Microwave Engineering Practices

Theory Class Instructor: Prof. Yu Ming

Lab Instructor: Dr. Wu Guang

Lab Timing: Thursday (10:00 – 12:00)

Office: 130, No.1 Teaching Building

Experiment: 1

Design of a Waveguide T-junction

Southern University of Science and Technology, Shenzhen, P.R. China

1. A Waveguide T-Junction

This guide leads you step-by-step through creating, solving, and analyzing the results of a waveguide T-junction.

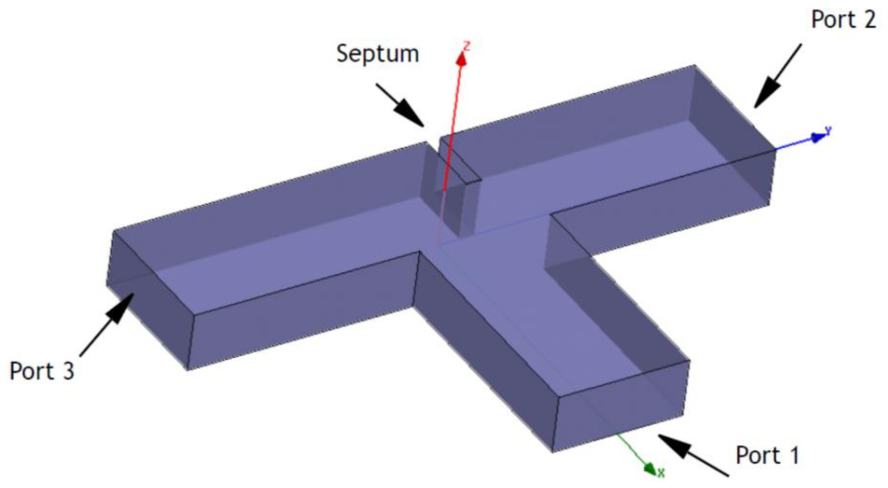
By following the steps in this guide, you will learn how to perform the following tasks in HFSS:

- 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.

About the T-Junction

The waveguide you will create is T-shaped with an inductive septum. 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 50.8 mm 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.

2. Set up the Design

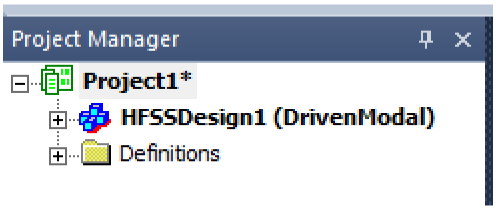
In this section you will complete the following tasks:

- 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.

Open HFSS and Save a New Project

1. Double-click the **HFSS 19** icon on your desktop to launch HFSS.

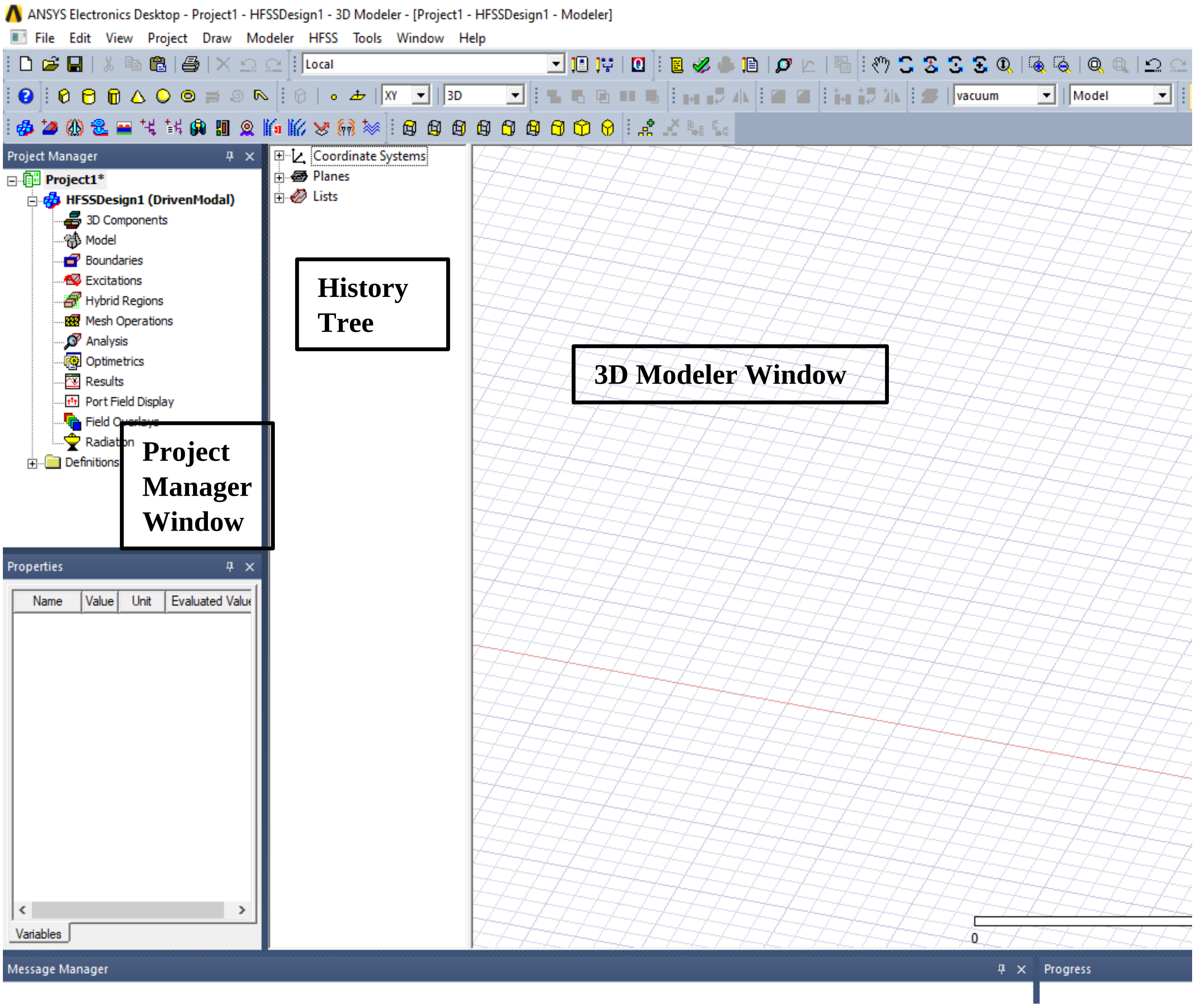
A new project is listed in the project tree in the **Project Manager** window and is named Projectn by default. Project definitions, such as material assignments, are stored under the project name.



