Getting Started with HFSS™ 3D Layout: Microstrip Filter

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

Conventions Used in this Guide

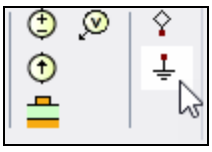
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates 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 (e.g., "**copy file1** " means type the word **copy**, then type a space, then type **file1**).
  - On-screen prompts and messages, names of options and text fields, and menu commands. Menu commands are often separated by greater than signs (>) (e.g., "Click **HFSS** > **Excitations** > **Assign** > **Wave Port**).
  - Labeled keys from 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 (e.g., "**copy***file name*" means type the word **copy**, then type a space, then type the name of the file).
- The plus sign (+) is used between keyboard keys to indicate that both keys should be pressed at the same time (e.g., "Press **Shift** +**F1** " means to press **Shift** and, while holding it down, press **F1**). Always depress the modifier key or keys first (e.g., **Shift**, **Ctrl**, **Alt**, or **Ctrl** +**Shift**), continue to hold it/them down, 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 **Layout** > **Interface Ground** "



This instruction means click the **Interface Ground** command from the **Layout** 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:

  "From the **File** menu, select **Open Examples**" means click the **File** menu and select **Open Examples** from the drop-down menu.

- Another alternative is to right-click and select from the *shortcut menu*. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means select an object, right-click, and click an option from the shortcut menu that appears.

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. This information can also be obtained by contacting an Ansys account manager.

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 are taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

From the Help menu, select **Help** and choose from the following:

- **[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** to open the selected help for the active product.

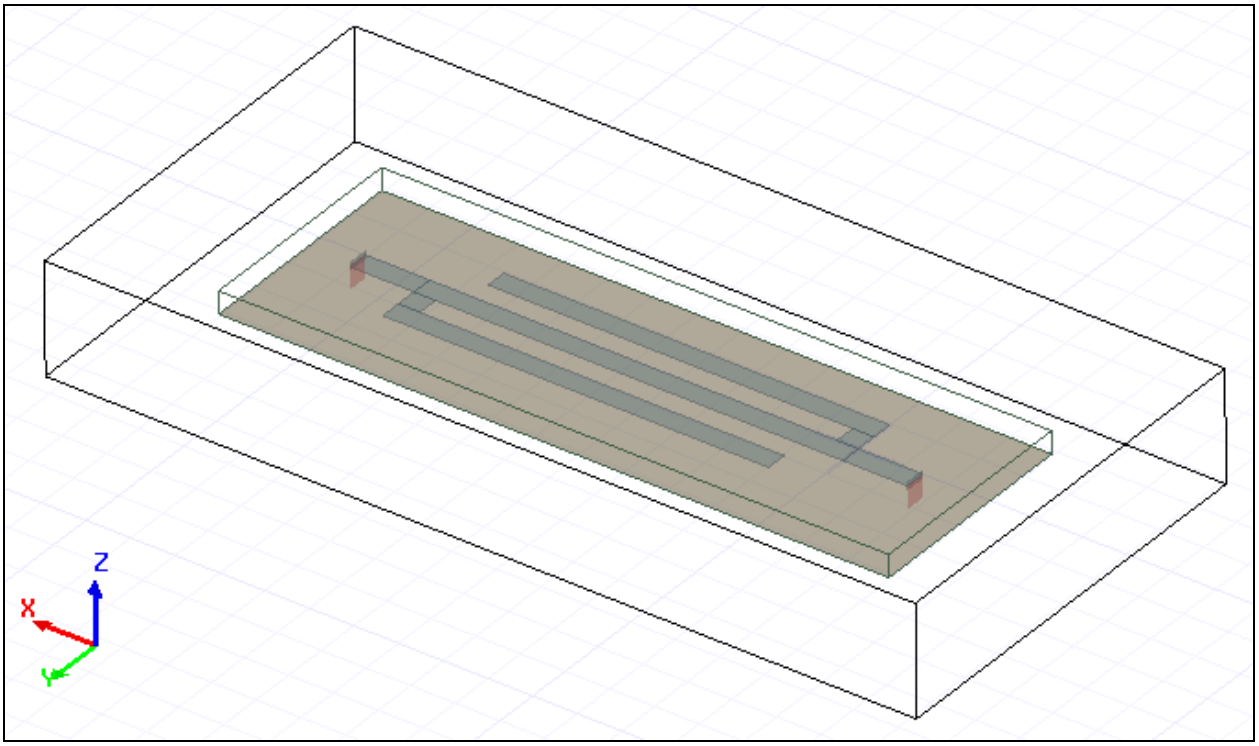
Press **F1** while the cursor is pointing at a menu command or while a particular window tab is open. In this case, the help page associated with the command or open window is displayed automatically.

Table of Contents

1 - Introduction

Complete the **Getting Started with HFSS 3D Layout: Microstrip Filter** guide to create, solve, and analyze a microstrip bandstop filter using the HFSS 3D Layout design type in the **Electronics Desktop** application. A bandstop filter (i.e., a band-rejection filter) is a device that attenuates signals within a target frequency band while passing higher and lower frequencies unaltered. It is the opposite of a bandpass filter.

> **Note:**
>
> The following screenshot was taken with **HFSS 3D Layout** > **HFSS Extents** enabled. The outer envelope represents the boundary of the default air box surrounding the model, which is the extent of the solution region meshed and solved when an HFSS analysis is performed.



> **Note:**
>
> In the *Getting Started with HFSS 3D Layout Low Pass Filter* guide, the ground plane layer of the stackup was defined as **Negative**, and any objects drawn on that layer represented areas of removed conductor. In this exercise, the ground layer is **notNegative**.

Draw the model using parametric design variables for the signal layer. Then evaluate and compare the filter response using an HFSS analysis.

Setting General Options

Before inserting an HFSS 3D Layout Design into the project, follow these steps to ensure the default unit of length measurement is set to **mil**.

1. From the **Desktop** ribbon tab, click **General Options**.
2. In the tree on the left side of the **Options** window, expand the **General** group and select the **Default Units** subgroup.
3. Select **mil** from the **Length** drop-down menu.
4. Click **OK** to close the **Options** window.

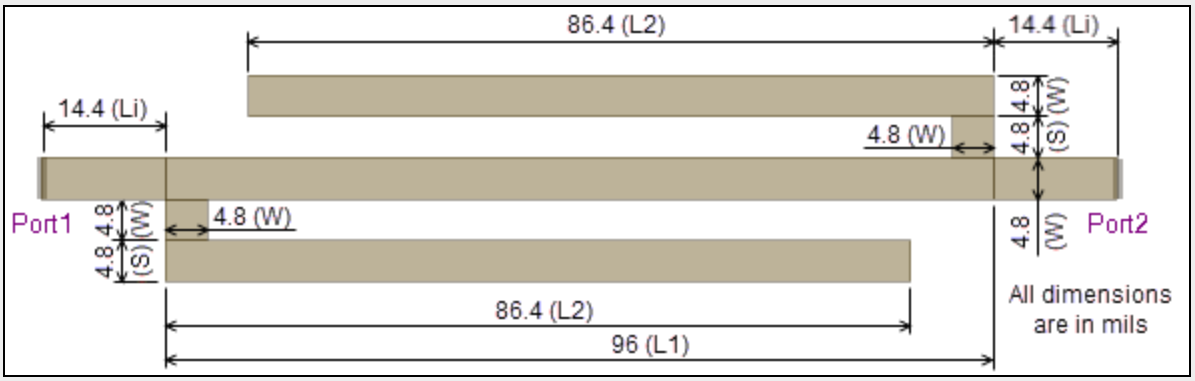
Create the Model

From here, continue to the following topics to define the stackup layers (i.e., topology) of the model, draw the geometry of the filter, and assign the excitation ports.

- Insert Layers
- Draw the Model Geometry
- Assign the Ports

Note:

The following figure is a diagram of the **Trace** (i.e., signal) layer of the microstrip bandstop filter model.



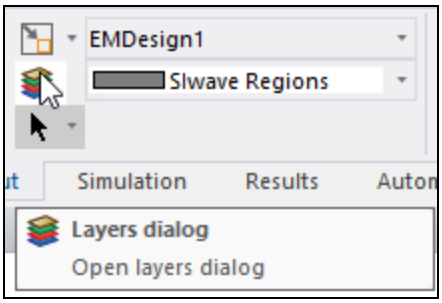
,

There is also a 150 x 50 mil rectangular ground plane conductor drawn on a separate layer.

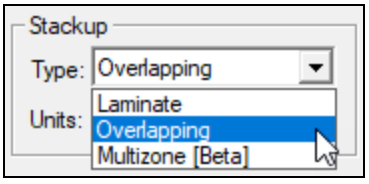
Inserting Layers

Follow these steps to insert layers in an HFSS 3D Layout design.

1. From the **Layout** tab, click **Layers dialog** to open the **Edit Layers** window.



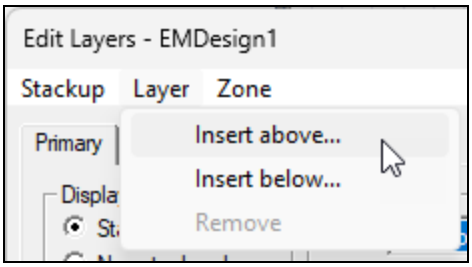
2. In the **Edit Layers** window > **Stackup** area, select **Overlapping** from the **Type** drop-down menu.



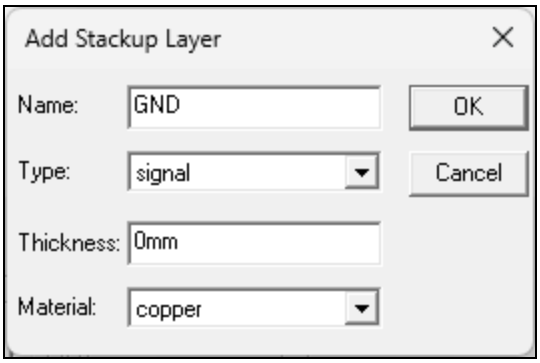
Add a Ground Layer to the Layer Table

> **Note:**
>
> When adding the first layer to the layers table, the actions of **Insert above** and **Insert below** are identical. Once there are one or more layers, select one, then choose the applicable option to add a new layer **above** or **below**.

1. Click **Layer** and select either **Insert above** or **Insert below** to open the **Add Stackup Layer** window.



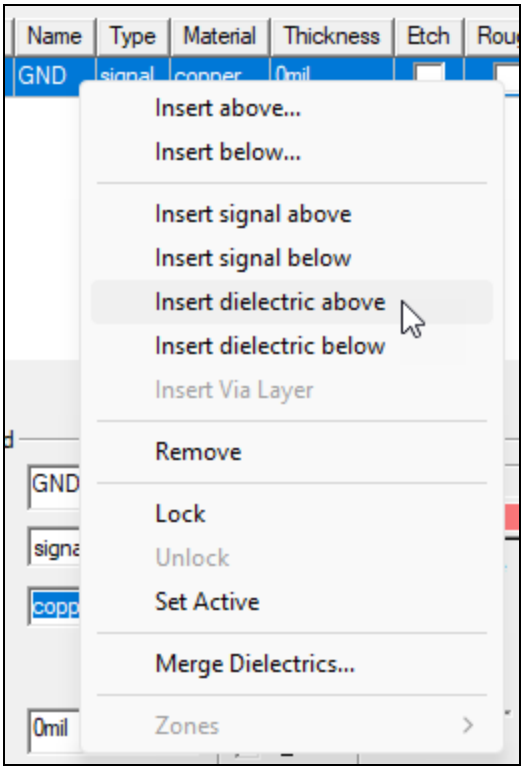
2. In the **Add Stackup Layer** window, do the following:
   a. Enter **GND** in the **Name** field.
   b. Click **OK** to close the **Add Stackup Layer** window add the new ground layer to the layer table.



> **Note:**
>
> The material *copper* is automatically assigned to signal layers.

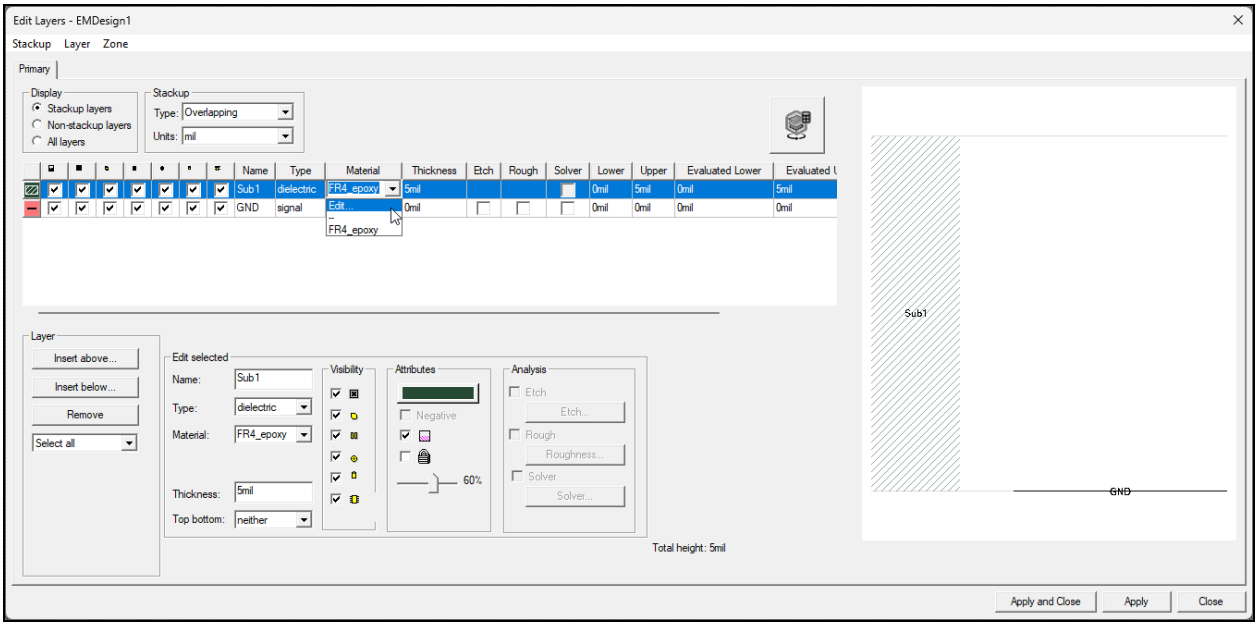
Add a Dielectric Layer to the Layer Table

1. Right-click anywhere in the **GND** layer and select **Insert dielectric above**. A new row appears in the layer table (default **Name**, **Dielectric**).

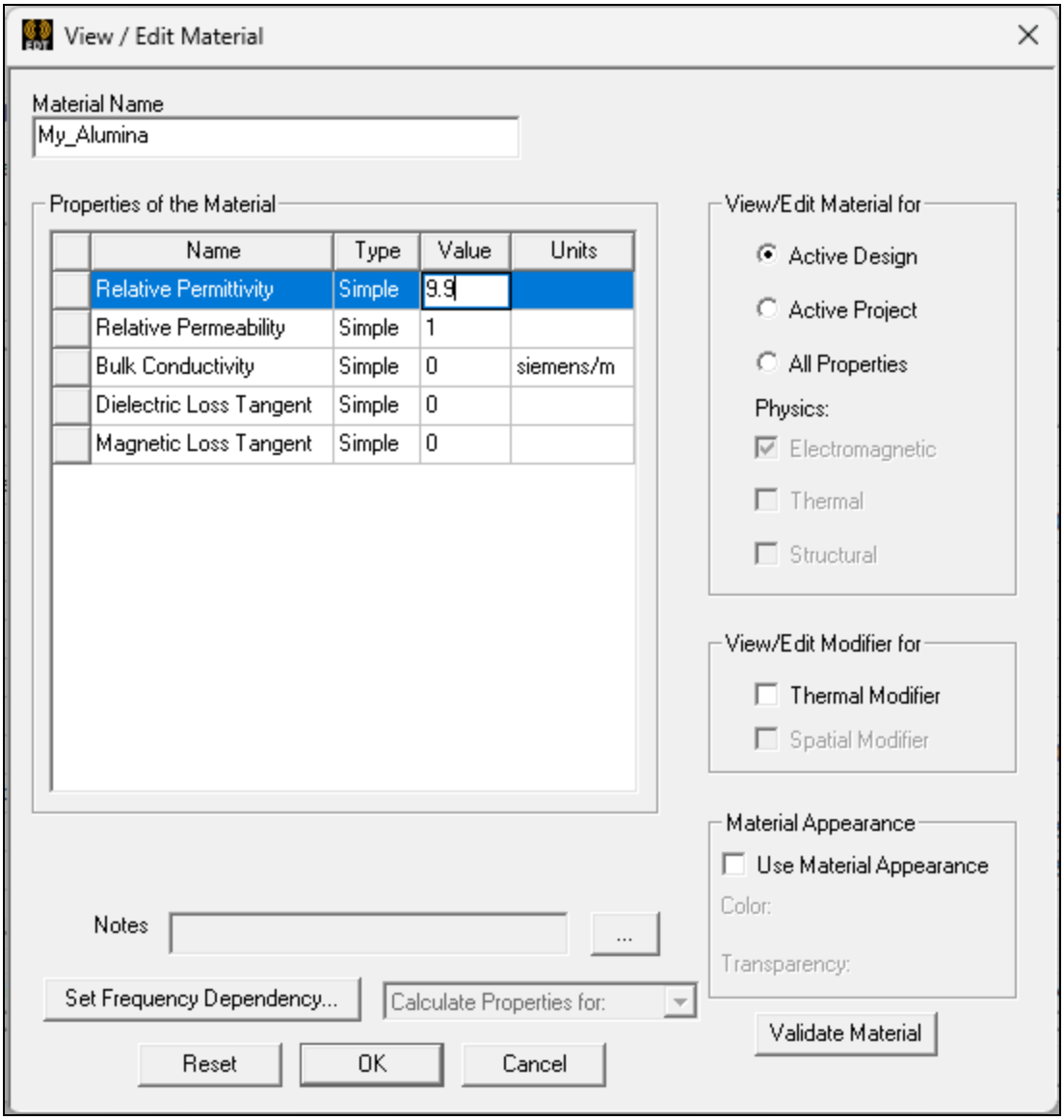


2. In the new **dielectric** row, do the following:
   a. In the **Name** field, replace **dielectric** with **Sub1**.
   b. Ensure **5mil** is entered in the **Thickness** field.
   c. Select **Edit** from the **Material** drop-down menu to open the **Select Definition**

window.



3. From the **Select Definition** window, do the following:
    a. Click **Add Material** to open the **View / Edit Material** window.
    b. In the **Material Name** field, replace **Material1** with **My_Alumina**.
    c. In the **Relative Permittivity Value** field, replace **1** with **9.9**.
    d. Click **OK** to save changes, close the **View / Edit Material** window, and return to the **Select Definition** window.

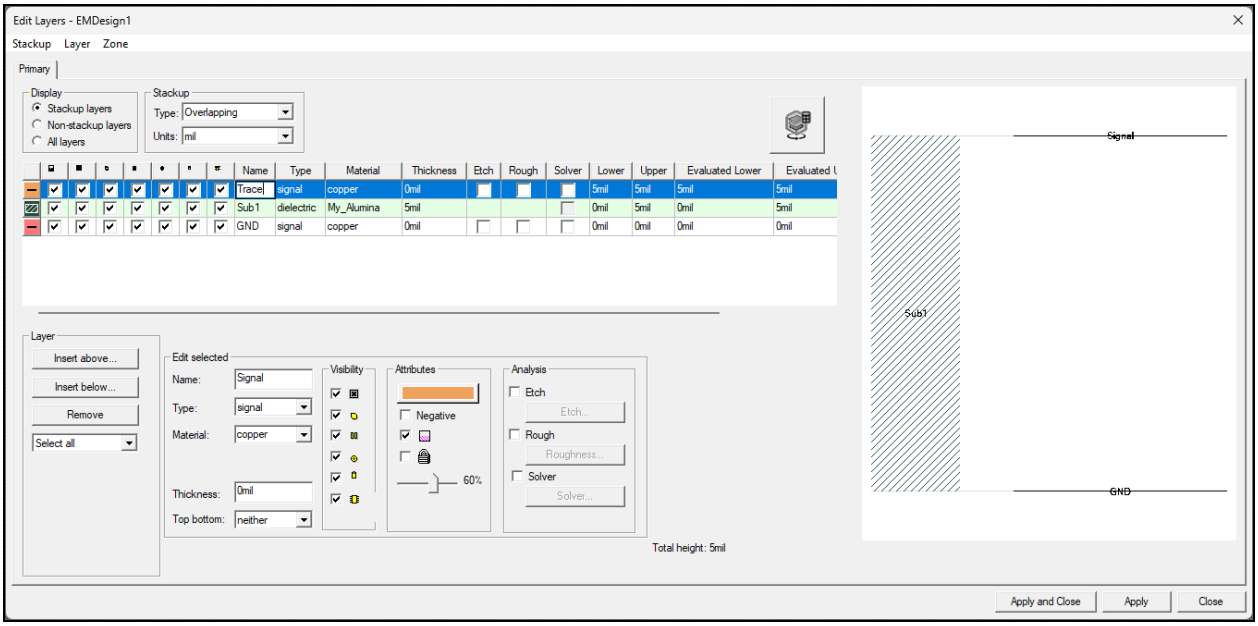


e. Click **OK** to close the **Select Definition** window.

Add a Trace (Signal) Layer to the Layer Table

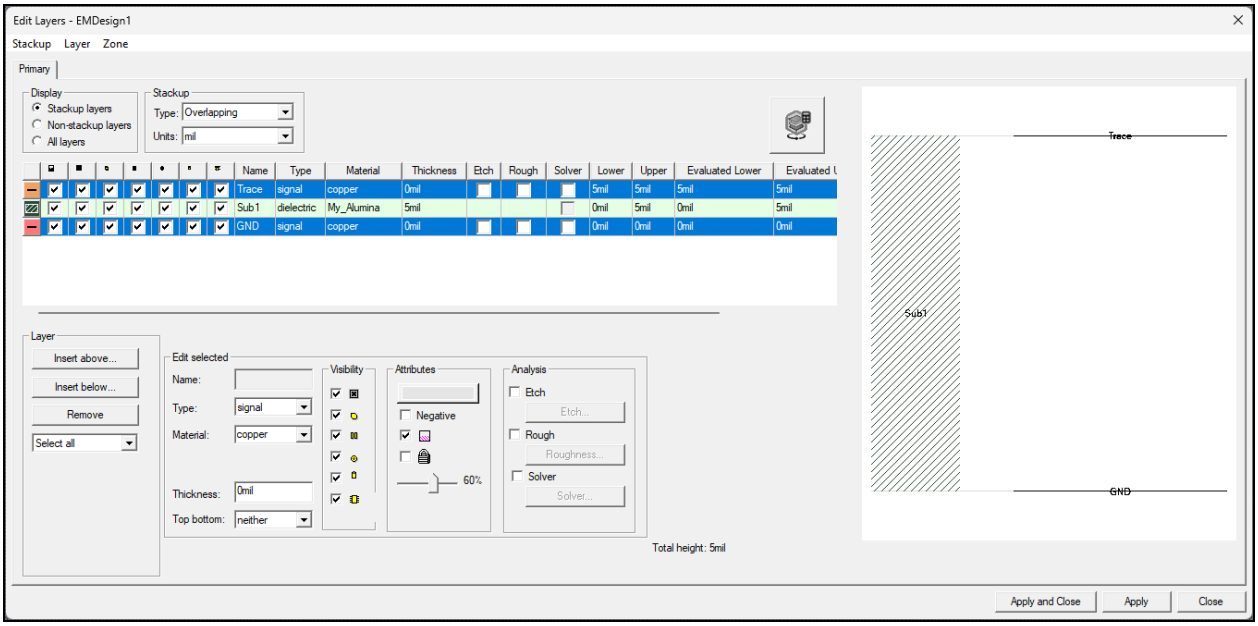
1. Right-click the **Sub1** layer and select **Insert signal above**. A new row appears in the table.

2. In the new **signal** row, replace **Signal** in the **Name** field with **Trace**.

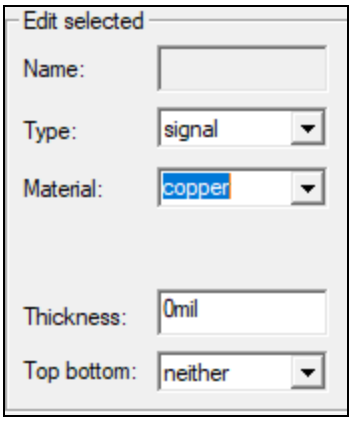


Make Changes to All Layers

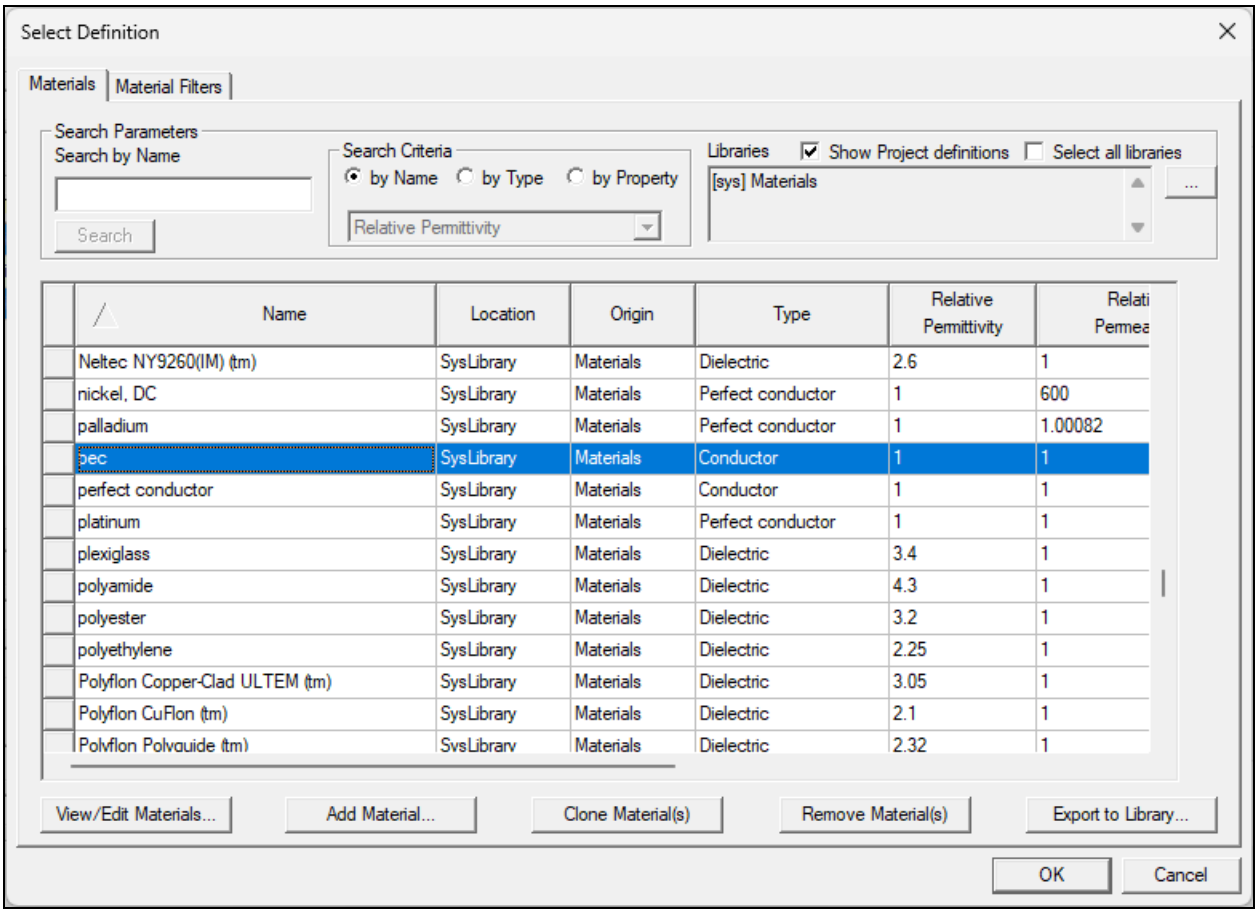
1. While the **Trace** layer is still selected, hold **Ctrl** and click the first field in the **GND** row. Both layers should be highlighted (i.e., selected).



2. From the **Edit selected** area, select **Edit** from the **Material** drop-down menu to open the **Select Definition** window.

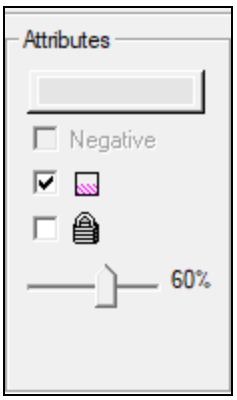


3. Select **pec** (perfect electrical conductor) from the list of library materials. Then click **OK** to close the **Select Definition** window.

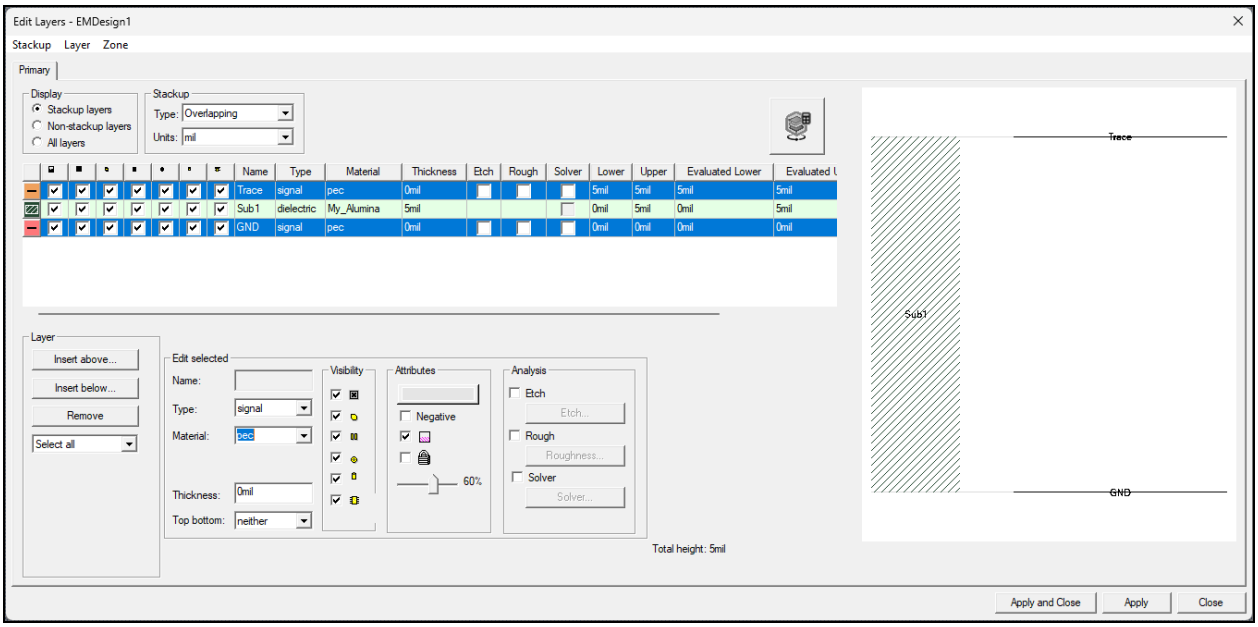


4. From the **Layer** area, re-select **Select All** from the drop-down menu to highlight all three rows.

5. Ensure the "shading" box in the **Attributes** area (i.e., the middle box) is checked. This ensures that all objects will be shaded, rather than only outlined (i.e., in wire frame).



