Ansys 2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

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

Getting Started with HFSS™: Waveguide T-Junction

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

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

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Conventions Used in this Guide

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

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

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

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

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

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

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

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

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

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

Getting Help: Ansys Technical Support

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

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

Help Menu

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

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

Context-Sensitive Help

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

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

Table of Contents

1 - Introduction

This document is intended as supplementary material to HFSS for beginners and advanced users. It includes instructions to create, solve, and analyze a T-junction.

 This chapter contains the following topics:

- Overview of the Interface
- Distinctive Features of HFSS
- Sample Project: The Tee Junction

What You Will Learn:

You will learn to perform the following HFSS tasks in the Ansys Electronics Desktop application:

- Set up the Tee project
- Create the Tee model
- Modify a model's design parameters
- Assign variables to a model's design parameters
- Specify solution settings for a design
- Validate a design setup
- Run an HFSS simulation.
- Create a 2D x-y plot of S-parameter vs. Frequency
- Create a field overlay plot of results
- Create a phase animation of results

Overview of the Interface

This topic provides an overview of the Ansys Electronics Desktop user interface in a typical configuration. You can enable or disable the display of individual windows to suit your usage and preferences. The following figure shows an HFSS design of a Tee Waveguide model within the interface:
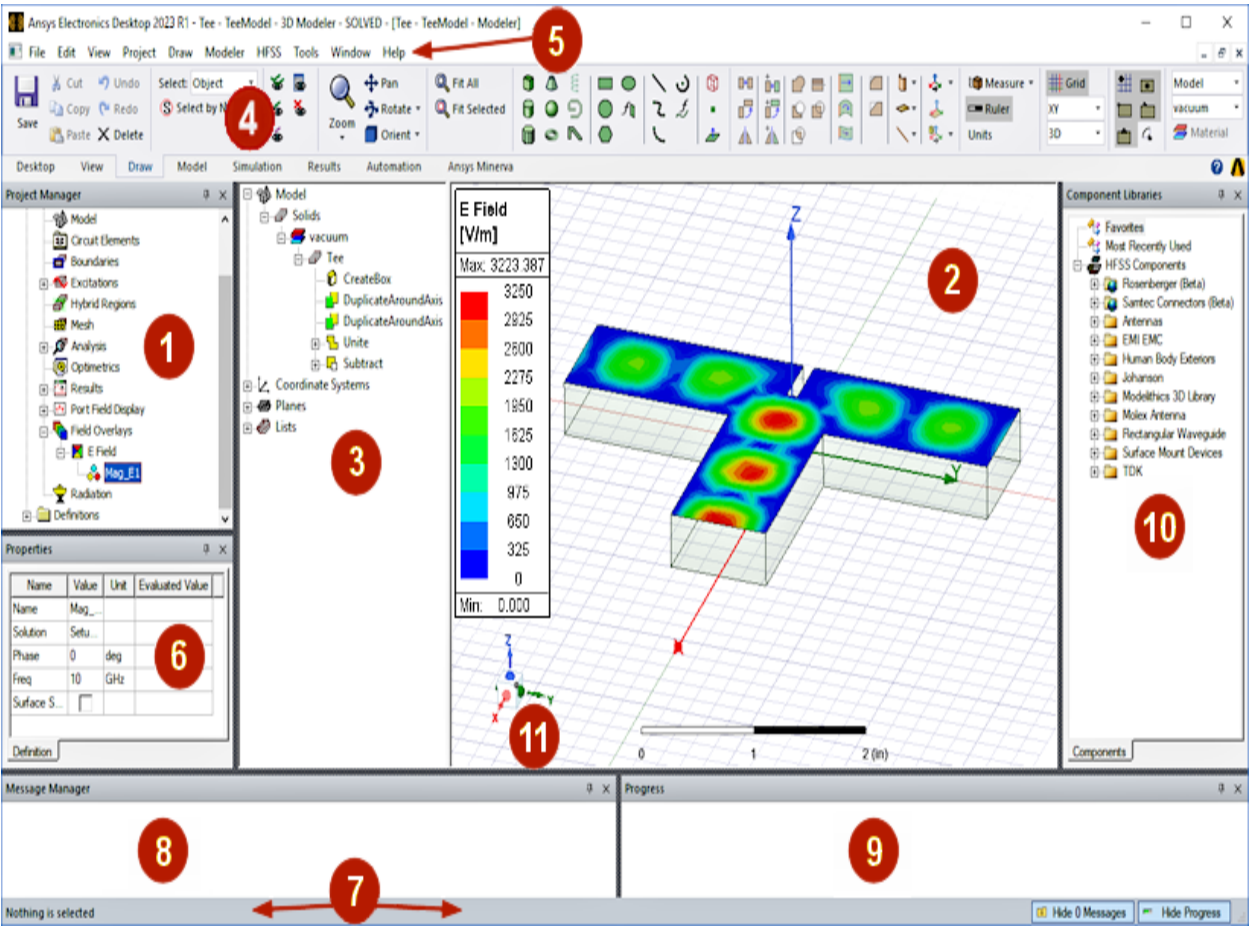
Figure 1-1: The Ansys Electronics Desktop User Interface – HFSS Design Example

The following table describes the different elements present within the user interface. You can control the visibility of individual windows from the menu bar by toggling the selection state of the corresponding options in the **View** menu.

1	*Project Manager* Window	Displays details of the design projects. Each project has its own main branch of the project tree, which ultimately includes various information and options (like the type of design, boundaries and excitations, analysis setups, and analysis results).
2	*Modeler* Window	Includes the drawing area of the active model and the *History Tree* (item 3). In this example, the E Field results of the simulation are overlaid on the model in the drawing area.
3	*History Tree*	Displays all operations and commands carried out on the active model (such as information about the model's objects, all actions associated with each object, and coordinate system information).
4	**Ribbon**	Contains various commands and options to create, manage, set up, analyze, and evaluate your designs and to customize program options. These commands and options are organized into related sets, with each logical group on a separate tab. The ordering of the tabs is designed to follow a left-to-right workflow. Certain popular commands (such as *Save* and various *View* options) appear on multiple tabs. Not all of the available commands can be accessed via the ribbon. Others are accessed from the *Menu Bar* (item 5) and shortcut menus that appear when you right-click.
5	**Menu Bar**	Provides various menus that enable you to perform all of the modeling, viewing, and HFSS tasks (such as managing project files, customizing the desktop components, drawing objects, and setting and modifying all project parameters).
6	*Properties* Window	Displays the attributes of a selected object in the active model (such as the object's name, material assignment, orientation, color, and transparency). This window also displays information about a selected command that has been carried out. For example, if a circle was drawn, the information about the command includes its name (*CreateCircle*), the type and name of the coordinate system the circle was drawn in, the coordinates of the circle's

		center position, its axis, and its radius.
		To differentiate this window from the similar *Properties* dialog box, it is typically referred to as the "docked *Properties* window" within the Getting Started Guides.
7	**Status Bar**	Shows current actions and provides instructions.
		Also, depending on the command being carried out, the status bar can display the X, Y, and Z (or dX, dY, and dZ) coordinate boxes. Use the **Absolute/Relative** drop-down menu to choose whether to enter a point's absolute or relative coordinates. Additionally, there's a drop-down menu to specify points in Cartesian, Cylindrical, or Spherical coordinates and the active model's unit setting.
8	***Message Manager* Window**	Displays errors, warnings, and informational messages for an active project.
9	***Progress* Window**	Displays solution progress information.
10	***Component Libraries* Window**	This window is one of several that you can optionally display or hide via the **View** menu. It displays 3D components that are stored in the SysLib and PersonalLib directories. From here you can easily add 3D components into your project.
		Other windows you can display here contain *Components, Layers,* and *Nets* information. Additionally, the *ACT Extensions* window (supported on Windows platforms only) has wizards to automate the creation of various models.
11	**Orientation Gadget**	This gadget is a graphical tool for quickly and easily manipulating the view orientation.

Distinctive Features of HFSS

The High-Frequency Structure Simulator (HFSS) offers some special features to help you create and work with your design and bring it into a desirable and effective condition. Some of the key features are as follows:

- **Numerous ways to perform similar tasks:**

  For example, you can select and assign design parameter values (variables), or you can work with fixed parameters (constants). Additionally, you can access commands via the

menu bar, the ribbon, or via shortcut menus that appear when you right-click. In the latter case, the contents of the menu depend on the context of where you right-click and what is selected at the time. Choose the tools that are the most comfortable or natural for you to use or that are best suited for a particular modeling task.

- **Create designs using your own discretionary workflow:**

You do not need to follow a rigid sequence of events to create your design. This guide shows a convenient method of creating the T-junction. However, you can execute the design steps in any logical order.

- **Quickly modify design properties:**

For example, you can draw a box freehand and then specify its exact dimensions in the docked *Properties* window. You can modify the properties at any time.

- **Track modifications easily:**

The *History Tree* and *Project Manager* help you keep track of your design modifications. The branches of these trees provide organized lists of design steps completed (such as applied boundaries, excitations, analysis setups, and created objects), and they provide access to setup dialog boxes, where you can modify design properties.

- **Take advantage of keyboard shortcuts:**

Special hotkey combinations, like **Ctrl+D** (to fit the model in the view window), and many others are available for frequently used commands. You can also create your own customized keyboard shortcuts.

For more information, see the separate *Getting Started* guide, *HFSS Keyboard Shortcuts*.

- **Parameterize design properties and save time:**

For example, in this *Getting Started* guide, you can assign a design variable to the septum's position. This design variable enables you to quickly modify the design and generate new results, facilitating "*what if*" analysis variants.

- **Optimetrics:**

Automate the process of optimizing your designs. Specify a cost function and optimization goals to achieve desired results in the most efficient way.

For more information, see the follow-up *Getting Started* guide, *HFSS Tee Waveguide Optimization*.

- **Evaluate results (post processing):**

Many 2D and 3D plots, overlays, and animations of simulation results are available. For example, in this sample model, the animations you create will help you visualize the difference in field pattern results for the two septum positions.

Sample Project – The Tee Junction

In this project you will design a T-junction with an inductive septum and analyze its behavior. This structure splits an incoming microwave signal into two outgoing signals. The septum divides the signal, directs it to the output ports, and minimizes reflection at its input port.

A high frequency (10 GHz) signal enters the waveguide at Port 1 (see below) and exits at Ports 2 and 3. The position of the septum determines the relative transmission to Ports 2 and 3 and the amount of signal reflection at Port 1.
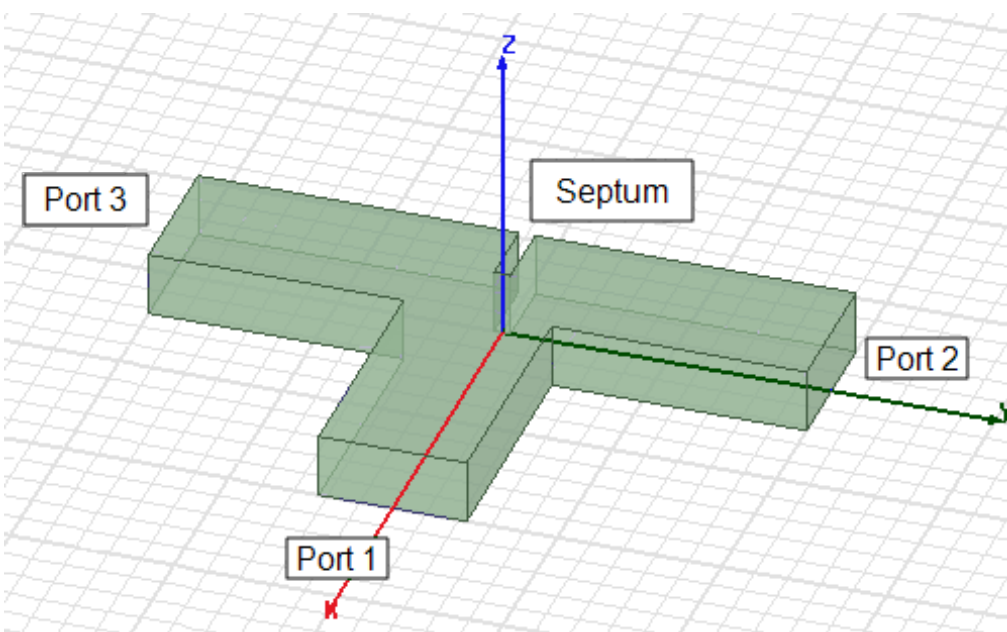
Figure 1-2: The Tee Junction Waveguide Model

Expected Results:

On the basis of prior simulations and product reviews (see footnote [1] at the bottom of this topic) the following results were observed at a solution frequency of 10 GHz:

1. If the septum is located centrally between output ports 2 and 3 (directly opposite from input port 1), the signal divides equally between ports 2 and 3 (S12 = S13 ≈ 0.7).

2. With the septum located centrally, the incidental reflection at port 1 (S11) was slightly more than 0.2.

3. If the septum is moved 0.2 inches closer to port 2, transmission through port 2 (S12) is reduced to slightly more than 0.4, and transmission through port 3 (S13) is increased to about 0.9.

4. At the same septum offset of 0.2 inches, the incidental reflection at port 1 (S11) decreases to slightly less than 0.1.

You will create a 2D x-y plot of the S-Parameters vs. Frequency for each septum position and verify that your results agree with the expected results. You can also compare the E-field pattern at each septum position by creating phase-animated field plots on the model geometry. These comparisons should indicate whether the field pattern changes with the septum's position as expected.

[1] "Parametrics and Optimization Using Ansoft HFSS," *Microwave Journal*, Product Reviews, November 1999.

2 - Set Up the Tee Project

This chapter contains the following topics:

- Launch Ansys Electronics Desktop
- Insert an HFSS Design
- Set the Unit of Measurement
- Set the Solution Type

Launch Ansys Electronics Desktop:

For convenience, a shortcut to the Ansys Electronics Desktop (EDT) application is placed on your desktop during program installation. Optionally, you may want to pin the shortcut to your Windows Start Menu too. Before proceeding to the next topic, launch EDT and add a blank project, as follows:

1. Double-click the **EDT** **Ansys Electronics Desktop** shortcut on your desktop (or the same shortcut on your Start Menu).

   The *Ansys Electronics Desktop* application opens.
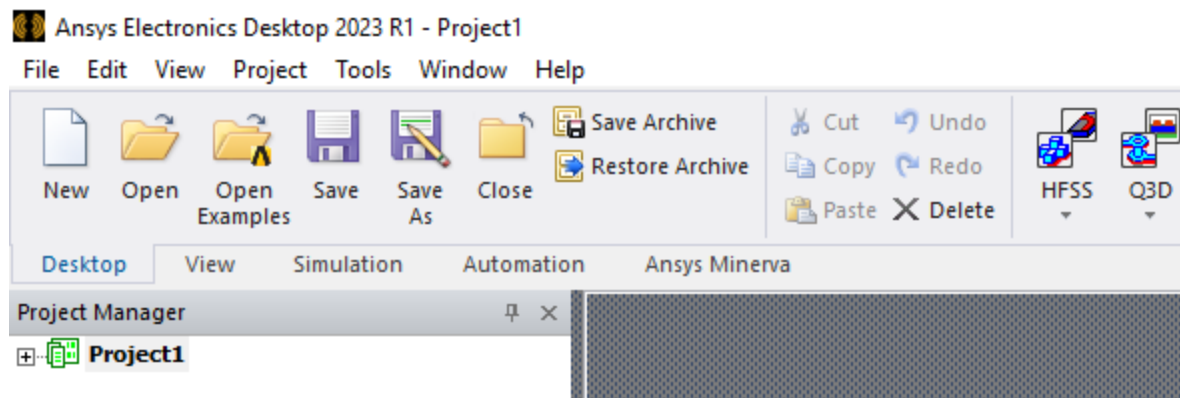
Figure 2-1: Ansys EDT Application Launched

2. 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.

> **Note:**
>
> Normally, a new, project is added automatically when you launch EDT. If you had the application open already and closed the model you were working on, you will have to add a new project manually.

3. Right-click **Project***x* at the top of the Project Manager and select **Rename** from the short-cut menu.

4. Type **Tee** and press **Enter**.

   The file *Tee.aedt* is saved to your default projects folder.

Insert an HFSS Design

1. Use one of the following methods to insert an HFSS design into the project:

   - From the **Desktop** ribbon tab, click [icon] **HFSS**.

