Ansys 2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT



# Getting Started with HFSS™: Monocone Antenna

Ansys

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 R2
July 2025

## 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 exporting 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 proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

### Conventions Used in this Guide

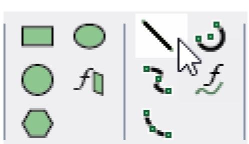
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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.

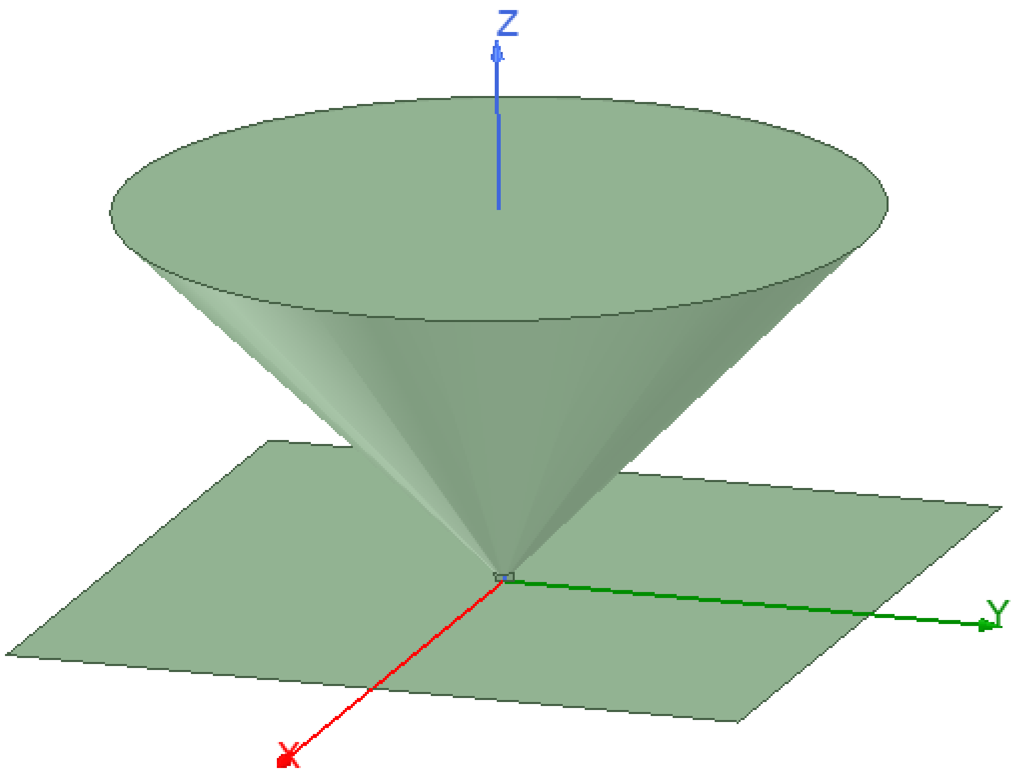
You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

# Table of Contents

# 1 - Introduction

This example looks at the radiation pattern from a monocone antenna using the HFSS design type with an IE hybrid region applied to the model objects. An *IE Region* results in a fullwave *Integral Equation* solver being used to analyze the model. This solver is particularly well suited for analyzing large conducting objects, and it eliminates the necessity of creating an open region (that is, a vacuum or air box) around the model to determine far field effects.

> **Note:**
>
> IE Regions in an HFSS design have replaced the legacy HFSS-IE design type. If you open a legacy HFSS-IE design, you are prompted with the option to automatically convert the model to an HFSS design with an IE Region applied to its objects.




Building this model is straightforward. You need to create a 3D solid cone and two rectangular sheet objects—one to serve as an infinite ground plane and one as a port.

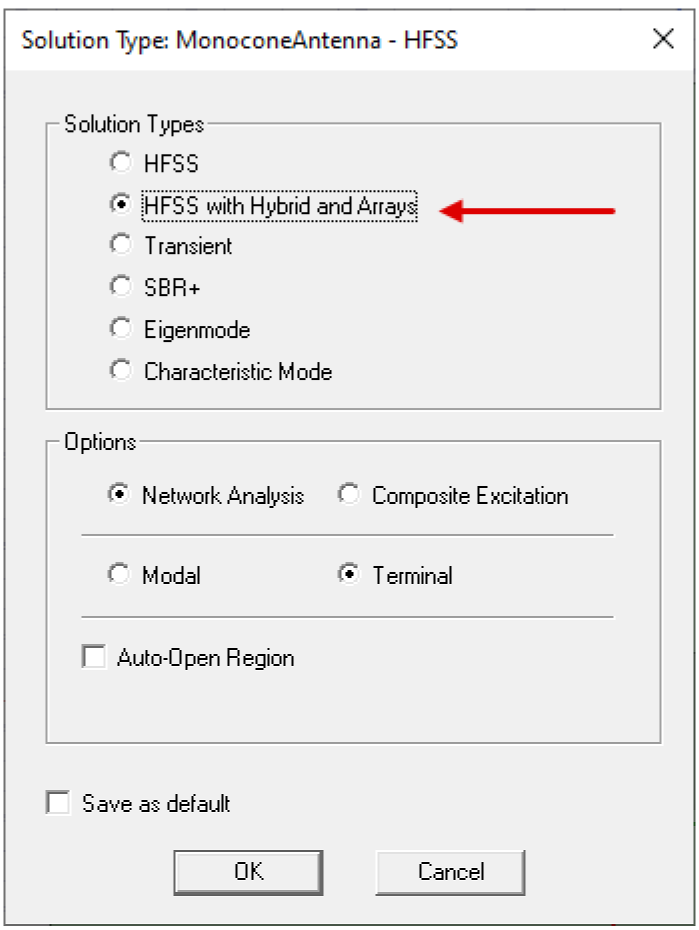
## Launch Application, Insert HFSS Design, and Set Solution Type

Prepare for building the model by completing the following steps:

1. Launch the **Ansys Electronics Desktop** application.
2. Insert a new project (if an empty one doesn't already exist), and name it **MonoconeAntenna**.
3. On the **Desktop** ribbon tab, click **HFSS** to insert an *HFSS* design.

> **Note:**
>
> You do not have to access the *HFSS* drop-down menu. Inserting an *HFSS* design (as opposed to an *HFSS 3D Layout* design) is the default action when you click this icon.

4. Using the menu bar, click **HFSS** > **Solution Type**.
5. In the *Solution Type* dialog box that appears, choose **HFSS with Hybrid and Arrays**. Also, ensure that the remaining settings match the following image:




6. Click **OK**.

# 2 - Create the Model

In this section you will draw the model geometry and assign the excitation. The following procedures are covered:

- Enable Legacy View Orientations
- Set Drawing Options
- Set Units and Default Material
- Draw the Ground Plane Rectangle
- Assign an Infinite Ground Plane
- Draw the Cone
- Draw the Port Rectangle
- Assign Lumped Port Excitation
- Assign an IE Region