6. The **Edit Layers** window should now match the following example.



> **Note:**
>
> If the stackup is not arranged in the correct hierarchy, rearrange the layers by **clicking+dragging** the selection handles in the left column. The **t1** layer should be at the top of the list, followed by the **d1** layer in the middle, and the **g1** layer at the bottom.

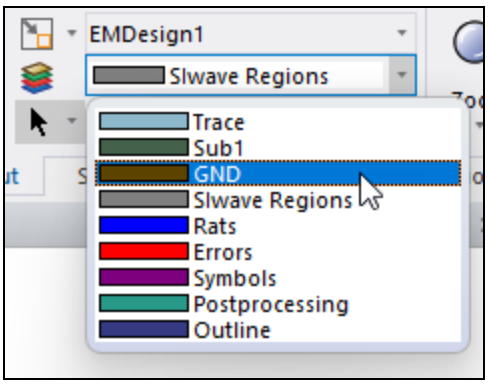
7. Click **Apply and Close** to apply the layer definitions and close the **Edit Layers** window.

Continue to Drawing the Ground Plane.

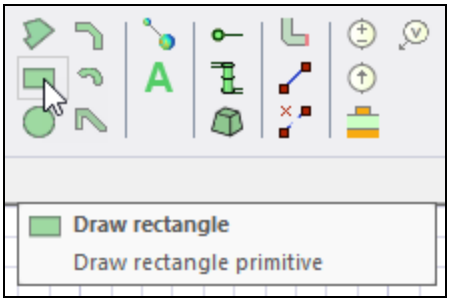
Drawing the Ground Plane

Follow these steps to draw a ground plane in the **Layout Editor**.

1. From the **Layout** tab, select **GND** from the **Active Layer** drop-down menu.



2. From the **Layout** tab, click **Draw rectangle**.

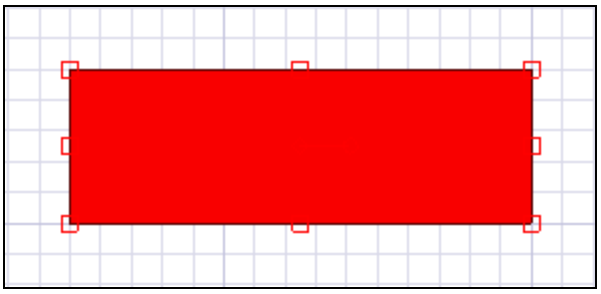


> **Note:**
>
> Coordinates entered via the coordinate fields specify opposite corners of a rectangle. Alternatively, entering the **Center**, **Width**, and **Height** of the proposed model in the **Properties** window determines the dimensions of the model dependent from the rectangle's centroid.

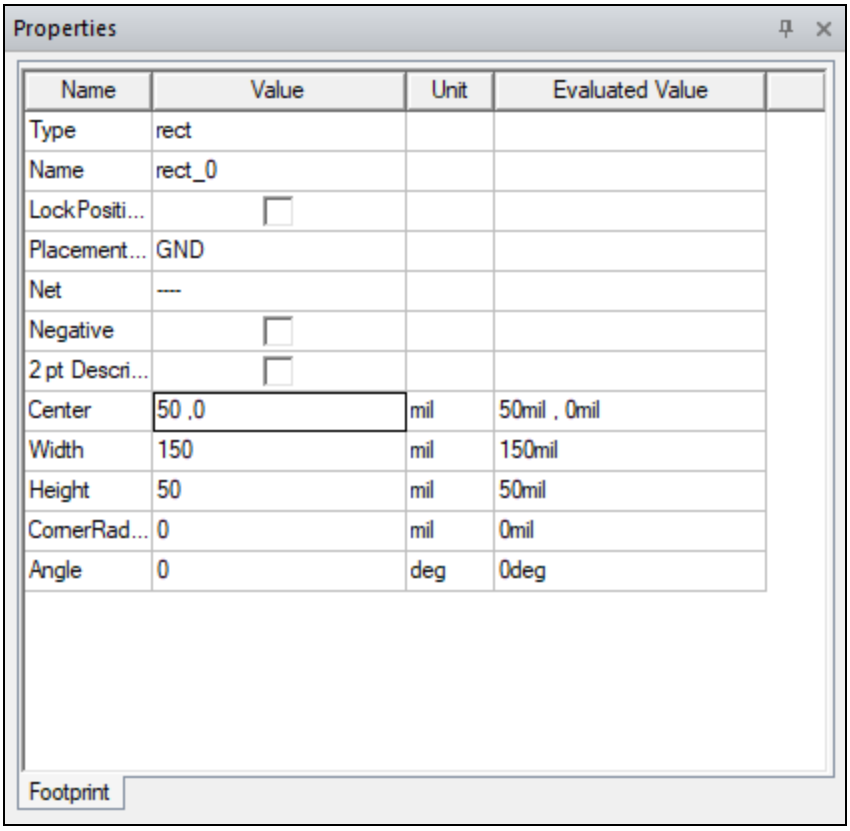
3. Do **not** **click+drag** in the **Layout Editor**. Instead, move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Click inside the field, delete the coordinates already present, and enter **50**.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.

5. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **150** in the field.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **50** in the field.

7. Press **Enter** to complete the ground plane.



8. With the new ground plane selected, ensure the **2 pt Description** option in the **Properties** window is **not** selected.
9. Enter **50 ,0** in the **Center** field and press **Enter** or **Tab** to relocate the ground plane, centered at **50, 0**.

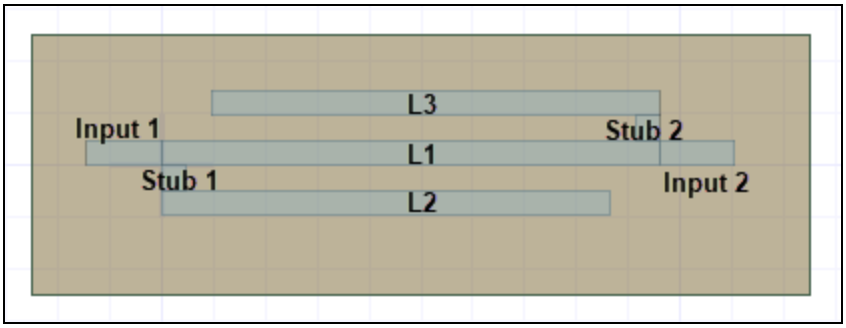


10. Press **Ctrl+D** to fit the drawing in the **Layout Editor** and clear the current selection.

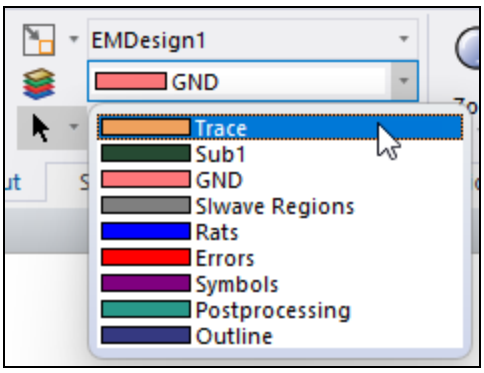
Continue to Creating L1.

Creating L1

Follow these steps to create the object L1 and then parameterize the object. When an object is parameterized, variables are defined for the coordinates and dimensions rather than absolute numeric values. In this way, define additional objects based on parameters of previously defined ones, to quickly alter the geometry of the model by editing the design parameters. All geometry directly or indirectly based on an altered parameter is automatically updated.

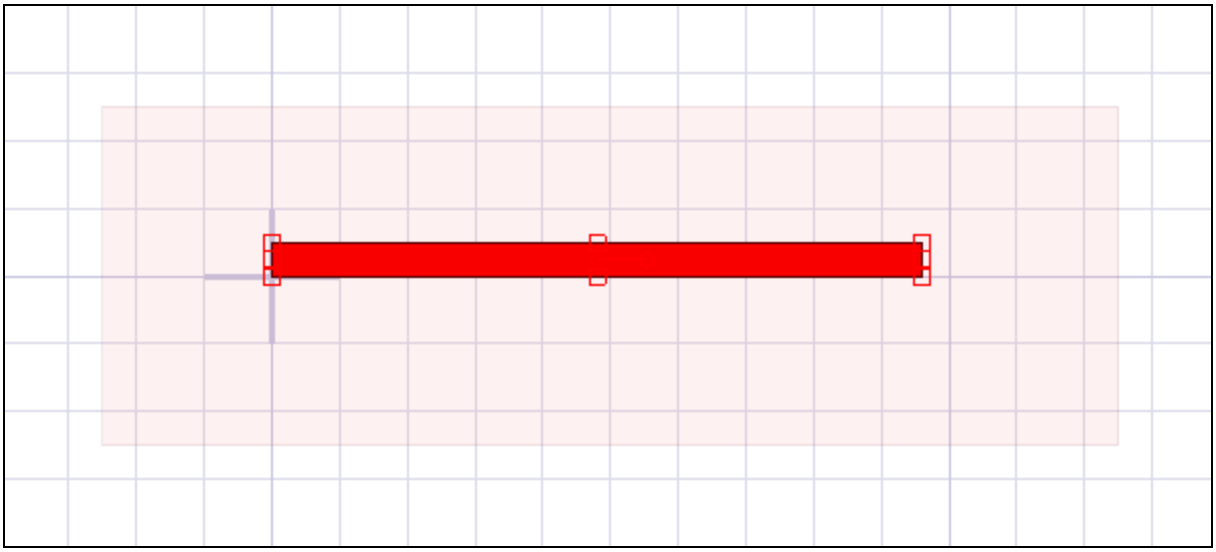


1. From the **Layout** tab, select **Trace** from the **Active Layer** drop-down menu.

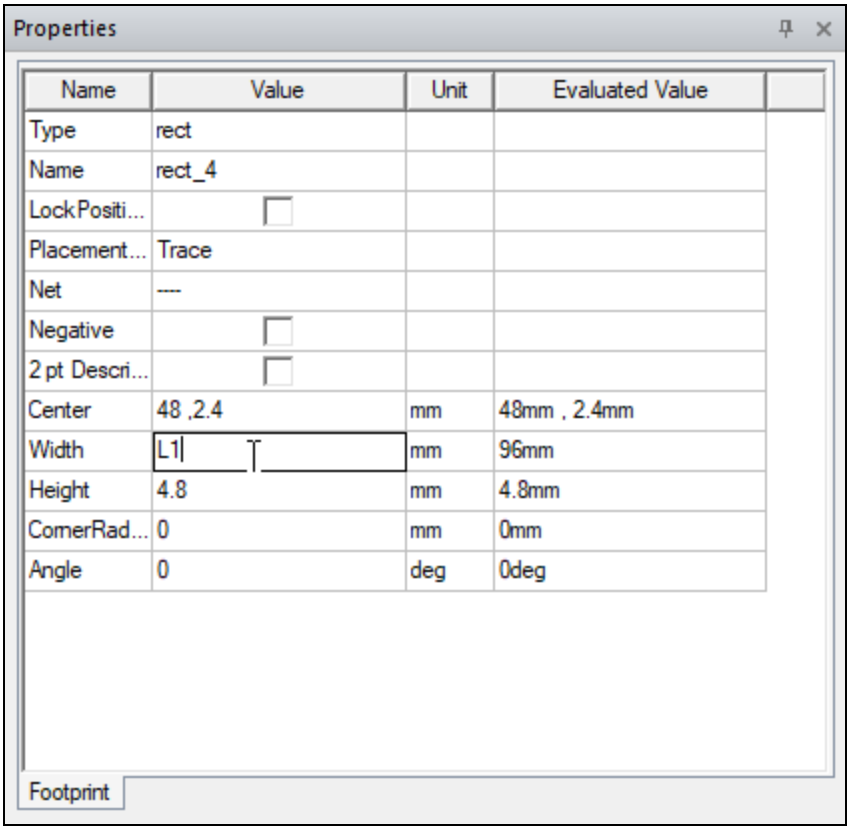


2. From the **Layout** tab, click **Draw rectangle**.
3. Do **not** **click+drag** in the **Layout Editor**. Instead, move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Click inside the field, delete the coordinates already present, and enter **0**.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.
5. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **96** in the field.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field.

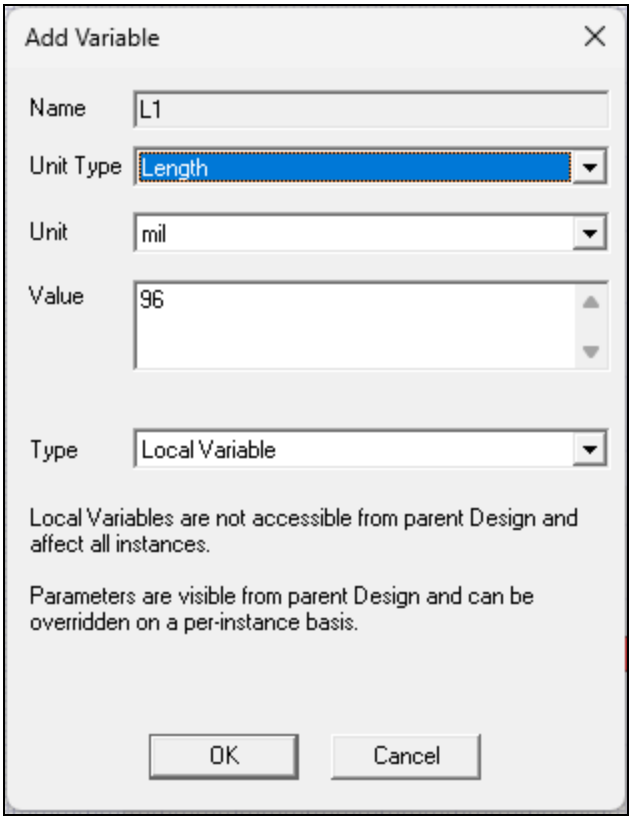
7. Press **Enter** to complete **L1**.



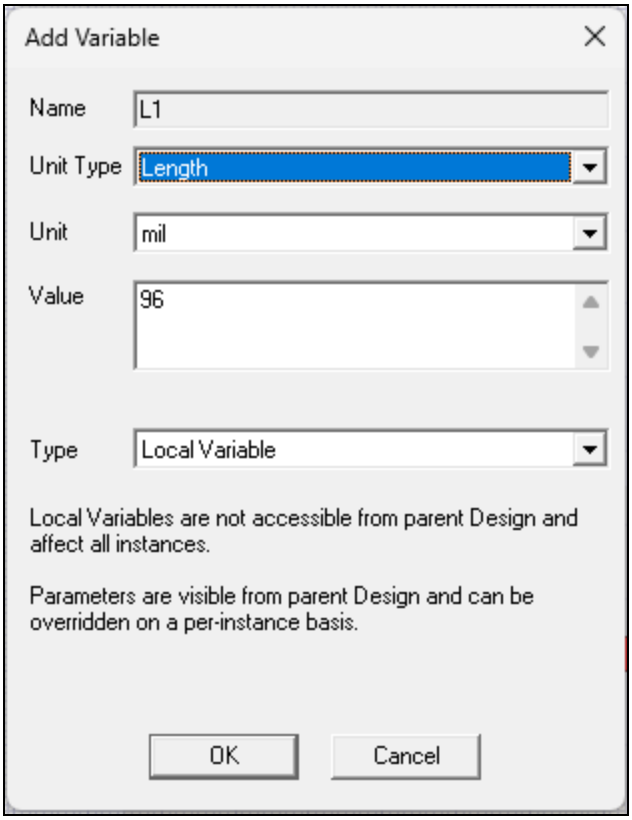
8. Select **L1** to populate the **Properties** window. Then make the following changes:
    a. Ensure the **2 pt Description** option is **not** selected.
    b. Enter **L1** in the **Width** field. Then press **Enter** or **Tab** to open the **Add Variable** window.



c. Ensure **mil** is selected from the **Unit** drop-down menu and **96** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

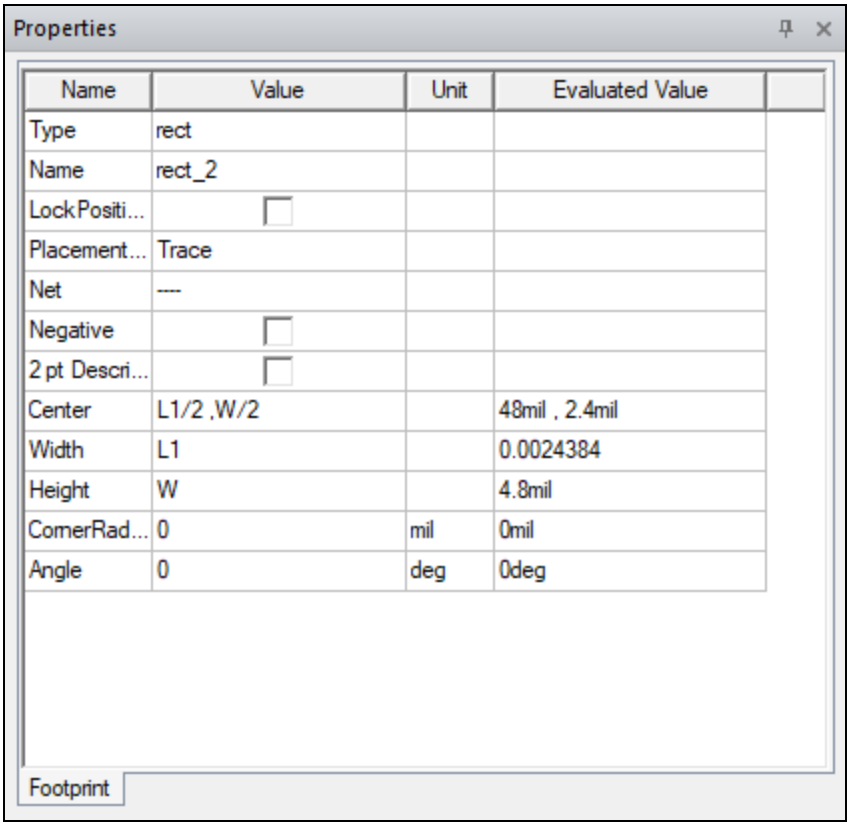


d. Enter **W** in the **Height** field. Then press **Enter** or **Tab** to open the **Add Variable** window.



e. Ensure **mil** is selected from the **Unit** drop-down menu and **4.8** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

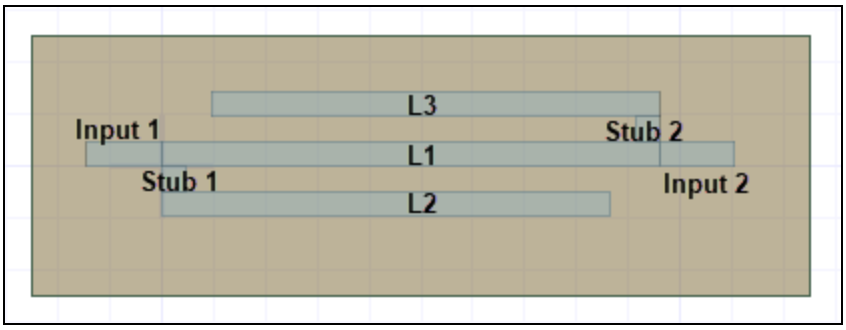
f. Enter **L1/2, W/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates



Continue to Creating Stub 1.

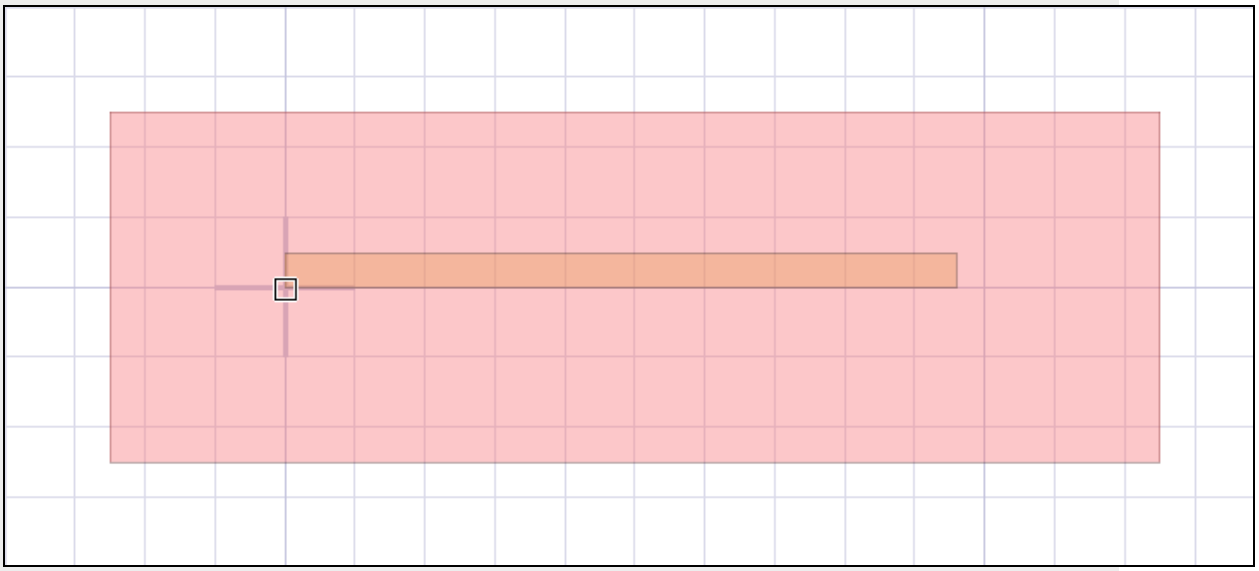
Creating Stub 1

Follow these steps to create the object Stub 1 and then parameterize the object.



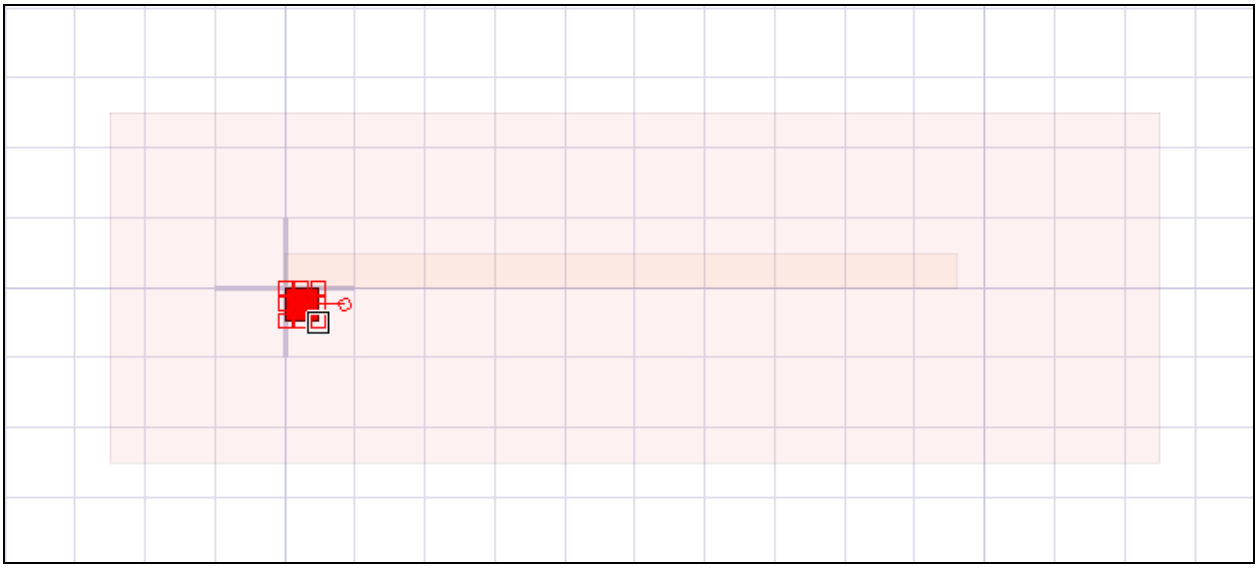
1. From the **Layout** tab, click **Draw rectangle**.
2. Click the lower-left corner of **L1** (i.e., **0, 0**).

> **Note:**
>
> The cursor changes from a rectangle to a square when a snapping point is detected.



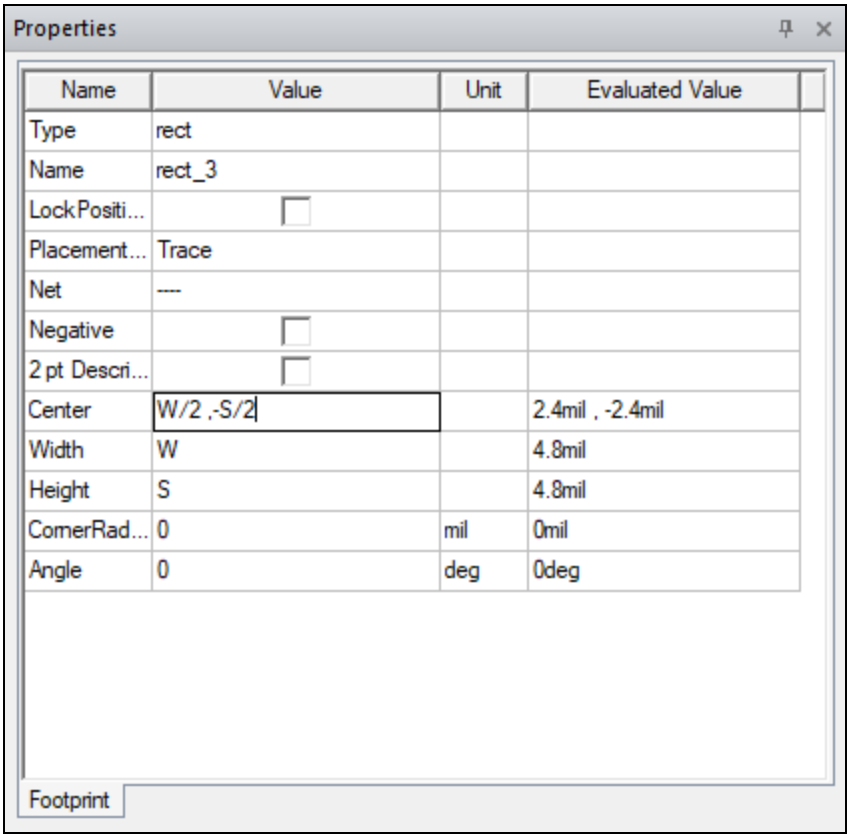
3. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **4.8** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **-4.8** in the field.

5. Press **Enter** to complete **Stub 1**.



6. Select **Stub 1** to populate the **Properties** window. Then make the following changes:
   a. Ensure the **2 pt Description** option is **not** selected.
   b. Enter **W** in the **Width** field. Then press **Enter** or **Tab**.
   c. Enter **S** in the **Height** field. Then press **Enter** or **Tab** to open the **Add Variable** window.
   d. Ensure **mil** is selected from the **Unit** drop-down menu and **4.8** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.
   e. Enter **W/2, -S/2** in the **Center** field and press **Enter** or **Tab** to accept the

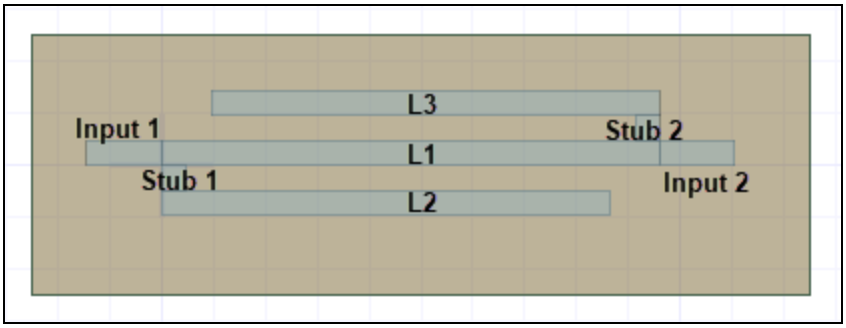
coordinates.



Continue to Creating L2.

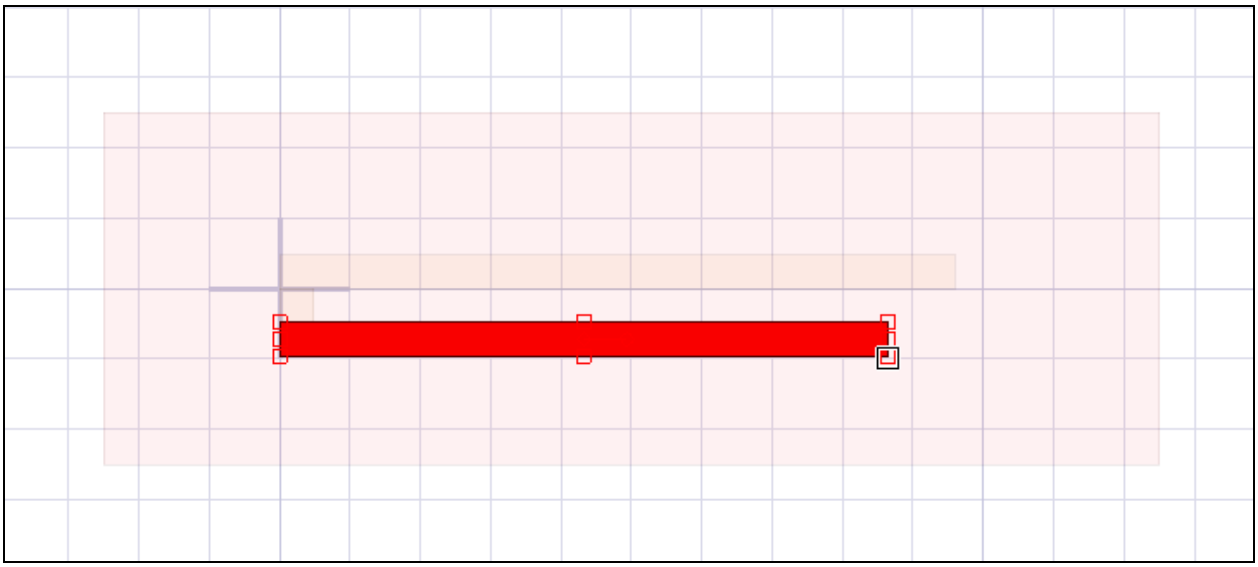
Creating L2

Follow these steps to create the object L2 and then parameterize the object.



