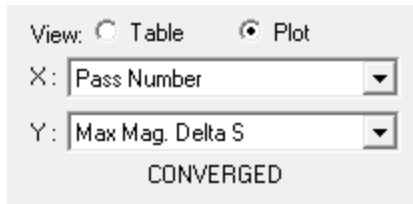2. Set the options as shown in the following figure:

View: ○ Table   ⊙ Plot

X: | Pass Number | ▼ |

Y: | Max Mag. Delta S | ▼ |

CONVERGED

Figure 5-3: Convergence View Mode and X and Y Axis Settings
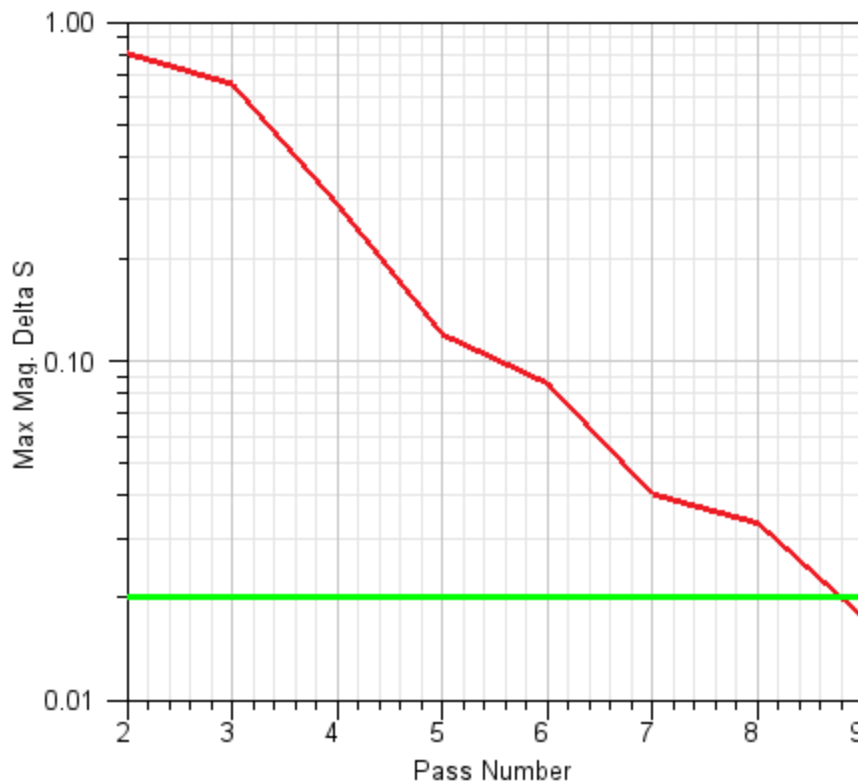
The Convergence *Plot* is displayed:

Figure 5-4: Convergence Plot

> **Note:**
>
> Convergence occurs in the 9th pass.

3. Click **Table** to view the convergence results listed numerically in a table:

Pass Number	Solved Elements	Max Mag. Delta S
1	1250	N/A
2	1630	0.79757
3	2121	0.64846
4	2761	0.28742
5	3590	0.11886
6	4671	0.085308
7	6074	0.040256
8	7898	0.033139
9	10278	0.017552

Figure 5-5: Convergence Table

Review the Matrix Data Tab

On the **Matrix Data** tab, you can view the S (scatter), Y (admittance), or Z (impedance) matrix coefficients; propagation constant (Gamma), or characteristic impedance (Zo) computed during each adaptive, nonadaptive, or sweep solution. Various display formats are available (such as Magnitude/Phase(deg), dB/Phase(deg), Real, Imaginary, and more).

1. On the *Solutions* dialog box, select the **Matrix Data** tab.

2. Select the **Display All Frequencies** option.

   By default, the *S Matrix* is displayed for the specified frequency sweep using the *Magnitude/Phase(deg)* format:

Figure 5-6: Matrix Data panel

Note:

- To see the available matrix format options, select the *Format* subtab.
- To view a real-time update of the Matrix Data while a solution is still being solved, set the *Simulation* options at the top of the dialog box to *Setup1* and *Last Adaptive*.

Review the Mesh Statistics Tab

This panel shows statistics such as the minimum and maximum edge lengths and volumes of the tetrahedral elements and the number of elements generated. RMS lengths, mean volumes, and standard deviations of the volumes are also given. Each quantity is reported separately for each of the individual solid objects comprising the model.

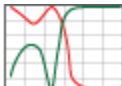1. On the *Solutions* dialog box, select the **Mesh Statistics** tab.