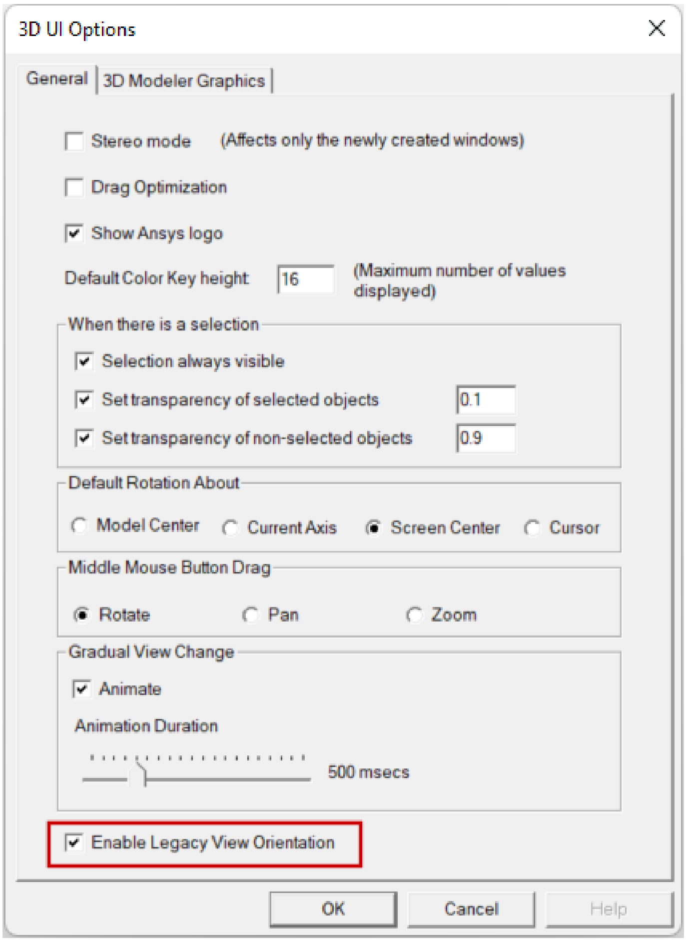
## Enable Legacy View Orientations

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 Drawing Options

Before creating any geometry, you must customize a couple of program options to ensure that your experience is consistent with the instructions in this guide.

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

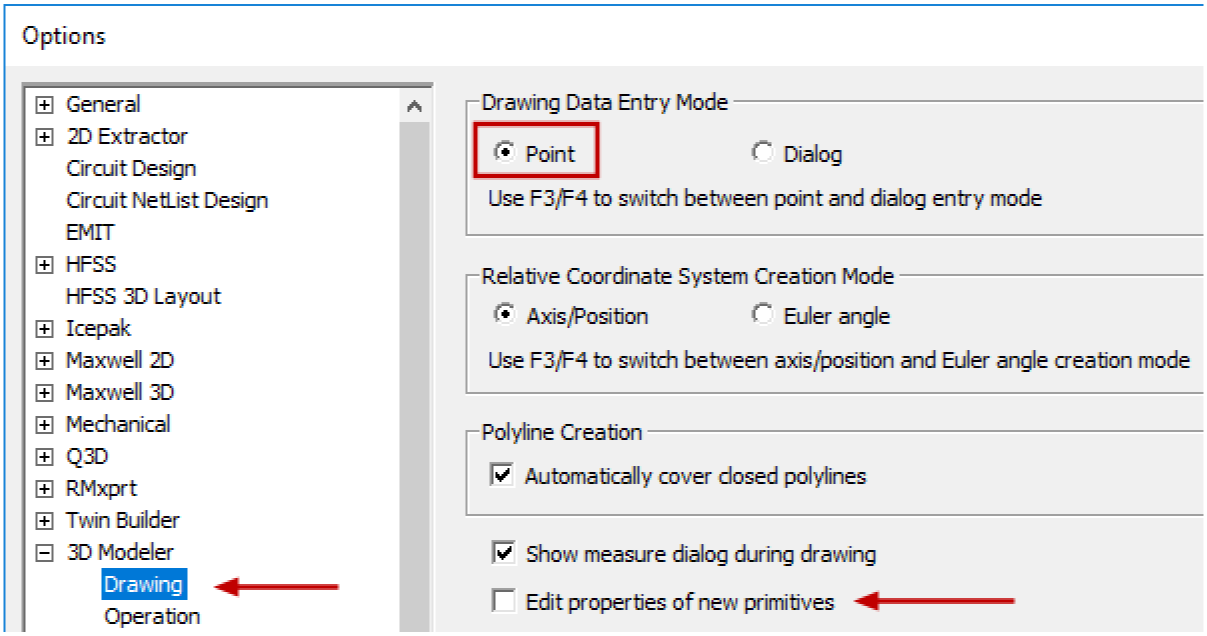
   The *Options* dialog box appears.

2. On the left side of the dialog box, expand the **3D Modeler** branch and select **Drawing**.
3. Specify the following 3D Modeler – Drawing settings:
   a. Select the **Point** option under *Drawing Data Entry Mode*.

   > **Note:**
   >
   > This option enable you to define objects by clicking grid points or snapping points in the Modeler window or by typing coordinates in the coordinate text boxes that are located in the status bar. The alternative method opens a dialog box as soon as you execute a drawing command, and you specify the properties entirely within the dialog box.

   b. Ensure that the **Edit properties of new primitives** option is *cleared* (that is, *not* selected).

   > **Note:**
   >
   > When selected, this option causes a dialog box to appear for editing the object's parameters whenever you create an object (such as a box, line, or cylinder). For this exercise, you will instead use the docked *Properties* window to edit the parameters of your drawn objects.




4. Click **OK**.

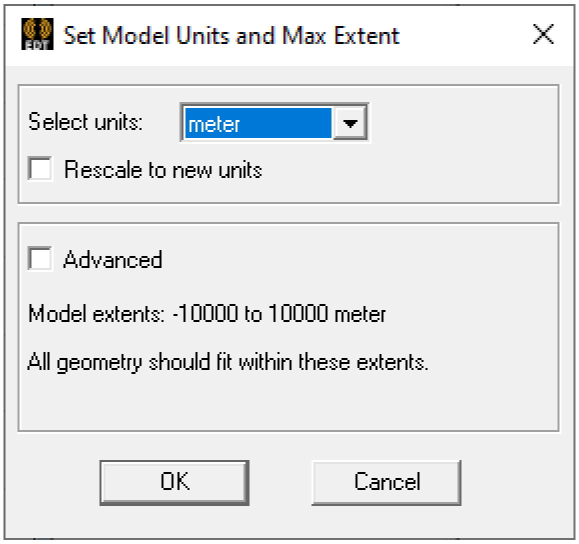
## Set Units and Default Material

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

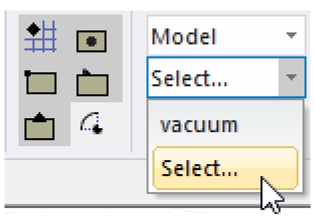
   The *Set Model Units and Max Extent* dialog box appears.

