



POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

© 2025 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Getting Started with HFSS™: Keyboard Shortcuts



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

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

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 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

Table of Contents	Contents-1
1 - Introduction	1-1
Using Keyboard Shortcuts	1-1
Sample Model: Slot Coupled Patch Antenna	1-1
2 - Set Up the Project	2-1
Launch Ansys Electronics Desktop (EDT)	2-1
Assign Custom Keyboard Shortcuts	2-2
Set General Options	2-7
Insert HFSS Design	2-9
Enable Legacy View Orientations	2-10
Set Model Units (cm)	2-11
Verify Solution Type (Terminal)	2-12
3 - Create the 3D Model	3-1
Set Default Material	3-1
Create Substrate	3-2
Create the Feed	3-3
Assign a Perfect E Boundary to the Feed	3-4
Create Ground	3-4
Draw Slot (Cut-Out)	3-5
Complete the Ground	3-6
Assign Perfect E Boundary to the Ground	3-7
Create Patch	3-8
Assign a Perfect E Boundary to the Patch	3-9
Create Air Body	3-10
Assign Radiation Boundary	3-11
Add Length Based Mesh Operation to Radiation Boundary	3-12
Create Radiation Setup	3-13
Create Source	3-14

Assign Excitation	3-15
4 - Analyze and Evaluate Results	4-1
Create Analysis Setup	4-1
Add Frequency Sweep	4-2
Validate and Analyze	4-2
Review Convergence History	4-3
Create Terminal S-Parameter Plot	4-5
Create and Animate E-Field Overlay	4-6
Create Gain Plot	4-8
5 - Examples	5-1
View Navigation Shortcuts	5-1
Geometry Creation and Selection Shortcuts	5-4
Plot Shortcuts	5-7
6 - Optionally, Restore Current View Orientations	6-1
A - Appendix: Default Keyboard Shortcuts	A-1

1 - Introduction

This chapter contains the following topics:

- Using Keyboard Shortcuts
- Sample Model: Slot Coupled Patch Antenna

Using Keyboard Shortcuts

For those who prefer communicating instructions using a keyboard more often than a mouse, this *Getting Started Guide* presents two different techniques for working with HFSS:

- Shortcut keys (either predefined or user-defined)
- Keystroke sequences for menu bar navigation to trigger desired actions.

Keyboard shortcuts can be a convenient means of performing common and repetitive tasks without having to navigate menus or command ribbons. In addition to the default keyboard assignments, you can customize the shortcuts. There is an extensive list of commands and operations available within Ansys Electronics Desktop, only a limited number of which have predefined shortcuts. So, you can add key assignments, or you can change predefined ones, to suit your preferences.

While menu navigation keystroke sequences are less convenient than keyboard shortcuts, they are useful when writing or recording scripts for automating modeling tasks, and they can be convenient when the desired target is located relatively high on a menu's main level. They also do not require that you define any shortcut key; their support within the user interface is intrinsic.

At the end of the guide, there are *Examples* showing how to use the keyboard shortcuts and an *Appendix* that contains a list of predefined shortcut key assignments.

Sample Model: Slot Coupled Patch Antenna

We assume you are familiar with the workings of HFSS within the Ansys Electronics Desktop software, and this is not your first project.

This example is intended to show you how to create, simulate, and analyze a probe fed, slot coupled patch antenna using the HFSS Design Environment, with an emphasis on using keyboard shortcuts.

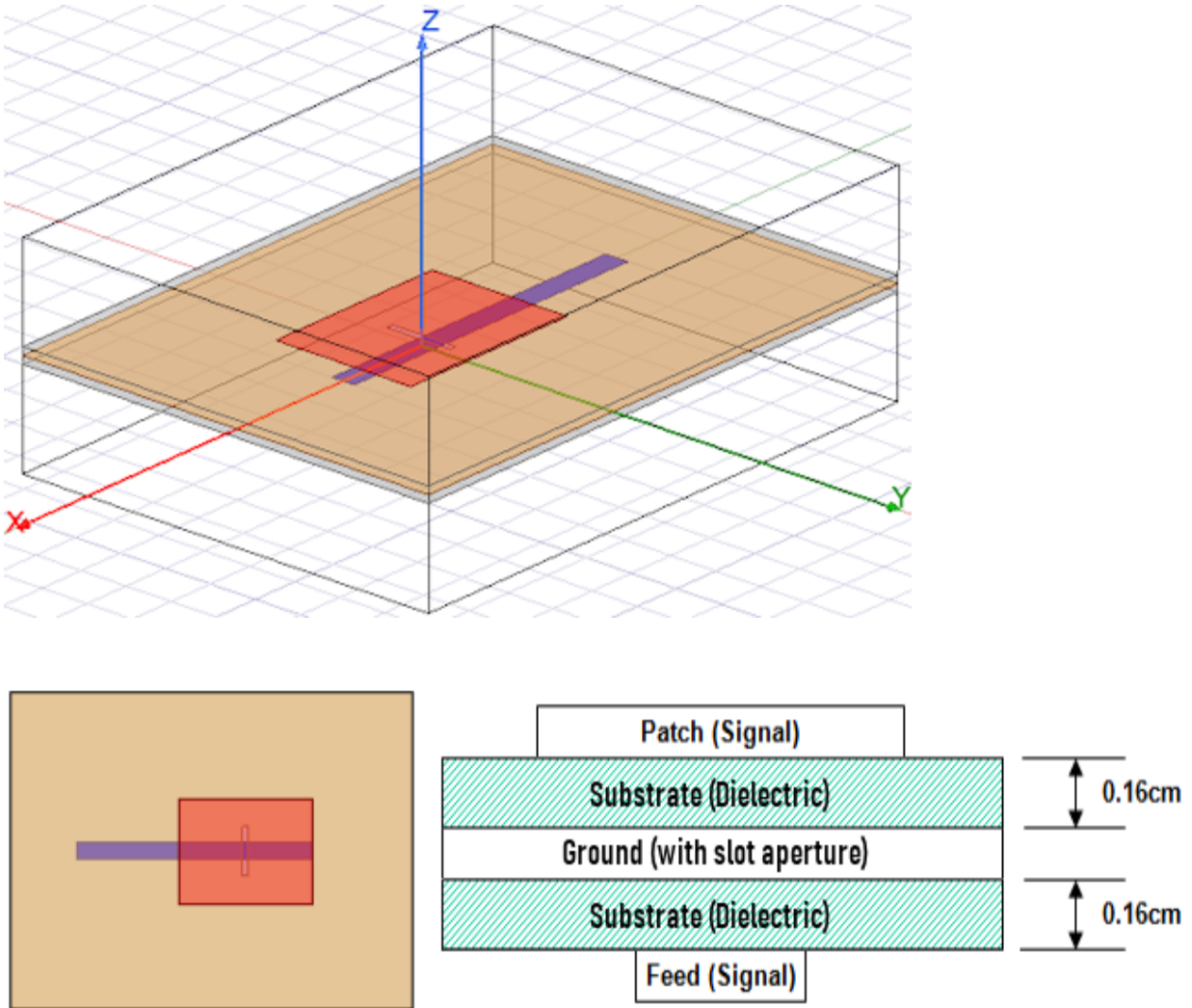


Figure 1-1: Slot Coupled Patch Antenna – Model Views and Layer Diagram


2 - Set Up the Project

This chapter contains the following topics:

- Launch Ansys Electronics Desktop (EDT)
- Assign Custom Keyboard Shortcuts
- Set General Options
- Insert an HFSS Design
- Enable Legacy View Orientations
- Set Model Units (cm)
- Verify Solution Type (Terminal)

Launch Ansys Electronics Desktop (EDT)



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

Ansys Electronics Desktop opens.

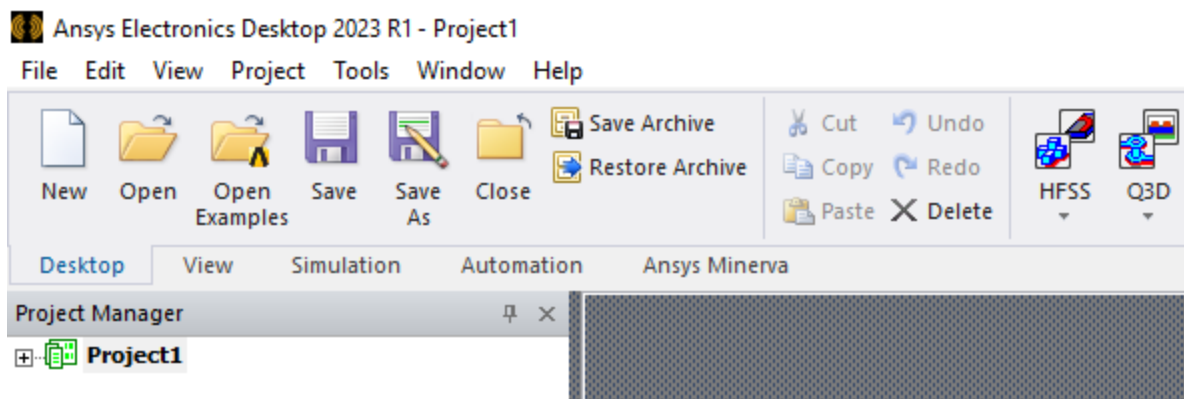



Figure 2-1: Ansys EDT Application Launched

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 (press **Alt** then **V**) and select the **Project Manager** option.
- After you assign the custom keyboard shortcuts detailed in the [next topic](#), you can use the hotkey combination **Ctrl+N** to create a new project folder.

Assign Custom Keyboard Shortcuts

In Ansys Electronics Desktop, you can configure the type of keyboard short-cuts that you want. Custom keyboard shortcuts can help you facilitate the process of setting up your design, simulating it, and evaluating the results.

The hot keys can be defined in the *Keyboard Shortcuts* dialog box. The following instructions walk you through defining the first two shortcuts that you will use in drawing, setting up, analyzing, and evaluating the results of the patch antenna model. Afterward, a table lists additional shortcuts, which you must define on your own.

Before altering the default or user-customized shortcuts, it's a good idea to make a backup copy of the current configuration.

1. Press **Alt**, then **T**, and then **K**. (The *Alt* key accesses menu bar items. Notice how a letter in each menu and command name is underlined after pressing the *Alt* key, indicating the additional keystroke that will open a menu or execute a command.)
2. In the *Keyboard Shortcuts* dialog box that appears, click **Save**. Then:
 - a. Navigate to a folder of your choice where you want to store the current shortcut configuration file.
 - b. Specify a suitable **File name**, such as *DefaultShortcuts*.
 - c. Click **Save**.

After completing this getting started guide, you can restore the prior keyboard shortcuts configuration, if desired.

3. Select **Insert** and **Insert HFSS Design** in the **Categories** and **Commands** lists, respectively.
4. Click inside the **Press new shortcut key** box, and then press the desired key combination (**Shift+1**).
5. Click **Assign** to complete the shortcut definition.

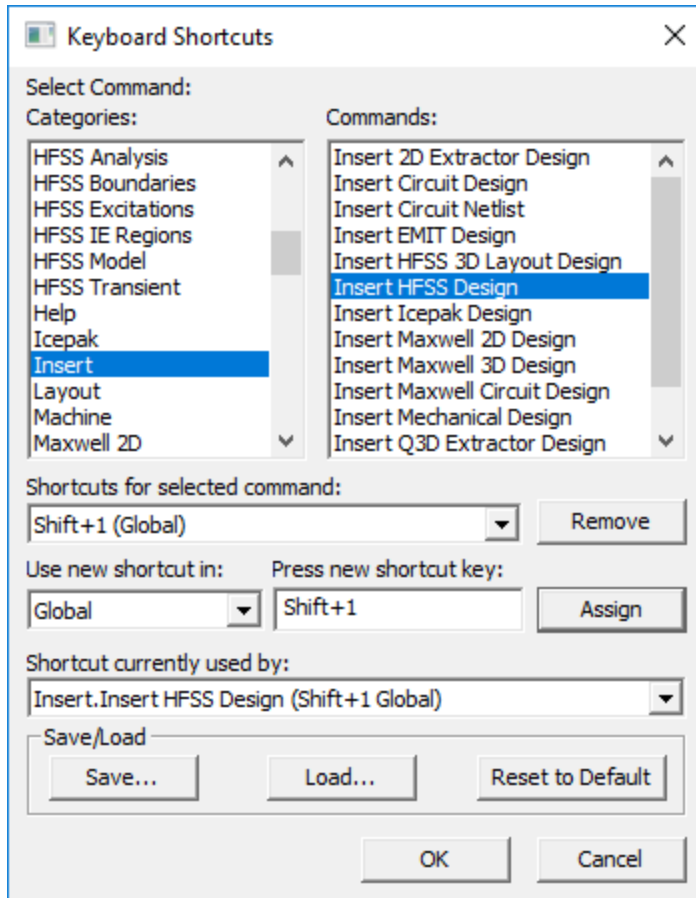


Figure 2-2: Assigning *Insert HFSS Design* Shortcut

Note:

- The **Shortcut currently used by** drop down menu lists the shortcut if it is already in use. To remove a specified shortcut key, click **Remove** and reassign a shortcut key per your choice.
- To maintain the states and functions of existing hotkeys in the program (such as **Ctrl +D** to fit the view), create your shortcuts carefully so that there is no conflict with the predefined ones. For example, defining **Ctrl+C** (Edit > Copy) or **F6** (wireframe view of geometry) as a shortcut for **Analyze All** would not be good choices.

Keep the *Keyboard Shortcuts* dialog box open as you continue to define additional hotkeys.

6. Select **3D Modeler** and **Units** under **Categories** and **Commands**, respectively.
7. Click in the **Press new shortcut key** box, and then press the desired shortcut key combination (**Shift+2**).

8. Click **Assign**.

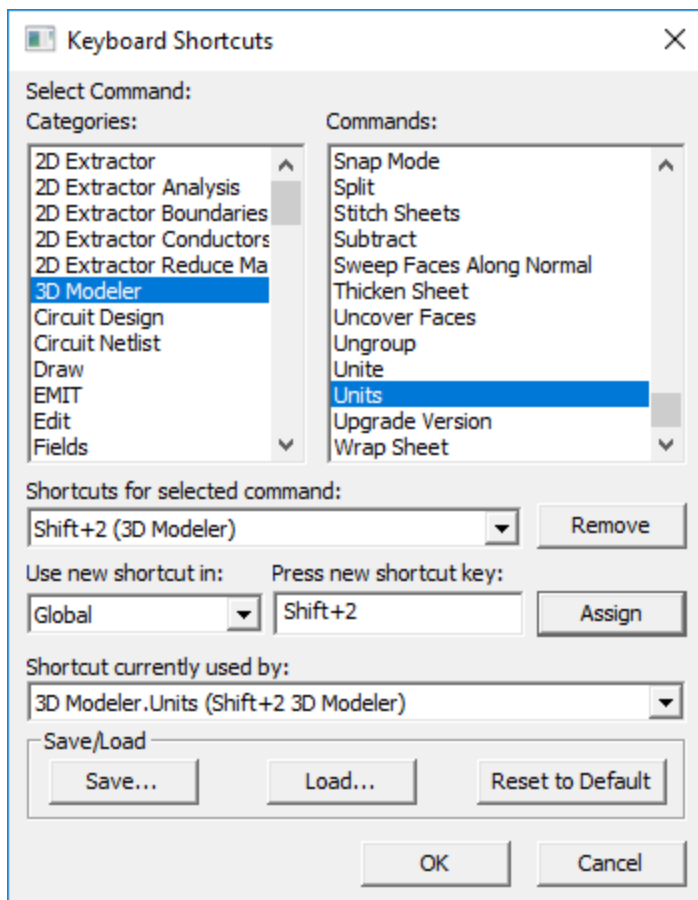


Figure 2-3: Assigning *Units* Shortcut

9. The following table lists all of the keyboard shortcuts that will be referenced within this guide. You have already defined the first two. Now, define the remaining keyboard shortcuts in Table 1 and keep the dialog box open when you're done.

Note:

Certain commands are applicable in two or more different contexts. For example, **Draw > Rectangle** is applicable to the *Schematic Editor*, *Layout Editor*, and *3D Modeler*. Only one context is listed at a time under **Shortcuts for selected command** and **Shortcut currently used by**. If you see a context different from what you expect (such as *2D Extractor* or *Schematic Editor*, this does not necessarily indicate a conflict with the chosen shortcut key combination. To assure yourself that there is no conflicting assignment, access the two drop-down lists to see all of the places where the command is applicable. *3D Modeler*, *HFSS*, or another context consistent with this HFSS getting started guide should be included in the list, and there is no need for concern.