Total number of elements: 10635

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Air	3889	0.0878576	3.75	0.978424	9.69179e-06...	2.94705	0.0694266	0.219373
Coax	605	0.0412709	0.289347	0.133381	2.49248e-06...	0.000200765...	5.23739e-05...	3.97215e-05
CoaxPin	218	0.0535757	0.289347	0.137883	1.05265e-06...	0.000213905...	3.44064e-05...	4.00568e-05
Probe	139	0.0535757	0.217968	0.139858	4.69994e-07...	0.000192125...	3.45351e-05...	3.69619e-05
Sub1	5784	0.072008	2.59	0.47787	3.53462e-06...	0.218976	0.00497842...	0.0195434

Figure 5-7: Mesh Statistics Tab

2. After you have viewed all of the results in the different panels on the *Solution Data* dialog box, click **Close** and proceed to the next section.

Create Terminal Solution Data Report

This section deals with report creation. In this example, you will create a rectangular plot of the terminal S-parameter at Coax Pin, Terminal 1 representing the signal reflection at this terminal [specifically, St(CoaxPin_T1,CoaxPin_T1)].

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

   The *Report* dialog box appears.

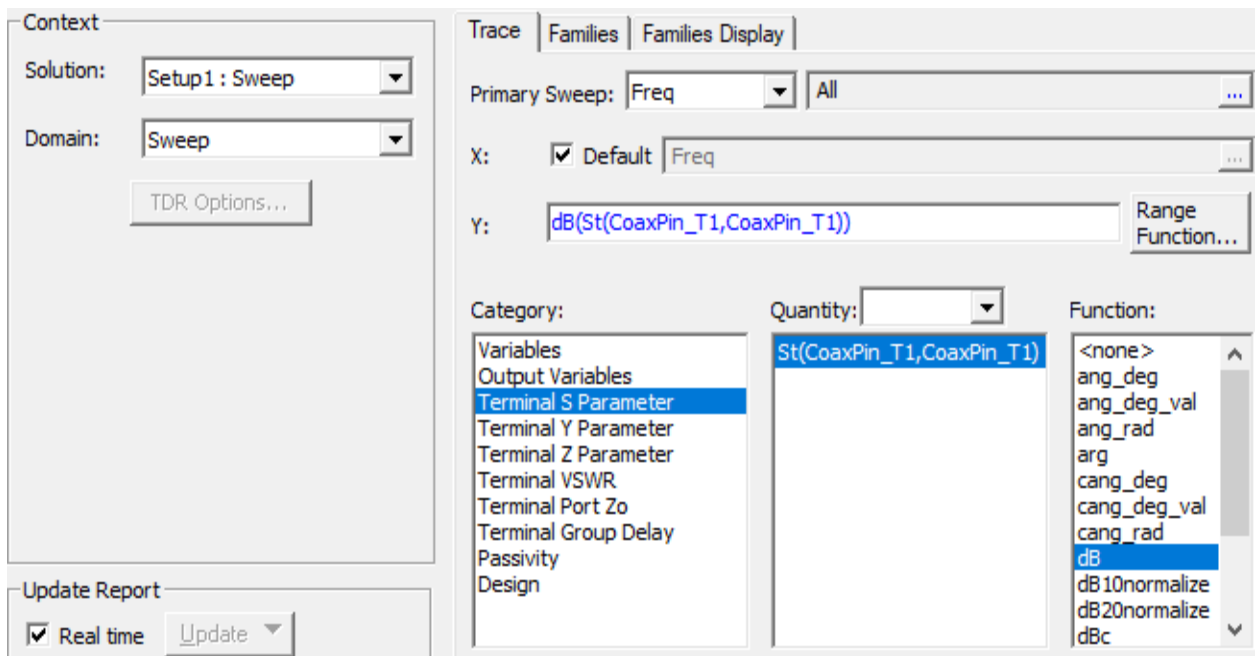2. Ensure that the settings are as shown in the following figure:

Figure 5-8: Report dialog box

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

    Terminal S Parameter Plot 1 appears in a new window.

4. Click on the trace to select it. (The trace line will become thicker to indicate that it is selected, and its properties will appear in the docked *Properties* window.)