2. From the **Select units** drop-down menu, choose **meter**.

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



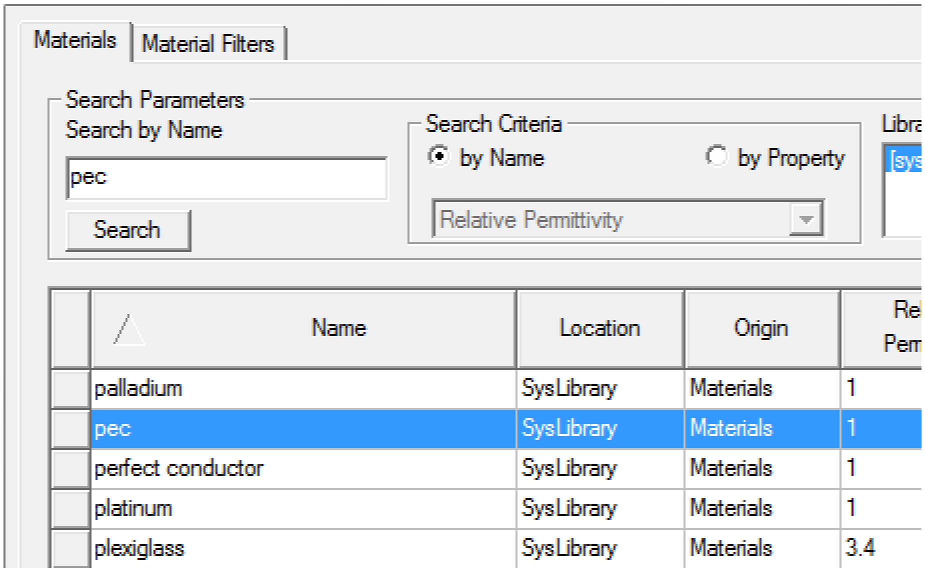

3. Click **OK**.
4. At the right end of the **Draw** ribbon tab, click **Default material** > **Select**:




The *Select Definition* dialog box appears.

5. Type **pec** in the **Search by Name** text box.

The *pec* material is highlighted in the list of library materials:




6. Click **OK** to select *pec* (perfect electrical conductor) as the default material and to close the dialog box.

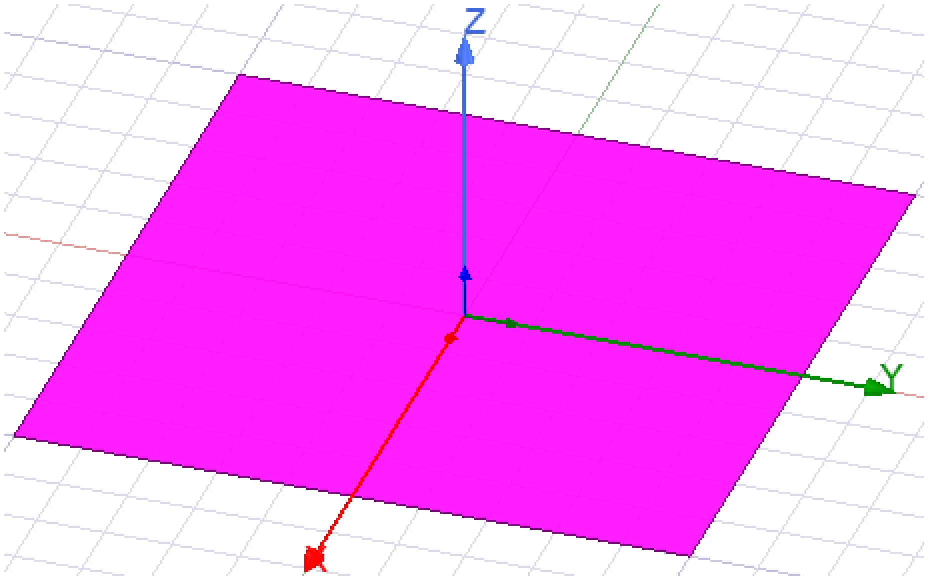
## Draw the Ground Plane Rectangle

1. On the **Draw** ribbon tab, click **Draw rectangle**.
2. Click on two different grid points to create an arbitrary rectangle.
3. In the **Command** tab of the docked *Properties* window, edit the values to match the following figure:

| Name | Value | Unit | Evaluated Value |
|---|---|---|---|
| Command | CreateRectangle | | |
| Coordinate System | Global | | |
| Position | -0.5 ,-0.5 ,0 | meter | -0.5meter , -0.5meter , 0meter |
| Axis | Z | | |
| XSize | 1 | meter | 1meter |
| YSize | 1 | meter | 1meter |

   Keep the rectangle selected.

4. On the **Draw** ribbon tab, click **Fit Selected**.

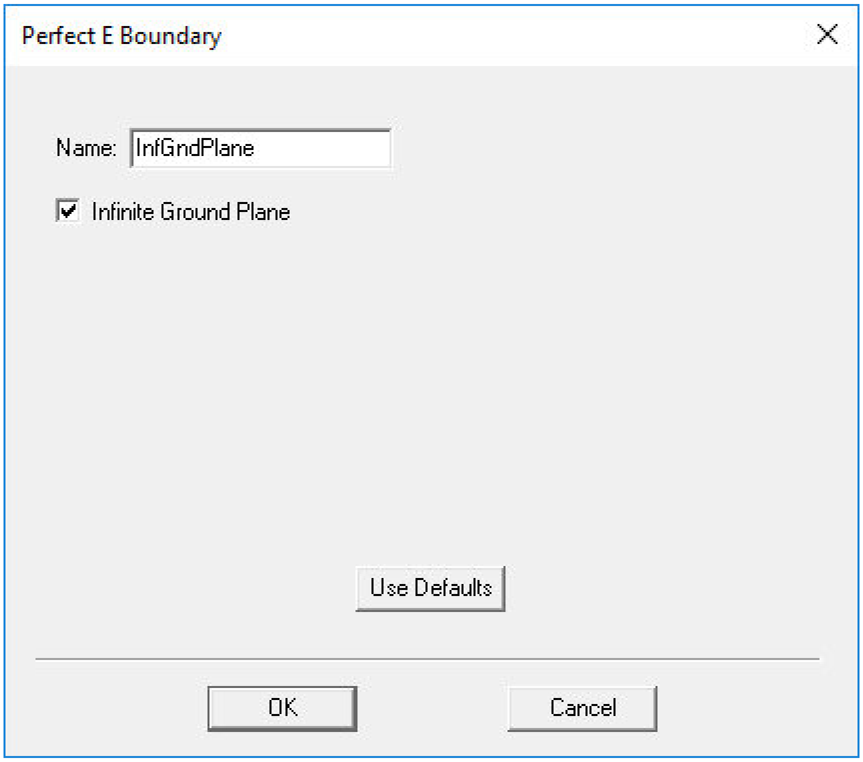



5. Continue keeping the rectangle selected. In the next topic, you will assign an infinite ground plane to it.

## Assign an Infinite Ground Plane

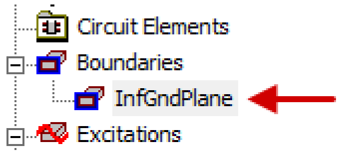
The next step is to assign a *Perfect E* boundary to the rectangle just drawn and declare it as an *Infinite Ground Plane*.

1. With *Rectangle1* still selected, right-click on **Boundaries** in the Project Manager and select **Assign** > **Perfect E** from the shortcut menu.
2. Make the following two changes in the *Perfect E Boundary* dialog box that appears:
   a. Change the **Name** to **InfGndPlane**.
   b. Select the **Infinite Ground Plane** option.