Table 1: Custom Keyboard Shortcuts

Hot Key	Function
Shift+1	Insert > Insert HFSS Design
Shift+2	3D Modeler > Units
Alt+S	3D Modeler > Subtract
Alt+B	Draw > Box
Alt+R	Draw > Rectangle
Alt+N	Edit > Select By Name
Shift+A	Fields > Animate
Alt+E	Fields > Mag_E
Shift+3	HFSS > Solution Type
Alt+7	HFSS > Validation Check
Alt+Z	HFSS > Analyze All
Shift+E	HFSS Boundaries > PerfectE
Shift+X	HFSS Boundaries > Radiation
Shift+4	HFSS Excitations > Lumped Port
Shift+5	HFSS Analysis > Add Solution Setup
Shift+6	HFSS Analysis > Add Frequency Sweep
Shift+L	Mesh > Assign on Selection Length Based
Insert	Radiated Fields > Infinite Sphere
Alt+F	Results > Far Fields Radiation Pattern
Alt+D	Results > Solution Data

10. In the *Keyboard Shortcuts* dialog box, click **Save**. Then, navigate to a folder of your choice, specify a suitable **File name** (such as *KeyboardShortcutsGuide*), and click **Save** to place the configuration file on your drive.

After completing this getting started guide, you may want to **Load** the default configuration that you saved in step 2 to restore it. Similarly, you can restore the version created for this exercise at any time. In this manner, you can create and switch easily between multiple shortcut configurations to suit different model types, workflows, or users.

Note:

Once you've defined the shortcuts, they appear within the program menus. For instance, **Shift+3** appears next to the **Solution Type** command in the **HFSS** menu:

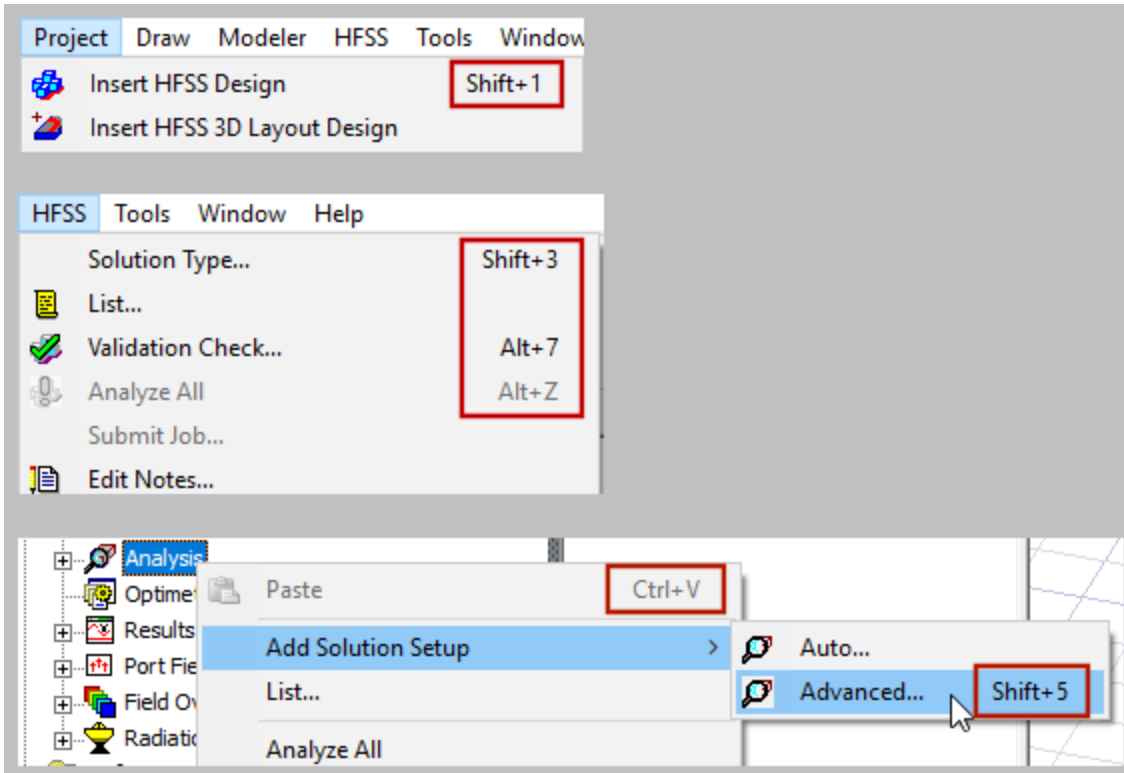


Figure 2-4: Menus with Shortcuts Indicated

Set General Options

Verify the options under the **Tools** menu as follows:

1. Press, in sequence, **Alt**, **T**, **O**, **G** (do not hold down the *Alt* key).

The *Options* dialog box appears.

2. Press **H** to jump to the *HFSS* options group.
3. Press the **right cursor key** to expand the *HFSS* branch and press **B** to jump to the **Boundary Assignment** subgroup:

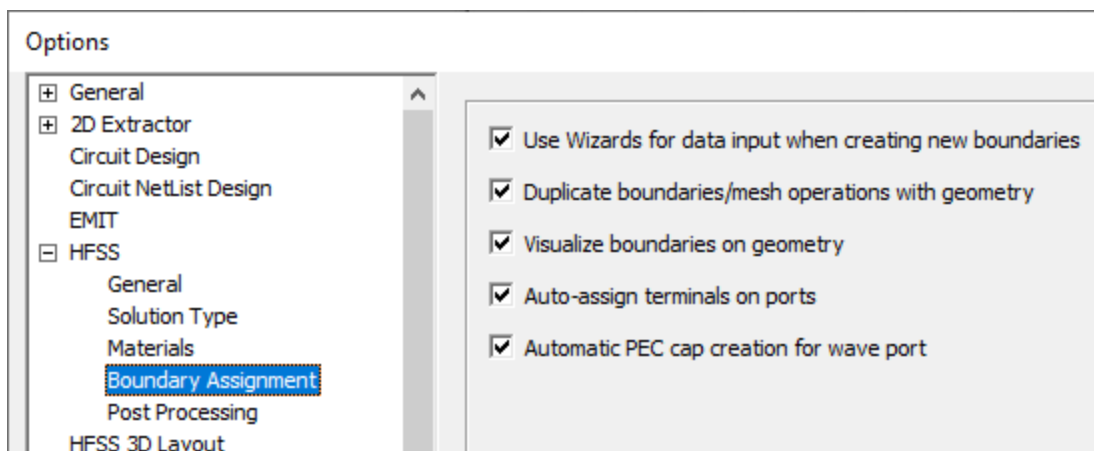


Figure 2-5: HFSS Boundary Assignment Options

4. Ensure that all options in the *Boundary Assignment* subgroup are checked.
5. Press **3** to jump to the *3D Modeler* options group and press the **right cursor key** to expand it.
6. Press **D** to jump to the **Drawing** subgroup. Then:
 - a. Verify that the **Automatically cover closed polylines** option is selected.
 - b. Select the **Edit properties of new primitives** option.

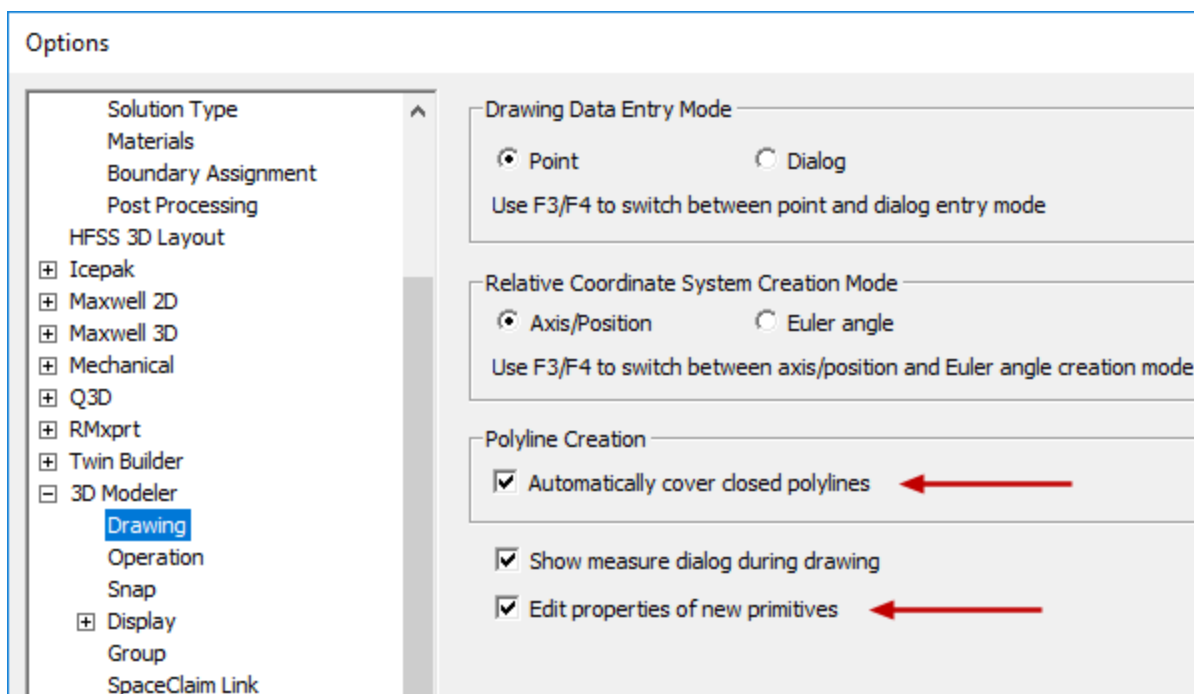


Figure 2-6: 3D Modeler Drawing Options

7. Click **OK**.

Insert HFSS Design

Skip this section if the project folder already includes an HFSS design. A design of a specified type can be automatically inserted into new projects if configured as such in the *Options* dialog box (under *General > Desktop Configuration > New Project Options*).

1. Press **Shift+1** to insert an HFSS design.

Note:

Alternatively, you can sequentially press **Alt, P, Enter** to access the command through keyboard navigation of the menu bar.

2. In the Project Manager, expand the **HFSSDesignx** branch:

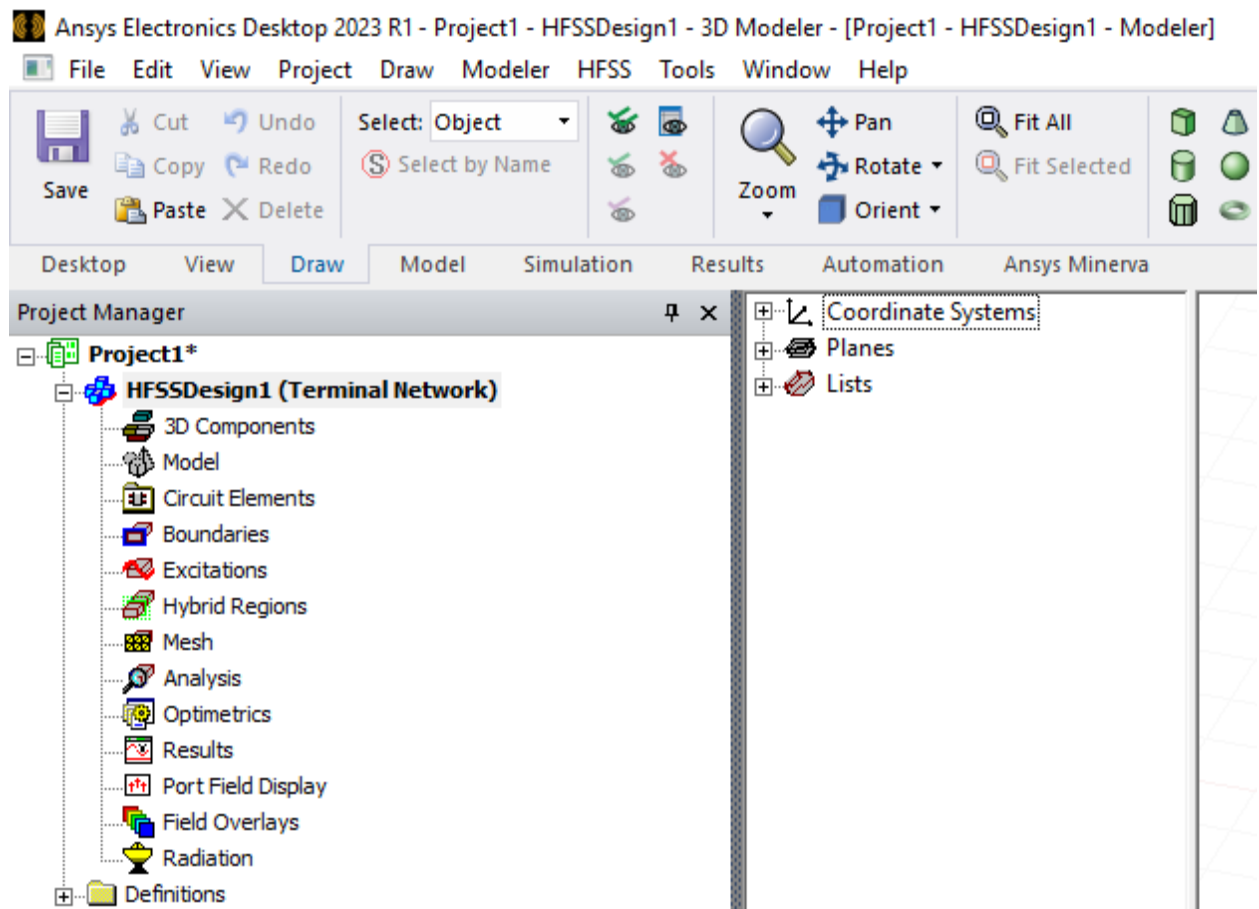


Figure 2-7: HFSS DesignX Included

3. Press **Ctrl+S**.

The **Save As** dialog box appears.

4. Navigate to your preferred project storage folder and specify a **File name** (such as *KeyboardShortcuts*).
5. Click **Save**.

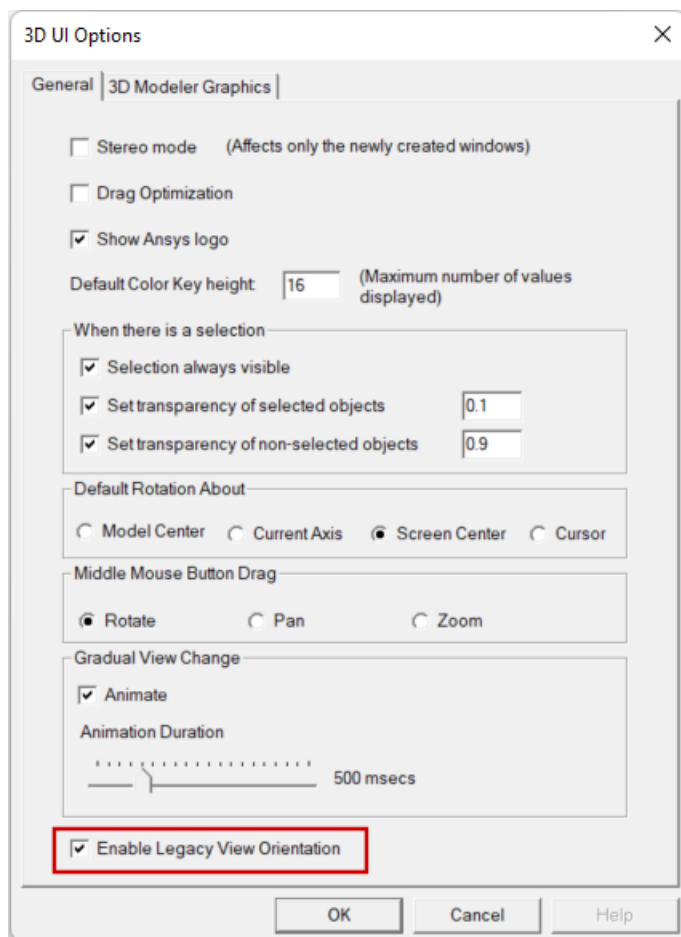
Enable Legacy View Orientations

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)

Commands to set the units are as follows:

1. Press **Shift+2**.

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

2. Type **cm** and press **Enter**.

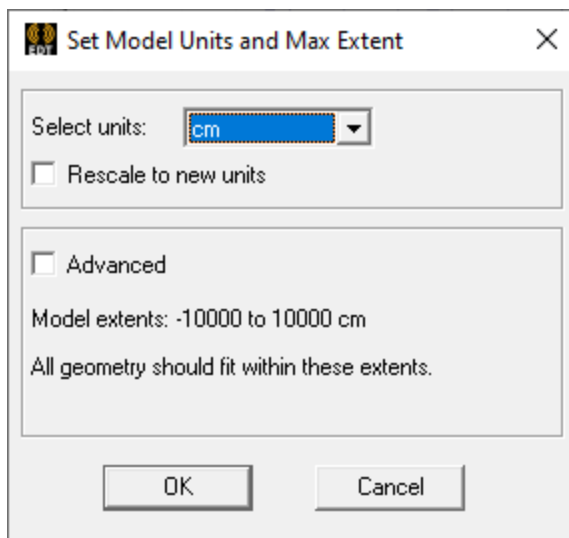


Figure 2-8: Specifying Model Units (cm)

Verify Solution Type (Terminal)

1. Press **Shift+3** to access the *Solution Type* dialog box.

Alternatively, navigate the menu bar by sequentially pressing **Alt, H, Enter** to access the *HFSS > Solution Type* command.