5. In the *Terminal S Parameter Plot 1* window, right-click and choose **Marker** > **Add Minimum**.
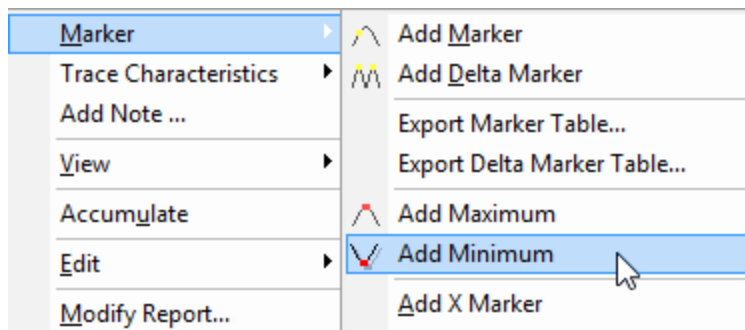

Figure 5-9: Plot Shortcut Menu

A marker symbol and label (*m1*) are added to the trace along with a table showing the X and Y values at that point.

6. Click and drag to reposition the plot legend and marker values to suitable locations:
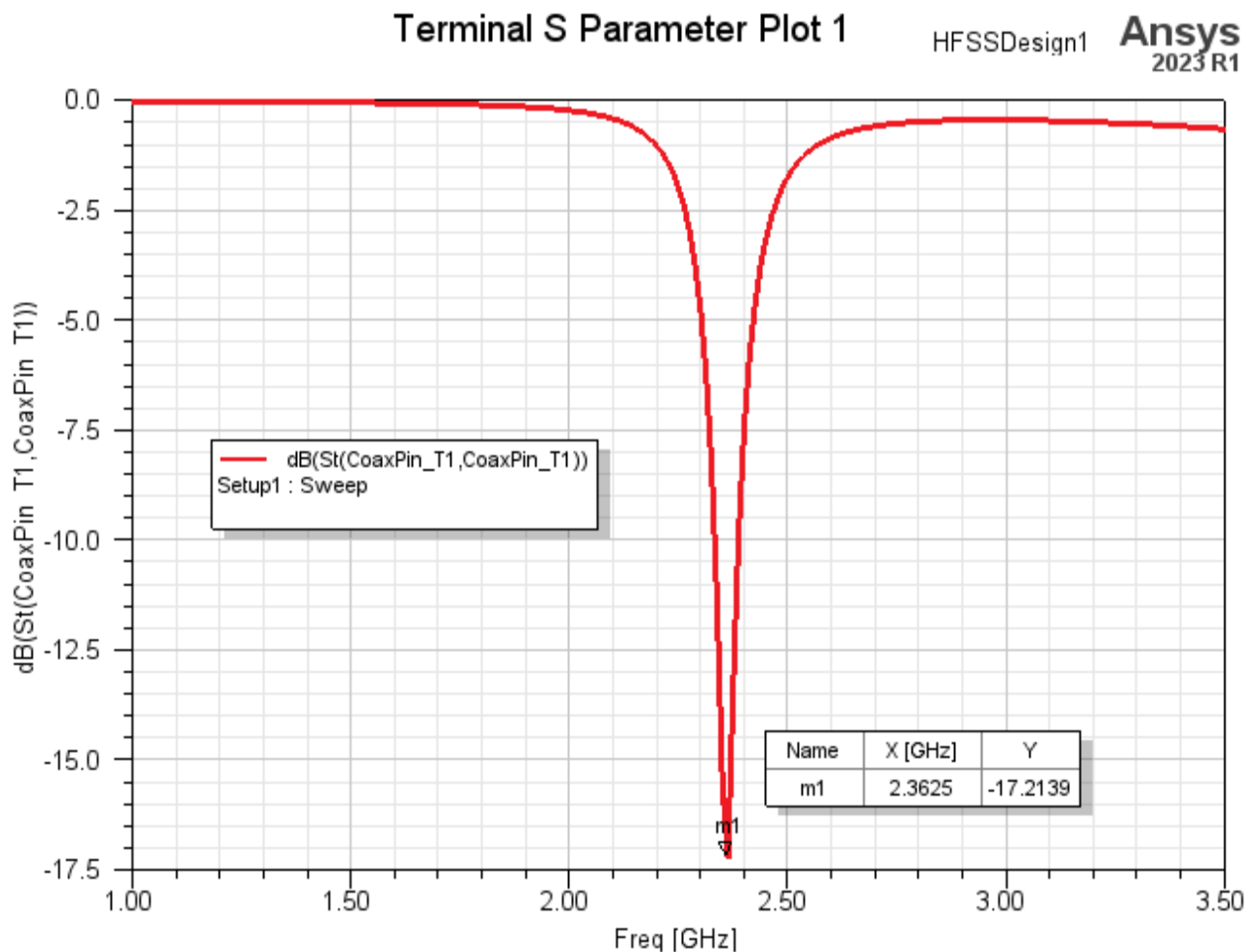
Figure 5-10: S Parameter vs. Frequency Plot – Signal Reflection at CoaxPin_T1