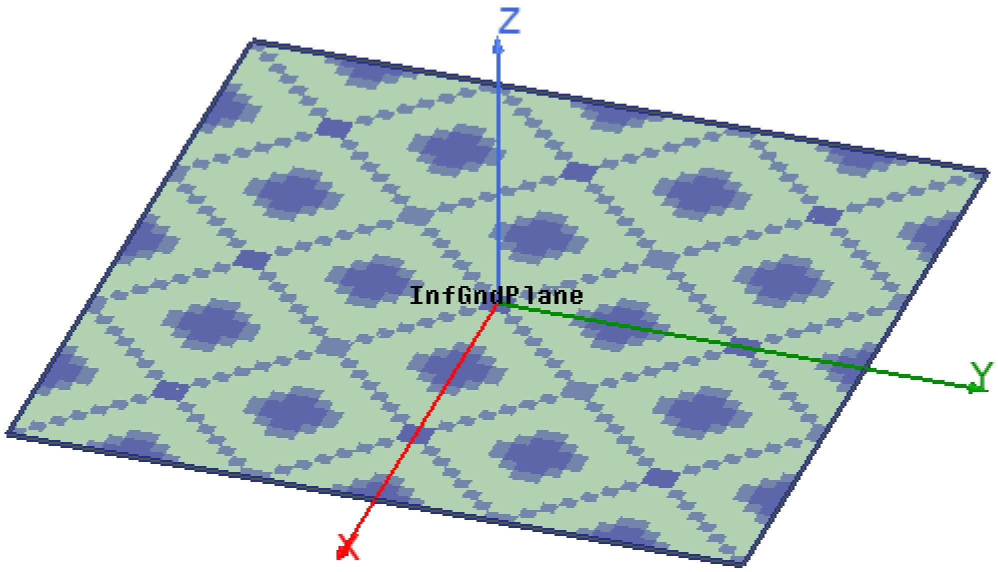



3. Click **OK**.

*InfGndPlane* appears under *Boundaries* in the Project Manager:




Also, while this boundary is selected, the following visualization appears on the model:

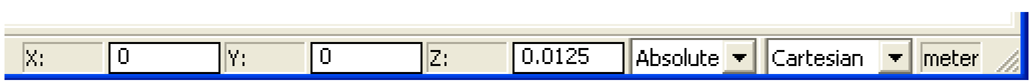



4. Click in the Modeler window's background area to clear the selection.

## Draw the Cone

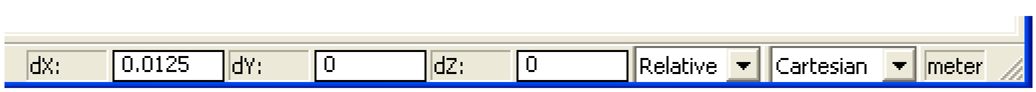
1. On the **Draw** ribbon tab, click **Draw cone**.

   Use the **X, Y,** and **Z** coordinate entry text boxes near the right end of the status bar to enter the center location as **0, 0, 0.0125** (the units should be meters) and then press **Enter**.

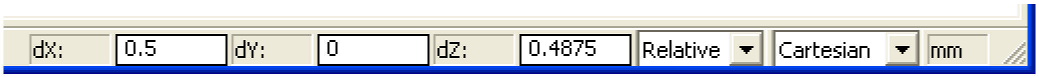



> **Important:**
>
> - Navigate between the text boxes using the **Tab** key.
> - When using the coordinate entry text boxes, be careful not to move your mouse, otherwise the cursor location will overwrite the values you've typed into the text boxes.

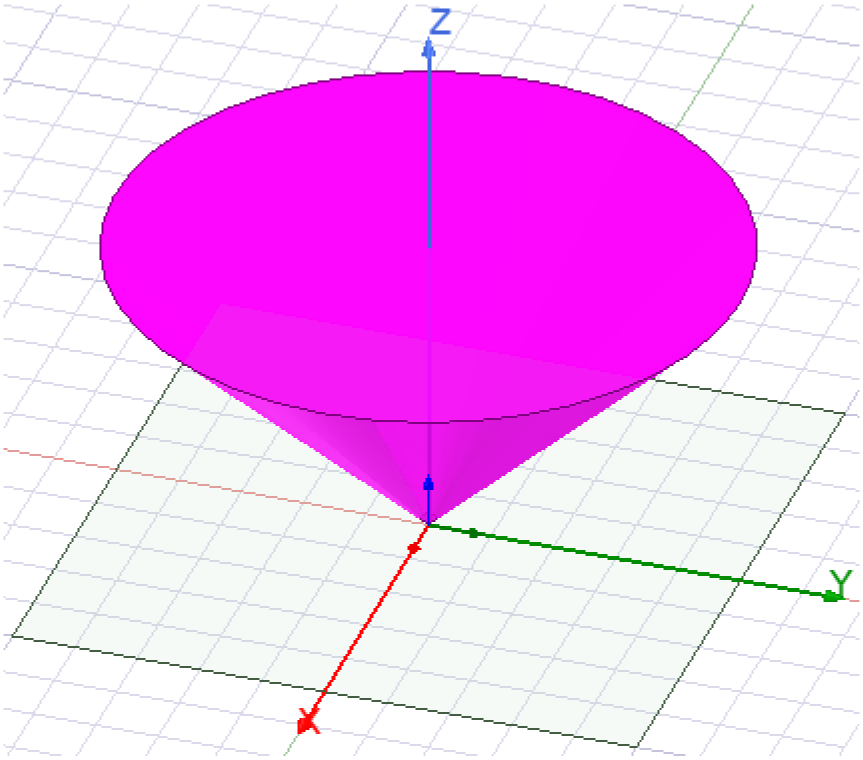
2. Use the **dX, dY,** and **dZ** coordinate entry text boxes to enter the lower radius as **0.0125, 0, 0** and then press **Enter**.




3. Use the **dX, dY,** and **dZ** coordinate entry text boxes to enter the upper radius as **0.5, 0, 0.4875** and then press **Enter**.




Keep the cone selected. It should appear as shown in the following image:




In the **Command** tab of the docked *Properties* window, you should see the following values:

| Name | Value | Unit | Evaluated Value |
|---|---|---|---|
| Command | CreateCone | | |
| Coordinate Sys... | Global | | |
| Center Position | 0 ,0 ,0.0125 | meter | 0meter , 0meter... |
| Axis | Z | | |
| Upper Radius | 0.5 | meter | 0.5meter |
| Lower Radius | 0.0125 | meter | 0.0125meter |
| Height | 0.4875 | meter | 0.4875meter |

4. If any of your *CreateCone* values do not match the preceding figure, edit them to correct the discrepancy.
5. Click in the Modeler window's background area to clear the selection.

# Draw the Port Rectangle

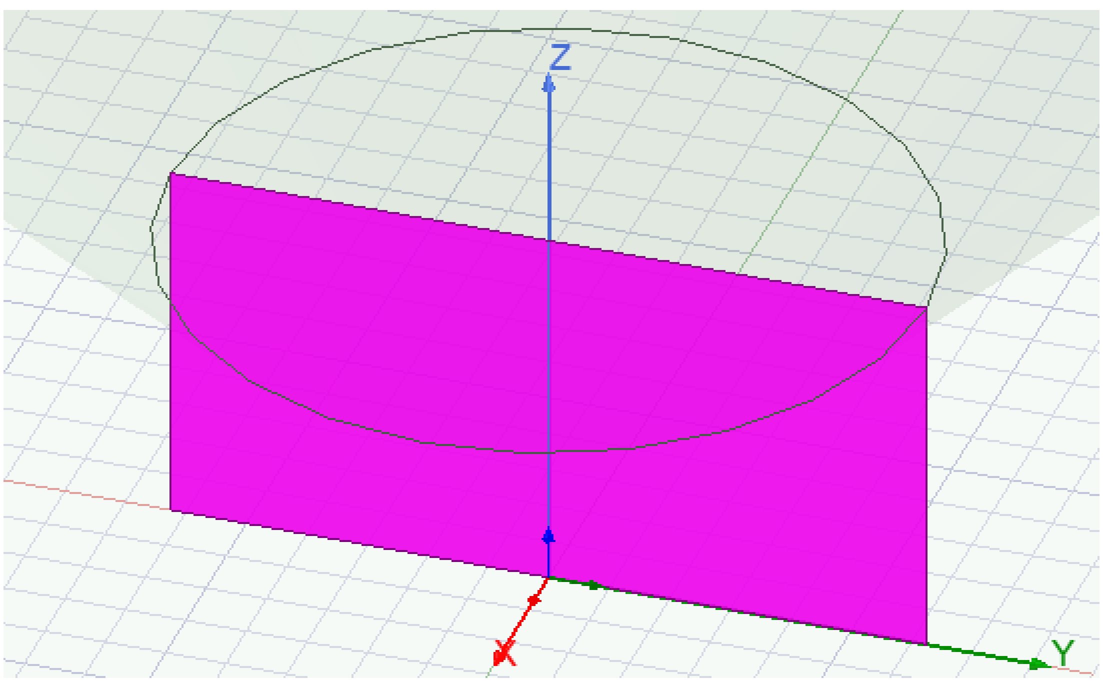
1. On the **Draw** ribbon tab, click **Draw rectangle**.
2. Click on two different grid points to create an arbitrary rectangle.
3. In the **Command** tab of the docked *Properties* window, edit the values to match the following figure:

| Name | Value | Unit | Evaluated Value |
|---|---|---|---|
| Command | CreateRectangle | | |
| Coordinate Sys... | Global | | |
| Position | 0 ,-0.0125 ,0.0125 | meter | 0meter , -0.012... |
| Axis | X | | |
| XSize | 0.025 | meter | 0.025meter |
| YSize | -0.0125 | meter | -0.0125meter |

Keep the rectangle selected.

4. On the **Draw** ribbon tab, click **Fit Selected**.

The rectangle should be visible under the cone and should lie on the global YZ plane. (The *X* value specified for the *Axis* is the direction normal to the plane of the objects.)

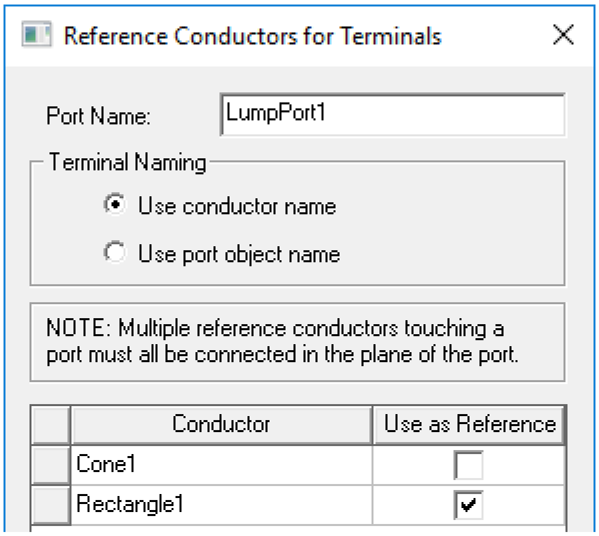



5. Continue keeping *Rectangle2* selected. In the next topic, you will add a lumped port excitation to it.

# Assign Lumped Port Excitation

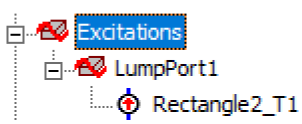
Next, assign a lumped port excitation to the rectangle, as follows:

1. With *Rectangle2* still selected, right-click in the Modeler window and choose **Assign Excitation** > **Port** > **Terminal Lumped Port** from the short cut menu.
2. In the *Reference Conductors for Terminals* dialog box that appears, make the following changes:
   a. Change the **Port Name** to **LumpPort1**.
   b. Ensure that the **Use conductor name** option is selected.
   c. Select the **Use as Reference** option for the **Rectangle1** *Conductor*.

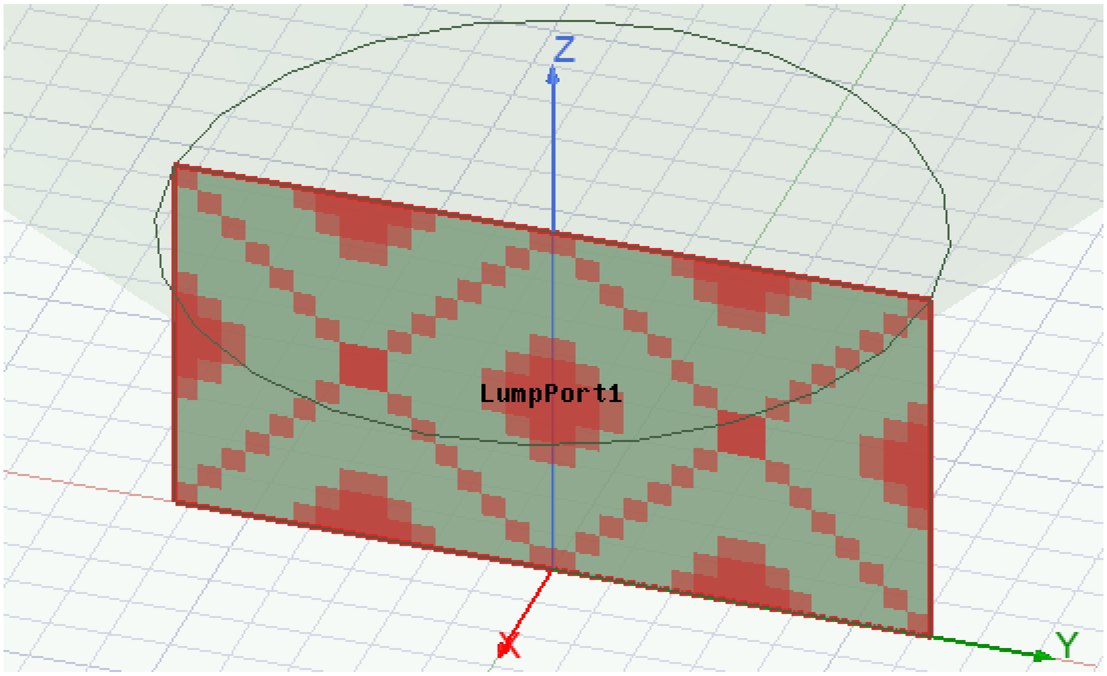



3. Click **OK**.

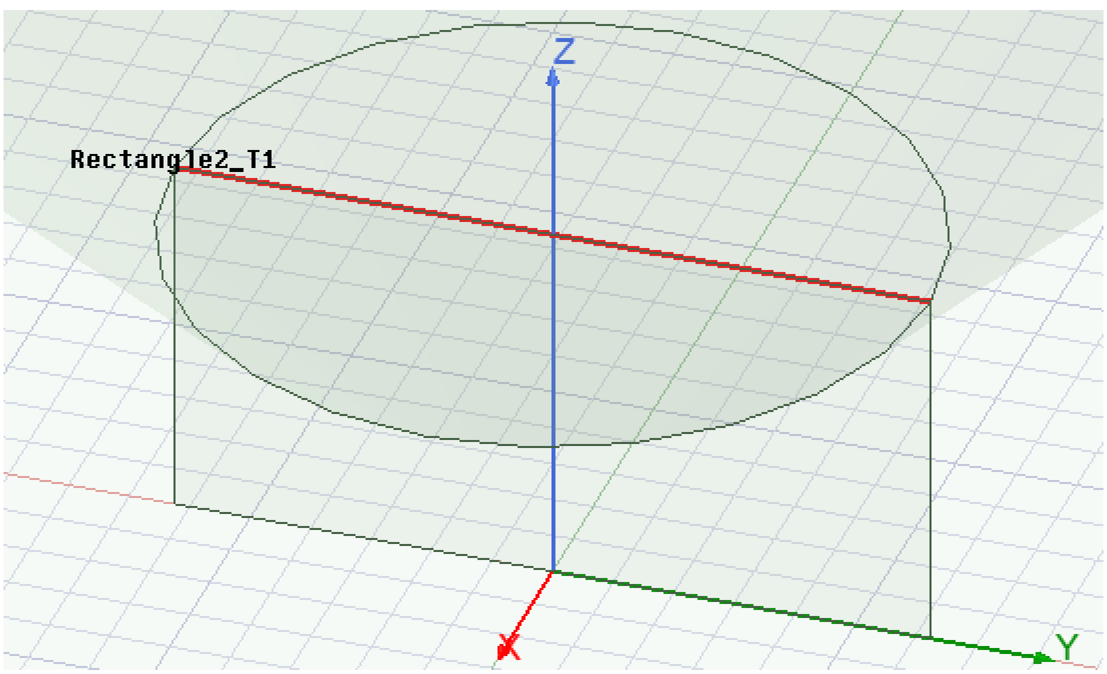
   The lumped port and associated terminal assignment appear under *Excitations* in the Project Manager:




4. In the Project Manager, select **LumpPort1** to see it visualized on the model:




5. Select **Rectangle2_T1** under *LumpPort1* to see the terminal visualized:



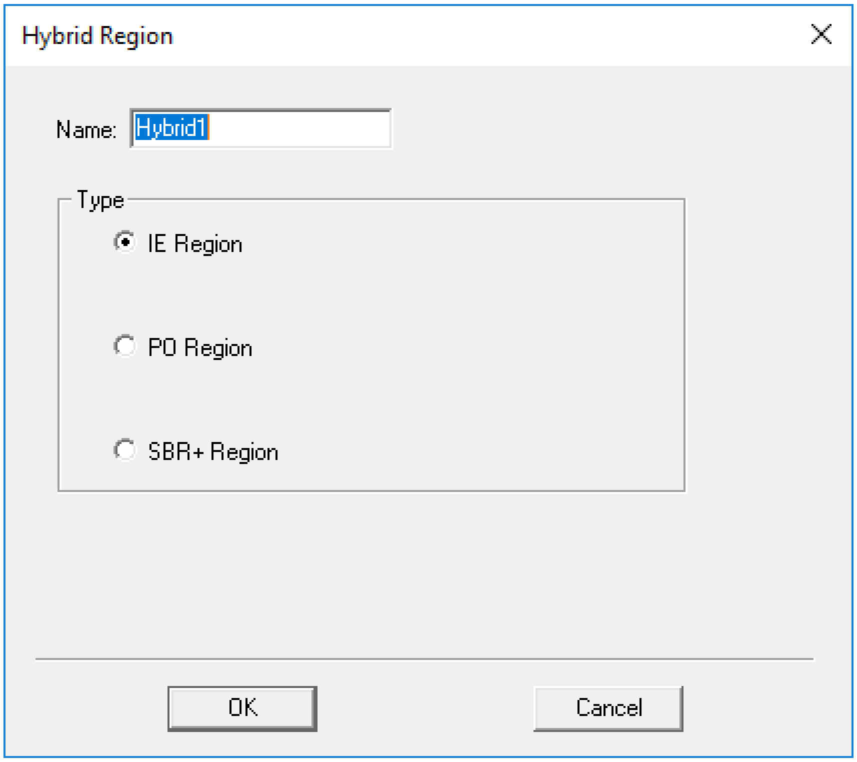

6. Press **Ctrl+D** to fit the view and click in the Modeler window background to clear the selection.

## Assign an IE Region

Next, assign a hybrid region of the type *IE Region* to all objects in the model.

1. Click in the Modeler window and press **O** to ensure that you are in the *Object selection* mode.
2. Press **Ctrl+A** to select all objects.
3. Right-click **Hybrid Regions** in the Project Manager and choose **Assign** > **IE Region** from the shortcut menu.

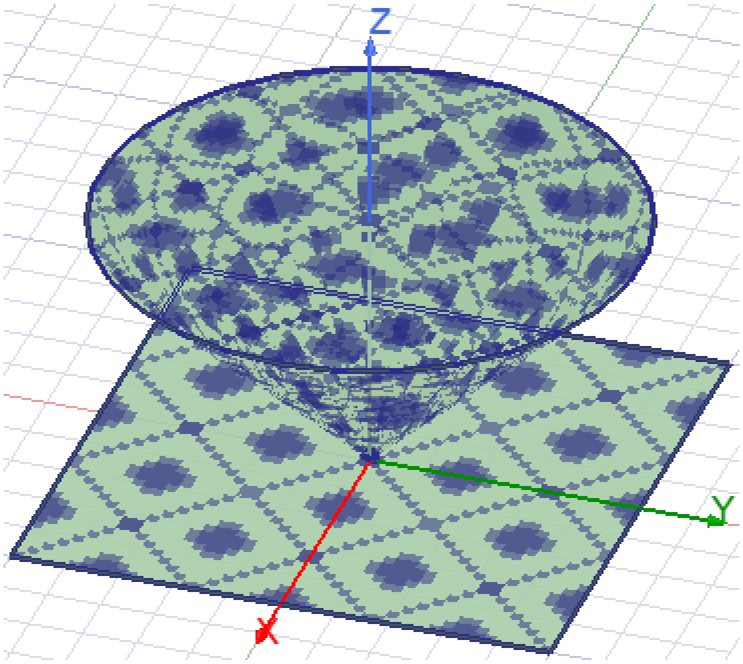
   The *Hybrid Region* dialog box appears:




4. Click **OK** to create the *IE Region* using the default type and name.

   *Hybrid1* appears under *Hybrid Regions* in the Project Manager:


   Also, the *IE Region* is visualized on the model when selected:




5. Clear the selection.

# 3 - Add Solution Setup and Analyze

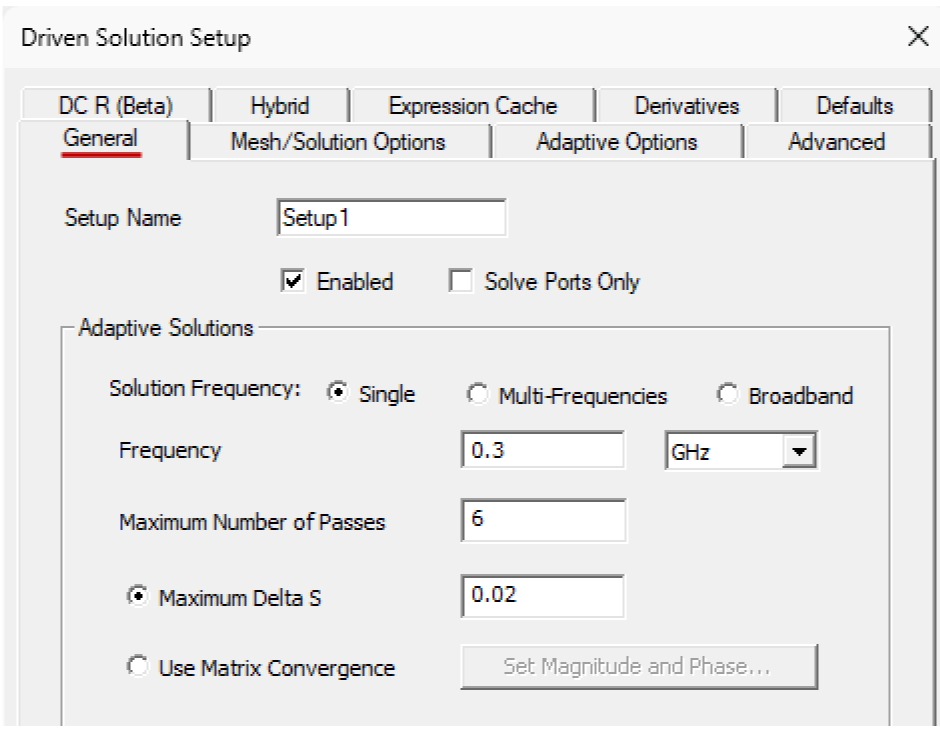
The last modeling step is to add the solution setup. Then, you can run the analysis.