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\HFSS19.0\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 HFSSDesignn by default. The **3D Modeler** window appears to the right of the Project Manager.



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

Select a Solution Type

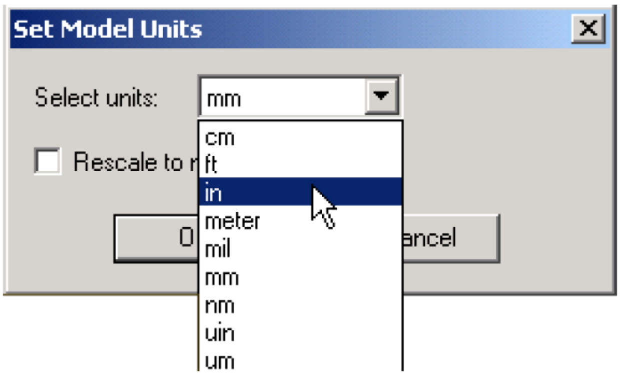
As you set up the design for analysis, available settings depend on the solution type. For this design, you will choose Driven Modal as the solution type, which is appropriate when calculating mode-based S-parameters of a passive, high-frequency waveguide that is being “driven” by a source.

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

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

1. Click **Modeler>Units**. The **Set Model Units** dialog appears
2. Select **mm** from the **Select units** pull-down list, and then click **OK**.



3. Create the Model

In this section 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.

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

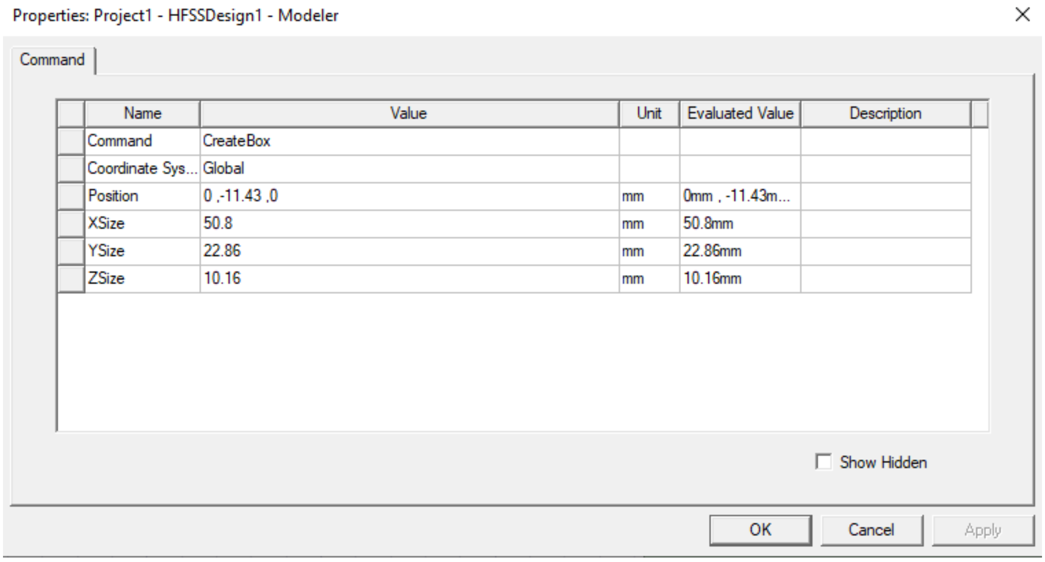
Draw a 3D box object to represent the first section of the tee.

