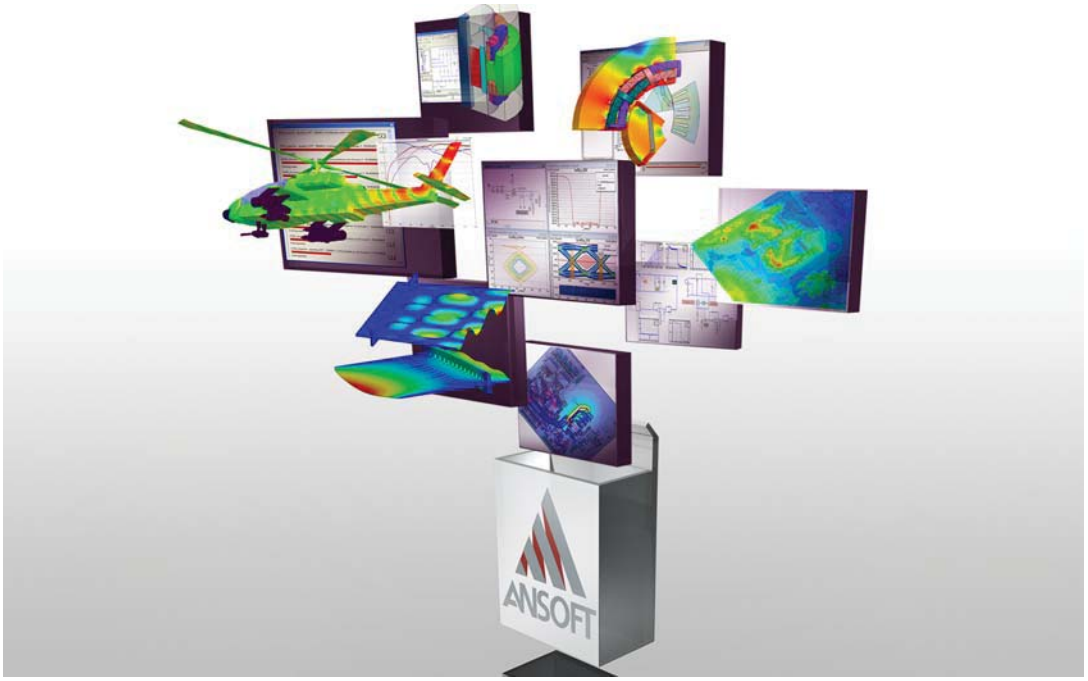



# Getting Started with HFSS:
## A Waveguide T-Junction

ANSYS Product Suite

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317
Tel: (+1) 724-746-3304
Fax: (+1) 724-514-9494
General Information: AnsoftInfo@ansys.com
Technical Support: AnsoftTechSupport@ansys.com

October 2010
Inventory 0000002934

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317
USA
Tel: (+1) 724-746-3304
Fax: (+1) 724-514-9494

General Information: AnsoftInfo@ansys.com
Technical Support: AnsoftTechSupport@ansys.com



New editions of this manual incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.

Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

| Edition | Date | Software Version |
|---|---|---|
| 1 | May 2003 | 9 |
| 2 | June 2005 | 10 |
| 3 | June 2007 | 11 |
| 4 | Sept 2009 | 12 |
| 5 | October 2010 | 13.0 |

# Conventions Used in this Guide

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

- 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 to type the word **copy**, to type a space, and then to type **file1**.

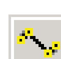
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, click **HFSS>Excitations>Assign>Wave Port**.

  - 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** *file name*" means to type the word **copy**, to type a space, and then to type a file name.
- 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 the **F1** key at the same time.
- Toolbar buttons serve as shortcuts for executing commands. Toolbar buttons are displayed after the command they execute. For example,

  "On the **Draw** menu, click **Line**  " means that you can click the Draw Line toolbar button to execute the **Line** command.

# Getting Help

## Ansys Technical Support

To contact Ansys technical support staff in your geographical area, please log on to the Ansys corporate website, https://www1.ansys.com. You can also contact your Ansoft account manager in order to obtain this information.

All Ansoft 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 online help from the HFSS menu bar, click **Help** and select from the menu:

- **Contents** - click here to open the contents of the online help.
- **Seach** - click here to open the search function of the online help.
- **Index** - click here to open the index of the online help.

## Context-Sensitive Help

To access online help from the HFSS user interface, do one of the following:

- To open a help topic about a specific HFSS menu command, press **Shift**+**F1**, and then click the command or toolbar icon.
- To open a help topic about a specific HFSS dialog box, open the dialog box, and then press **F1**.

# Table of Contents

## 4. Set Up and Generate Solutions



## 5. Compare the Solutions

# 1 Introduction

This *Getting Started* guide is written for HFSS beginners as well as experienced users who are using version 13 for the first time. This guide leads you step-by-step through creating, solving, and analyzing the results of a waveguide T-junction.

By following the steps in this guide, you will learn how to perform the following tasks in HFSS:

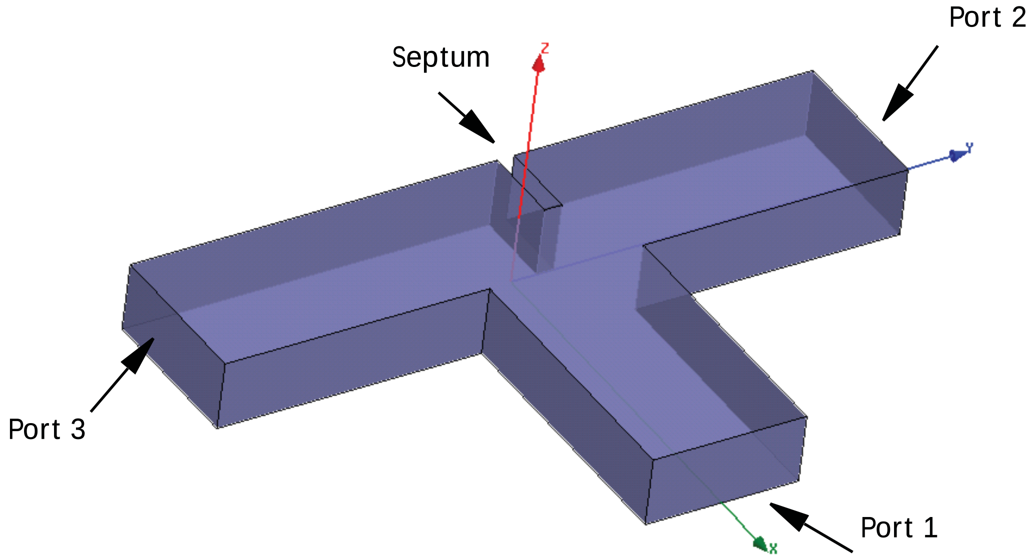
- Draw a geometric model.
- Modify a model’s design parameters.
- Assign variables to a model’s design parameters.
- Specify solution settings for a design.
- Validate a design’s setup.
- Run an HFSS simulation.
- Create a 2D x-y plot of S-parameter results.
- Create a field overlay plot of results.
- Create a phase animation of results.

*Estimated time to complete this guide: 45 minutes.*

## About the T-Junction

The waveguide you will create is T-shaped with an inductive septum.[1] This type of structure is used to split an incoming microwave signal into two outgoing signals. The septum divides the signal and directs it to the outgoing ports, while minimizing reflection at the signal's point of entry.

A signal at a frequency of 10 GHz enters the waveguide at Port 1 (see below) and exits at Port 2 and Port 3. The waveguide's transmission and reflection of the signal depends on the position of the septum.




### Expected Results

When the septum is located centrally opposite Port 1, it divides the signal and directs it evenly towards the output ports, Port 2 and Port 3. The magnitude of S-parameters at the output ports is expected to be about 0.7. Incidental reflection is expected at Port 1.

Moving the septum 0.2 inches closer to Port 2 reduces the transmission through Port 2 to about 0.1 and increases the transmission through Port 3 to about 0.9.

To determine if the results are as expected, you compare HFSS's S-parameter calculations at each septum position on a 2D x-y plot. You also compare the E-field pattern at each septum position by creating phase-animated field plots on the model geometry. These comparisons will indicate if the field pattern changes as expected with the septum's position.