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

   The *Driven Solution Setup* dialog box appears.

2. In the **General** tab, change the **Frequency** to **0.3 GHz**.

   Leave the remaining settings at their default values:




3. Click **OK**.
4. Click **Cancel** to dismiss the *Edit Frequency Sweep* dialog box that appears. You do not need to define a frequency sweep for this exercise.

   The Monocone Antenna setup is complete.

5. **Save** the project.

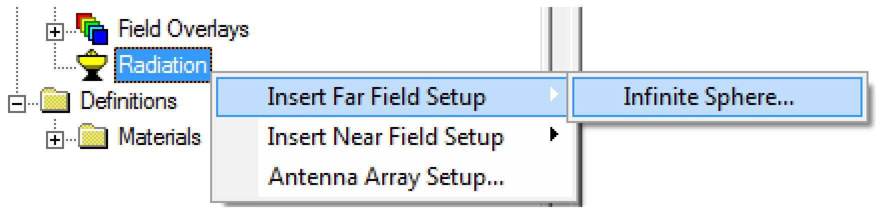
6. On the **Simulation** ribbon tab, click **Analyze All**.

   The solving process will take approximately five minutes to complete, depending on your computer system.

# 4 - Plot the Radiation Pattern

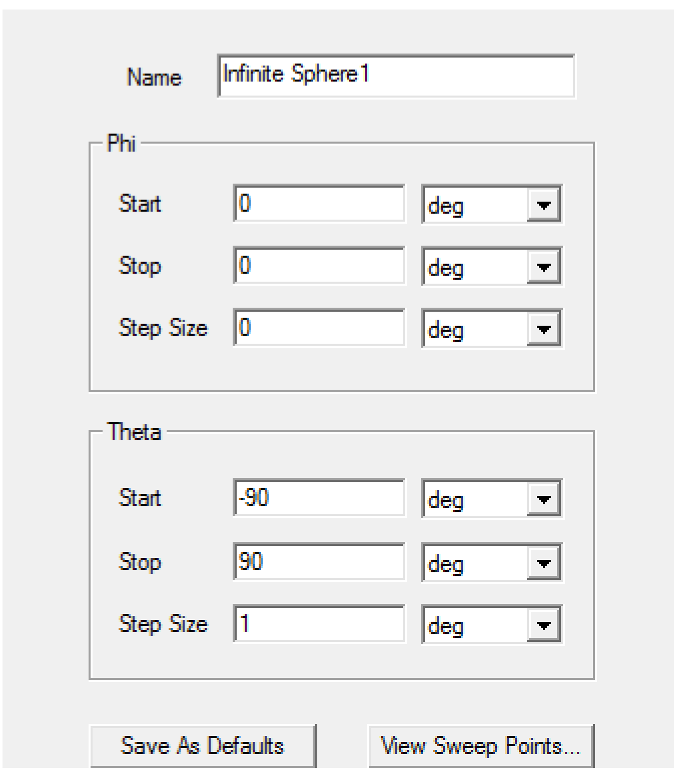
To plot the radiation you first must create a far field setup (the same as required for HFSS designs). Create a far field setup with a single Phi (ϕ) value of 0°.

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



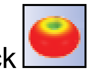
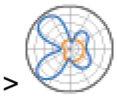

The *Far Field Radiation Sphere Setup* dialog box opens.

2. Enter **0** for all three *Phi* values (**Start, Stop,** and **Step Size**).
3. Enter the following settings for *Theta*:
   a. *Start* = **-90 deg**
   b. *Stop* = **90 deg**
   c. *Step Size* = **1 deg**



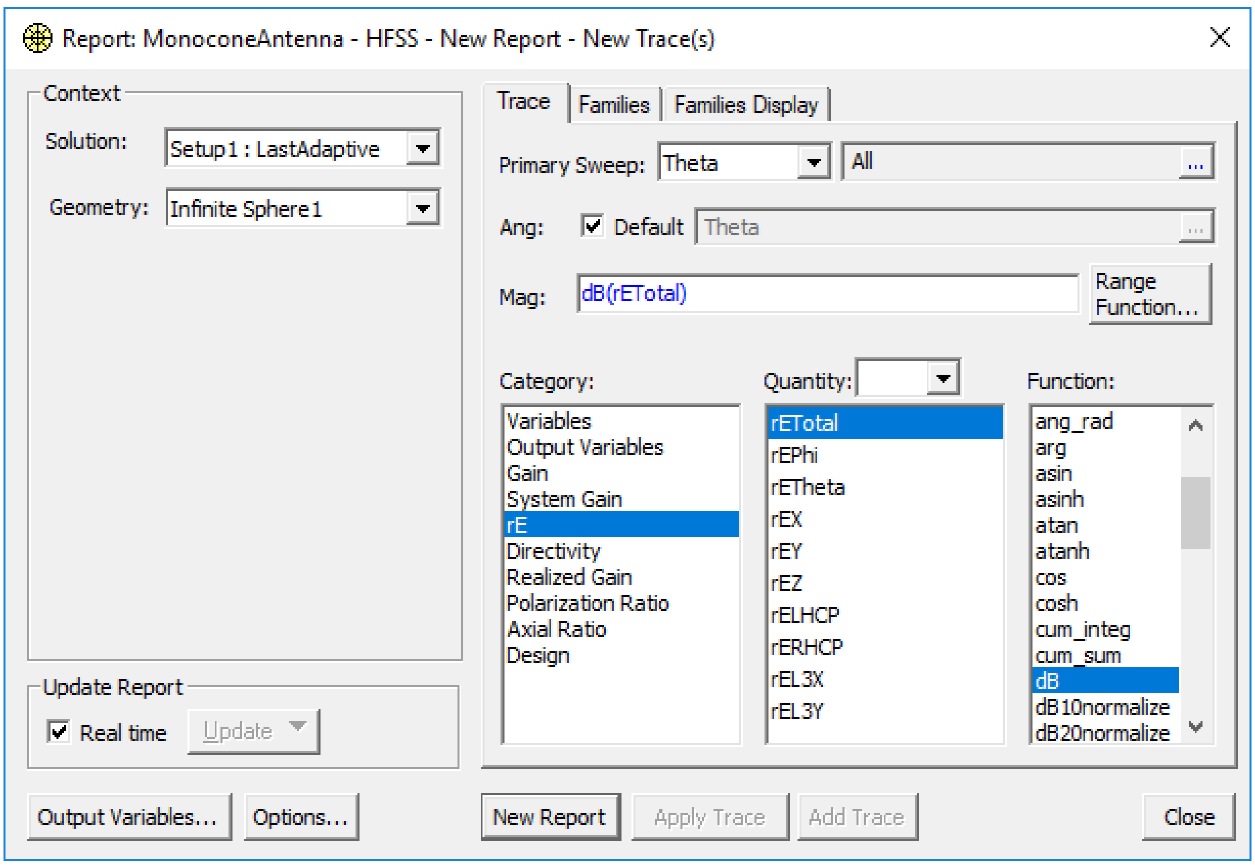
4. Click **OK**.