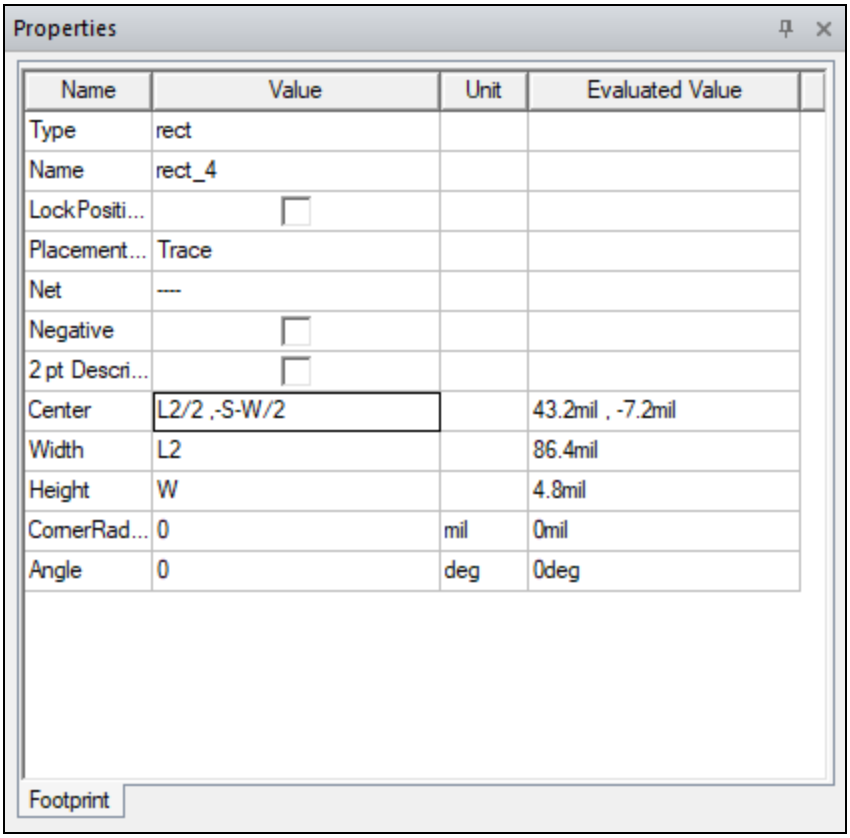
1. From the **Layout** tab, click **Draw rectangle**.
2. Click the lower-left corner of **Stub 1** (i.e., **0, -4.8**).

3. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **86.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **-4.8** in the field.
5. Press **Enter** to complete **L2**.



6. Select **L2** to populate the **Properties** window. Then make the following changes:
   a. Ensure the **2 pt Description** option is **not** selected.
   b. Enter **L2** in the **Width** field. Then press **Enter** or **Tab** to open the **Add Variable** window.
   c. Ensure **mil** is selected from the **Unit** drop-down menu and **86.4** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.
   d. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.
   e. Enter **L2/2, -S-W/2** in the **Center** field and press **Enter** or **Tab** to accept the

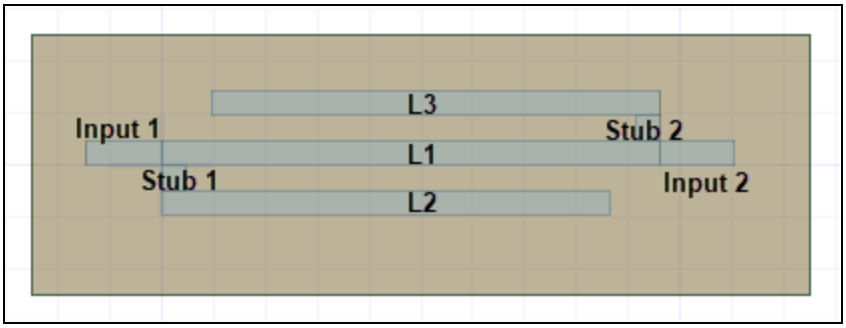
coordinates.



Continue to Creating Stub 2.

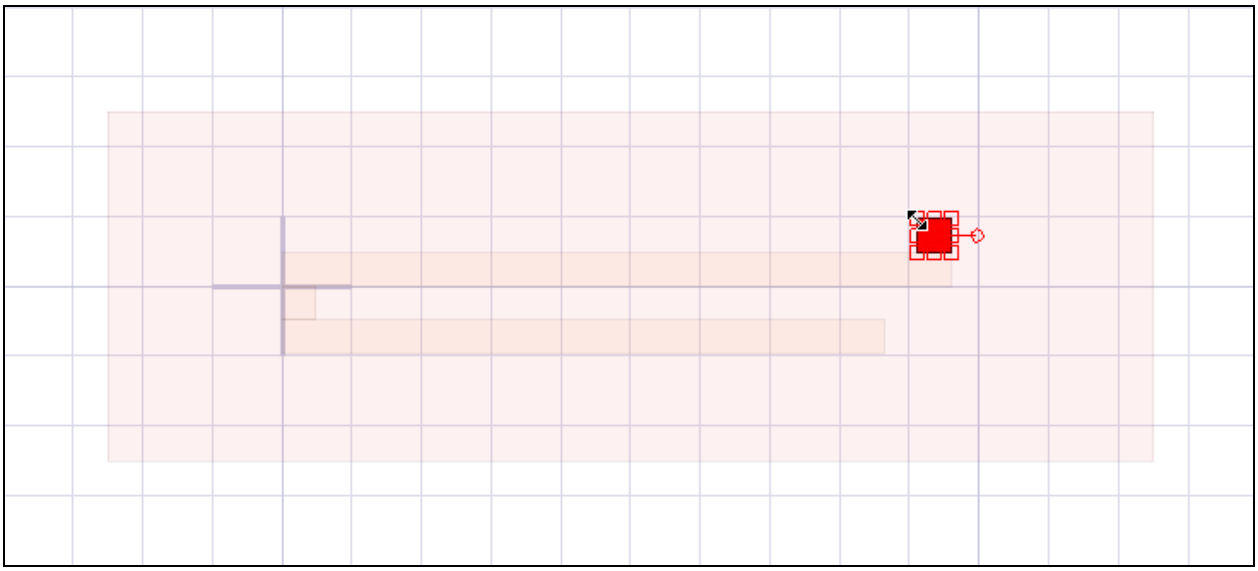
Creating Stub 2

Follow these steps to create the object Stub 2 and then parameterize the object.



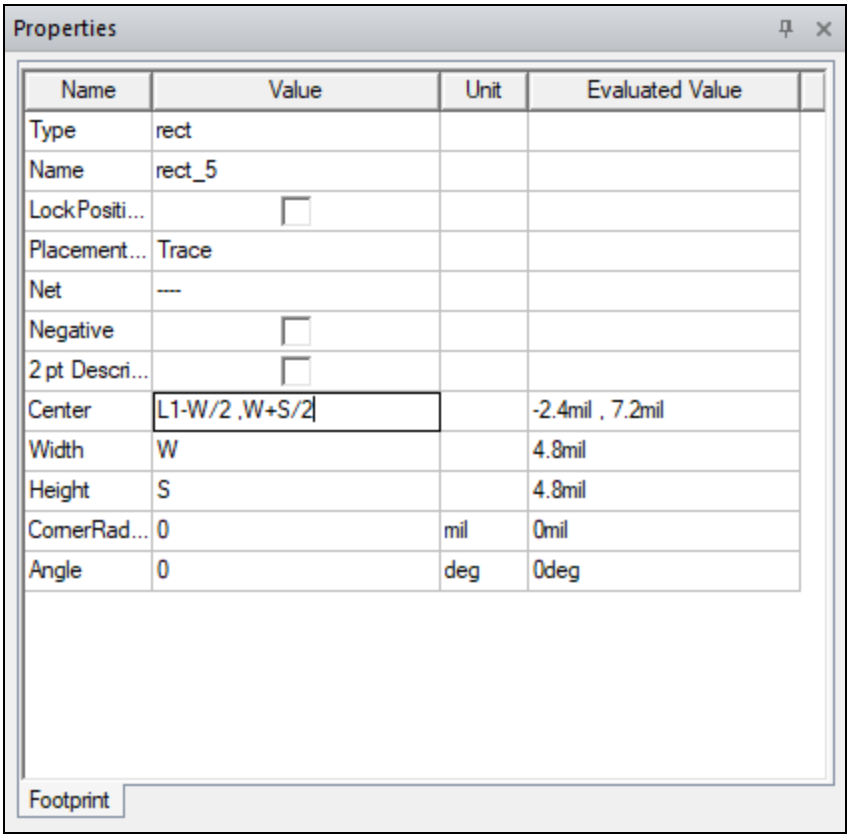
1. From the **Layout** tab, click **Draw rectangle**.
2. Click the upper-right corner of **L1** (i.e., **96, 4.8**).

3. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **-4.8** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field.
5. Press **Enter** to complete **Stub 2**.



6. Select **Stub 2** to populate the **Properties** window. Then make the following changes:
   a. Ensure the **2 pt Description** option is **not** selected.
   b. Enter **W** in the **Width** field. Then press **Enter** or **Tab**.
   c. Enter **S** in the **Height** field. Then press **Enter** or **Tab**.
   d. Enter **L1-W/2 , W+S/2** in the **Center** field and press **Enter** or **Tab** to accept the

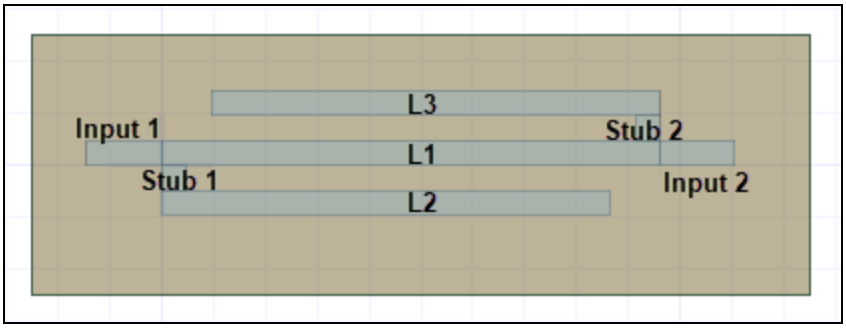
coordinates.



Continue to Creating L3.

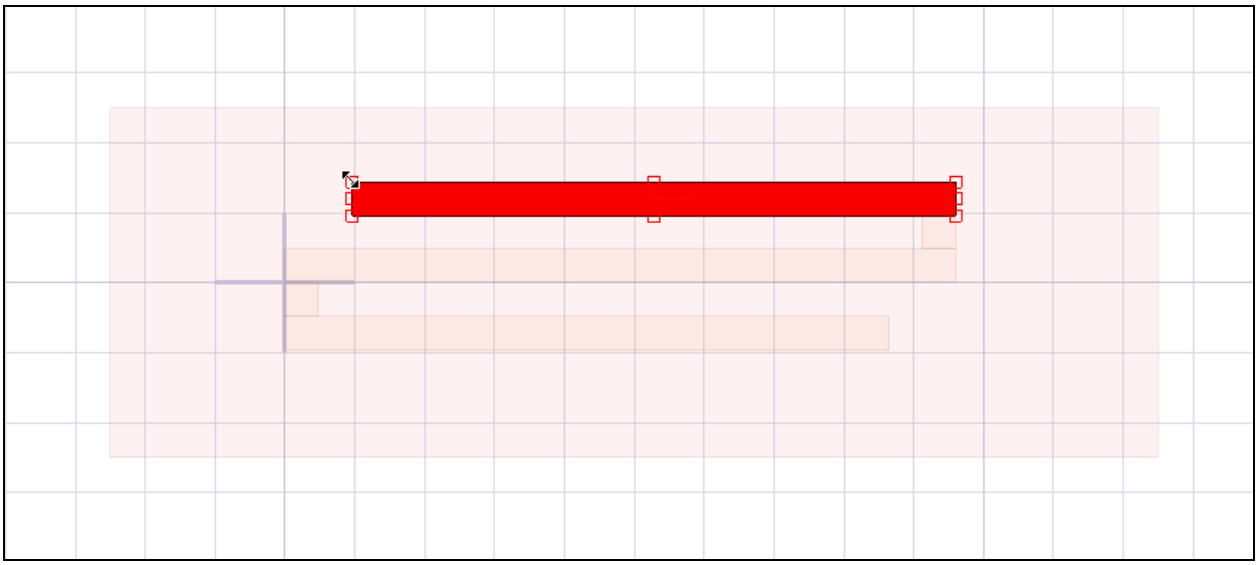
Creating L3

Follow these steps to create the object L3 and then parameterize the object.



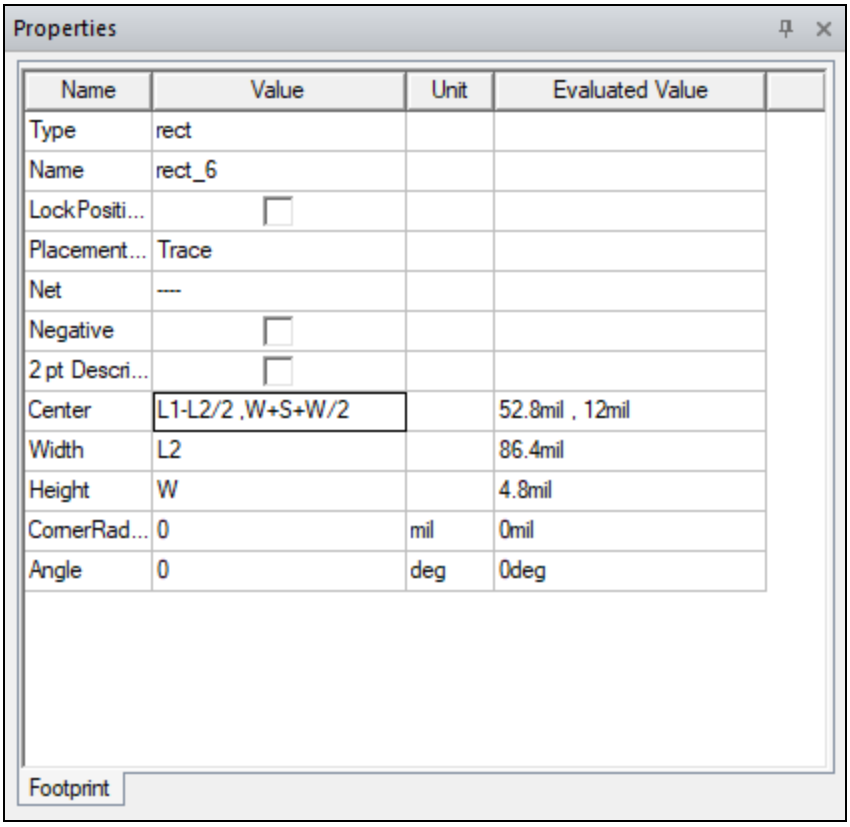
1. From the **Layout** tab, click **Draw rectangle**.
2. Click the upper-right corner of **Stub 2** (i.e., **96, 9.6**).

3. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **-86.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field.
5. Press **Enter** to complete **L3**.



6. Select **L3** to populate the **Properties** window. Then make the following changes:
    a. Ensure the **2 pt Description** option is **not** selected.
    b. Enter **L2** in the **Width** field. Then press **Enter** or **Tab**.
    c. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.
    d. Enter **L1-L2/2, W+S+W/2** in the **Center** field and press **Enter** or **Tab** to accept the

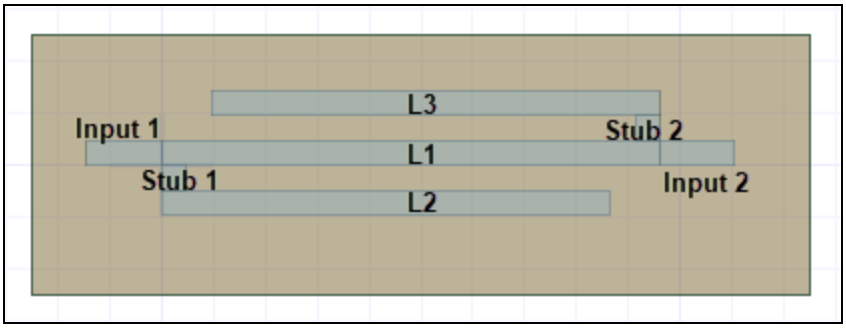
coordinates.



Continue to Creating Input 1.

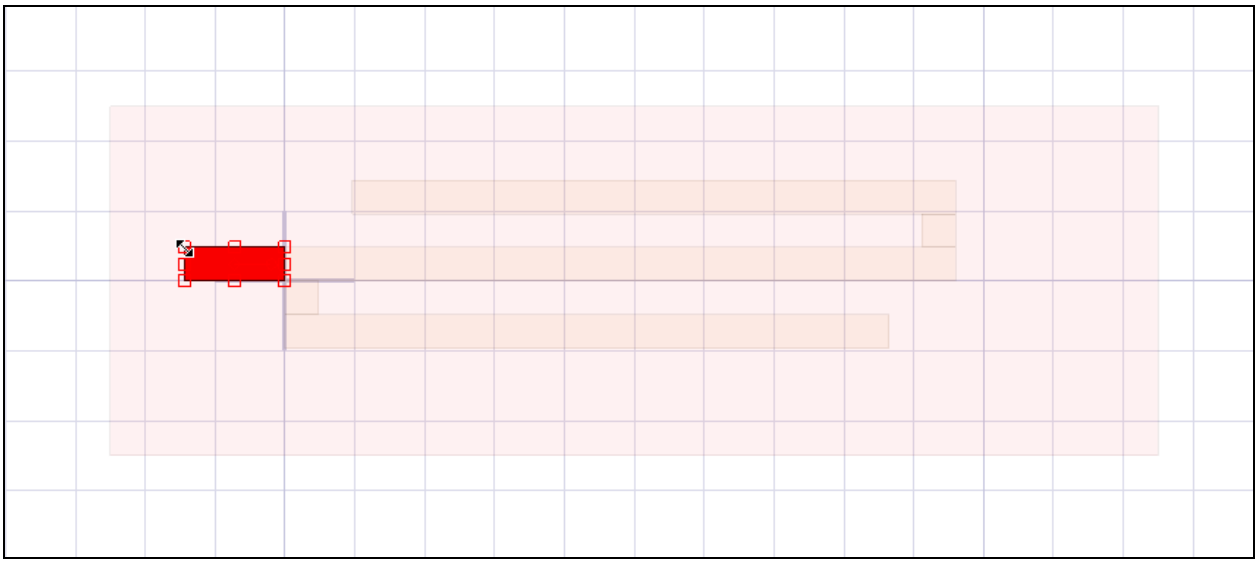
Creating Input 1

Follow these steps to create the object Input 1 and then parameterize the object.



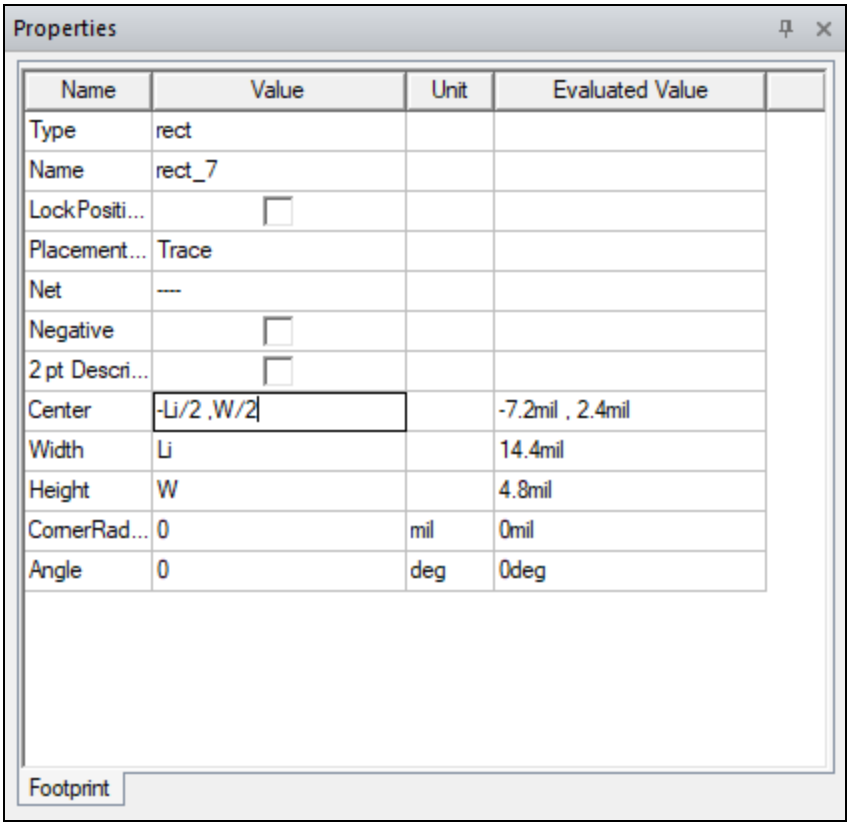
1. From the **Layout** tab, click **Draw rectangle**.
2. Click the lower-left corner of **L1** (i.e., **0, 0**).

3. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **-14.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field.
5. Press **Enter** to complete **Input 1**.



6. Select **Input 1** to populate the **Properties** window. Then make the following changes:
    a. Ensure the **2 pt Description** option is **not** selected.
    b. Enter **L2** in the **Width** field. Then press **Enter**or **Tab** to open the **Add Variable** window.
    c. Ensure **mil** is selected from the **Unit** drop-down menu and **14.48** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.
    d. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.
    e. Enter **-Li/2, W/2** in the **Center** field and press **Enter** or **Tab** to accept the

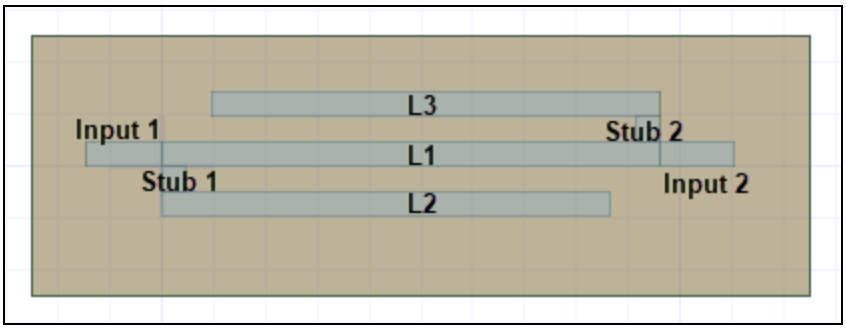
coordinates.



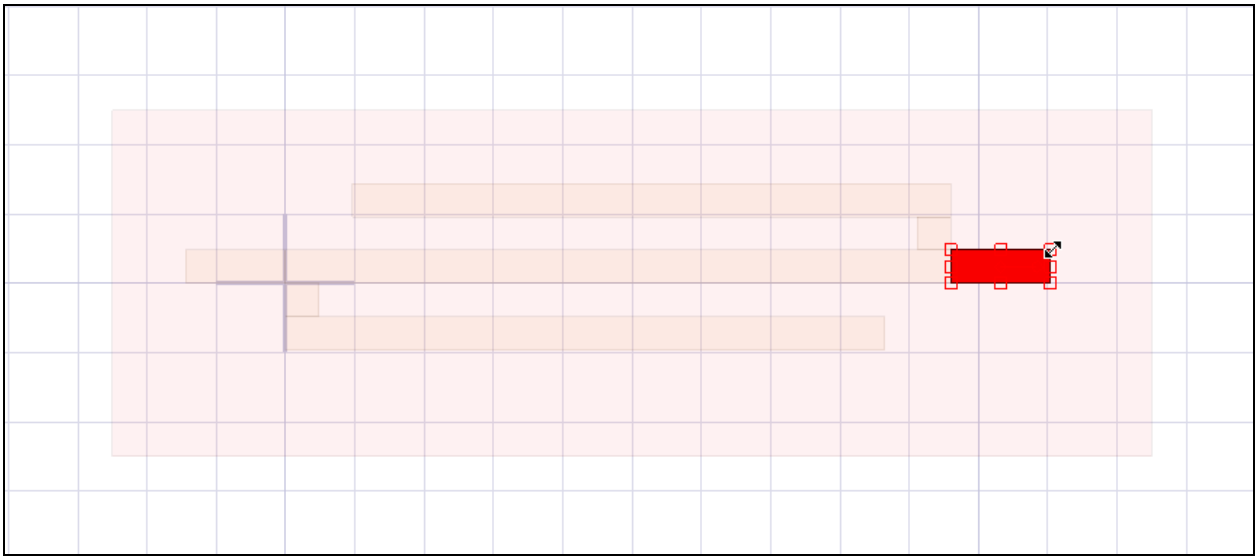
Continue to Creating Input 2.

Creating Input 2

Follow these steps to create the object Input 2 and then parameterize the object.

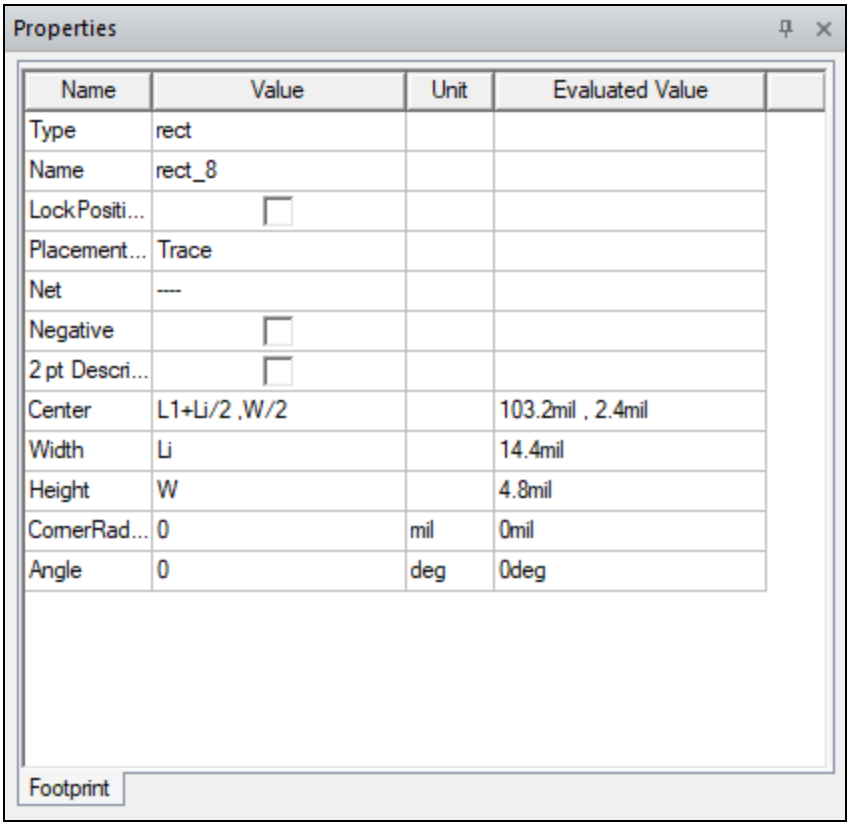


1. From the **Layout** tab, click **Draw rectangle**.
2. Click the lower-right corner of **L1** (i.e., **96, 0**).
3. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **14.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field.
5. Press **Enter** to complete **Input 2**.



6. Select **Input 2** to populate the **Properties** window. Then make the following changes:
   a. Ensure the **2 pt Description** option is **not** selected.
   b. Enter **Li** in the **Width** field. Then press **Enter**or **Tab**.
   c. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.
   d. Enter **L1+Li/2, W/2** in the **Center** field and press **Enter** or **Tab** to accept the

coordinates.



Continue to Creating Edge Ports.

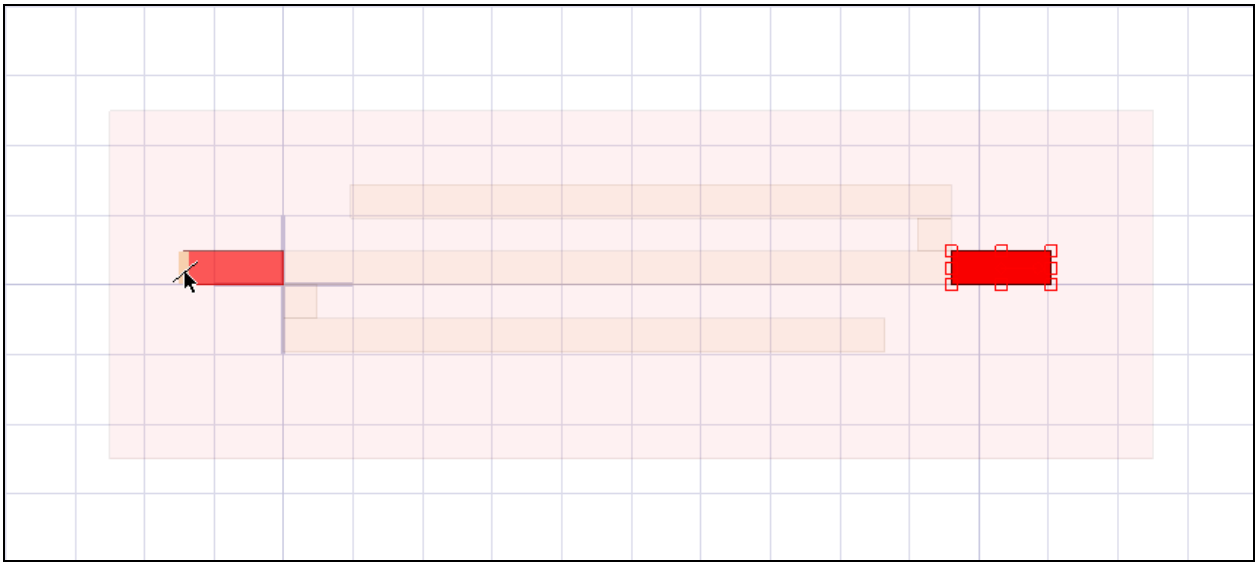
Creating Edge Ports

Follow these steps to add two edge ports to the model.