     > **Note:**
     >
     > There is no need to access the *HFSS* drop-down menu in this ribbon tab. The default action is to insert a regular HFSS design.

   - From the menu bar, click **Project** > **Insert HFSS Design**:
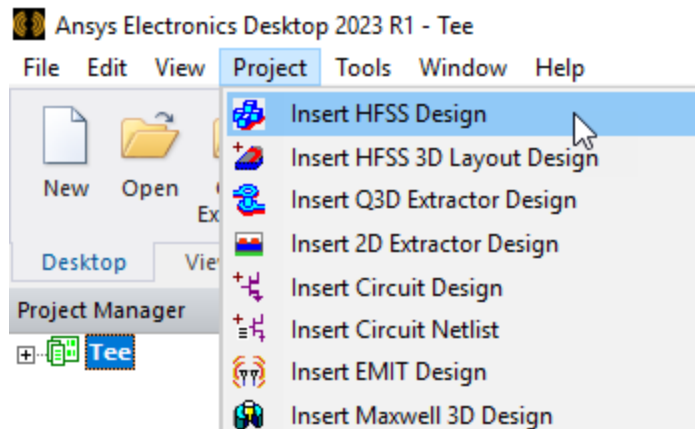
   "HFSSDesign*x*" appears in the Project Manager.

2. Expand the **HFSSDesign***x* branch of the Project Manager.

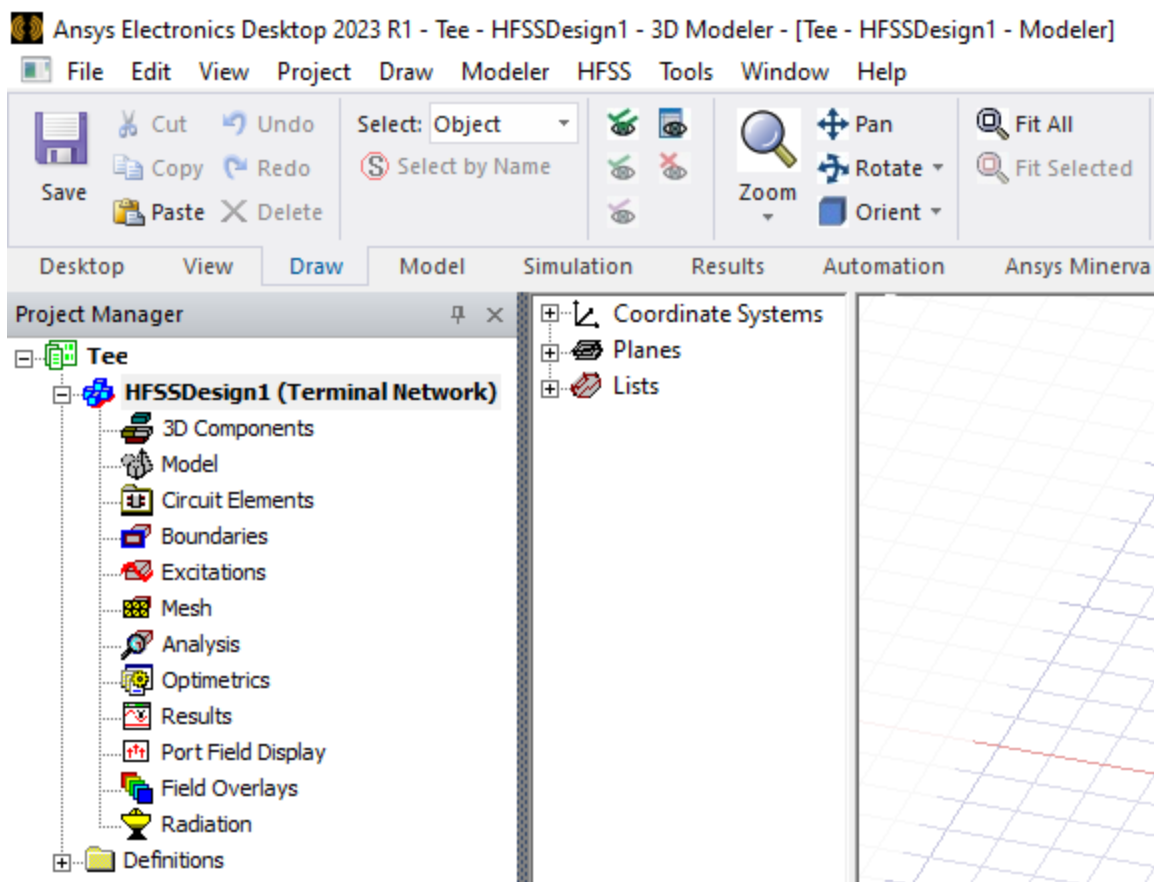   Your screen should resemble the following figure:

Figure 2-2: "HFSS Design*x*" Inserted into Tee Project

3. On any ribbon tab, click ![save] **Save**.

   It's a good idea to save your model frequently as you add geometry, boundaries, excitations, parameterizations, or analysis setups. By default, the program will autosave the model after each set of ten (10) editing operations, and you can customize the autosave interval (*General Options*).
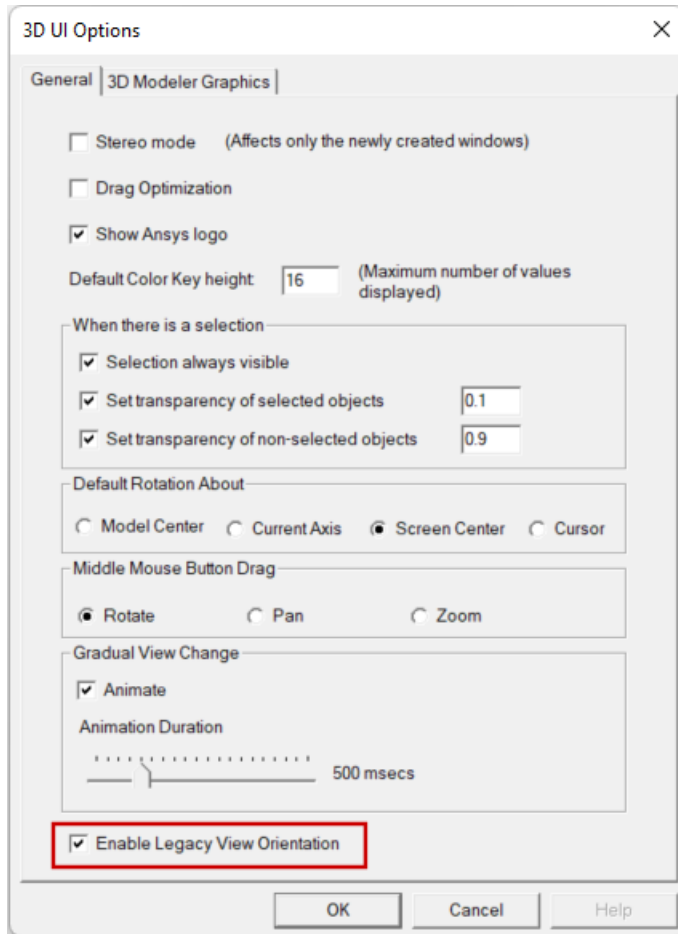
Enable Legacy View Orientations

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 the Units of Measurement

Set the length unit to *inches* for drawing the *Tee* model, as follows:
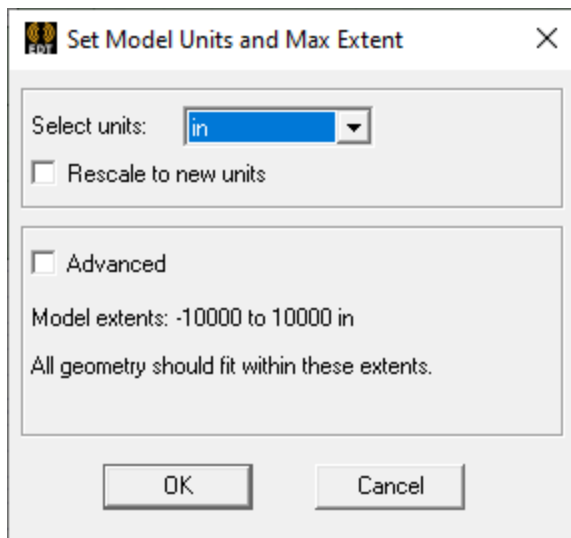
1. On the **Draw** ribbon tab, click **Units**:
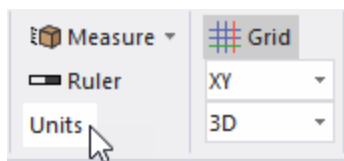
Figure 2-3: Location of *Units* Command on *Draw* Ribbon Tab

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

2. Select **in** (inches) from the **Select units** drop-down menu.

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

Figure 2-4: *Set Model Units and Max Extent* Dialog Box
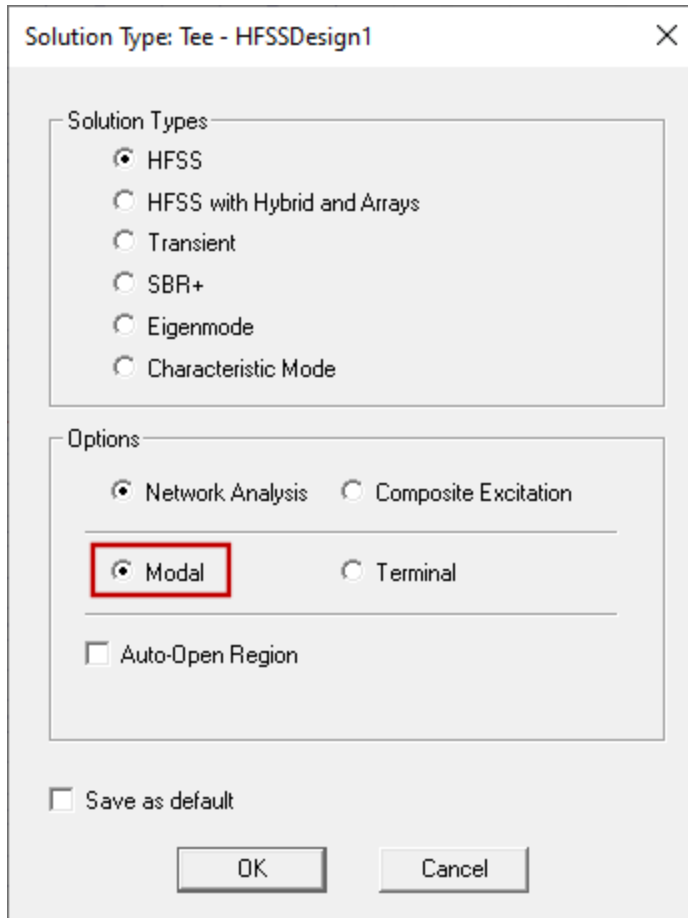
3. Click **OK**.

Set the Solution Type

The settings that HFSS offers in the user interface depend on the chosen solution type. For the Tee waveguide junction, select the *HFSS* solution type and the *Modal* and *Network Analysis* options, as follows:

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

   The *Solution Type* dialog box appears.

2. Under *Options*, select **Modal**. Also, verify that the remaining settings match the following image:

Figure 2-5: *Solution Type* **Dialog Box**

3. Click **OK**.

   This option ensures that HFSS will calculate the modal-based S-parameters using the driven Modal method. The S-matrix solutions will be expressed in terms of the incident and reflected powers of the waveguide modes (or natural resonances).

   > **Note:**
   >
   > The convergence criterion for Modal solutions is the maximum Delta S (change in modal S-matrix values) between consecutive iterations.

Description of Solution Types and Associated Options:

- **Modal:** This option is applicable to *HFSS* and *HFSS with Hybrid and Arrays* solution types. It is used for calculating the mode-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines which are "driven" by a source, and for computing incident plane wave scattering.

- **Terminal:** This option is applicable to *HFSS* and *HFSS with Hybrid and Arrays* solution types. It is used for calculating the terminal-based S-parameters of passive, high-frequency structures with multi-conductor transmission line ports which are "driven" by a source. This solution type results in a terminal-based description in terms of voltages and currents.

- **Transient:** Used for calculating problems in the time-domain (transient, short-duration events). Use it for simulations with pulsed excitations (such as ultra-wide band antennas, lightning strikes, and electrostatic discharge).

- **Eigenmode:** Used to calculate resonant frequencies of structures (for example, filters) and the fields at those frequencies. Frequency-dependent design parameters are not allowed.

- **Characteristic Mode:** Used to calculate the characteristic modes (equiphase current modes) of a conducting or dielectric structure, which are independent of any excitation. Only discrete sweeps and lossless boundaries are supported. Results produced include characteristic value, characteristic current, characteristic angle, and modal significance. The results are useful for problems involving syntheses, analyses, and optimization of antennas and scatterers.

- **SBR+:** Used to simplify design creation for SBR+ analyses (Shooting and Bouncing Ray technology). A typical application is to determine the reflection signature of radar targets. Ports and infinite ground planes are not allowed. At least one Incident Plane Wave excitation must be defined. Yo do not have to define Hybrid Regions. Far field results are calculated from current sources and the defined geometry.

3 - Create the Tee Model

This chapter contains the following topics:

- Draw a Box
- Alternative Method
- Assign a Wave Port to the Box
- Duplicate the Box
- Duplicate Boundaries and Mesh Operations
- Create the Second Section
- Create the Third Section
- Unite the Three Boxes
- Create the Septum
- Subtract Septum from T-junction

How To Draw the T-Junction:

The T-junction is made up of three box objects that are united through a Boolean operation and a fourth box used to cut the septum. The geometry creation workflow is summarized as follows:

- Draw a box to represent a section of the tee.
- Assign a wave port to one of its faces.
- Duplicate the box and wave port twice to create the second and third sections of the tee.
- Unite the three sections to complete the Tee.
- Draw a much smaller box and subtract it from the Tee to create the septum.

> **Note:**
>
> You will assign a design variable to control the septum position. This step will facilitate changing the position of the septum along the Y-axis, which will affect the relative signal transmission to the second and third ports of the tee.

The ensuing topics detail the above-summarized construction process.

Draw a Box

Before drawing any objects, check the 3D Modeler's Drawing options to ensure that the *Properties* dialog box appears whenever you create a new object. Then, you can draw a box object to represent the first section of the Tee.

Set the Desired Drawing Options:

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

2. In the tree on the left side of the *Options* dialog box, expand the **3D Modeler** branch and select the **Drawing** subheading.