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

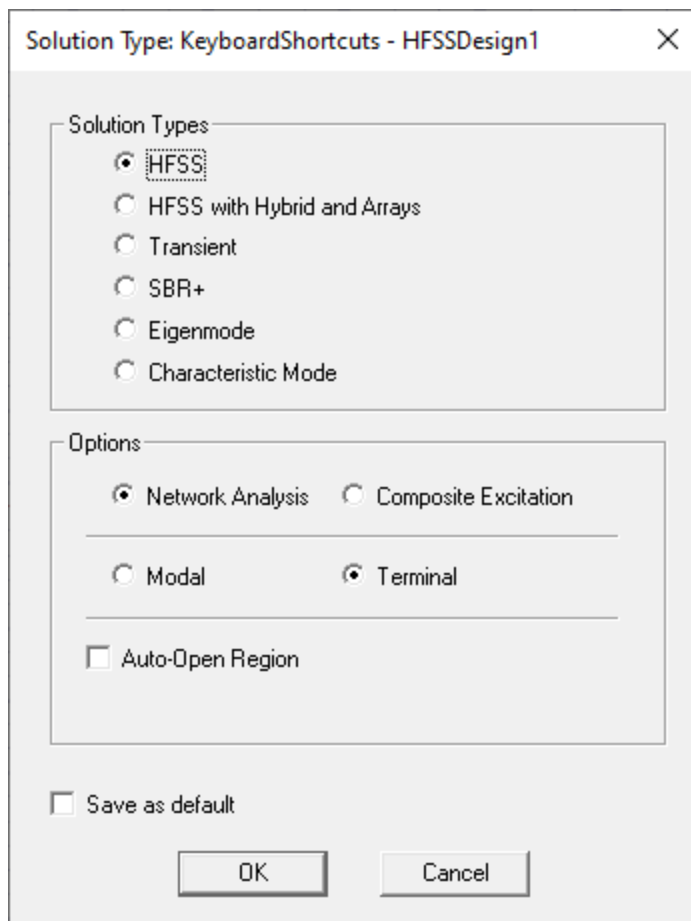


Figure 2-9: Solution Type Dialog Box

The **HFSS** solution type and the **Network Analysis** and **Terminal** options are the default settings for a clean installation of the program. However, users can save a different group of settings as the default.

3. Click **OK**.

3 - Create the 3D Model

This document contains the following topics:

- Set Default Material
- Create Substrate
- Create Feed
- Assign Perfect E Boundary to the Feed
- Create Ground
- Draw Slot Cut-Out
- Complete the Ground
- Assign Perfect E Boundary to the Ground
- Create Patch
- Assign Perfect E Boundary to the Patch
- Create an Air Body
- Assign Radiation Boundary
- Add Length Based Mesh Operation to Boundary
- Create Radiation Setup
- Create Source
- Assign Excitation

Set Default Material

1. Copy the material name, *Rogers RT/duroid 5880(tm)*, from this guide. (Select the text and press **Ctrl+C**).
2. On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu to open the *Select Definition* dialog box:

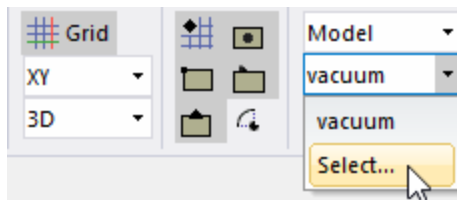


Figure 3-1: Default Material Drop-Down Menu

3. The cursor should be in the **Search by Name** text box. If not, press **Alt+N** to jump to this location.
4. Press **Ctrl+V** to paste the material name into the text box. The list will scroll to the specified material, and it will be selected.
5. Press **Enter**.

The material is selected as the default for new objects, and the *Select Definition* window closes.

Create Substrate

To create the substrate, construct a box as follows.

1. Press **Alt+B**.
2. If the *CreateBox* dialog box does not appear, press **F4**.
3. On the **Command** tab of the dialog box, edit the values as shown in the following figure:

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	Global		
Position	-7,-4.5,0	cm	-7cm , -4.5cm , 0cm
XSize	12	cm	12cm
YSize	9	cm	9cm
ZSize	0.32	cm	0.32cm

Figure 3-2: Substrate Properties – Command Tab

Keep the dialog box open.

4. Edit the properties within the **Attribute** tab as shown below and then click **OK**.
 - a. **Name = Substrate**
 - b. Ensure that **Material = Rogers RT/duroid 5880(tm)**.
 - c. **Material Appearance** is **cleared** (not selected).
 - d. **Color = Light Gray** (Red: 192, Green: 192, Blue: 192)
 - e. **Transparent = 0.6**
5. Press **Ctrl+D** to fit the view.

Note:

Use this command as needed after constructing every component.

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

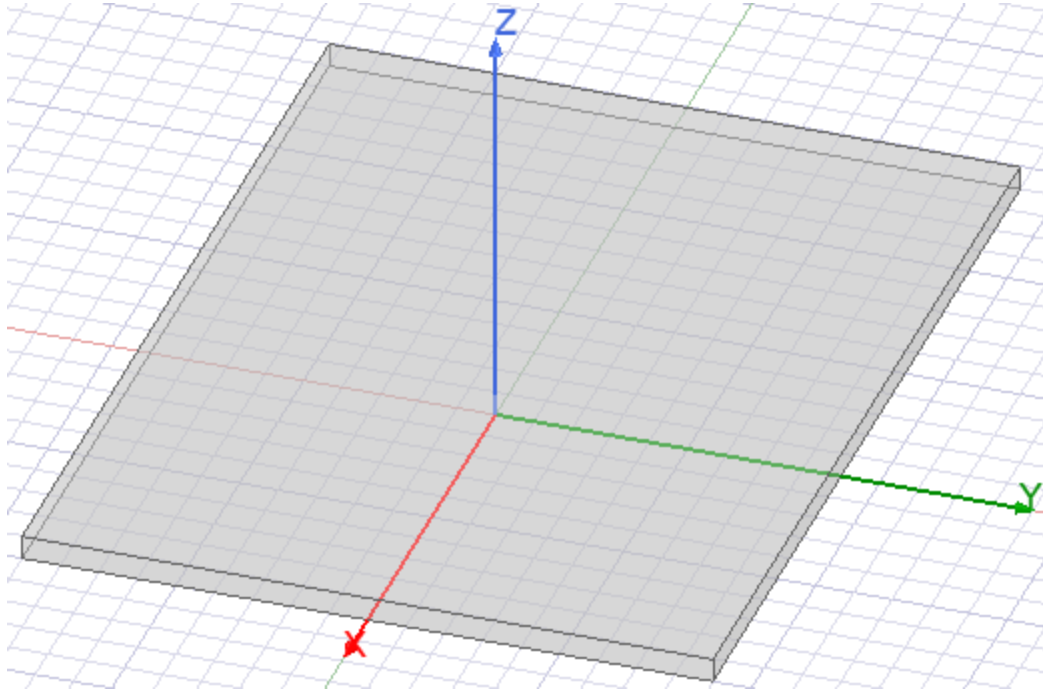


Figure 3-3: Substrate Created

Create the Feed

To create the feed, construct a rectangle as follows.

1. Press **Alt+R** and edit the values in the **Command** tab of the *CreateRectangle* dialog box as shown below:

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate Sys...	Global		
Position	-5 , -0.2475 , 0	cm	-5cm , -0.2475cm , 0cm
Axis	Z		
XSize	7	cm	7cm
YSize	0.495	cm	0.495cm

Figure 3-4: Feed Properties – Command Tab

2. Edit the properties in the **Attribute** tab as follows and then click **OK**.
 - a. **Name = Feed**
 - b. **Color = Blue** (Red: 0, Green: 0, Blue: 255)
3. Clear the current selection.

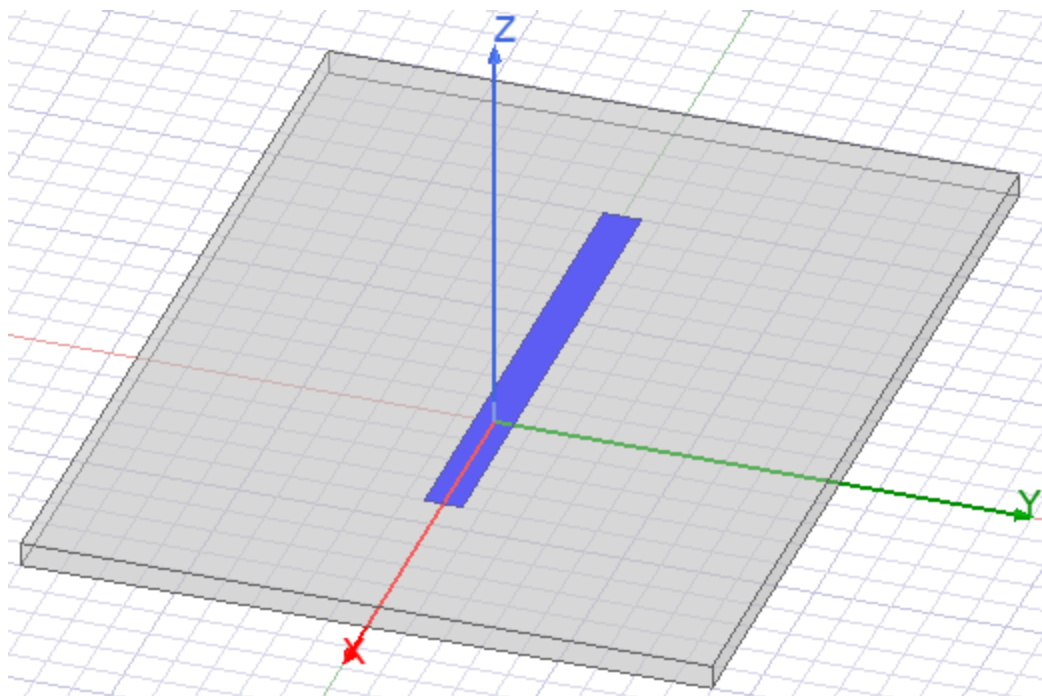


Figure 3-5: *Feed Created*

Assign a Perfect E Boundary to the Feed

Reselect the *Feed* and assign a perfect E boundary to it, as follows:

1. Press **Alt+N** to access the *Select Object* dialog box.
2. Select **Feed** in the **Name** list and click **OK**.
3. Press **Shift+E** to access the *Perfect E Boundary* dialog box.
4. Change the **Name** to **PerfE_Feed** and press **Enter**.

PerfE_Feed appears in the Project Manager under *Boundaries*.

Create Ground

To create the ground, construct a rectangle as follows:

1. Press **Alt+R** and edit the values in the **Command** tab of the CreateRectangle dialog box, as shown in the following figure:

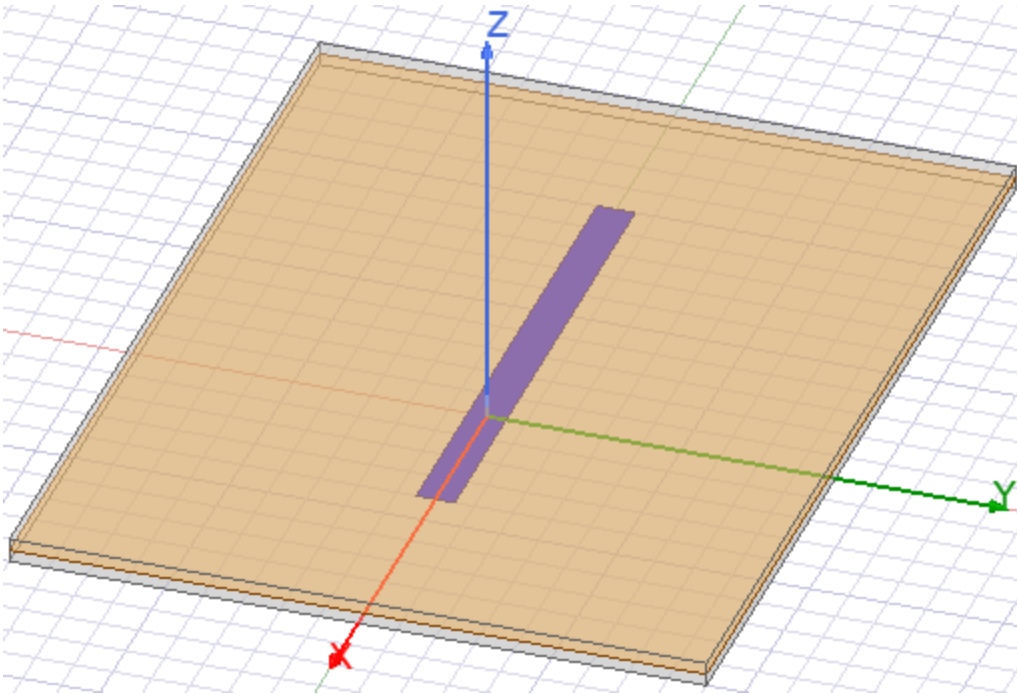
Note:

Remember to press **F4** if, while constructing a 2D or 3D object, the appropriate create object dialog box does not appear. The **F3** and **F4** keys switch between the graphical and numerical object definition methods, respectively.

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate System	Global		
Position	-7,-4.5,0.16	cm	-7cm,-4.5cm,0.16cm
Axis	Z		
XSize	12	cm	12cm
YSize	9	cm	9cm

Figure 3-6: *Ground Properties – Command Tab*

2. Edit the values in the **Attribute** tab as follows and click **OK**.
 - a. **Name = Ground**
 - b. **Color = Orange** (Red: 255, Green: 128, Blue: 0)
 - c. **Transparent = 0.7**
3. Clear the selection.

Figure 3-7: *Ground Created*

Draw Slot (Cut-Out)

To make a slot (cut-out) in the ground plane, draw a rectangle, as follows:

1. Press **Alt+R** but this time, press **F3** to switch to the graphical specification mode.
2. Draw a rectangle of any arbitrary size and location by clicking twice in the Modeler window (once for each of two opposite corners of the rectangle).

As soon as you click the second corner location, the Properties dialog box appears.

3. Edit the values in the **Command** tab as shown in the following figure:

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate Sys...	Global		
Position	-0.0775 ,-0.7 ,0.16	cm	-0.0775cm , -0.7cm , 0.16cm
Axis	Z		
XSize	0.155	cm	0.155cm
YSize	1.4	cm	1.4cm

Figure 3-8: Slot Properties – Command Tab

Do not close the *Properties* dialog box yet.

4. In the **Attribute** tab, change the **Name** to **Slot** and then click **OK**.
5. Clear the selection.

A small slot that is parallel to the Y axis and has default color and transparency attributes should be visible near the origin. In the next procedure, you will subtract the *Slot* from the *Ground* rectangle to produce the desired cut-out.

Complete the Ground

To complete the ground, you will perform Boolean subtraction, as follows:

1. Press **Alt+N** to access the *Select Object* dialog box.
2. In the **Name** list, select **Ground** and **Slot** (in that specific order) and then click **OK**:

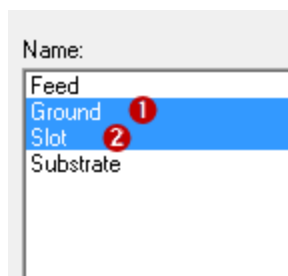


Figure 3-9: Selecting Objects for Subtraction

3. Press **Alt+S** to access the *Subtract* window.
4. Ensure that the *Blank Parts* and *Tool Parts* entries are specified as shown in the following figure and click **OK**.

Note:

The object selection order determines which object is the blank part and which is the tool part.