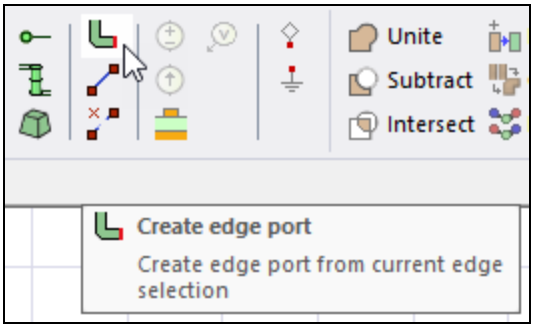
1. To create the first port (i.e., ***Port1***), first press **E** to immediately enter **Select Edges** mode.

> **Note:** Once **Select Edges** is selected, the cursor changes: a diagonal line crosses the tip of the arrow.
>
> 

2. Click the left edge of **Input 1** to select it.

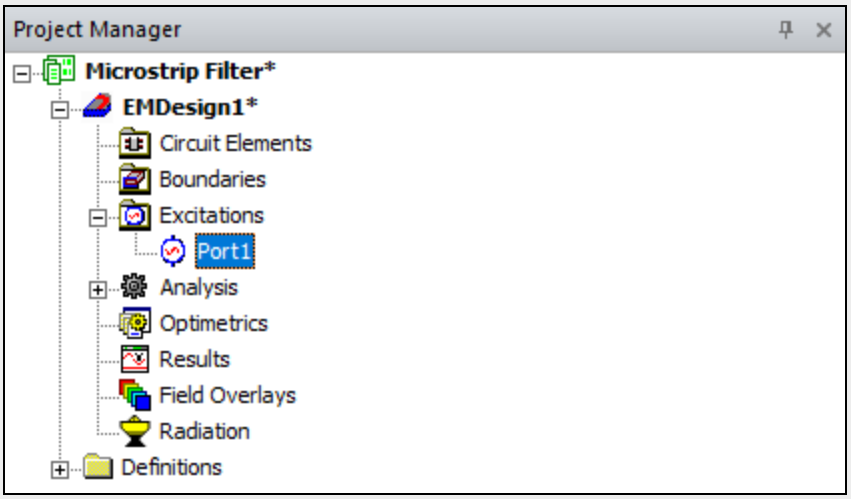


3. From the **Layout** tab, select **Create edge port**.

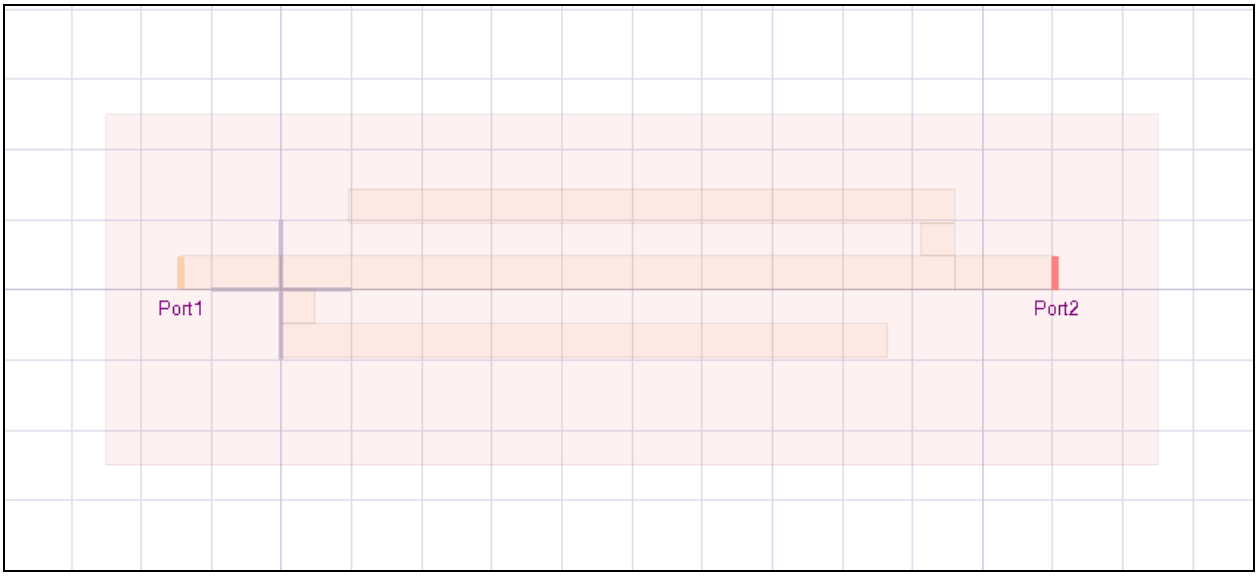


Note:

Once a port is created, it appears in the **Project Manager** > **Project Tree** > [***active design folder***] > **Excitations** folder.



4. Repeat steps 1-3 to create the second port (i.e., ***Port2***) on the right edge of **Input 2**.



Note:

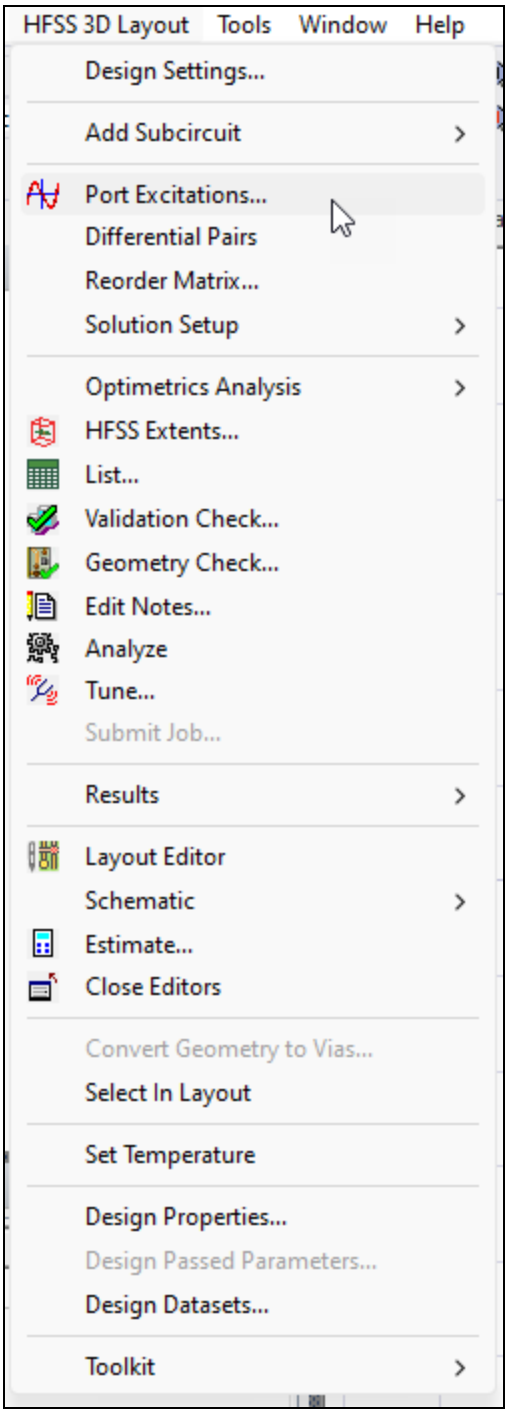
In the **Low Pass Filter Getting Started Guide**, the individual objects comprising the trace layer were united into a single object. However, because this model is parametric, do **not** unite the objects. Unlike the 3D Modeler used for conventional HFSS designs, the **Layout Editor** does not maintain the full parametric history of the model construction. If the trace objects are united, the parameters defined for the individual rectangles no longer have any effect on the geometry. Once the individual rectangles are merged into a complex polygon, the vertices of that polygon are defined using their absolute numerical coordinates. Therefore, altering the design variables no longer has an effect on the model because the variables are no longer used to define the united object. All of the edges of the conducting objects in this exercise are coincident with their adjacent objects' edges. Whether united or not, the trace objects behave like a single conducting part and the solver treats the objects as a contiguous conducting part.

Continue to Setting Port Excitations.

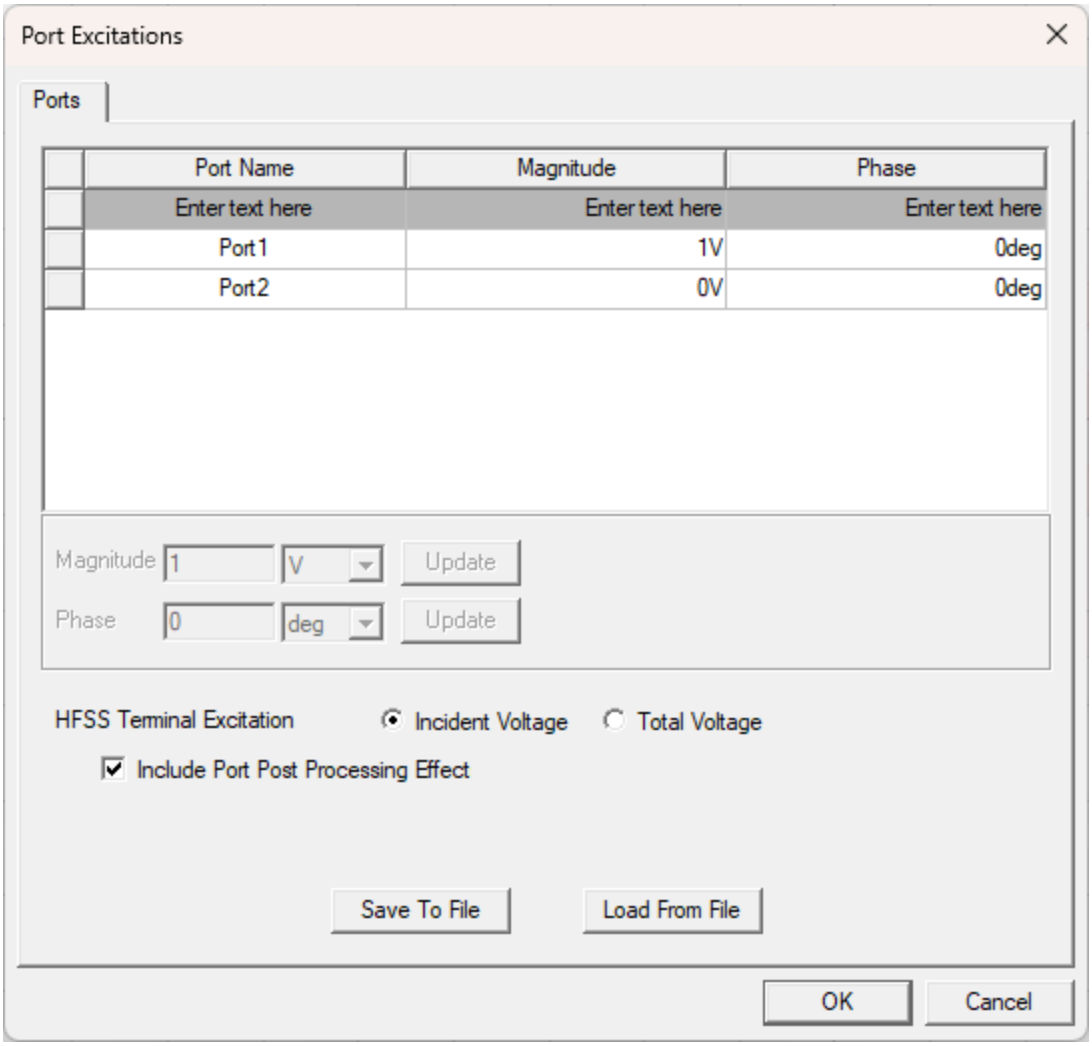
Setting Port Excitations

In order to create meaningful results with the microstrip filter design, one edge port should act as the filter input and the other as the filter output. By default, a **1** volt excitation at **0** degrees phase is applied to each edge port (i.e., **Port1** and **Port2**). Follow these steps to modify the port excitations to specify **0** volts at **Port2**, allowing it to act as the filter output.

1. From **HFSS 3D Layout**, select **Port Excitations** to open the **Port Excitations** window.



2. From the **Port Excitations** window, enter **0V** in the **Port2 Magnitude** field.



3. Click **OK** to close the **Port Excitations** window and return to the **Layout Editor**.
4. **Save** the project, either by navigating to **File** > **Save** or clicking the **Save** button on any of the ribbons.

Continue to Creating an HFSS Analysis Setup.

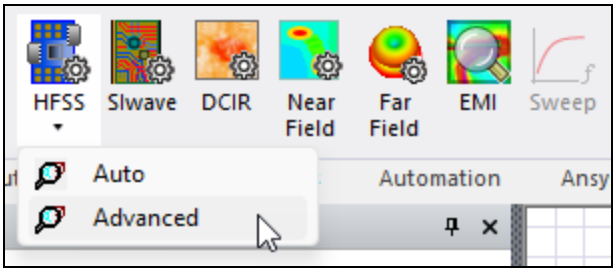
2 - Analysis and Post-Processing

This chapter contains the following topics:

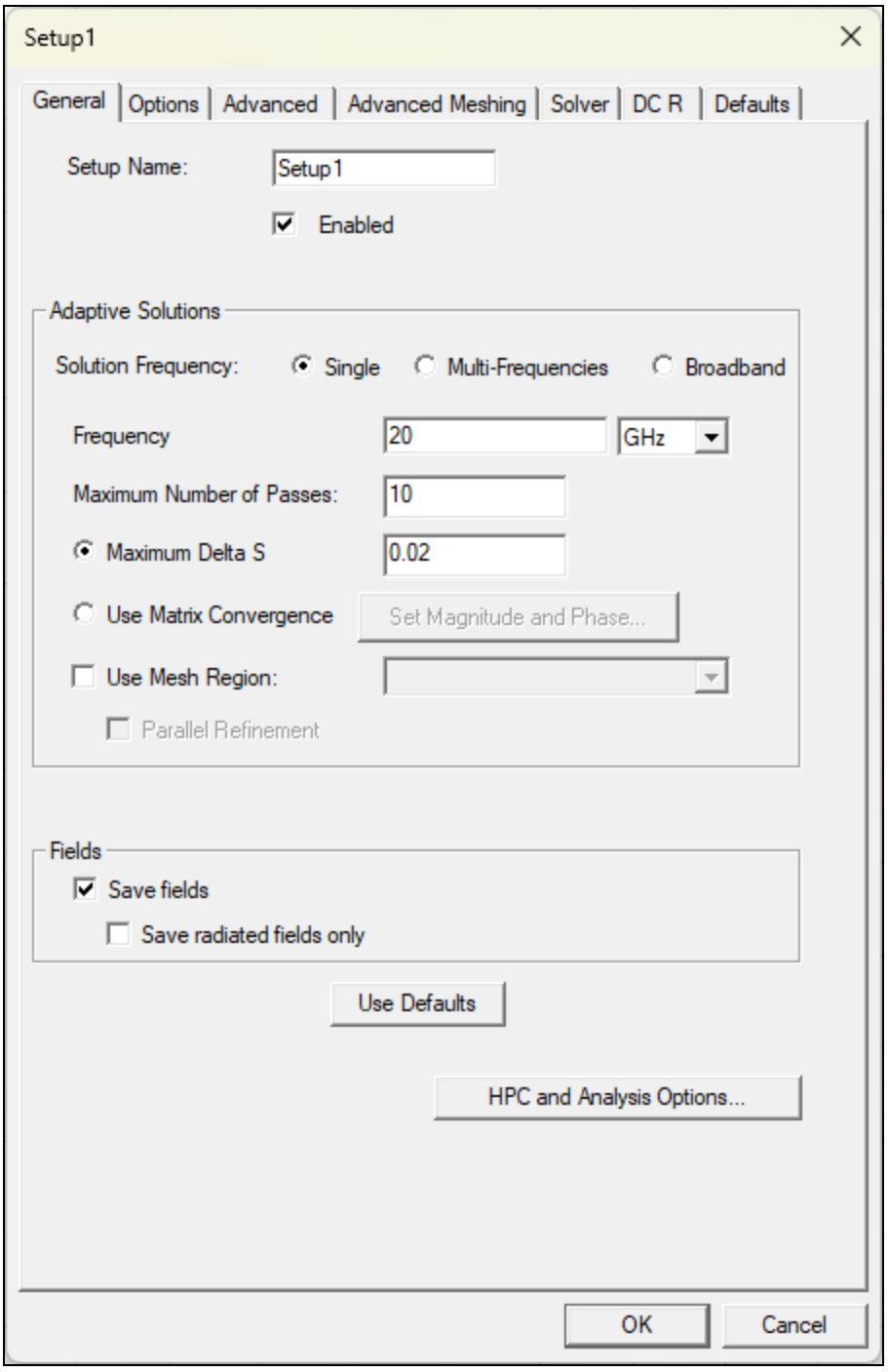
Creating an HFSS Analysis Setup

Solution setups are listed in the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Analysis**). Follow these steps to add an **HFSS Analysis** setup solution to this project using basic, initial meshing tools.

1. From the **Simulation** ribbon tab, navigate to **HFSS** > **Advanced** to open the **Setup** window.

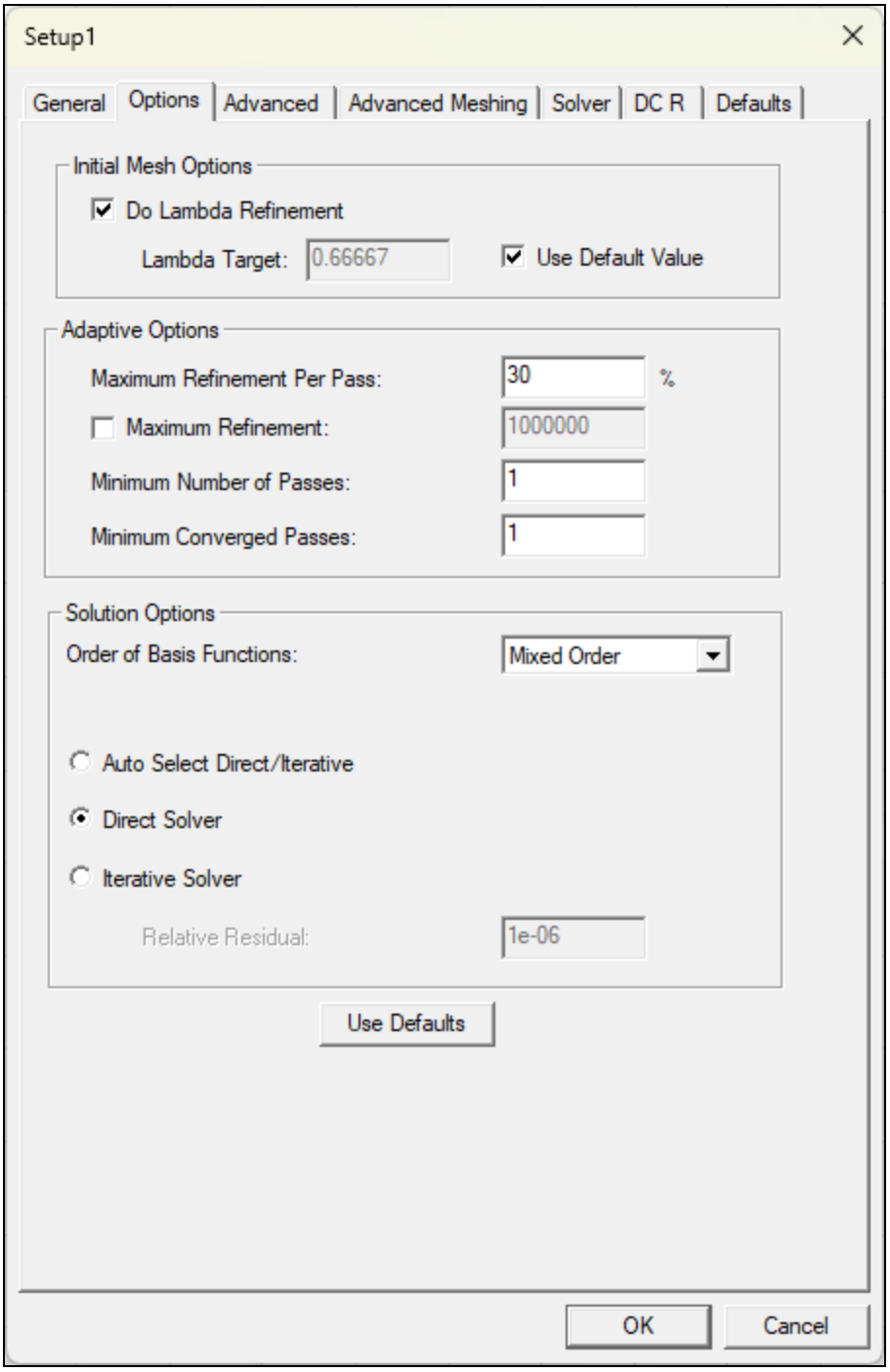


2. Enter **20** in the **Frequency** field.
3. From the **Fields** area, check the **Save fields** box.

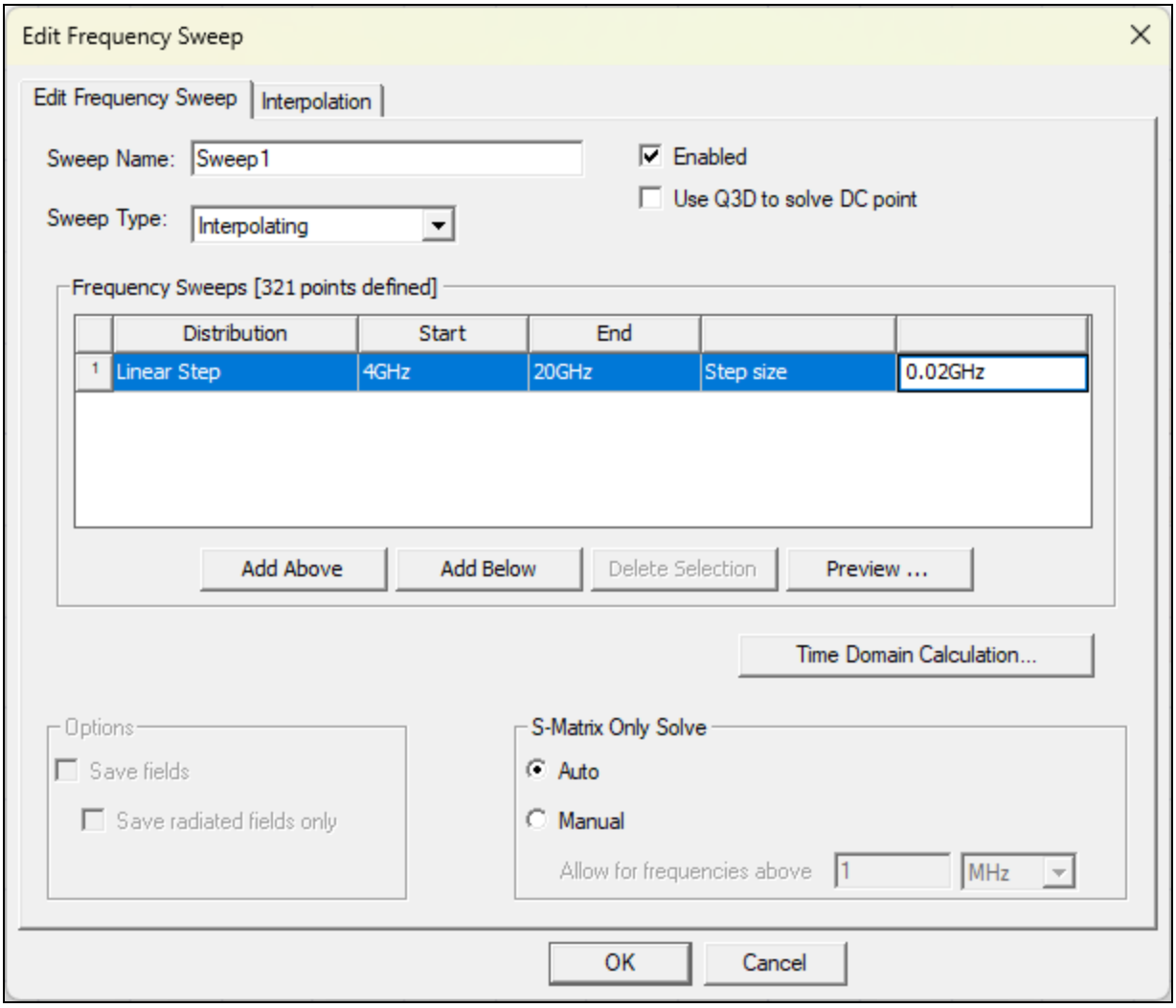


4. Navigate to the **Options** tab.

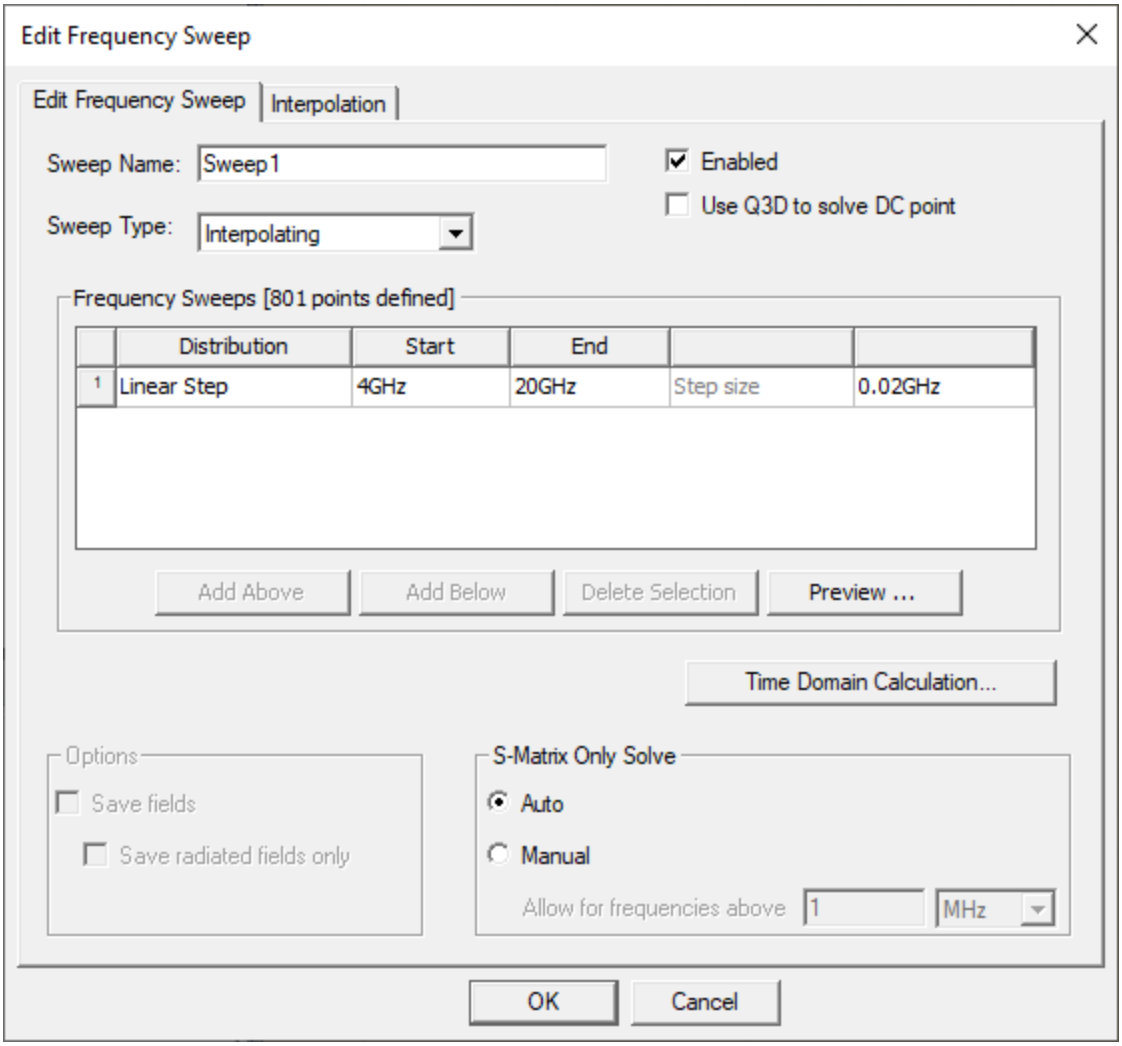
5. Ensure **Mixed Order** is selected from the **Order of Basis Functions** drop-down menu.



6. Click **OK** to close the **Setup** window and open the **Edit Frequency Sweep** window.



7. Ensure **Interpolating** is selected from the **Sweep Type** drop-down menu and **Linear Step** is selected from the **Distribution** drop-down menu.
8. Enter the following parameters in the first row of the **Frequency Sweeps** table:
   - **4** (GHz) in the **Start** field.
   - **20** (GHz) in the **End** field.
   - **0.02** (GHz) in the **Step size** field.



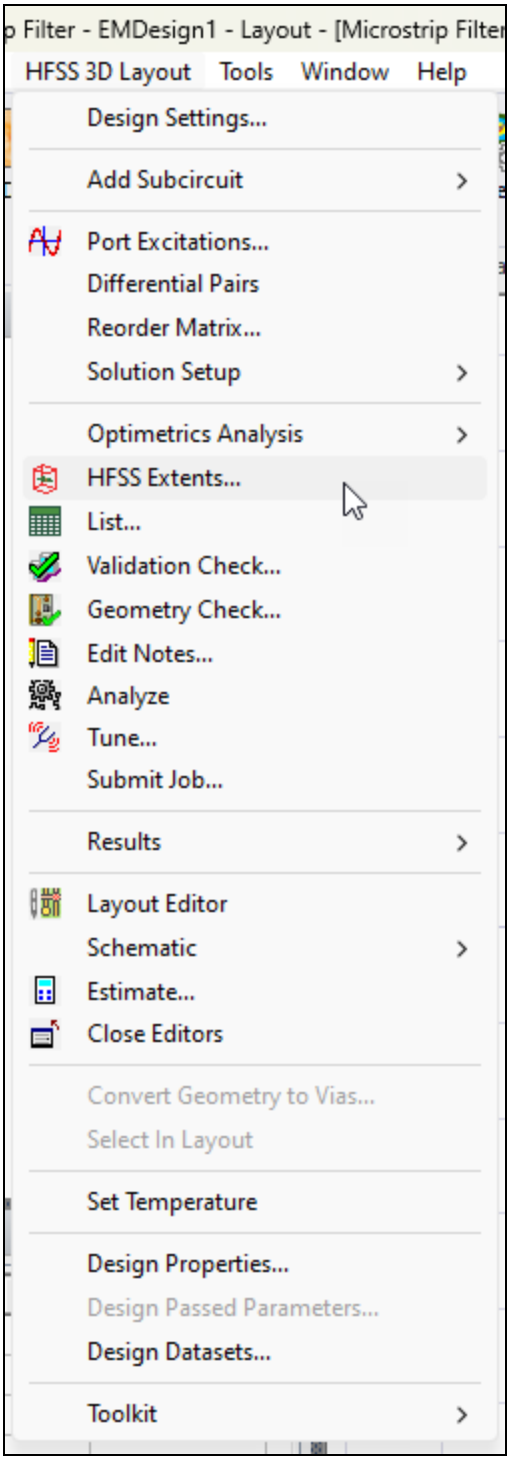
9. Click **OK** to add the interpolating sweep and close the **Edit Frequency Sweep** window.