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

# Using HFSS to Create and Improve the Design

As you step through this *Getting Started* guide, you will be introduced to several key concepts:

- *There are numerous ways to perform most tasks.* For example, several methods are presented for selecting and for assigning design parameter values.
- *There is no required sequence of events when creating a design.* A convenient method for creating the T-junction will be demonstrated, but the design setup steps can be completed in any logical order.
- *You can quickly modify design properties at any time.* For example, you can draw a box freehand, then specify its exact dimensions in the **Properties** window.
- *You can easily track modifications to your design in the history tree and the project tree.* The branches provide access to setup dialogs, where you can modify design properties.
- *You can modify the model view at any time.* You will learn shortcut keys like **Ctrl+D**, which fits the model in the view window.
- *You can save time by parameterizing design properties.* For example, you can assign a design variable to the septum's position. This enables you to quickly modify it and generate new results.
- *You can use HFSS's extensive post-processing features to evaluate solution results*. For example, the animations you create will help you visualize the difference in field pattern results for the two septum positions.

*Parameterizing is most effective when paired with Ansoft's Optimetrics software. Optimetrics allows you to define and solve a series of variable values within a range, called parametric analysis. You can also perform an optimization analysis, in which Optimetrics changes the design parameter values to meet a user-defined goal. Both of these capabilities are demonstrated in "Getting Started with Optimetrics: Optimizing a Waveguide T-Junction Using HFSS and Optimetrics."*

# 2 Set up the Design

In this chapter you will complete the following tasks:

- Save a new project.
- Rename the HFSS design in the project.
- Select a solution type for the project.
- Set the drawing units for the design.

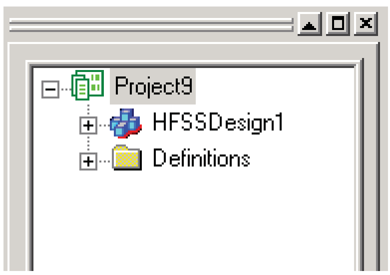
*Estimated time to complete this chapter: 5 minutes.*

# Open HFSS and Save a New Project

A project is a collection of one or more designs saved in a single *.hfss file. A new project is automatically created when HFSS is launched. Open HFSS and save the default project by a new name.

1. Double-click the **HFSS 13** icon on your desktop to launch HFSS.

   A new project is listed in the project tree in the **Project Manager** window and is named Project*n* by default. Project definitions, such as material assignments, are stored under the project name.

*If HFSS was already open and a default project is not listed in the project tree, add a new HFSS project: On the **File** menu, click **New***

.

2. Click **File>Save As**.

   The **Save As** dialog appears.

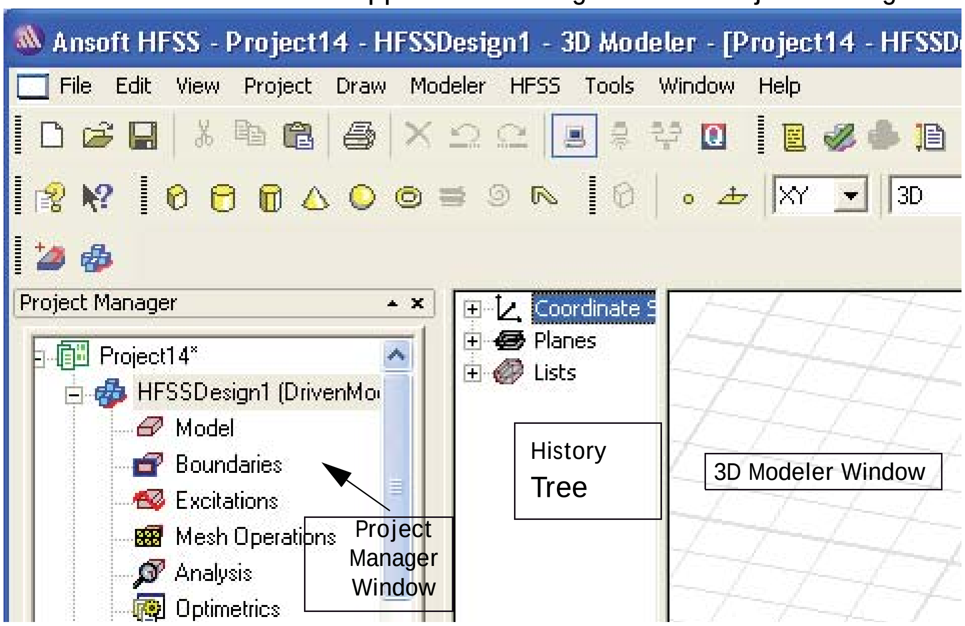
3. Use the file browser to locate the folder in which you want to save the project, (such as C:\Program Files\Ansoft\HFSS13.0\Projects), and then double-click the folder's name.

4. Type Tee in the **File name** text box, and then click **Save**.

   The project is saved in the folder you selected to the file name *Tee.hfss*.

## Rename the Design

You will now rename the default HFSS design in the project. The design is already listed in the project tree when HFSS opens. It is named HFSSDesign*n* by default. The **3D Modeler** window appears to the right of the Project Manager.




*If the Project Manager does not appear after you insert a new design, click* ***View>Project Manager****.*

1. To rename the design: Right-click **HFSSDesign*n*** in the project tree, and then click **Rename** on the shortcut menu.
2. Type **TeeModel**, and then press **Enter**.

## Select a Solution Type

*To automatically expand the project tree when an item is added to the project: Click* ***Tools>Options> General Options****. Under* ***Project Options****, select* ***Expand Project Tree on Insert****.*

As you set up the design for analysis, available settings depend on the solution type. For this design, you will choose Driven Modal as the solution type, which is appropriate when calculating mode-based S-parameters of a passive, high-frequency waveguide that is being “driven” by a source.

1. To specify the design solution type, click **HFSS>Solution Type**.

   The **Solution Type** dialog appears.
2. In the **Solution Type** dialog box, select **Driven Modal**, and

then click **OK**.

## Set the Drawing Units

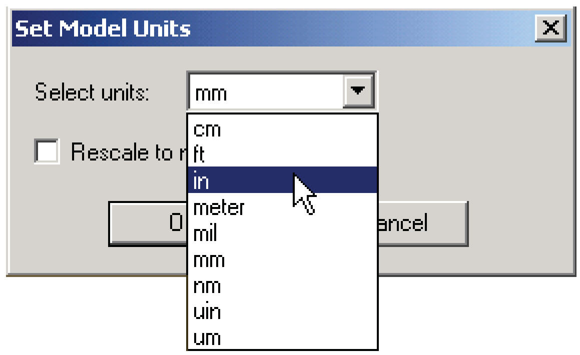
*The* ***Rescale to new units*** *option in the dialog changes the current units of all objects in the design to the new units. For example, 1 mm would become 1 in.*

To set the units of measurement for drawing the geometric model.

1 Click **Modeler>Units**.

The **Set Model Units** dialog appears

2 Select **in** from the **Select units** pull-down list, and then click **OK**.



# 3 Create the Model

In this chapter you will complete the following tasks:

- Draw a section of the T-junction.
- Assign a wave port with an integration line to the section.
- Duplicate the section to create the other two sections of the T-junction.
- Unite the three sections to create the complete T-junction.
- Draw the septum.
- Assign a variable to the septum's position.
- Subtract the septum from the T-junction.

*Estimated time to complete this chapter: 15 minutes.*

# Create the T-Junction

The T-junction is made up of three joined box objects. First you will draw a box that represents one section of the tee. You will assign it a name, confirm its material assignment, and assign a wave port to one of its faces.

You will then duplicate the box two times to create the second and third sections of the tee. Last, you will unite the three sections to create the complete T-junction.

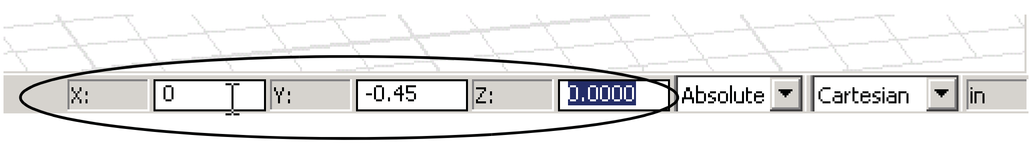
## Draw a Box

Draw a 3D box object to represent the first section of the tee.