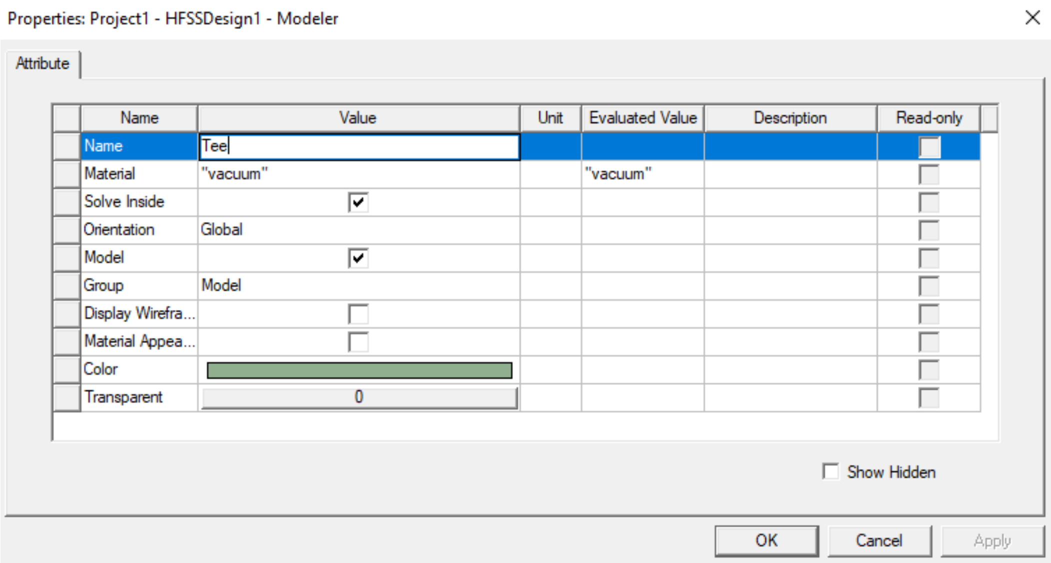
1. Ensure the option **Edit Properties of new primitives** is selected in **Tools->Options>Modeler Options->Drawing Tab**.
2. On the **Draw menu**, click **Box** .
3. Find the coordinate fields at the bottom of the HFSS window, labeled "Enter the box position," and specify the base corner of the box as (**0, −11.43, 0**):

   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 **−11.43** in the Y box, and then press **Tab**.
   d. Type **0** in the **Z** box, and then press **Enter**.

4. Specify the length and width of the box by entering a point relative in distance to the base corner: Type (**50.8, 22.86, 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, 10.16**) 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.





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

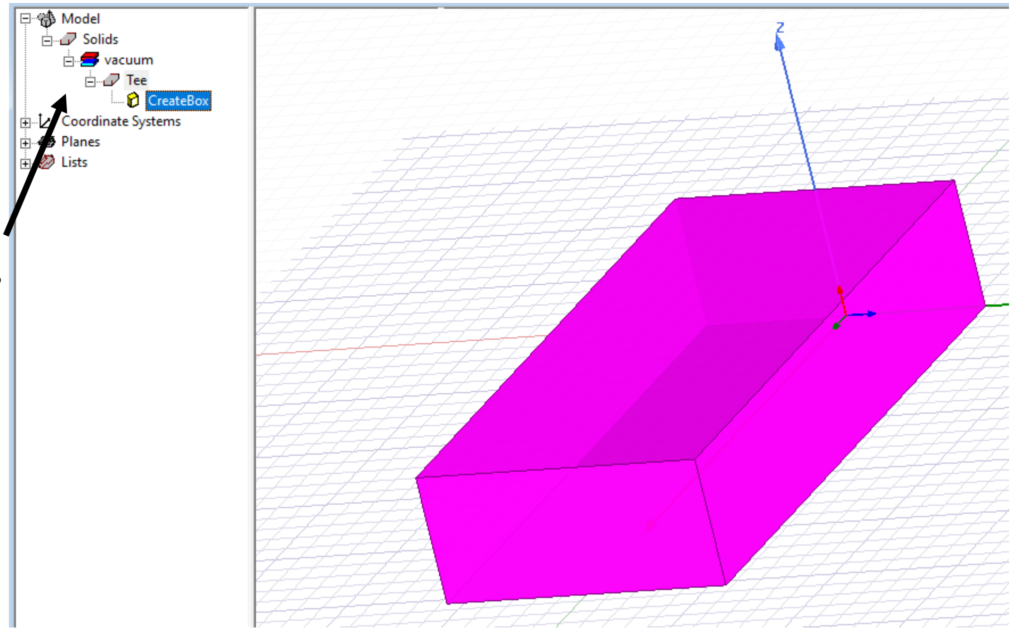
By default, the material assigned to the box is “vacuum”. This is the material you will use for the T-junction. Confirm that vacuum is the value in the Material row so you do not need to change it.

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.

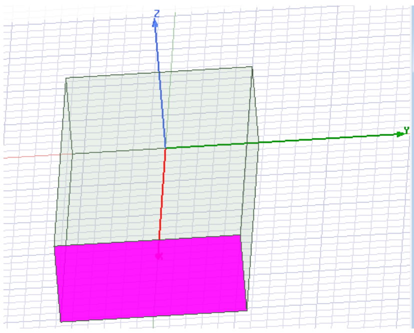
The name *Tee* was assigned to the box. The commands performed on the box are tracked in the history tree.