3. Verify that the last option listed, **Edit properties of new primitives**, is selected and then click **OK**.
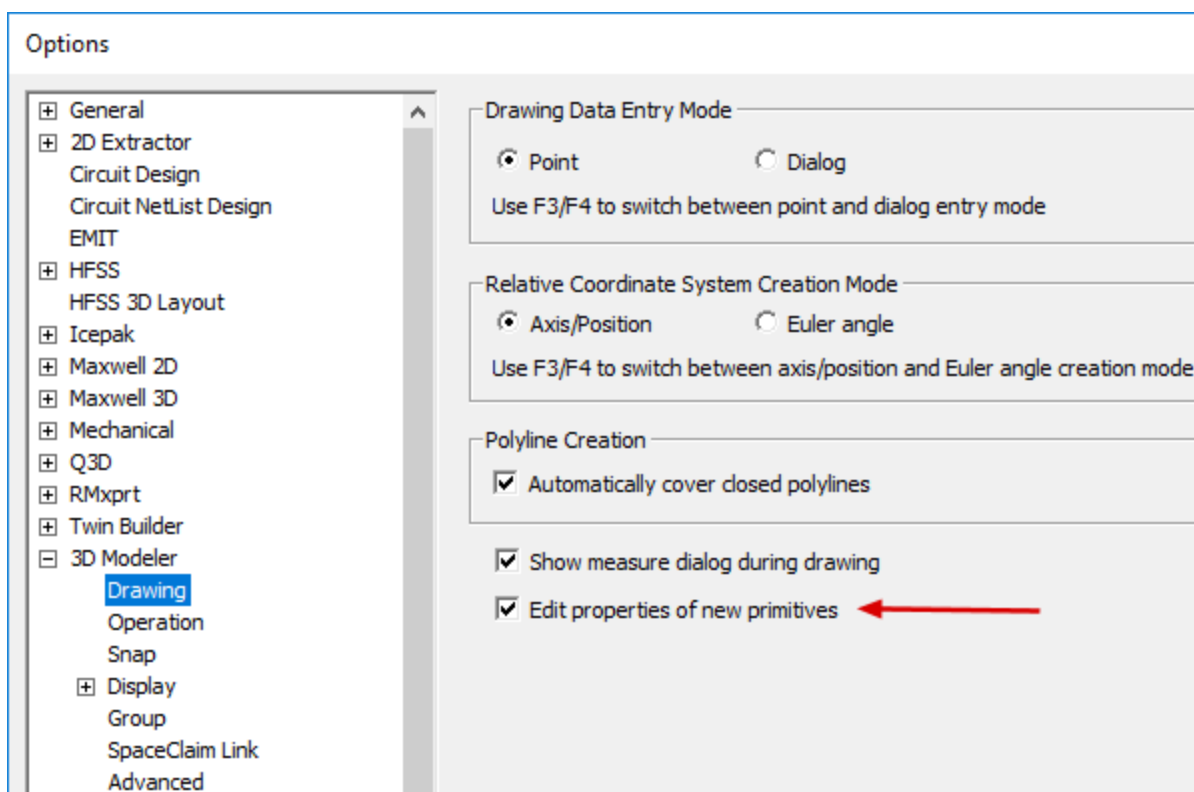
Figure 3-1: *Options* **Dialog Box –** *3D Modeler > Drawing* **Options**

Note:

When this option is selected, you can edit the properties of the objects in your design within the *Properties* dialog box as soon as you create each one of them. This method is an alternative to selecting items in the History Tree and editing the properties in the docked *Properties* window.
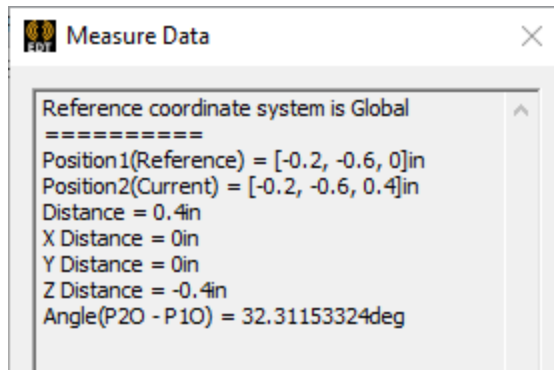
Draw the First Box:

4. On the **Draw** ribbon tab, click 🗍 **Draw box**.

5. Press **F3** to specify the object size and location graphically or via the coordinate text boxes in the status bar.

> **Note:**
>
> You only need to press this key the first time you draw an object. The preference is retained for future operations.
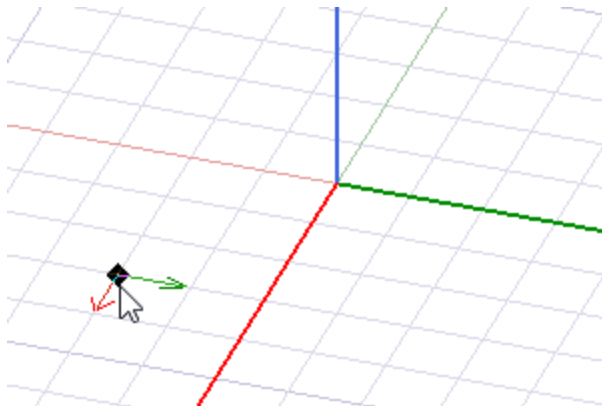
The *Measure Data* window appears and updates as you move the cursor.

Figure 3-2: *Measure Data* **Window**

6. Click an arbitrary point on the grid in the Modeler window to establish the starting corner position of the box.

Mini xy axes appear as shown below:

Figure 3-3: Mini XY Axes

7. Drag the mouse to form a rectangle and click again.

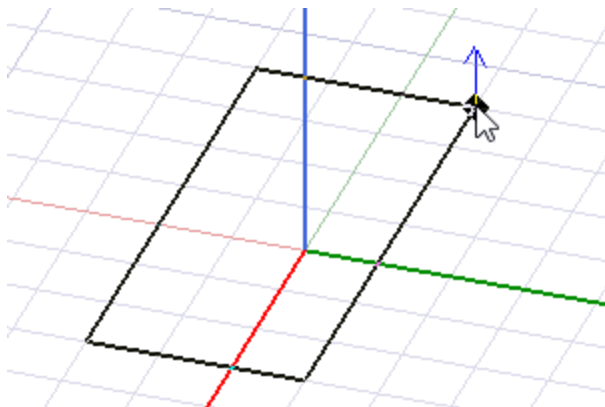An arrow pointing upwards along the z-axis prompts you to set the height:

Figure 3-4: The Mini Z-axis to Set Height

8. Drag the mouse along the z-axis and click once more to create the box.

    The *Properties* dialog box appears.

9. In the **Command** tab of the *Properties* dialog box, enter values of **Position, XSize, YSize**, and **ZSize** as shown in the following figure:

Properties: Project116 - HFSSDesign1 - Modeler

Name	Value	Unit	Evaluated Value	Description
Command	CreateBox			
Coordinate Sys...	Global			
Position	0 ,-0.45 ,0	in	0in , -0.45in , 0in	
XSize	2	in	2in	
YSize	0.9	in	0.9in	
ZSize	0.4	in	0.4in	

**Figure 3-5: *Properties* Dialog Box – *Command* Tab**

10. Select the **Attribute** tab, change the **Name** to **Tee**, and press **Enter**

11. Verify that the **Material** value is **vacuum** and the **Material Appearance** option is *not* selected.

> **Note:**
>
> The default material is *vacuum*.

12. Click the button to the right of **Transparent**. Then, in the *Set Transparency* dialog box that appears, set the value to **0.4**, as shown below, and click **OK**.
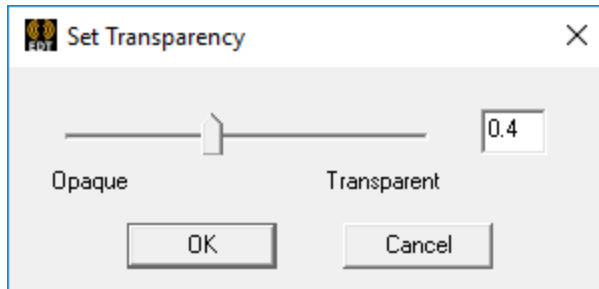
Figure 3-6: Set Transparency dialog box

The *Attribute* tab values should now match the following figure:

Figure 3-7: *Properties* Dialog Box – *Attributes* Tab

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

The first section of the *Tee* in the Modeler window should now resemble the following figure:
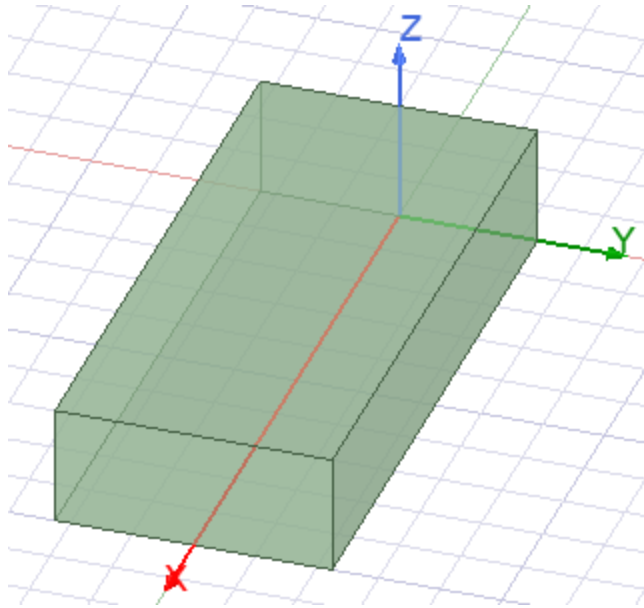
Figure 3-8: The First Section of the *Tee* Structure

Alternative Method

In the previous topic, you drew a box of arbitrary location and size and then edited its properties to produce the desired box. Another method of drawing the box and setting its coordinates and dimensions is as follows:

1. Verify that the status bar at the bottom of the EDT application window has one of the following two messages displayed:
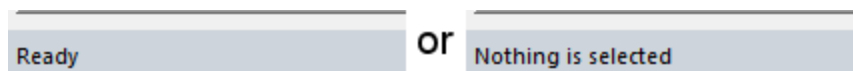

    Figure 3-9: Status Bar Messages when Ready to Draw an Object

2. On the **Draw** ribbon tab, click 🗋 **Draw box**.

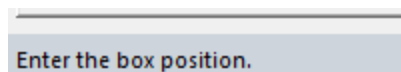   The status bar displays a new message:

    Figure 3-10: Prompt to Specify Box Position

3. Press **F3** to specify the object size and location graphically or via the coordinate text boxes in the status bar.

> **Note:**
>
> You only need to press this key the first time you draw an object. The preference is retained for future operations.

4. Ensure that the indicated length unit at the far right end of the status bar is **in** (inches), as previously set.

5. **Tab** into the X, Y, and Z coordinate entry text boxes near the right end of the status bar and type in the following coordinates:

   - **X: 0**
   - **Y: -0.45**
   - **Z: 0**

X:	0	Y:	-0.45	Z:	0	Absolut ▼	Cartesian ▼	in

Figure 3-11: Coordinate Text Boxes – Box Position

6. Press **Enter**.

   The status bar message changes as follows:

   Enter the opposite corner of the base rectangle.
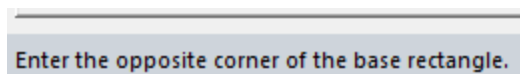
Figure 3-12: Prompt for Opposite Corner of Base Rectangle

7. **Tab** into the dX, dY, and dZ text boxes and type in the following coordinates, which specify the *XSize* and *YSize* of the box:

   - **dX: 2**
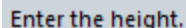   - **dY: 0.9**
   - **dZ: 0**

dX:	2	dY:	0.9	dZ:	0	Relative ▼	Cartesian ▼	in
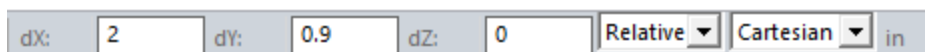
Figure 3-13: Coordinate Text Boxes – Opposite Corner Relative Location

> **Tip:**
>
> If you enter a non-zero *dZ* value for the opposite corner (in this case, *0.4*), you can also simultaneously specify the height of the box, eliminating the necessity of completing steps 9 and 10.

8. Press **Enter**.

The status bar message changes as follows:

Enter the height.

Figure 3-14: Prompt for Box Height

9. **Tab** into the dX, dY, and dZ text boxes and type in the following coordinates, which specify the *ZSize* (height) of the box:

    - **dX: 0**
    - **dY: 0**
    - **dZ: 0.4**

| dX: | 0 | dY: | 0 | dZ: | 0.4 | Relative ▼ | Cartesian ▼ | in |

Figure 3-15: Coordinate Text Boxes – Height of Box

10. Press **Enter**.

    The *Properties* dialog box appears.

   The values in the *Command* tab are already properly defined, since you specified the coordinates numerically.

11. Return to [step 9 of the prior *Draw a Box* topic](#) and complete steps 9 through 12 to define the box attributes. The process of editing the attributes is the same as for the first box-drawing method described.

Assign a Wave Port to the Box

You will assign a wave port to the +X face of the box. You will also define an integration line, which is a vector that specifies the direction of the excitation field pattern at the port. Integration lines ensure that the field pattern is consistent at all ports.

Figure 3-16: Face of the box parallel to the yz plane at x=2

1. Click in the modeler window and press **F** to begin the *face selection* mode.
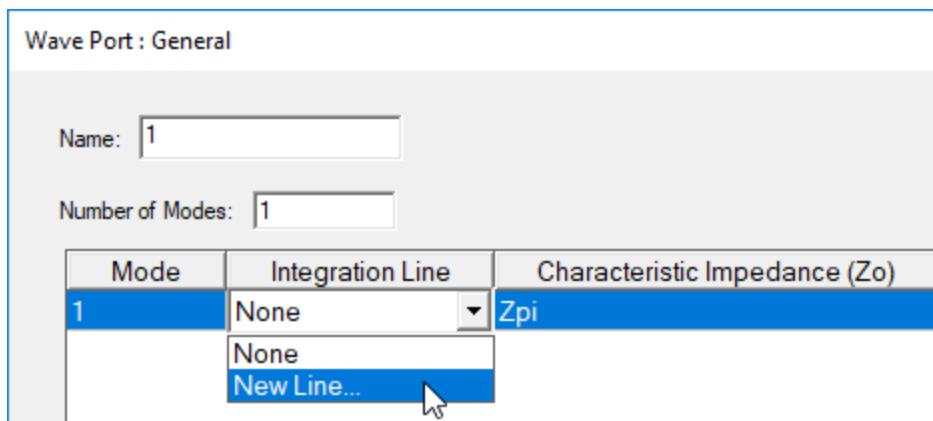2. Click the face at the **+X** end of the box to select it:
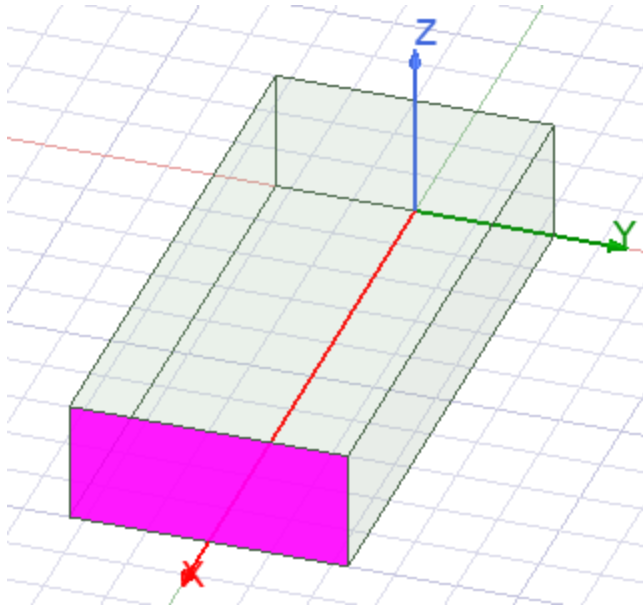
Figure 3-17: Wave Port Face Selected

3. Right-click the *Modeler* window and choose **Assign Excitation** > **Port** > **Wave Port** from the shortcut menu.
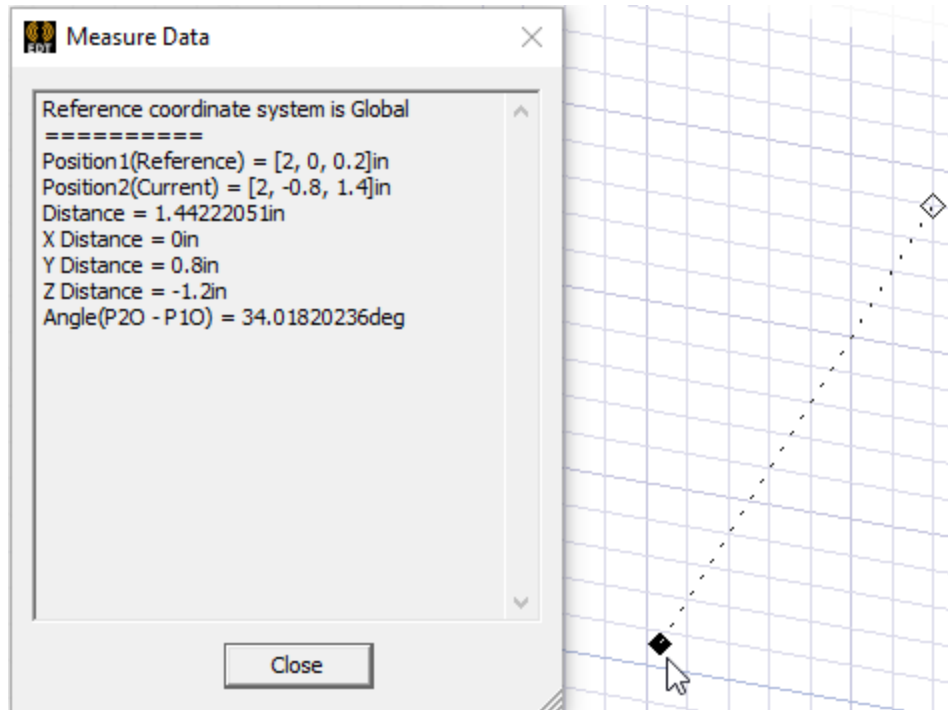
   The *Wave Port : General* dialog box appears.