1. Ensure the option **Edit Properties of new primitives** is selected in **Tools->Options>Modeler Options->Drawing Tab**.
2. On the **Draw** menu, click **Box** .
3. Find the coordinate fields at the bottom of the HFSS window, labeled "Enter the box position," and specify the base corner of the box as (0, -0.45, 0):

*To move to the previous coordinate box, press **Shift+Tab**.*

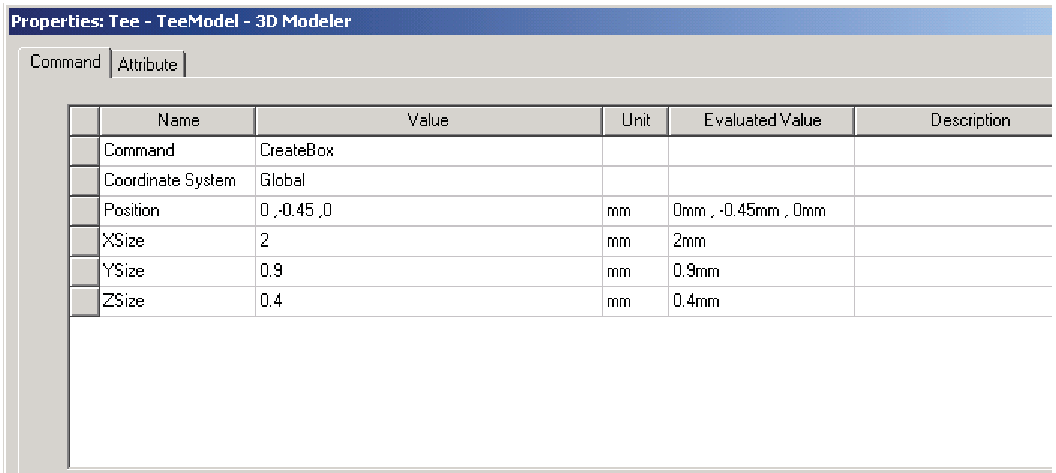
   a. Press **Tab** to move to the X text box in the status bar.
   b. Type **0** in the **X** box, and then press **Tab** to move to the **Y** box.
   c. Type **-0.45** in the **Y** box, and then press **Tab**.
   d. Type **0** in the **Z** box, and then press **Enter**.




*If you make a mistake, click **TeeModel** in the project tree, and then click **Undo** on the **Edit** menu to undo design operations. HFSS lets you undo every command performed since the last save.*

4. Specify the length and width of the box by entering a point relative in distance to the base corner: Type (**2**, **0.9**, **0**) in the **dX**, **dY**, and **dZ** boxes, and then press **Enter**.
5. Specify the height of the box by entering a point on the z-axis relative in distance to the previously entered point: Type (**0**, **0**, **0.4**) in the **dX**, **dY**, and **dZ** boxes, and then press **Enter**.

The **Properties** window appears, with the **Command** tab selected, enabling you to modify the dimensions and position of the box.




| Name | Value | Unit | Evaluated Value | Description |
|---|---|---|---|---|
| Command | CreateBox | | | |
| Coordinate System | Global | | | |
| Position | 0 ,-0.45 ,0 | mm | 0mm , -0.45mm , 0mm | |
| XSize | 2 | mm | 2mm | |
| YSize | 0.9 | mm | 0.9mm | |
| ZSize | 0.4 | mm | 0.4mm | |

*If you do not want the* ***Properties*** *dialog box to appear after you draw an object: Click* ***Tools>Options>3D Modeler Options****. In the* ***3D Modeler Options*** *dialog, click the* ***Drawing*** *tab, and then clear the* ***Edit property of new primitives*** *option.*

While the **Properties** window is open, you will use it to assign a name to the box, confirm its material assignment, and make it more transparent.

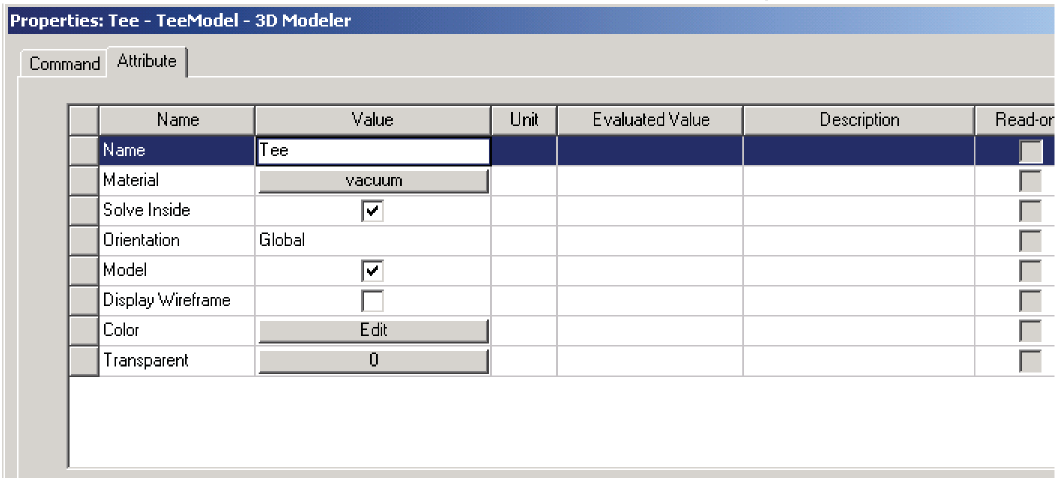
## Assign a Name to the Box

Assigning a name to the box makes it easier to track modifications you make to the design.

1 In the **Properties** window, click the **Attribute** tab.

2 Change the name of the box to *Tee*: Type **Tee** in the **Value**

text box in the **Name** row, and then press **Enter**.




## Confirm the Material Assigned to the Box

By default, the material assigned to the box is “vacuum”. This is the material you will use for the T-junction. Confirm that **vacuum** is the value in the **Material** row so you do not need to change it.

## Increase the Transparency of the Box

Increasing the box’s transparency makes it easy for you to distinguish separations between other objects.

1. Click the value in the **Transparent** row.

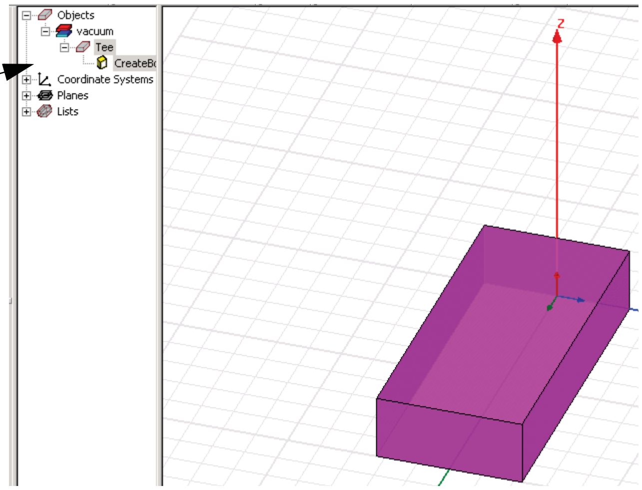
   The **Set Transparency** dialog appears.

2. Move the slider until the transparency level is 0.4, and then click **OK**.

**3** Click **OK** to close the **Properties** dialog.

The name *Tee* was assigned to the box. The commands performed on the box are tracked in the history tree.

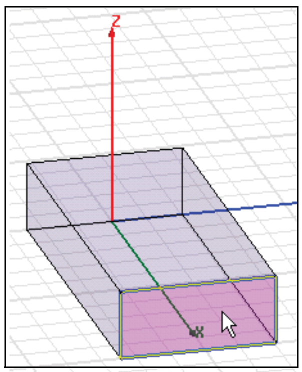
*To magnify the view of the port face, press **Alt+Shift** while dragging the mouse towards the top of the view window. Drag the mouse towards the bottom of the window to zoom out.*




The first box object in the **3D Modeler** window.
It is selected by default when you exit the **Properties** window.

