Getting Started with HFSS™ 3D Layout: Low Pass 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

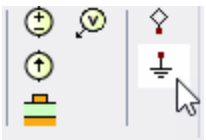
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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 (>). For example, "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: Low Pass Filter** guide to create a low pass filter in HFSS 3D Layout, then analyze the design. The low pass filter model consists of three layers (i.e., a ground layer, a dielectric layer, and a signal layer). The user will need to define the layers, assign a custom material to the dielectric layer, draw the model, define the ports, and set up the solution. After solving the model, the user will review the S matrix results, plot the return loss, create a Smith chart, a current density overlay, a far field display, and animate the results.

> **Note:**
>
> For more information refer to the HFSS 3D Layout Simulator in the main help.

The following topics cover the physical design and EM analysis of a low-pass filter, specifically the following subtopics:

- How to start the software and explore the HFSS 3D Layout tools
- How to use the ribbon, menu, and shortcut menu
- Terms and concepts essential to the simulation of an HFSS 3D Layout design
- How to add a custom-defined dielectric material to a design
- How to create a report to display simulation results

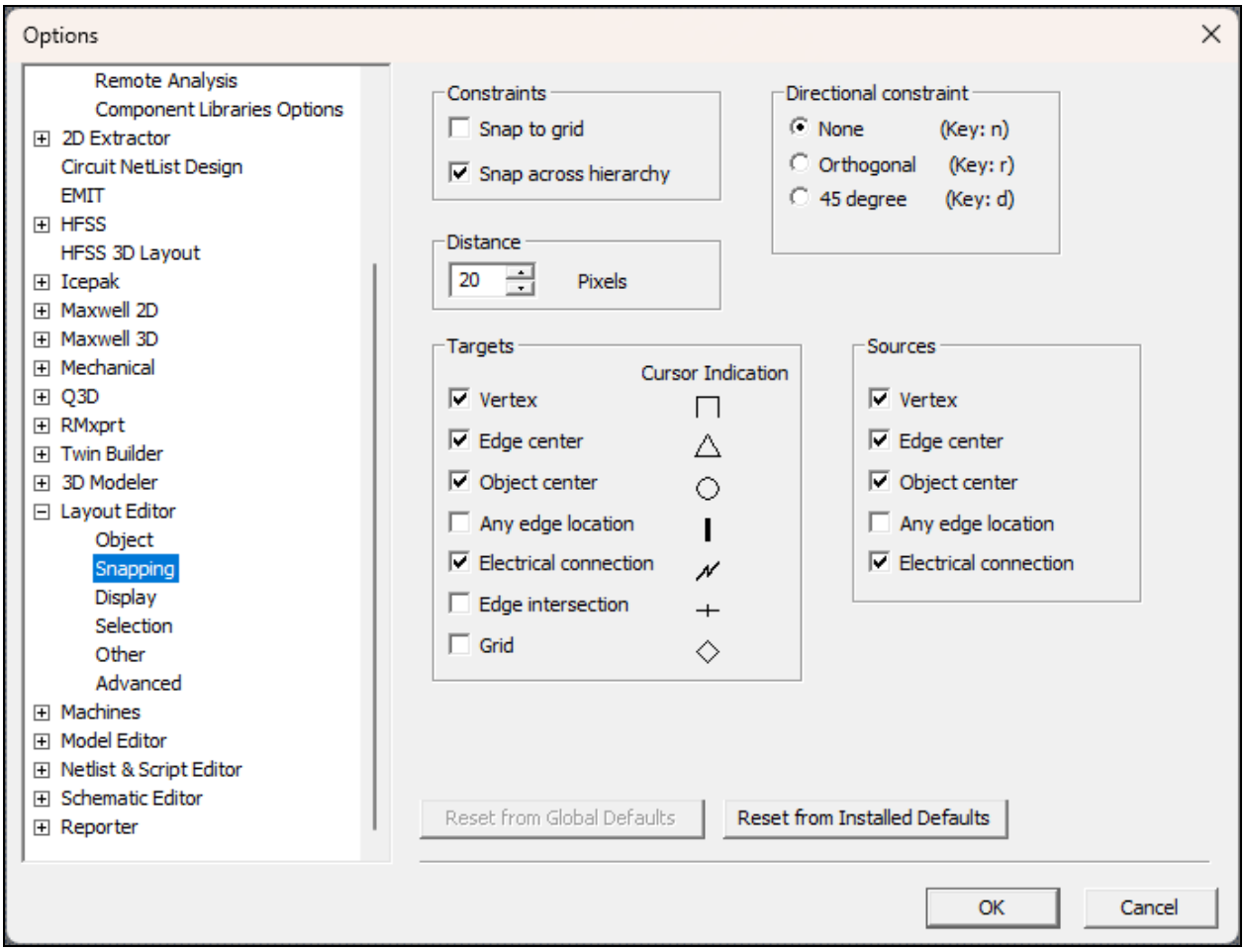
> **Note:**
>
> In many cases, there are multiple ways to access a tool or reach a particular window. To keep this guide concise and easy to follow, only one method is described for each step. If an alternative method offers a specific advantage or is better suited to a scenario, it will be noted accordingly.

Setting General Options

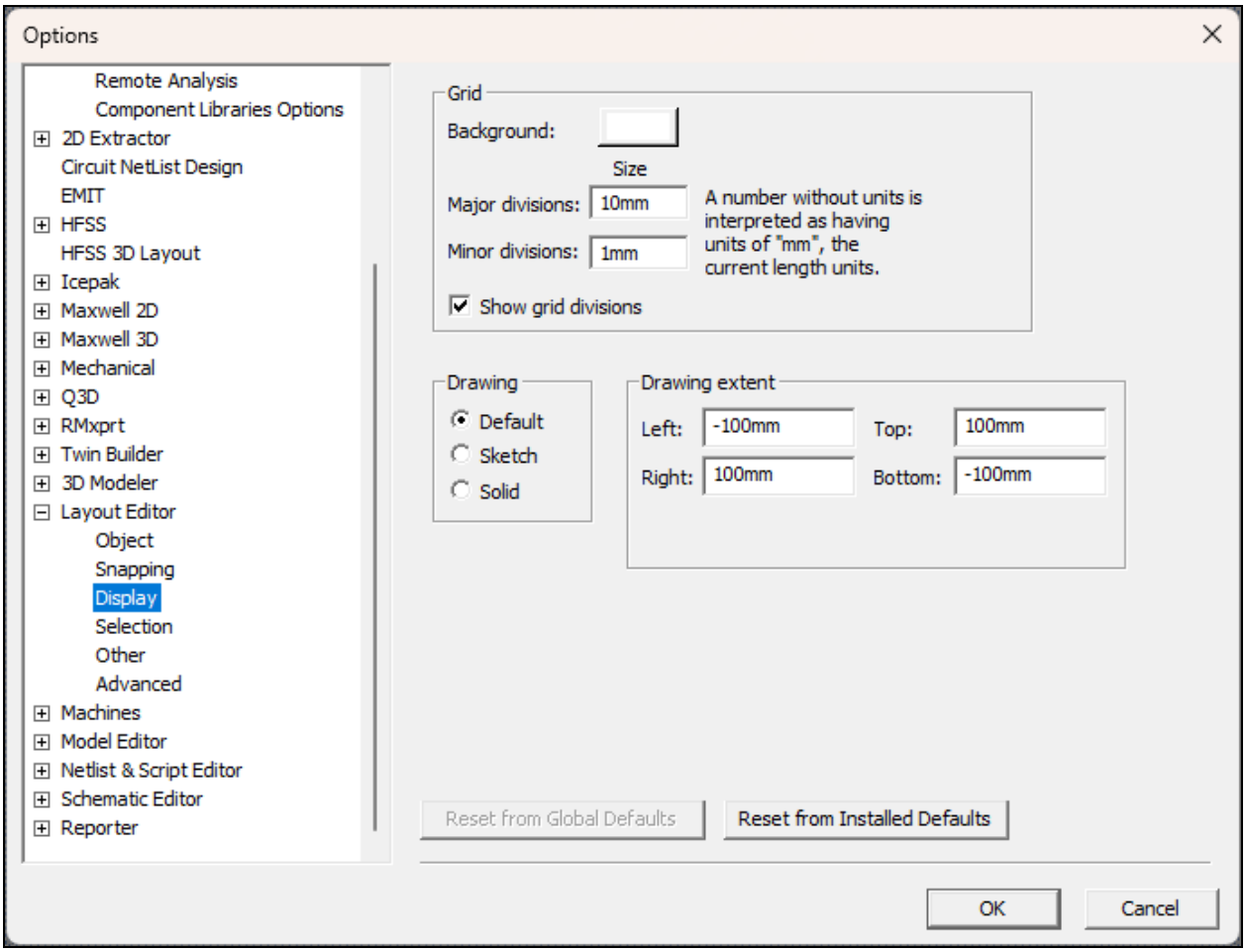
Before inserting a design into the project, follow these steps to ensure the **Electronics Desktop** options are set correctly for this exercise.

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 **mm** from the **Length** drop-down menu to use millimeters as the default length unit.
4. From the **Options** tree, select the **Layout Editor** > **Snapping** subgroup.

5. From the **Targets** area, clear the **Grid** option and ensure that your other selected options match the following settings:



6. From the **Options** tree, select the **Layout Editor** > **Display** subgroup.
7. In the **Grid** area, enter **10 mm** in the **Major** field and **1 mm** in the **Minor** field, making sure the unit used for each is millimeter (i.e., mm), the default setting.



8. Click **OK** to close the **Options** window.

Create the Model

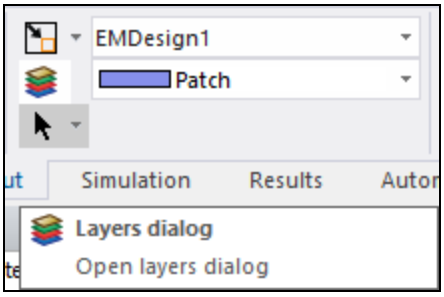
From here, continue to the following topics to define the stackup layers (i.e., topology) of the model, draw the geometry of the filter, and assign the excitation ports.

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

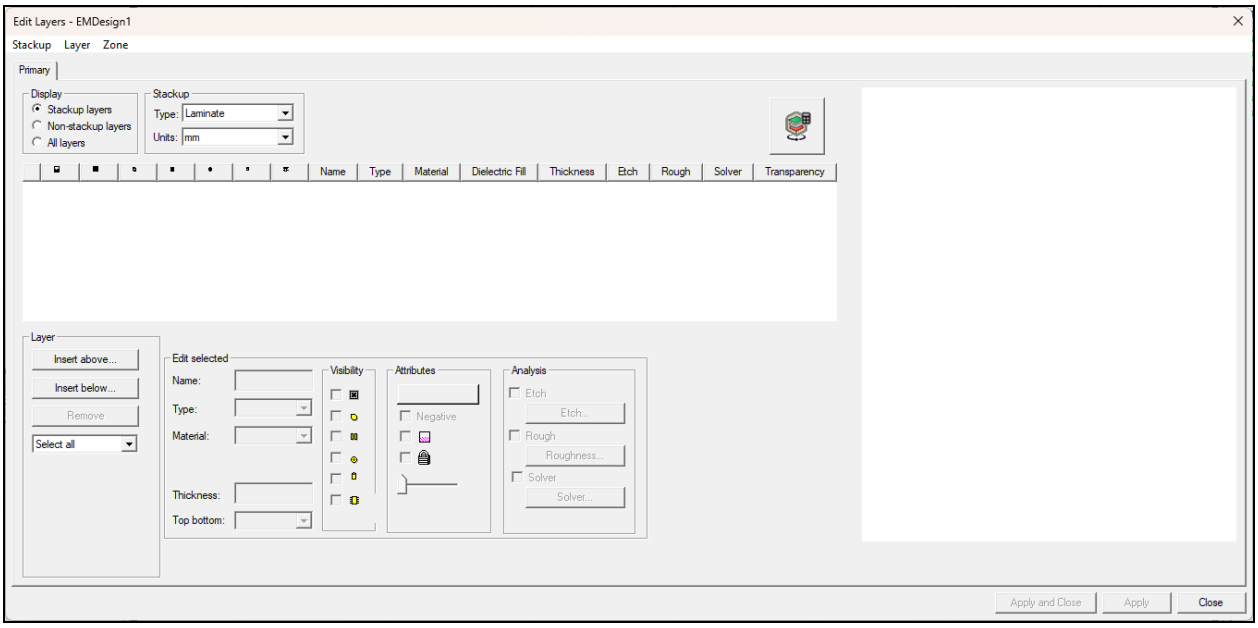
Insert Layers

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.



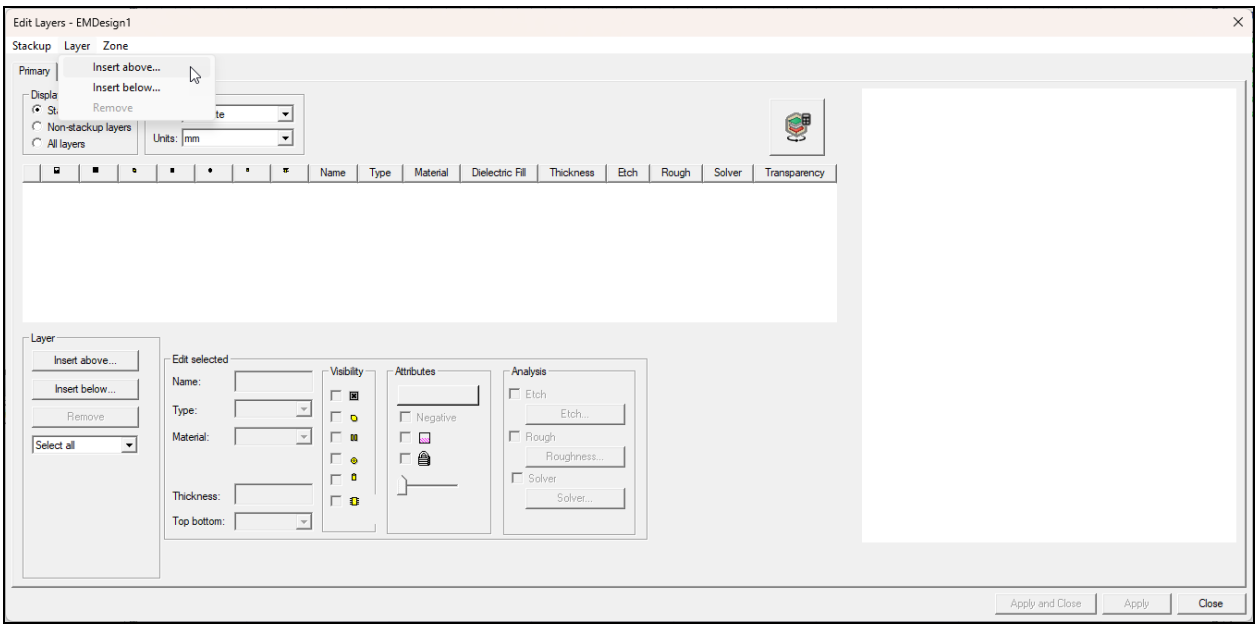
2. Ensure **Laminate** is selected from the **Stackup** area > **Type** drop-down menu.



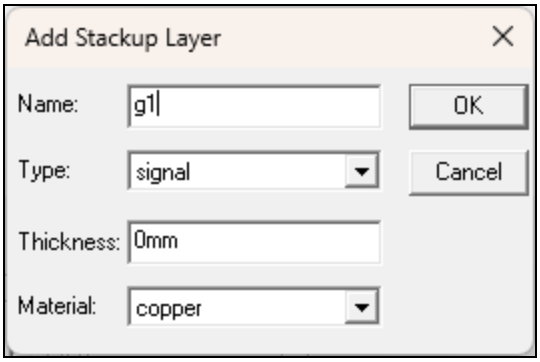
Add a Ground Layer to the Layer Table

> **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 in the table, the **Insert above** and **Insert below** options will be inactive until a layer is selected from the table. After selecting a layer from the table, select **Layer** > ***where to add the new layer*** (i.e., **Insert above** or **below** the highlighted layer).

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 **g1** in the **Name** field.
   b. Select **signal** from the **Type** drop-down menu.
   c. Click **OK** to close the **Add Stackup Layer** window add the new infinite ground layer to the layer table.

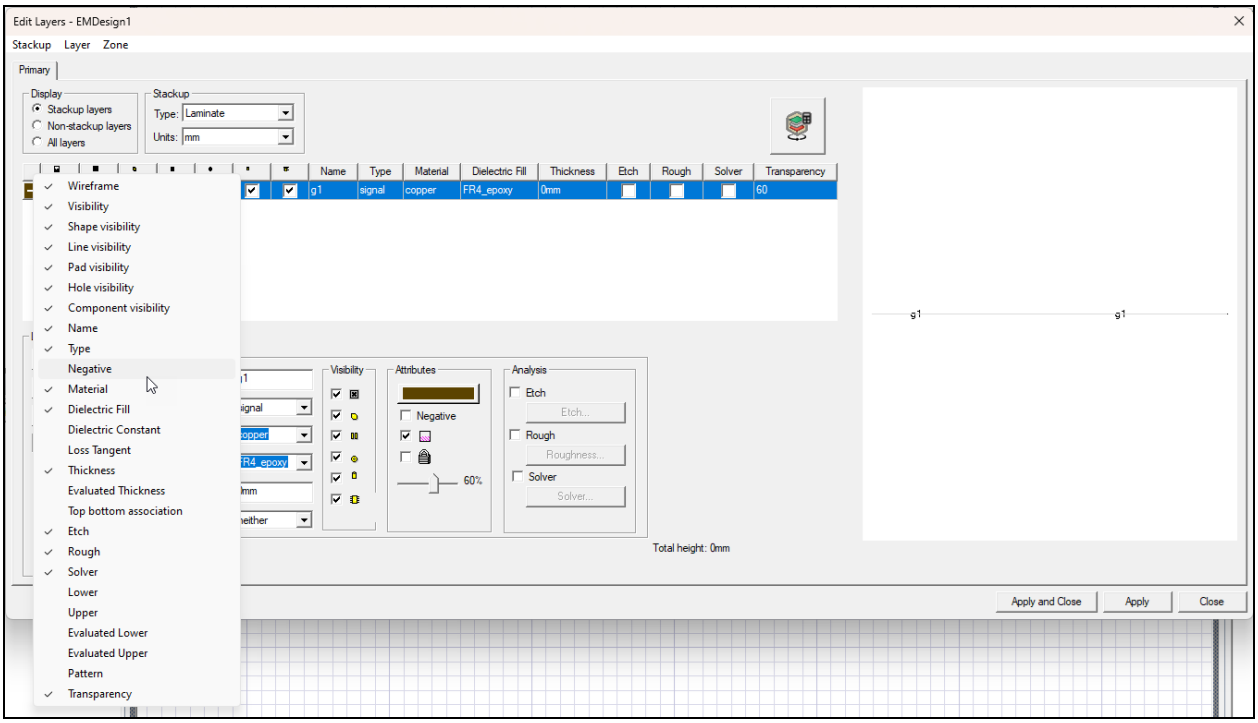


> **Note:**
>
> The material *copper* is automatically assigned to signal layers.

3. Ensure the box in the **Rough** column is not checked.

> **Note:**
>
> If the **Rough** box is checked, the surface roughness of the conductors is taken into account when approximating the impedance of the signal traces. However, surface roughness is ignored for ground layers. Removing the check from the box prevents a warning to that effect during validation and solution setup.

4. If applicable, add the **Negative** column to the table. Right-click any column header (e.g., **Name**, **Type**, **Material**, et cetera) to open the shortcut menu. Then select **Negative**. The **Negative** column will appear in the table.