Continue to Displaying the HFSS Bounding Box.

Displaying the HFSS Bounding Box

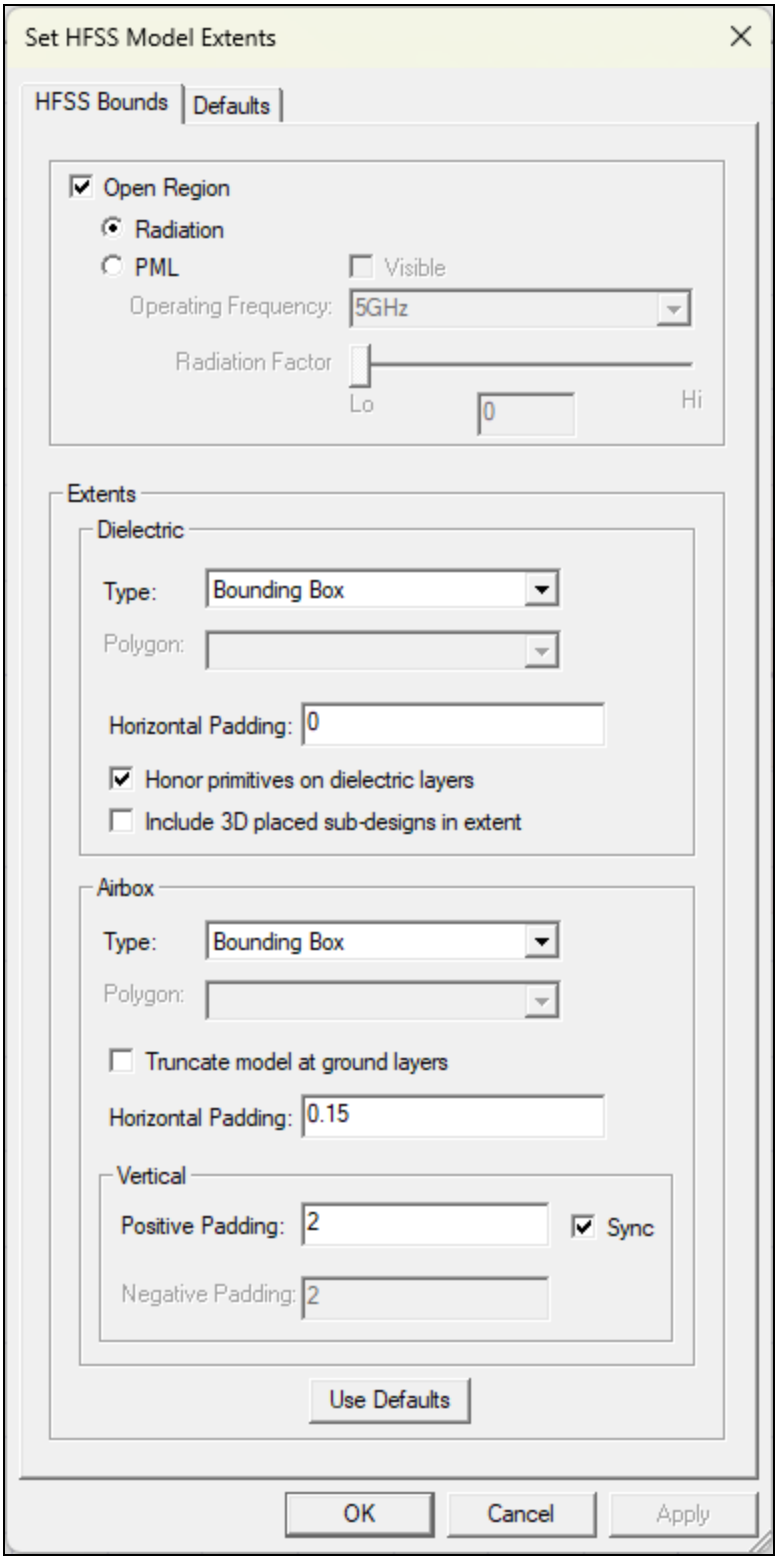
Follow these steps to display and then hide a bounding box around the complete design in the **Layout Editor**.

1. From **HFSS 3D Layout**, select **HFSS Extents** to open the **Set HFSS Model Extents** window.

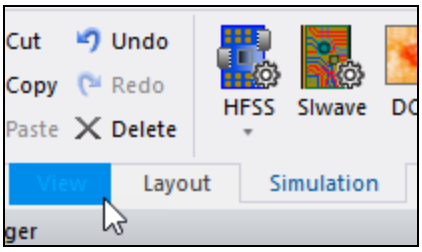


2. Ensure the **Open Region** box is checked and **Radiation** is selected.
3. From the **Dialectric** area, enter **0** in the **Horizontal Padding** field.

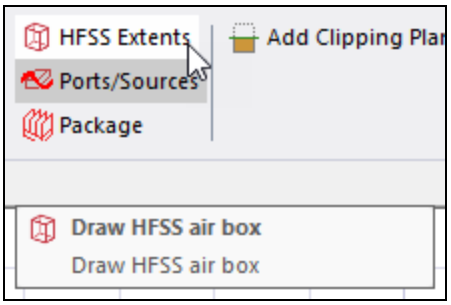
4. Click **Apply** to save changes.



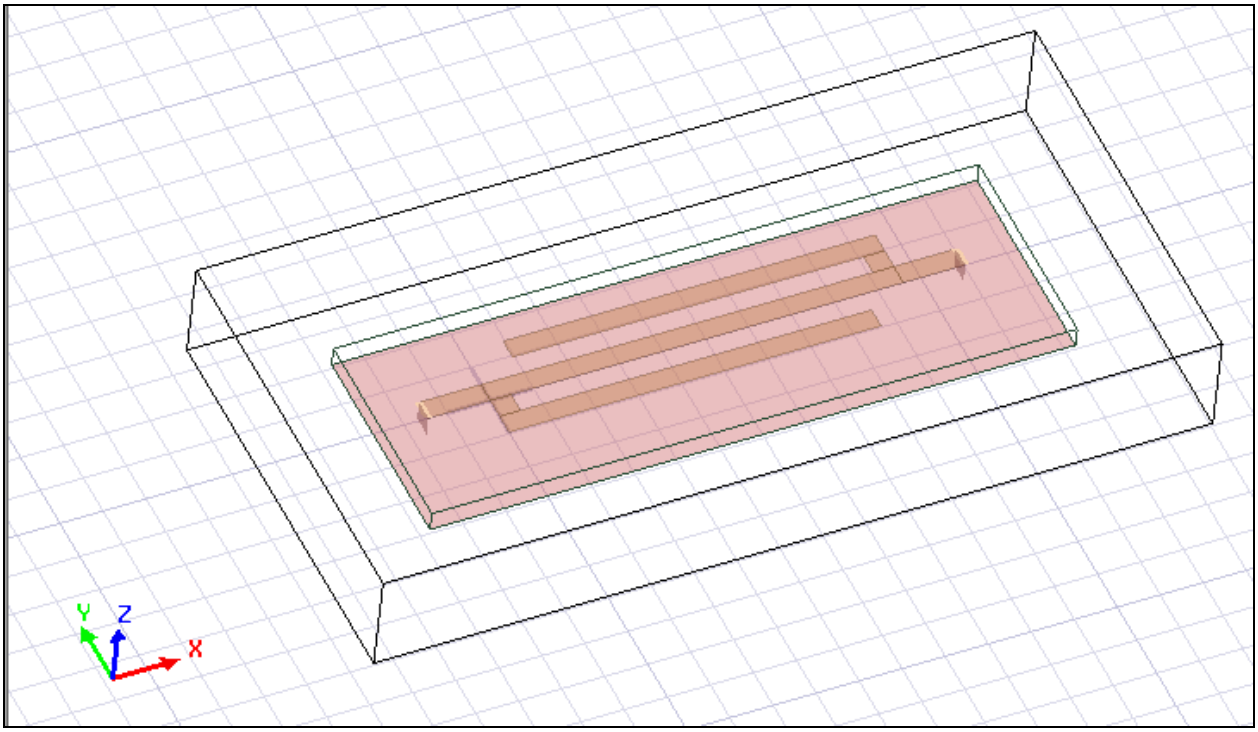
5. Click **OK** to close the **Set HFSS Model Extents** window.
6. Navigate to the **View** ribbon.



7. Click **HFSS Extents** to display a bounding box around the design in the **Layout Editor**.



8. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.



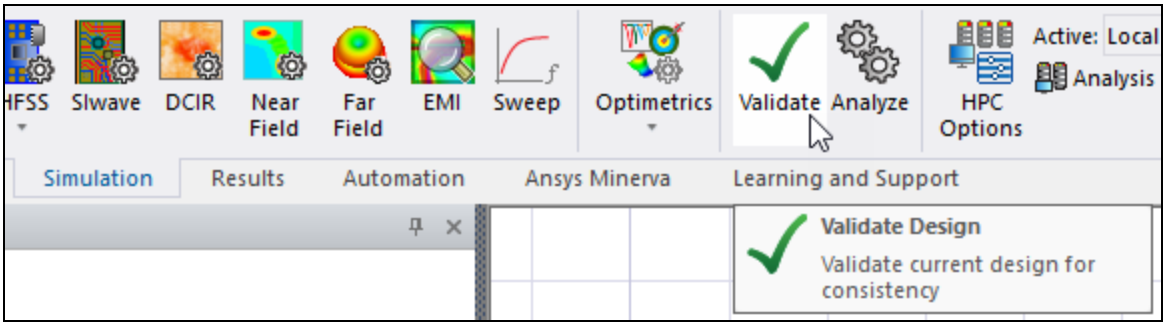
9. Repeat step **7** to hide the bounding box.

Continue to Validating and Analyzing the Design.

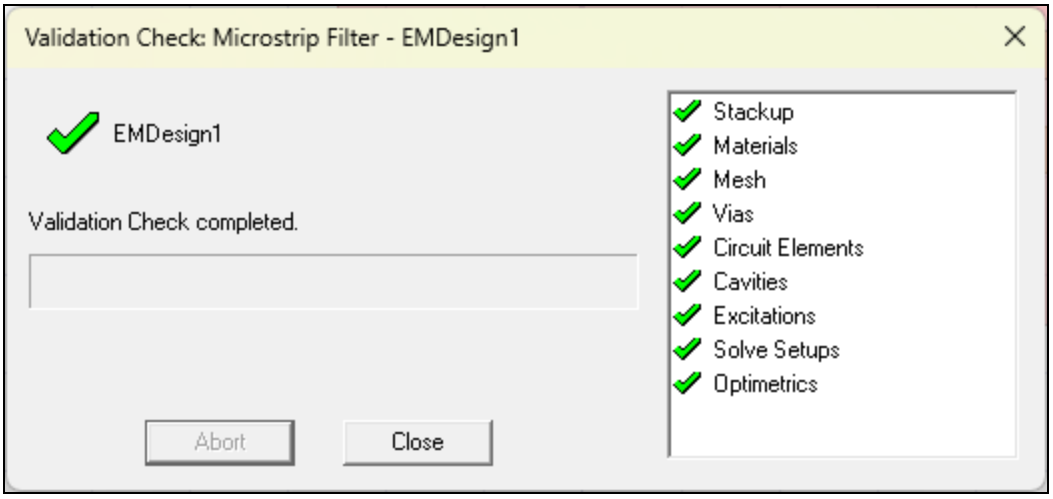
Validating and Analyzing the Design

Follow these steps to validate the active design and analyze the HFSS setup.

1. From the **Simulation** ribbon tab, click **Validate** to open the **Validation Check** window.



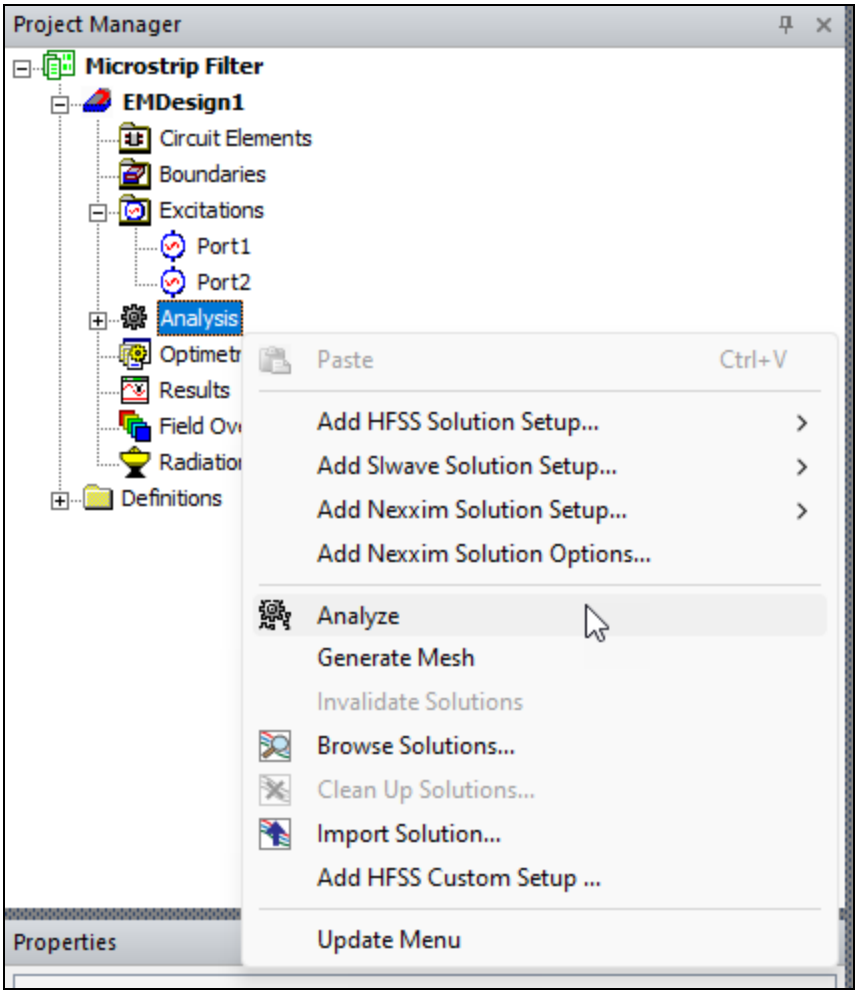
Assuming the design is valid, the **Validation Check** window will display the following result.



2. Click **Close**.

> **Note:**
>
> If the validation check detects any errors or warnings, view detailed feedback in the **Message Manager** window.

3. To sequentially run all analysis setups and associated frequency sweeps in the active design, navigate to the **Project Manager** window, right-click **Analysis**, and select **Analyze**.



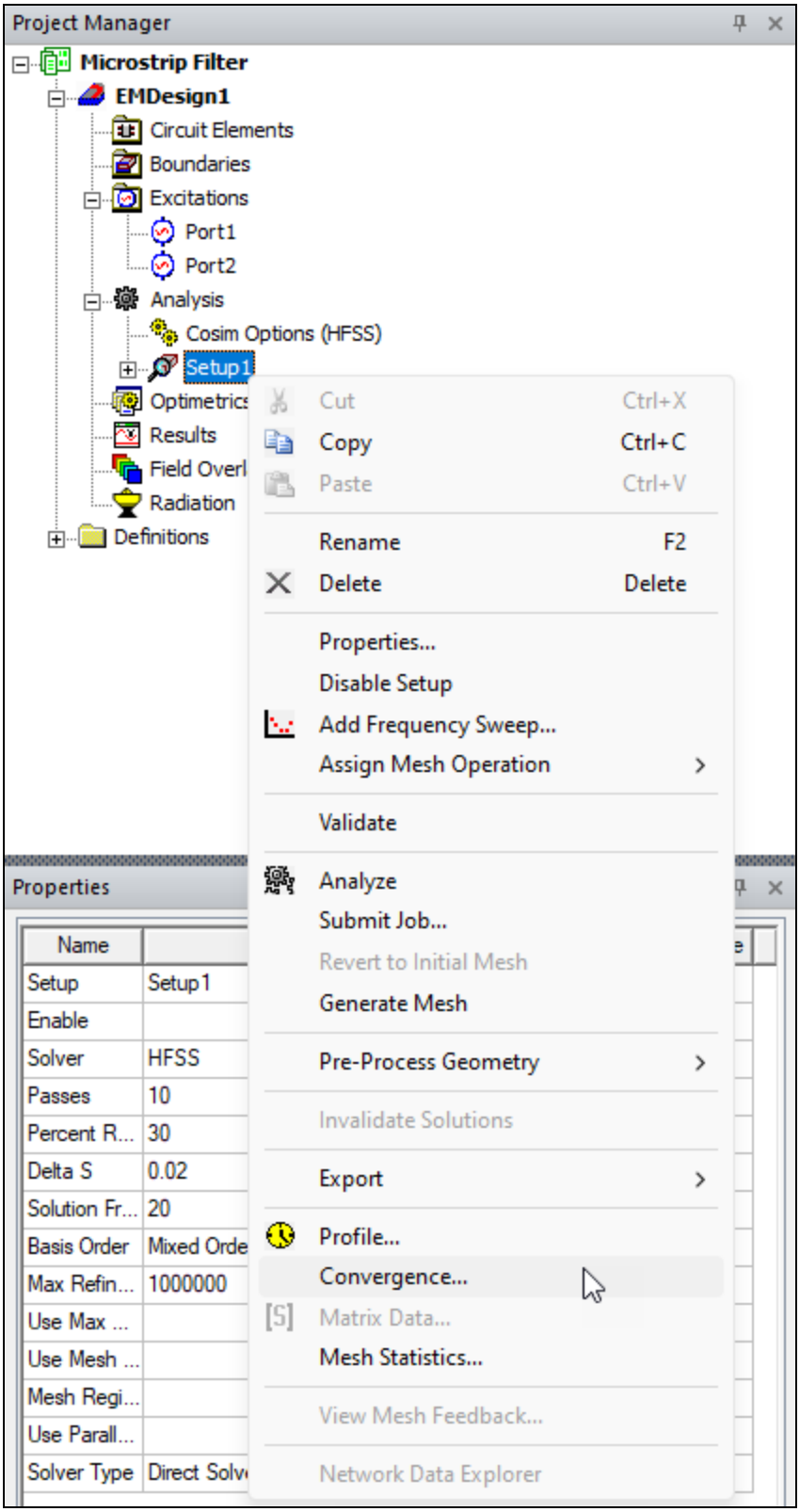
> **Note:**
>
> Alternatively, run a single setup or sweep by right-clicking the applicable setup or sweep and select **Analyze**.

Continue to Viewing Convergence History.

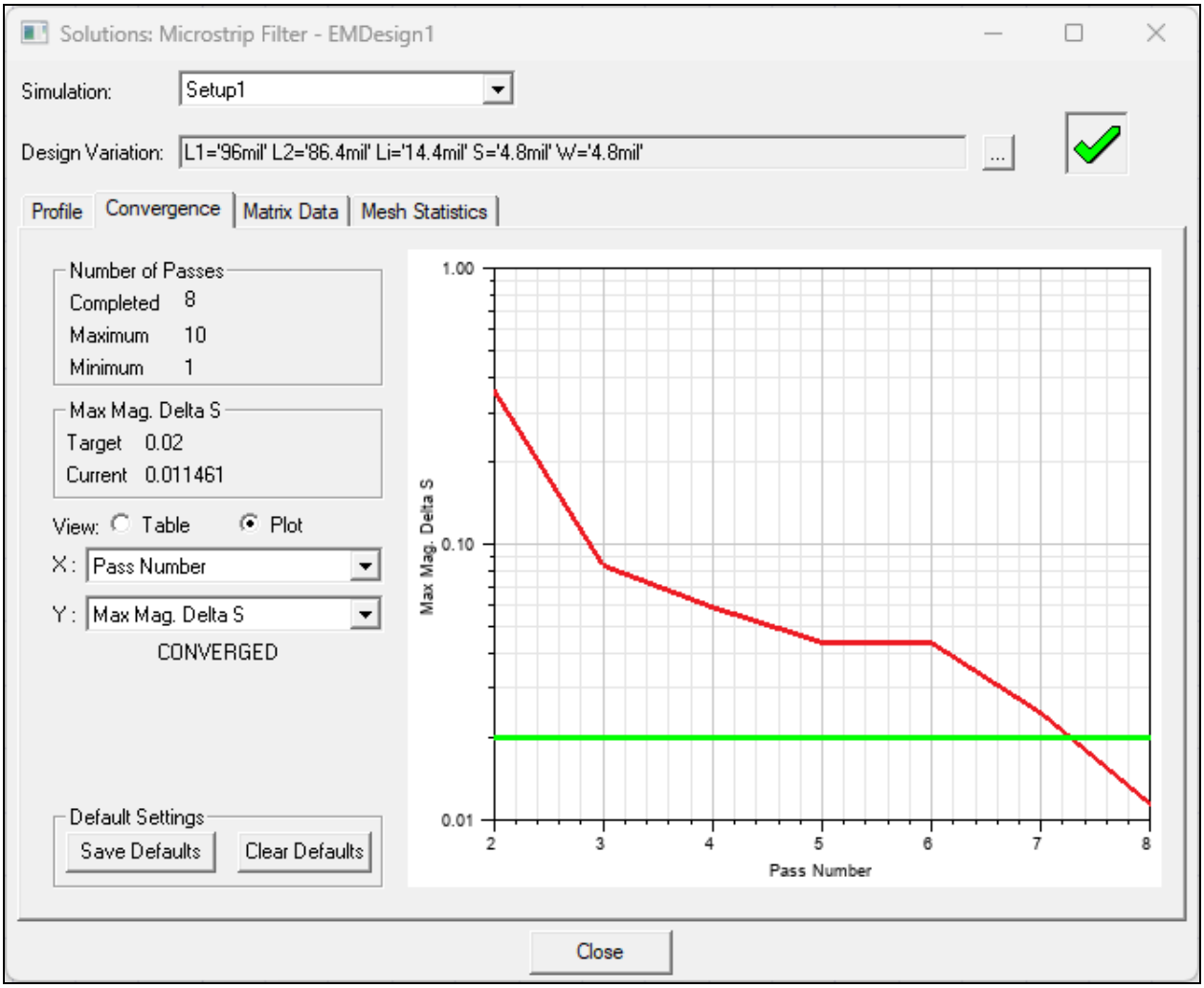
Viewing Convergence History

Follow these steps to view the active design's analysis convergence history.

1. From the **Project Manager** window, expand **Analysis**. Then right-click **Setup1** and select **Convergence** to open the **Solutions** window.



2. Select **View: Plot**.



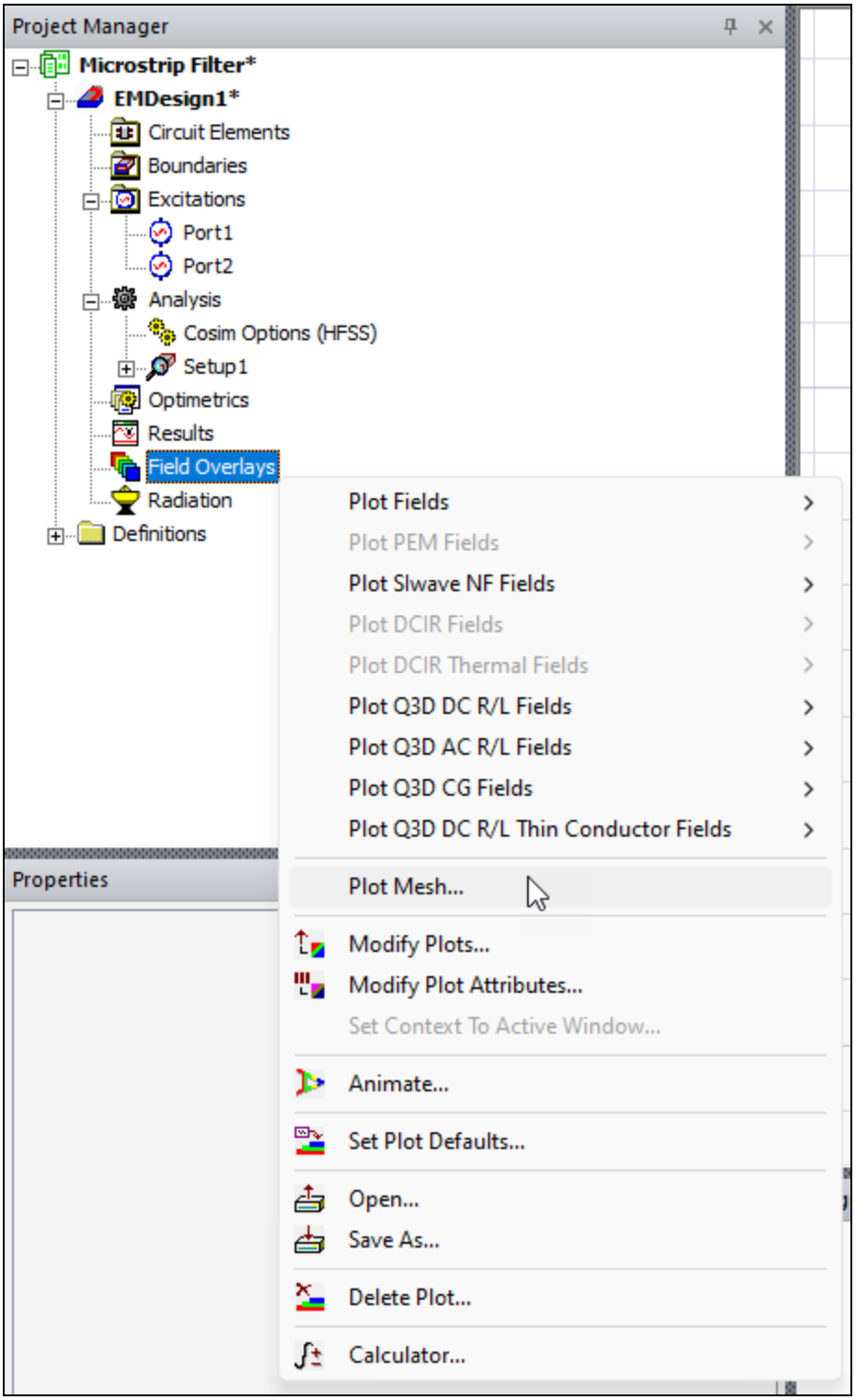
3. Click **Close**.

Continue to Plotting the HFSS Mesh.

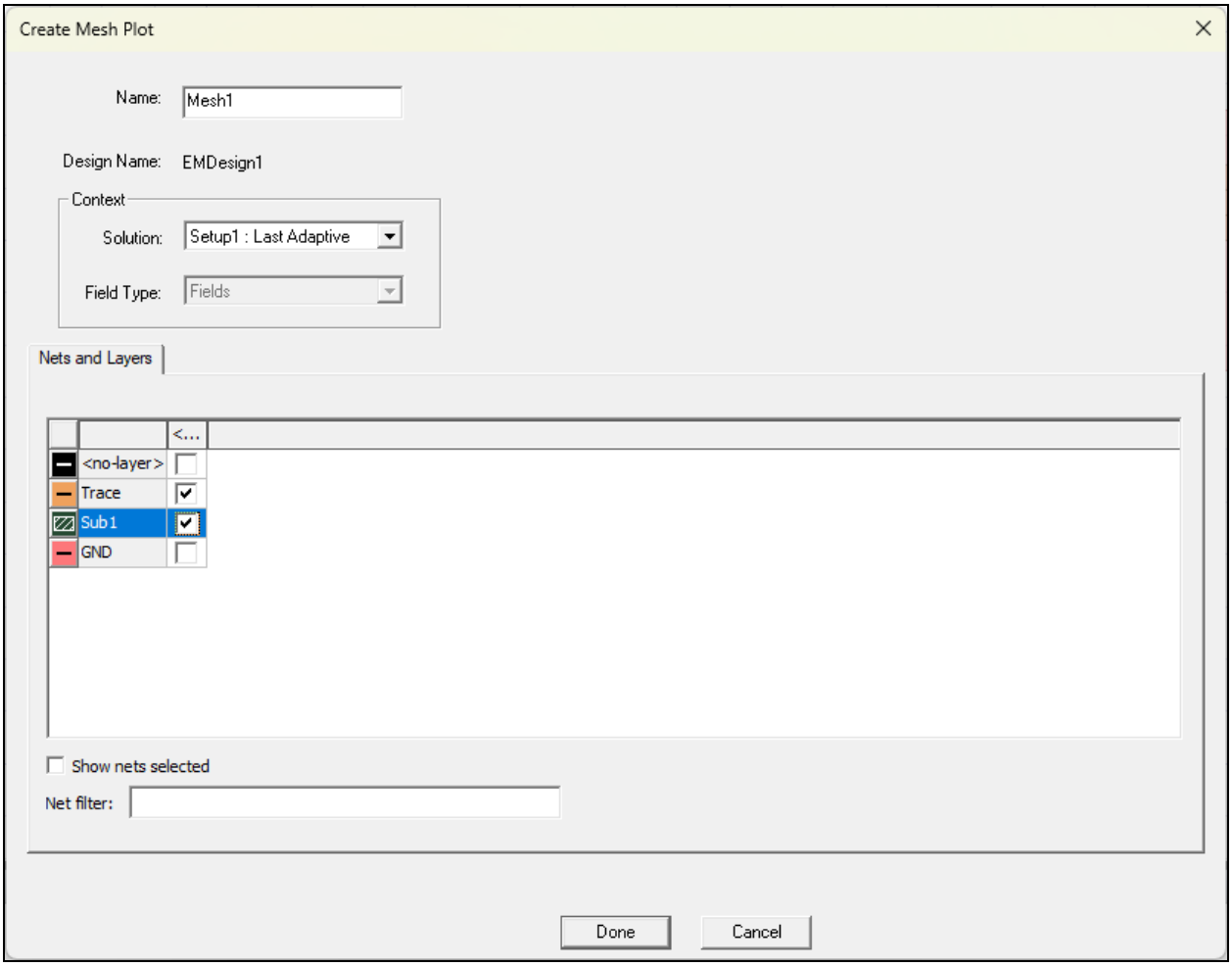
Plotting the HFSS Mesh

Follow these steps to plot and view a mesh of the HFSS Setup solution.