## Assign a Wave Port to the Box

Now you will assign a wave port to the face of the box that is parallel to the yz plane at x = 2. As part of the setup process, you will define an integration line, which is a vector that specifies the direction of the excitation field pattern at the port. These lines ensure that the field pattern is consistent at all ports.



The face of the box that is parallel to the yz plane at x = 2.

1 Switch to face selection mode by pressing the shortcut key **F**.

2 Click the face of the box that is parallel to the yz plane at x = 2, as shown to the right.

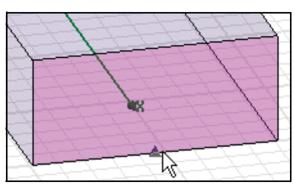
3 Right-click the **3D Modeler** window, and then click **Assign Excitation>Wave Port** on the shortcut menu.

The **Wave Port** wizard appears.

4 Type **Port1** in the **Name** text box, and then click **Next**.

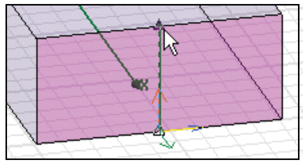
5 Select **New Line** from the **Integration Line** pull-down list.

6 In the **3D Modeler** window, select the start point of the vector, (2, 0, 0), by clicking the edge center at the bottom of the face. By default, the cursor should snap to this point, appearing as a triangle.



7 Select the end point (2, 0, 0.4) by clicking the edge center

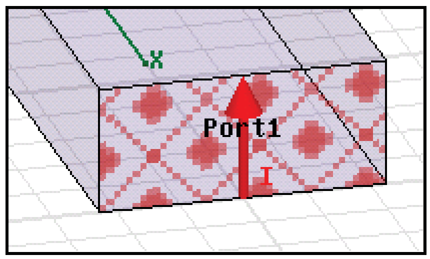
at the top of the face.



The **Wave Port** dialog box reappears.

8 To accept the remaining default settings, click **Next**.

9 Click **Finish**.




The assigned port.

## Duplicate the Box

Now you will duplicate the box to create the second and third sections of the T-junction. The attributes of the box will be duplicated along with its geometry, boundary assignments, and excitations, including wave port settings, can be duplicated along with the geometry if the option is set in the **HFSS Options** dialog box. In this example, you will make sure this setting is selected.

### Set Duplicates to Copy Boundaries

1 Click **Tools>Options>HFSS Options**.

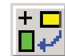
2 Under the **General** tab of the **HFSS Options** dialog box, select **Duplicate boundaries with geometry**, and then click **OK**.

### Duplicate the Box to Create the Second Section

Duplicate the box 90 degrees around the z-axis to create the second section.

1 Click **Tee** in the history tree to select the object.

2 In the 3D modeler window, right-click to display the short-

cut menu and select **Edit>Duplicate> Around Axis**  .

**3** In the **Duplicate Around Axis** dialog box, select **Z**.

**4** Type **90** in the **Angle** box. A positive angle causes the object to be placed in the counter-clockwise direction.

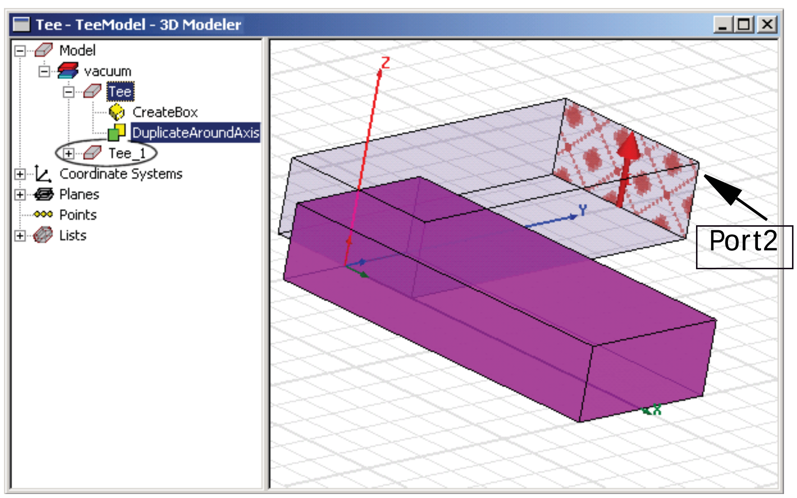
**5** Type **2** in the **Total number** box. This is the total number of objects, including the original, that will be created.

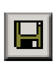
**6** See that **Attach to original object** is not checked in this panel.

**7** Click **OK**.

*The history tree shows that the Tee object was duplicated and a new object, named Tee_1, was created.*

The parent object, *Tee*, is duplicated, and the duplicate, named *Tee_1* by default, is placed around the z-axis at a 90-degree angle. The attributes of the parent object, including its dimensions, material, color, transparency, port, and integration line are duplicated with the box.




*Save your project frequently: Click **File>Save**.*  .

Port1 was duplicated with the geometry of the box. The new port is named Port2 by default, which you can verify under **Excitations** in the project tree.

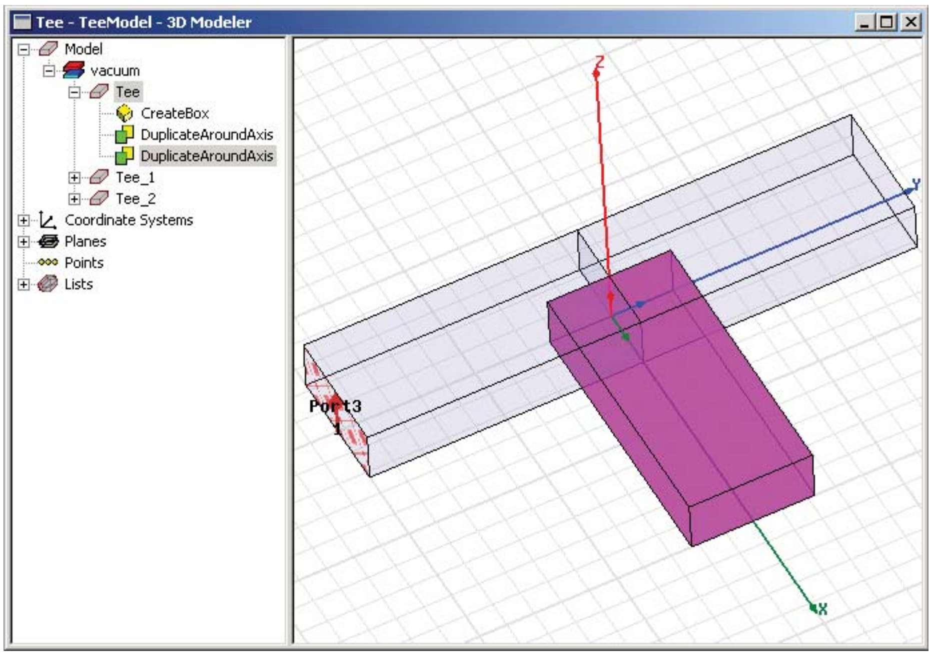
**8** Click **OK** to close the Properties dialog.

**9** Press **Ctrl+D** to fit the objects in the view window.

## Duplicate the Box to Create the Third Section

**1** Duplicate the first box again using the same procedure, but this time, type **-90** in the **Angle** box. A negative angle causes the object to be placed in the clockwise direction.

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

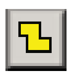



The parent object, still selected, and its duplicates.