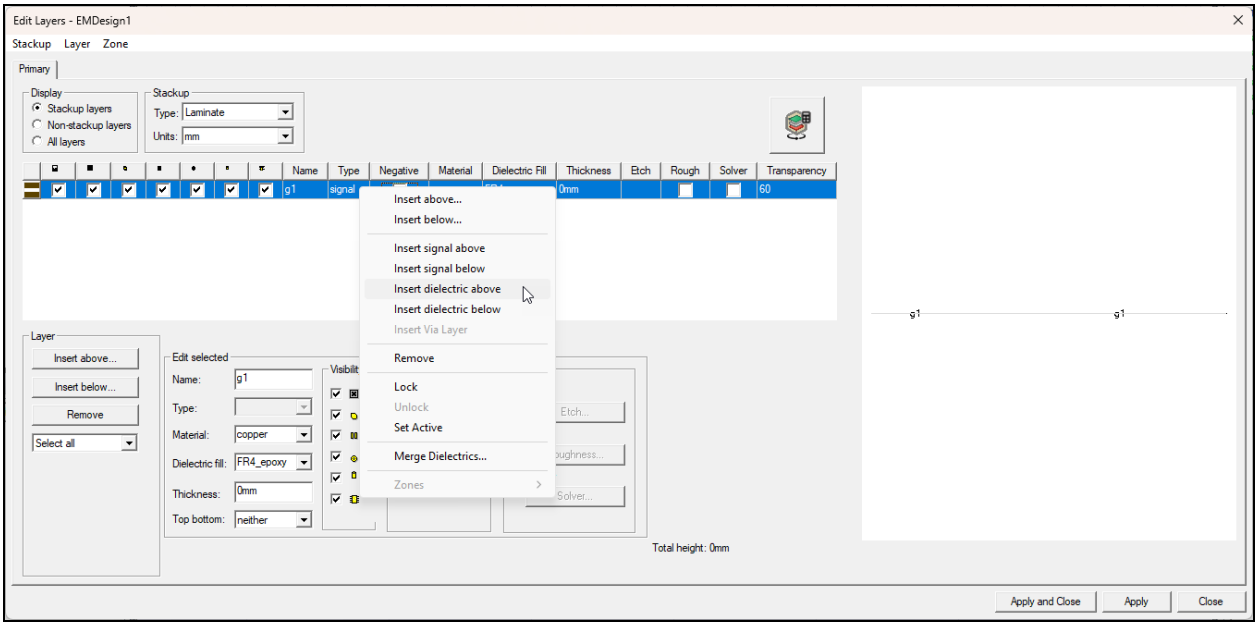


5. Check the box in the **Negative** column.

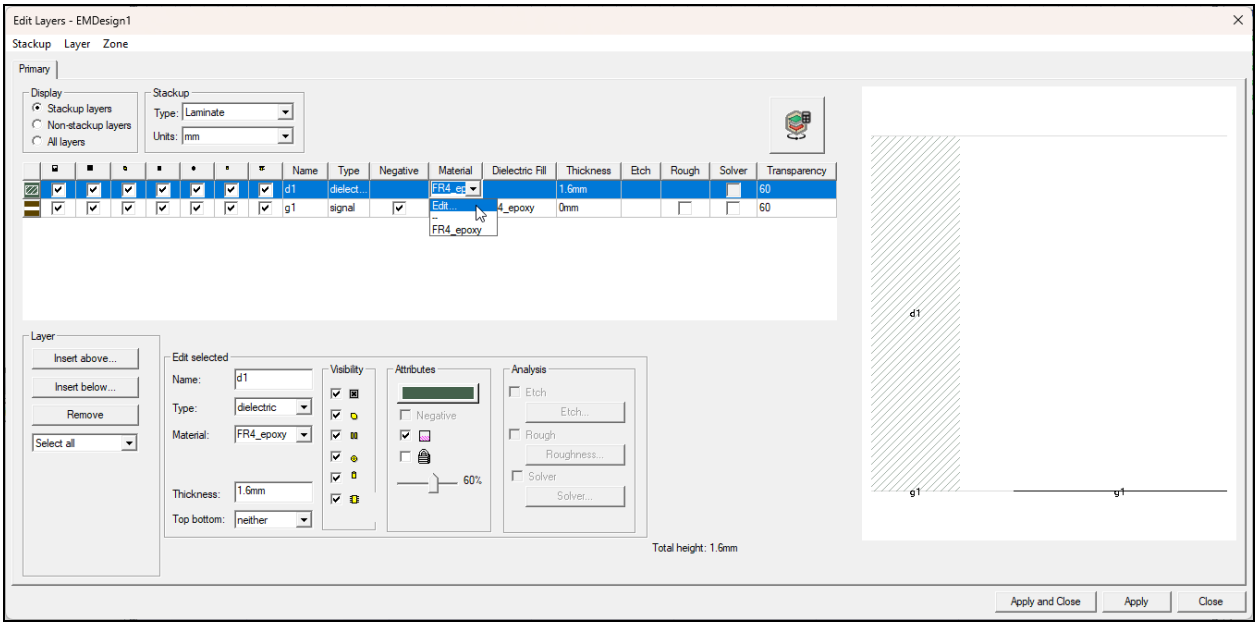
> **Note:**
>
> Checking the box in the **Negative** column identifies the layer as a ground plane layer. Any object drawn on a negative layer becomes a cutout in the ground layer (conductor removed). However, no objects can be drawn on the **g1** layer for this model.

Add a Dielectric Layer to the Layer Table

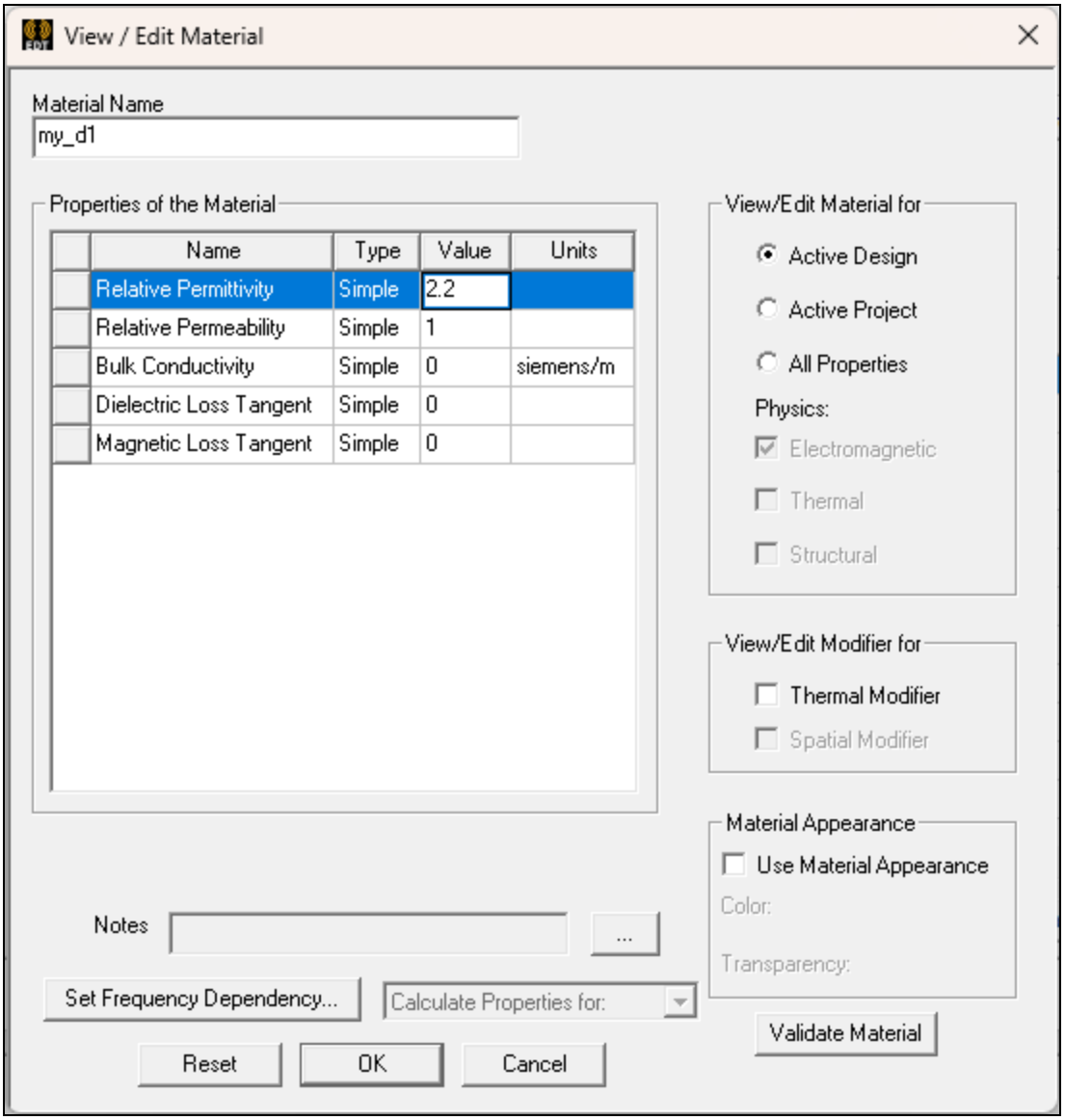
6. Right-click anywhere in the **g1** layer and select **Insert dielectric above**. A new row appears in the layer table (default **Name**, **Dielectric**).



7. In the new **dielectric** row, do the following:
   a. In the **Name** field, replace **dielectric** with **d1**.
   b. Ensure **1.6mm** is entered in the **Thickness** field.
   c. Select **Edit** from the **Material** drop-down menu to open the **Select Definition** window.



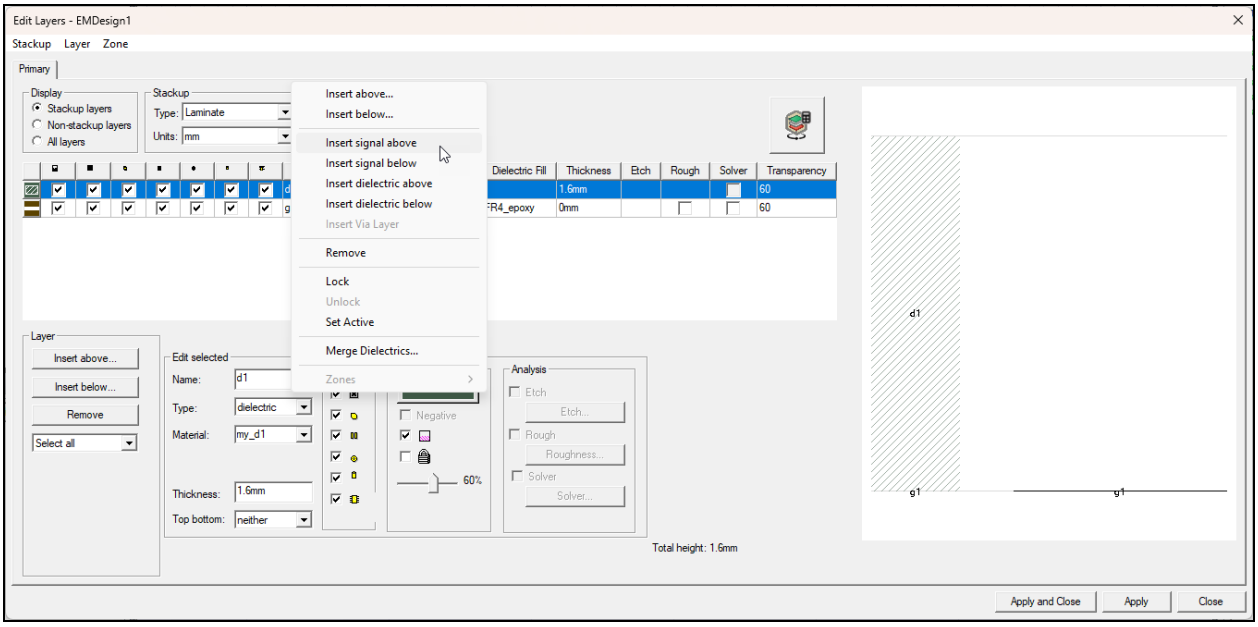
8. From the **Select Definition** window, do the following:
   a. Click **Add Material** to open the **View / Edit Material** window.
   b. In the **Material Name** field, replace **Material1** with **my_d1**.
   c. In the **Relative Permittivity Value** field, replace **1** with **2.2**.
   d. Click **OK** to save changes, close the **View / Edit Material** window, and return to the **Select Definition** window.



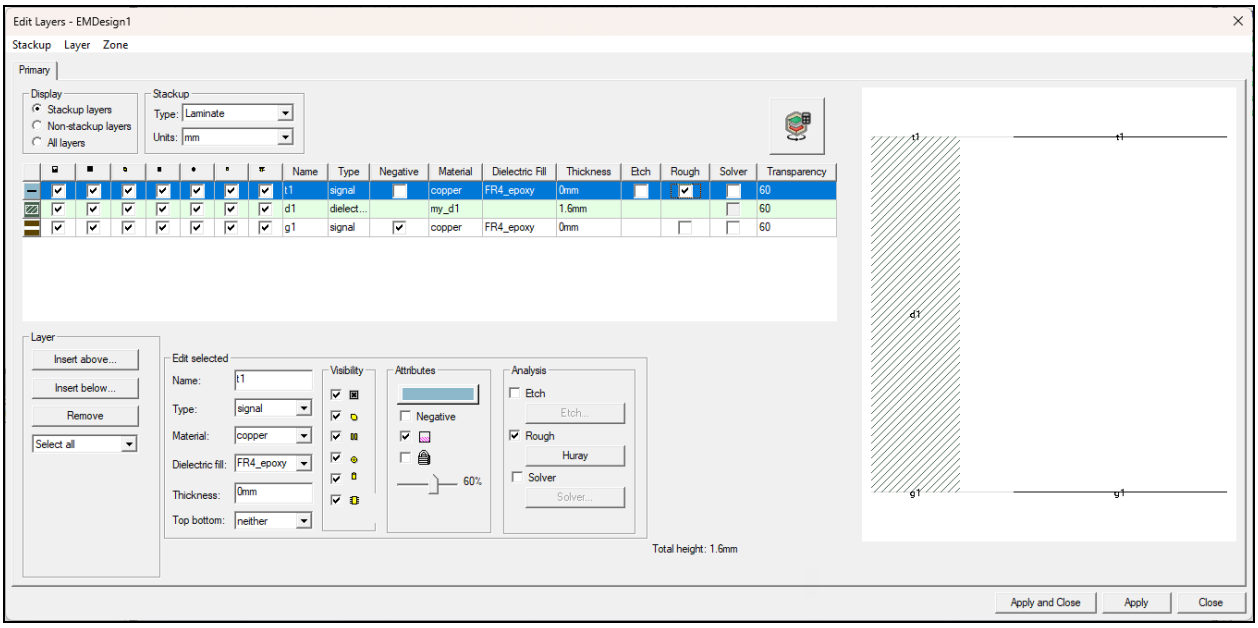
e. Click **OK** to close the **Select Definition** window.

Add a Trace (Signal) Layer to the Layer Table

1. Right-click the **d1** layer and select **Insert signal above**. A new row appears in the table.

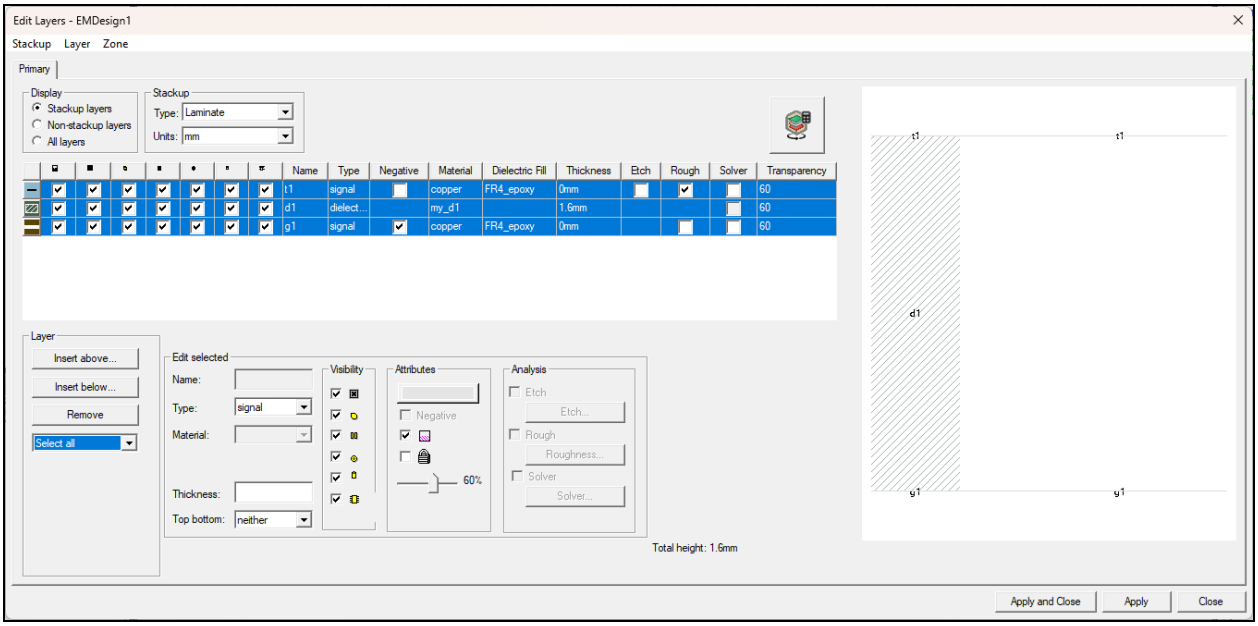


2. In the new **signal** row, do the following:
    a. In the **Name** field, replace **Signal** with **t1**.
    b. Check the box in the **Rough** column.

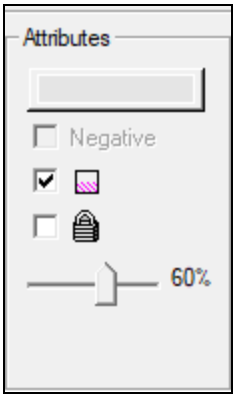


Make Changes to All Layers

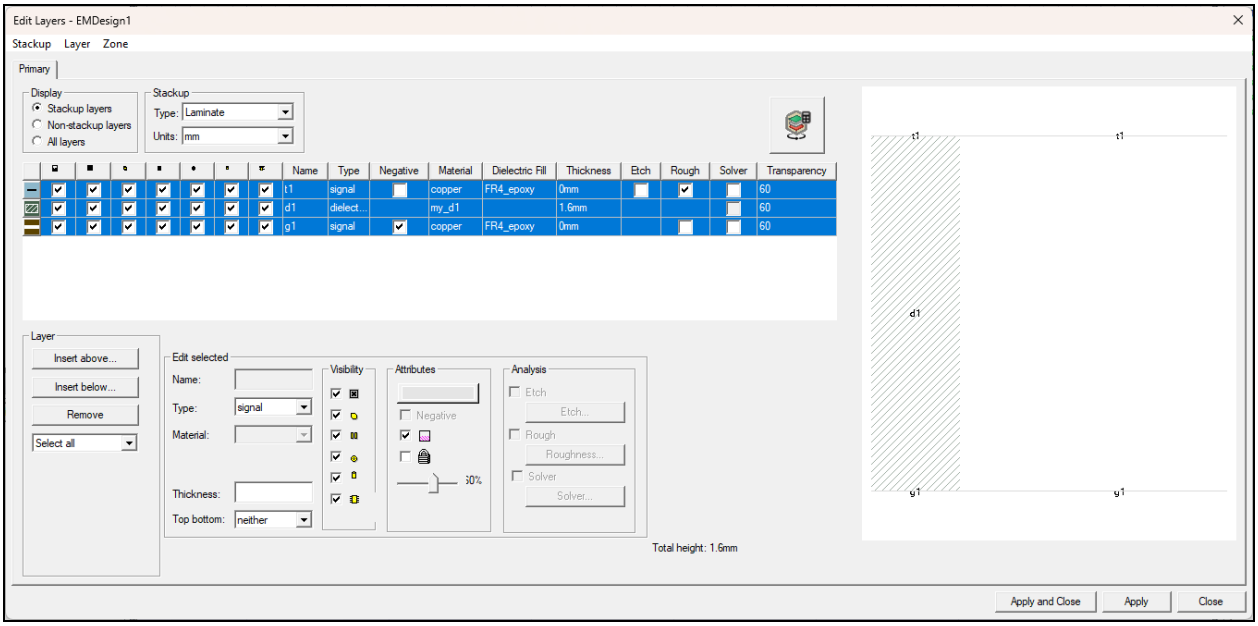
1. From the **Layer** area, re-select **Select All** from the drop-down menu to highlight all three rows. Ensure the **Solver** check box is **not** checked for all layers.



2. 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., wire frame).



3. 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 on the bottom.

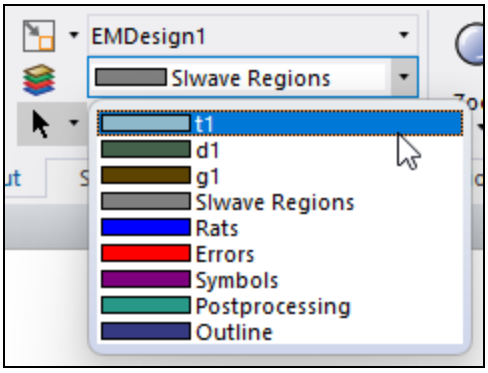
4. Click **Apply and Close** to apply the layer definitions and close the **Edit Layers** window.

Continue to Draw the Model.