1. From the **Project Manager** window, right-click **Field Overlays** and select **Plot Mesh** to open the **Create Mesh Plot** window.



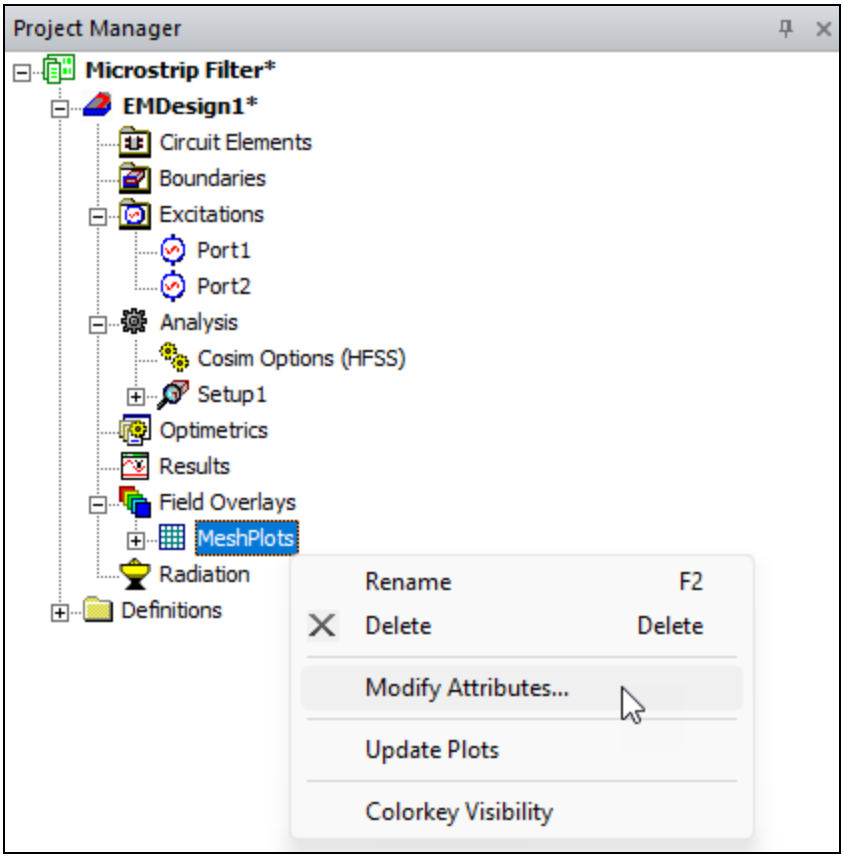
2. Under the **Nets and Layers** tab, check the **Trace** and **Sub1** boxes to only plot the mesh on those layers.



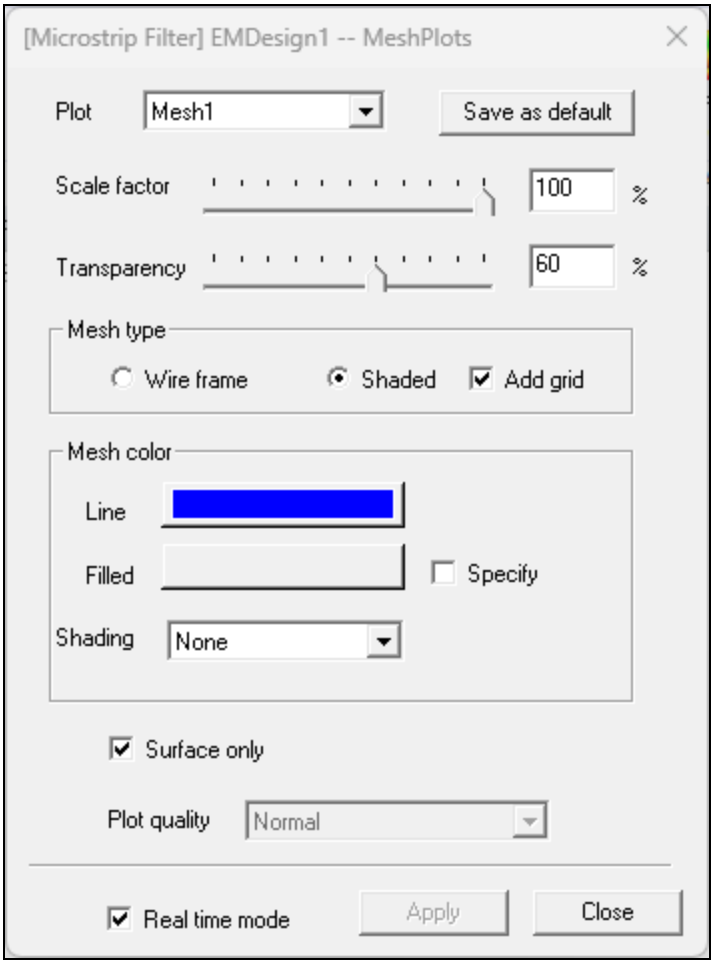
3. Click **Done** to close the **Create Mesh Plot** window and display the mesh for the **Setup1 : Last Adaptive** solution in the **Layout Editor**.

> **Note:**
>
> To change a mesh, from the **Project Manager** window, expand **Field Overlays** > **MeshPlots**. Then right-click the mesh (e.g., **Mesh1**) and select **Reassign**.

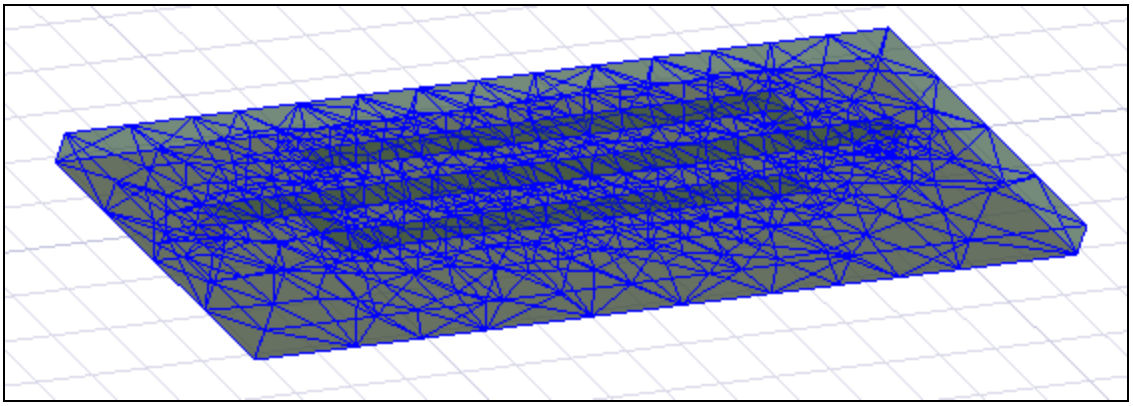
4. From the **Project Manager** window, expand **Field Overlays**. Then right-click **MeshPlots** and select **Modify Attributes** to open the **MeshPlots** window.



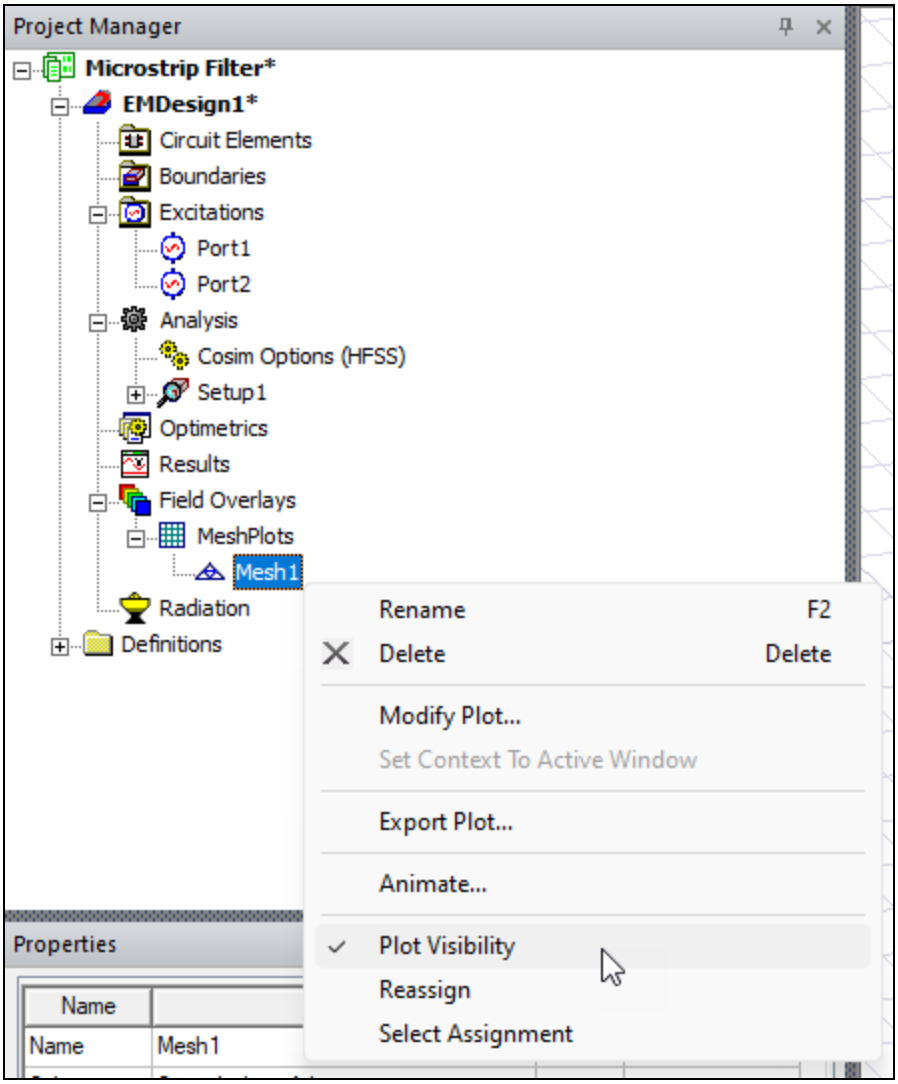
5. Either **click+drag** the **Transparency** scale marker or enter **60** in the field to improve the visibility of the mesh lines (i.e., element edges).



6. Click **Apply**.
7. Click **Close** to return to the **Layout Editor**.
8. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.



9. From the **Project Manager** window, right-click the mesh (e.g., **Mesh1**) and select **Plot Visibility** to remove the adjacent check mark and hide the mesh.

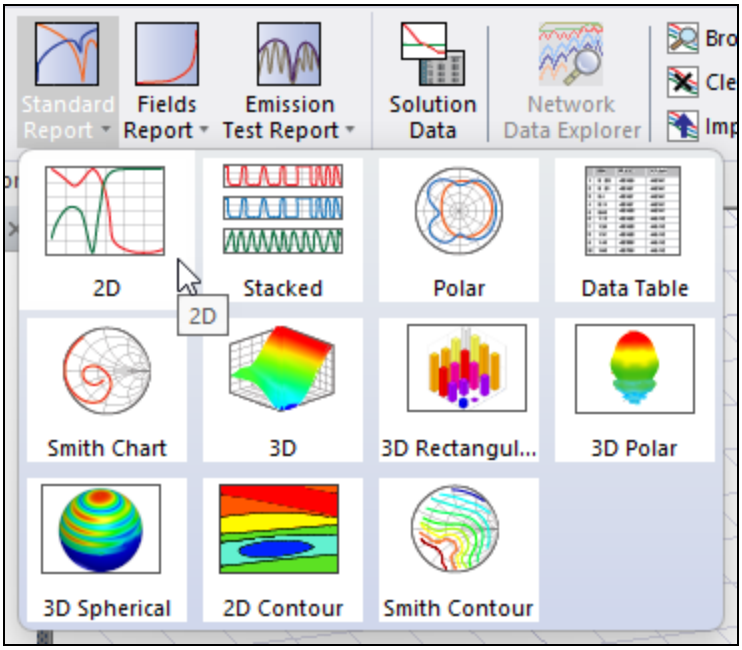


Continue to Creating the S-Parameter Plot.

Creating the S-Parameter Plot

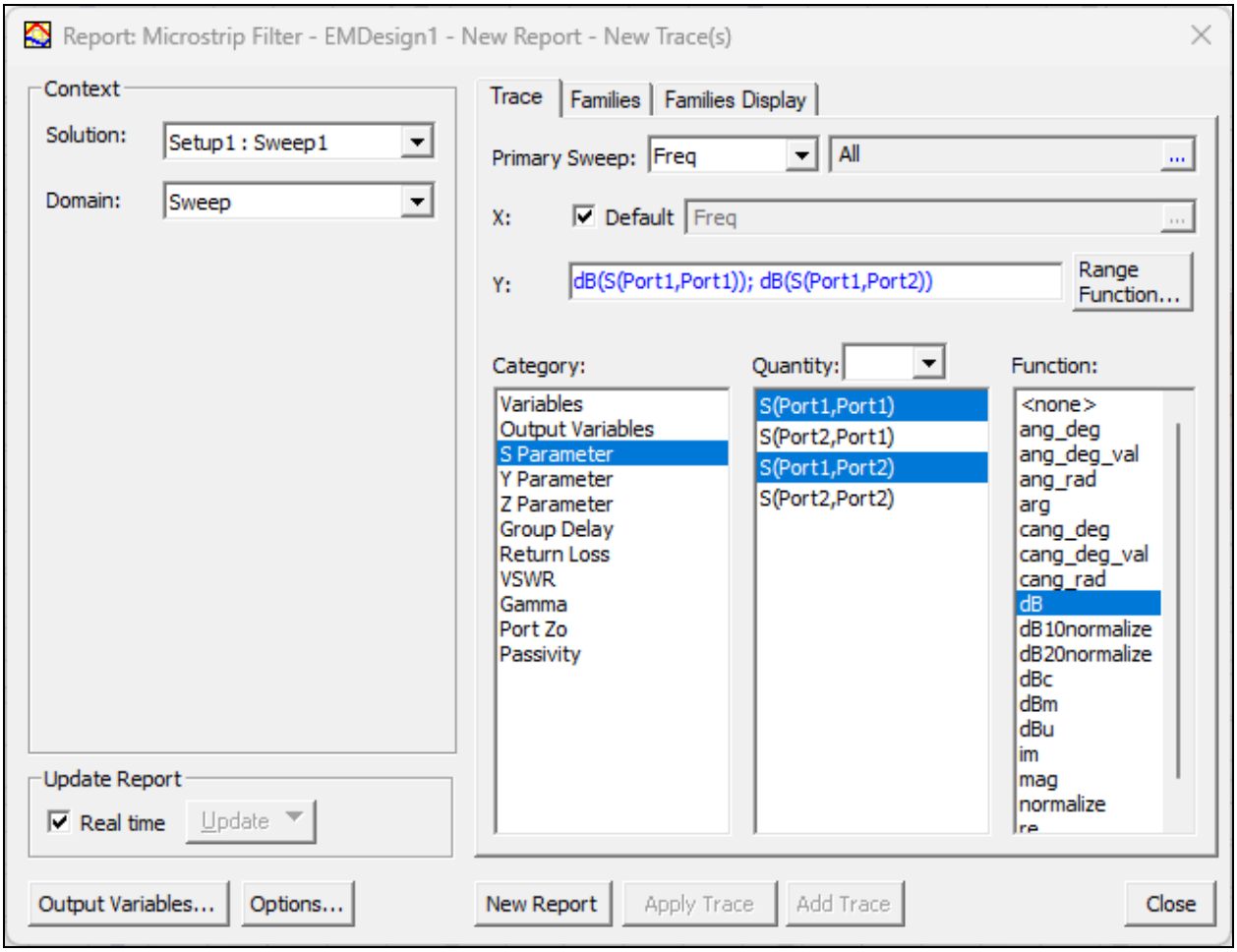
Follow these steps to create an S-Parameter plot with two traces, two each, then compare the result of the solution types.

1. From the **Results** ribbon tab, select **Standard Report** > **2D** to open the **Report** window.



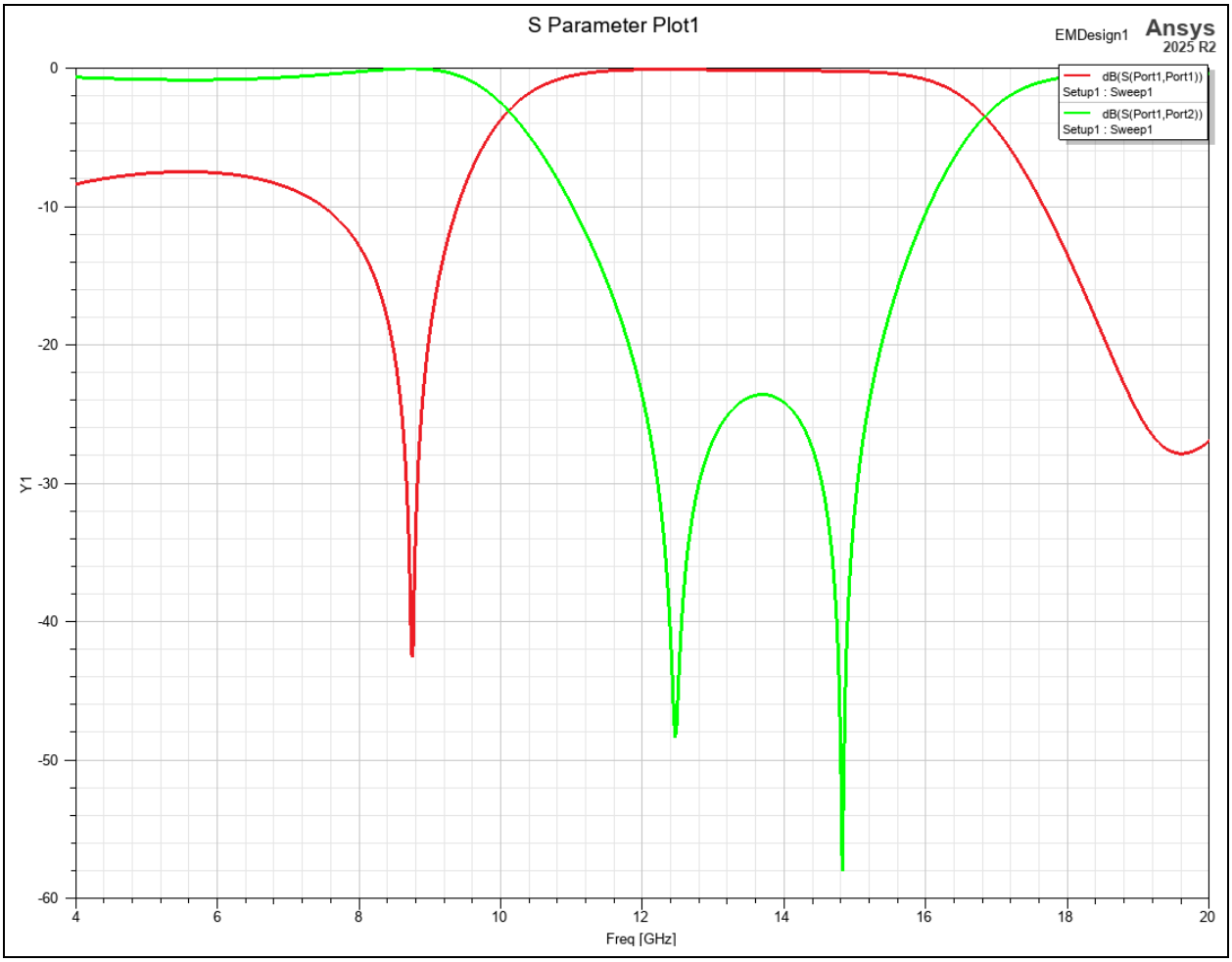
2. From the **Report** window, ensure the following settings are selected:
   - From the **Solution** drop-down menu, **Setup1 : Sweep1**.
   - From the **Domain** drop-down menu, **Sweep**.
   - From the **Category** list, **S Parameter**.
   - From the **Function** list, **dB**.

3. From the **Quantity** list, **Ctrl+click** to select **S(Port1,Port1)** and **S(Port1,Port2)**.

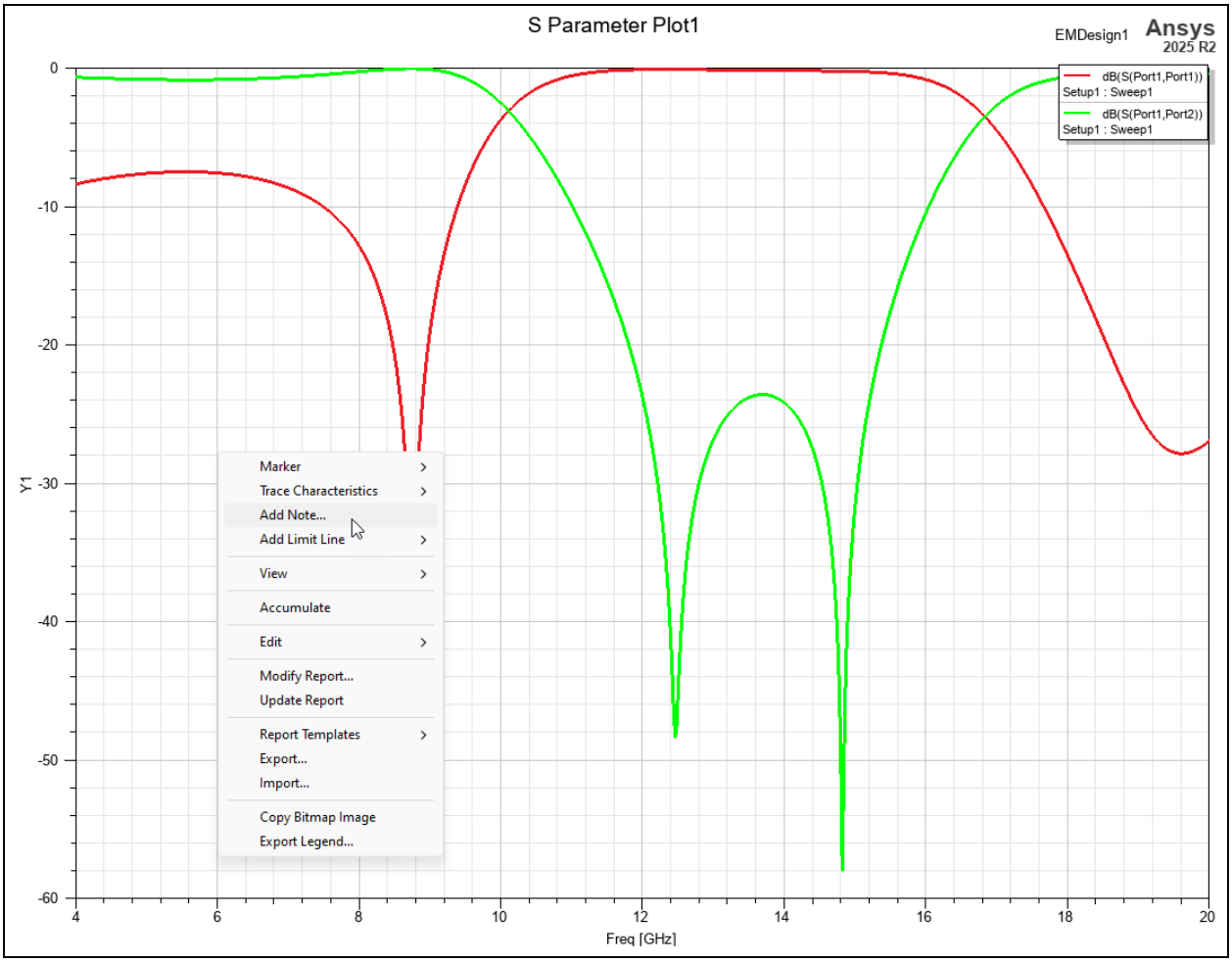


4. Click **New Report** and the return loss plot opens under the **Report** window.

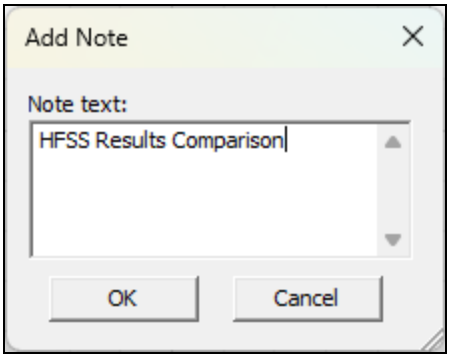
5. **Close** the **Report** window to view the plot.



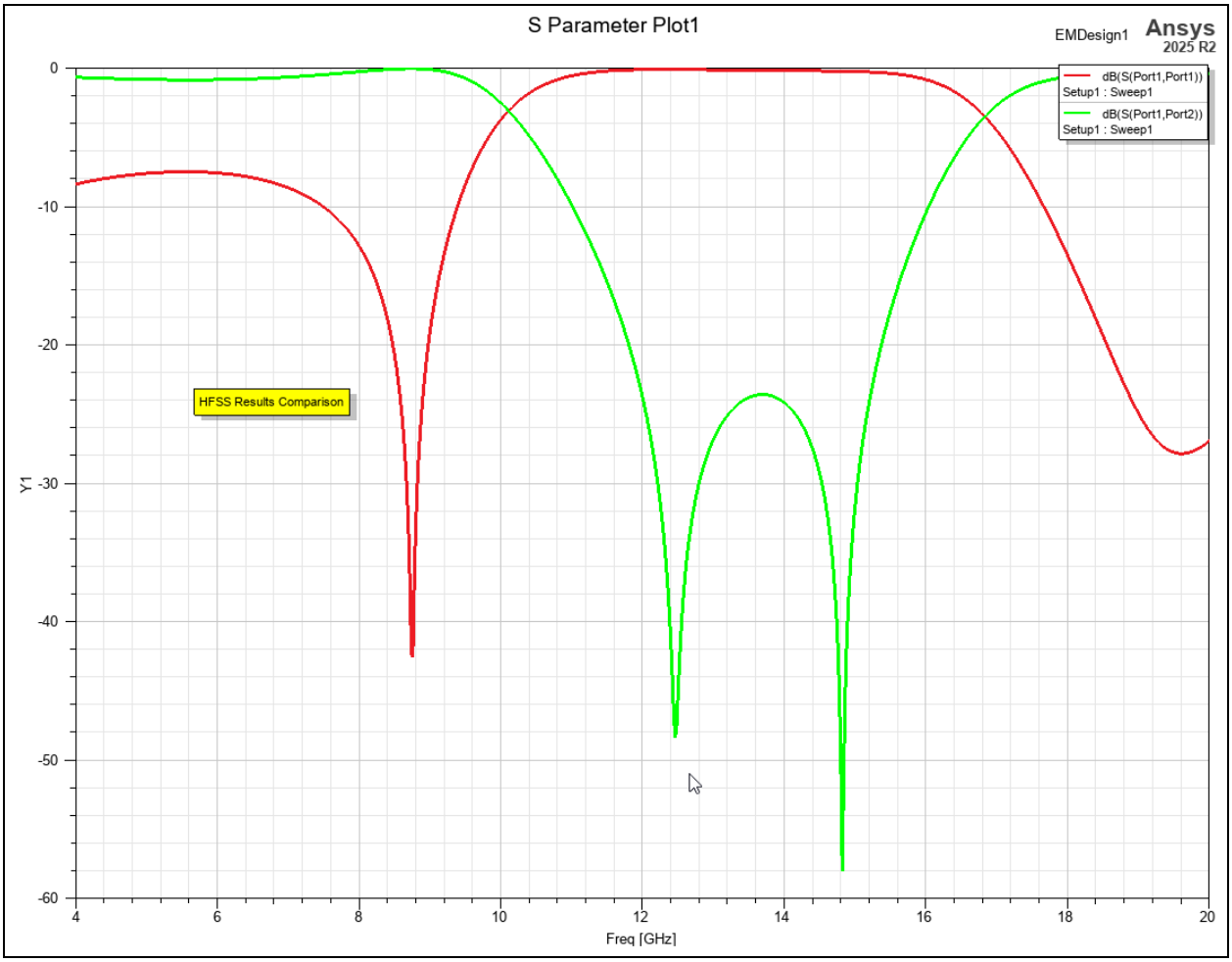
6. Right-click within the report window and select **Add Note** to open the **Add Note** window.



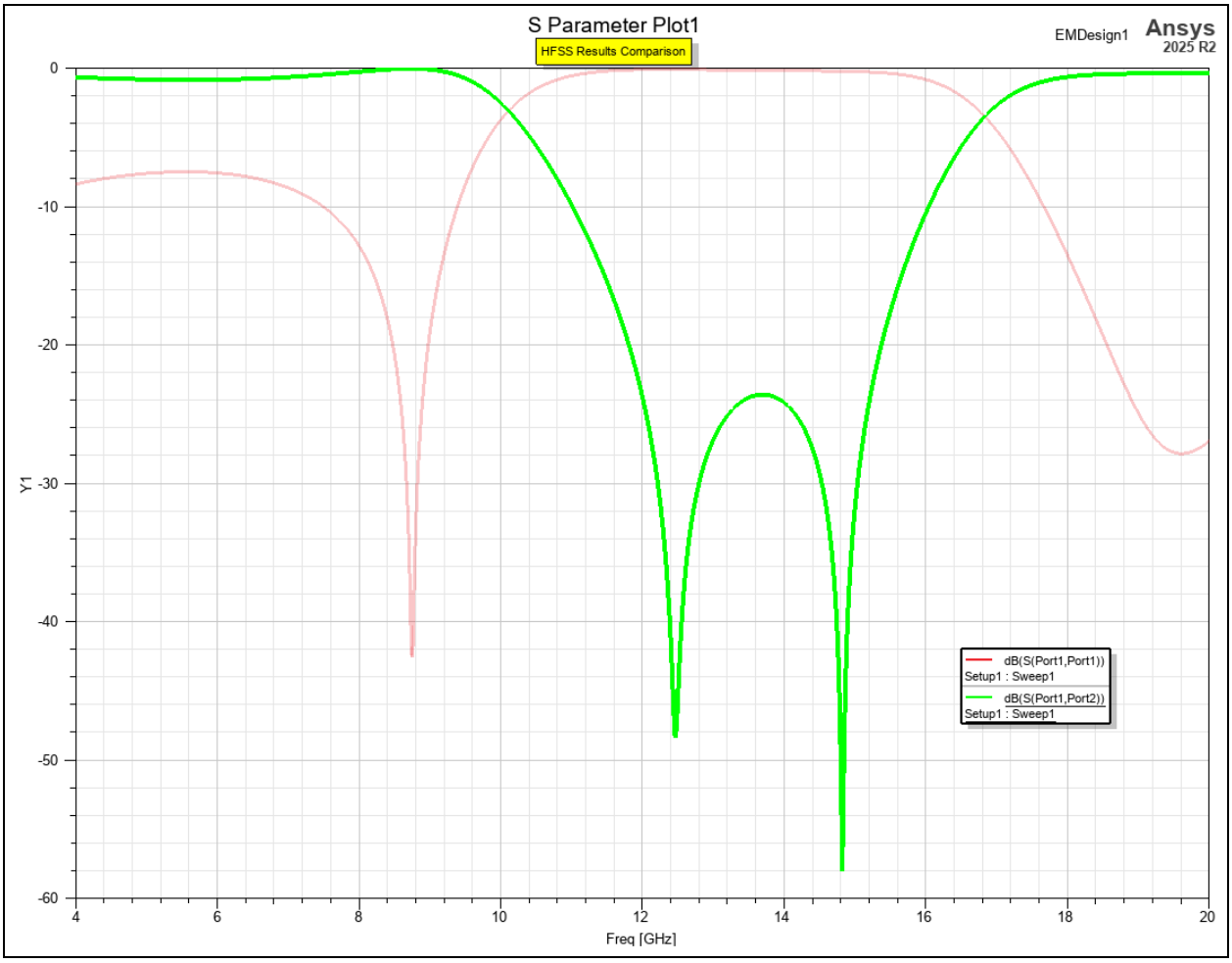
7. Enter **HFSS Results Comparison** in the **Add Note** field.