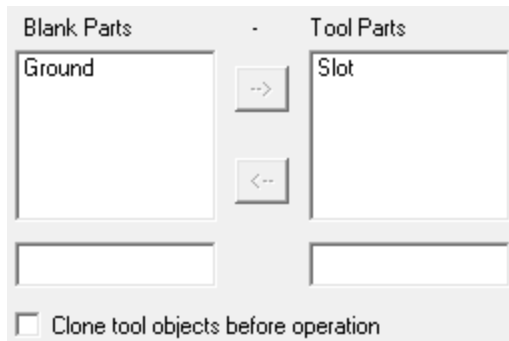


Figure 3-10: Subtract Dialog Box

5. Clear the selection. The model should look like the following figure:

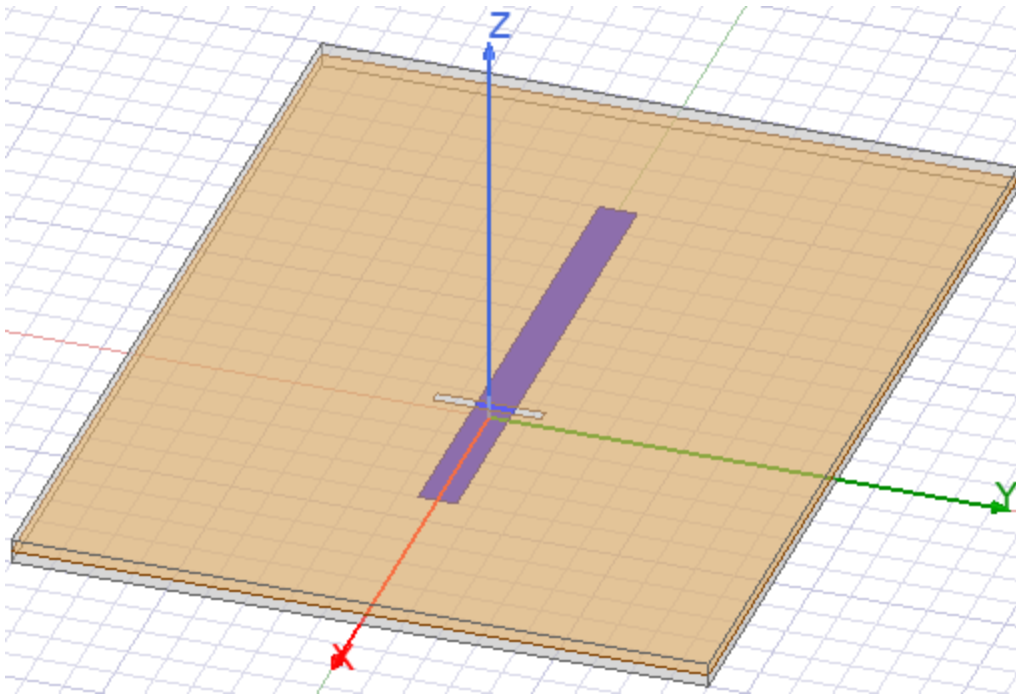


Figure 3-11: Model with Slot Subtracted from Ground

Assign Perfect E Boundary to the Ground

Select *Ground* and assign a perfect E boundary to it as follows:

1. In the History Tree, under *Model > Sheets > Unassigned*, select **Ground**.
2. Press **Shift+E**.

The *Perfect E Boundary* dialog box appears.

3. Change the default **Name** to **PerfE_Ground**:

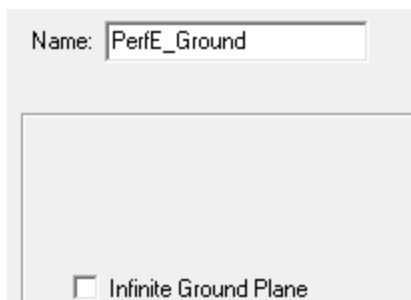


Figure 3-12: Perfect E Boundary

4. Press **Enter** to apply the boundary.
5. Clear the selection if the boundary visualization is displayed.

Create Patch

To create the antenna patch, construct a rectangle as follows:

1. Press **Alt+R** and then press **F4** to switch back to the numerical object definition mode.

The *CreateRectangle* dialog box appears.

2. Edit the values in the **Command** tab as shown in the following figure and then click **OK**.

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate Sys...	Global		
Position	-2, -1.5, 0.32	cm	-2cm, -1.5cm, 0.32cm
Axis	Z		
XSize	4	cm	4cm
YSize	3	cm	3cm

Figure 3-13: Patch Properties – Command Tab

3. Set the following properties under the **Attribute** tab:
 - a. **Name = Patch**
 - b. **Color = Red** (Red: 255, Green: 0, Blue: 0)
 - c. **Transparency = 0.6**
4. Clear the selection.

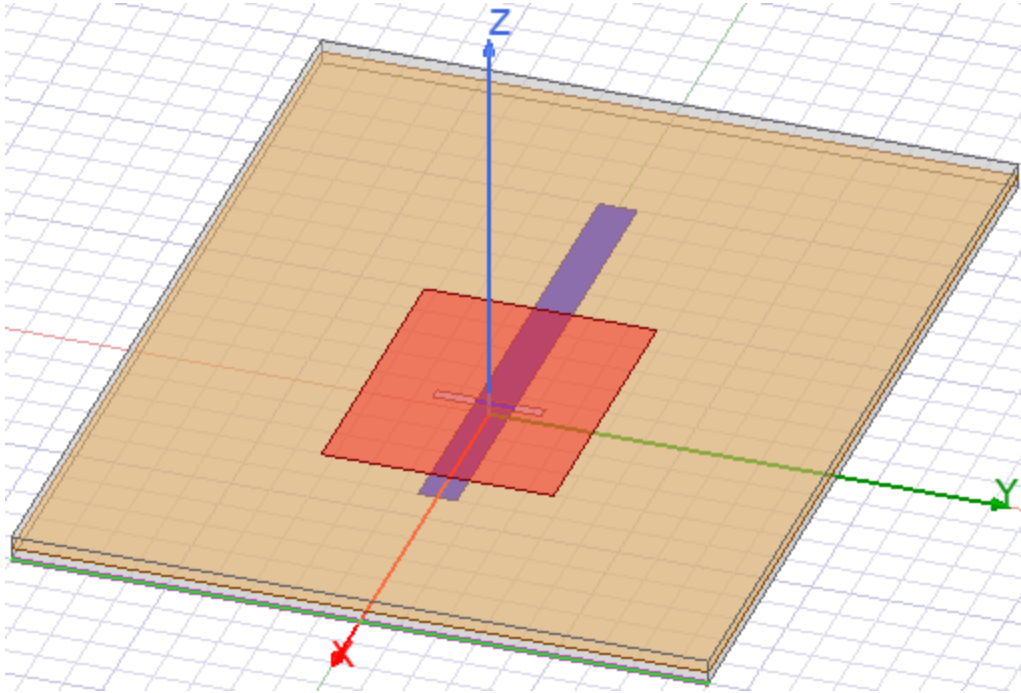


Figure 3-14: *Patch Created*

Assign a Perfect E Boundary to the Patch

Reselect the *Patch* and assign a perfect E boundary to it, as follows:

1. In the History Tree, under *Model > Sheets > Unassigned*, select **Patch**
2. Press *Shift+E*.

The *Perfect E Boundary* dialog box appears.

3. Change the Name to *PerfE_Patch*.

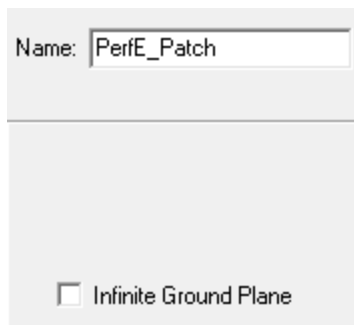


Figure 3-15: *Perfect E Boundary* dialog box

4. Press **Enter** to apply the boundary and close the dialog box.
5. Clear the selection if the boundary visualization is displayed.

Create Air Body

The air body is an automatically or manually created solution region surrounding the antenna within which electromagnetic fields can be computed.

You can automate the process of creating a vacuum volume of an appropriate size (based on the solution frequency), and assigning the necessary boundary condition to its faces via the menu command **HFSS > Model > Create Open Region**. (The difference between vacuum and air is insignificant with regard to electromagnetic wave propagation.) Alternatively, you can select the **Auto-Open Region** option within the *Solution Type* dialog box. This option, in addition to creating an invisible bounding region and assigning the necessary boundary condition, also automatically creates an analysis setup with a default solution frequency. You can then edit the default value to specify a different solution frequency, and the invisible bounding region is automatically resized accordingly.

While you are encouraged to take advantage of the automatic open region capabilities, this model uses a customized air body with zero padding between the model and radiation boundary on the front, back, and sides. Padding is only provided above and below the model (that is, in the +Z and -Z directions). In later procedures, you will assign a radiation boundary to the faces of this region (to absorb all outgoing radiation from the antenna), and you will apply mesh refinement to its faces. This manual air box and radiation boundary creation process provides the demonstration of a few more keyboard shortcuts.

Use the *Draw box* command to create the *Air* body, as follows:

1. Press **Alt+B** to access the *CreateBox* dialog box and edit the values in the **Command** tab as shown in the following figure:

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate Sys...	Global		
Position	-7,-4.5,-2	cm	-7cm,-4.5cm,-2cm
XSize	12	cm	12cm
YSize	9	cm	9cm
ZSize	4.32	cm	4.32cm

Figure 3-16: Air Body Properties – Command Tab

2. Edit the values under the **Attribute** tab as follows and then click **OK**:
 - a. **Name = Air**
 - b. **Material = air**
 - c. **Color = Black** (Red: 0, Green: 0, Blue: 0)
 - d. **Display Wireframe** option = **Selected**
3. Press **Ctrl+D** to fit the view.
4. Clear the current selection.

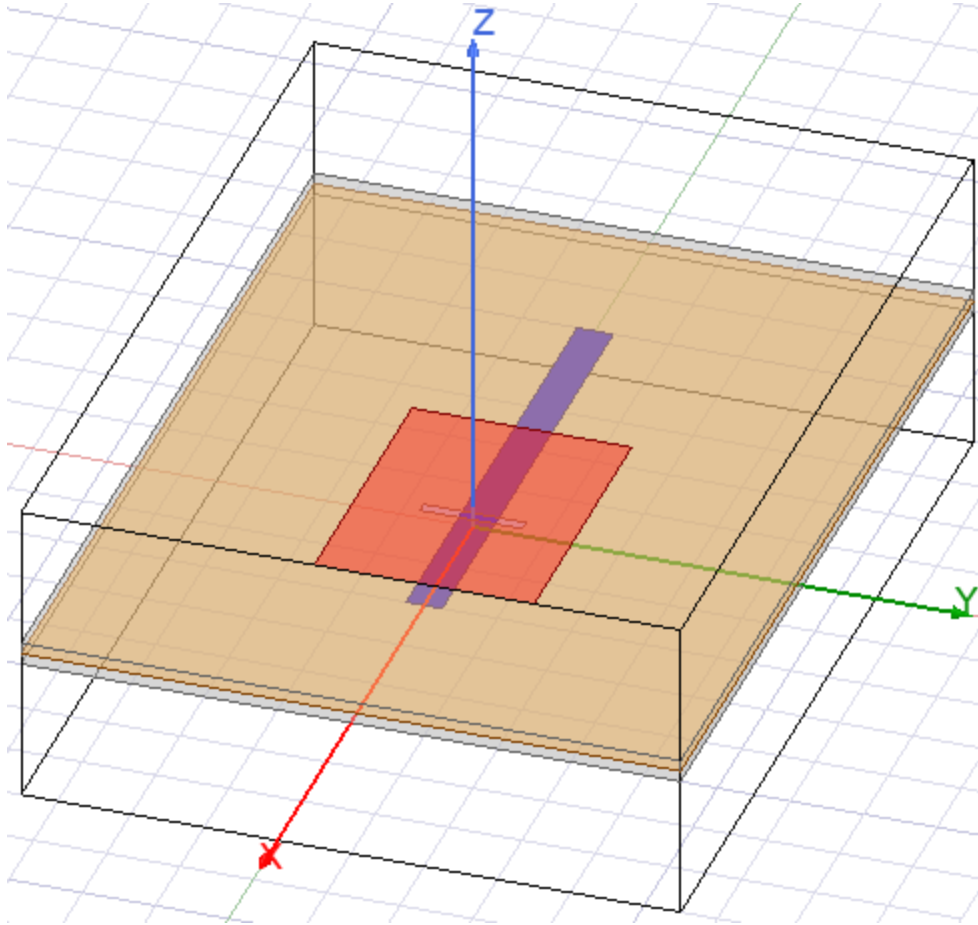


Figure 3-17: *Air Body Created*

Assign Radiation Boundary

To assign radiation boundary to the air body perform the following steps:

1. Under *Model > Solids > air* in the History tree, select **Air**.
2. Press **Shift+X** to access the Radiation Boundary dialog box.
3. Keep the default **Name** as **Rad1** and press **Enter** to apply the boundary.
4. If the boundary visualization is not displayed, select **Rad1** under *Boundaries* in the Project Manager.

Your model should look like the following figure:

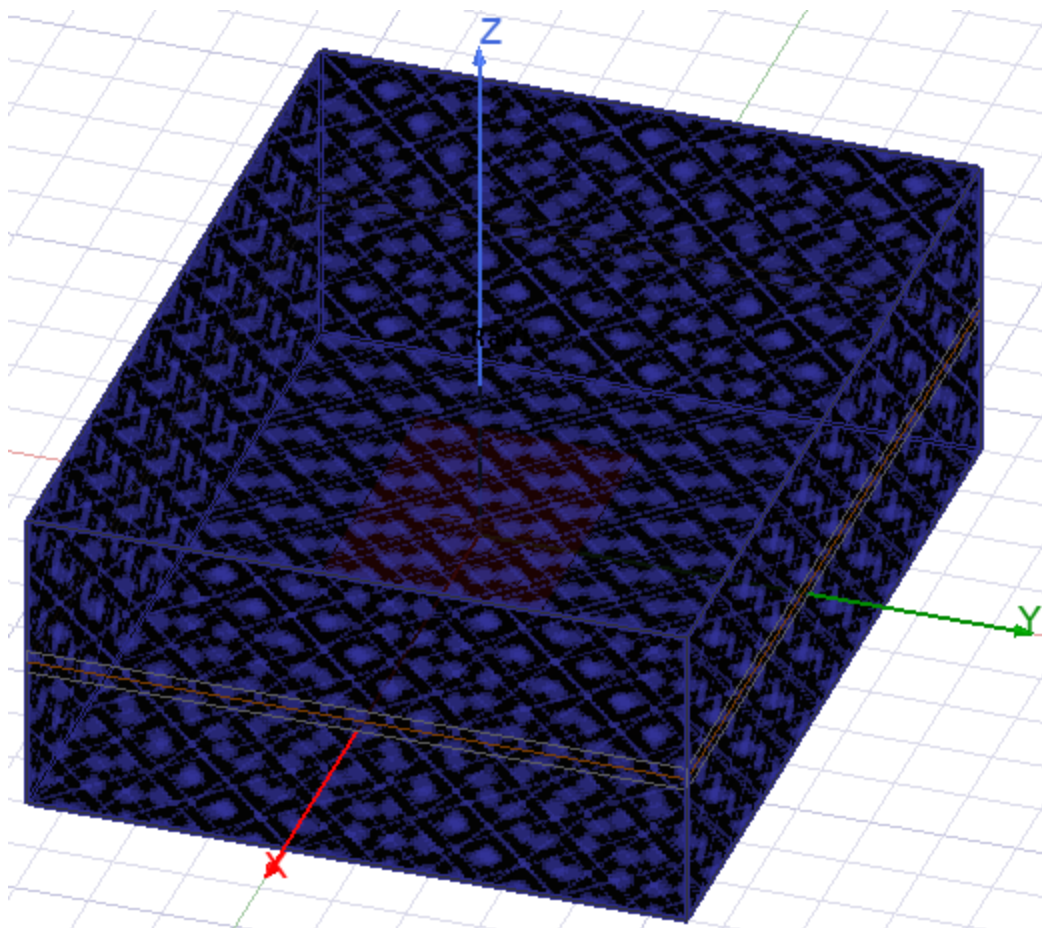


Figure 3-18: Radiation Boundary Applied to the *Air* Body

5. Clear the selection to restore the normal view of the model.

Add Length Based Mesh Operation to Radiation Boundary

Far fields are calculated by integrating the fields on the radiation surface. To obtain accurate far fields for the antenna problems, the integrating surface should be forced to have a $\lambda/6$ to $\lambda/8$ maximum tetrahedra length.

1. Under *Model > Solids > air* in the History tree, select **Air**.
2. Press **Shift+L** to access the *Element Length Based Refinement* dialog box.
3. Specify the following properties in the dialog box:
 - a. **Name = LengthOnRadiation**
 - b. **Set maximum element length** is **selected** and set to **2 cm**.

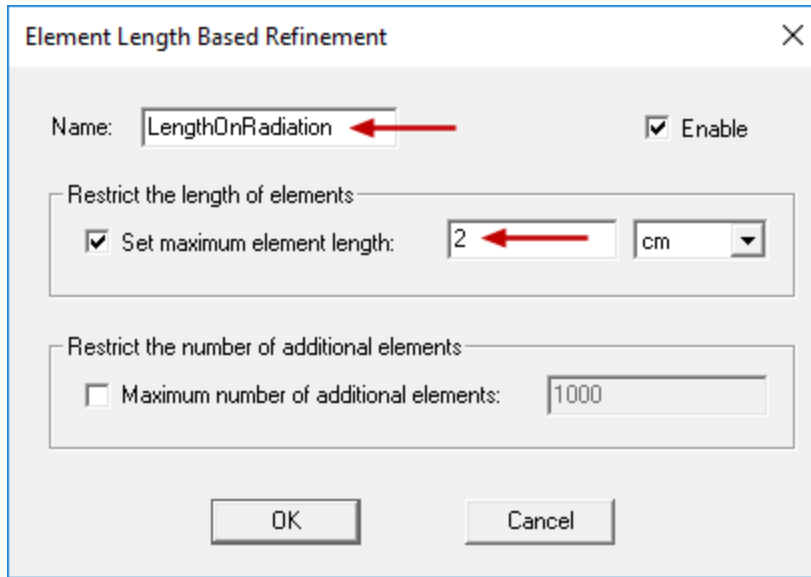


Figure 3-19: Element Length Based Refinement for Air Body

4. Press **Enter** to complete the operation.

LengthOnRadiation appears under *Mesh* in the Project Manager.