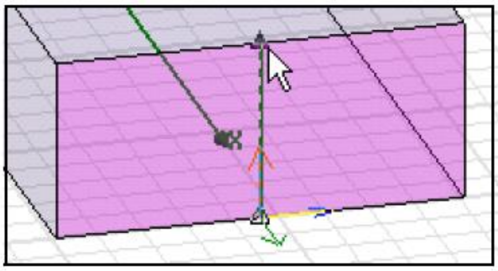
The first box object in the **3D Modeler** window. It is selected by default when you exit the **Properties** window.

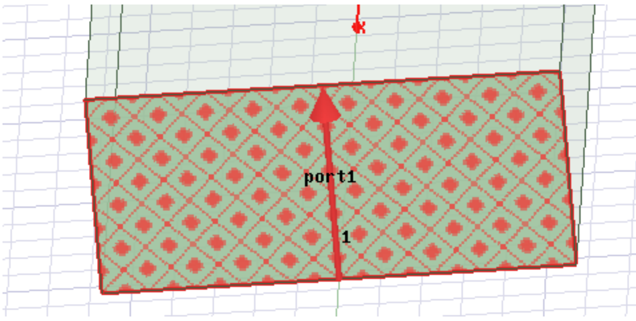
Assign a Wave Port to the Box

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.



7. Switch to face selection mode by pressing the shortcut key F.
8. Click the face of the box that is parallel to the yz plane at x = 50.8, as shown to the right.
9. Right-click the **3D Modeler** window, and then click **Assign Excitation>Wave Port** on the shortcut menu. The **Wave Port** wizard appears.
10. Type **Port1** in the **Name** text box, and then click **Next**.
11. Select **New Line** from the **Integration Line** pull-down list.
12. In the **3D Modeler** window, select the start point of the vector, (**50.8, 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.
13. Select the end point (50.8, 0, 10.16) by clicking the edge center The face of the box that is parallel to the yz plane at x = 50.8. at the top of the face.





The **Wave Port** dialog box reappears.

The assigned port.

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

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**.

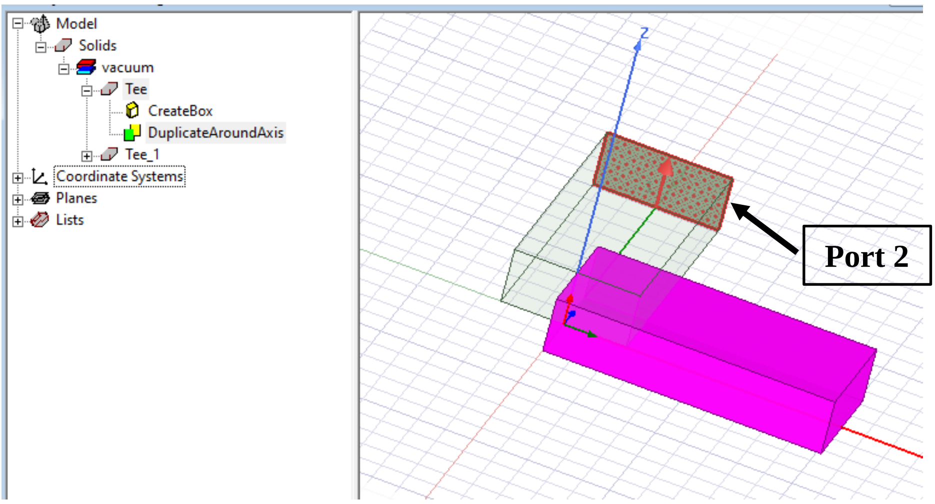
Duplicate the Box to Create the Second Section

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 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.

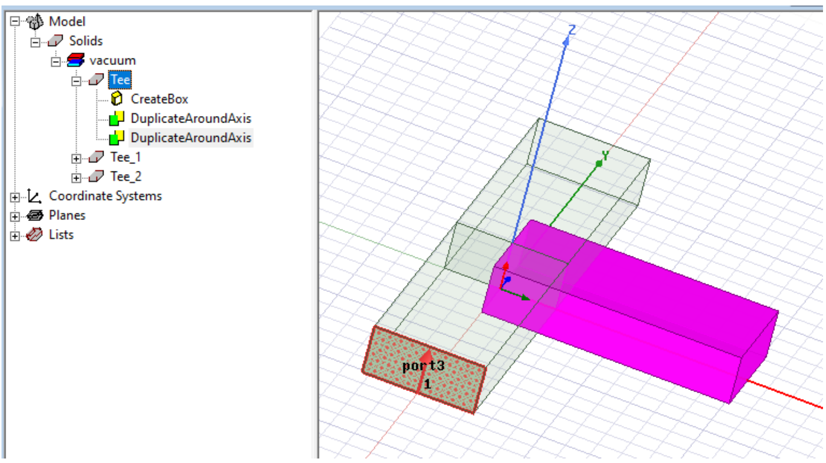


Port1 was duplicated with the geometry of the box. The new port is named Port2 by default, which you can verify under Excitations in the project tree.