4. Type **Port1** in the **Name** text box.

5. Select **New Line** from the drop-down menu in the **Integration Line** column, as shown below:


Figure 3-18: Selecting *New Line* Command for Integration Line

The *Measure Data* dialog box appears, and the cursor is accompanied by a dotted line.

Measure Data

Reference coordinate system is Global
==========
Position1(Reference) = [2, 0, 0.2]in
Position2(Current) = [2, -0.8, 1.4]in
Distance = 1.44222051in
X Distance = 0in
Y Distance = 0.8in
Z Distance = -1.2in
Angle(P2O - P1O) = 34.01820236deg

Close

Figure 3-19: *Measure Data* **Dialog Box and Integration Line Cursor**

6. Select the starting point of the vector (**2, 0, 0**), by clicking the center of the selected face's bottom edge. By default, the cursor should appear as a triangle and snap to this point:
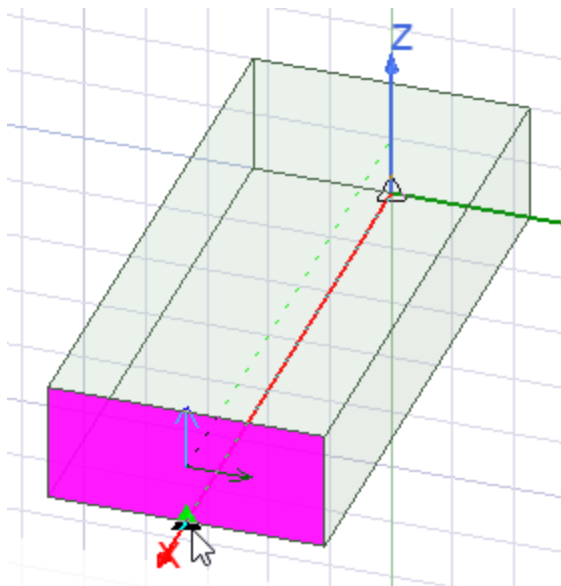
**Figure 3-20: Integration Line Starting Point**

7. Select the end point (**2, 0, 0.4**) by clicking the center of the selected face's top edge:
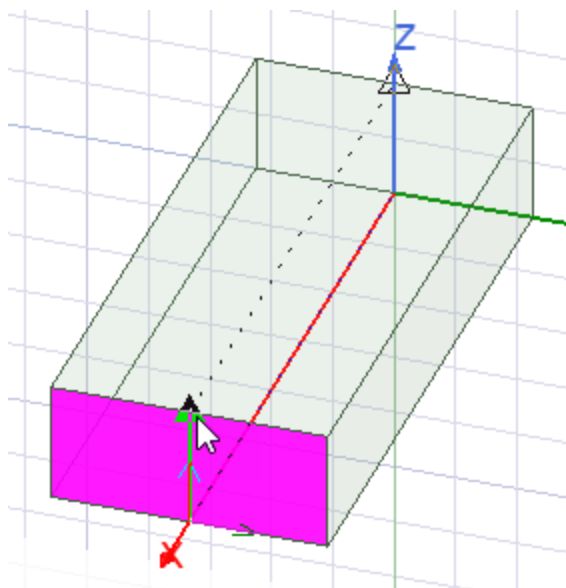
Figure 3-21: Integration Line Ending Point

The *Wave Port : General* dialog box reappears.

8. Click **Next** to accept the remaining default settings.
9. Click **Finish**.
10. Under *Excitations* in the Project Manager, select **Port1** to display its visualization on the model:
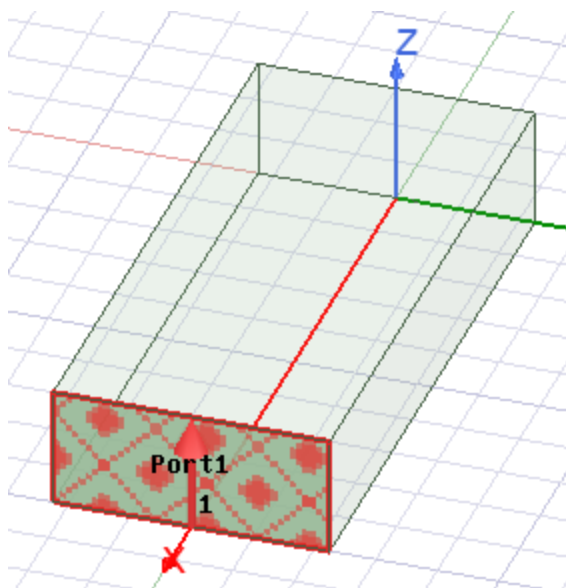
Figure 3-22: The Assigned Wave Port

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

Duplicate the Box and Excitation

You must duplicate the box twice to create the second and third sections of the Tee. For convenience, you can also duplicate the boundary assignments, excitations, and any mesh operations that have been defined along with duplicating the geometry. A program option controls whether these extra items are copied with the geometry. Before duplicating the box, you will select the needed option.

For the subject Tee, you will copy the box object and the wave port that was assigned to its face (including the integration line and any other port settings). The names of the duplicated object and wave port will be incremented automatically.

Enable Duplication of Boundaries and Mesh Operations with the Geometry

1. On the **Desktop** ribbon tab, click ▧ **General Options**.
2. Expand the *HFSS* group and select the **Boundary Assignment** subitem.
3. Select the **Duplicate boundaries/mesh operations with geometry** option:
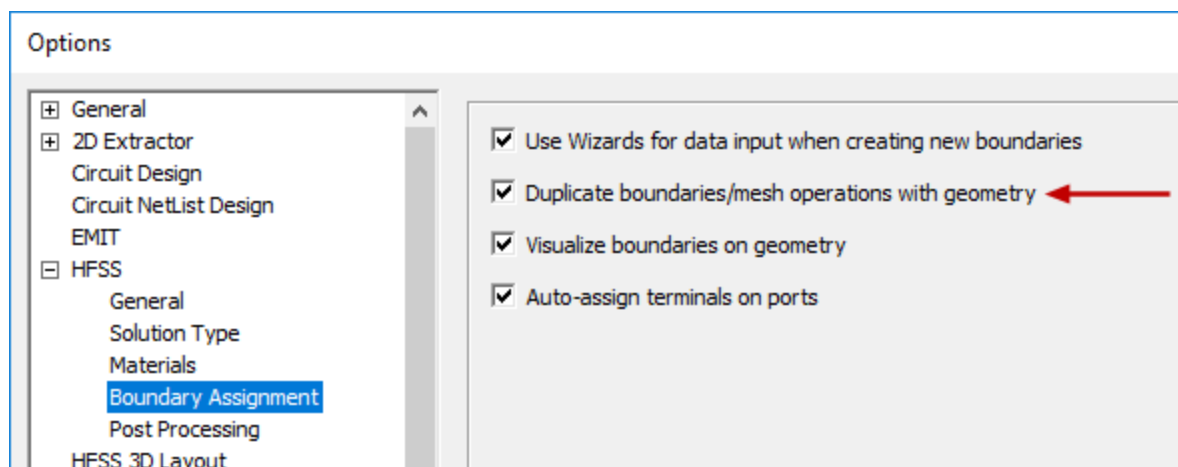
Figure 3-23: *HFSS > Boundary Assignment* **Options**

4. Click **OK**.

Create the Second Section

You will duplicate the first box to create the second section.

5. Under *Model > Solids > vacuum* in the History Tree, click **Tee** to select the object.
6. On the **Draw** ribbon tab, click 🗗 **Around Axis** (Duplicate Around Axis).

   The *Measure Data* and *Duplicate Around Axis* dialog boxes appear.

7. Ensure that the settings in the *Duplicate Around Axis* dialog box match the following specifications and figure:

    - **Axis: Z**
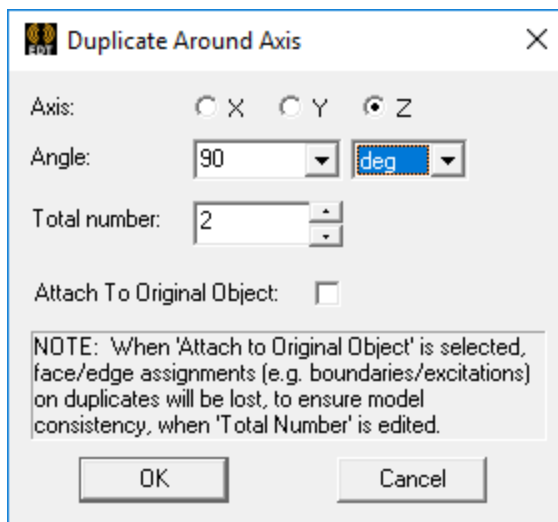   - **Angle: 90 deg**

    > **Note:**
    >
    > A positive angle causes the copied object to be rotated in the counter-clockwise direction about the specified axis.

    - **Total number: 2**

    > **Note:**
    >
    > This value is the total number of objects after duplication, including the original one.

    - **Attach To Original Object: Not selected**

Figure 3-24: Settings for the First Duplication

8. Click **OK** to create the duplicate box and excitation.
9. Click **Cancel** to dismiss the *Properties* dialog box that appears.
10. Press **Ctrl+D** to fit the view, if needed.

    The preview of the duplicate box in the Modeler window should look like the following figure:
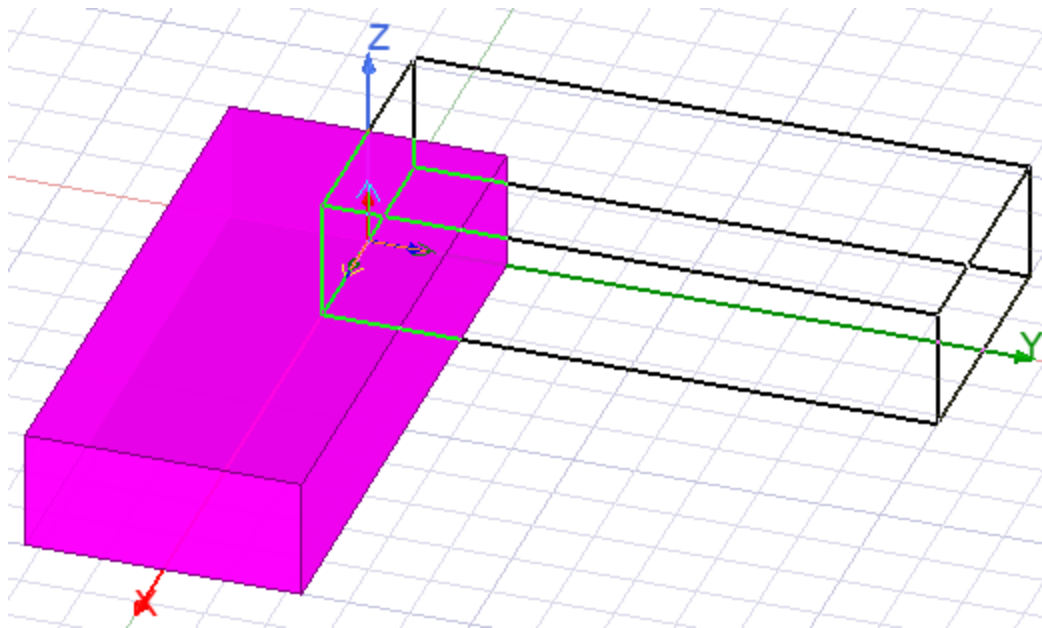
Figure 3-25: Preview of the First Box Duplication

11. Click **OK**.

    The box is duplicated, the duplicated wave port and its integration line are displayed, and the *Properties* dialog box appears.

12. Click **OK** to dismiss the *Properties* dialog box.

13. Clear the current selection.

Note:

The first box (*Tee*) was duplicated at the specified location, and the rotated copy named *Tee_1*. The attributes of the parent object (including its dimensions, material, color, and transparency) were duplicated. *Port1* was also duplicated along with its integration line. The duplicate was named *Port2* and listed under *Excitations* in the Project Manager.


Create the Third Section

1. Using the same procedure as detailed in the previous topic, duplicate the first box (*Tee*) again, but this time, specify **-90 deg** for the **Angle**.

> **Note:**
>
> A negative angle causes the copied object to be rotated in the clockwise direction about the specified axis.

2. Press **Ctrl+D** to fit the objects within the view window.

3. Clear the current selection.

   Your model should now look like the following figure:
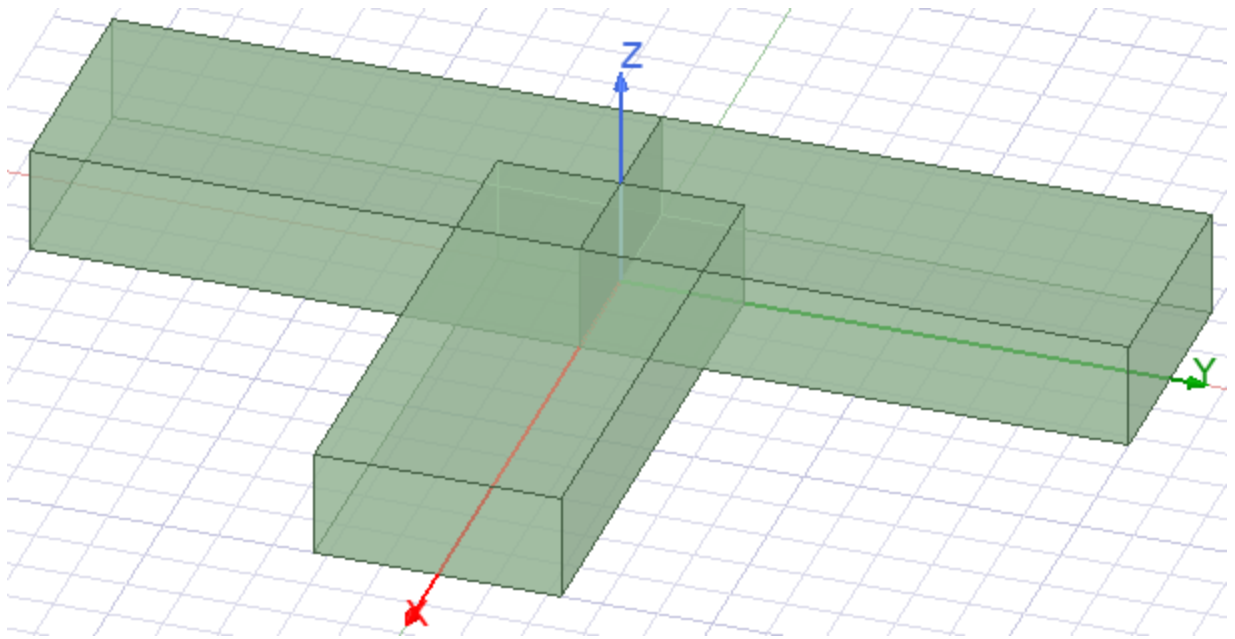
Figure 3-26: The *Tee* Model after Duplications are Completed

In the next procedure, you will unite the three separate boxes into a single object.

Unite the Three Boxes

To create the complete T-junction, you must unite the three separate boxes into a single object. Before doing so, there's a program option that you must verify is disabled.

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

   The *Options* dialog box appears.