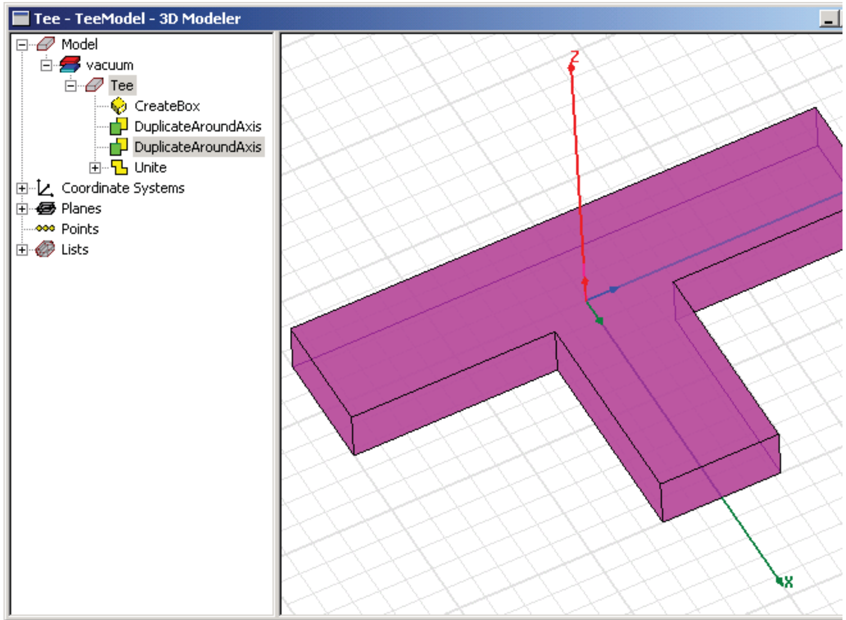
## Unite the Boxes

Now you will unite the three sections to create the complete T-junction. Before doing this, you want to be sure that HFSS will not create copies of the original objects before joining them, so you will clear the “clone before unite” option in the **3D Modeler Options** dialog box.

1. Click **Tools>Options>Modeler Options**.

   The **3D Modeler Options** dialog appears.
2. Under the **Operation** tab, make sure the **Clone tool objects before uniting** option is clear
3. Click **OK**.
4. Switch to object selection mode by pressing the shortcut key **O**.
5. Select the first box by clicking it in the view window.
6. Hold the **Ctrl** key and click the second and third boxes.
7. On the **3D Modeler** menu, point to **Boolean**, and then click **Unite**  .

   The objects are united at the points of intersection. The new object has the same attributes as the first object selected.




The united object.

# Create the Septum

The septum is a 3D box object that will be subtracted from the T-junction. When you draw the septum, you make its y position dependent on a variable.

## Draw a Box

This time when you draw a box, you will draw it freehand, and then modify its dimensions and position in the **Properties** window.

1 On the **Draw** menu, click **Box** .

*As a guideline, aim for the first point to be near the coordinates (-0.45, 0, 0), the second point near (0.45, 0.1, 0), and the third point near (0, 0, 0.4).*

2 Draw an arbitrarily shaped box in the **3D Modeler** window: Select a corner of the base rectangle, then select a second corner of the base rectangle, and then select a point on the axis perpendicular to the base rectangle.

   When you have selected the last point of the box, the **Properties** window appears, with the **Command** tab selected.

   Now you will assign the box's exact position and dimensions.

## Parameterize the Position of the Box

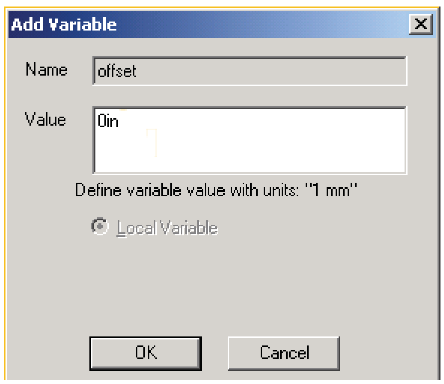
When you specify the box's position, you will enter the following expression for the y position: *offset - 0.05*, where *offset* is the name of a variable you will define. Because the variable *offset* is not yet defined when you type it in the expression, the **Add Variable** dialog box appears, enabling you to define value for *offset*.

When you specify the variable's value, you must include its unit of measurement as part of the value.

1 In the **Properties** window, under the **Command** tab, in the **Position** text box, type **-0.45in, offset - 0.05in, 0in** and then press **Enter**.

The **Add Variable** dialog box appears.

*Alternatively, you could define the variable offset before you draw the septum. Local variables can be defined in the* ***Properties*** *window, which is accessed by right-clicking the design in the project tree, and then clicking* ***Design Properties.***




2 Type **0in** in the **Value** text box, and then click **OK**.

You return to the **Properties** window.

Now you will set the exact dimensions of the box.

## Modify the Dimensions of the Box

1 In the **Properties** window, under the **Command** tab, type **0.45** in the **Xsize** box.

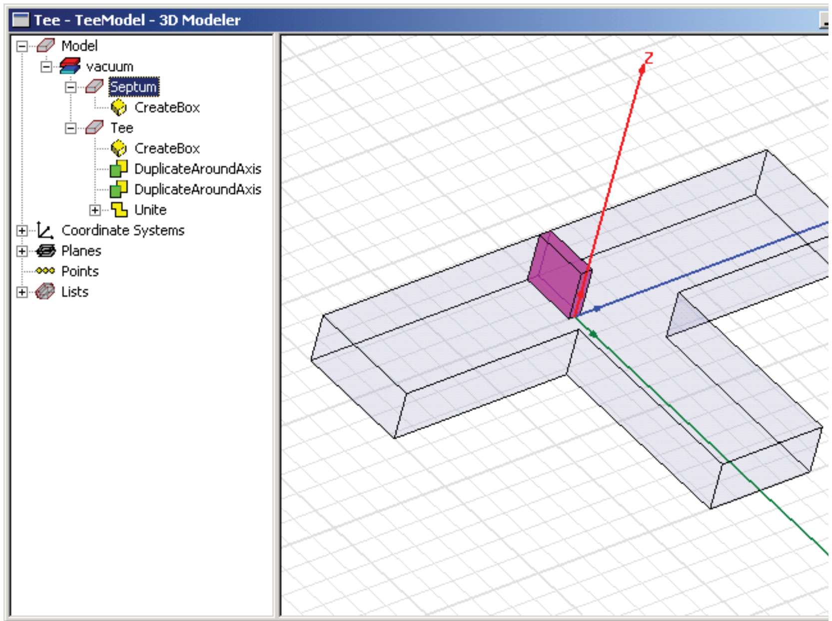
2 Type **0.1** in the **Ysize** box.

3 Type **0.4** in the **Zsize** box.

While the **Properties** window is open, you will assign a name to the box.

## Assign a Name to the Box

1. In the **Properties** window, click the **Attribute** tab.
2. Type **Septum** in the **Value** text box in the **Name** row.
3. Click **OK**.

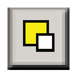



The septum object in the **3D Modeler** window.

Optionally, rotate the view to get a better view of the septum object: Press **Alt** and drag the mouse in the direction you want to rotate the view.

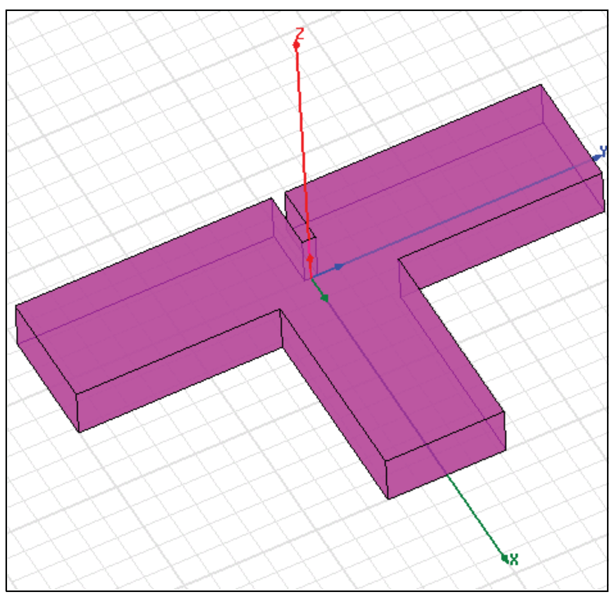
## Subtract the Septum from the T-Junction

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

1. Click **Tee** in the history tree to select the tee object.
2. Hold down the **Ctrl** key and click **Septum** in the history tree to select the septum.
3. On the **Modeler** menu, point to **Boolean**, and then click **Subtract** .

   The **Subtract** dialog box appears. **Septum** is listed in the **Tool Parts** list, and **Tee** is listed in the **Blank Parts** list, indicating that the septum object will be subtracted from the tee object.
4. Make sure the **Clone tool objects before subtracting** option is clear.
5. Click **OK**.