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

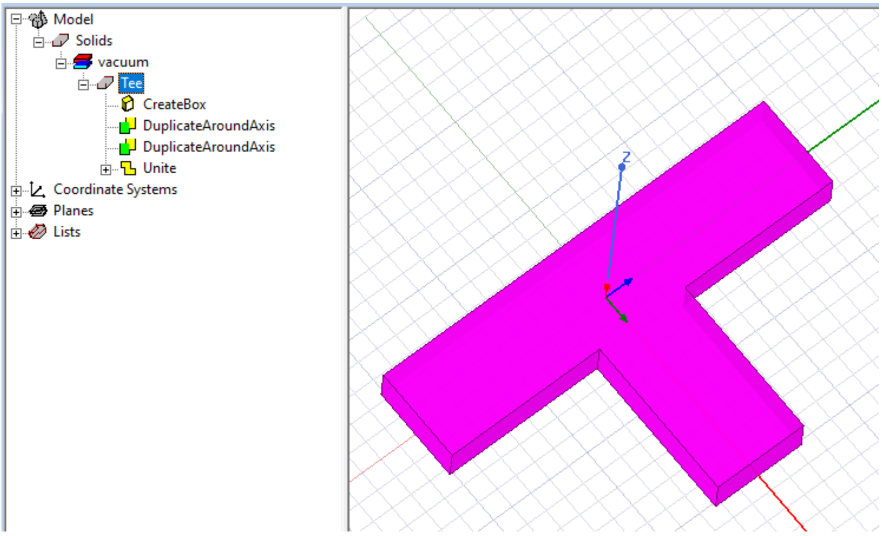
Now you will unite the three sections to create the complete T-junction. Before doing this, you want to be sure that HFSS will not create copies of the original objects before joining them, so you will clear the “clone before unite” option in the **3D Modeler Options** dialog box.

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.



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** .
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.

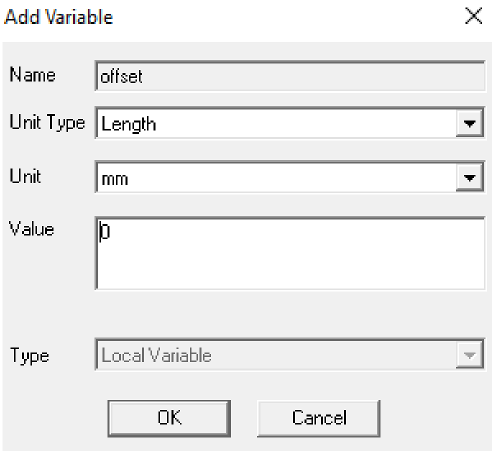
Now you will assign the box’s exact position and dimensions.

Parameterize the Position of the Box

When you specify the box’s position, you will enter the following expression for the y position: *offset* – 1.27, 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 **−11.43mm**, **offset – 1.27mm**, 0mm and then press **Enter**.
   The **Add Variable** dialog box appears.



2. Type **0mm** 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.

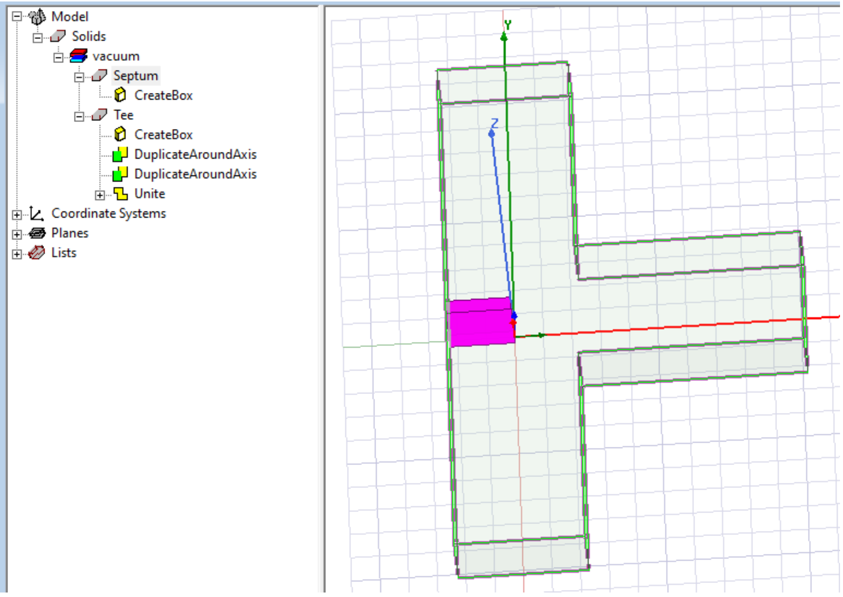
Modify the Dimensions of the Box

1. In the Properties window, under the Command tab, type **11.43** in the **Xsize** box.
2. Type **2.54** in the **Ysize** box.
3. Type **10.16** 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