2. Expand the *3D Modeler* group and select the **Operation** subitem.

3. Verify that the options, **Clone tool objects before uniting** and **Clone tool objects before subtracting**, are **not** selected.
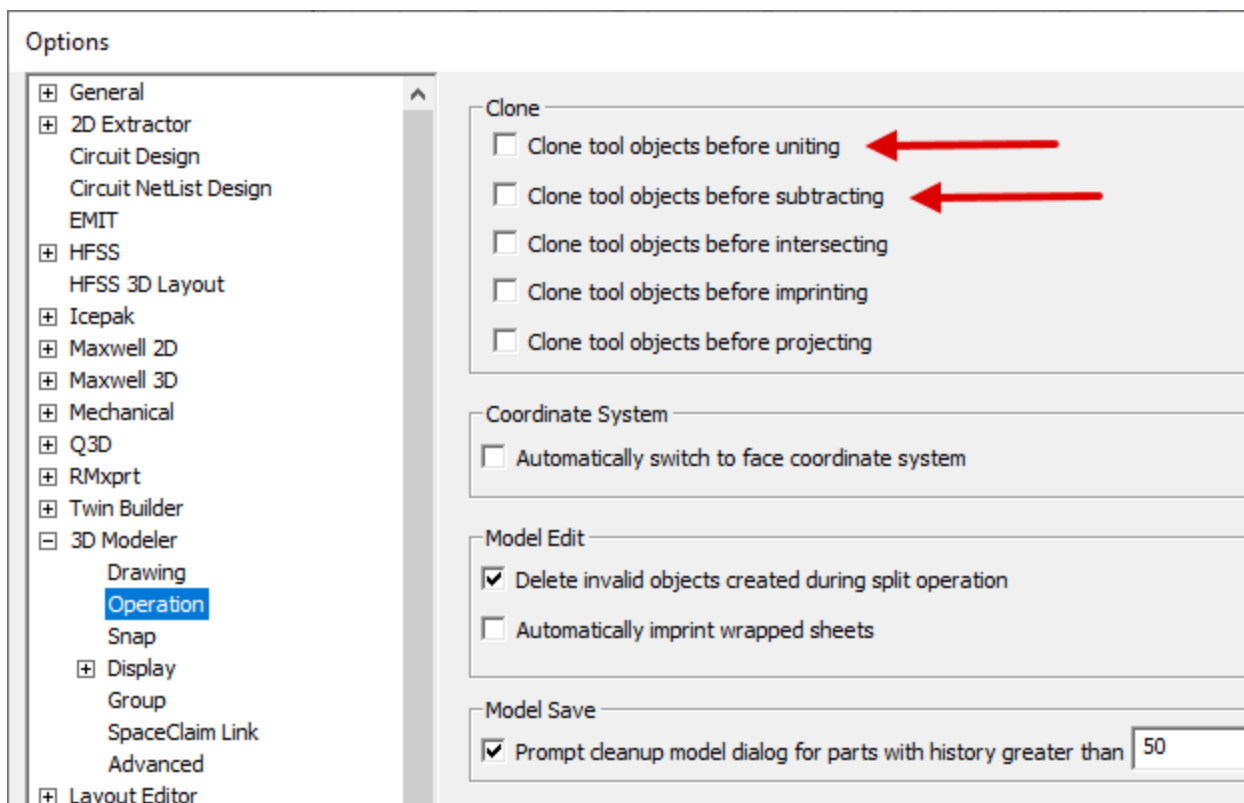
Figure 3-27: *3D Modeler > Operation* **Options**

> **Note:**
>
> You don't want the Ansys Electronics Desktop application to create copies of the original objects before uniting them. The original parts, as separate boxes, will not be a part of the analysis model. Therefore, you must deselect the *Clone tool objects before uniting* option if it is currently selected. In a later procedure, you will also perform a subtraction. For that operation, you likewise do not want the tool object cloned. Since you're already visiting the *3D Modeler > Operation* settings, you can check the state of both options now.

4. Click **OK**.

5. Press **O** to begin the *object selection* mode.

6. Click the first box to select it (the one running in the direction of the X-axis).

   The selection order of the objects is important because it determines the name of the united object.

7. Hold the **Ctrl** key and click the second and third boxes.

8. On the **Draw** ribbon tab, click 🟫 **Unite**.

   The objects are united into a single contiguous volume. The new object has the same name (*Tee*) and attributes as the first object selected.

9. Clear the selection.
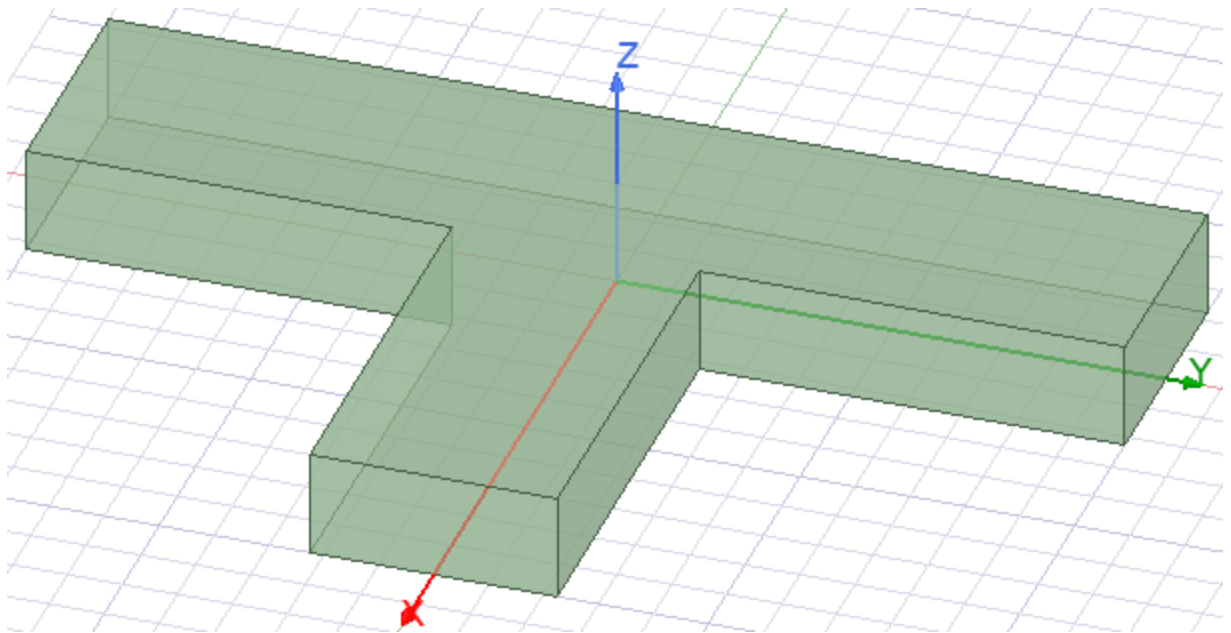
Figure 3-28: The United *Tee* Object

Create the Septum

You will create a small box to use as a cutting tool. You will also define a design variable to control the Y-position of this box so that the septum's location can be fine-tuned easily. (In the subsequent topic, you will subtract the small box from the T-Junction to complete the septum.)

1. On the **Draw** ribbon tab, click 🟦 **Draw box**.

2. Click at three different points in the Modeler window to draw an arbitrary box.

   As soon as you click the third point, the *Properties* dialog box appears.
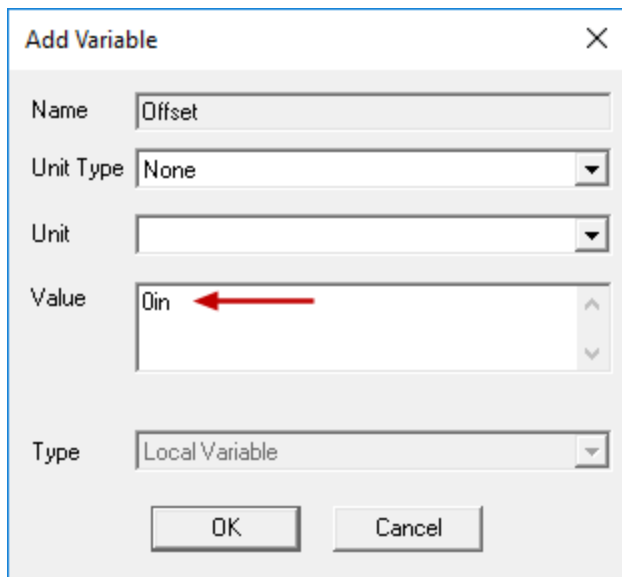
3. Under the **Command** tab of the *Properties* window, specify the following set of coordinates for the **Position** *Value*:

   • **-0.45in, Offset-0.05in, 0in**

4. Press **Enter**.

   The *Add Variable* dialog box appears.

5. In the **Value** text box, specify **0in** and click **OK** to define the initial *Offset* value.

> **Note:**
>
> You must define the unit of measurement, but you can do that as part of the *Value* entry (without using the *Unit Type* and *Unit* drop-down menus):

Figure 3-29: *Add Variable* Dialog Box

6. Specify the following additional values in the **Command** tab of the *Properties* dialog box:

   - **XSize** = **0.45**
   - **YSize** = **0.1**
   - **ZSize** = **0.4**

The settings under the *Command* tab should agree with the following figure:

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate System	Global		
	Position	-0.45in ,Offset-0.05in ,0in		-0.45in , -0.05in , 0in
	XSize	0.45	in	0.45in
	YSize	0.1	in	0.1in
	ZSize	0.4	in	0.4in

Figure 3-30: Cutting Box Properties – *Command* Tab

7. Under the **Attribute** tab of the *Properties* dialog box, change the **Name** to **Septum**.

8. Click **OK**.

   With the cutting box (*Septum*) still selected, your model should look like the following figure:
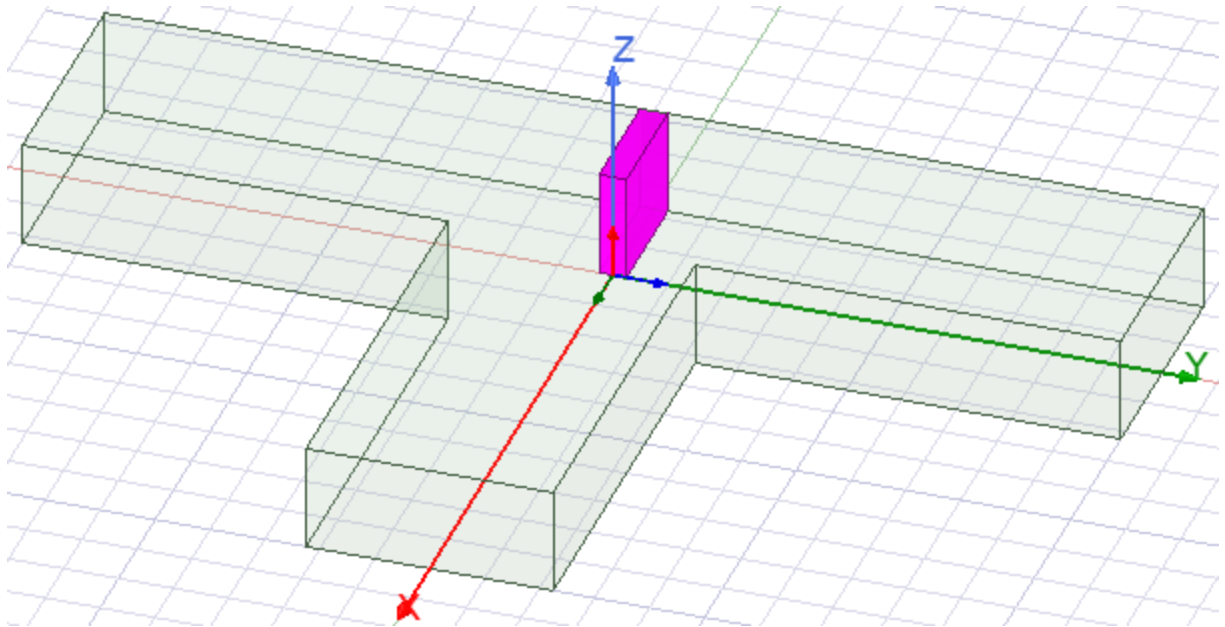
Figure 3-31: *Septum* Cutting Box Created

9. Clear the selection.

Subtract Septum from the T-Junction

To complete the model geometry, you must subtract the septum object from the T-junction.

1. Under *Model > Solids > vacuum* in the History Tree, click **Tee** to select it.

2. Hold down the **Ctrl** key and also click **Septum** in the History Tree.

   > **Note:**
   >
   > The order of object selection determines which part is the *Blank Part* and which is the *Tool Part* when you perform a subtraction.

3. On the **Draw** ribbon tab, click ⬚ **Subtract**.

   The *Subtract* dialog box appears. Ensure that each object is listed in the correct parts list, as shown below:
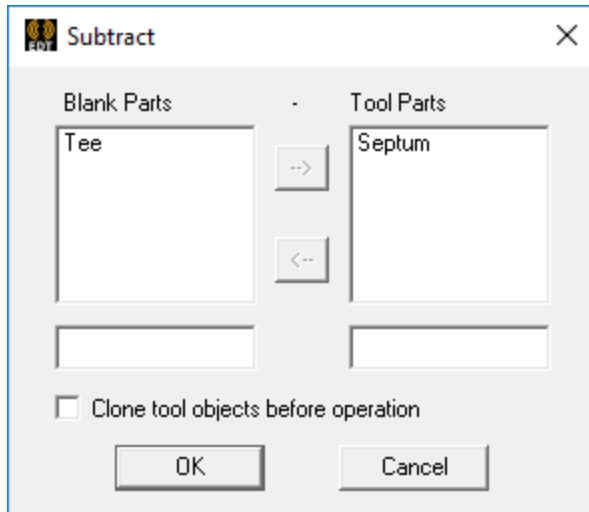
Figure 3-32: *Subtract* Dialog Box
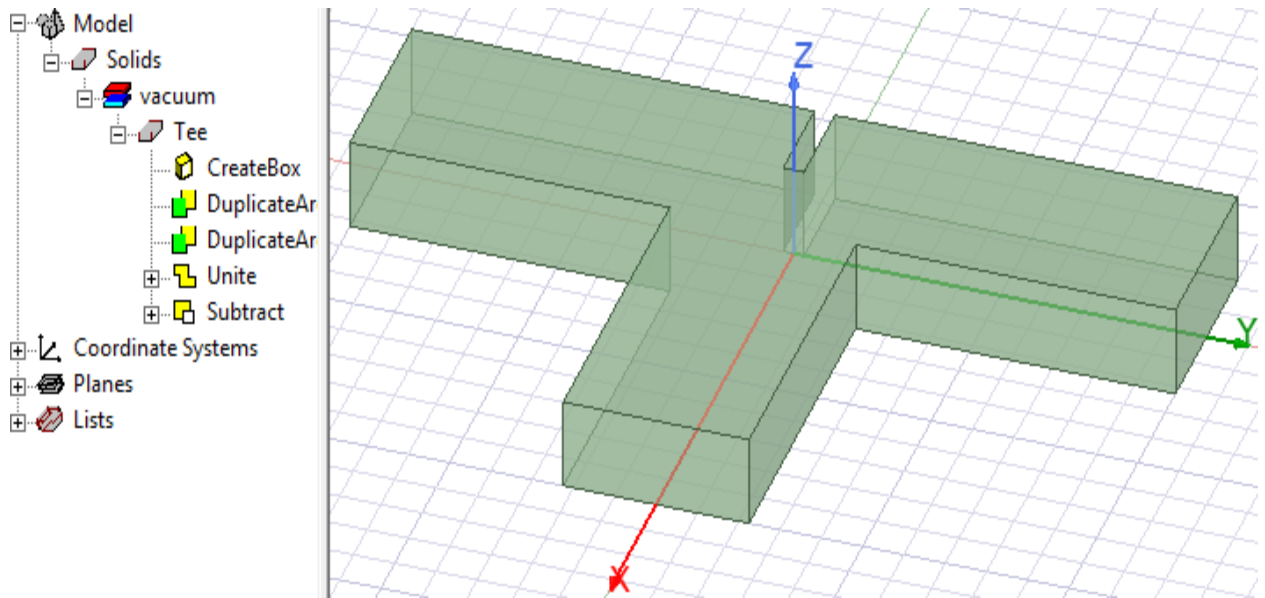
> **Note:**
>
> The Tool Part (Septum) will be subtracted from the Blank Part (Tee).

4. Verify that the   **Clone tool objects before subtracting** option is **not** selected.
5. Click **OK**.

   The *Septum* is subtracted from the *Tee*. The resulting object has the same attributes as the first object you selected.

6. Clear the selection.

Figure 3-33: The Completed Model Geometry

4 - Solve the Design

This chapter covers the following topics:

- Add Solution Setup to the Design
- Validate and Analyze the Tee
- Modify the Septum's Position and Reanalyze

Add Solution Setup to the Design

In this procedure, you will specify how HFSS should compute the solution by adding a *solution setup* to the design. Set up the solution such that HFSS performs an adaptive analysis at a frequency of 10 GHz.

> **Note:**
>
> During an adaptive analysis, HFSS refines the mesh iteratively in the areas of highest error.

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

   The *Driven Solution Setup* dialog box appears.

2. On the **General** tab of the dialog box:
   a. Type **10** in the **Frequency** text box. (Leave the default frequency unit of *GHz*.)
   b. Type **0.01** in the **Maximum Delta S** text box.