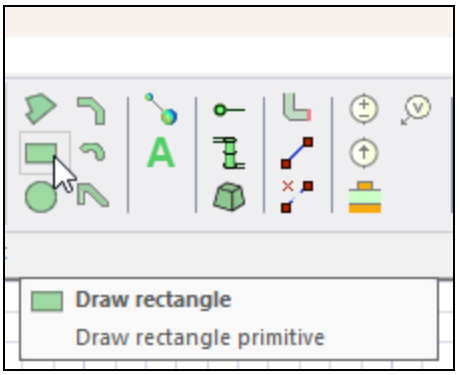
Draw the Model

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

1. From the **Layout** tab, select **t1** from the **Active Layer** drop-down menu.

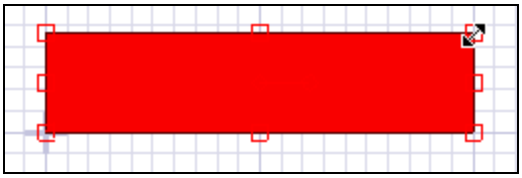


2. From the **Layout** tab, click **Draw rectangle**.



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 **20** in the field.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.6** in the field.

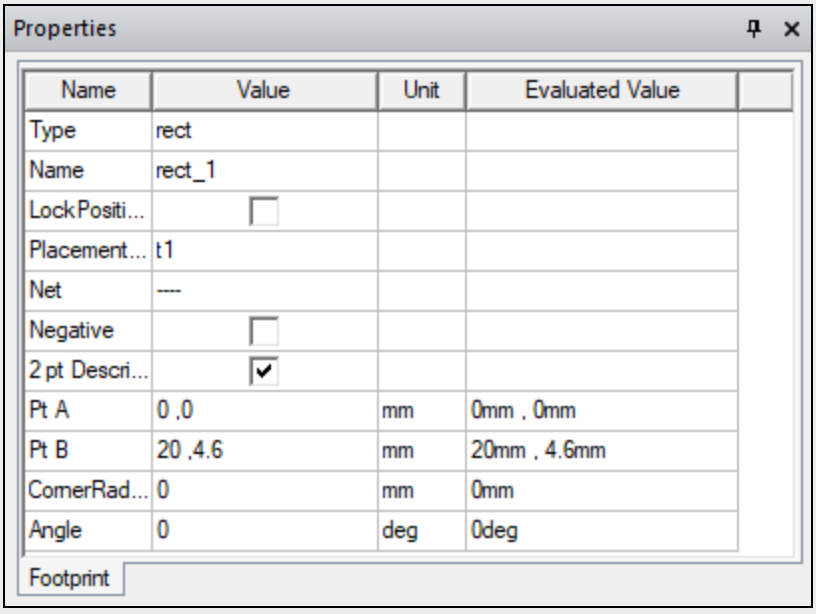
7. Press **Enter** to complete the shape.



Note:

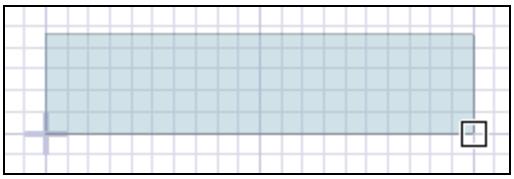
Alternatively, create a rectangle to the exact dimensions required by first creating a rectangle of any size, in any location, and then modifying its perimeters in the **Properties** window. Coordinates entered via the coordinate fields specify opposite corners of a rectangle. Alternatively, entering 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. For example, to create the rectangle described previously, do the following:

a. From the **Layout** tab, click **Draw rectangle**.
b. In the **Layout Editor**, **click+drag** to draw a rectangle.
c. Select the newly-drawn rectangle to display its perimeters in the **Properties** window.
d. From the **Properties** window, do the following:
    i. Ensure the **2 pt Description** box is checked.
    ii. Enter **0, 0** in the **Pt A** field.
    iii. Enter **20, 4.6** in the **Pt B** field.
    iv. Press **Enter** to save changes.

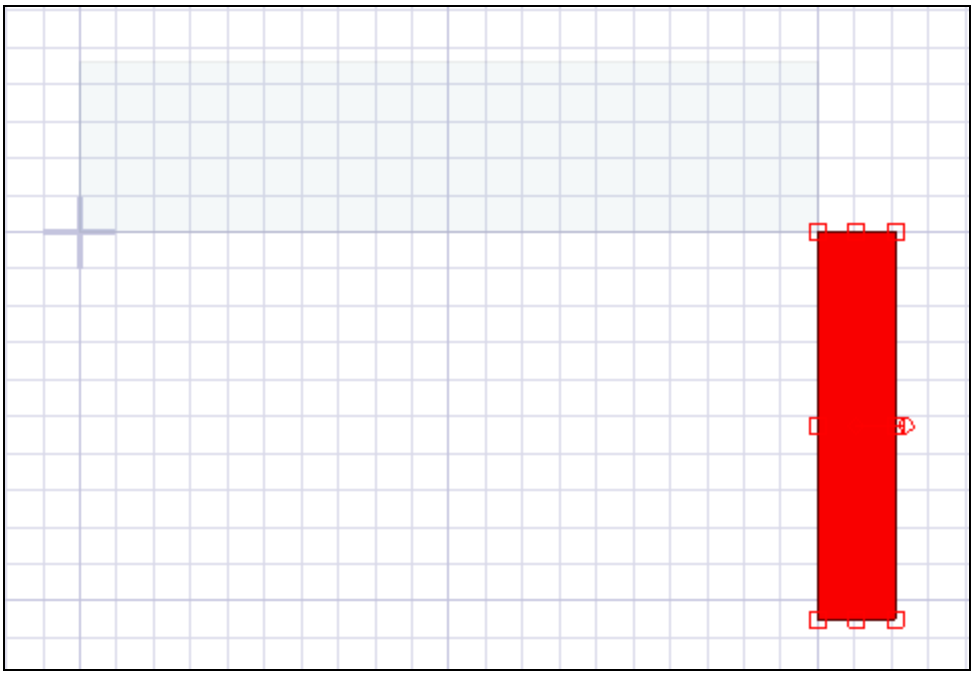


e.

8. Create a second rectangle by doing the following:
    a. From the **Layout** tab, click **Draw rectangle**.
    b. Hover over the lower-right corner of the first rectangle until the cursor turns into a square.

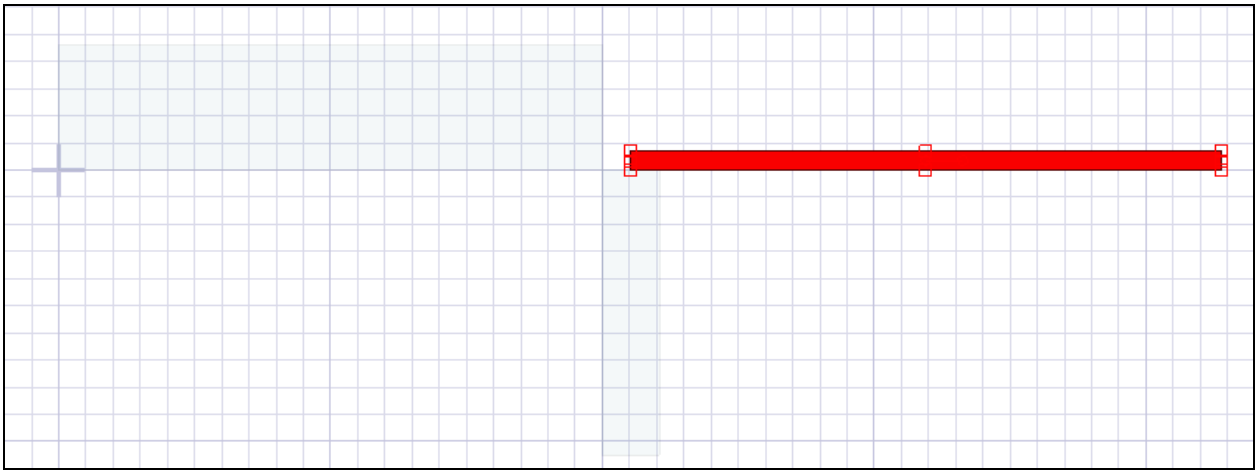


    c. Click to snap the first corner of the second rectangle to that point. This effectively designates the **X**, **Y** coordinates for the second rectangle **20**, **0**.
    d. Click inside the **Delta X** coordinate field, delete the coordinates already present, and enter **2.1**.
    e. Press **Tab** to move the cursor to the **Delta Y** coordinate field, Then type **-10.5** in the field.
    f. Press **Enter** to complete the shape.

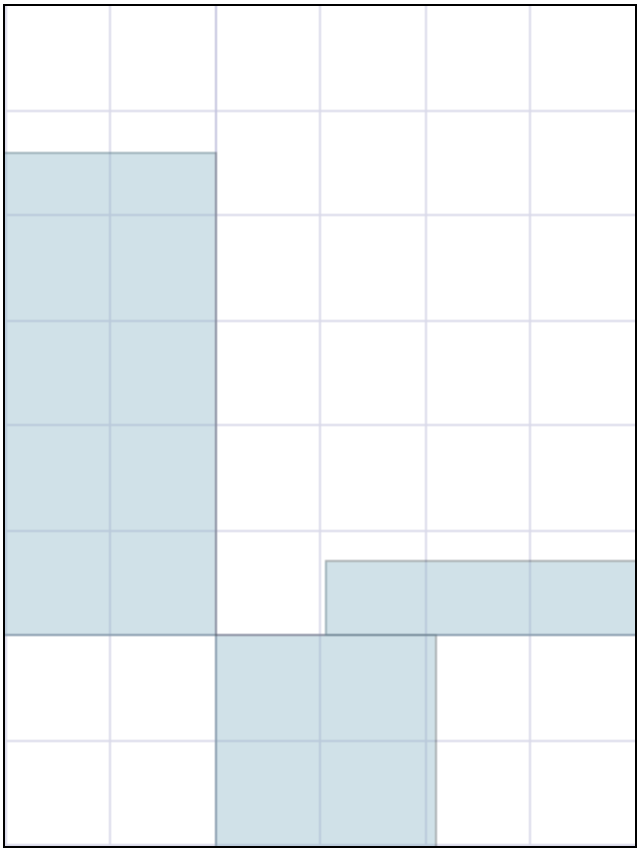


9. Create a third rectangle by doing the following:
    a. From the **Layout** tab, click **Draw rectangle**.
    b. Move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Click inside the field, delete the coordinates already present, and enter **21.05**.

c. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.
d. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **21.7** in the field.
e. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **0.7** in the field.
f. Press **Enter** to complete the shape.



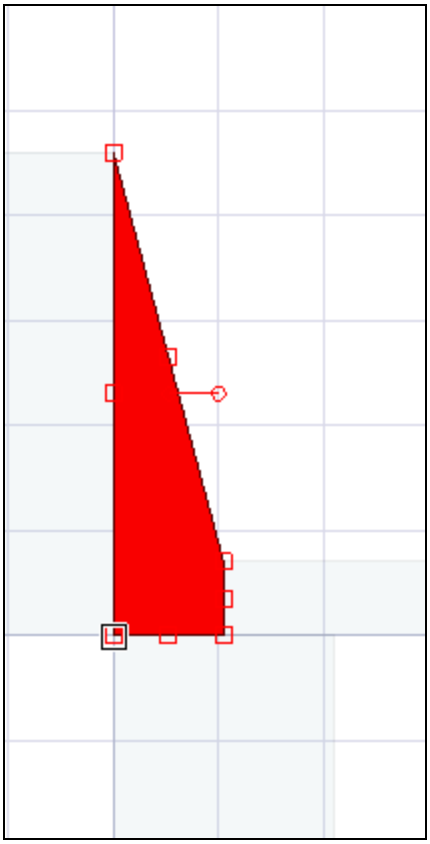
10. Click anywhere else in the **Layout Editor** to clear the current selection.
11. Spin the mouse wheel to **Zoom In**/**Out** on the space between the right edge of the first rectangle, the top edge of the second rectangle, and the left edge of the third rectangle.



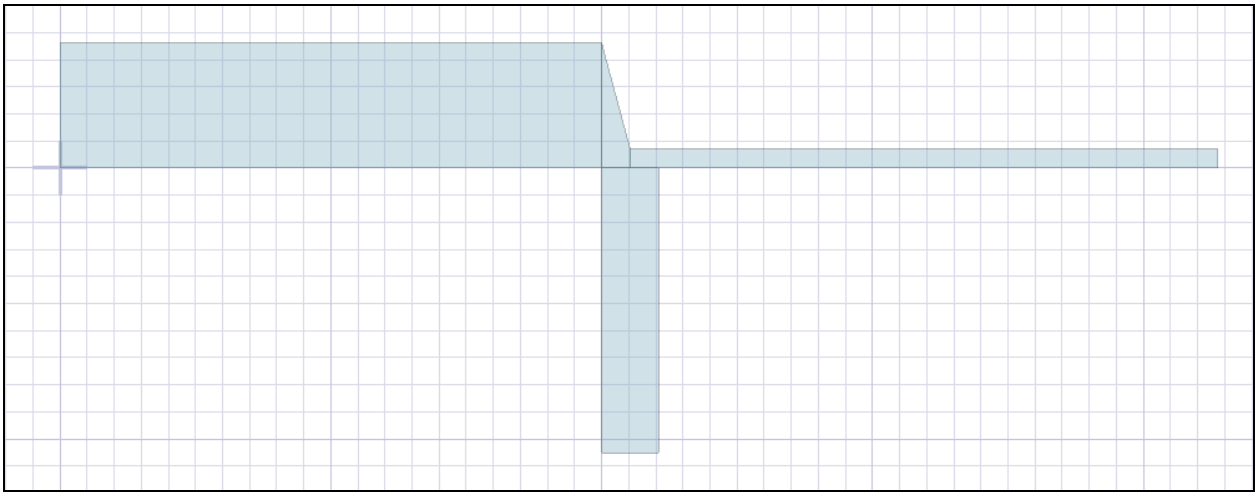
12. Create a polygon by doing the following:

> **Note:**
>
> During step c, the cursor will become a triangle, since the snapping point corresponds to the midpoint between the second rectangle's top edge and the end point of two of the third rectangle's edges.

    a. From the **Layout** tab, click **Draw polygon**.
    b. Hover over the upper-right corner of the first rectangle until the cursor turns into a square when the snap point is found. Then click (values displayed in the **X**, **Y** coordinate fields will be **20.0**, **4.6**).
    c. Click the upper-left corner of the third rectangle (values displayed in the **X**, **Y** coordinate fields will be **21.05**, **0.7**)
    d. Click the lower-left corner of the third rectangle (values displayed in the **X**, **Y** coordinate text fields will be **21.05**, **0.0**)
    e. Double-click the lower-right corner of the first rectangle (values displayed in the **X**, **Y** coordinate text fields will be **20.0**, **0.0**)

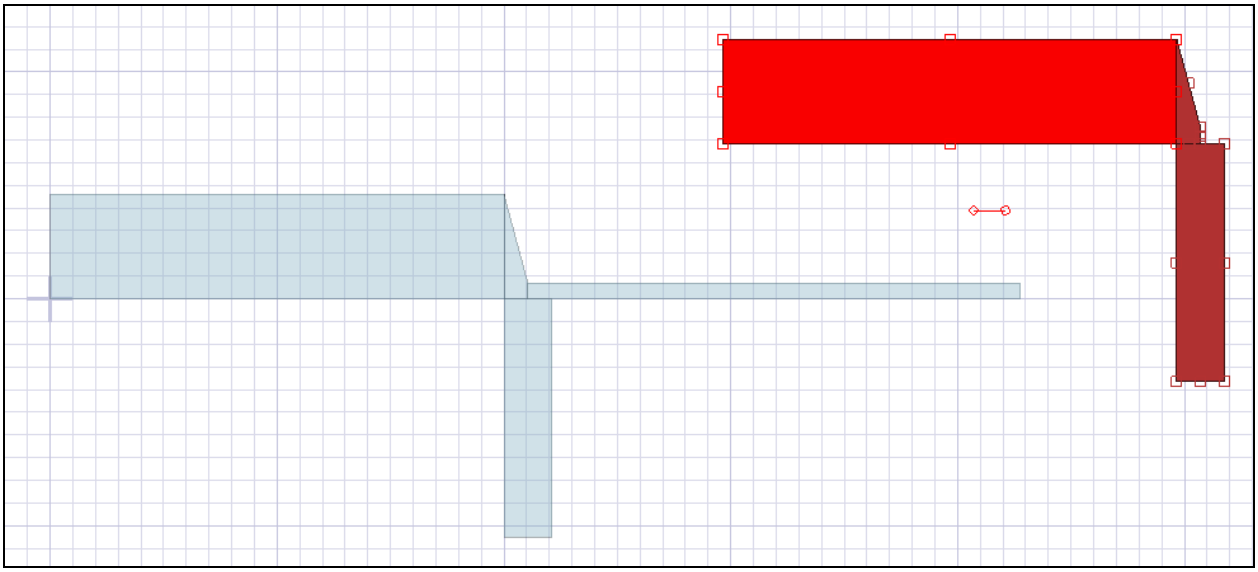


13. Press **Ctrl**+**D** to fit the drawing in the **Layout Editor** and clear the current selection.

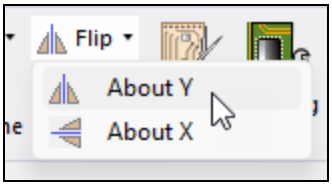


14. Create duplicate copies of the first two rectangles and the new polygon by doing the following:

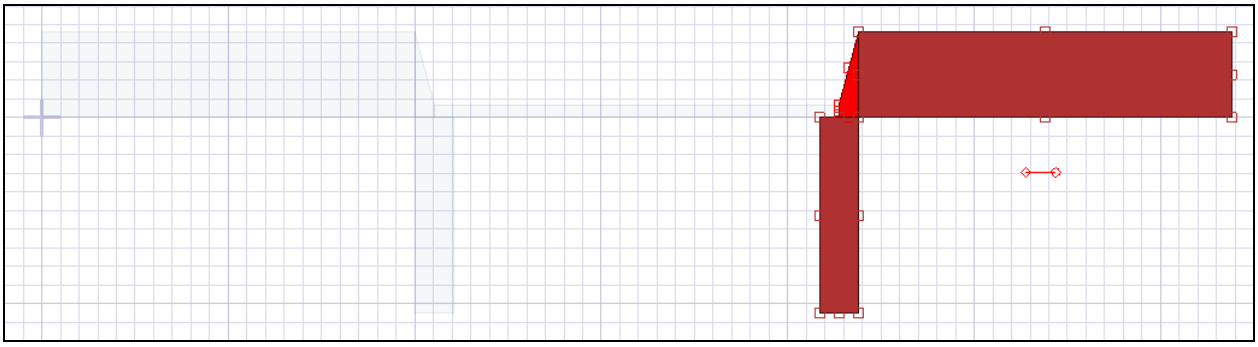
a. Press **Ctrl**+**A** to select all the objects.
b. While holding down **Ctrl**, click the third rectangle to deselect it. The first two rectangles and the polygon should still be selected.
c. Press **Ctrl**+**C** to copy the selected objects.
d. Press **Ctrl**+**V** to paste a duplicate set of objects into the **Layout Editor**. The location of the pasted objects moves as the mouse is moved.
e. Choose an area in the **Layout Editor** that does not overlap the original objects. Then click to drop the new objects. Ensure the new objects remain selected for the next step.



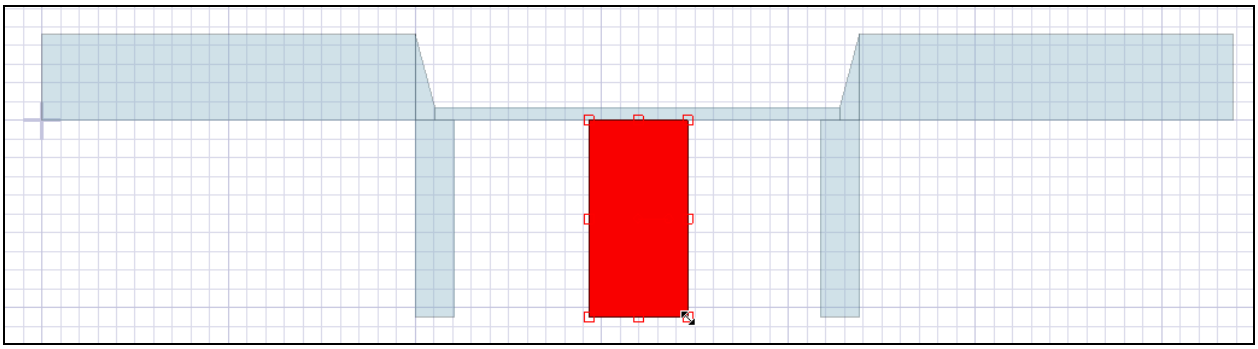
15. From the **Layout** tab, select **Flip** > **About Y** (or, from **Draw**, select **Flip Horizontal**).



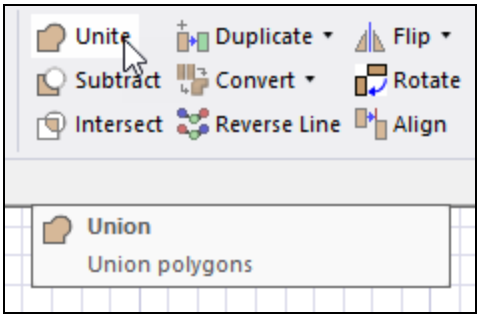
16. **Click**+**drag** the selected objects to align with the bottom right corner of the third rectangle. Once the cursor becomes a square, release-click and the objects will snap into place.