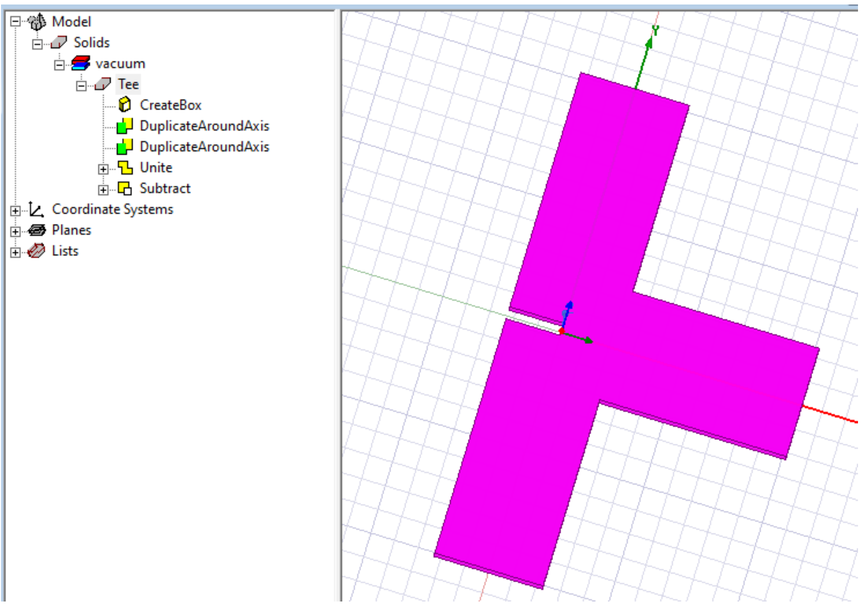
To complete the model geometry, you will now subtract the septum object from the T-junction.

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 section 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.

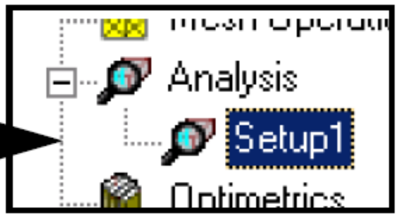
Add a Solution Setup to the Design

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

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.
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

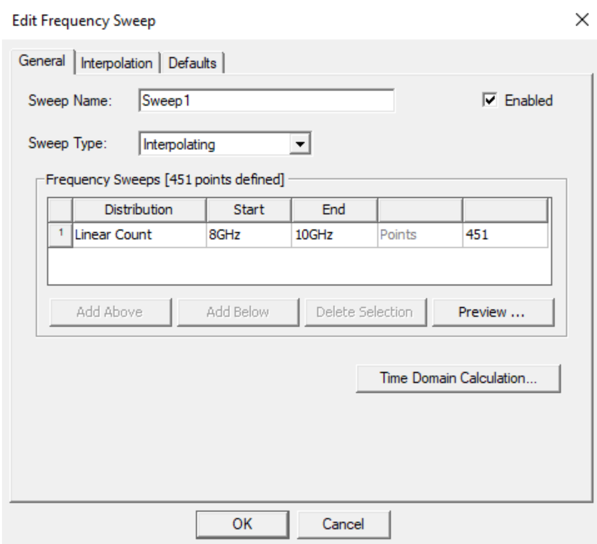
A smooth frequency response is expected for this design, so you will select an interpolating frequency sweep. An Interpolating sweep estimates a solution for an entire frequency range. HFSS chooses the frequency points at which to solve the field solution so that the entire interpolated solution lies 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
   Points 451

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



6. Click **OK**.

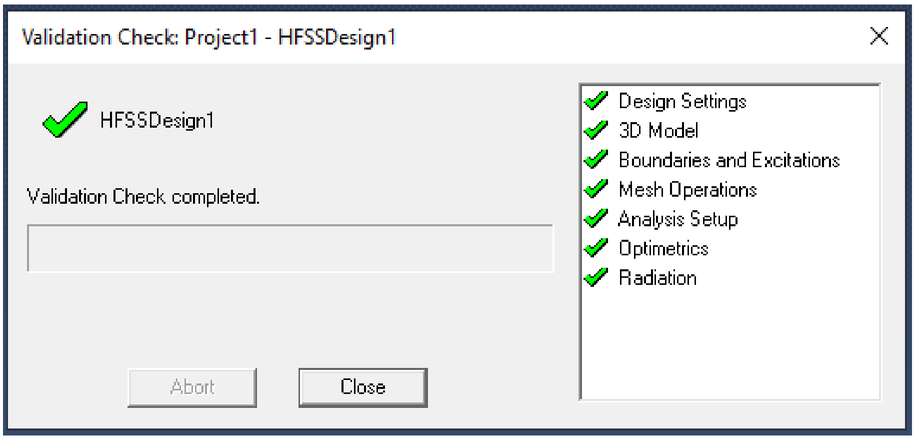
The frequency sweep is listed in the project tree under **Setup1** named **Sweep1** by default.

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.



2. Click **Close**.

Now you are ready to run the simulation.

Analyze the Design

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.

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

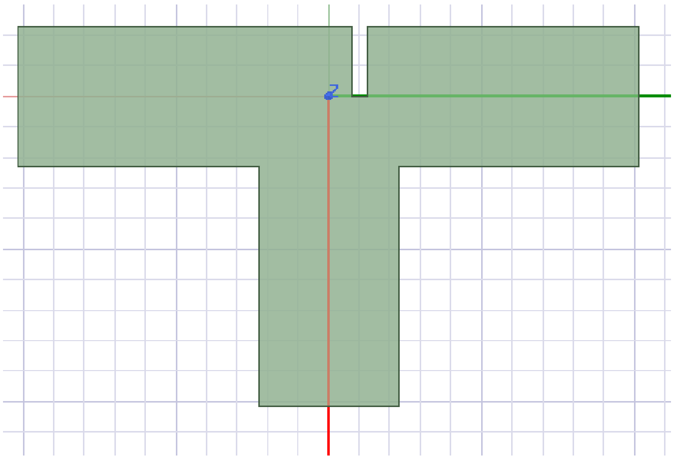
When the analysis is complete, modify the septum's position by changing the value of the variable *offset*.