> **Note:**
>
> The *Maximum Number of Passes*, which is *6* by default, is the maximum number of iterative passes that HFSS will perform. From the second pass on, if the convergence criterion (*Max Delta S* in this case) is not satisfied, the mesh is refined and another iteration performed. The solution stops after the specified maximum number of passes has been completed, whether the solution has converged or not.
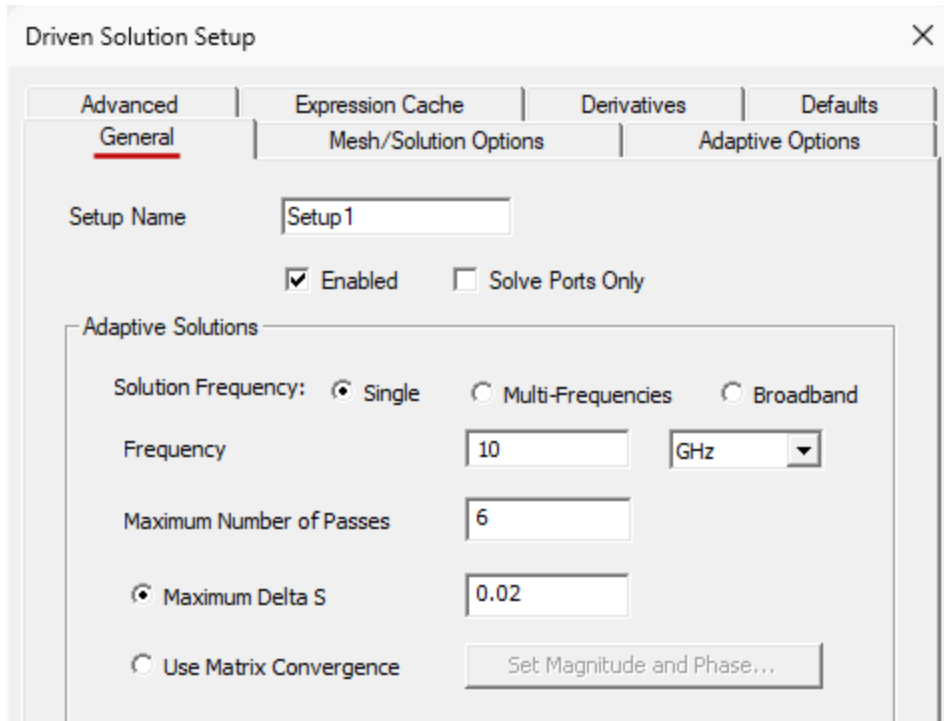
Figure 4-1: Driven Solution Setup

3. Accept the remaining default settings and click **OK**.

   The *Edit Frequency Sweep* dialog box appears. Keep this dialog box open. In the next procedure you will add a frequency sweep.

Add a Frequency Sweep to the Setup

When you complete a solution setup for an HFSS design in which at least one port has been assigned, the *Edit Frequency Sweep* dialog box opens automatically. You will now add a sweep to see the HFSS solution results over a specified range of frequencies. A smooth frequency response is expected for this design, so you will select an interpolating frequency sweep.

> **Note:**
>
> An Interpolating sweep estimates a solution for an entire frequency range. HFSS chooses the frequency points at which to solve the field solution so that the entire interpolated solution lies within a specified error tolerance.

The sweep is complete when the solution meets the error tolerance criterion or generates the maximum number of solutions. The sweep is solved after the adaptive analysis is complete.

1. If you accidentally closed the *Edit Frequency Sweep* dialog box, right-click **Setup1** under *Analysis* in the Project Manager, and choose **Add Frequency Sweep** from the shortcut

menu to reopen it. Otherwise, skip this instruction and proceed to step 2.

Alternatively, on the **Simulation** ribbon tab, click ⌐ƒ **Sweep** (Add Frequency Sweep). But, to make this command available, you must first select the analysis setup, in the Project Manager, to which the sweep will apply (*Setup1* in this case).

The *Edit Frequency Sweep* dialog box appears.

2. Specify the following settings under the **General** tab of the dialog box:

    a. **Sweep Name: Sweep1**

    b. **Sweep Type : Interpolating**

    c. **Distribution:** Choose **Linear Step** from the drop-down menu.

    d. **Start: 8Ghz**

    e. **End: 10 GHz**

    f. **Step Size** (this column depends on the *Distribution* menu selection): **0.05GHz**
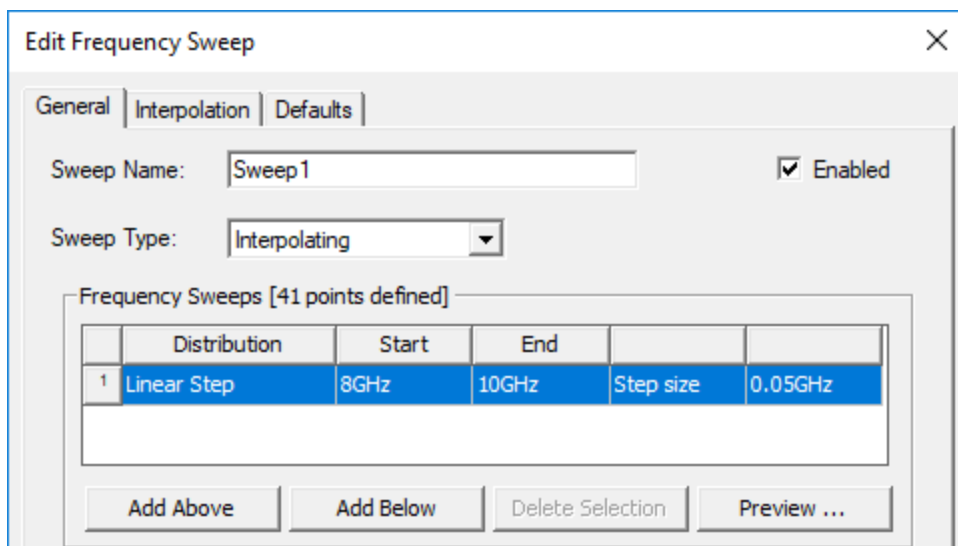
Figure 4-2: Frequency Sweep Settings

> **Note:**
>
> HFSS will solve the frequency point at each step in the specified range, including the start and stop frequencies.

3. Click **OK**.

    Sweep1 is listed under *Analysis > Setup1* in the Project Manager.

> **Note:**
>
> Before adding the frequency sweep, you must assign wave ports to the model. Otherwise, the *Edit Frequency Sweep* dialog box will not contain an *Interpolating* choice in the *Sweep Type* drop-down menu.
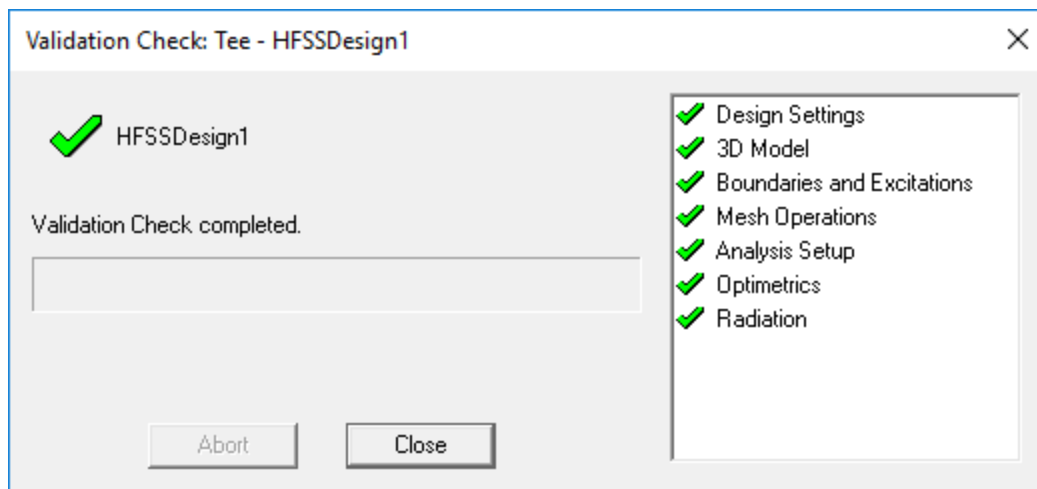
Validate and Analyze the Tee

Because you parameterized the septum position, you can easily move the septum to various positions, run the simulations to generate the corresponding results, and review the resulting behavior of the Tee-structure. Before you run an analysis, verify that all of the necessary setup steps have been completed and their parameters are reasonable.

Validate the Design:

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

    The *Validation Check* window appears, and HFSS checks the project setup. There should be no errors or warnings:

Figure 4-3: Validation Check

2. Click **Close**.

    You are ready to run the simulation.

Analyze the Tee:

Next, analyze the design with the septum placed centrally, opposite from *Port1*:

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

    HFSS computes the solution for every analysis setup in the project. In this exercise, *Setup1* is the only setup you defined. In situations where you have a design with multiple setups and prefer to run only one, you can right-click on a single setup in the Project Manager and select **Analyze** from the shortcut menu.

    > **Note:**
    >
    > The solution process takes approximately 30 seconds to a few minutes to complete, depending on your computer hardware. When the solution is complete, a confirmation message appears in the *Message Manager* window.
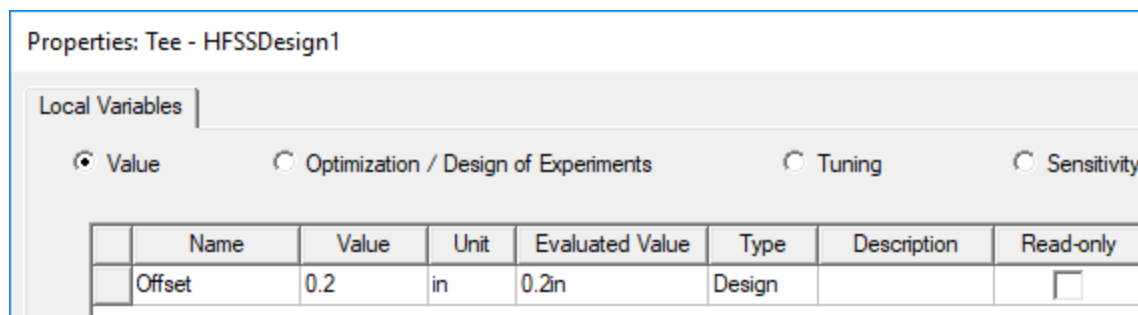
Modify the Septum's Position and Reanalyze

Change the Septum Offset:

You can modify the septum's position by varying the *offset*, as follows:

1. From the menu bar, click **HFSS** > **Design Properties** or right-click **HFSSDesign*x* (Modal Network)** in the Project Manager and choose **Design Properties** from the shortcut menu.

    The *Properties* dialog box appears.

Figure 4-4: Design Properties – Local Variables

2. Ensure that the **Value** option is selected.
3. For the variable **Offset**, type **0.2** in the **Value** column.
4. Click **OK**.

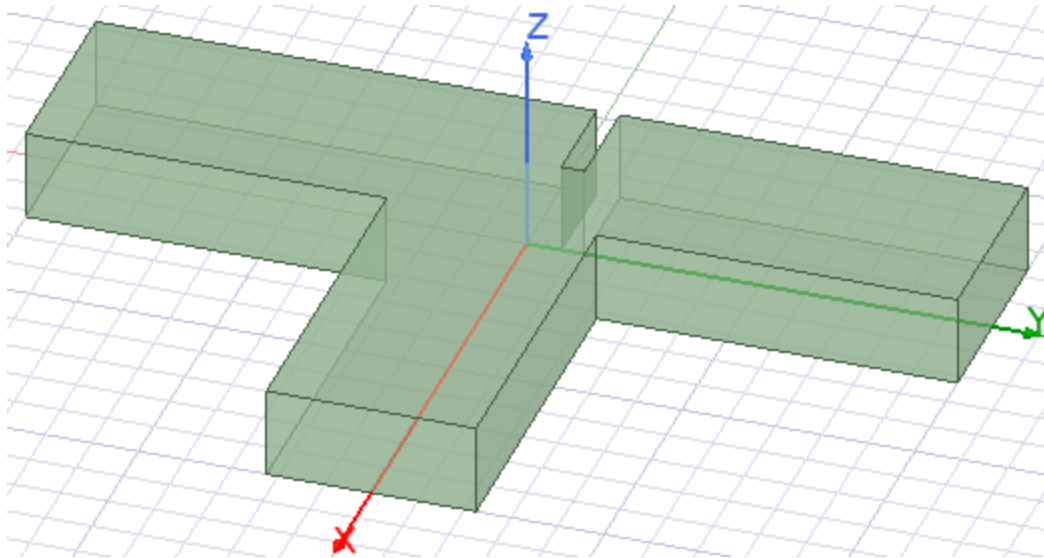   The Tee is updated with the new 0.2 inch *Offset* value:

Figure 4-5: Tee with Septum Offset = 0.2 Inch

Reanalyze the Tee:

You will run a fresh simulation to generate results for the T-junction variant with the septum located off-center, closer to *Port2*.

> **Note:**
>
> The previous solution is saved and the original results remain available for post processing.

5. Right-click **Setup1** under *Analysis* in the Project Manager and select **Analyze**.

    HFSS computes the setup and sweep solution for the offset Septum variant of the T-junction.

5 - Compare the Solutions

This chapter covers the following topics:

- S-Parameter vs. Frequency Plot
- Add Data Markers
- Visualize E-Field at Zero Offset
  - Modify Septum Position
  - Create E-Field Overlay
  - Animate the E-Field
- Visualize E-Field at 0.2 Inch Offset
- What's Next?
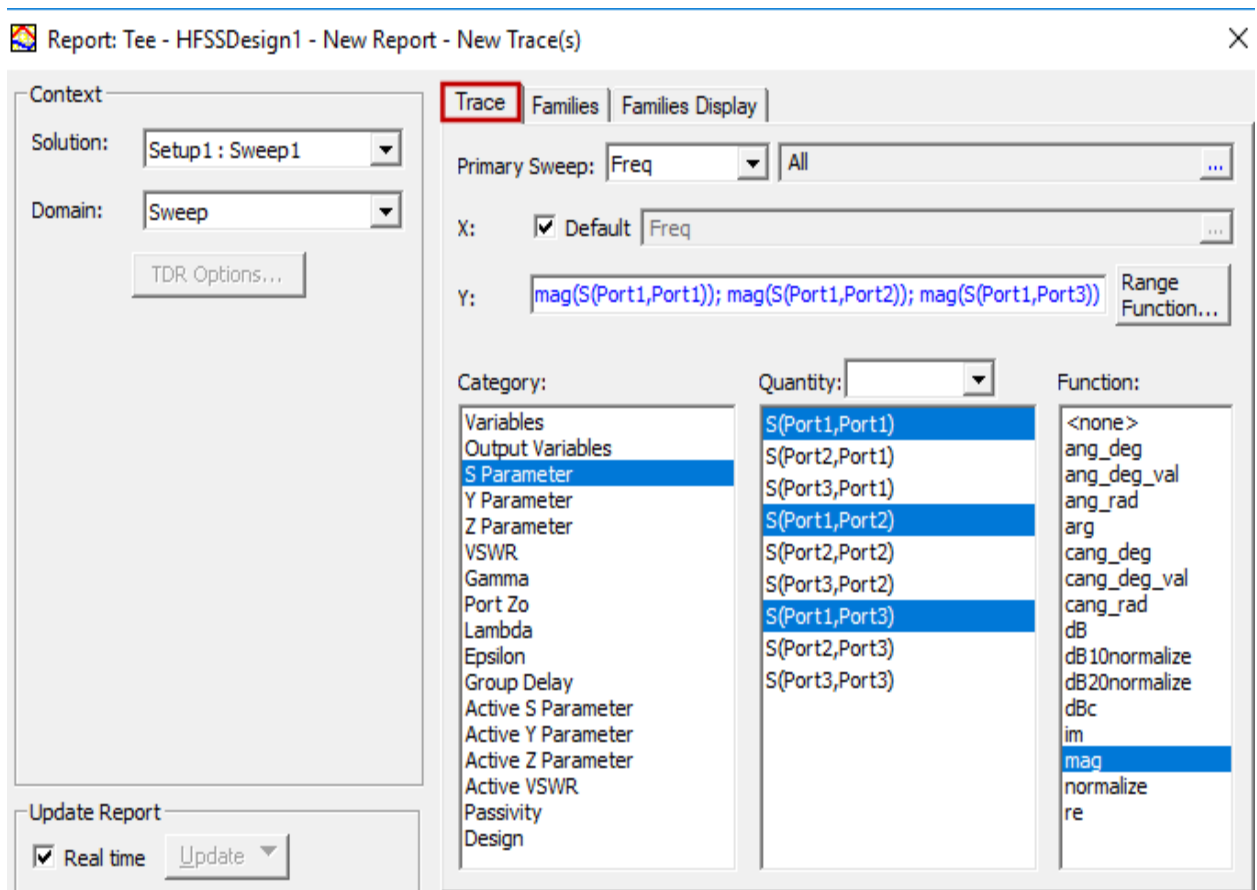
S-Parameter vs. Frequency Plot

You will create a 2D x-y (rectangular) plot to compare the S-parameter results at each port for the two septum positions.