   The septum is subtracted from the tee. The new object has the same attributes as the first object you selected, the tee object.




The complete model geometry.

# 4

# Set Up and Generate Solutions

In this chapter you will complete the following tasks:

- ✔ Add a solution setup.
- ✔ Add a frequency sweep to the solution setup.
- ✔ Validate the design.
- ✔ Run the analysis.
- ✔ Modify the septum's position.
- ✔ Re-run the analysis using the new septum position.

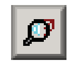
*Estimated time to complete this chapter: 15 minutes.*

# Add a Solution Setup to the Design

Specify how HFSS will compute the solution by adding a *solution setup* to the design.

*To learn more about solution parameters, see the HFSS online help.*

In the solution setup, you will instruct HFSS to perform an adaptive analysis at 10 GHz. During an adaptive analysis, HFSS refines the mesh iteratively in the areas of highest error.

1 In the project tree, under the TeeModel design, right-click **Analysis**, and then click **Add Solution Setup**  on the shortcut menu.

   The **Solution Setup** dialog box appears.

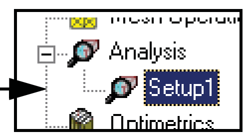
*The adaptive analysis will be performed at the solution frequency, 10 GHz.*

2 Under the **General** tab, type **10** in the **Solution Frequency** text box, and leave the default unit set to **GHz**.

3 Leave the **Maximum Number of Passes** set to 6. This is the maximum number of mesh refinement cycles that HFSS will perform.

4 Leave the default settings and click **OK**.

   The solution setup is listed in the project tree under **Analysis**. It is named *Setup1* by default.

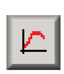



You want HFSS to solve over a range of frequencies, so you will now add a frequency sweep to the solution setup.

## Add a Frequency Sweep to the Solution Setup

*For the frequency sweep, HFSS will use the finite element mesh refined during the adaptive solution.*

A smooth frequency response is expected for this design, so you will select an interpolating frequency sweep. 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 Right-click **Setup1** in the project tree, and then click **Add Sweep** .

The **Edit Sweep** dialog box appears.

2 Select **Interpolating**.

3 Leave the default settings for **Error Tolerance** and **Max Solutions**.

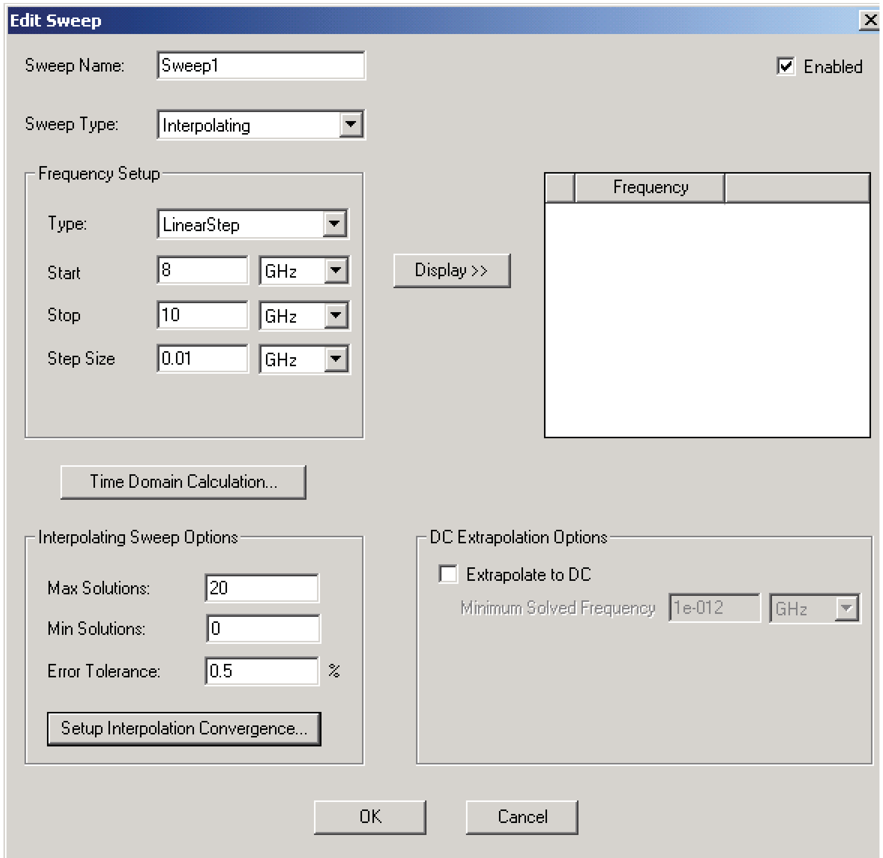
4 Click **Linear Step** in the **Type** pulldown list.

5 Specify the following range of frequencies:

| | |
|---|---|
| **Start** | **8 GHz** |
| **Stop** | **10 GHz** |
| **Step Size** | **0.01 GHz** |

HFSS solves the frequency point at each step in the specified frequency range, including the start and stop frequencies.

*Click **Display** to verify the frequency points that will be solved.*




**6** Click **OK**.

The frequency sweep is listed in the project tree under **Setup1** named *Sweep1* by default.

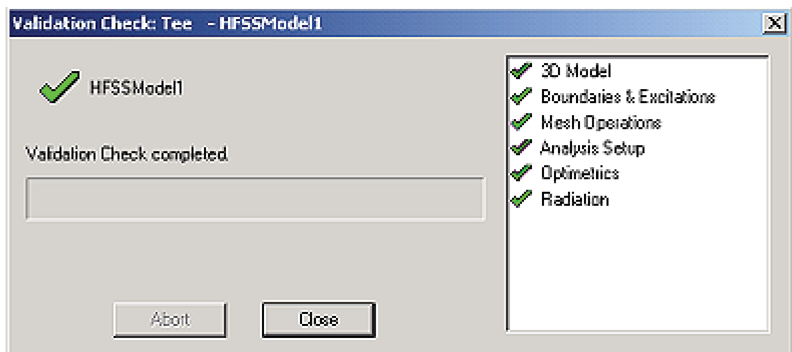
## Validate the Design

Before you run an analysis, it is helpful to verify that all of the necessary setup steps have been completed and their parameters are reasonable.

**1** On the **HFSS** menu, click **Validation Check** .

HFSS checks the project setup, and then the **Validation Check** window appears.

*If there is a problem with the design setup, the Message Manager lists detailed error or warning messages. Click **View>Message Manager** to display the Message Manager.*



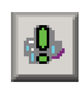

**2** Click **Close**.

Now you are ready to run the simulation.

## Analyze the Design

*You should save the project before running the simulation. To set automatic save of projects before solving, click **Tools>Options>HFSS Options**. Under the **General** tab, select **Save before solving**. Click OK*

Now you will run the simulation, which will generate results for the T-junction when the septum is located centrally opposite Port 1.

- On the **HFSS** menu, click **Analyze All** .
  HFSS computes the 3D field solution for every solution setup in the project. In this problem, Setup1 is the only setup.

*You can monitor the solution's progress in the **Progress** window at the bottom of the screen. If the Progress window is not visible, click **View>Progress Win-***

The solution process is expected to take approximately 1 - 5 minutes.

When the solution is complete, a confirmation message appears in the **Message Manager**.

## Move the Position of the Septum

When the analysis is complete, modify the septum's position by changing the value of the variable *offset*.

1 Right-click the design name **TeeModel** in the project tree, and then click **Design Properties**.

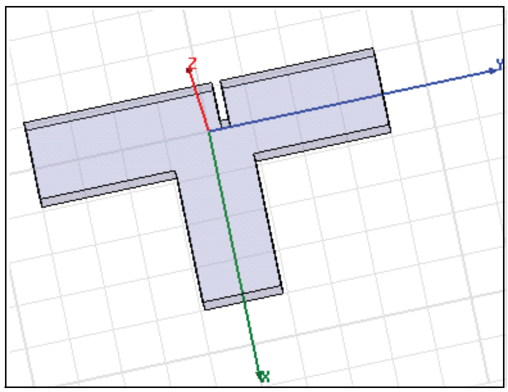
The **Properties** dialog box appears.