17. Press **Ctrl**+**D** to fit the drawing in the **Layout Editor** and clear the current selection.
18. Create a fourth and final rectangle by doing the following:
    a. From the **Layout** tab, click **Draw rectangle**.
    b. 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 **29.3**.
    c. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.
    d. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Then type **5.3** in the field.
    e. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **-10.5** in the field.
    f. Press **Enter** to complete the shape.



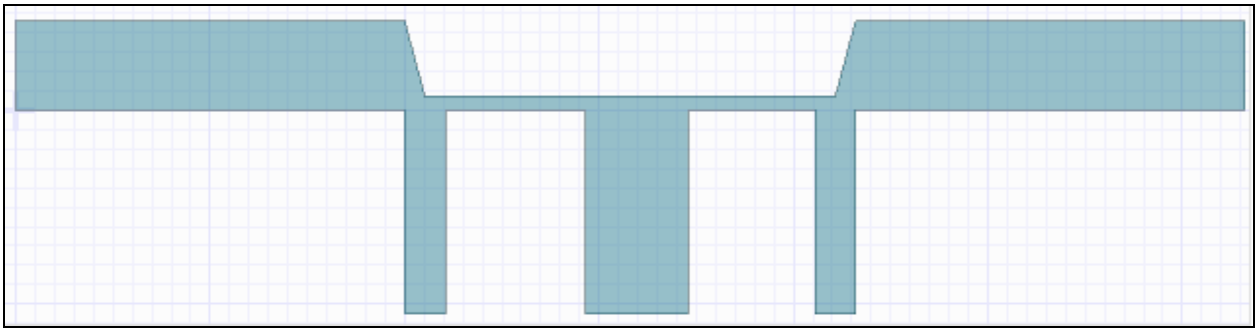
19. Press **Ctrl**+**A** to select all objects.
20. From the **Layout** tab, select **Unite** to form all the rectangles and polygons into a single object.



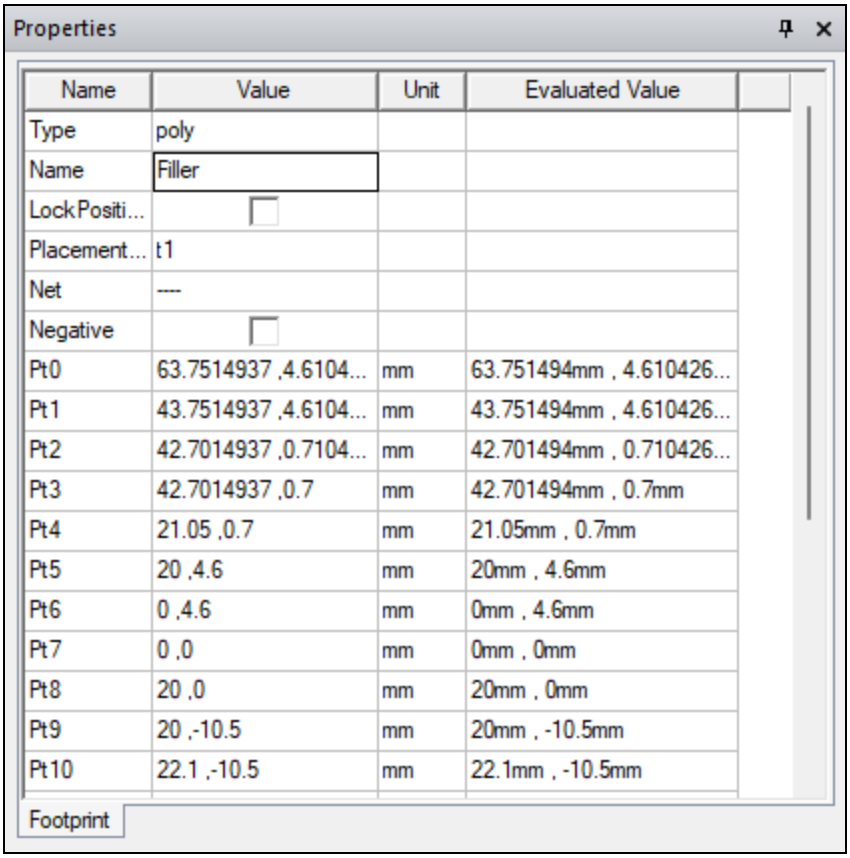
Note:

It is not mandatory to unite the individual shapes. Where they meet, the solver treats them as a contiguous object, regardless. However, uniting them produces a simpler model and eliminates the possibility of accidentally dragging one shape out of its proper position.

21. The design should match the following figure.



22. Click the object to select it. From the **Properties** window, replace the text in the **Name** field with **Filter**. Then press **Enter** to save changes.



23. **Save** the design, either by navigating to **File** > **Save** or clicking the **Save** button on any of the ribbons.

Continue to Assign the Ports.

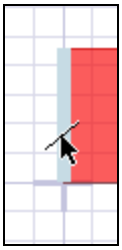
Create Edge Ports

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

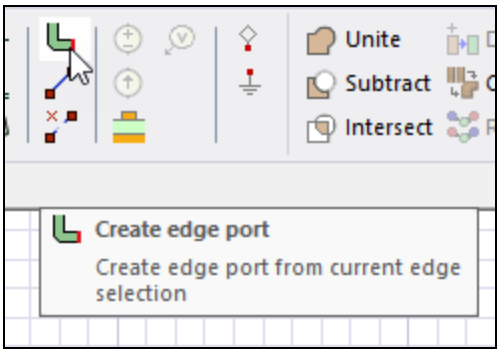
1. To create the first port (i.e., ***Port1***), press **E** to enter **Select Edges** mode.

> **Note:**
>
> Once **Select Edges** mode is active, the cursor changes: a diagonal line crosses the tip of the arrow. Refer to the following step.

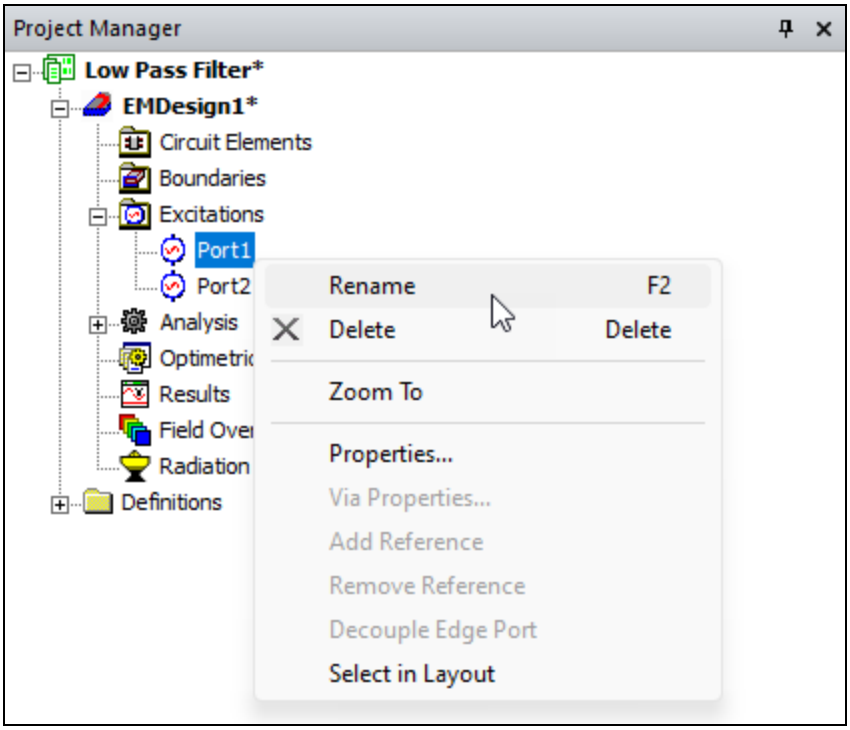
2. Click the left side of the leftmost rectangle to select it.



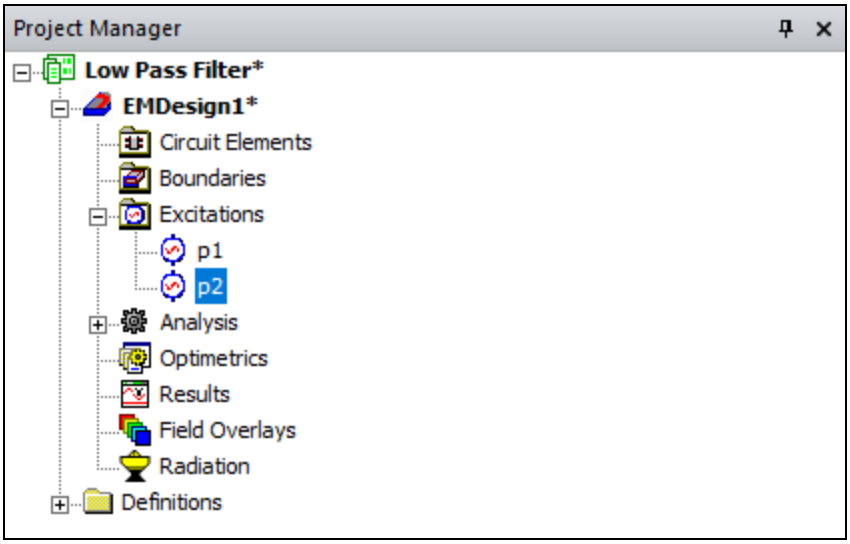
3. From the **Layout** tab, select **Create edge port**.



4. To create the second port (i.e., ***Port2***), press **E** to reenter **Select Edges** mode.
5. Click the right side of the rightmost rectangle to select it.
6. From the **Layout** tab, select **Create edge port**.
7. Once a port is created, it appears in the **Project Manager** > **Project Tree** > **[*active design folder*]** > **Excitations** folder. Right-click **Port1** and select **Rename**.



8. Rename **Port1** to **p1** and **Port2** to **p2**.



Continue to Set Up an Analysis and Frequency Sweeps.

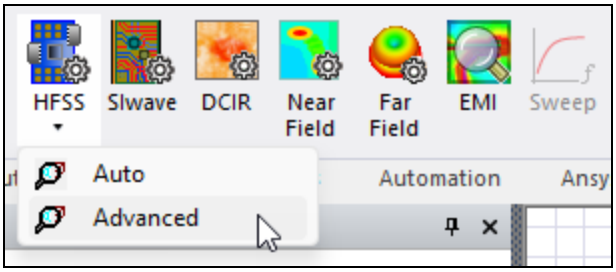
2 - Set Up Solution and Analyze

This chapter contains the following topics:

Set Up an Analysis and Frequency Sweeps

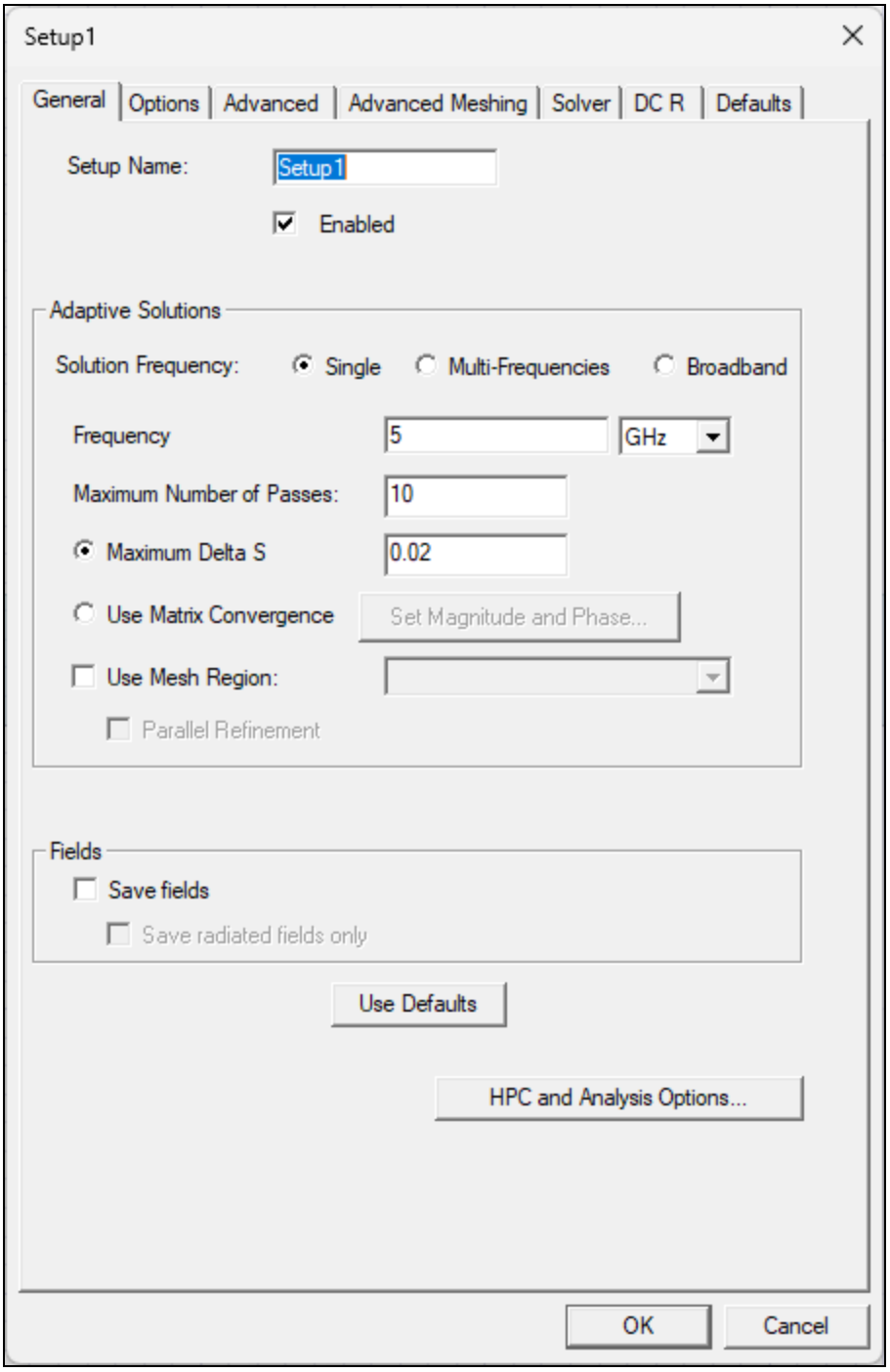
Solution setups are listed in the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder). Follow these steps to add a new solution setup to the project using basic, initial meshing tools. Then add, set up, and define an interpolating frequency sweep and a discrete frequency sweep.

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

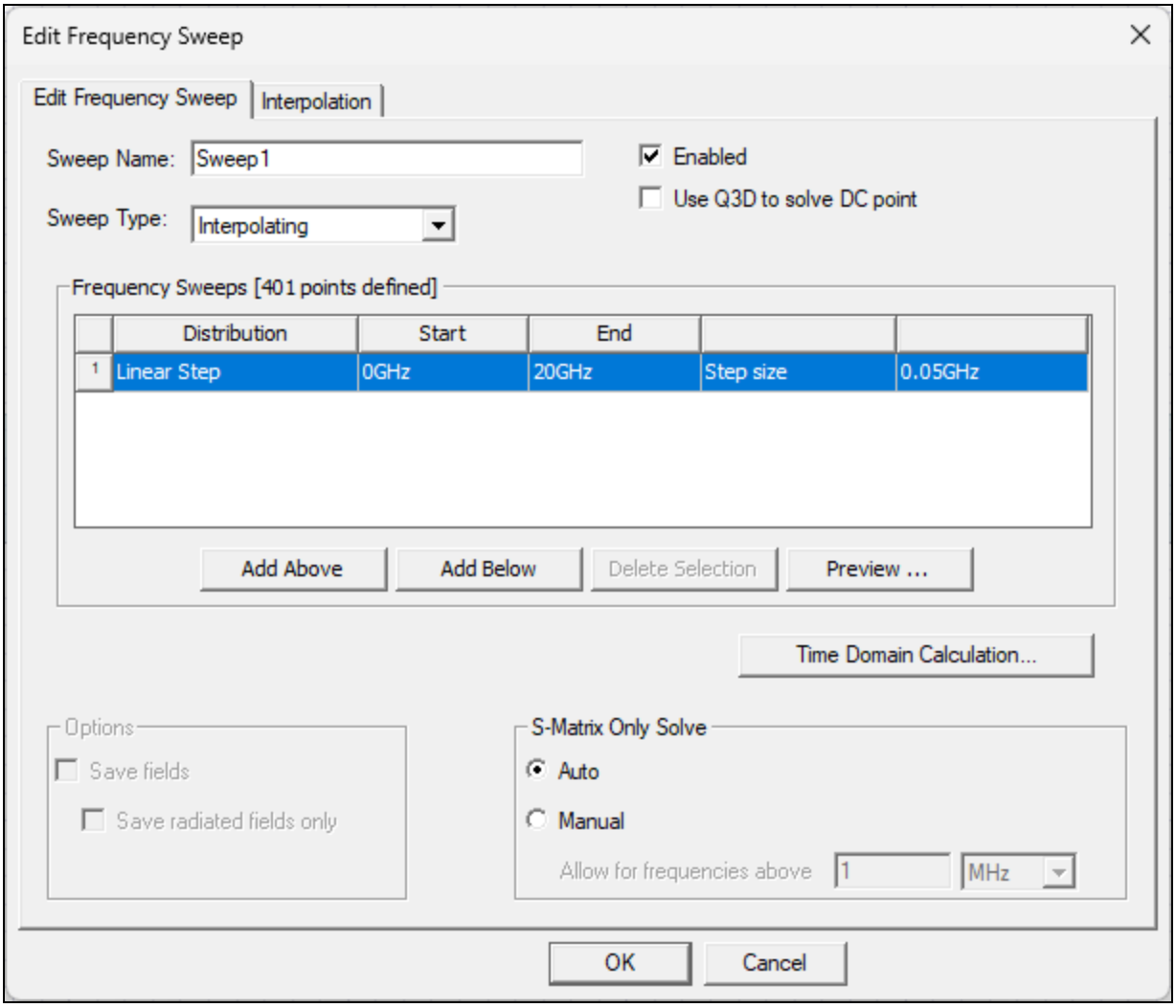


2. From the **HFSS Setup** window, click **OK** to accept the default settings and open the **Edit**

Frequency Sweep window.