Create Gain Plot

Now, you can plot far-field quantities, such as antenna gain. To create a 2D polar far field plot:

1. On the **Results** ribbon tab, click **Far Fields Report** > **Mag/Ang Polar**.

   The *Report* dialog box appears.
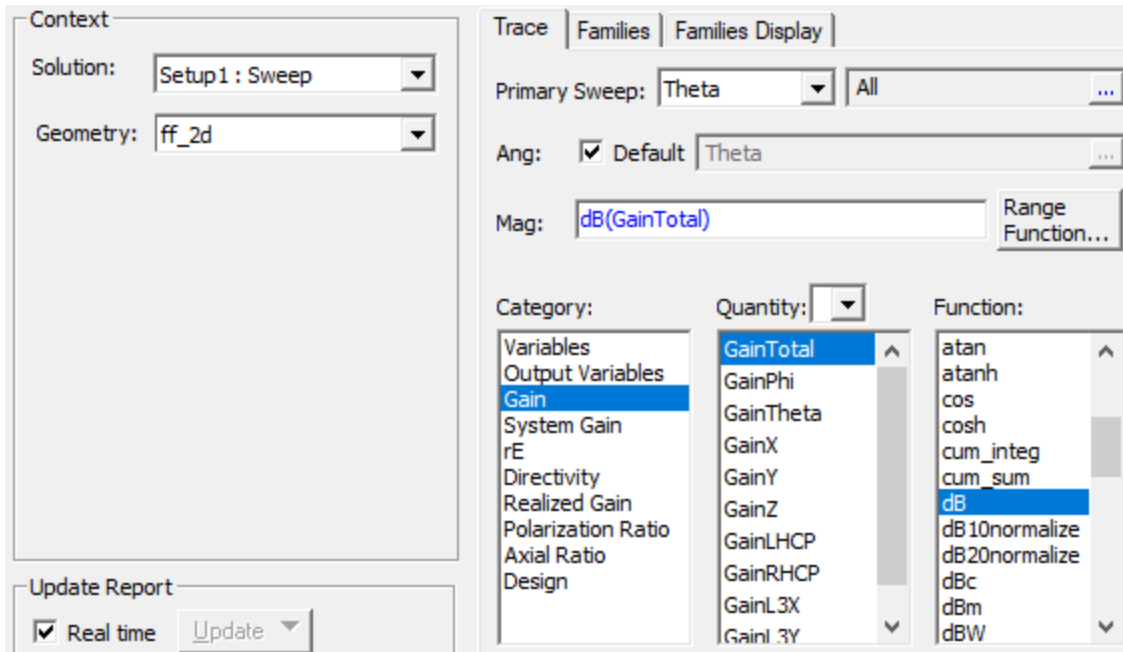
2. Edit the settings as shown in the following figure:

Figure 5-11: Gain Plot Report Settings

3. Click **New Report** and **Close**.
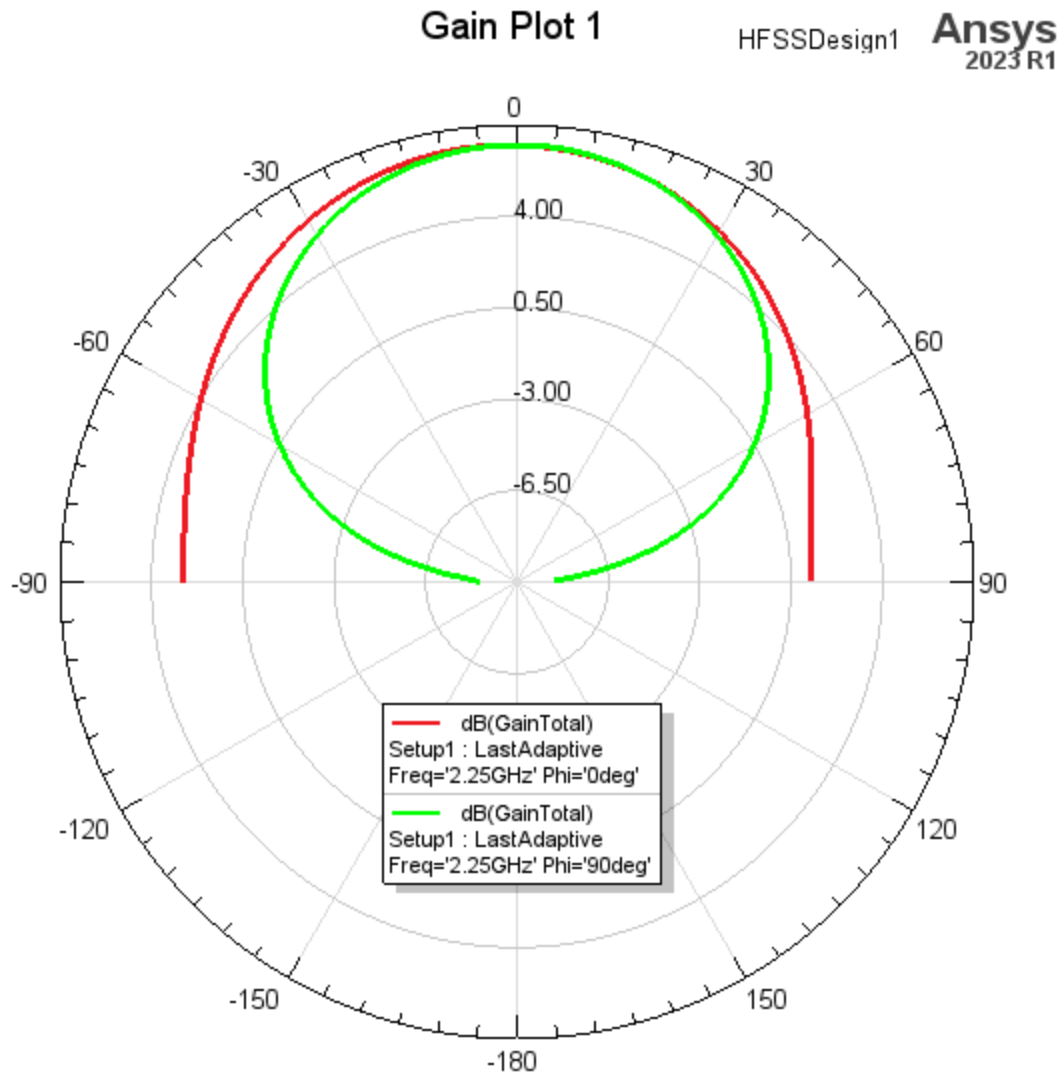
   Gain Plot 1 appears in a new window:

Figure 5-12: Gain Plot

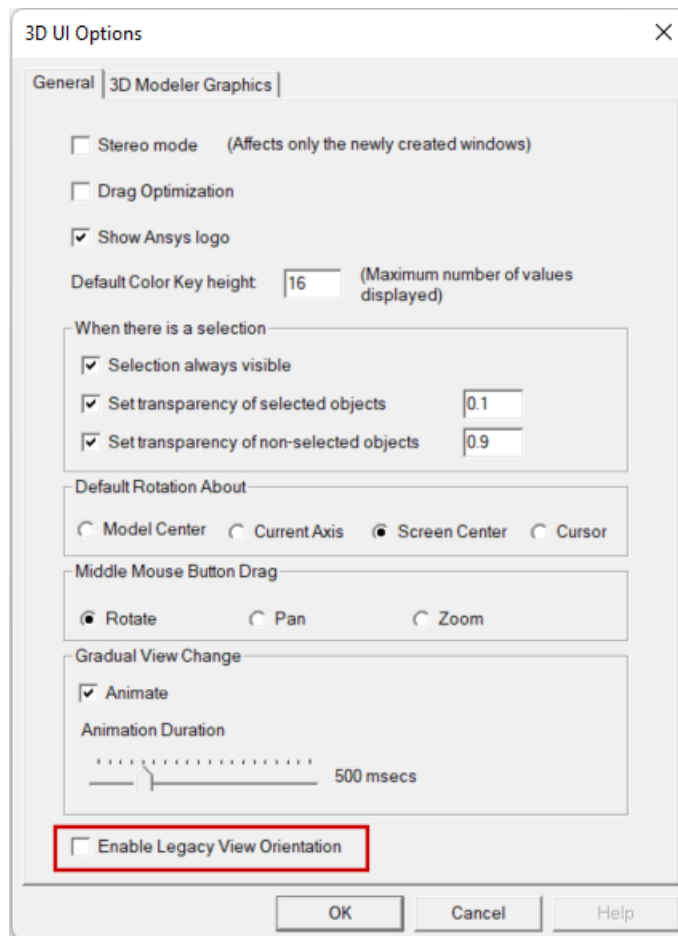# 6 - Optionally, Restore Current View Orientations

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.