8. Click **OK** to close the **Add Note** window and add the note to the plot.



9. Drag the note to a more desirable location. The legend can also be relocated, as necessary.



Note:

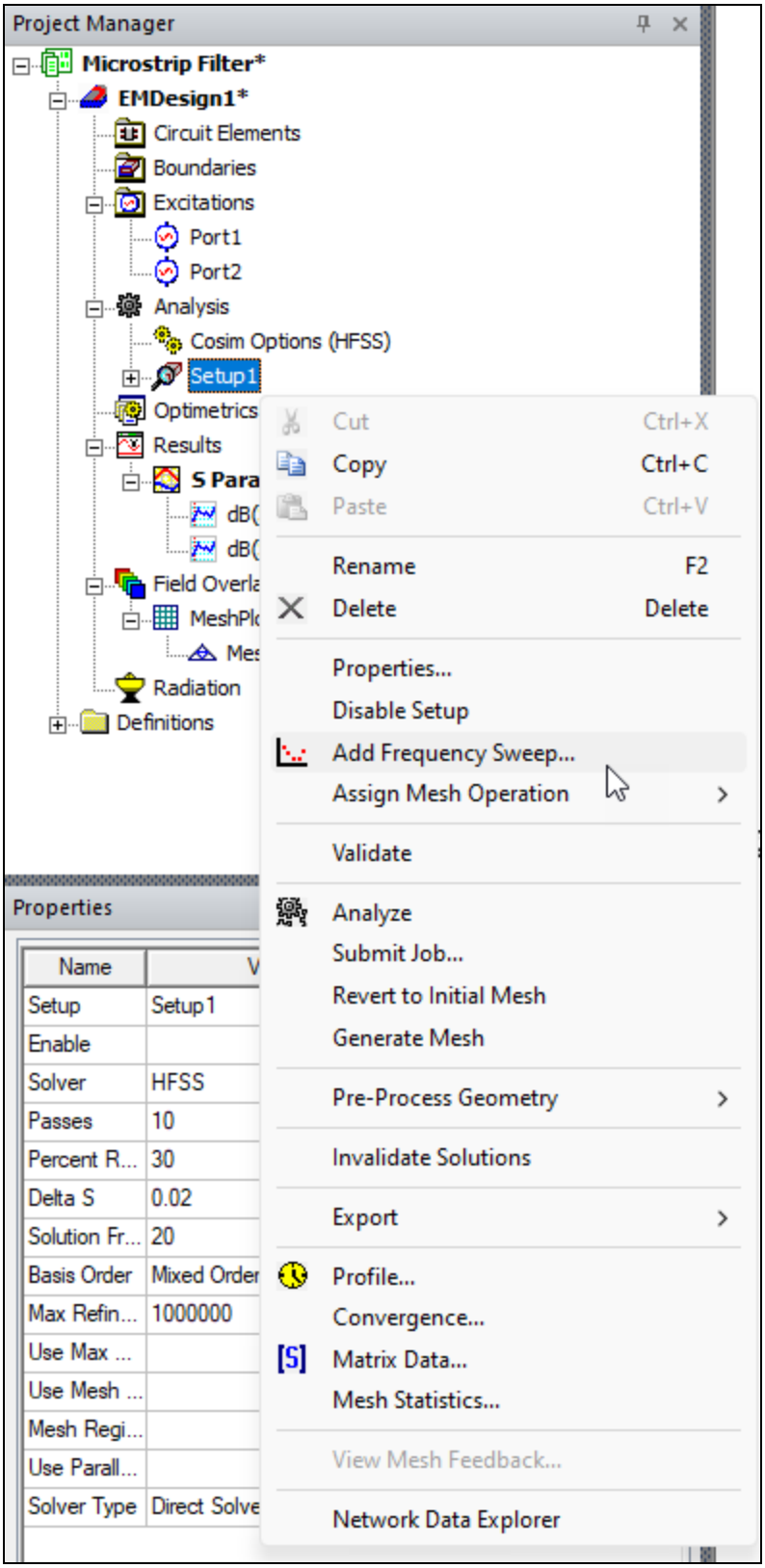
Both solutions demonstrate the bandstop behavior over the range of about 10 to 17 GHz. The only significant difference is in the magnitude of the three minimal points along the traces (at 8.75, 12.5, and 15.2 GHz).

Continue to Adding and Analyzing a Discrete Sweep.

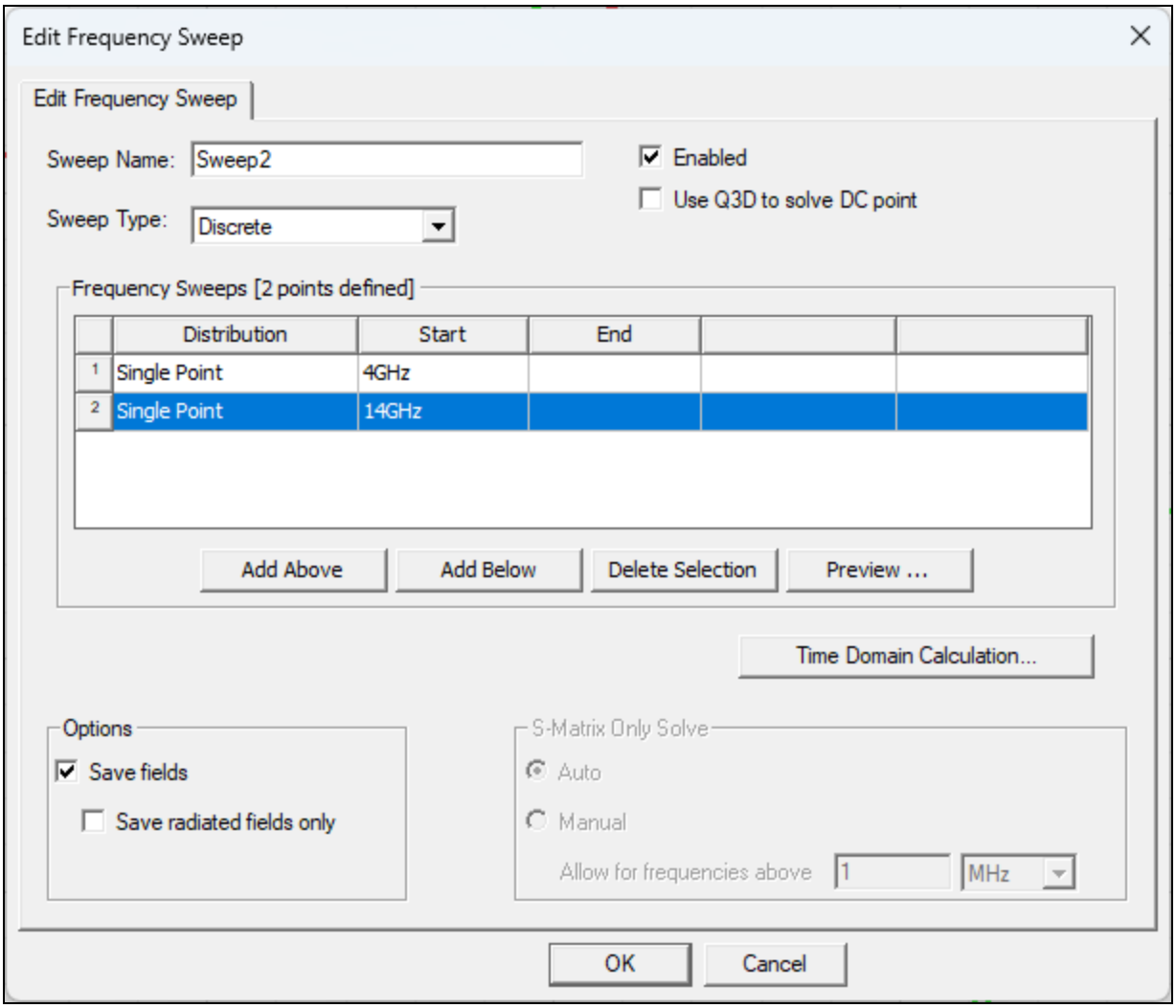
Adding and Analyzing a Discrete Sweep

In HFSS 3D Layout designs, surface current results are only available for the last pass of an adaptive mesh or for discrete sweeps. Before viewing surface current results from the conducting layers, follow these steps to define a discrete sweep with results at two frequencies: one at a low pass-through frequency and one in the middle of the bandstop range.

1. From the **Project Manager** window, right-click **Setup1** and select **Add Frequency Sweep** to open the **Add Frequency Sweep** window.

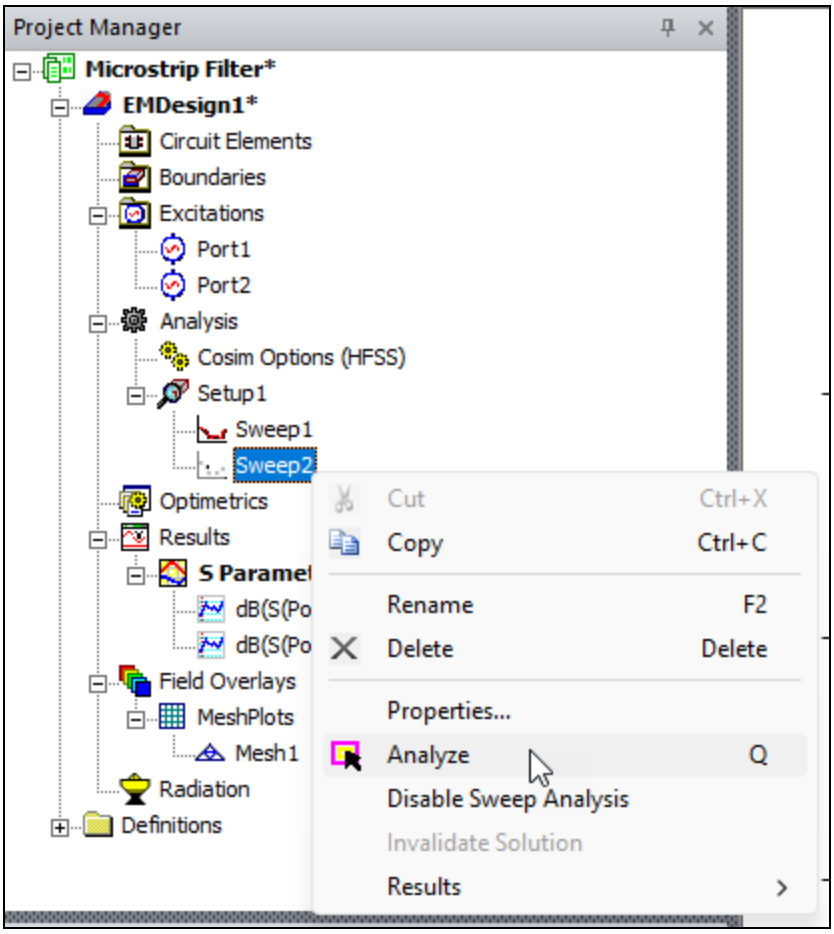


2. From the **Edit Frequency Sweep** window, do the following:
   - Select **Discrete** from the **Sweep Type** drop-down menu.
   - Select **Single Point** from the **Distribution** drop-down menu.
   - From the **Frequency Sweeps** table, enter **4** (GHz) in the **Start** column.
   - Click **Add Below**.
   - In the new row, enter **14** (GHz) in the **Start** column.
   - From the **Options** area, check the **Save fields** box.



3. Click **OK** to add the discrete sweep and close the **Edit Frequency Sweep** window.

4. From the **Project Manager** window, expand **Setup1** and right-click **Sweep2**. Then select **Analyze**.

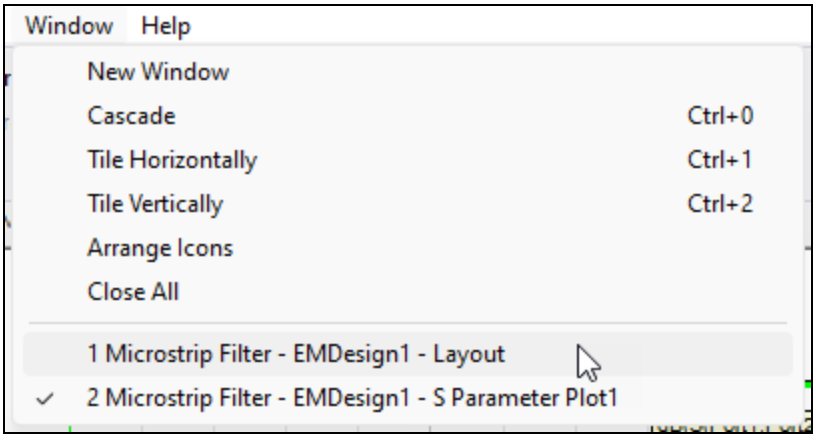


Continue to Creating and Animating the Current Overlay.

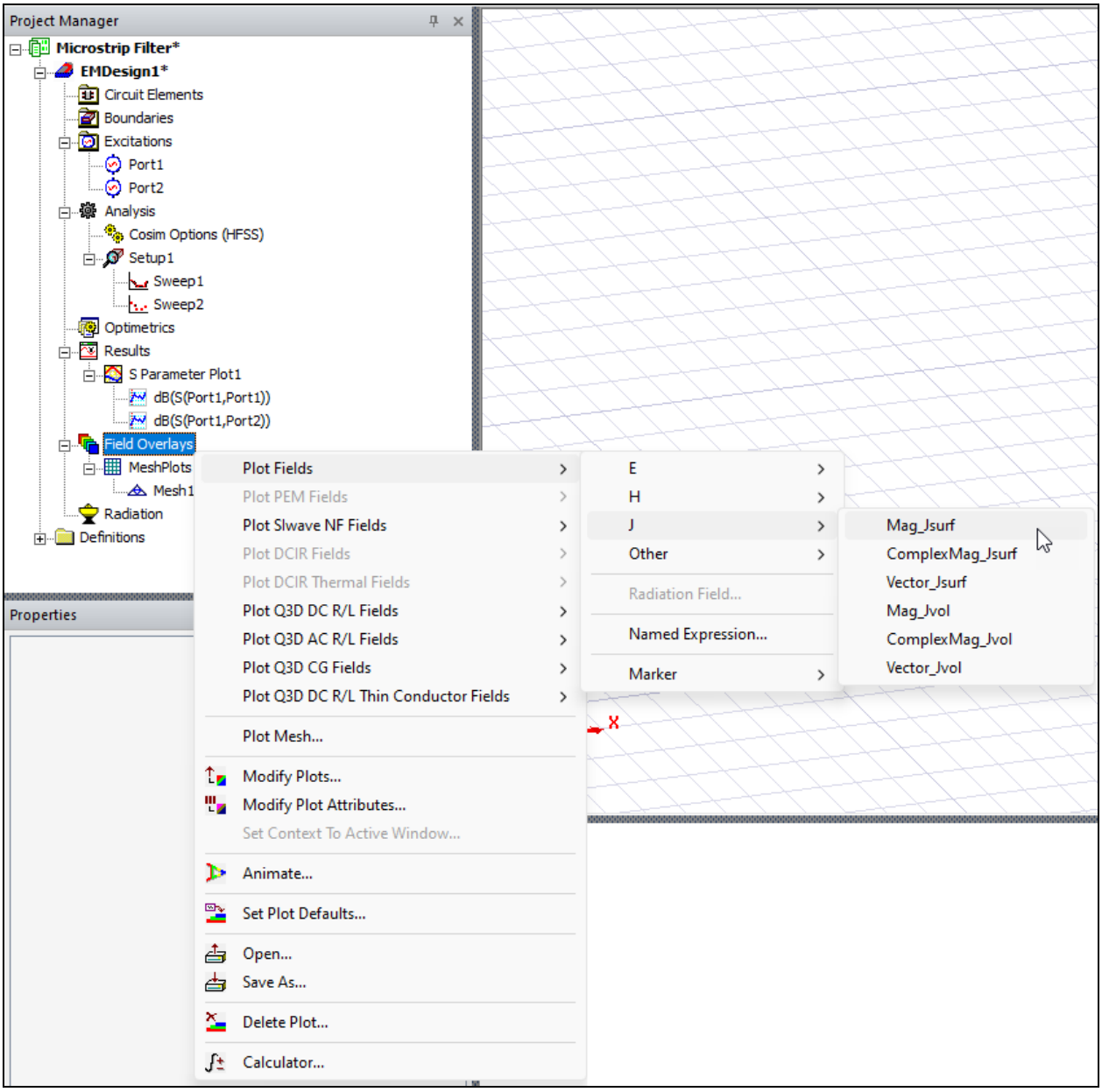
Creating and Animating the Current Overlay

Follow these steps to create and animate the current overlay.

1. Return to the design by navigating to **Window** > **Layout**.



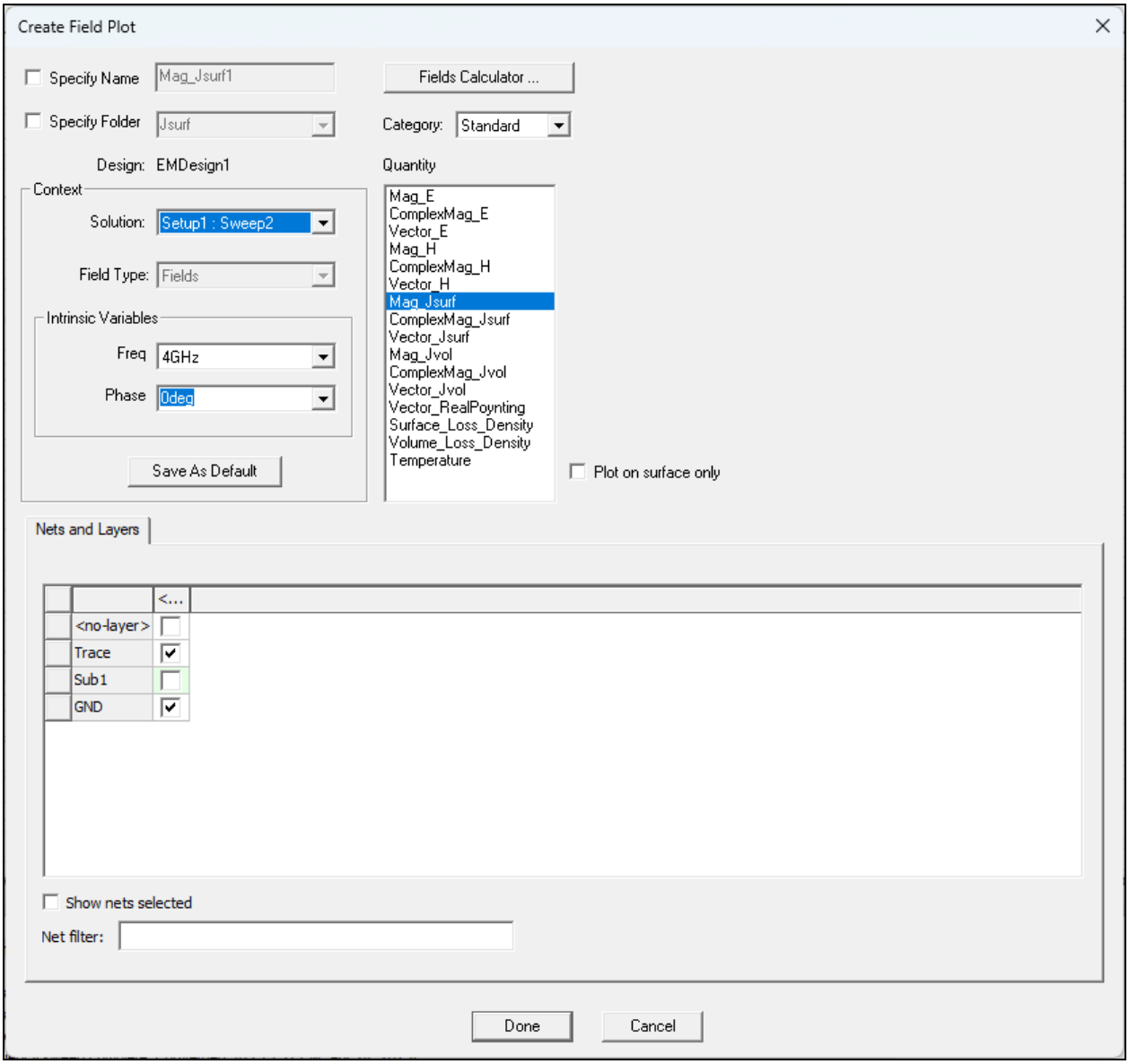
2. From the **Project Manager** window, right-click **Field Overlays** and select **Plot Fields** > **J** > **Mag_Jsurf** to open the **Create Field Plot** window.



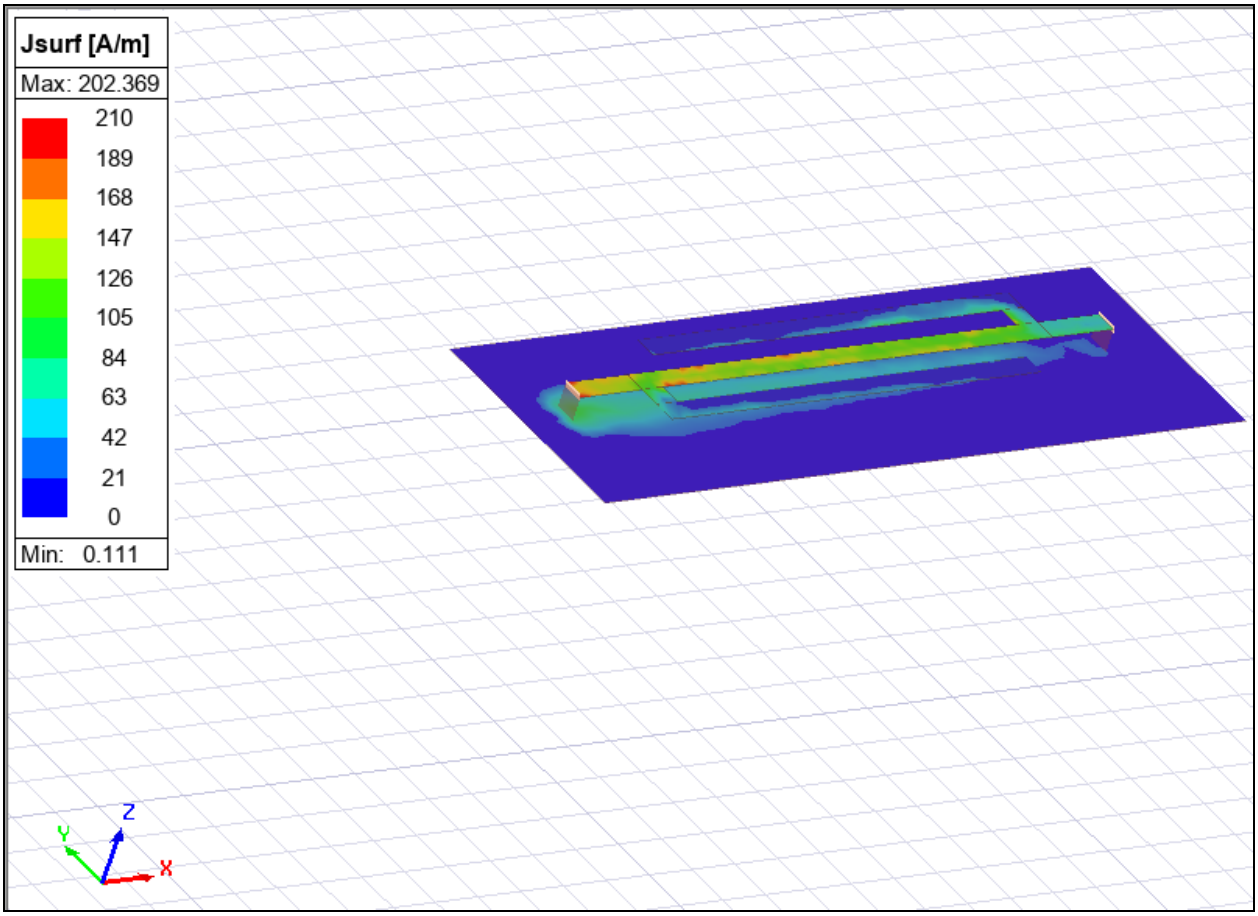
3. From the **Create Field Plot** window, do the following:

- Select **Setup1 : Sweep2** from the **Solution** drop-down menu.
- Ensure **4Ghz** is selected from the **Freq** (i.e., frequency) drop-down menu and **Mag_Jsurf** is selected from the **Quantity** list.

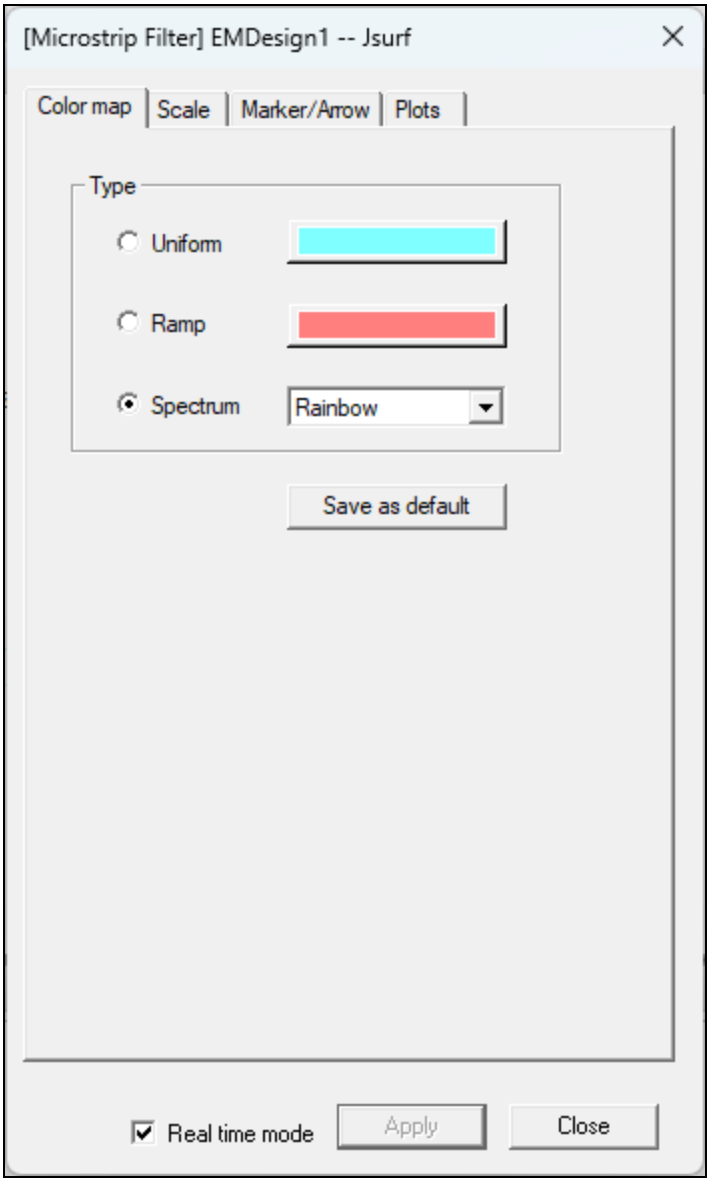
- Under the **Nets and Layers** tab, check the **Trace** and **GND** boxes.



4. Click **Done** to save changes, close the **Create Field Plot** window, and view the overlay.

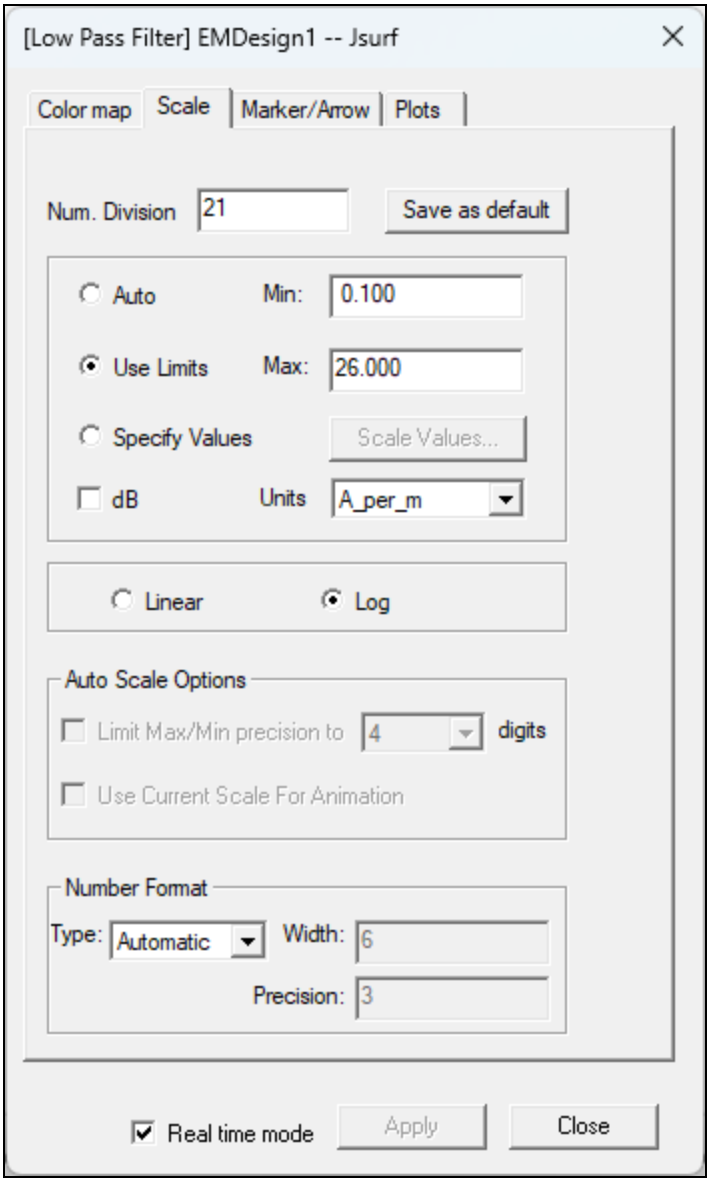


5. From the **Layout** tab, double-click in the current plot overlay legend to open the (**Jsurf**) plot settings window.