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

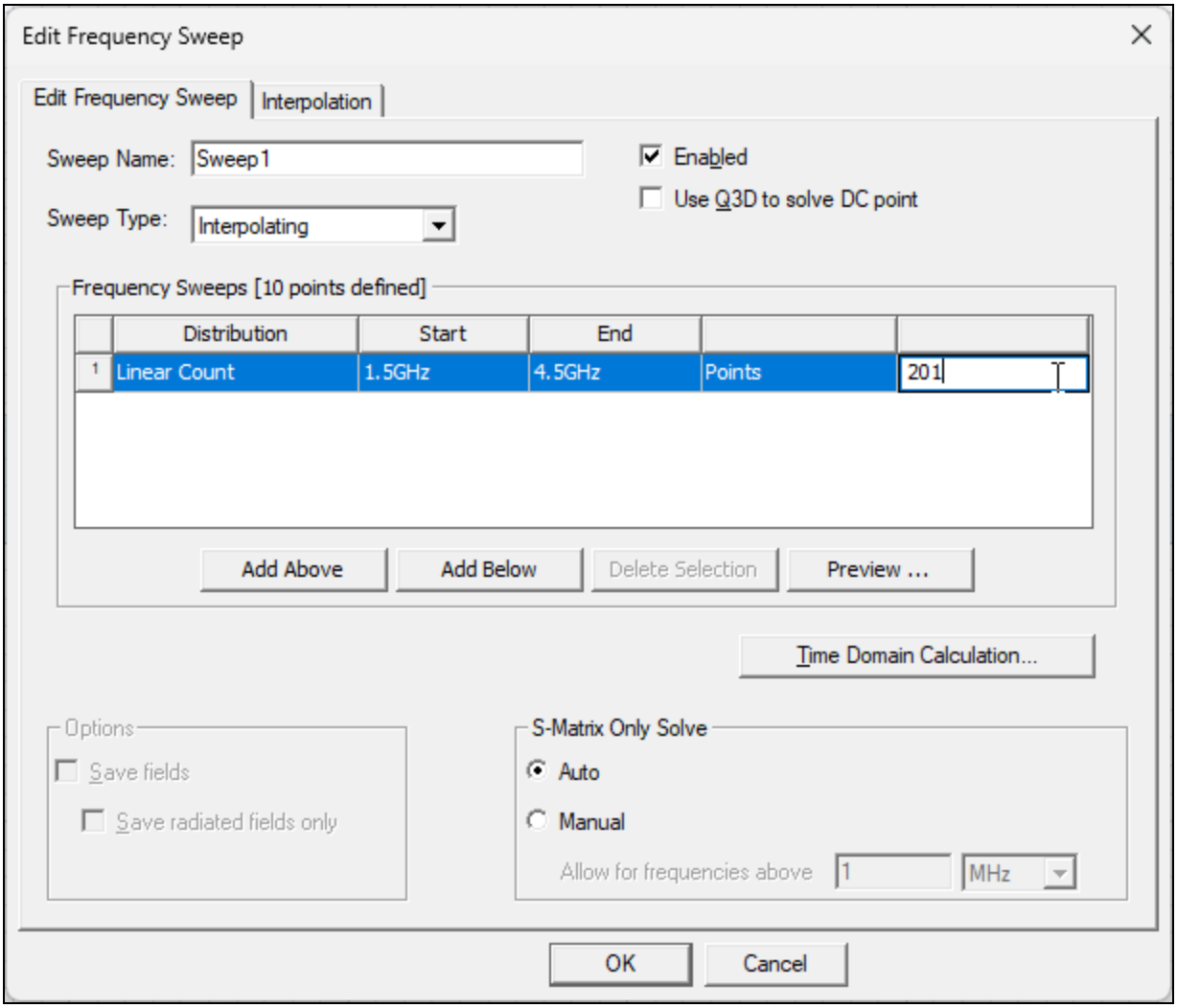


Add an Interpolating Frequency Sweep

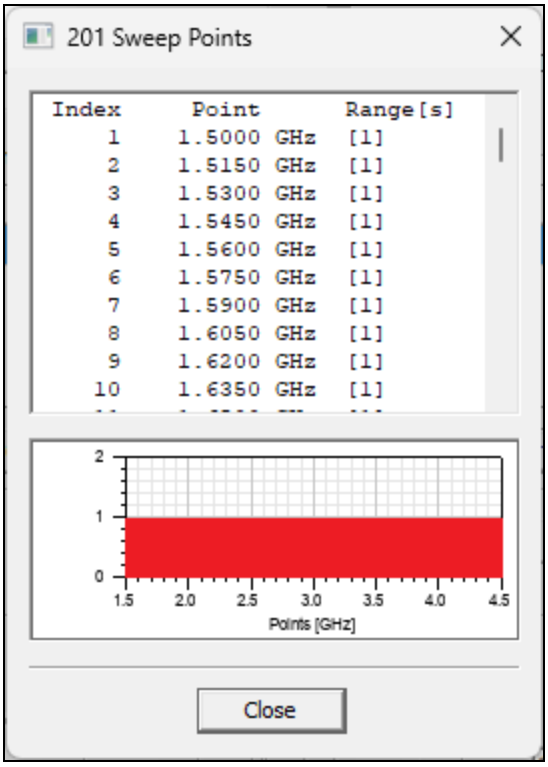
> **Note:**
>
> For interpolating sweeps, generating surface current data is only good for the last adaptive pass of the solution frequency. Surface current data cannot be saved for fixed mesh analyses.

1. Ensure **Interpolating** is selected from the **Sweep Type** drop-down menu.
2. Select **Linear Count** from the **Distribution** drop-down menu in the **Frequency Sweeps** table.

3. Enter **1.5** (GHz) in the **Start** column.
4. Enter **4.5** (GHz) in the **Stop** column.
5. Enter **201** in the **Points** field.



6. Click **Preview** to display the **Sweep Points** window.



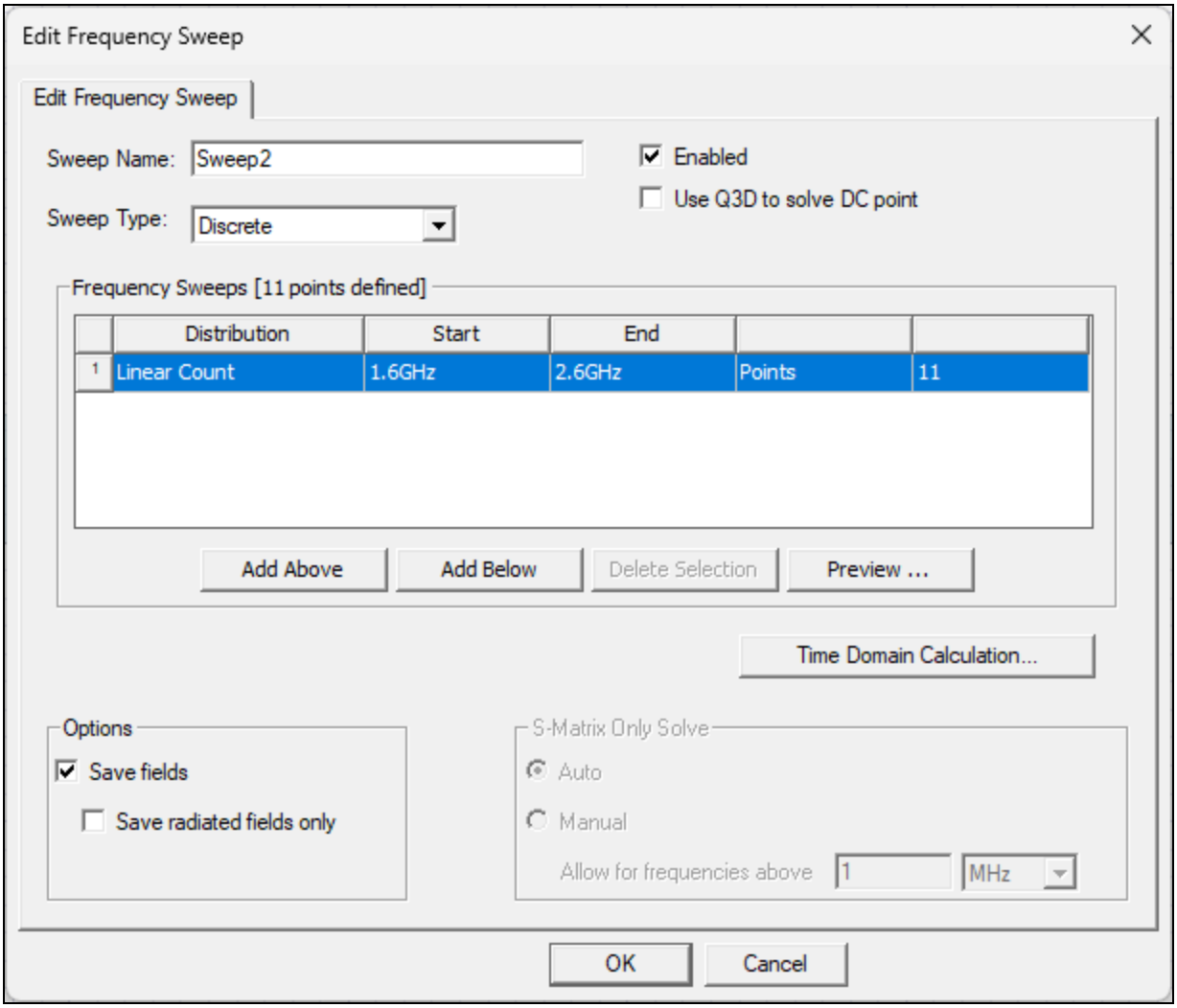
7. Click **Close** to return to the **Edit Frequency Sweep** window.
8. Click **OK** to add the interpolating sweep and close the **Edit Frequency Sweep** window. The sweep is added to the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Analysis** > **Setup1**).

Add a Discrete Frequency Sweep

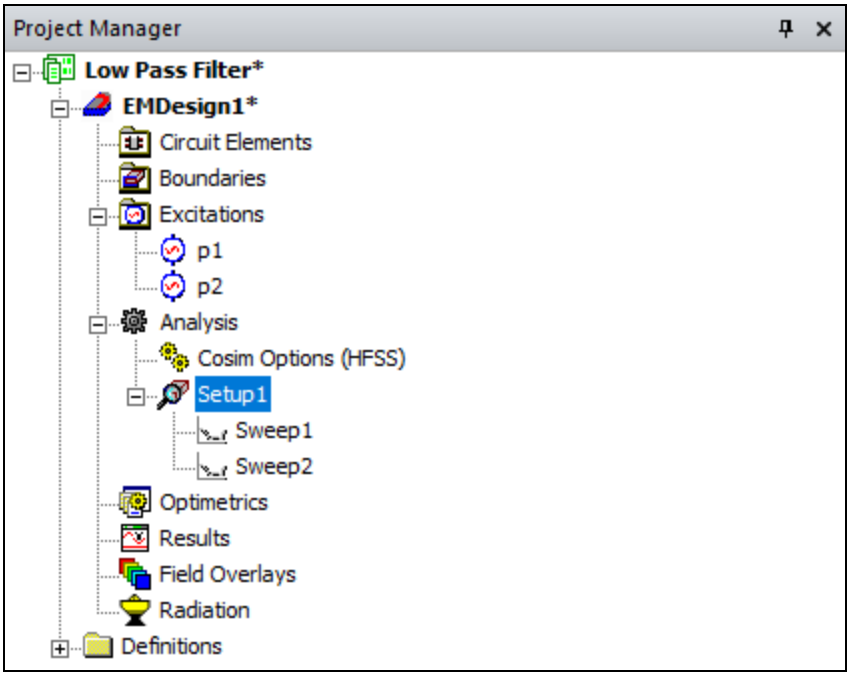
> **Note:**
>
> For discrete sweeps, generating surface current data enables viewing of currents and calculating far field effects at multiple frequencies in later post-precessing steps.

1. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click the analysis setup (e.g., **Setup1**) and select **Add Frequency Sweep** to reopen the **Edit Frequency Sweep** window.
2. Select **Discrete** from the **Sweep Type** drop-down menu.
3. Select **Linear Count** from the **Distribution** drop-down menu in the **Frequency Sweeps** table.
4. Enter **1.6** (GHz) in the **Start** column.

5. Enter **2.6** (GHz) in the **Stop** column.
6. Enter **11** in the **Points** field.
7. In the **Options** area, check the **Save fields** box.



8. Click **OK** to add the discrete sweep and close the **Edit Frequency Sweep** window. The sweep is added to the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Analysis** > **Setup1**).



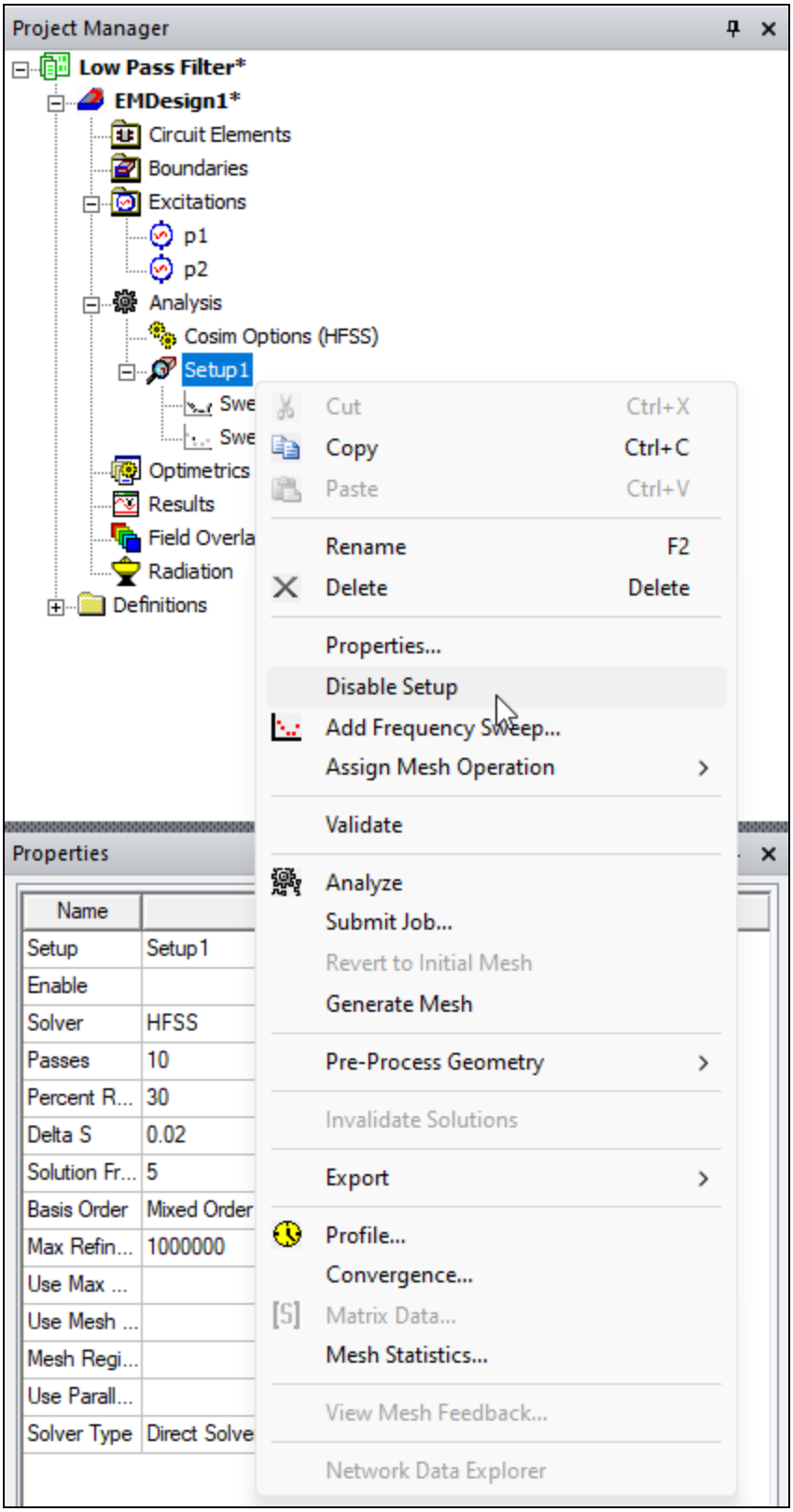
Continue to Explore Disabling Sweeps and Setups.

Deactivate/Activate Setups and Frequency Sweeps

If necessary, follow these instructions to activate or deactivate setup definitions and frequency sweeps.

Deactivate or Enable a Setup Definition

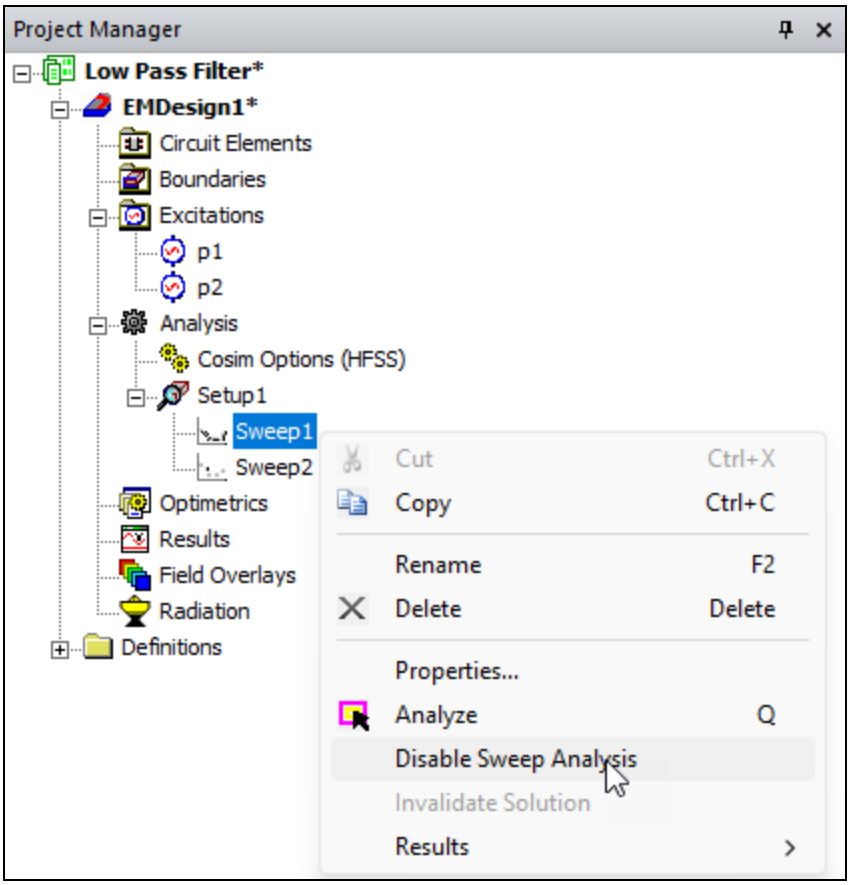
From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click **Setup1** and select **Disable Setup** or **Enable Setup**. Only one option will be available, depending on the current status of the setup (e.g., if the setup is currently deactivated, only **Enable Setup** will appear in the shortcut menu).



When an analysis setup is deactivated, any sweep associated with it has no effect, even if an associated sweep is enabled.

Deactivate or Activate a Sweep Definition

From the **Project Manager** window, expand **Setup1**. Then right-click the sweep (e.g., **Sweep1**) and select **Disable Sweep Analysis** or **Enable Sweep Analysis**. Only one option will be available, depending on the current status of the setup (e.g., if the sweep is currently deactivated, only **Enable Sweep Analysis** will appear in the shortcut menu).

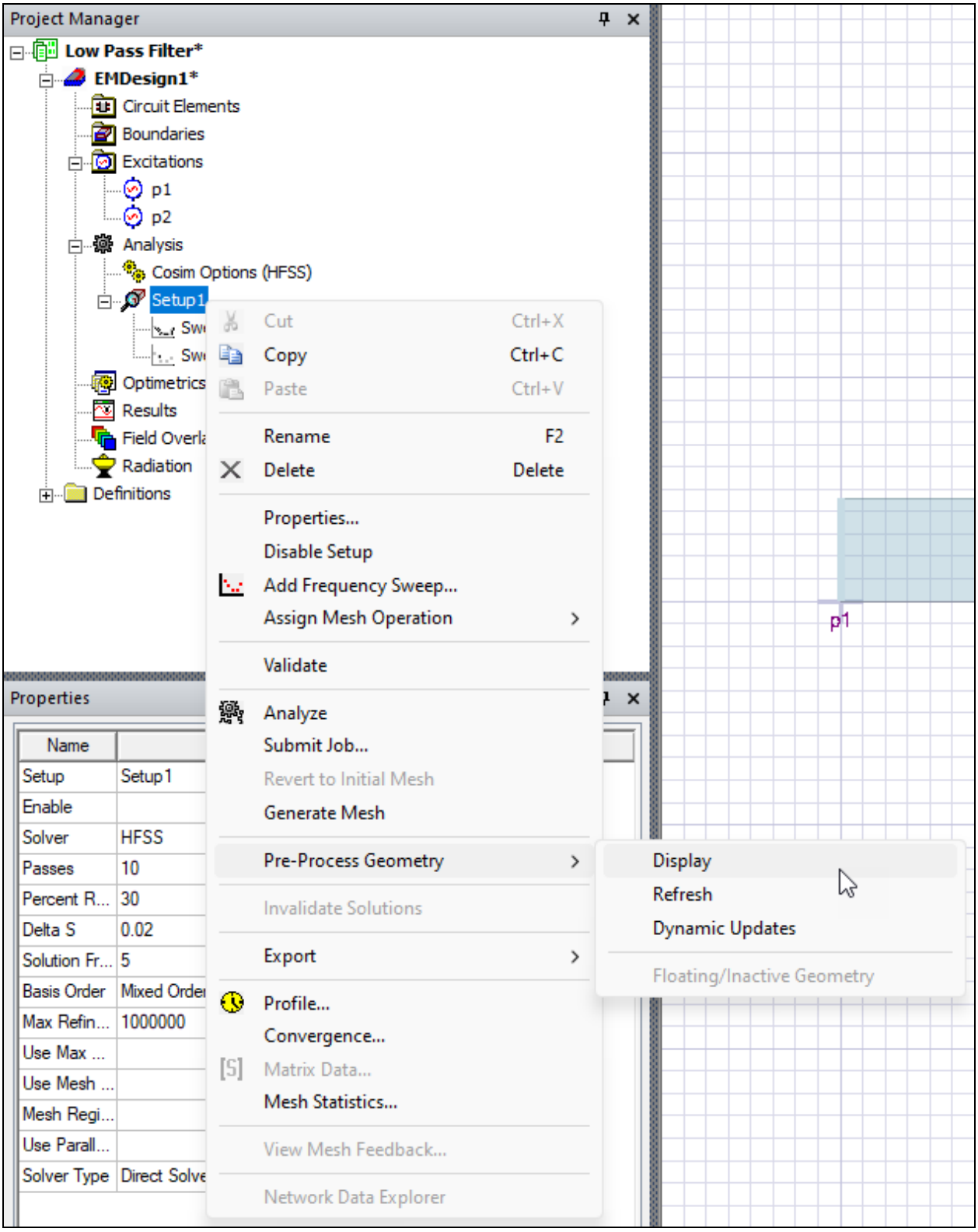


Continue to View the Mesh.