2 Under the **Local Variables** tab, select **Value**.

3 Type **0.2** in the **Value** text box for the variable *offset*.

4 Click **OK**.

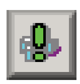
The geometry is updated in the **3D Modeler** window.




Top-down view of the septum in its new position, closer towards Port 2.

## Re-analyze the Design

Now you will run a second simulation to generate results for the T-junction when the septum is located closer to Port 2. The previous solution is saved and available for post processing.

- Right-click **Analysis** in the project tree, and then click **Analyze All**  .

HFSS computes the new 3D field solution.

The solution process is expected to take approximately 1 - 5 minutes.

Proceed to the next step in the next chapter, creating a 2D x-y plot of S-parameter results, while the analysis is running.

HFSS will populate the plot with data when the solution is complete.

5

# Compare the Solutions

In this chapter you will complete the following tasks:

- Create a 2D x-y plot of S-parameters.
- Create a field overlay plot on a surface of the T-junction.
- Animate the field overlay plot.
- Modify the septum's position and re-animate the field overlay plot.
- Close the project and exit HFSS.

*Estimated time to complete this chapter: 10 minutes.*

# Create a Rectangular Plot of S-Parameter Results

Now you will create a 2D x-y (rectangular) plot that compares the S-parameter results at each port for the two septum positions.

1. Right-click **Results** in the project tree, and then click **Create Modal Solution Data Report>Rectangular Plot**.

   The **Report** dialog appears with the **Trace** tab selected.
2. In the Context area select the Solution as Setup1:Sweep1 and the Domain as Sweep.
3. In the **X** area, select *Freq*, which HFSS recognizes as the frequency points solved during the frequency sweep.
4. In the **Y** area, specify the information to plot along the y-axis:

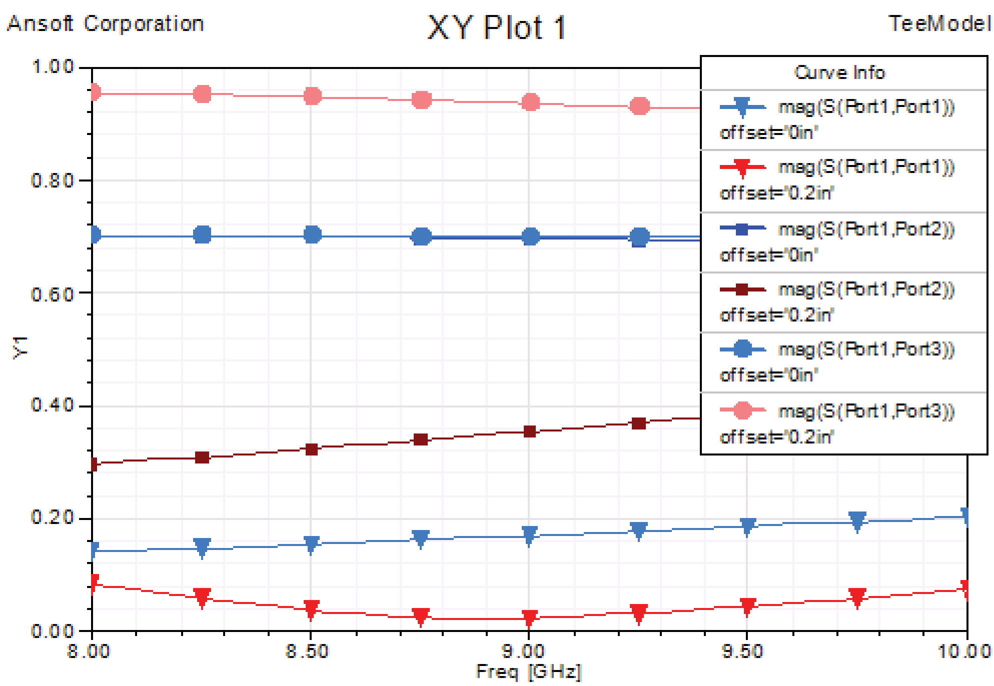
   a. In the **Category** list, click **S parameter**.

   b. In the **Quantity** list, press **Ctrl** and click **S(Port1, Port1)**, **S(Port1, Port2)**, and **S(Port1, Port3)**.

   c. In the **Function** list, click **mag**.
5. Click **New Report**.

   A trace represents a line connecting data points on the plot.
6. Click **Close**.

   The magnitude of the S-parameters at each *offset* value will be plotted against frequency on an x-y graph, as shown on the next page. The plot is listed under **Results** in the project tree along with its three corrsponding trace definitions.




*The line styles in the plot were modified in the **Trace Properties** dialog box for better visualization.*

*To add data markers to all lines on the plot as shown: Double-click a line. In the **Trace Properties** dialog box, click the **Attributes** tab, select **Show Symbols**, and then type **5** in the text box for Symbol Frequency. To change a line's color: Click the **Color** property and modify the selected line's color.*

The three blue-shaded lines show the S-parameter values at each port when offset = 0 in. The three red-shaded lines show the S-parameter values at each port when offset = 0.2 in. The line styles in the plot above were modified in the**Traces Properties** dialog box for better visualization.

As expected, minor reflection near 0.2 is occurring at the input port, Port 1, when the value of the *offset* variable is 0in—that is, when the septum is located centrally opposite to Port 1. At the same time, an equal transmission near 0.7 occurs at the two output ports, Port 2 and Port 3.

The reflection at Port 1 decreases slightly when the value *offset* is 0.2 inches—that is, when the septum is moved 0.2 inches toward Port 2. The transmission at Port 2 decreases and the transmission at Port 3 increases with the septum at this position.

Next you will create and animate a field overlay plot that displays the difference in field pattern between the two septum positions.

# Create a Field Overlay Plot

A field overlay plot is a representation of 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 T-junction. First, move the septum back to its original position centrally opposite Port 1.

## Modify the Position of the Septum

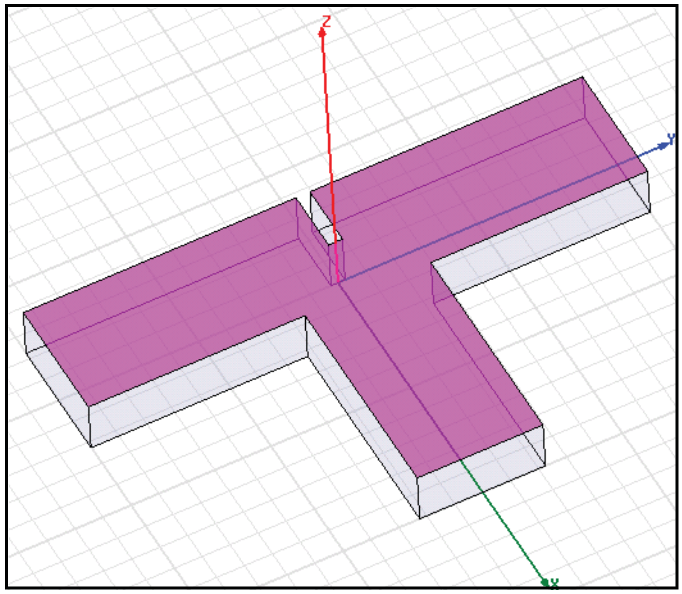
Change the value of the variable *offset* back to 0 inches:

1. Make sure the **Property** window is displayed. If it is not, click **View> Property Window**.
2. Click the design name **TeeModel** in the project tree.
3. Under the **Variables** tab in the **Properties** window, type **0** in the **Value** text box for the variable *offset*, and then press **Enter**.

## Create the Field Plot

1. Return to the **3D Modeler** window.
2. Switch to face selection mode: Right-click in the view window, and then click **Select Faces** on the shortcut menu.