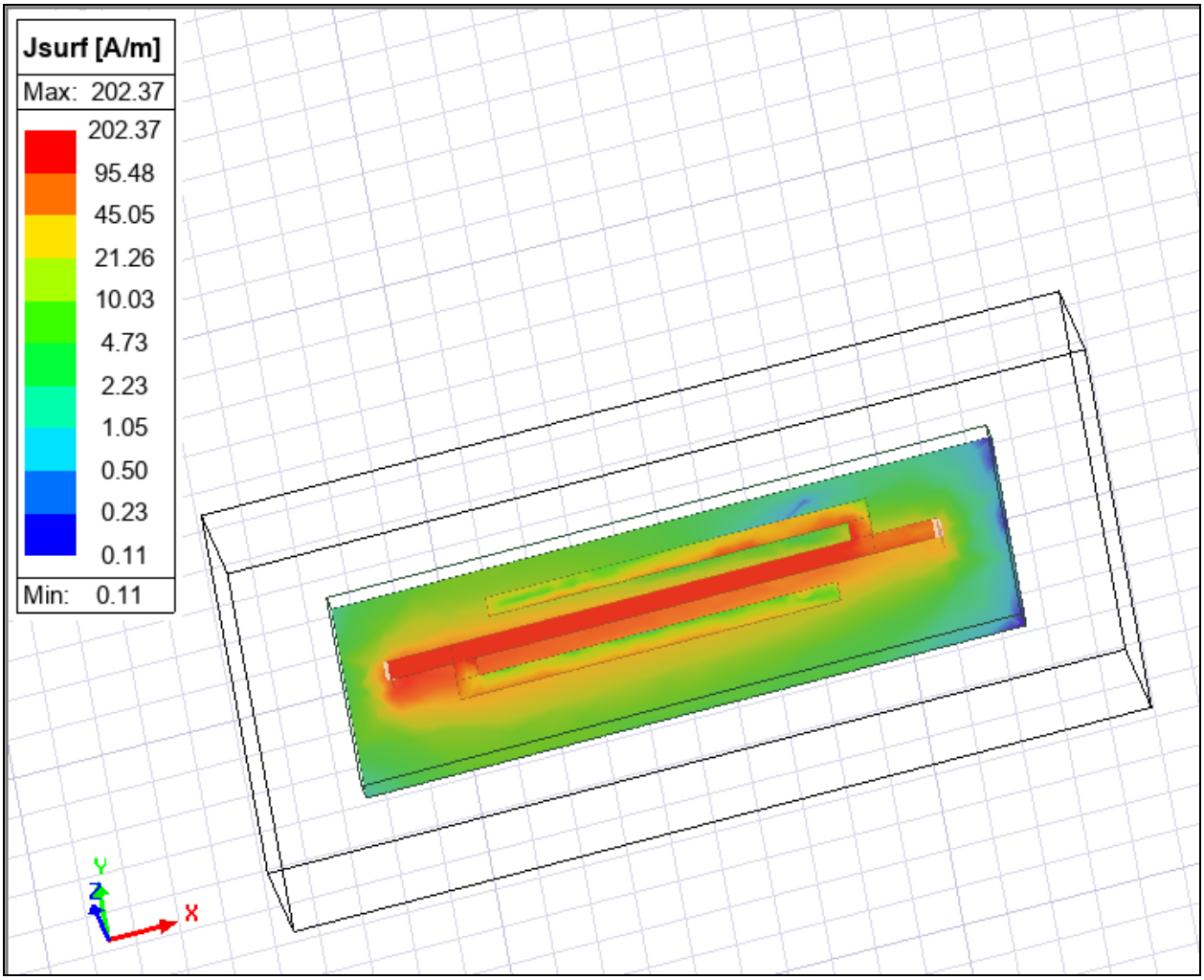


6. Select the **Scale** tab and do the following:
    a. Enter **10** in the **Num. Division** field.
    b. Select **Log** to produce a logarithmic scale.
    c. Select **Automatic** from the **Type** drop-down menu.
    d. Enter **2** in the **Precision** field.

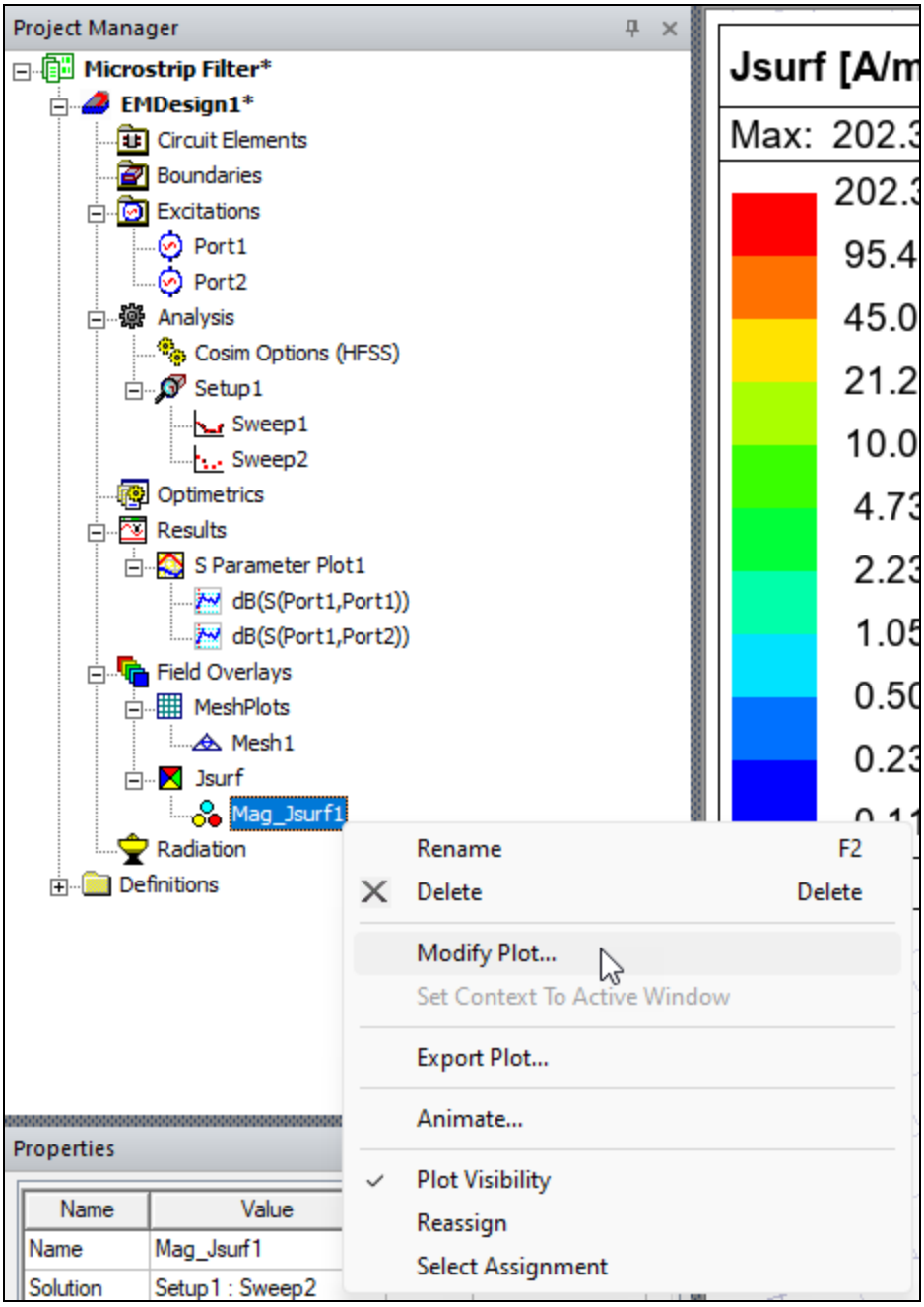
e. Click **Apply** to save changes.



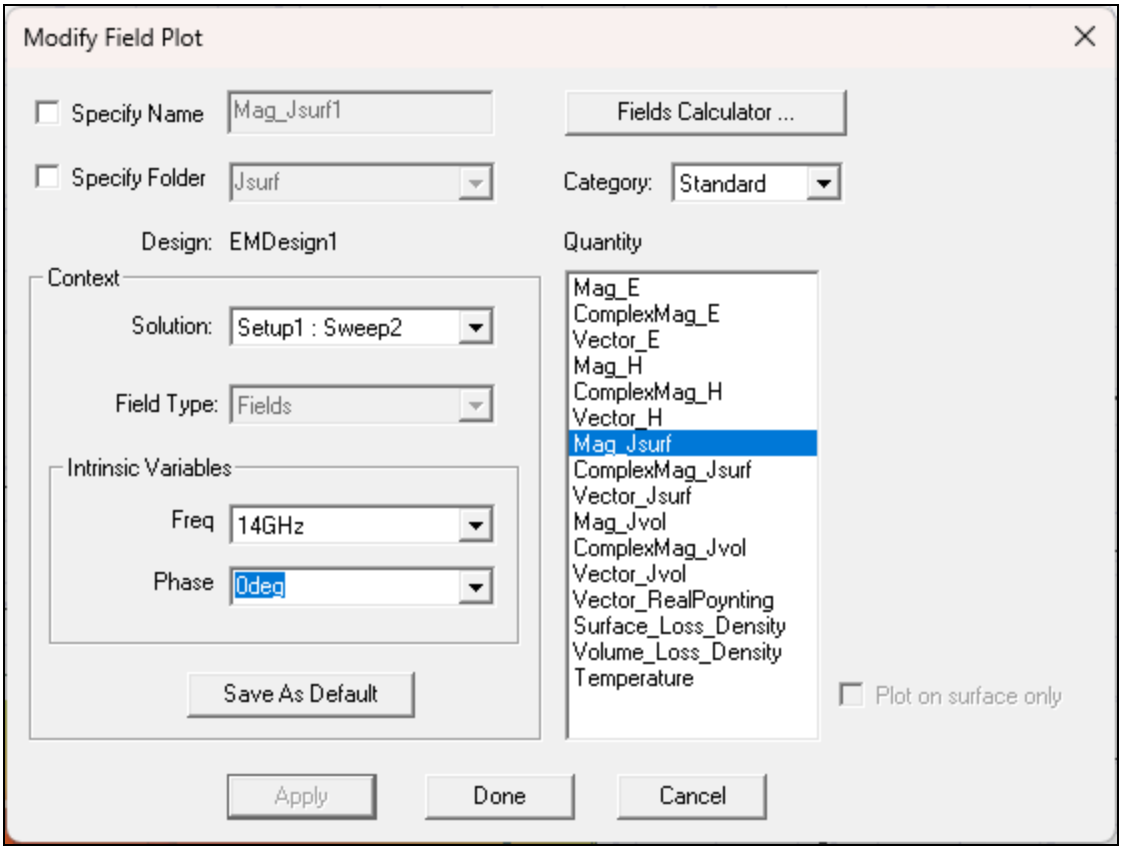
7. **Close** the plot settings window.
8. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.



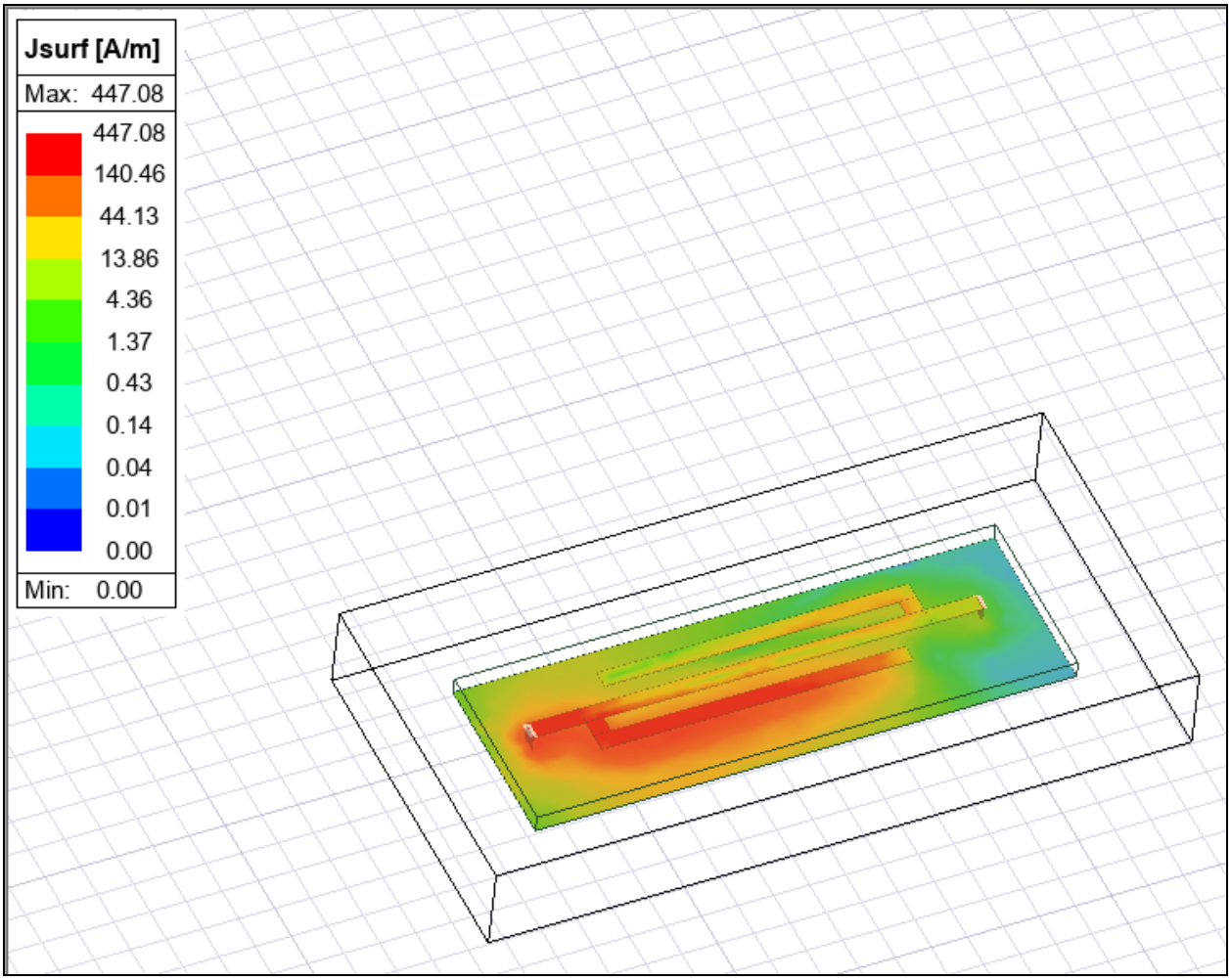
9. From the **Project Manager** window, expand **Jsurf**. Then right-click **Mag_Jsurf1** and select **Modify Plot** to open the **Modify Field Plot** window.



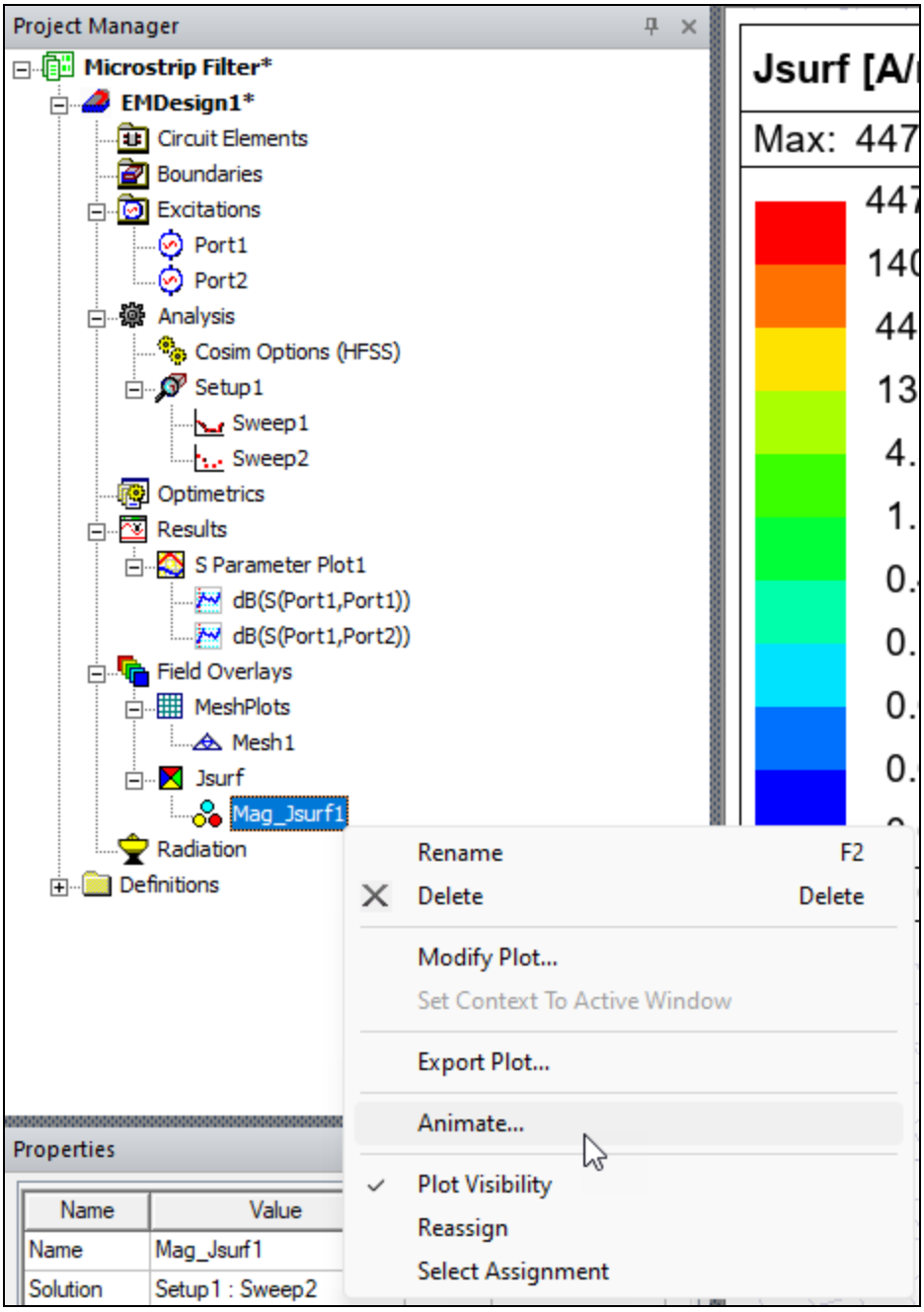
10. From the **Modify Field Plot**, do the following:
    - Select **14GHz** from the **Freq** drop-down menu in the **Intrinsic Variables** area.
    - Click **Apply**.



11. Click **Done** to close the window and return to the **Layout Editor**.



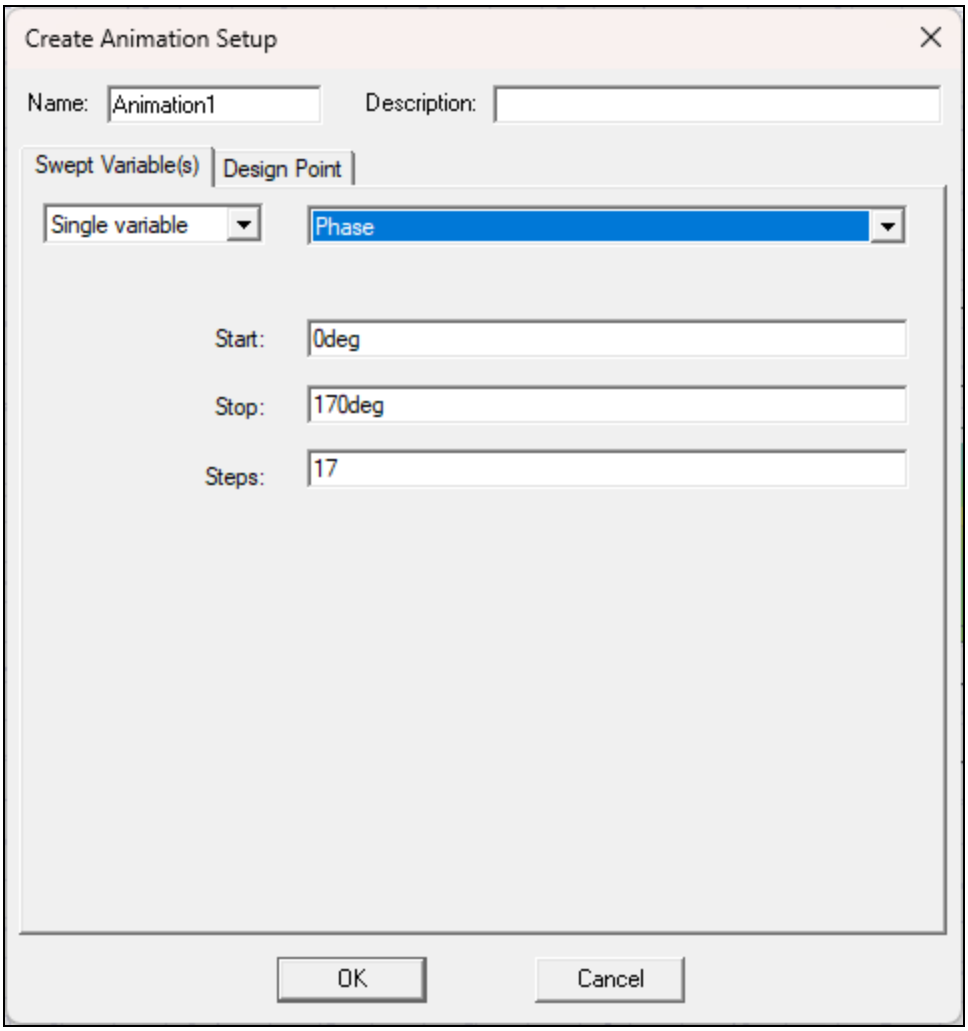
12. From the **Project Manager** window, right-click **Mag_Jsurf1** and select **Animate** to open the **Create Animation Setup** window.



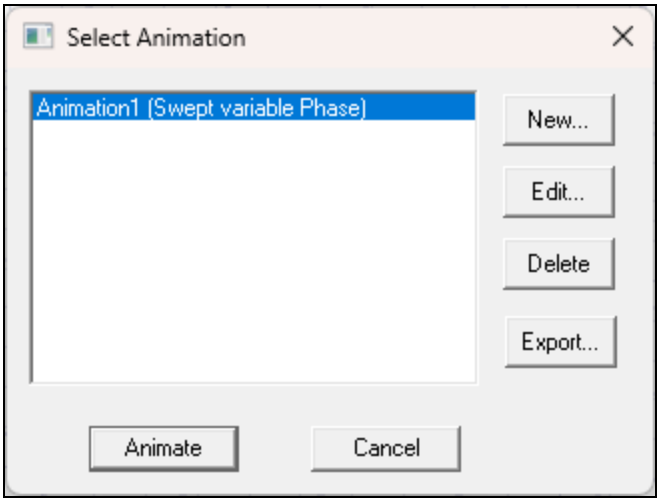
13. From the **Create Animation Setup** window, ensure the following settings were selected by default:

    - **Single variable** and **Phase** are selected from the drop-down menus.
    - **0deg** is entered in the **Start** field.
    - **170deg** is entered in the **Stop** field.

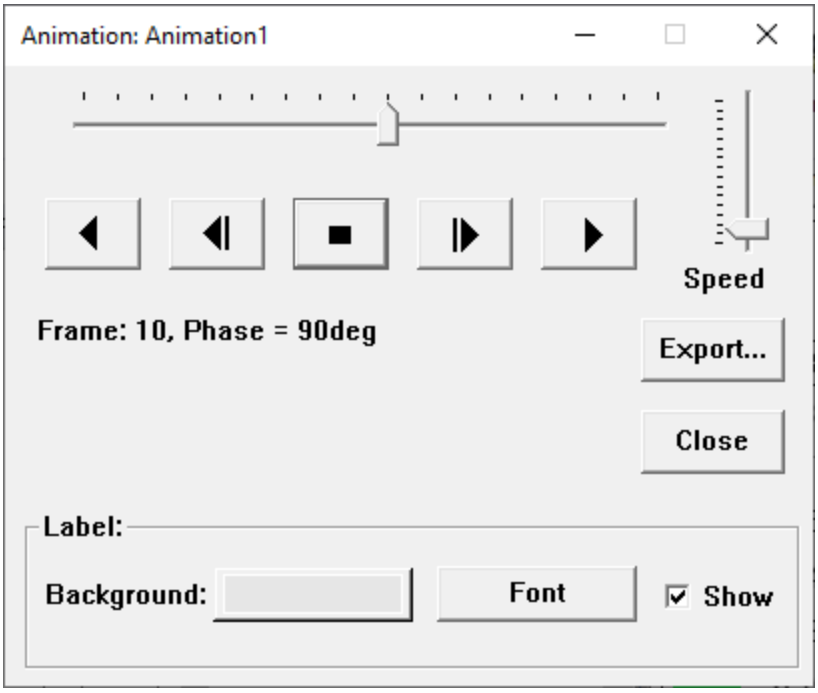
- **17** is entered in the **Steps** field.



14. Click **OK** to close the **Create Animation Setup** window and open the **Selection Animation** window. The only available option (i.e., **Animation1**) is selected by default.

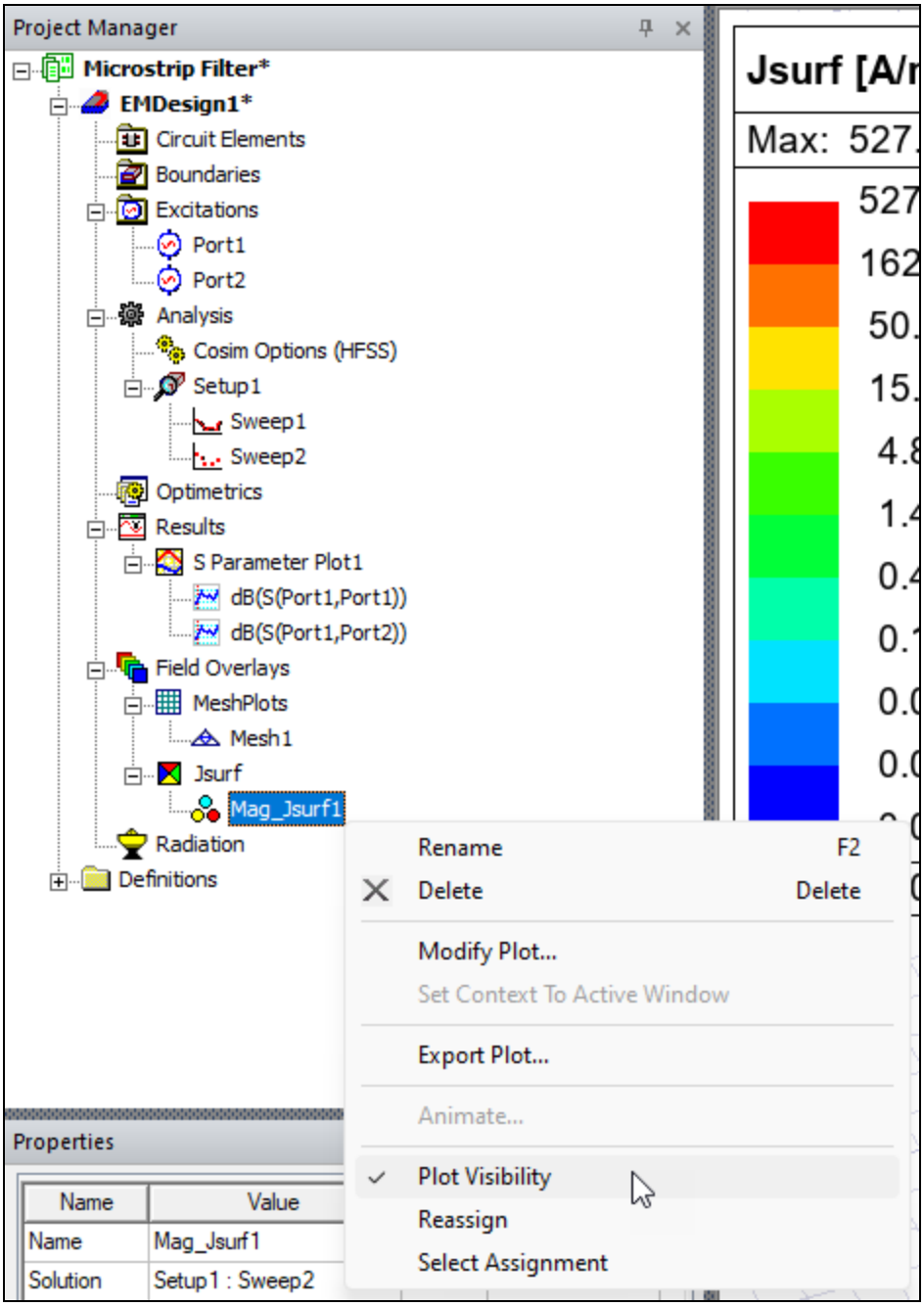


15. Click **Animate** to close the **Select Animation** window, open an animation control panel, and start the animation in the **Layout Editor**.



> **Note:**
>
> The current at Port 2 remains relatively low throughout the animation. This is expected.

16. Use the animation controls to reverse, stop, and change the speed of the animation, among other settings.
17. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.
18. From the animation control panel, click **Close** to end the animation.
19. Before continuing, navigate to the **Project Manager** window. Then right-click **Mag_ Jsurf1** and select **Plot Visibility** to remove the adjacent check mark and hide the overlay.



Continue to Creating and Animating an E Field Overlay.