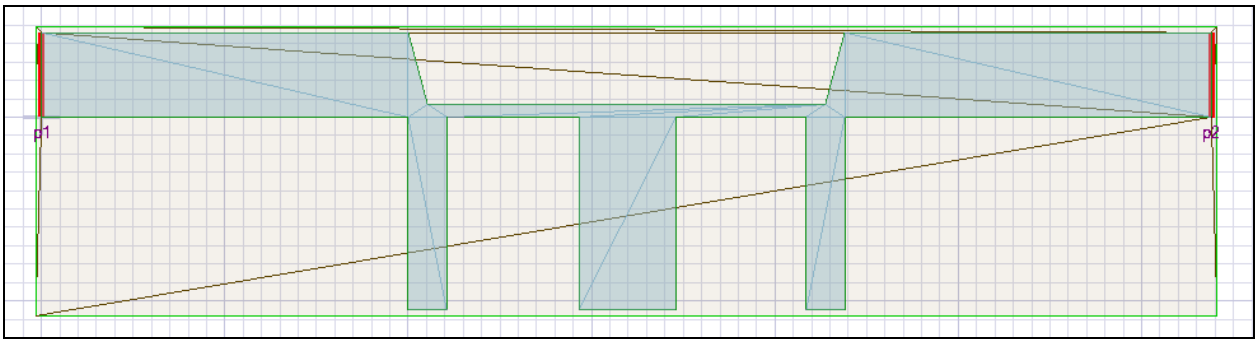
View the Mesh

Follow these instructions to view the active design in a number of planar or 3D views.

From the **Project Manager** window, right-click **Setup1** and select **Pre-Process Geometry > Display**.



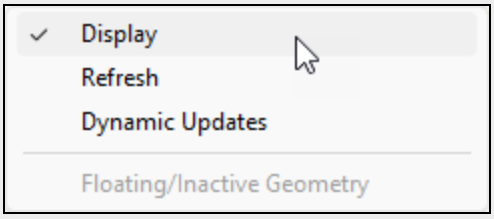
A 3D mesh appears on the model in the **Layout Editor**.



The mesh will **not** display on a layer that is not visible.

Note:

A check mark appears adjacent to **Display** when the mesh is visible and disappears when the mesh is hidden. To hide the mesh, uncheck **Display**.



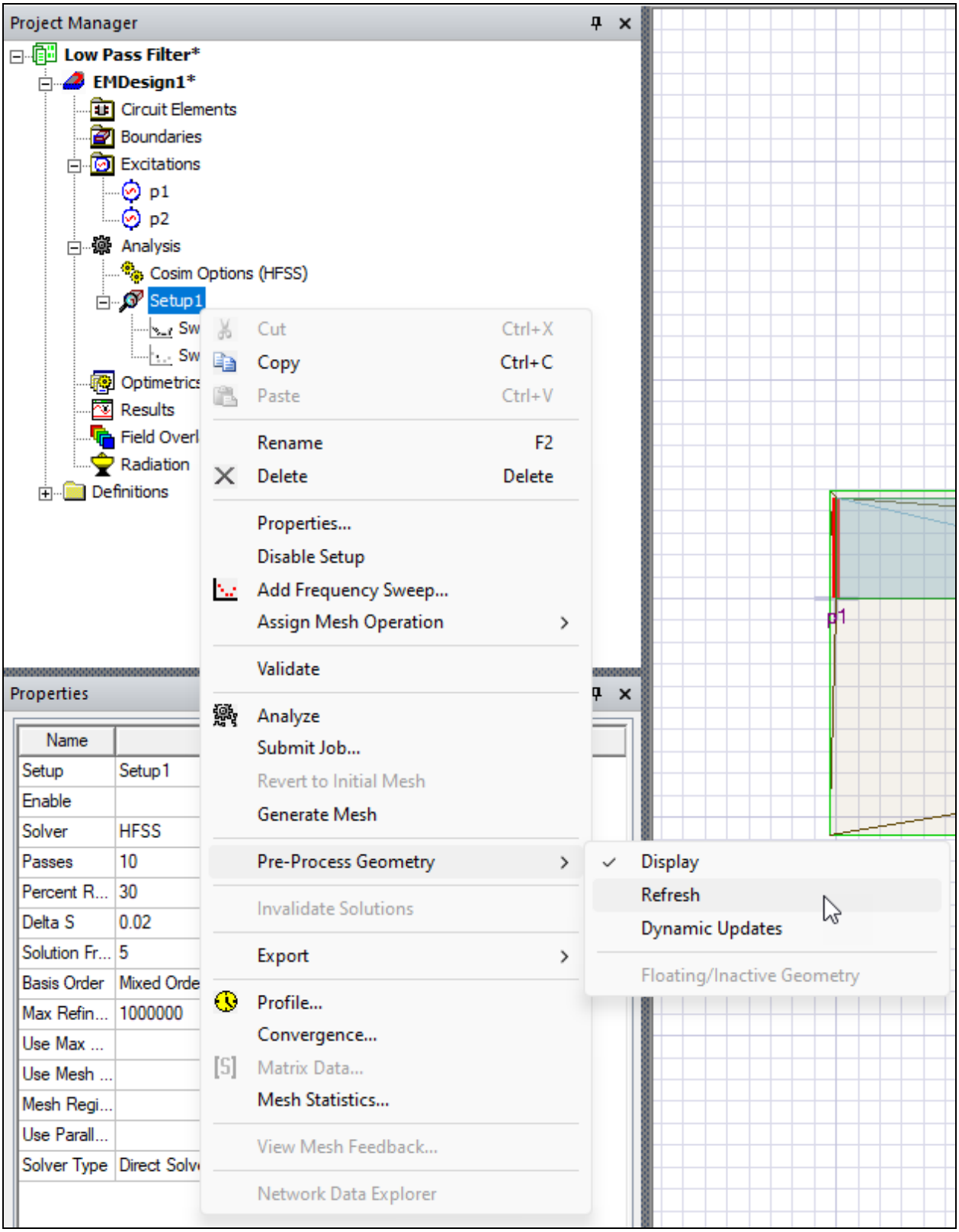
Continue to Explore Dynamic Mesh Updates.

Choose Manual or Dynamic Mesh Updates

Objects in the **Layout Editor** can be stretched, compressed, skewed, and otherwise manipulated by **click+dragging** the handles that surround the objects at the edges and midpoints. By default, an object's mesh is not dynamically updated when the geometry of the object is altered. The following instructions explain how to refresh the mesh manually or enable **Dynamic Mesh** to update the mesh in real time.

Update the Mesh Manually

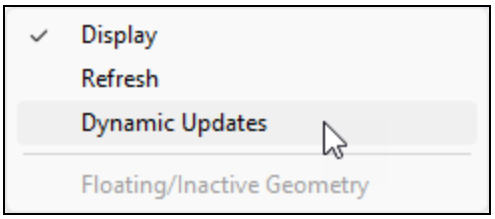
From the **Project Manager** window, right-click **Setup1** and select **Pre-Process Geometry > Refresh**.



> **Important:**
>
> The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including those with light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

Enable Dynamic Updates

From the **Project Manager** window, right-click **Setup1** and select **Pre-Process Geometry** > **Dynamic Updates**.



Reset the Mesh After Using Undo or Redo

Even if **Dynamic Updates** is enabled, using the **Undo** or **Redo** commands to reset the geometry of the design will disrupt the dynamic update process and the mesh will not immediately conform to the reset geometry. **Refresh** to reset the mesh.

> **Note:**
>
> A check mark appears adjacent to **Dynamic Updates** when it is active and disappears when **Dynamic Updates** is inactive. To deactivate **Dynamic Updates**, uncheck **Dynamic Updates**.

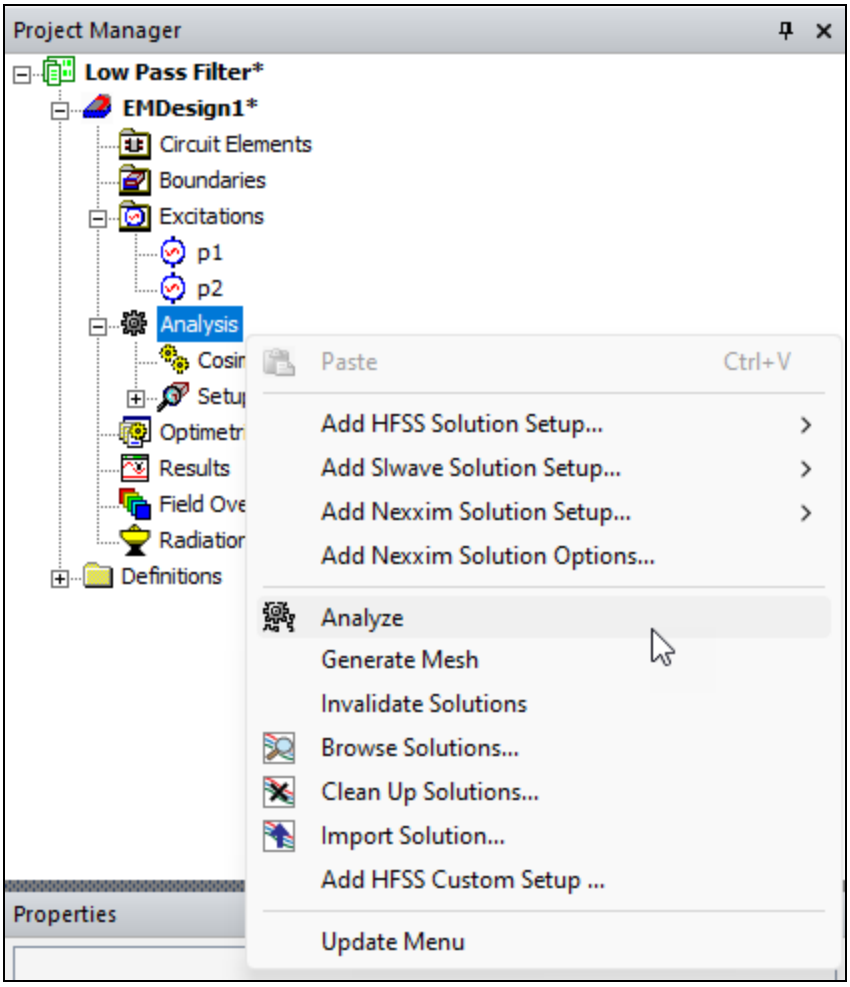
Experiment with the **Layout Editor** by **activating the mesh**, enabling dynamic updates, and then distorting or even adding new geometry to the model. Use the **Select (O)** and **Handles (H)** modes to update the cursor, then **click+drag** to alter the model, as applicable.

After experimenting, **deactivate the mesh**.

Continue to Run the Analysis.

Run Analyses and Choose Analyses Options

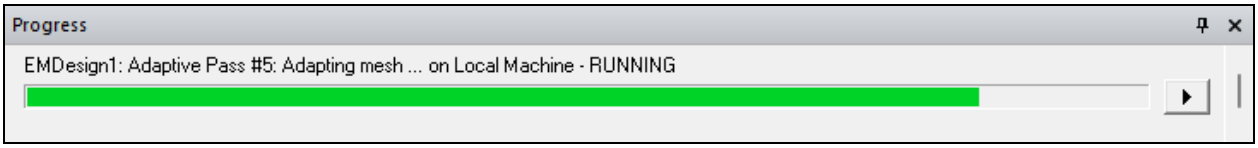
To sequentially run all analysis setups and associated frequency sweeps in the active design, navigate to the **Project Manager** window. Then right-click **Analysis** and select **Analyze**.



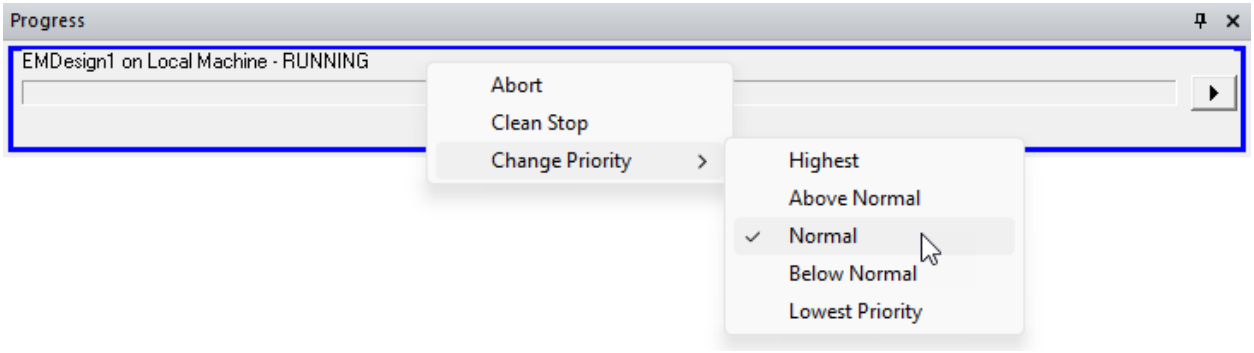
> **Note:**
>
> Alternatively, run a single setup or sweep by right-clicking the applicable setup or sweep and select **Analyze**.

View Progress

View simulation progress in the **Progress** window.



If necessary, perform the following actions during analysis: right-click within the **Progress** window and select from: **Abort**, **Clean Stop**, or **Pause**. **Clean Stop** completes the computation of the current frequency point, then analysis ends. Select **Change Priority** > (**Highest**, **Above Normal**, **Normal**, **Below Normal**, or **Lowest Priority**), to change the priority of the associated solution. Altering the priority can be useful when multitasking, to free up resources from a computationally intensive application. Conversely, it can prevent less important programs from excessively slowing down the more intensive application.



Continue to View S Matrix Data.

3 - Evaluate the Results (Post-processing)

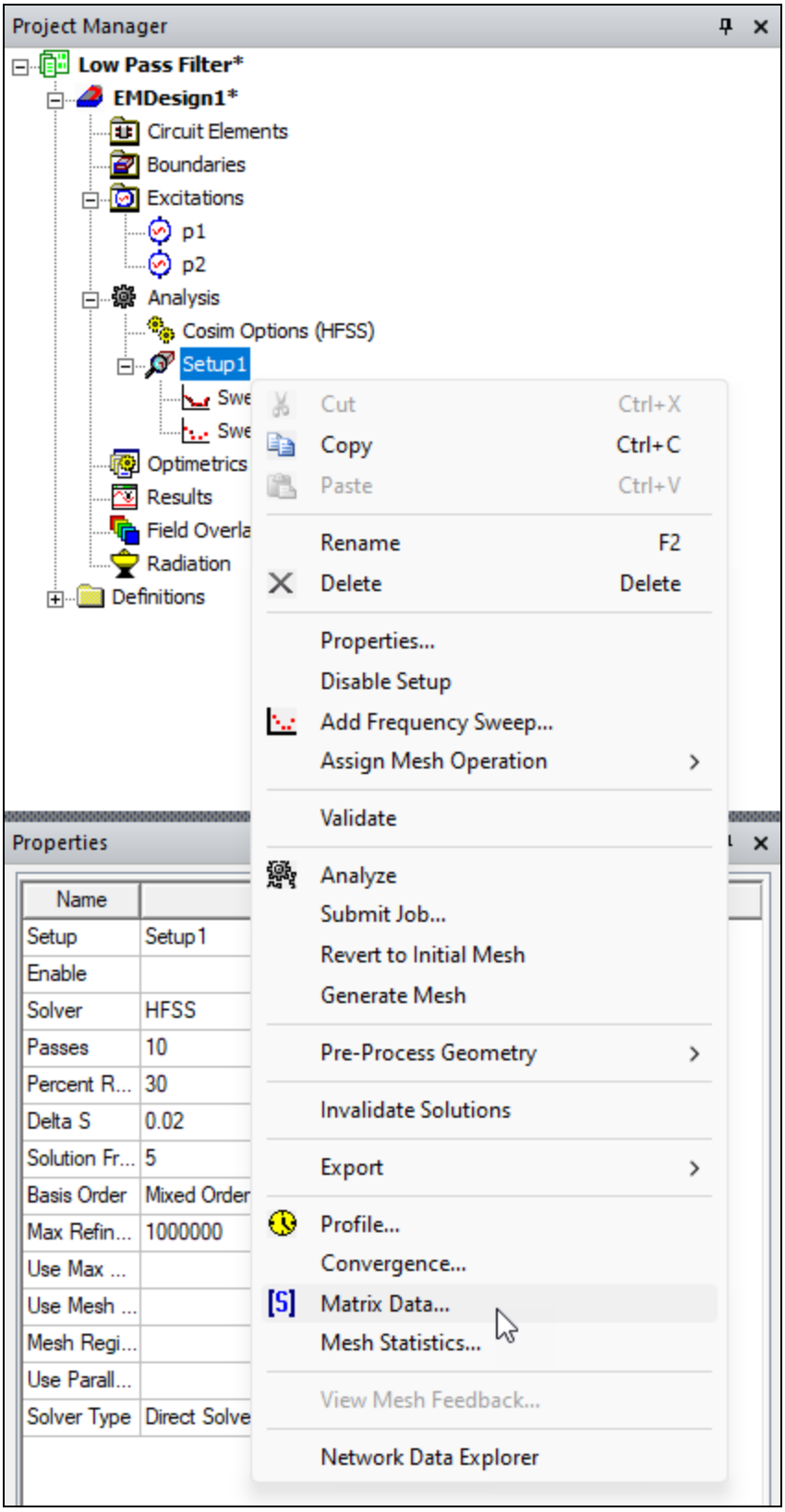
Use the post-processing capabilities of Ansys Electronics Desktop to display the results of a simulation. Also use the export features to save the analysis data (and an equivalent circuit) in various industry-standard file formats.

The sections in this topic include:

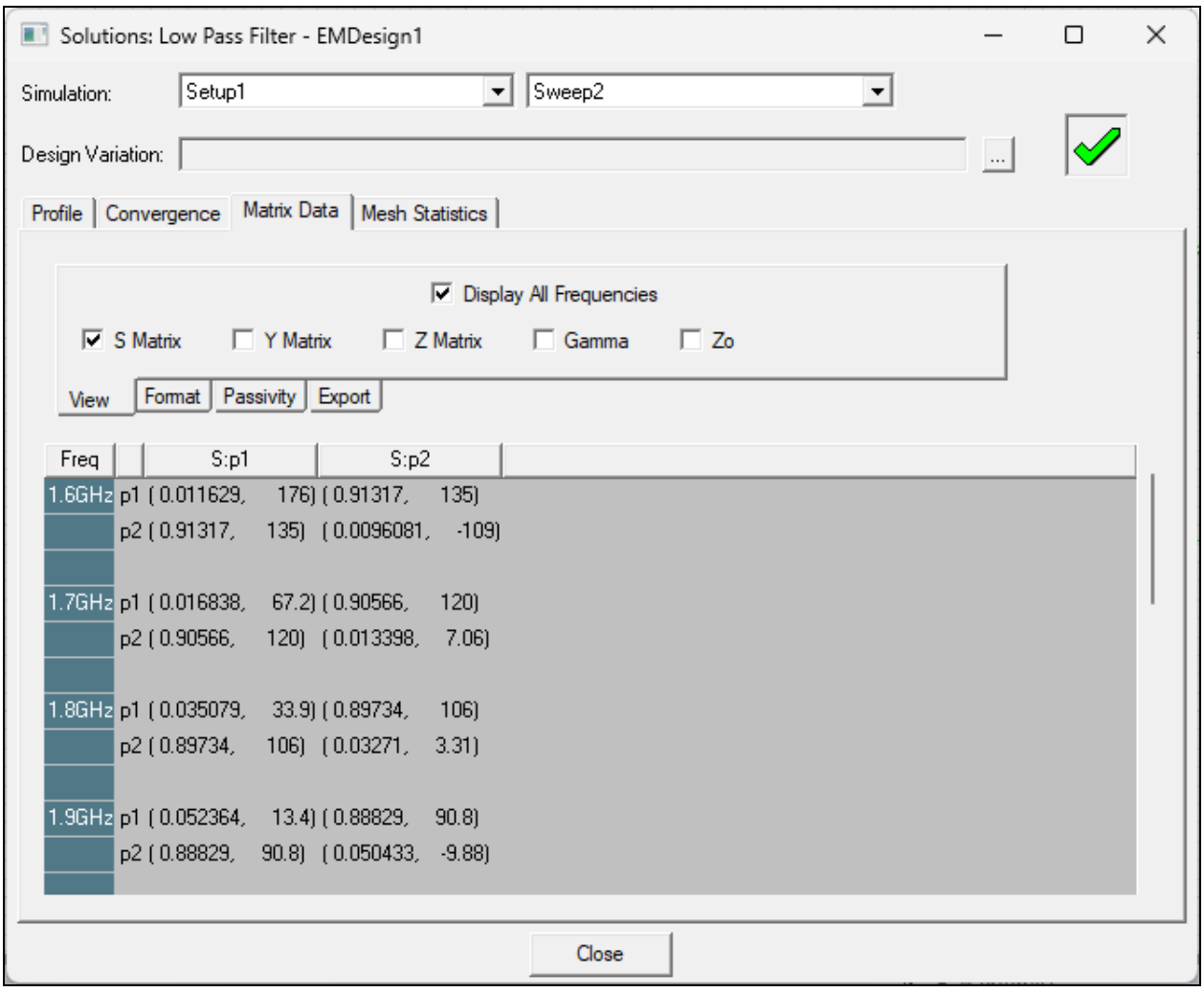
View *S* Matrix Data

Follow these instructions to review the scatter matrix (S matrix) data at all sweep frequencies or at a selected frequency, as well as Y and Z matrix data.