**3** Select the top face of the T-junction:




*You can also open the* ***Create Field Plot*** *dialog by right-clicking the view window and then clicking* ***Plot Fields>Mag_E*** *on the shortcut menu.*

**4** Click **HFSS>Fields>Plot Fields>E>Mag_E**.

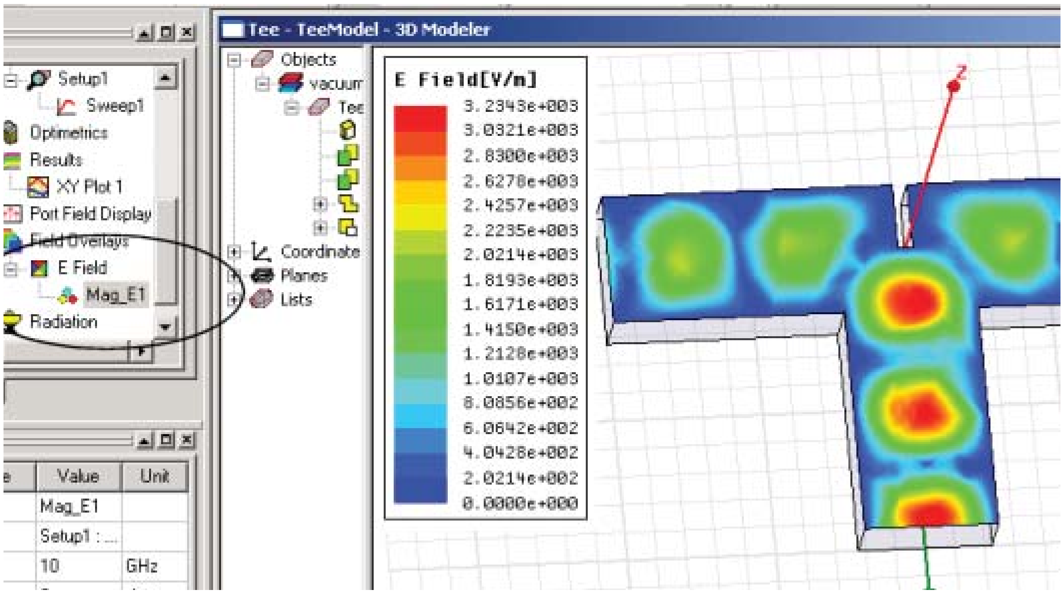
The **Create Field Plot** dialog box appears.

**5** Select **Setup1:LastAdaptive** as the solution to plot in the **Solution** pull-down list.

**6** Accept the default settings by clicking **Done**.

The plot appears on the top surface of the T-junction. It shows the E-field distributed evenly towards Port 2 and Port 3.

The new plot is listed under **Field Overlays** in the project tree. It is named *Mag_E1*, which was the default name set in the **Create Field Plot** dialog box.




*To hide the color key that appears in the upper-left corner of the **3D Modeler** window: Right-click the color key, and then click **Hide**.*

The *Mag_E1* plot of the E-field when the septum is located opposite Port 1. The new plot is listed in a default folder under **Field Overlays** in the project tree.

Now you will animate the field overlay plot.

## Animate the Field Overlay Plot

An animated plot is a series of frames that displays a field, mesh, or geometry at varying values. You specify the values of the plot that you want to include, just as an animator takes snapshots of individual drawings that make up a cartoon. Each value is a frame in the animation.

1. Right-click **Mag_E1** in the project tree, and then click **Animate**.

   The **Select Animation** dialog box appears.

2. Click **New...**

   The **Setup Animation** dialog appears.

3. Under the **Swept Variable** tab, click **Phase** on the **Swept Variable** list.

4. Specify the phase values to include in the animation:

   a.Type **0deg** in the **Start** text box.

   b.Type **160deg** in the **Stop** text box.

   c.Type **8** in the **Steps** text box.

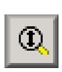
*Animations can be exported to animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format by clicking **Export** in the **Animation** dialog box. that appears.*

5. Click **OK**.

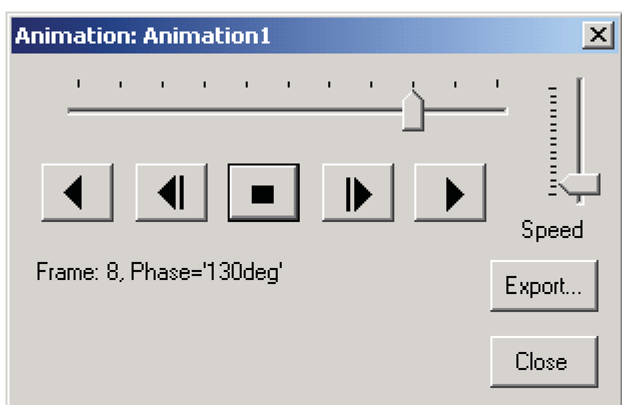
   The animation begins in the view window. It shows the septum steering the electromagnetic wave evenly toward Port 2 and Port 3.

   The **Animation** dialog box appears in the upper-left corner of the desktop, enabling you to stop, restart, and control the speed and sequence of the frames.

*You can modify the view of the animation while it is running. For example, click the Zoom In or Zoom Out button*



*in the toolbar and drag the mouse towards the top (to zoom in) or bottom (to zoom out) of the view window.*



The animation displays the plot at 8 phase values between 0 and 160. The start value is the first frame displayed, resulting in a total of 9 frames in the animation.

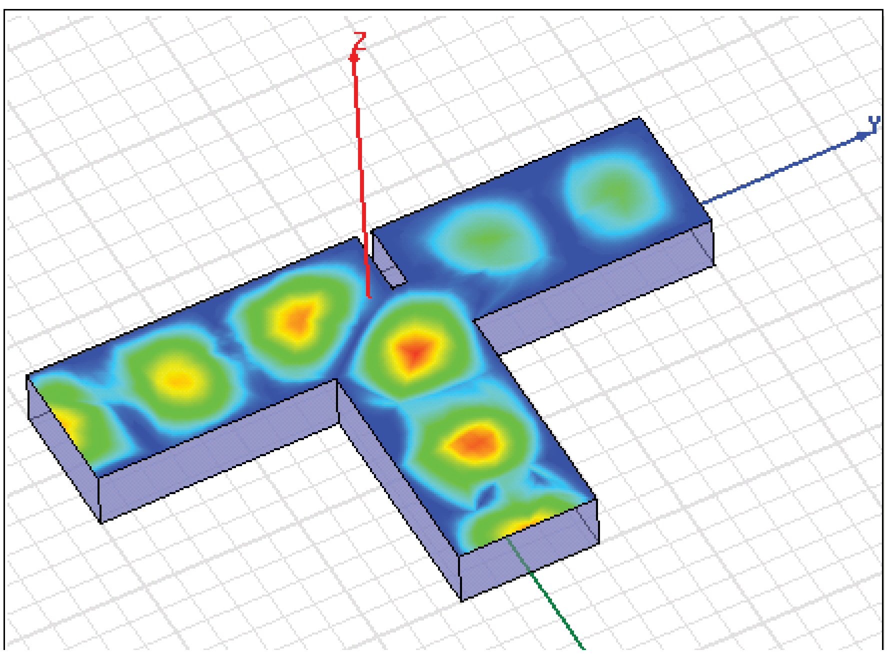
6 In the **Animation** dialog box, click the stop button .

## Modify the Septum's Position and Re-animate

Now you will move the septum's position closer to Port 2 to see its effect on the E-field pattern on the T-junction's top surface.

1 Click the design name **TeeModel** in the project tree. You may need to drag the **Animation** dialog box aside to see the design name in the project tree.

2 Under the **Variables** tab in the **Properties** window, type **0.2** in the **Value** text box for *offset*, and then press **Enter**.
Click the play button to start the updated animation. It shows more of the electromagnetic wave moving towards Port 3 than toward Port 2.

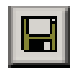



The animated *Mag_E1* plot of the E-field when the septum is located 0.2 inches closer to Port 2.

## Close the Project and Exit HFSS

Congratulations! You have successfully completed this HFSS *Getting Started* guide! You may stop the animation, close the *Tee* project, and exit the software.

1 In the **Animation** dialog box, click the stop button , and then click **Close**.

2 Save the project .

3 Click **File>Close**.

4 Click **File>Exit**.

*At this point, you may choose to go on to Getting Started with Optimetrics: Optimizing a Waveguide T-Junction Using HFSS and Optimetrics. It uses the T-junction design you created in this guide to demonstrate Ansoft's Optimetrics software.*

# Index

## R



## S



## T



## U



## V



## W