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

   The *Report* dialog box appears.

2. In the *Context* section and under the **Trace** tab, specify the following settings:
   a. **Solution** = **Setup1:Sweep1**
   b. **Domain** = **Sweep**
   c. **Primary Sweep** = **Freq**
   d. **Category** = **S Parameter**
   e. **Quantity** = **S(Port1,Port1), S(Port1,Port2), S(Port1,Port3)** (for the **Y** axis)
   f. **Function** = **mag**
   g. **X** axis value = **Default** (*Freq*)

**Figure 5-1:** *S Parameter vs. Frequency* **Report Settings –** *Trace* **Tab**

3. Under the **Families** tab, ensure that the **Value** column for the **Offset** variable is set to **All**. If not, click the ellipsis button [...] and select the **Use all values** option.
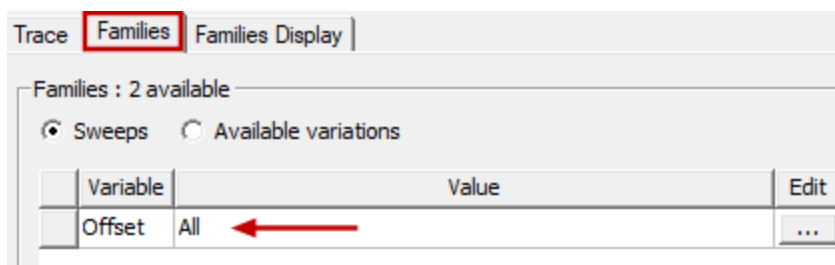
Figure 5-2: *S Parameter vs. Frequency* **Report Settings –** *Families* **Tab**

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

The magnitude of the specified S-parameters at each *Offset* value are plotted against frequency on an x-y graph:
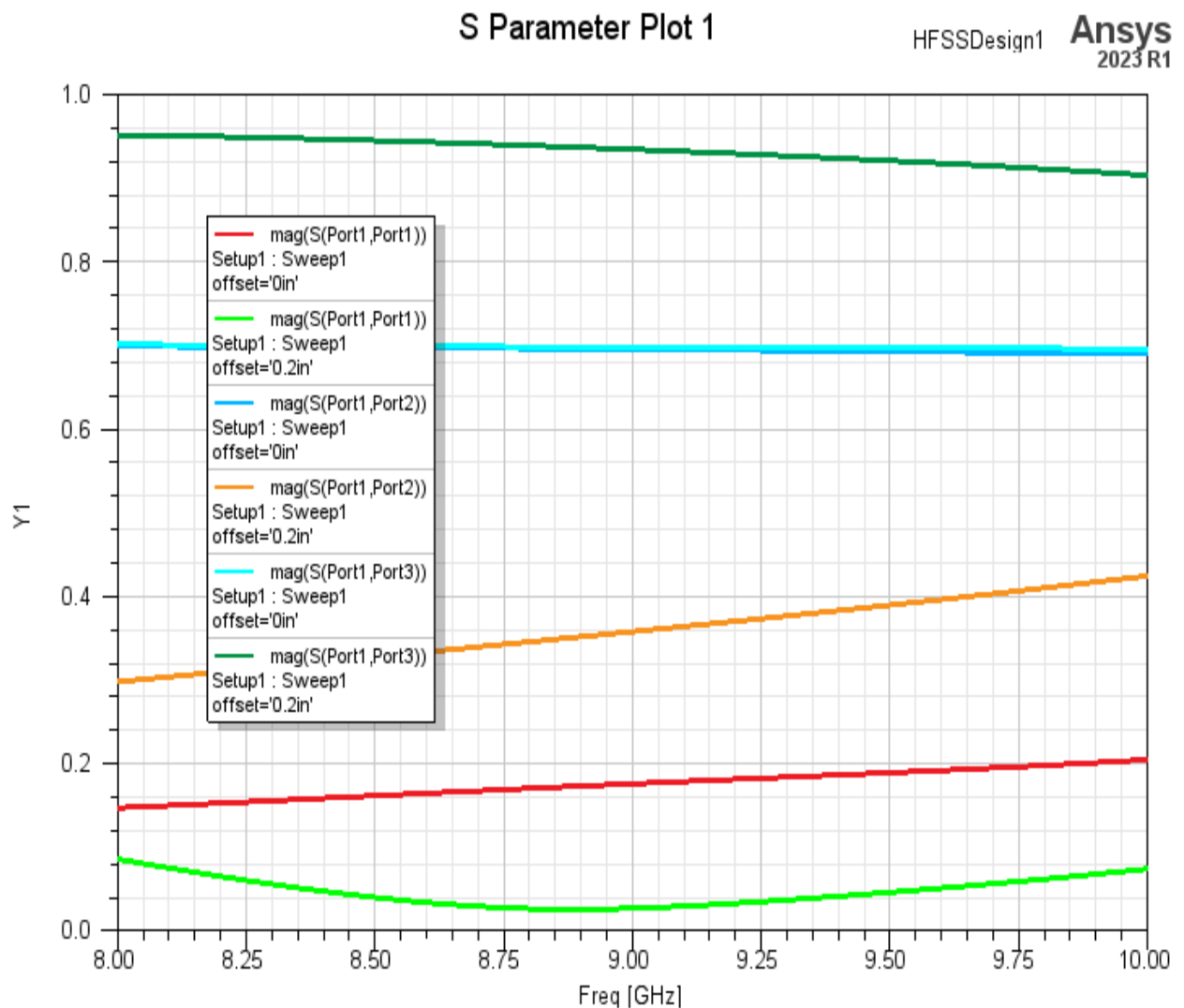
Figure 5-3: S-Parameter vs. Frequency Plot

This plot is listed under *Results* in the Project Manager along with its three corresponding trace definitions. The plot legend shows the *Curve Info* for each trace. Since you specified three S-parameters, and two Offset values were solved, there are six traces.

Observations:

- Two of the traces are directly on top of each other (cyan and medium blue). These two curves represent the relative signal transmission from Port1 to Port2 and from Port 1 to Port3 with zero Offset of the Septum. The signal divides evenly and is fairly constant at approximately 0.7 across the swept frequency range, only decreasing slightly with increased frequency. The magnitude of this result at the adaptive solution frequency of 10 GHz is approximately **0.69**.

- At 0.2 inch Septum offset, the transmission to Port3 (dark green curve) is greater than the transmission to Port2 (orange curve) across the full sweep. The magnitude of the signal transmitted to Port3 decreases as the frequency increases, whereas the magnitude at Port2 increases as the Frequency increases. At the adaptive solution frequency of 10 GHz, S (Port1,Port3) is about **0.9** (an increase of 0.21 relative to the zero Offset result). For the same conditions, S(Port1,Port2) is **0.42** (a decrease of about 0.27 relative to the zero Offset result).

- At zero Offset, the reflection at Port1 (red curve) increases across the full swept range as the Frequency increases. The S(Port1,Port1) parameter at the adaptive solution frequency of 10 GHz is approximately **0.2**.

- With an Offset of 0.2 inch there is less reflection at Port1 throughout the swept range (light green curve), and it is minimal at 8.9 GHz. At the adaptive solution frequency of 10 GHz, S(Port1,Port1) is about **0.07**.

The current results are consistent with those observed from prior simulations and product reviews. (See the earlier-presented [Expected Results](#) discussion.)

Add Data Markers

You can add data point markers along any of the traces, if desired. Additionally, you can control how frequently the marker symbols are added. For example, a *Symbol Frequency* of 1 places a marker at every calculated data point; whereas a *Symbol Frequency* of 5 places a marker on every fifth calculated data point.

1. Click a trace on the plot to which you intend to add data markers (for example, the orange trace).

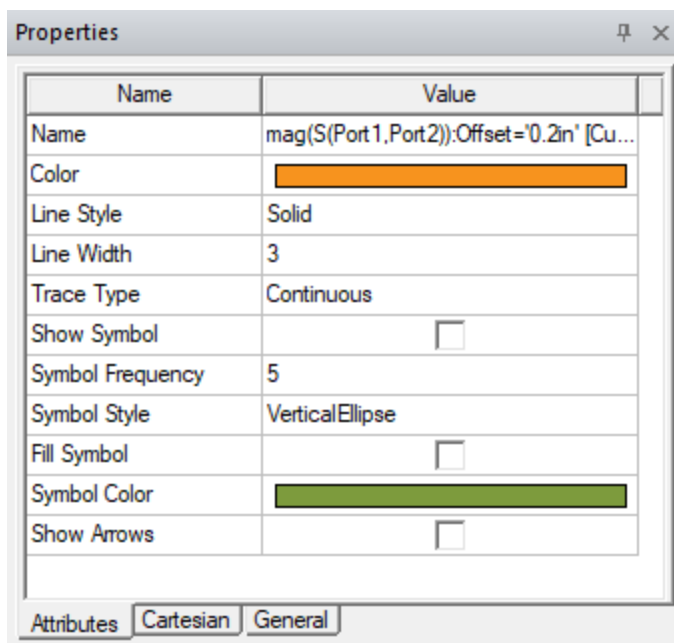   The trace properties appear in the docked *Properties* window:

Figure 5-4: Trace Properties – *mag(S(Port1,Port2)):Offset='0.2in'*

2. Select the **Show Symbol** option.

3. Enter **5** as the **Symbol Frequency** value and press **Enter**.

4. Click in the plot background area to clear the selection.

   Data point markers appear on the trace, as shown below:

Figure 5-5: Data Markers Added to a Trace

Visualize E-Field at Zero Offset

Create and animate an E-Field overlay to look at the signal propagation with the Septum positioned directly opposite Port1 (exactly centered between Port2 and Port3). A field overlay plot represents a field quantity on a surface or within an object. You will plot the magnitude of the E-field on the top surface of the Tee.

Before creating the overlay, you must first modify the *Offset* design variable again, to return the Septum to its original location.

Modify the Septum Position

1. Use the **Window** menu to return to the **Modeler** window, which may presently be covered by the S-parameter plot.

   Alternatively, if the Modeler window is partially visible, simply click it to bring the window to the foreground.

2. From the menu bar, click **HFSS** > **Design Properties**.

   The *Local Variables* tab of the *Properties* dialog box appears.

3. Ensure that the **Value** option is selected.

4. For the **Offset** variable, type **0** in the **Value** column.

5. Click **OK**.

   The Tee geometry is updated with the original (centered) position of the septum restored.

Create the E-Field Overlay

1. On the **Draw** ribbon tab, click ⚏ **Grid** to toggle off the drawing grid visibility.

2. Press **F** or right-click in the Modeler window and choose **Selection Mode** > **Faces** from the shortcut menu.
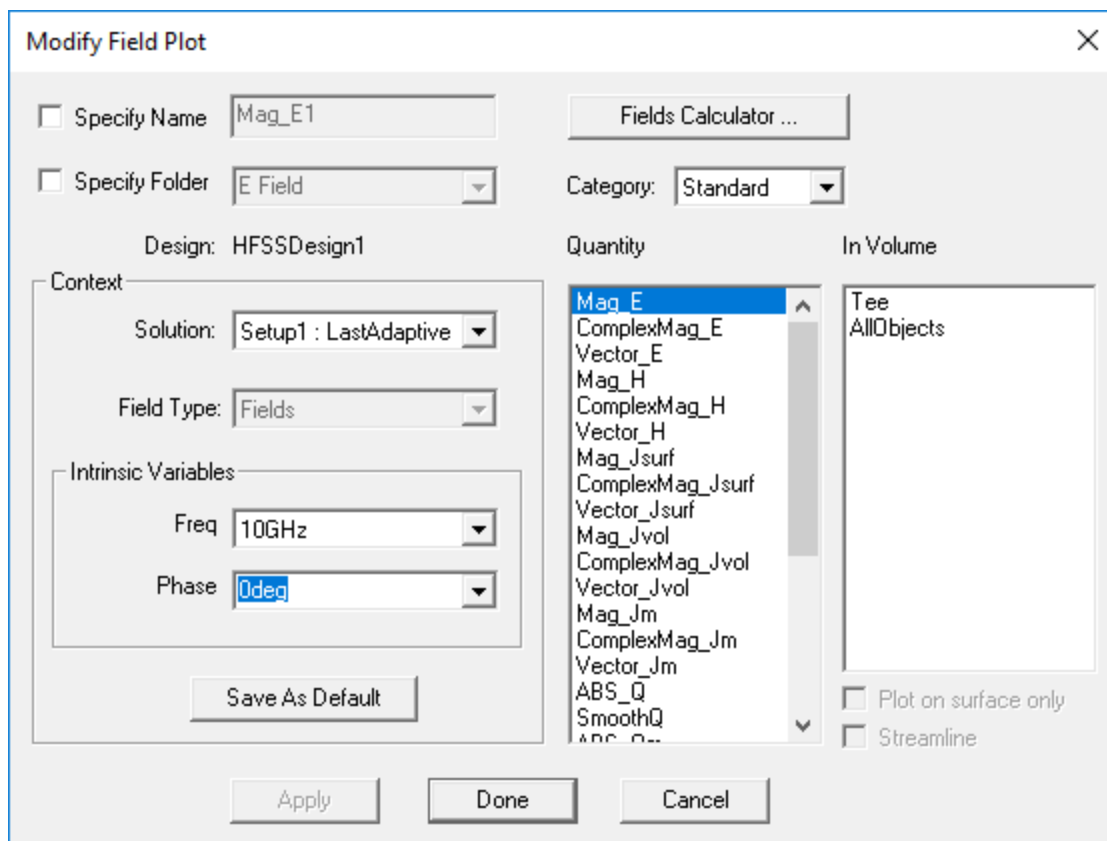
3. Select the **top** face of the Tee:
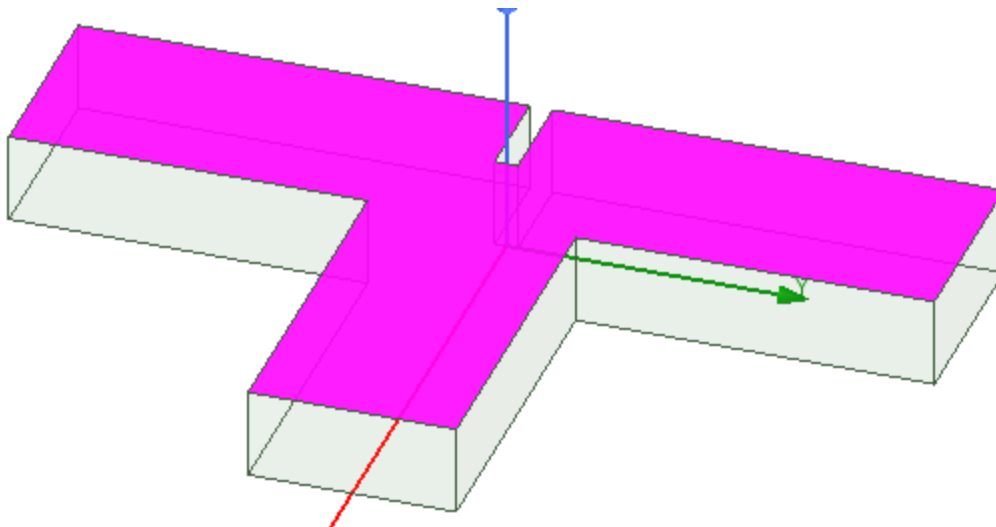
Figure 5-6: Top Face of Tee Selected

4. In the Project Manager, right-click **Field Overlays** and select **Plot Fields** > **E** > **Mag_E**.

   The *Create Field Plot* dialog box appears.