   *Infinite Sphere1* appears under *Radiation* in the Project Manager.

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

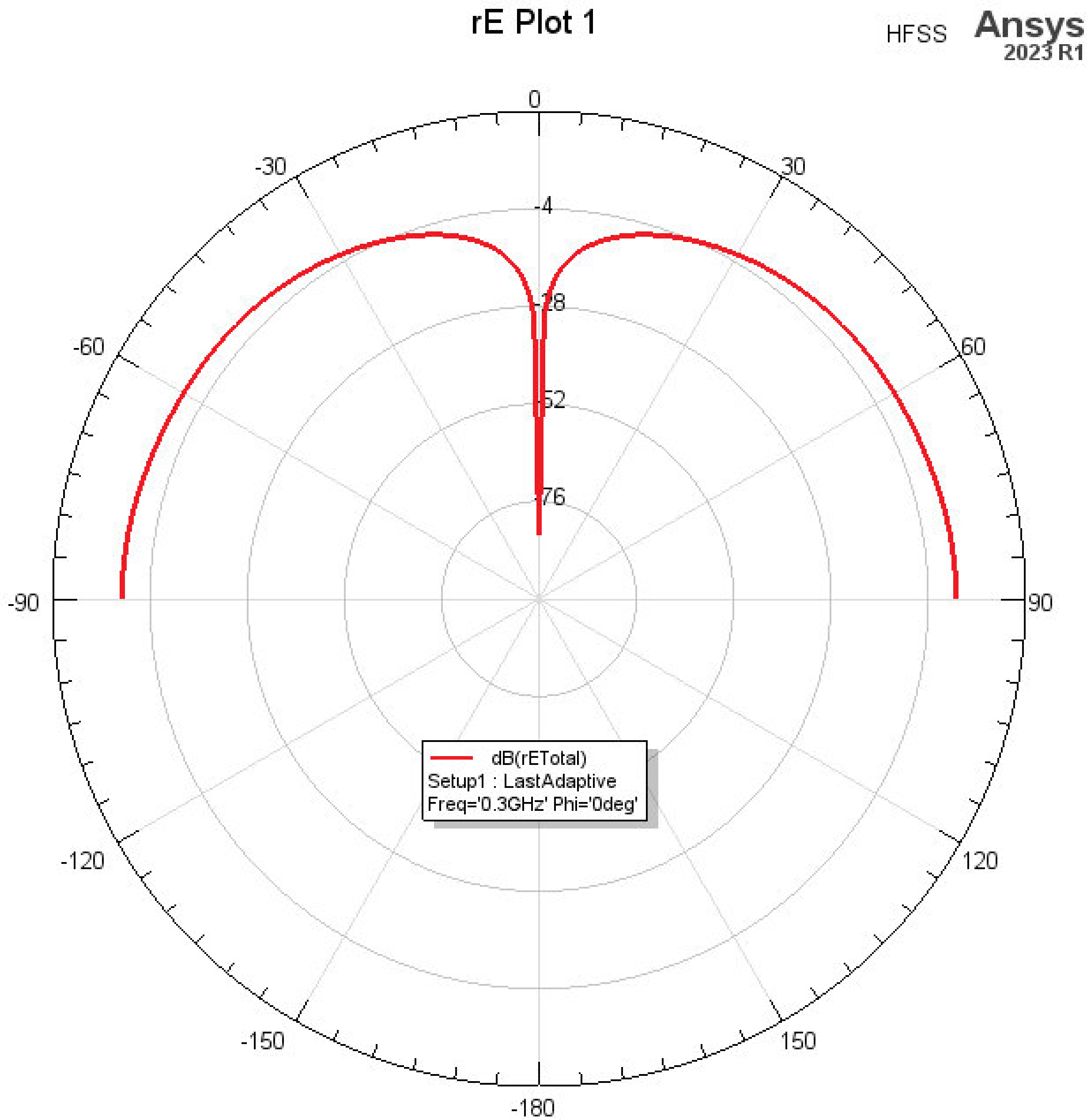
   The *Report* dialog box appears.

6. Specify the following settings:
   a. For *Category*: **rE**
   b. For *Quantity*: **rETotal**
   c. For *Function*: **dB**.




7. Click **New Report** and then click **Close**.
8. Click and drag the *Curve Info* legend to position it as desired.

   The resulting plot should resemble the following figure:


rE Plot 1
HFSS
Ansys
2023 R1
0
-30
30
-60
60
-90
90
-120
120
-150
150
-180
-4
-28
-52
-76
dB(rETotal)
Setup1 : LastAdaptive
Freq='0.3GHz' Phi='0deg'

> **Note:**
>
> If you click the circular axis, its settings are displayed in the docked *Properties* window. In the *Axis* tab, you can change the major and minor divisions, axis line width and color, and more. Click one of the radial or circumferential grid lines to see a different set of settings. In the *Grid* tab, you can change the Rho (radial) minimum scale and maximum scale values. Select the trace to customize its line width, color, or other settings in the *Attribute* tab. Finally, click the title to change its font style, size, or color in the *Header* tab.

9.  **Save** your project.

# 5 - Optionally, Restore Current View Orientations

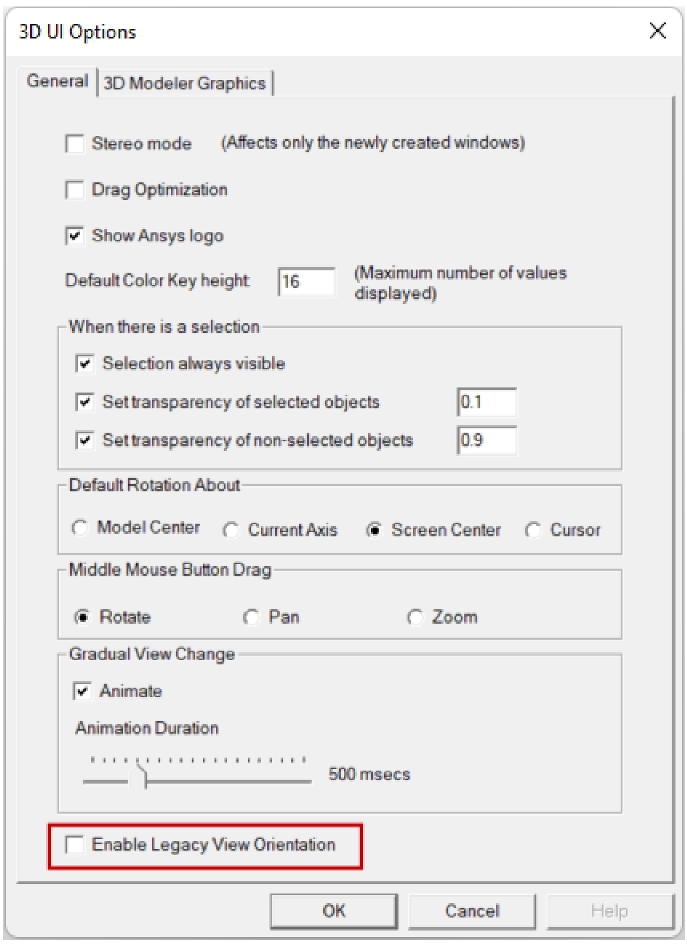
You have completed this getting started guide.

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.