5. Clear the selection.

Create Radiation Setup

While the previously created *Air* body is an actual region of finite elements, and therefore part of the solution process, the far field setup is purely a post-processing tool. You can create a far field setup after solving an analysis and then produce reports based on the far field setup without having to rerun the analysis.

In a finite air or vacuum region surrounding the model, you can overlay current density, electric field, and magnetic field results. The far field setup is used to produce 2D polar and 3D radiation pattern or gain plots (to determine the directionality of an antenna, for example).

Commands to create an infinite sphere radiation setup are as follows:

1. Press the **Insert** key to access the *Far Field Radiation Sphere Setup* dialog box.
2. Edit the properties under the *Infinite Sphere* tab as shown in the following figure:

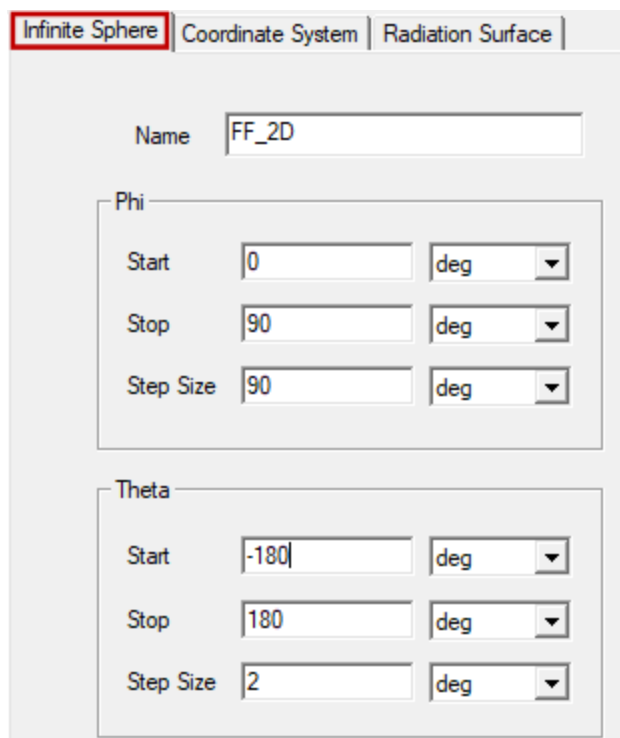


Figure 3-20: Far Field Radiation Sphere Setup

3. Press **Enter** to complete the setup and close the dialog box.

FF_2D appears under *Radiation* in the Project Manager.

Create Source

In this procedure you will create a Source rectangle for applying excitation to the model. First, you will change the drawing plane, since this rectangle will be drawn on the YZ rather than the XY plane.

Make YZ the Active Drawing Plane:

1. Sequentially press **Alt, M, G, Enter**, and **Y**.

The **Drawing Plane** drop-down menu on the **Draw** ribbon tab now indicates the YZ plane:

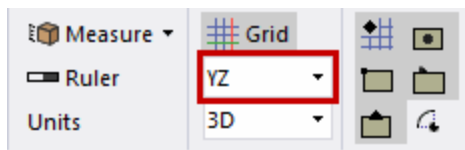


Figure 3-21: Drawing Plane Indicated on *Draw* Ribbon Tab

Create the Source Rectangle:

2. Press **Alt+R** to access the *CreateRectangle* command.
3. Edit the values under the **Command** tab as shown in the figure:

Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate Sys...	Global		
Position	-5,-0.2475,0	cm	-5cm,-0.2475cm,0cm
Axis	X		
YSize	0.495	cm	0.495cm
ZSize	0.16	cm	0.16cm

Figure 3-22: Source Properties – Command Tab

4. Set the properties under the **Attribute** tab as follows and then click **OK**.
 - a. **Name = Source**
 - b. **Color = Bright Green** (Red: 0, Green: 255, Blue: 0)
 - c. **Transparency = 0**

Keep the Source rectangle selected. In the next procedure, you will add excitation to it:

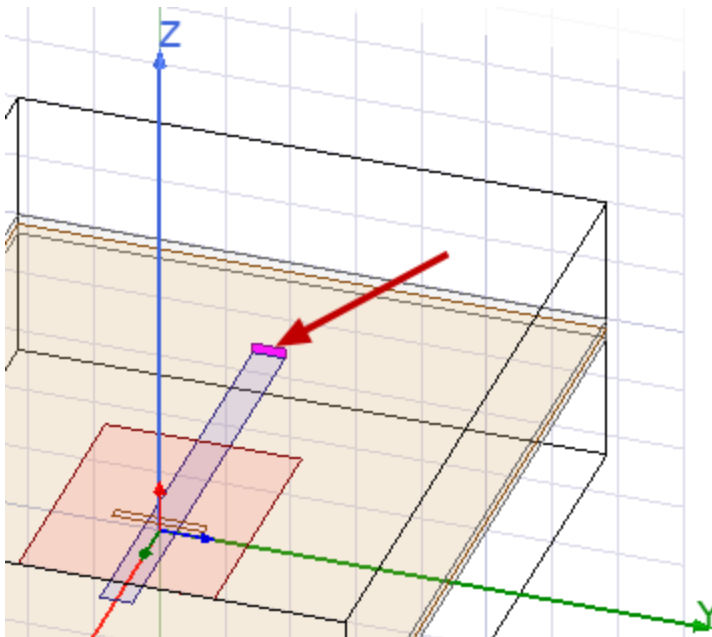


Figure 3-23: Source Created

Assign Excitation

For this model we will use a lumped port to excite the *Source*.

1. Ensure that *Source* is still selected (magenta highlight).
2. Press **Shift+4**.

The *Reference Conductors for Terminals* dialog box appears.

3. Select the **Use as Reference** option in the **Ground** row of the table:

Port Name:

Terminal Naming

☒ Use conductor name

☐ Use port object name

NOTE: Multiple reference conductors touching a port must all be connected in the plane of the port.

	Conductor	Use as Reference
<input type="checkbox"/>	Feed	<input type="checkbox"/>
<input checked="" type="checkbox"/>	Ground	<input checked="" type="checkbox"/>

☒ Highlight selected conductors

Figure 3-24: Reference Conductor for Source Terminal (Port 1)

4. Keep the default *Port Name* (1) and click **OK**.
5. Select **Feed_T1** under *Excitations* > 1 in the Project Manager, press **F2**, rename it as **T1**, and press **Enter**.

This procedure completes the model construction. You are now ready to set up, validate, and analyze the model.

4 - Analyze and Evaluate Results

This section contains the following topics:

- Create Analysis Setup
- Add Frequency Sweep
- Model Validation
- Analyze
- Review Convergence Panel
- Create Terminal S Parameter Plot - Magnitude
- Create Far Field Overlay

Create Analysis Setup

1. Press **Shift+5** to access the *Driven Solution Setup* dialog box.
2. Under the **General** tab, specify the settings shown in the following figure and then click **OK**:

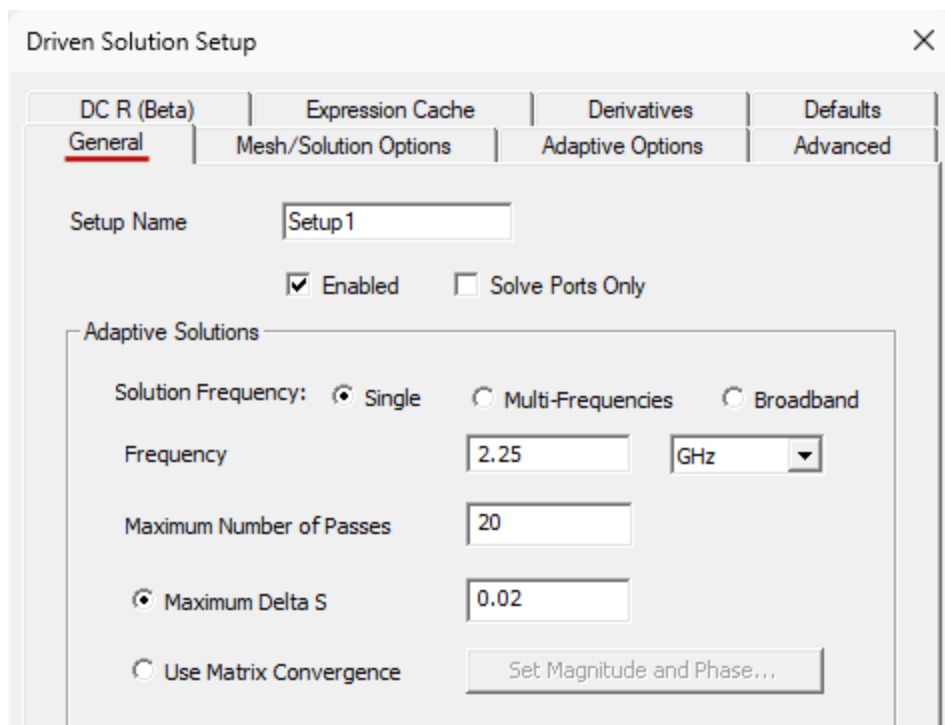


Figure 4-1: Driven Solution Setup Dialog Box

Because a port excitation has already been assigned, the *Edit Frequency Sweep* dialog box opens automatically at the completion of the solution setup.

Add Frequency Sweep

3. If you accidentally closed the *Edit Frequency Sweep* dialog box, press **Shift+6** to reopen it.

You can also use this command to add a sweep when the solution setup is performed before any ports are assigned, in which case the dialog box does not appear automatically.

4. Under the **General** tab, specify the settings shown in the following figure and then click **OK**:

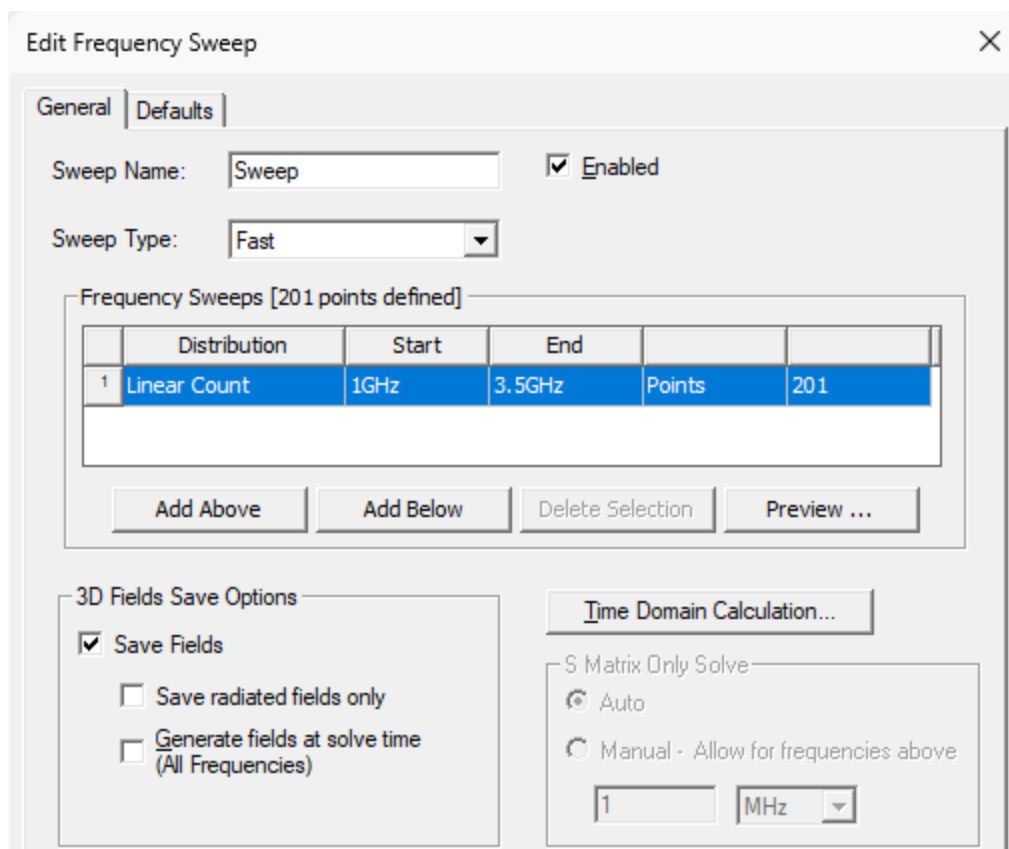


Figure 4-2: *Edit Frequency Sweep* Dialog Box

Validate and Analyze

You are ready to verify that the model is set up correctly and to perform the analysis.

Model Validation:

1. Press **Ctrl+S** to save your work.
2. Press **Alt+7** to perform a *Validation Check*.

If you performed all the steps detailed in this getting started guide correctly, the model will pass the validation check:

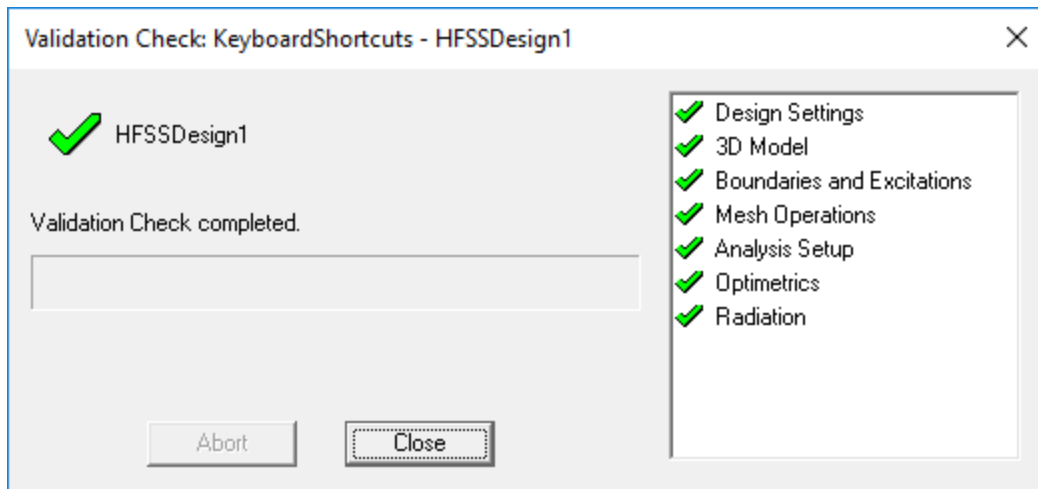


Figure 4-3: Successful Validation Check

3. Press **Enter** to close the *Validation Check* window.

Analyze:

4. Press **Alt+Z** to perform the analysis.

HFSS solves the model.

Review Convergence History

Solution convergence information is provided as part of the *Solutions* dialog box. To view the convergence history for the analysis just run, do as follows:

1. Press **Alt+D** to access the *Solutions* dialog box.

Note:

This command is an example of a custom keyboard shortcut being particularly more efficient than navigating the menu bar via the keyboard, which would require nine keystrokes in this case. Specifically, to execute this command via menu navigation keystrokes, you would have to sequentially press **Alt, H, Enter, R, R, Enter, D, D, and Enter**. However, as stated in the *Introduction*, menu navigation keystrokes can be useful when writing or recording scripts to automate modeling tasks.

2. Click the **Convergence** tab.

- For the **View** option, select **Plot**.

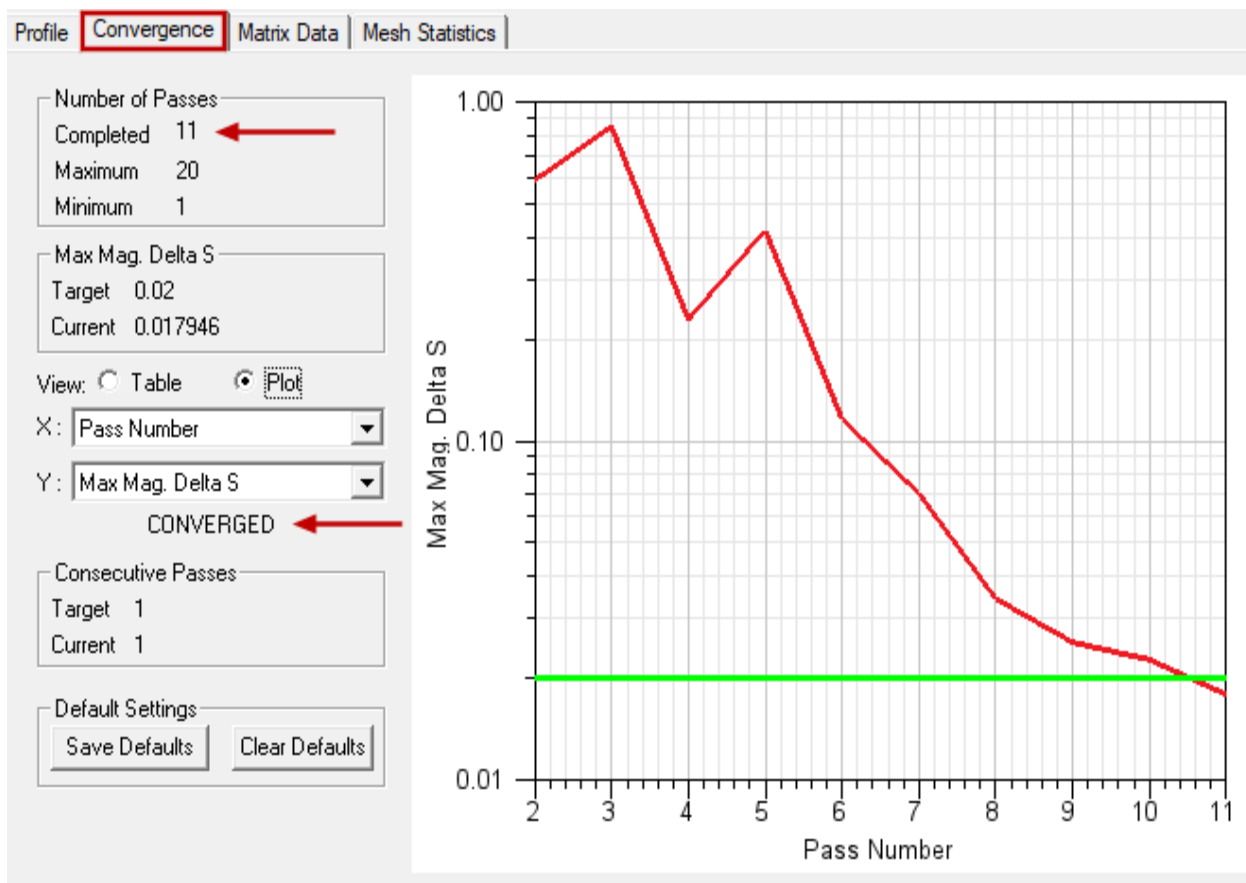


Figure 4-4: Convergence History – Max Mag. Delta S vs. Pass Number

From the data presented in this figure, you can see that the solution included eleven iterative passes, and the convergence criterion (*0.02 Max Mag. Delta S*) was satisfied.