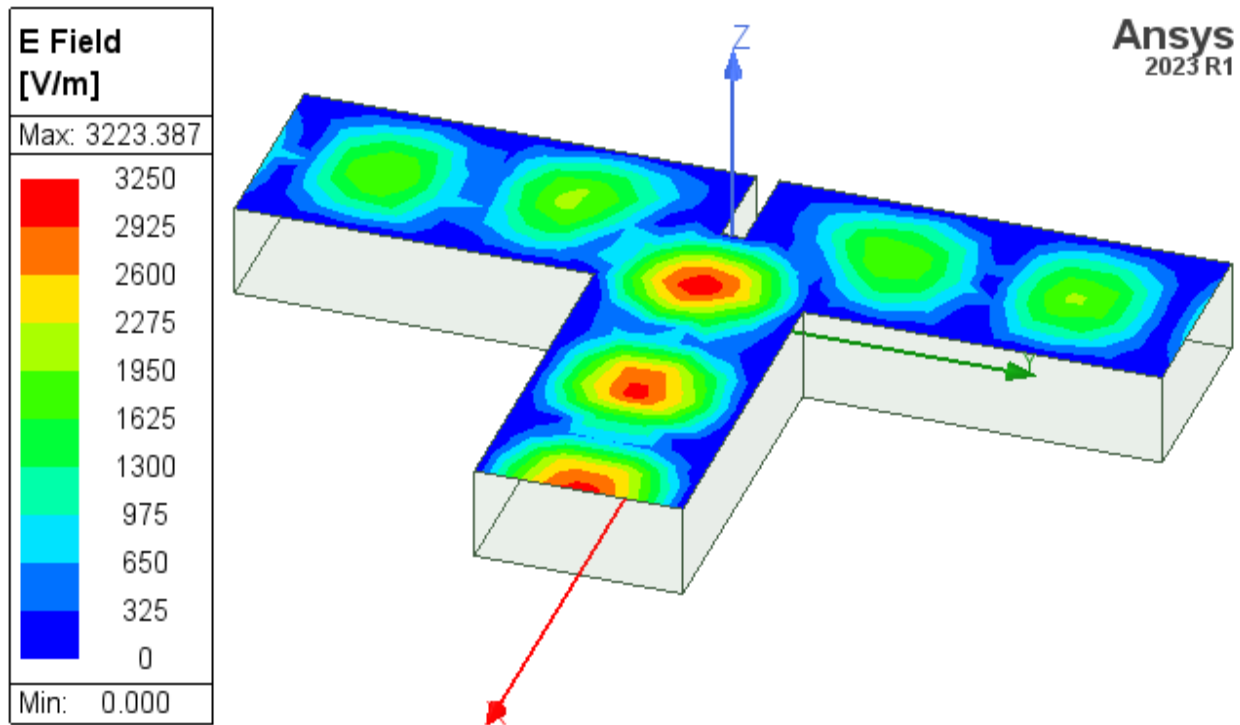5. Verify that the settings are as shown in the following figure:

Figure 5-7: *Create Field Plot* **Dialog Box**

6. Click **Done** to accept the default settings.

    The overlay appears on the top face of the T-junction and it is listed as *Mag_E1* (the default name) under *Field Overlays* in the Project Manager.

    The overlay shows the E-field distributed evenly toward Port 2 and Port 3.

Figure 5-8: *Mag_E1* **Overlay with Septum at Zero Offset**

7. Optionally, you can modify the plot display by double-clicking within the legend box to access a variety of settings (such as color map and scale options). Click **Close** when done.

Animate the E-Field

An animated plot is a series of frames that displays a field, mesh, or geometry at varying values (in this case, varying phase angle). You must specify the values of the plot that you want to include. Each value is a frame in the animation.

1. Right-click **Mag_E1** under *Field Overlays > E Field* in the Project Manager and choose **Animate**.

    The *Create Animation Setup* dialog box appears.

2. Verify the default settings, as shown in the following figure, and click **OK**.
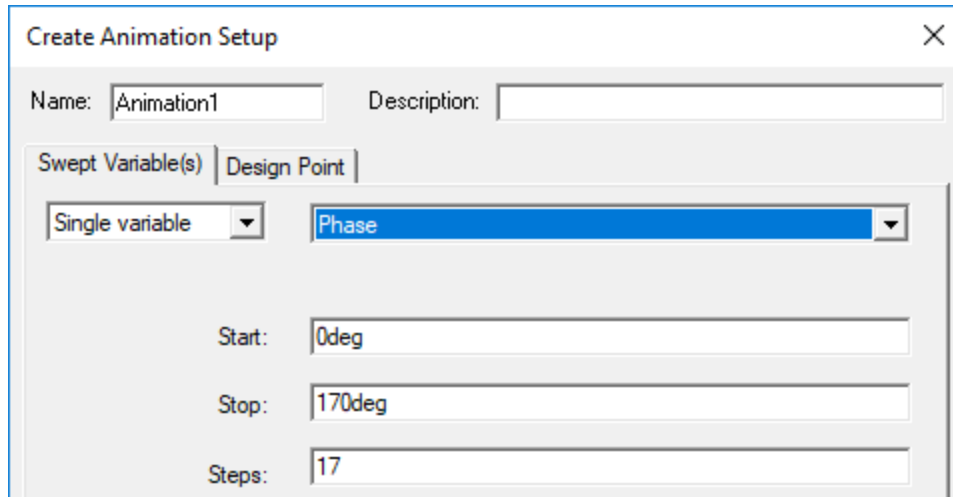
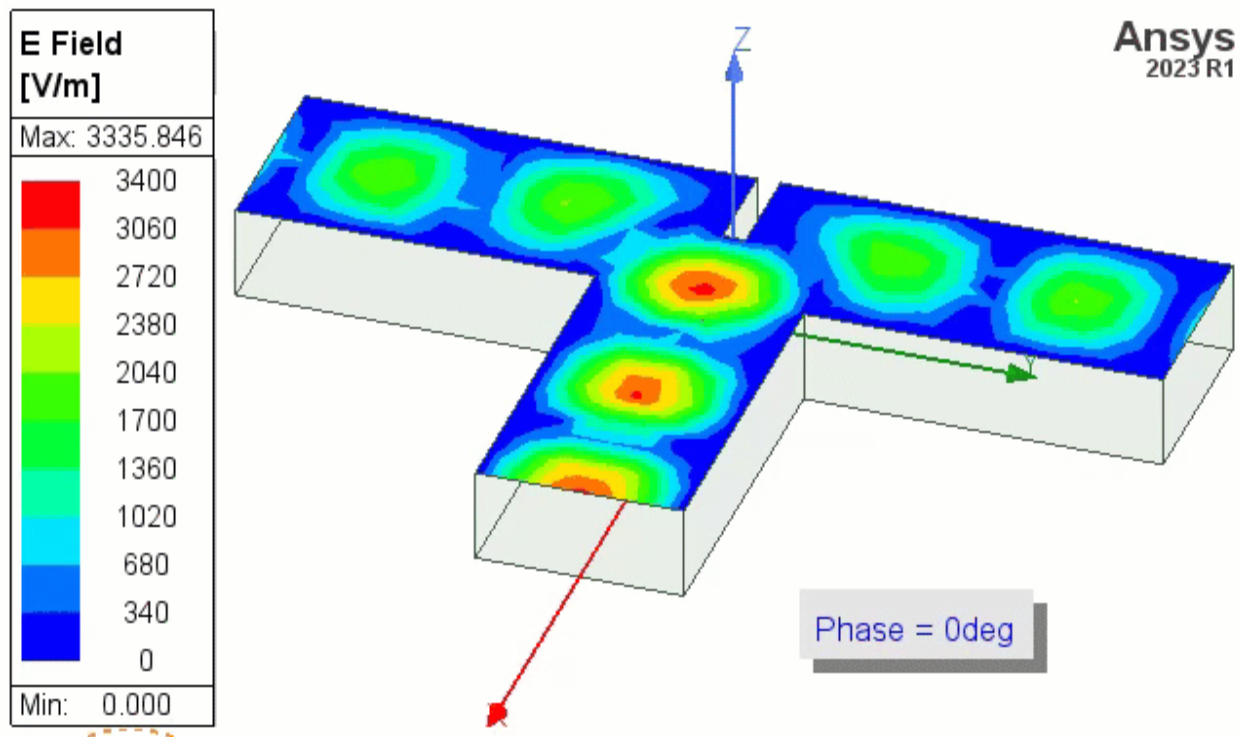**Figure 5-9:** *Create Animation Setup* **Dialog Box**

The *Animation* dialog box appears, enabling you to stop, restart, reverse, and control the speed of the frames.
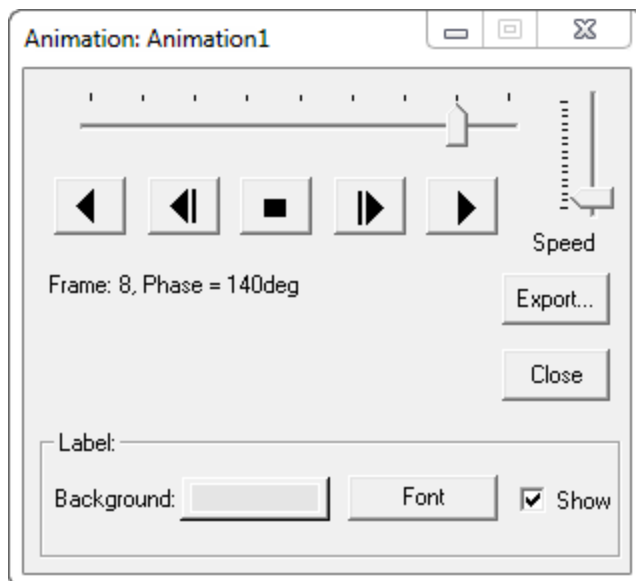
Figure 5-10: *Animation* **Dialog Box**

The animation begins and shows the Septum dividing the electromagnetic wave evenly toward Port 2 and Port 3. The field is displayed at 18 phase values between 0 and 170 degrees, inclusively (17 increments of 10° each).

Figure 5-11: E-Field Animation – Zero Septum Offset

3. In the *Animation* dialog box, click **Close**.

Visualize E-Field at 0.2 Inch Offset

Create and animate an E-Field overlay to look at the signal propagation with the Septum Offset 0.2 inches toward Port2.

Before creating the overlay, you must first modify the *Offset* design variable again to place it at the desired position for the animation.

Modify the Septum Position:

1. Select **HFSSDesign*x* (Driven Modal)** in the Project Manager.

    The design variables are shown in the **Variables** tab of the docked *Properties* window.

2. For the **Offset** variable, type **0.2** in the **Value** column and press **Enter**.

    The Tee geometry is updated to place the Septum 0.2 inches offset toward Port2. Also, the E-field overlay is immediately updated. Now, more energy is flowing toward Port3 than Port2.

Animate the Revised E-Field Overlay:

3. Right-click **Mag_E1** under *Field Overlays > E Field* in the Project Manager and choose **Animate**.

    The *Select Animation* dialog box appears and previously defined *Animation1* is selected.

4. Click **OK** to access the *Animation* dialog box and start the animation:
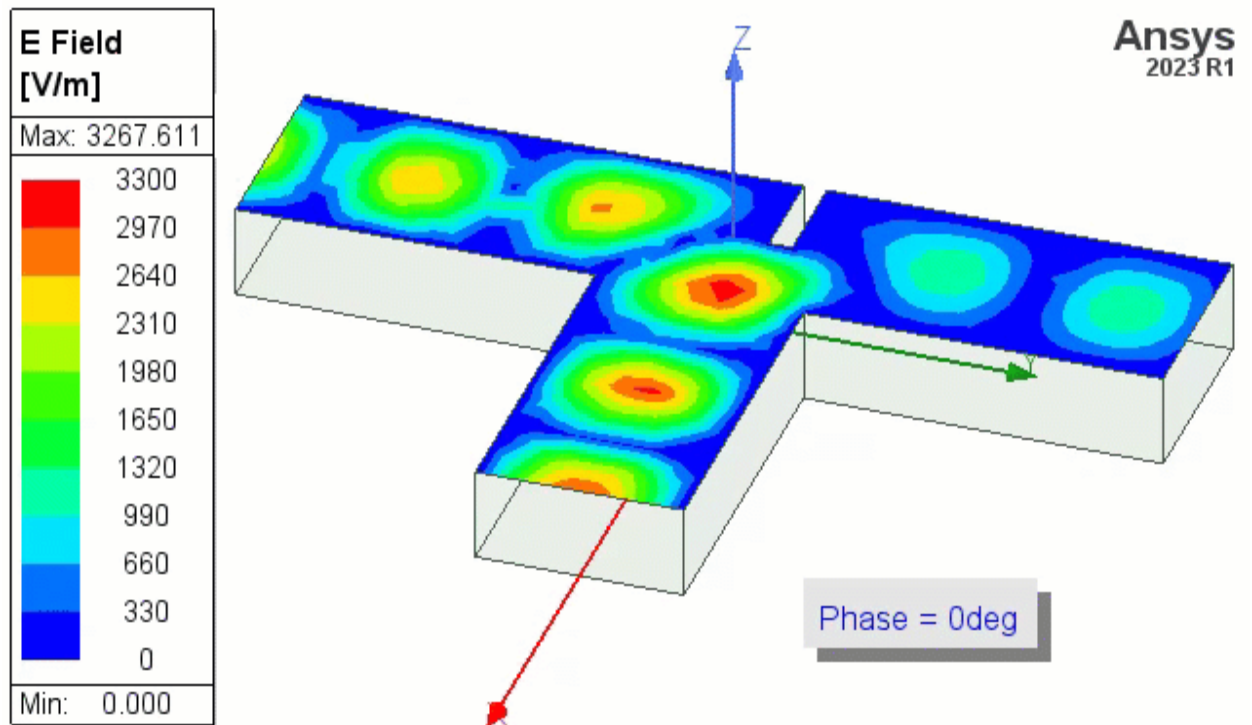

Figure 5-12: E-Field Animation – 0.2 Inch Septum Offset

5. 💾 **Save** your project.

6 - What's Next?

You have successfully completed creating the T-Junction Waveguide and evaluating the behavior of its electromagnetic field. If desired, you can proceed immediately to the follow-up guide, *Getting Started with HFSS: Waveguide T-Junction Optimization*, where you will learn about *Optimetrics* using this model.

Choose *one* of the following two actions:

- Leave this project open and proceed to the *Introduction* page of the *Waveguide T-Junction Optimization*.

  Skip the topic at the end of this guide because you will continue to use legacy view orientations for the optimization process.

- If you do *not* wish to continue to the *Waveguide T-Junction Optimization*, proceed to the next topic to restore current view orientations before closing this project.

# 7 - 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:
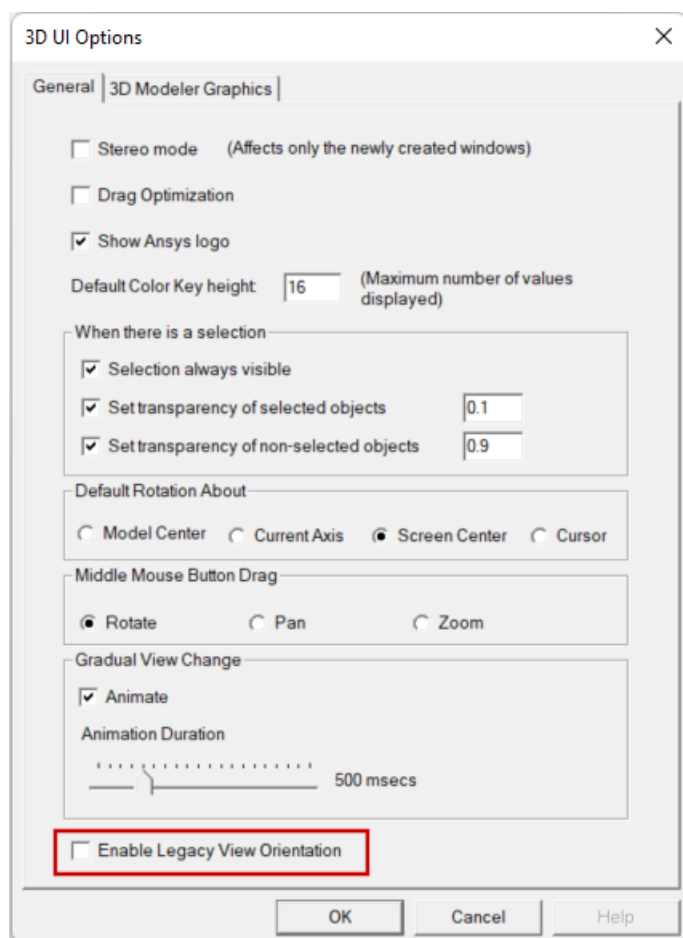
3. Click **OK**.

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.