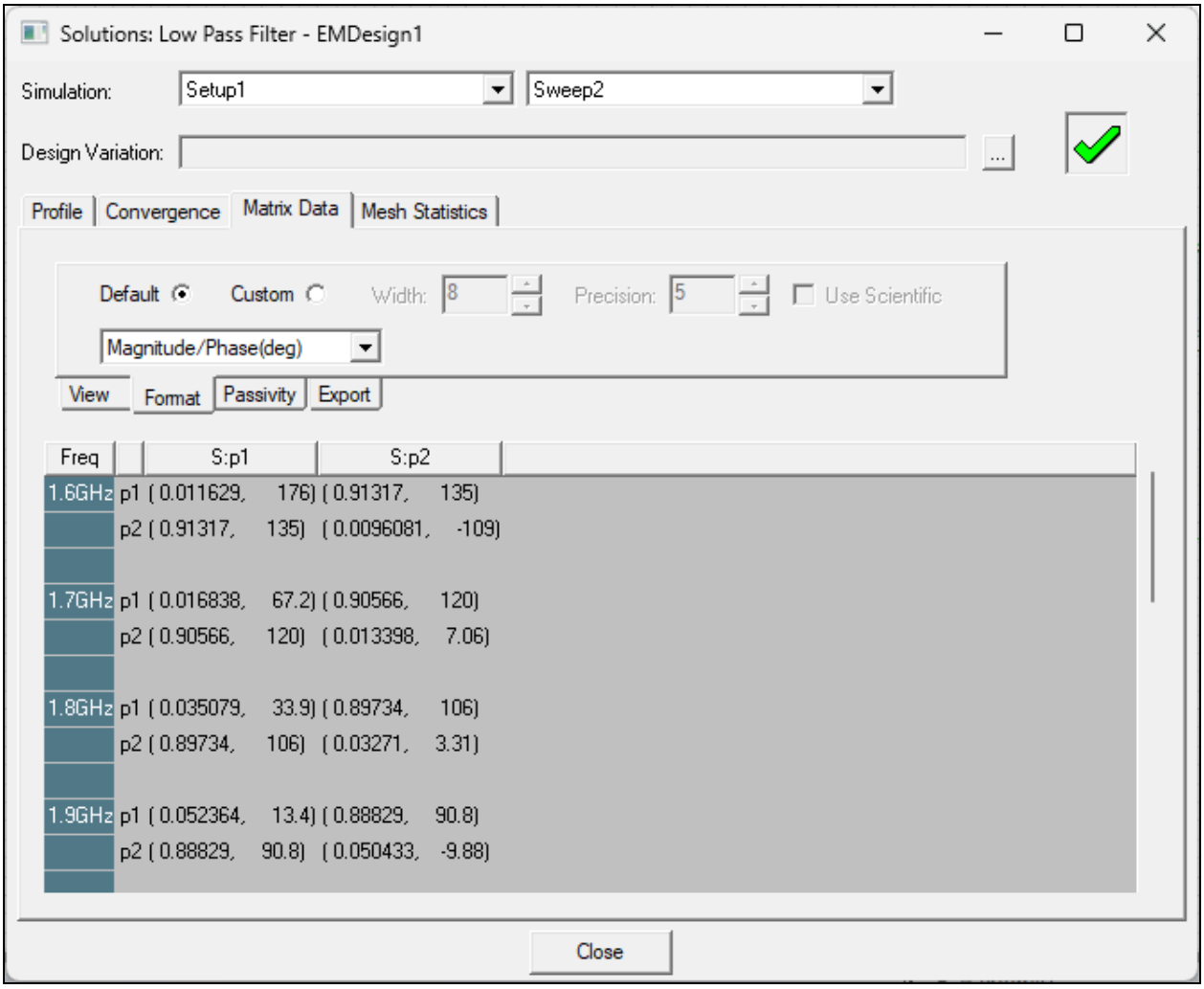
1. From the **Project Manager** window, right-click **Setup1** and select **Matrix Data** to open the **Solutions** window.



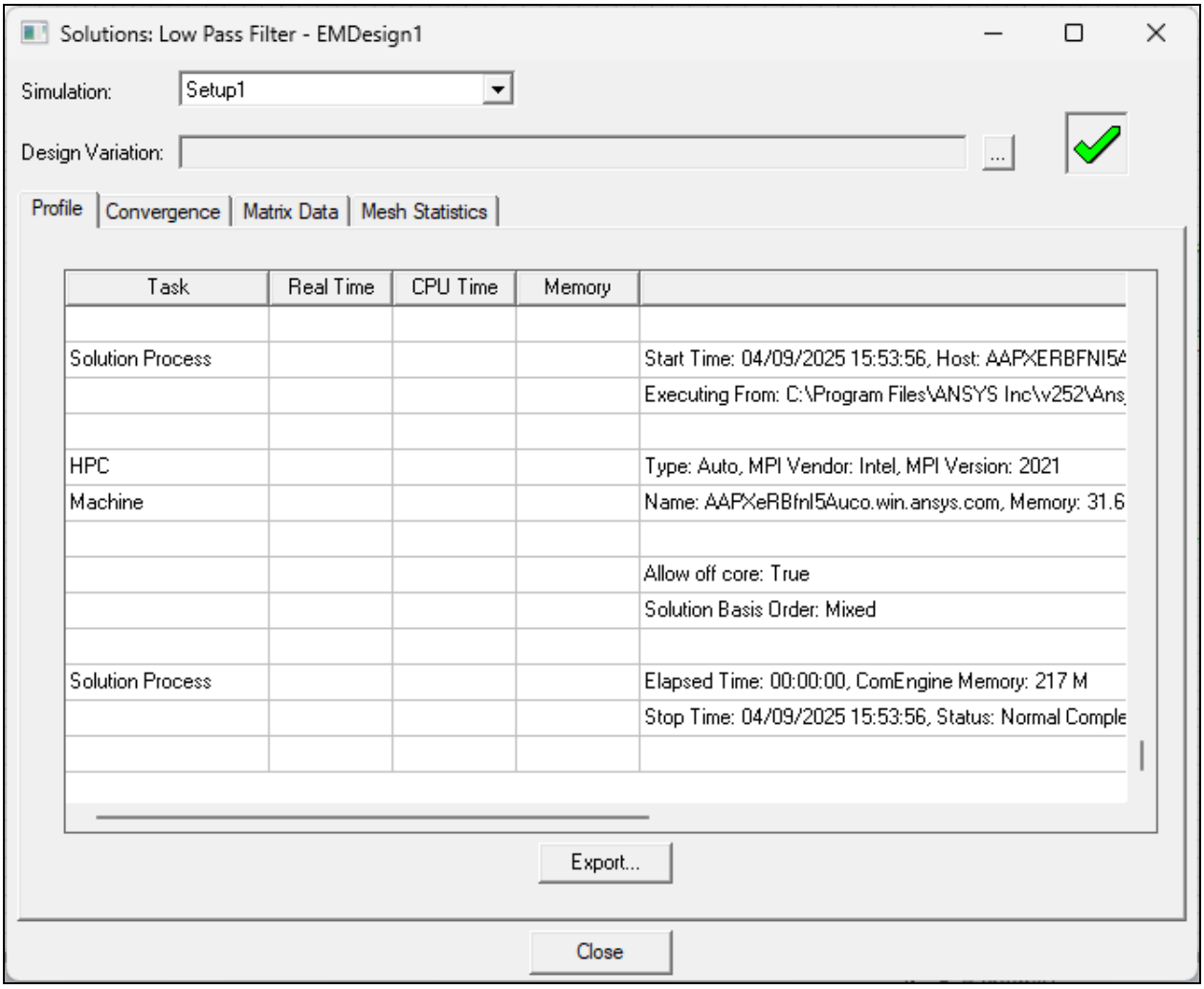
2. From the **Matrix Data** tab, ensure the **Display All Frequencies** and **S Matrix** boxes are checked.



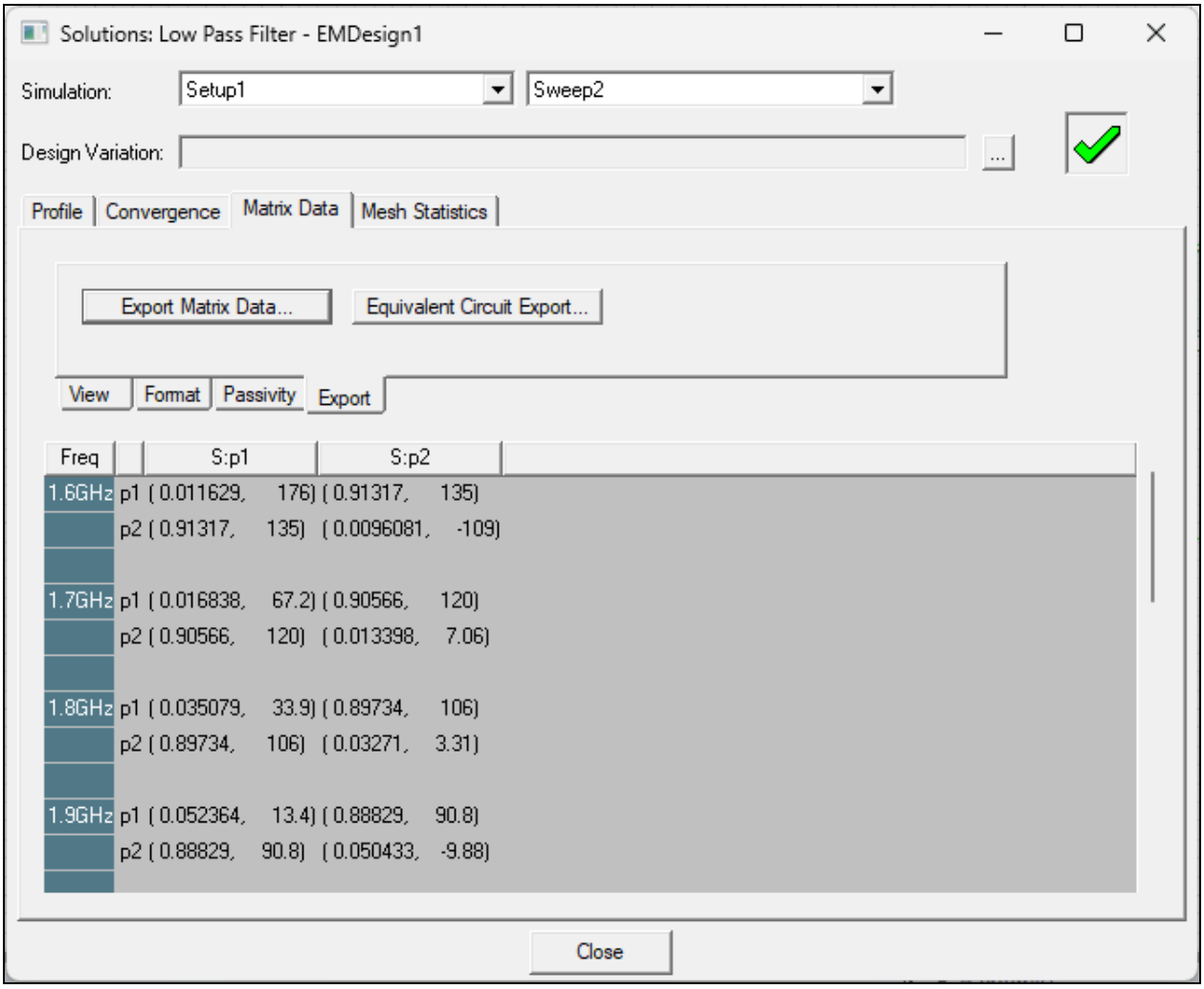
3. Navigate to the **Format** subtab. Then choose the selected data format from the drop-down menu (i.e., **Magnitude/Phase(deg)**, **dB/Phase(deg)**, **Real**, **Imaginary**, **Magnitude**, **Phase(deg)**, or **dB**).



4. To see runtime data pertaining to the analysis, navigate to the **Profile** tab.

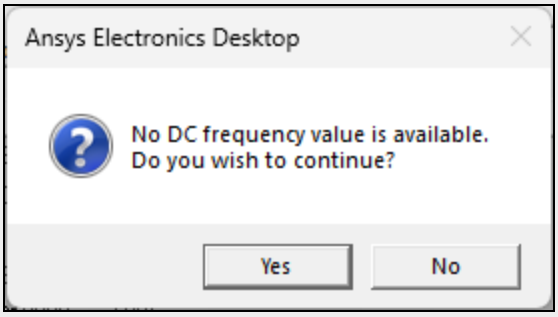


5. To export data, do the following:
    a. Navigate to the **Matrix Data** > **Export** subtab and click **Export Matrix Data** to open an explorer window.

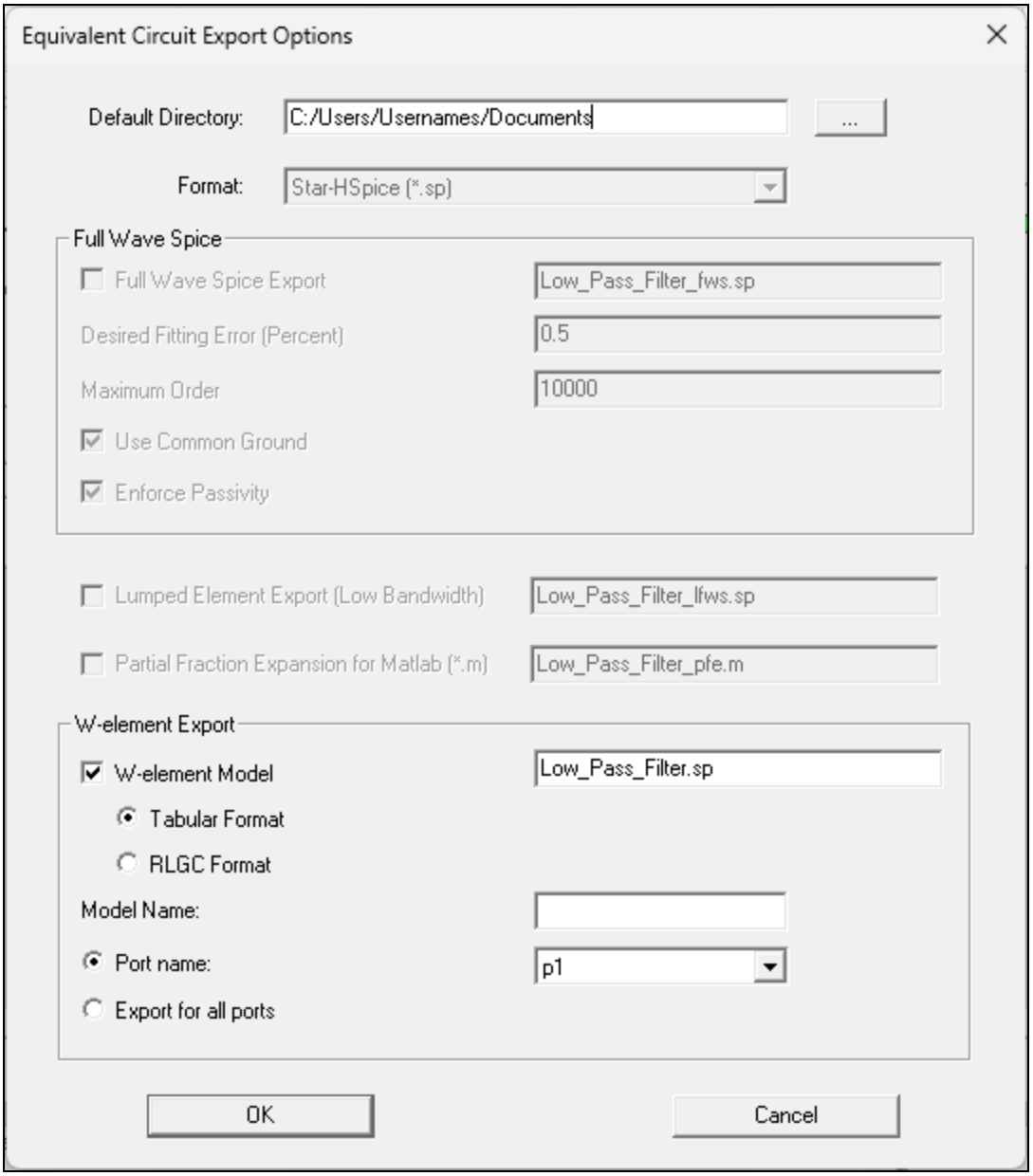


b. From the explorer window, do the following:
   i. Type a **File name** or use the default (e.g., <**ProjectName_DesignName**>).
   ii. Select a file format from the **Save as type** drop-down menu (i.e., **Touchstone (*.sN*p*)**, **Data Table (spreadsheet) (*.*tab*)**, **Neutral Model Format (*.*nmf*)**, **MATLAB (*.*m*)**, or **Citifile (*.*cit*)**).
   iii. Navigate to an applicable save location. (The default save location is the folder where the model is saved).
   iv. Click **Save** to save changes, close the explorer window, and return to the **Solution** window, or click **Cancel** to close the explorer window without exporting anything.

6. To export the matrix data as an equivalent SPICE model, do the following:
   a. From the **Export** subtab, click **Equivalent Circuit Export**.

> **Note:** SPICE models require solution data at DC (0Hz) conditions. If there is no DC frequency value available, a warning will appear.
>
> 
>

   b. If necessary, click **Yes** to close warning and open the **Equivalent Circuit Export Options** window.



c. Select an applicable save location, file format, and other equivalent circuit options.

d. Click **OK** to save changes, close the **Equivalent Circuit Export Options** window, and return to the **Solution** window, or click **Cancel** to close the window without exporting anything.

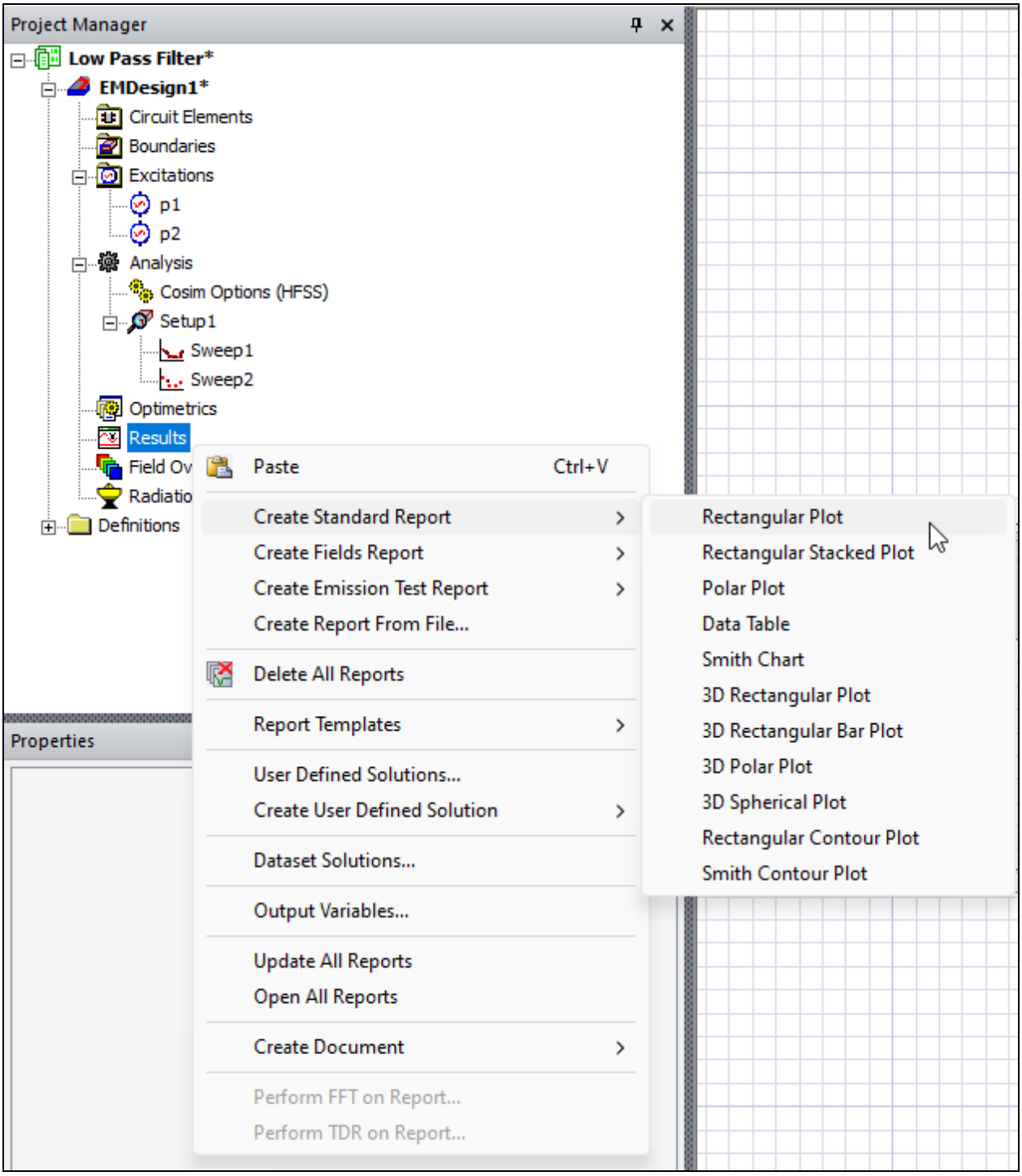
7. **Close** the **Solutions** window.