Note:

Your convergence history may differ somewhat from the preceding figure due to variations between software releases and between different computing platforms.

- Optionally, review the solution data presented within the **Profile**, **Matrix Data**, and **Mesh Statistics** tabs.
- Click **Close**.

Create Terminal S-Parameter Plot

1. Alt, H, Enter, R, R, Enter, Enter, Enter.

The Report dialog box appears.

2. Ensure that the settings are as shown in the following figure and then press **N** (for *NewReport*).

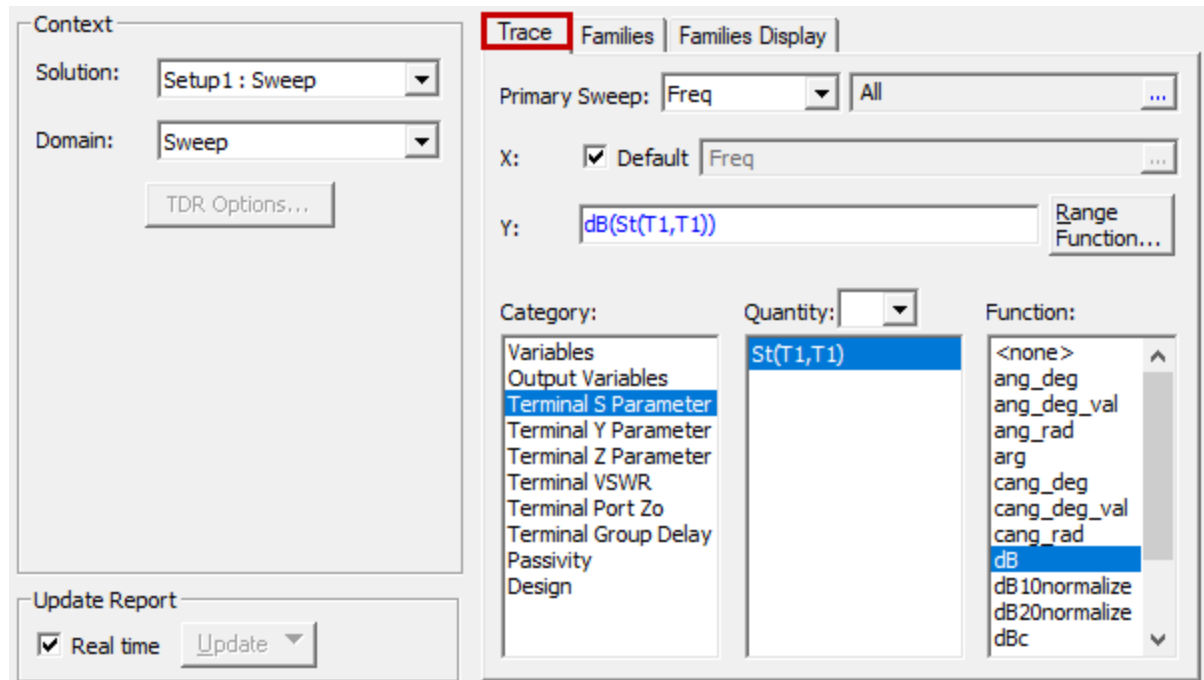


Figure 4-5: Report Settings for Terminal S-Parameter Plot

3. Press **C** to close the dialog box.

Your plot should resemble the following figure:

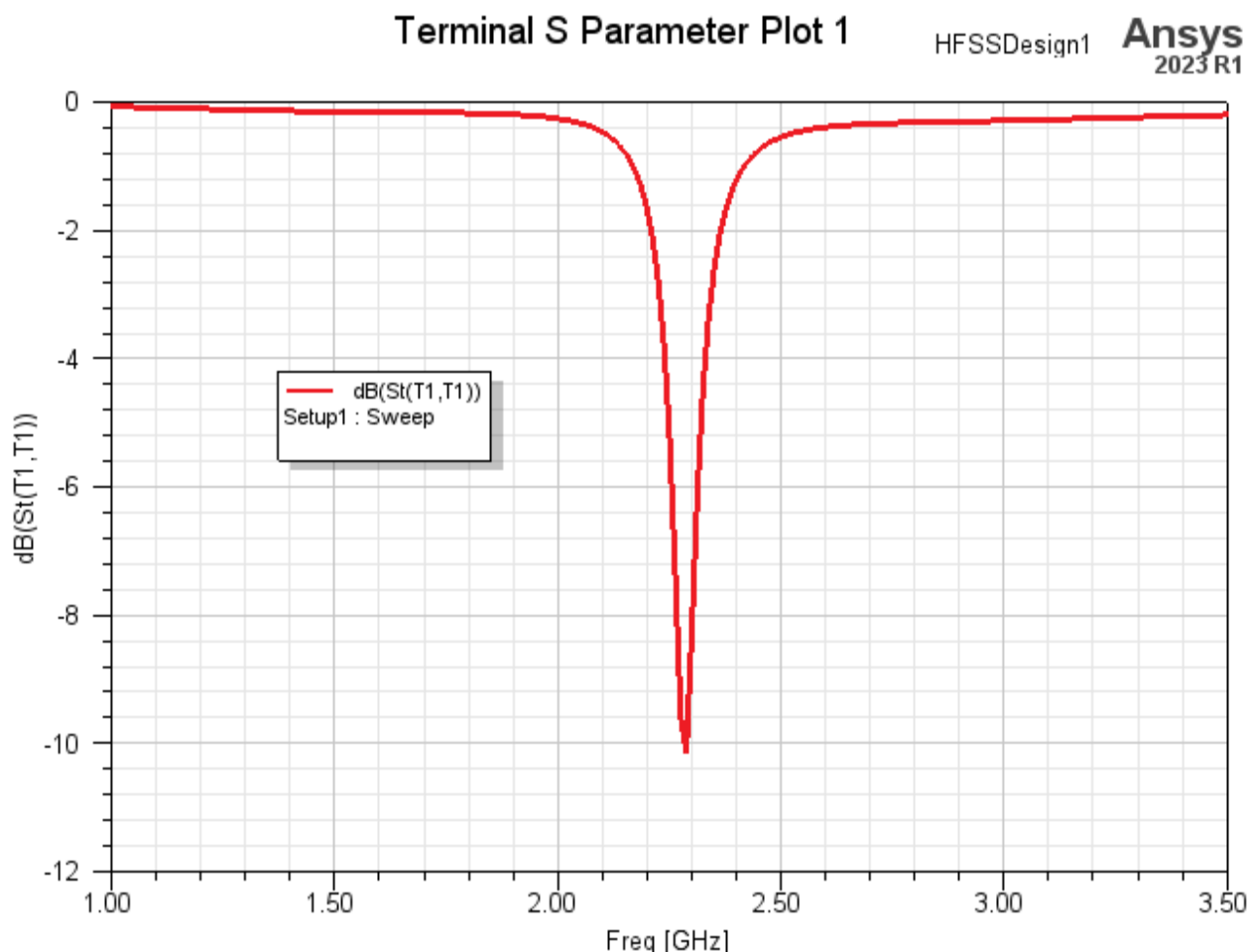


Figure 4-6: S Parameter Plot

Notice that the minimum S-parameter result occurs at slightly less than 2.3 GHz.

Create and Animate E-Field Overlay

Create an E-field overlay on the *Air* body, adjust the legend scale, and animate the field, as follows:

1. Make the Modeler window the active window.
2. In the History Tree, under *Model > Solids > air*, select **Air**.
3. Press **Alt+E** to begin creating an E-field overlay.
4. Specify the following overlay properties in the *Create Field Plot* dialog box:
 - a. From the **Solution** drop-down menu, select **Setup1 : Sweep**.
 - b. From the **Frequency** drop-down menu, select **2.2875GHz**, which is the frequency that most closely corresponds to the minimum point on the S-parameter plot.

- c. **Quantity = Mag_E**
- d. **In Volume = Air**

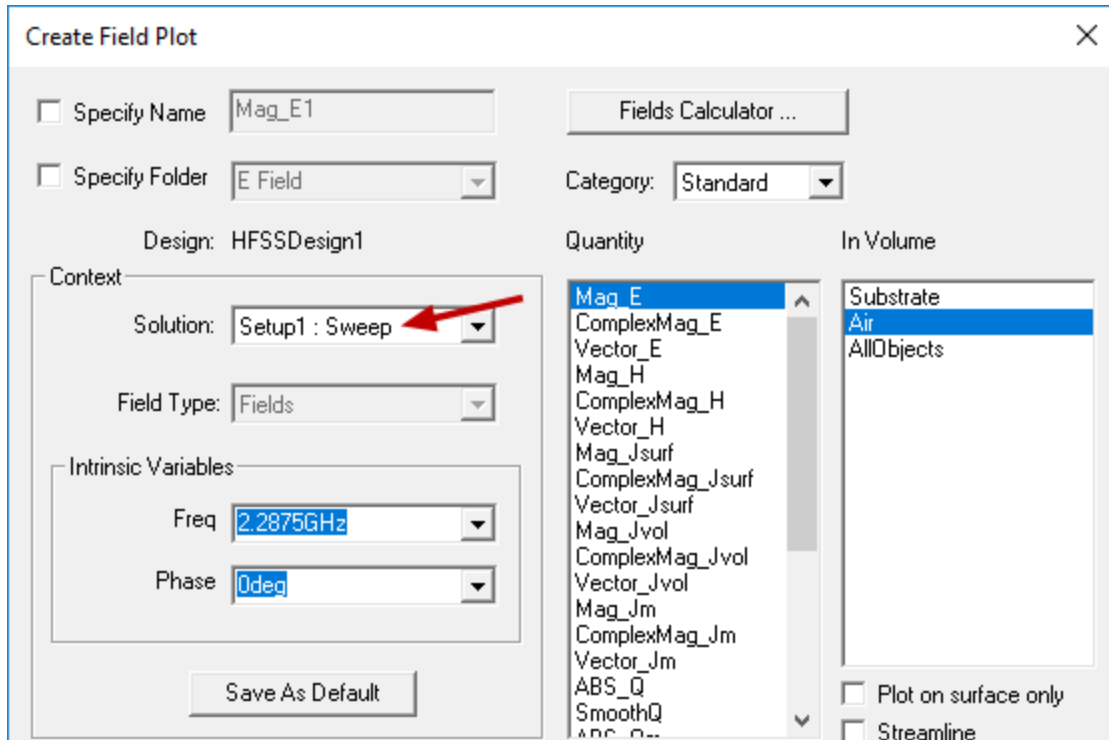


Figure 4-7: E-Field Plot Settings

5. Click **Done** to generate the overlay plot, which will appear after field computations are complete.
6. Double-click within the *E Field plot legend* and select the **Scale** tab of the *E Field* dialog box that appears.
7. Make the following changes to the plot *Scale* settings and then click **Close**:
 - a. **Num. Division = 10**
 - b. Select **Use Limits**
 - c. **Min = 0.04**
 - d. **Max = 4000**
 - e. Select the **Log** option for a logarithmic scale.
 - f. In the *Number Format* section, choose **Decimal** from the **Type** drop-down menu.
 - g. **Precision = 2**
8. Press **Shift+A** to animate the E-field overlay.

The *Create Animation Setup* dialog box appears.

9. Accept the default animation settings and click **OK**.

The *Animation* window appears and the E-field results are phase-animated:

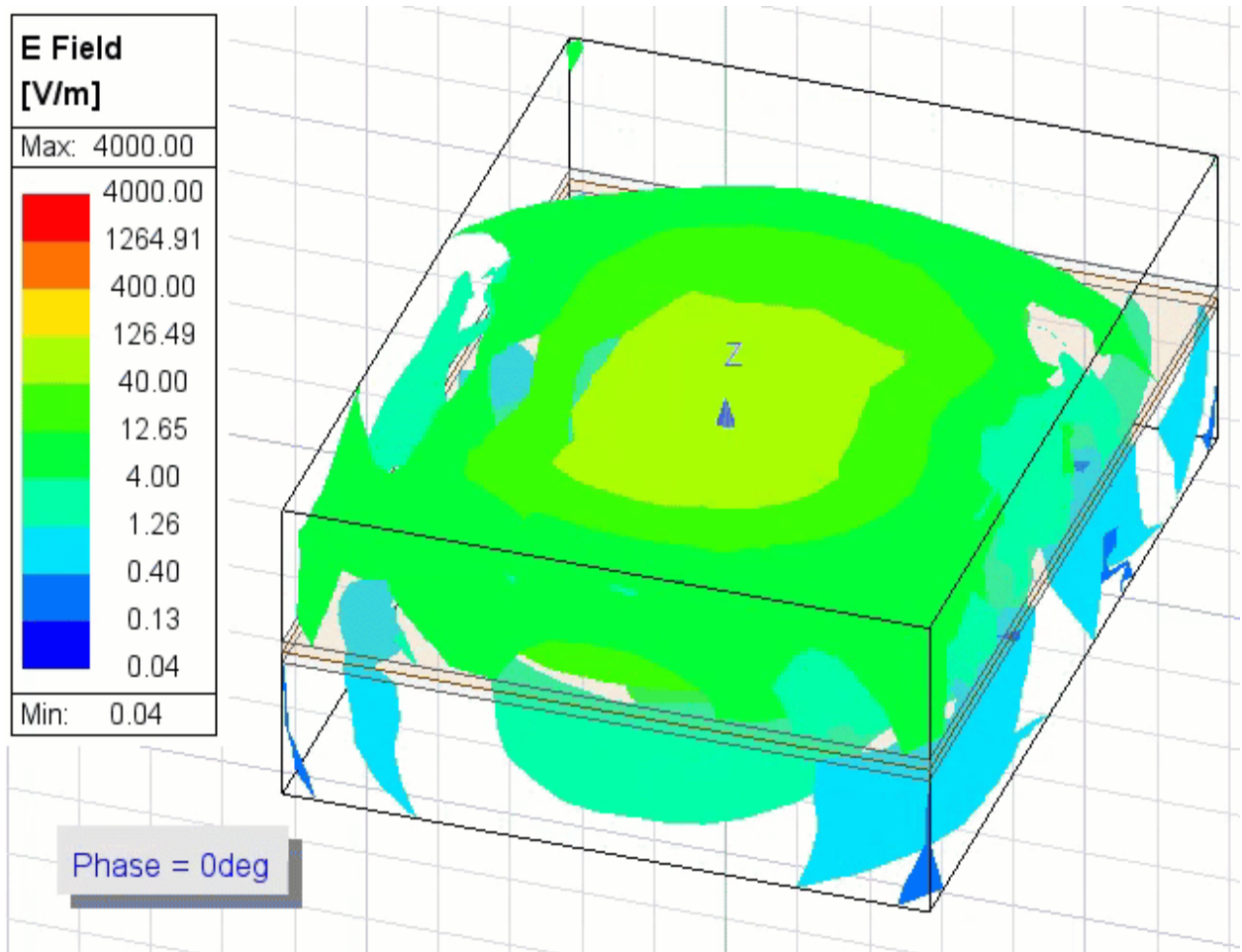


Figure 4-8: E-Field Phase-Animation

10. Use the animation controls to stop, restart, reverse, or adjust the speed of the animation.
11. Click **Close** when your finished viewing the animated results.