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 Variable**s tab, select Value.
3. Type **5.08** 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 xy 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 Section 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.

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**.

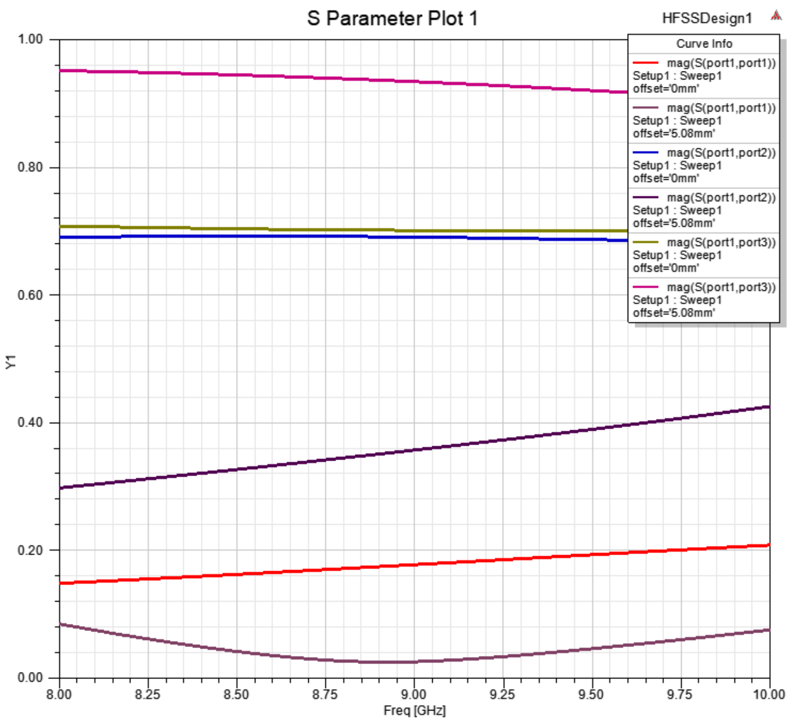
   The **Report** dialog appears with the **Trace** tab selected.

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 corresponding trace definitions.



The three blue-shaded lines show the S-parameter values at each port when offset = 0 mm. The three red-shaded lines show the S-parameter values at each port when offset

= 5.08 mm. 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 0mm—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 5.08mm—that is, when the septum is moved 5.08mm 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.

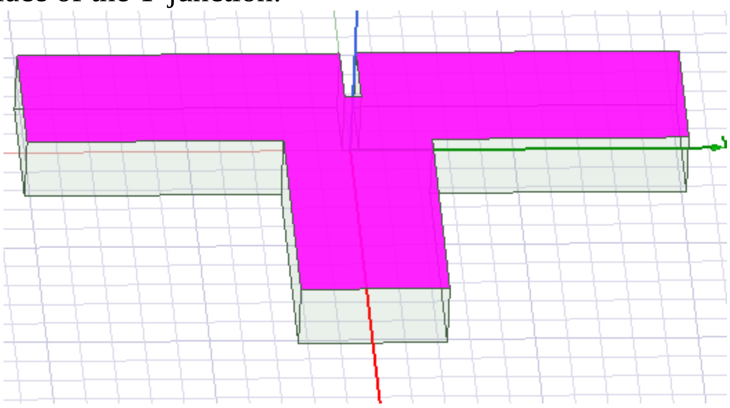
Modify the Position of the Septum

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

1. Make sure the **Property** window is displayed. If it is not, click **View**> **Property Window**.
2. Click the design name **Tee Model** 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:

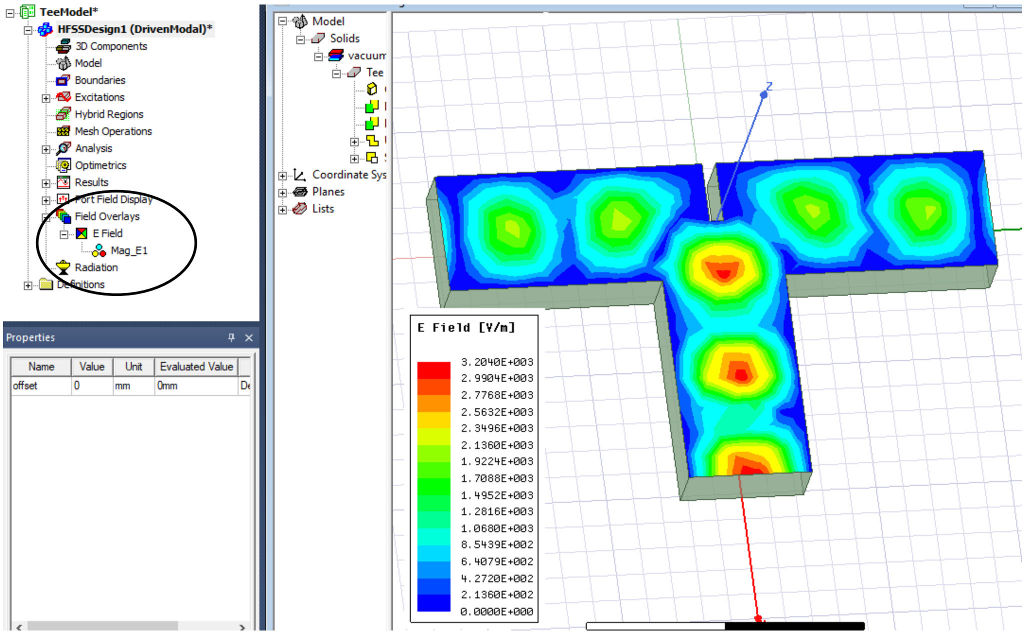


4. Click **HFSS>Fields>Plot Fields>E>Mag_E**.
   The **Create Field Plot** dialog box appears.
5. Select **Setup1:LastAdaptive** as the solution to plot in the Solution pull-down list.

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.



   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**.

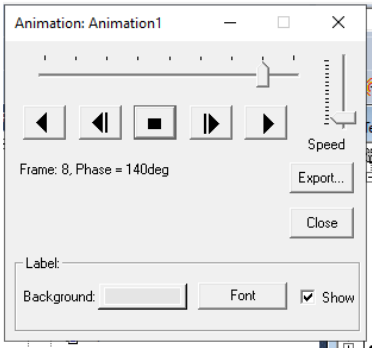
   The **Select Animation** dialog box appears.

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.
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.

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.

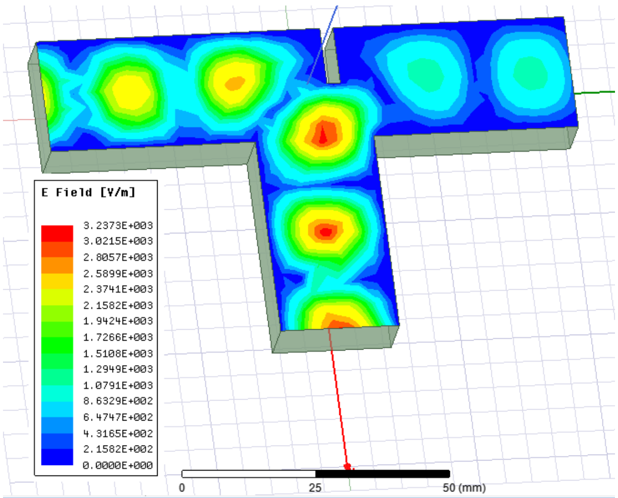
6. In the **Animation** dialog box, click the stop button .

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 5.08mm 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 5.08mm closer to Port 2.

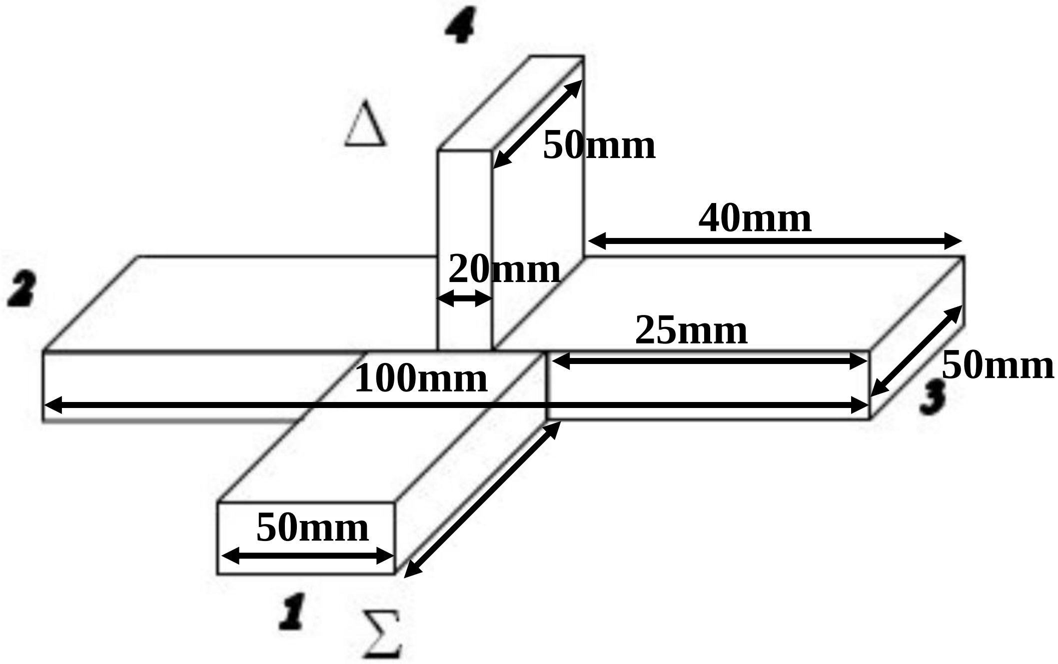
Close the Project and Exit HFSS

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**.

Prepare a report on T-junction magic tee with four port using HFSS

1. The above procedure can be followed to model and analyse the schematic as shown in figure below.



A magic T-junction with four port