Create Gain Plot

1. Press **Alt+F** to begin creating a gain plot representing the far field radiation pattern.
2. In the Trace tab of the Report dialog box, specify the following settings:
 - a. **Solution = Setup1 : Sweep**
 - b. **Geometry = FF_2D** (This is the only choice because it's the only far field setup that has been defined.)
 - c. **Category = Gain**

- d. **Quantity** = **GainPhi** and **GainTheta**
- e. **Function** = **dB**

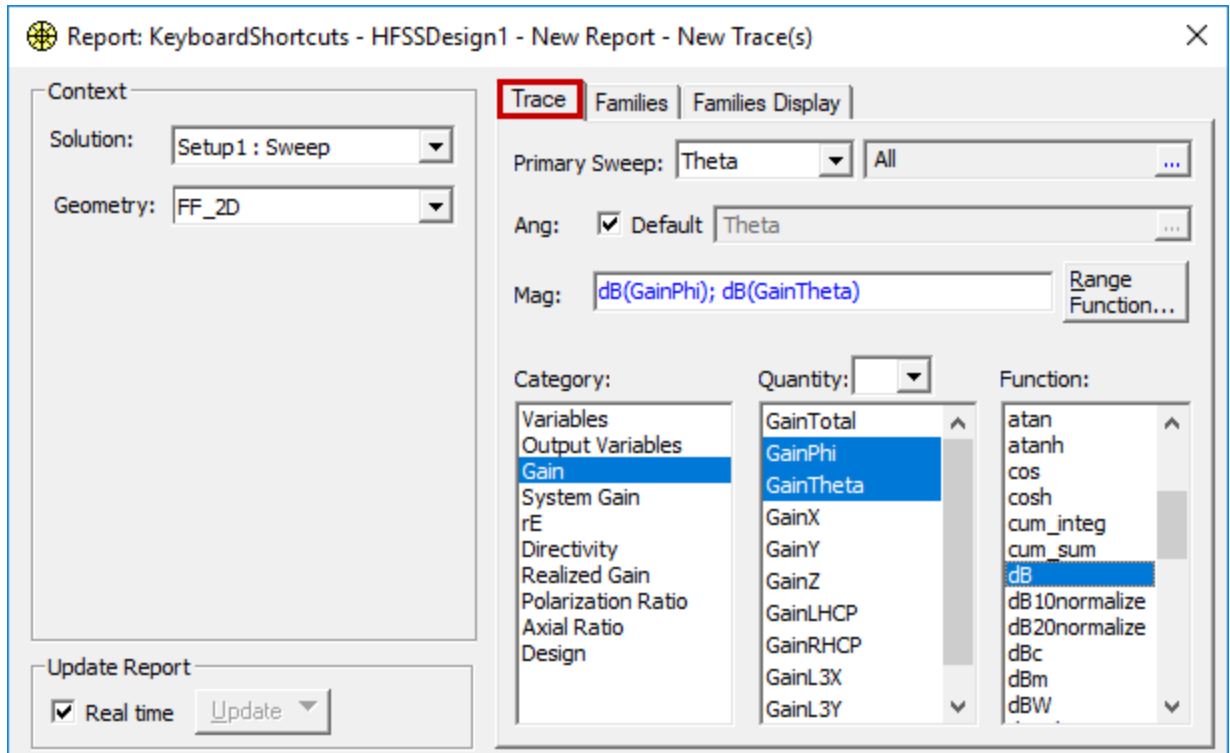



Figure 4-9: Gain Plot Settings – Trace Tab

3. In the **Families** tab, click the elipsis button () at the right end of the **Freq** row and select only the **2.2875GHz** frequency.
4. Click **New Report**. The plot appears when the necessary post-processing computations are done.
5. **Close** the *Report* dialog box.

Your gain plot should resemble the following figure:

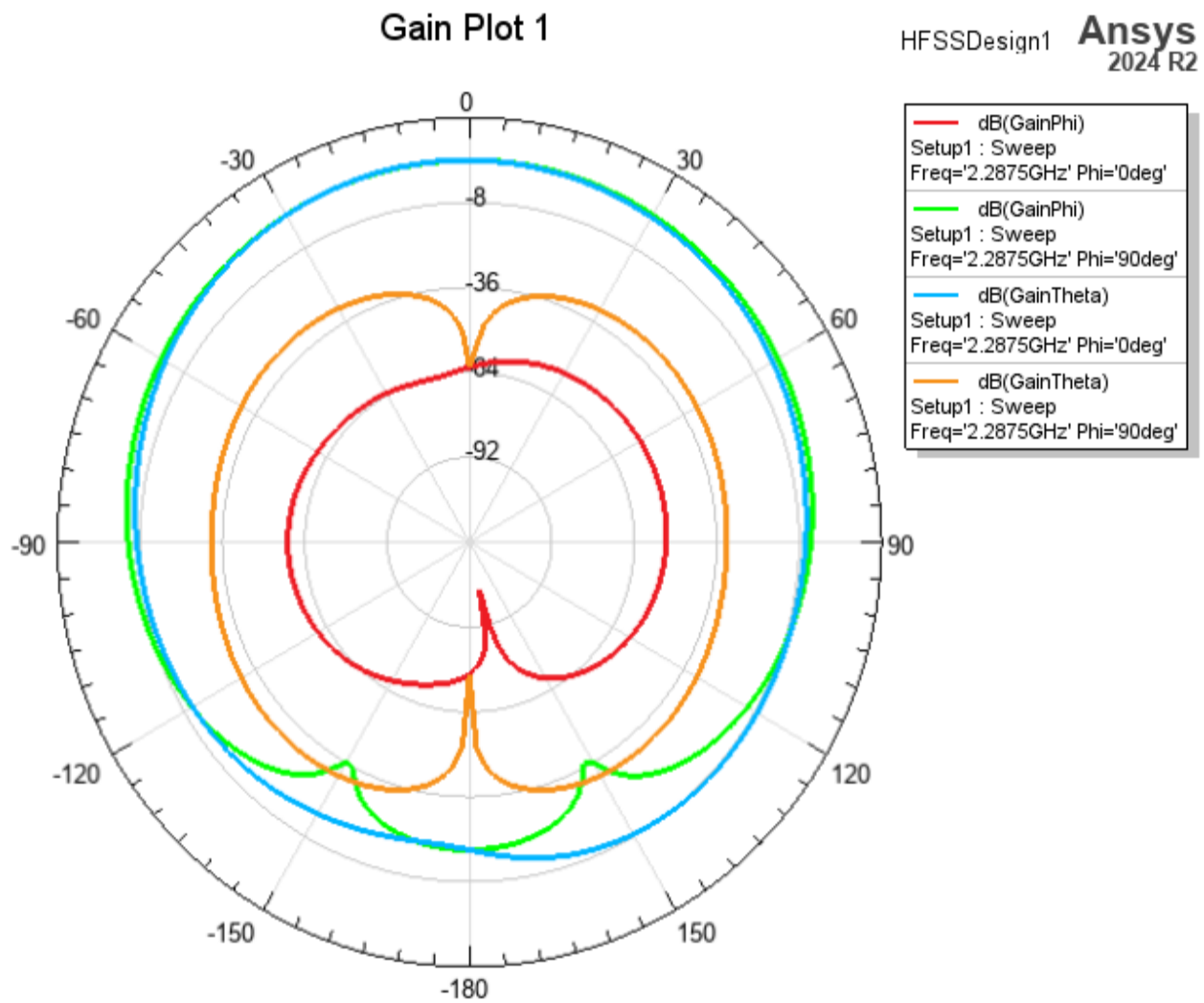


Figure 4-10: Gain Plot (Radiation Pattern)

5 - Examples

This section describes the usage of a few of the keyboard shortcuts. It is by no means a comprehensive set of examples, but rather, it highlights some useful and commonly used shortcuts.

The following topics are included:

- View Navigation Shortcuts
- Geometry Creation and Selection Shortcuts
- Plot Shortcuts

View Navigation Shortcuts

The following view navigation shortcuts are presented below:

- Ctrl+H
- Esc
- F6
- F7
- Plus Sign (+)
- Minus Sign (–)

Ctrl + "+" and Ctrl + "-":

Ctrl + "+" zooms in, and **Ctrl + "-"** zooms out. You can use the "+" or "-" keys on the numeric keypad or the "=+" or "-_" keys in the row below the function keys. If you use the "=+" key, do **not** press the *Shift* key.

The zoom level adjustments are in fixed increments and work both when no drawing operation is active and while a drawing operation is active. So you can manipulate the model view while creating geometry without terminating the drawing operation. Similarly, the *Zoom* commands in the menu bar and ribbon are also available whether or not a geometry creation operation is active.

You can use these shortcuts as an alternative to rolling the mouse wheel or dragging the mouse using the *Ctrl + middle-click* combination for zooming (or the *Shift + Alt + left-click* combination when legacy view navigation is enabled). The focal point (that is fixed point) for the zoom increments is at the center of the Modeler window's display area. Therefore, objects not at the center of the drawing area will move outward as you zoom in (toward the edge of the window or even off-screen, if you zoom in far enough).

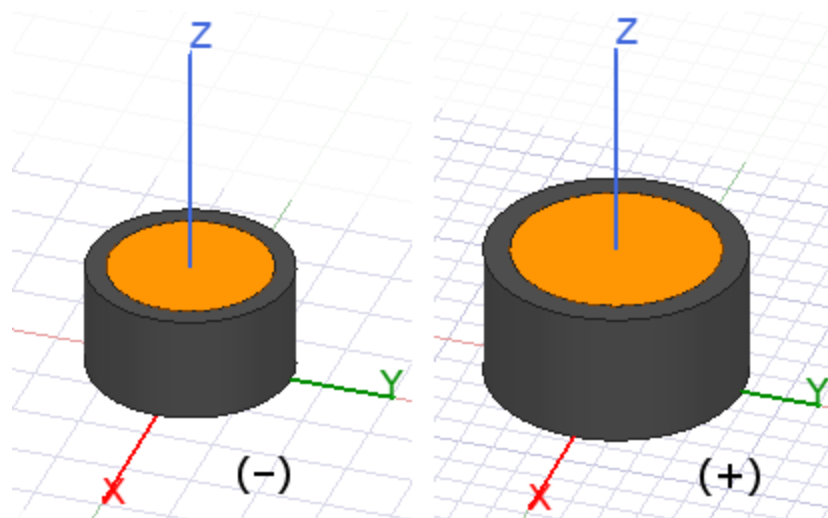


Figure 5-1: Zooming Out (-) or In (+)

Notice how the grid spacing is updated when you zoom in far enough for finer increments to be displayed.

Ctrl+H:

Use the **Ctrl+H** keyboard shortcut to quickly hide selected objects, as shown in the following figure:

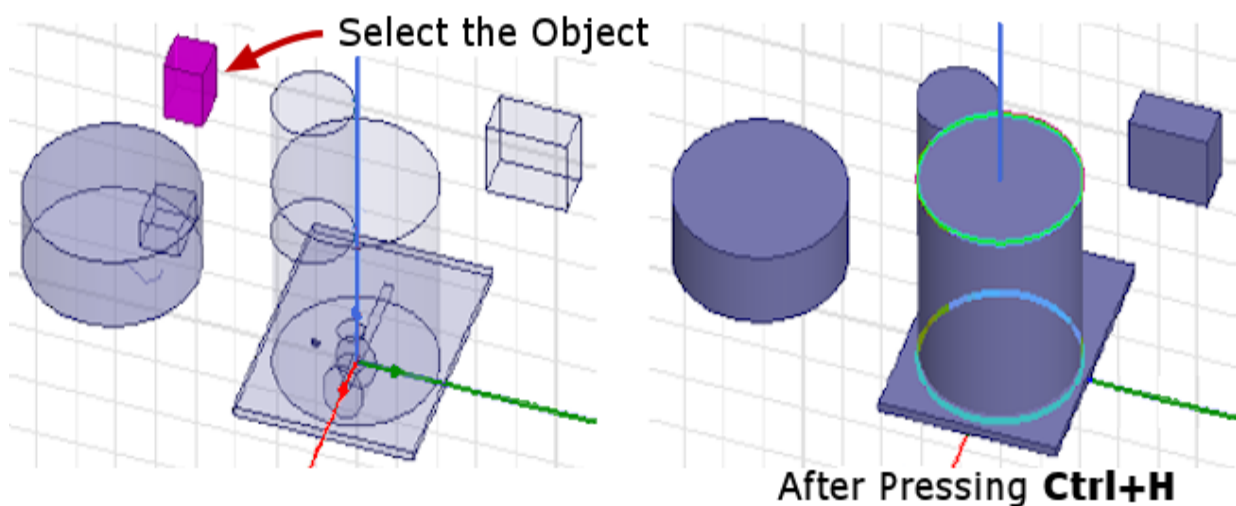


Figure 5-2: Selected Object Hidden using *Ctrl+H*

Escape (Esc):

Use the escape key (**Esc**) to cancel view navigation commands and return to the draw/select cursor mode. The following view navigation modes can be terminated using the **Esc** key:

- **Zoom** (*drag up for zoom in , down for zoom out*)
- **Zoom Area**
- **Pan**
- **Rotate**

The following image shows the view navigation command icon, the associated cursor, and the return to the normal drawing and selecting cursor when **Esc** is pressed:

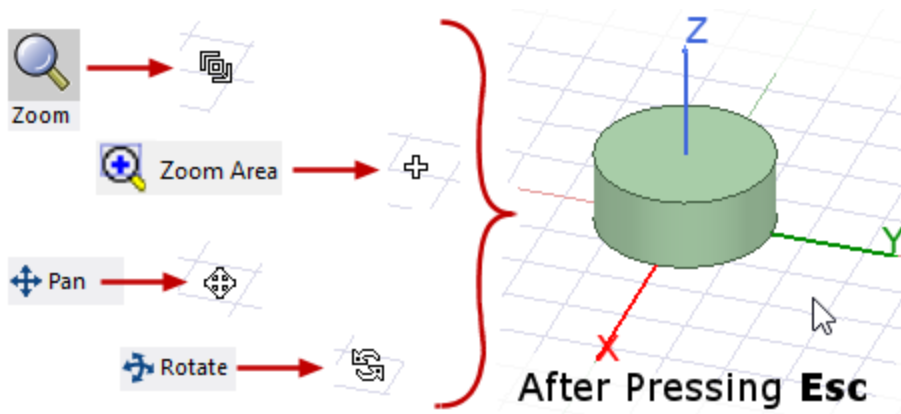


Figure 5-3: Zoom Mode Cursors and Canceling View Navigation Commands with the Esc Key

There is no cursor change associated with **Zoom +** and **Zoom -** commands.

F6 and F7:

The **F6** and **F7** keys determine the visual rendering mode of the model. These keys function globally, affecting the rendering of all objects in the Modeler window, as follows:

- **F6:** Displays a wireframe view of all objects.
- **F7:** Renders all objects with smooth shading using the specified object colors and transparency settings.

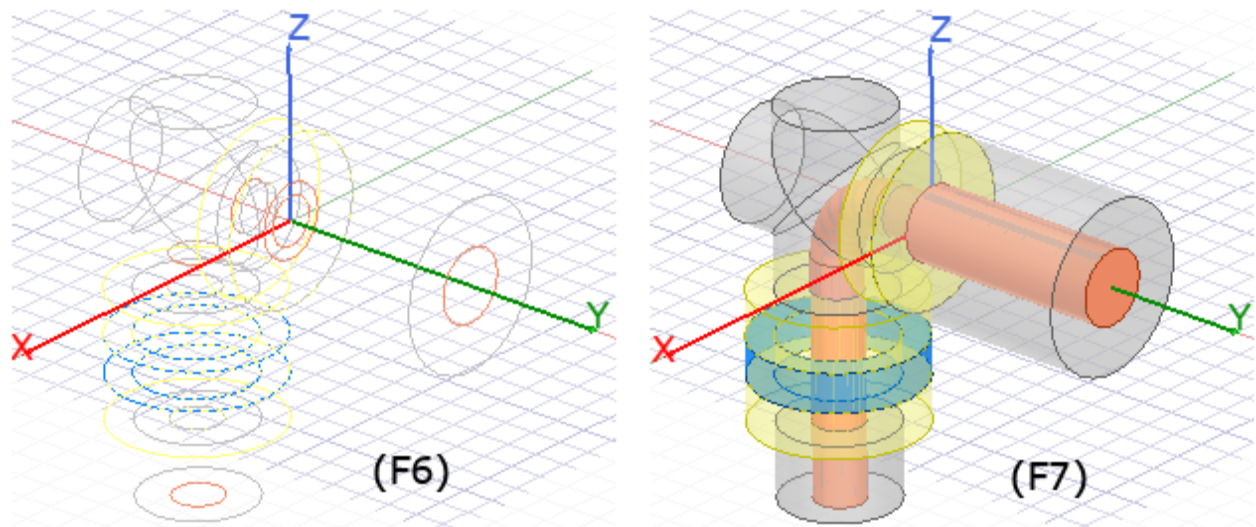


Figure 5-4: Effect of *F6* and *F7* Keys

Geometry Creation and Selection Shortcuts

The following view navigation shortcuts are presented below:

- F3
- F4
- B
- E
- F
- M
- O
- V

F3 and F4:

These keys control the definition mode when creating new objects (such as boxes, rectangles, polylines, and more). You have to press the **F3** or **F4** key while a drawing command is active to change the definition mode.

- **F3**: Specify coordinates graphically (by mouse clicks) or via the coordinate text boxes in the status bar:

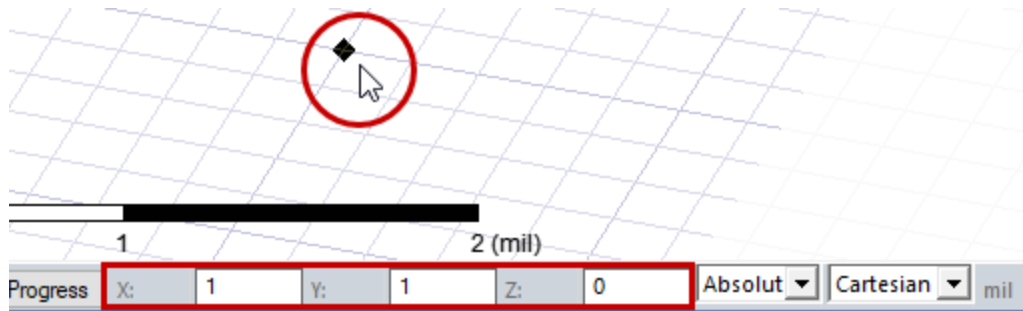


Figure 5-5: F3 Geometry Definition Mode – Mouse or Coordinate Text Boxes

- **F4:** Specify location and size in the associated *CreateObject* properties dialog box:

Command	Attribute		
Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	0 , 0 , 0	mm	0mm , 0mm , 0mm
XSize	0	mm	0mm
YSize	0	mm	0mm
ZSize	0	mm	0mm

Figure 5-6: F4 Geometry Definition Mode – CreateObject Dialog Box

B (Next Behind):

This shortcut key is very handy. Frequently, you will have to select an object or face that is hidden behind or within another object. One way to perform this task would be to hide the obstructing object, select the desired object, and then restore visibility of the obstructing object, but this is an inconvenient method.

Alternatively, click with the cursor pointed at the desired background entity, which causes the foreground entity to initially be selected. Then, press **B** to select the next entity *Behind* the initially selected one. If the desired object or face is behind two or more intervening objects or faces, simply press **B** repeatedly until you've drilled down to the desired entity.

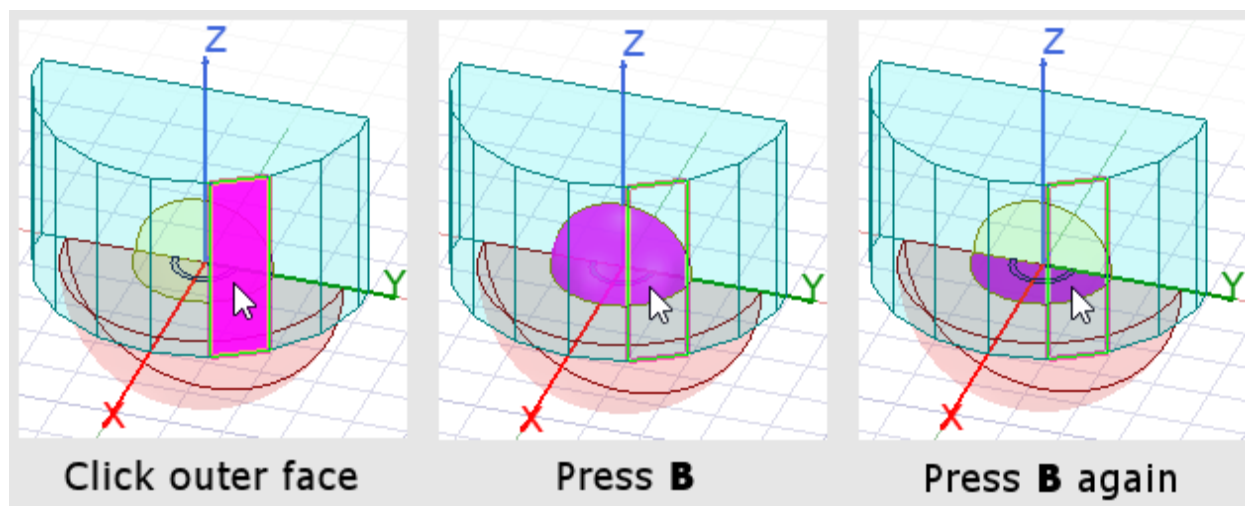


Figure 5-7: Using **B** Shortcut to Select Background Faces

E, F, O, V, and M (Selection Modes):

These four keyboard shortcuts control the type of model entity that is selected when you click on the model, as follows:

- **E:** Select *Edges*
- **F:** Select *Faces*
- **O:** Select *Objects*
- **V:** Select *Vertices*
- **M:** Select entities of any type, depending on where you click, as follows:
 - To select a vertex, click within a 10 pixel radius of the vertex.
 - To select an edge, click near an edge (and at least 10 pixels away from any vertex).
 - To select an object, click a little farther from any edge (between 10 and 20 pixels away).
 - To select a face, click anywhere else on the interior of a face. (You may have to zoom in to smaller faces so that no other foreground vertex or edge is within 20 pixels.)

Note:

When selecting two or more entities, subsequent selections must be of the same type as the first item clicked. Mixed selection types are not permitted.

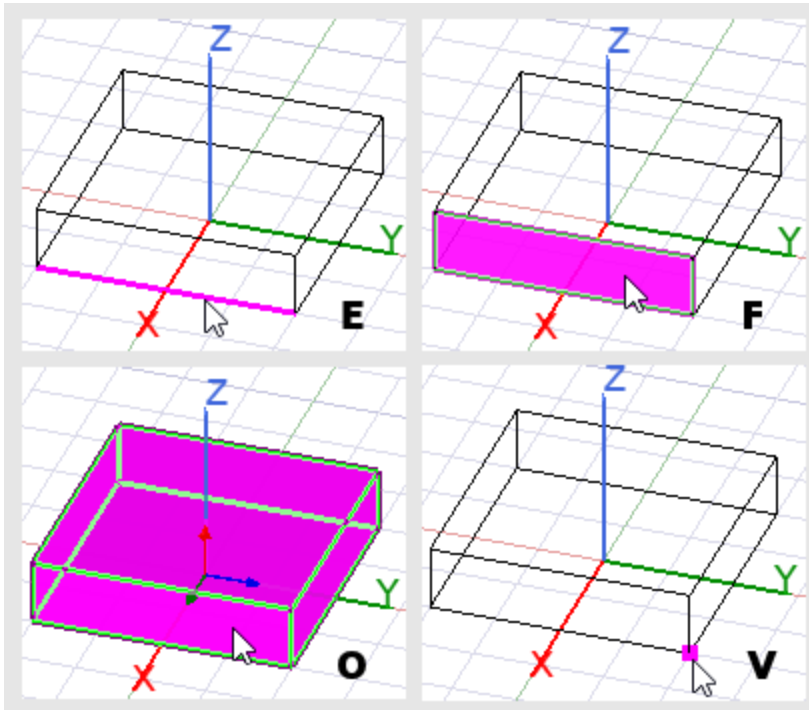


Figure 5-8: Edge (E), Face (F), Object (O), and Vertex (V) Selection Modes

Note:

The Modeler must be the currently active window for these keyboard shortcuts to work. Click anywhere in the drawing area before pressing a selection mode shortcut key if it doesn't have any effect on the first attempt.

Plot Shortcuts

The following shortcuts, applicable to plots, are presented below:

- Shift+Ctrl+N
- Ctrl+Insert
- Shift+Insert

Shift+Ctrl+N (Add Note):

You may want to place custom annotations on your result plots, such as a customer name, order number, or assumptions and parameters used in the modeling and analysis setup. To do so, press the **Shift+Ctrl+N** keyboard to access the *Add Note* dialog box :

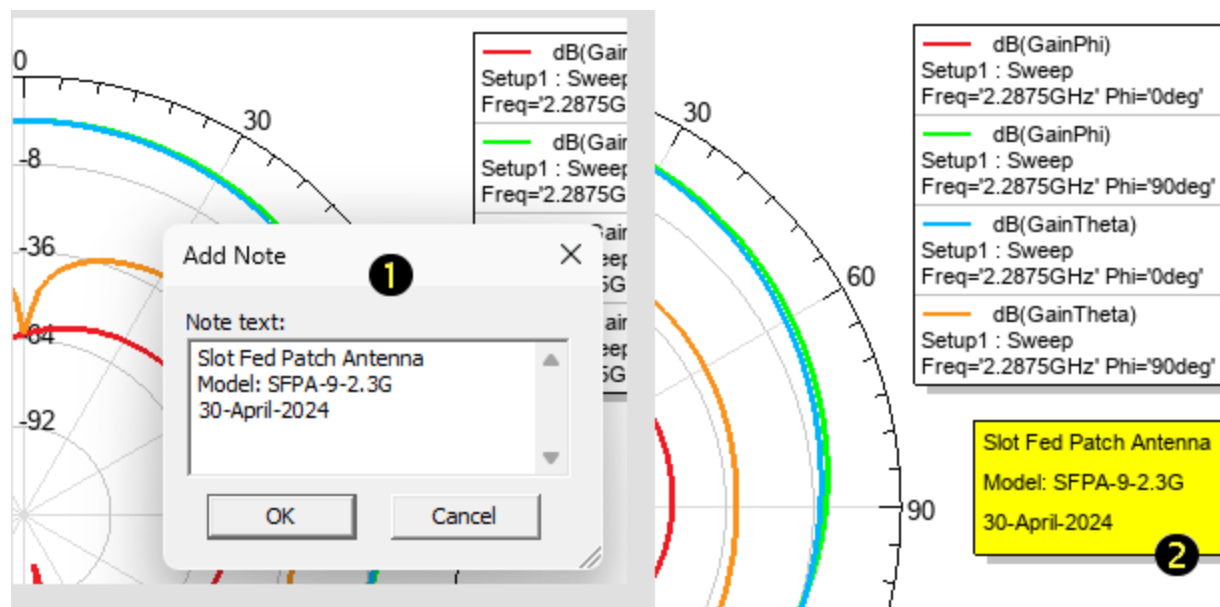


Figure 5-9: Shift+Ctrl+N (Add Note to Plot)

Ctrl+Insert and Shift+Insert:

After selecting a plot in the Project Manager, press **Ctrl+Insert** to copy it to the clipboard and **Shift+Insert** to paste a duplicate plot into the project. This method can be useful when subsequent plots share a majority of settings with an earlier plot. After pasting in the duplicate plot, edit its properties as needed for the results you want to graph.

The action of these two shortcuts is essentially the same as the standard **Ctrl+C**, **Ctrl+V** copy and paste operation, except that **Ctrl+Insert** and **Shift+Insert** shortcuts are only applicable to results plots within Ansys Electronics Desktop.

6 - Optionally, Restore Current View Orientations

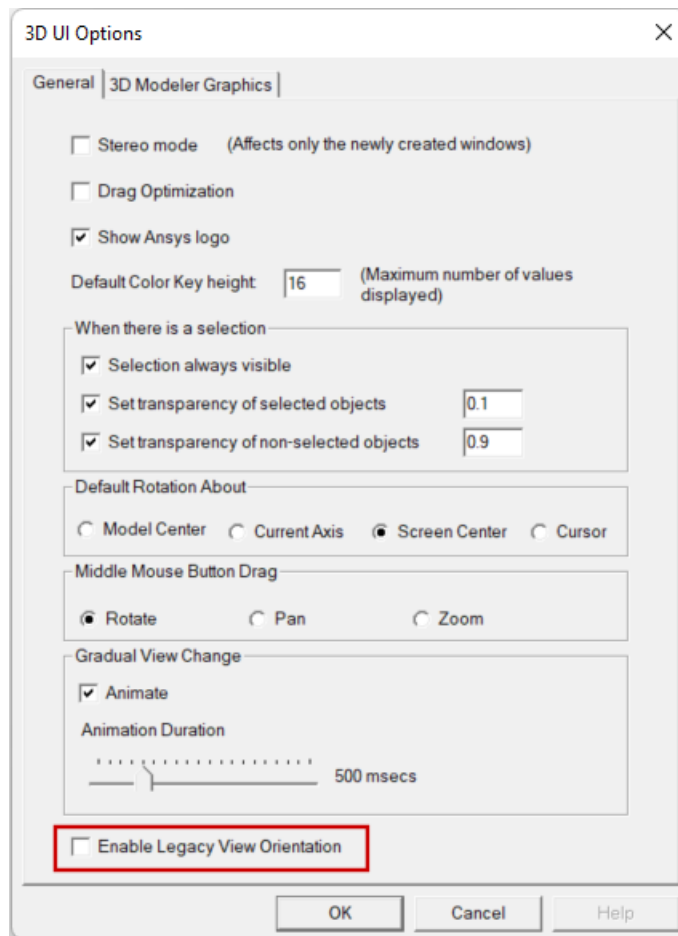
You have completed this getting started guide.

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.

A - Appendix: Default Keyboard Shortcuts

The following tables list the default keyboard shortcuts by category. Some shortcuts are supported for the 3D Modeler, 3D Layout Editor, and Plot windows, and some are applicable only to a particular window. Therefore, there are separate tables for the Modeler, Layout, and Plot windows. Separate tables are also provided for view navigation, drawing and selection, and general desktop function shortcuts.

Note:

If your Windows system or another application has reserved one of the Ansys Electronics Desktop keyboard shortcuts for another function, the key or key combination may not perform the function listed in the tables below. In such cases, you can choose a different key combination in one of the applications with conflicting keyboard shortcut assignments.

Table 2: Modeler Window – 3D View Shortcuts

Command	Function
Ctrl+D	Fit All
Ctrl+E, Ctrl++	View Zoom-in one step, center focus
Ctrl+F, Ctrl+-	View Zoom-out one step, center focus
Ctrl+H	View Hide selection
Shift+Ctrl+Z	View Previous
Shift+Ctrl+Y	View Next
Esc	Escape from Zoom (drag mode)
+	Step-wise zoom-in
-	Step-wise zoom-out
Alt+Double-Click	Changes the model viewpoint to one of nine standard planar or isometric projections depending on the cursor location when double-clicking (Center, N, NE, E, SE, S, SW, W, or NW).

Table 3: Modeler Window – 3D Drawing and Selection Shortcuts

Command	Function
Esc	Abort command in progress
F3	Data Entry Mode in the Status Bar fields
F4	Data Entry Mode in the Properties dialog
F6	Render wire frame to geometry
F7	Render smooth shade to geometry
B	Select Next Behind
E	Select Edges
F	Select Faces
M	Select Multi
O	Select Object
V	Select Vertices
Delete	Edit > Delete
Ctrl+A	Select All objects
Ctrl+Shift+A	Deselect All objects
Ctrl+C	Edit > Copy
Ctrl+X	Edit > Cut
Ctrl+V	Edit > Paste

Table 4: Layout Window – 3D View Shortcuts

Command	Function
Ctrl+D	Fit All
Ctrl+E, Ctrl++	View Zoom-in one step, center focus
Ctrl+F, Ctrl+-	View Zoom-out one step, center focus
Alt+Double-Click	Changes the model viewpoint to one of nine standard planar or isometric projections depending on the cursor location when double-clicking (Center, N, NE, E, SE, S, SW, W, or NW).

Table 5: Layout Window – 3D Drawing and Selection Shortcuts

Command	Function
Esc	Abort command in progress
B	Cycle Selection (between foreground and background objects)
E	Select Edges
H	Select Handles
O	Select Object
Delete	Edit > Clear
Ctrl+A	Select All objects
Ctrl+Shift+A	Deselect All objects
Ctrl+C	Edit > Copy
Ctrl+X	Edit > Cut
Ctrl+V	Edit > Paste
Ctrl+W	Add Connection (Wire)

Table 6: Plot Shortcuts

Command	Function
Ctrl+D	Fit All
Ctrl+H	View > Hide selection
Esc	Escape from Drag mode
Shift+Ctrl+N	Add Note
Ctrl+C	Edit > Copy Plot
Ctrl+Insert	Edit > Copy Plot
Ctrl+V	Edit > Paste Plot
Shift+Insert	Edit > Paste Plot

Table 7: Desktop General Shortcuts

Command	Function
Ctrl+0	Window > Cascade
Ctrl+1	Window > Tile Horizontally
Ctrl+2	Window > Tile Vertically
Ctrl+C	Edit > Copy
Ctrl+Insert	Edit > Copy
Ctrl+N	File > New
Ctrl+O	File > Open
Ctrl+P	File > Print
Ctrl+S	File > Save
Ctrl+V	Edit > Paste
Shift+Insert	Edit > Paste
Ctrl+X	Edit > Cut
Ctrl+Y	Edit > Redo
Ctrl+Z	Edit > Undo
Alt+Back	Edit > Undo
Delete	Edit > Clear
Shift+Delete	Edit > Cut
F1	Context-Sensitive Help
F2	Edit > Rename
F6	Next Pane
Shift+F6	Previous Pane