Continue to Plot Return Loss.

Plot Return Loss (a Standard Report)

Follow these steps to create a report and plot return loss.

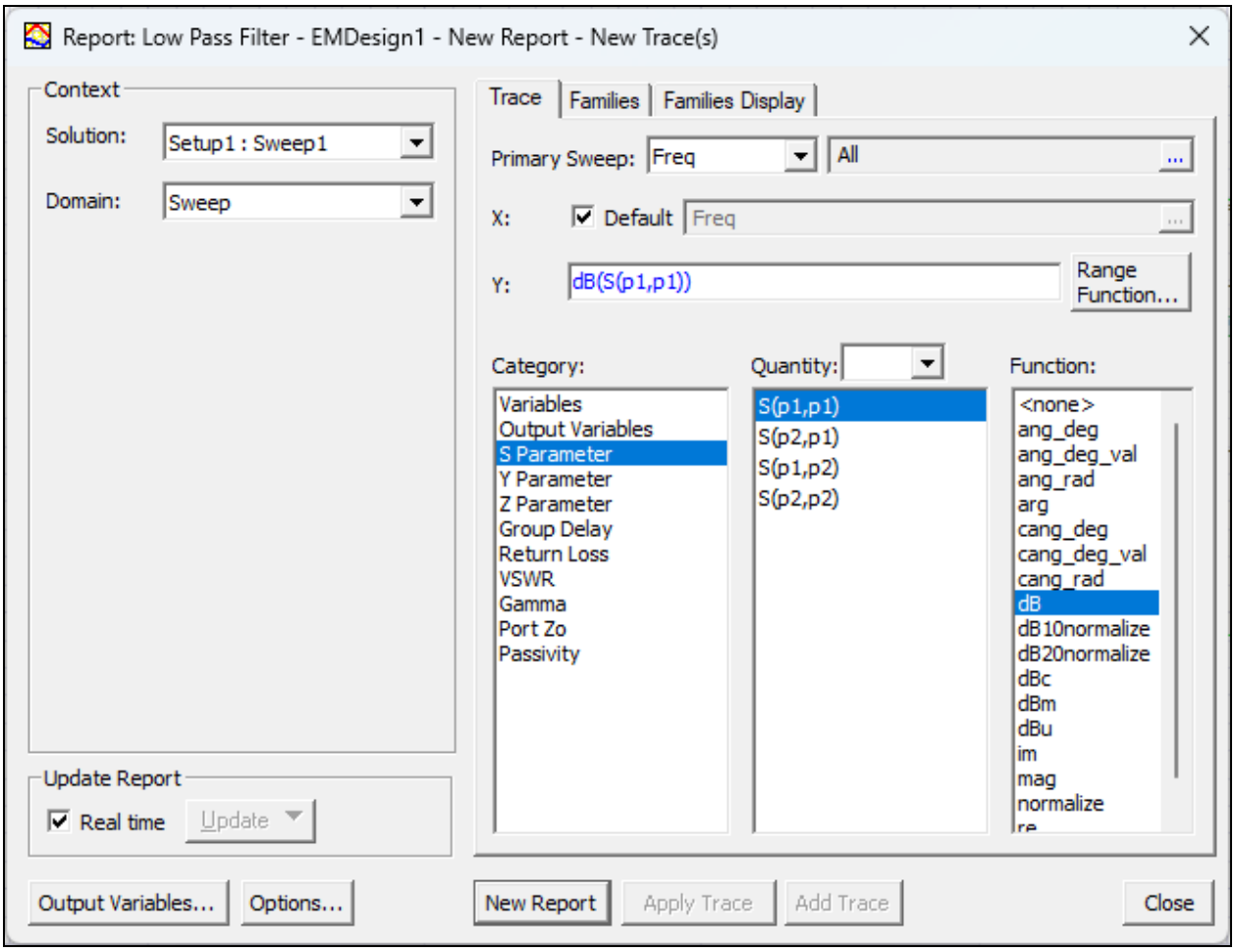
1. From the **Project Manager** window, right-click **Results** and select **Create Standard Report** > **Rectangular Plot** to open the **Report** window.



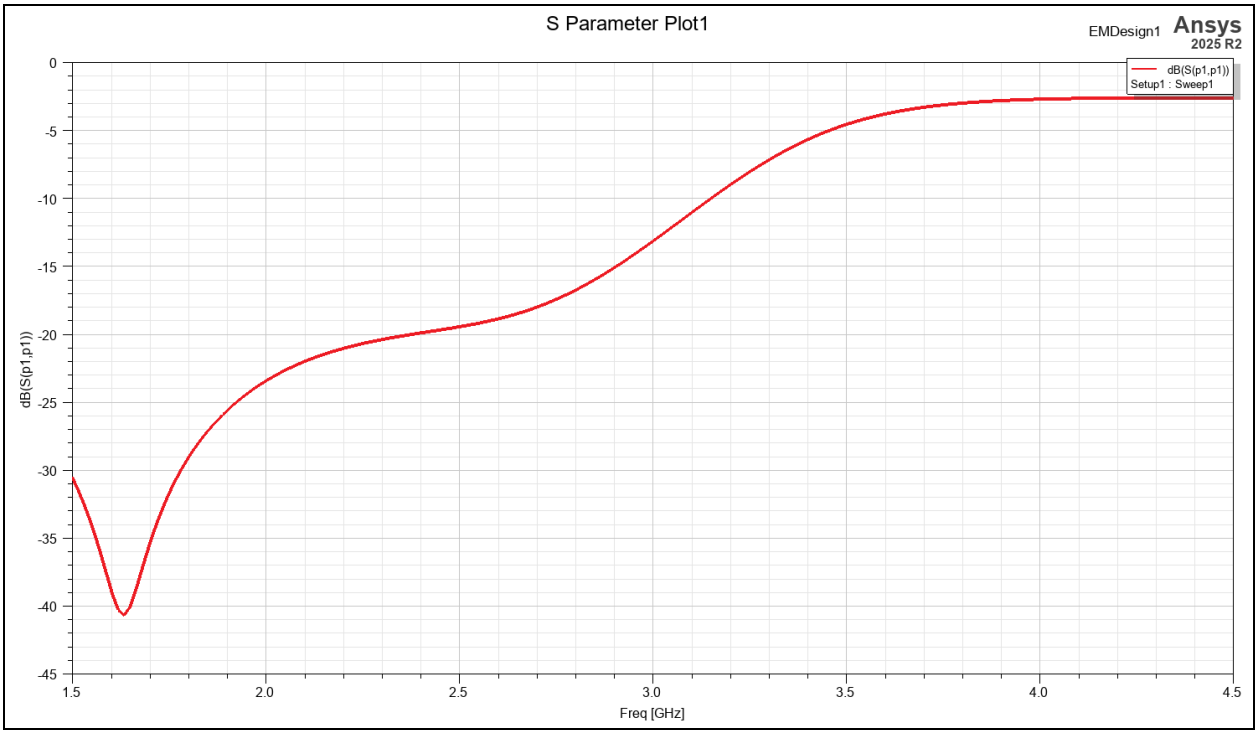
2. From the **Report** window, ensure the following settings are selected:
   a. From the **Solution** drop-down menu, **Setup1 : Sweep1**.
   b. From the **Category** list, **S Parameter**.

c. From the **Quantity** list, **S(p1,p1)**.

d. From the **Function** list, **dB**.



3. Click **New Report** and the return loss plot opens under the **Report** window.
4. **Close** the **Report** window to view the plot.



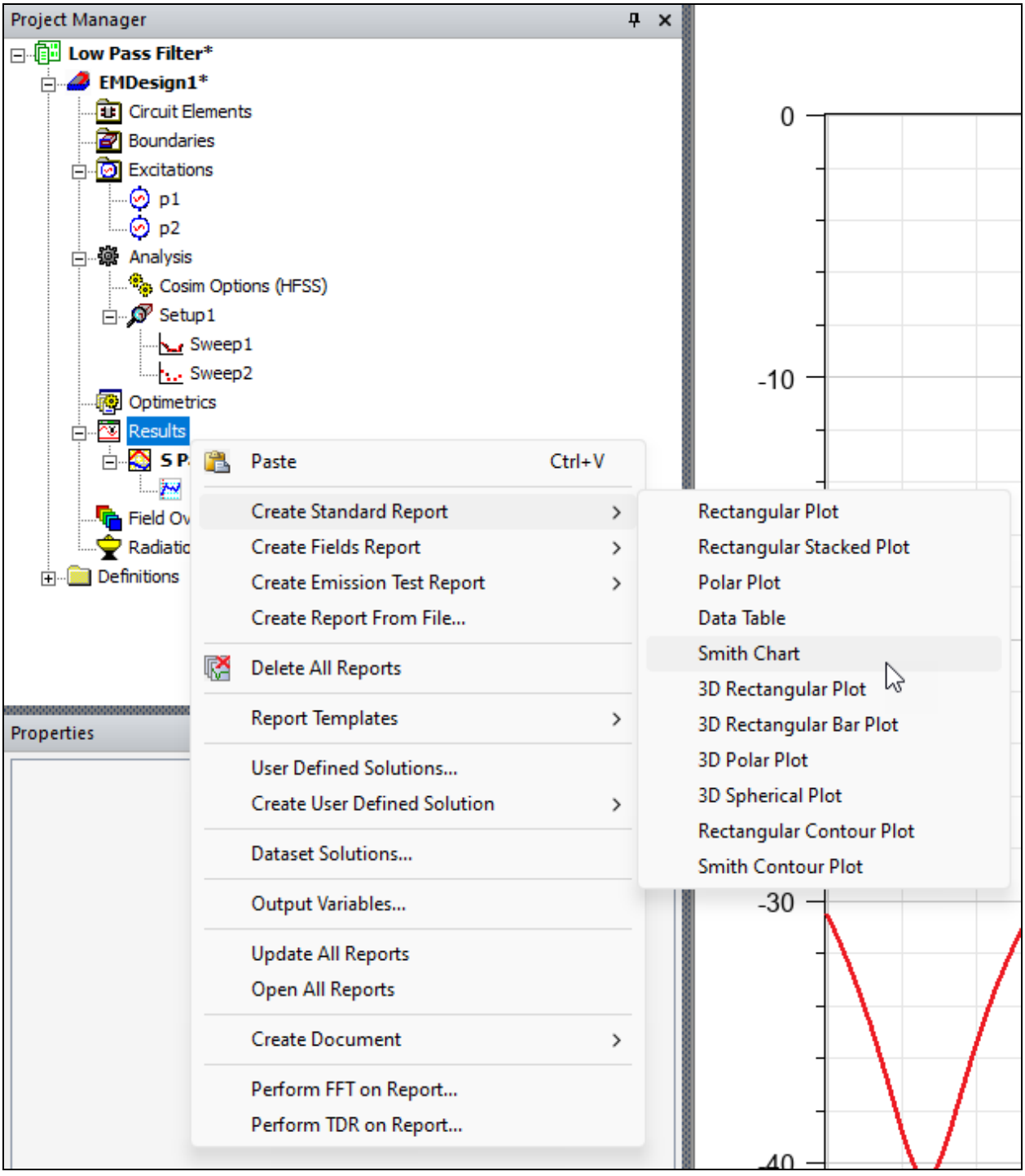
> **Note:**
>
> Return to the plot any time from the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Results** folder, and double-click the applicable plot).

Continue to Plot a Smith Chart.

Plot a Smith Chart

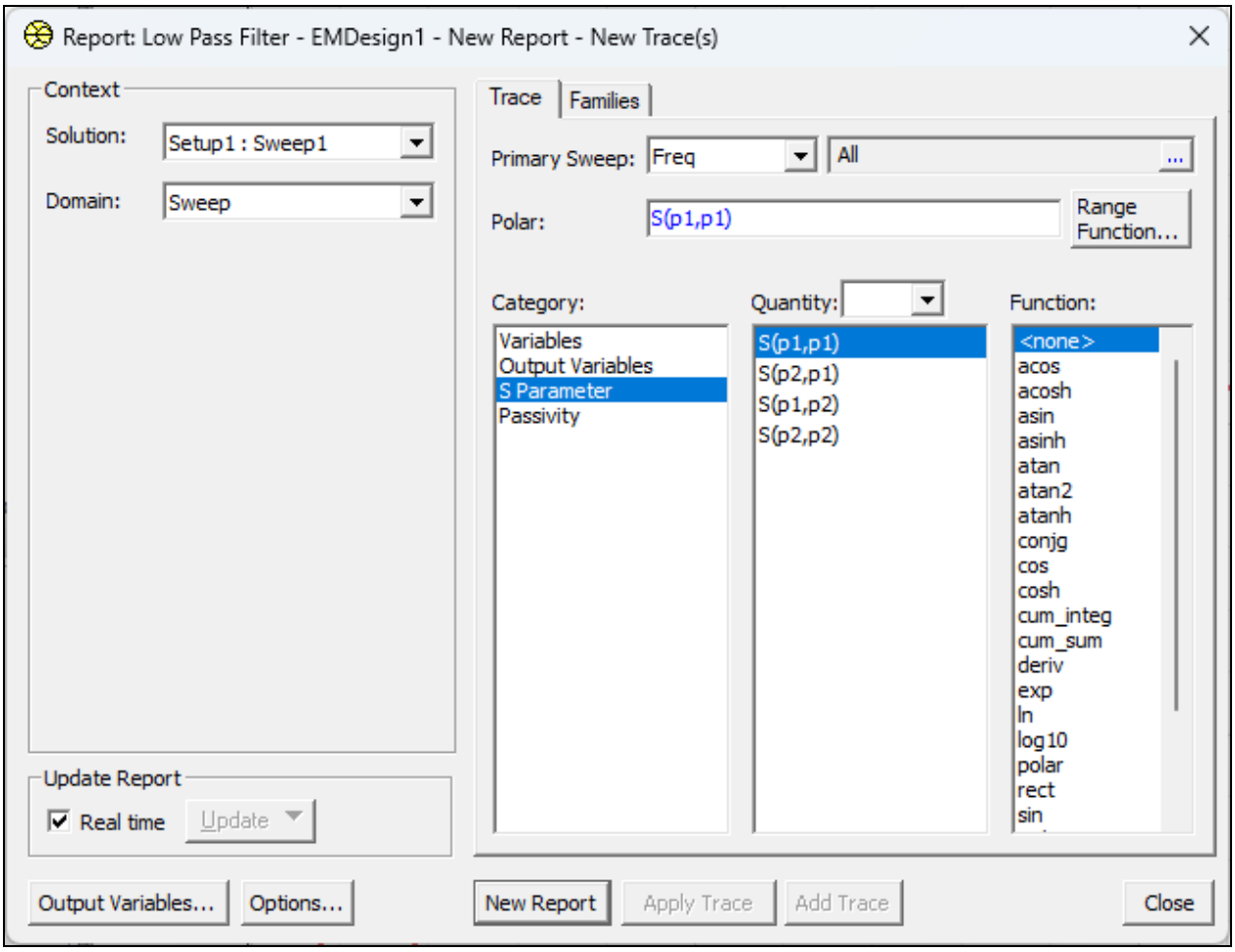
A Smith chart is a convenient means of simultaneously displaying multiple parameters (e.g., impedance, reflection coefficients, scattering parameters, constant gain contours, et cetera). It is a useful graphical aid for electronics engineers working in the radio frequency disciplines. Follow these steps to create a Smith chart.

1. From the **Project Manager** window, right-click **Results** and select **Create Standard Report** > **Smith Chart** to open the **Report** window.

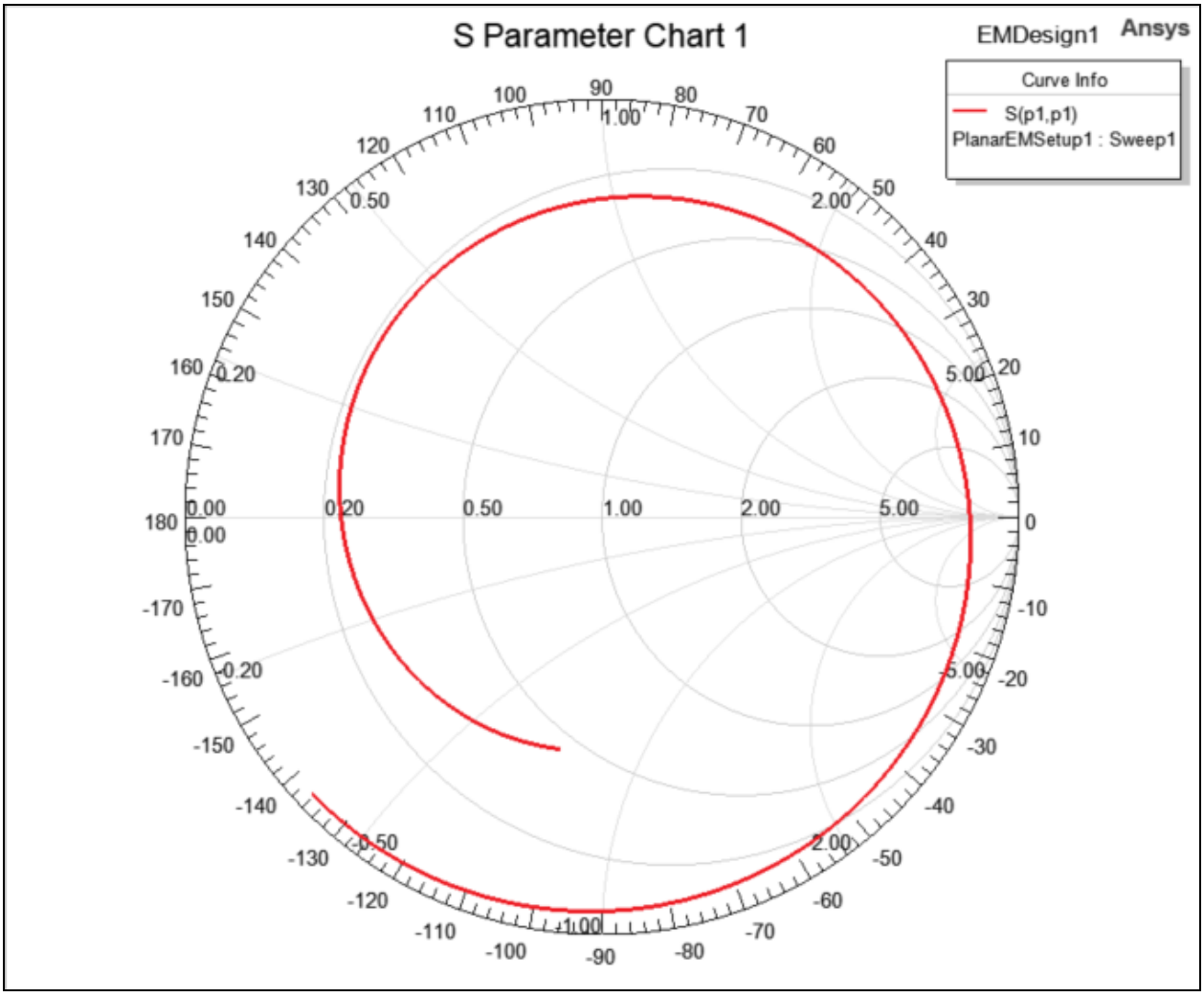


2. From the **Report** window, ensure the following settings are selected:
   a. From the **Solution** drop-down menu, **Setup1 : Sweep1**.
   b. From the **Category** list, **S Parameter**.

c. From the **Quantity** list, **S(p1,p1)**.
d. From the **Function** list, **<none>**.



3. Click **New Report** and the return loss plot opens under the **Report** window.
4. **Close** the **Report** window to view the plot.



Note:

Return to the plot any time from the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Results** folder, and double-click the applicable plot).

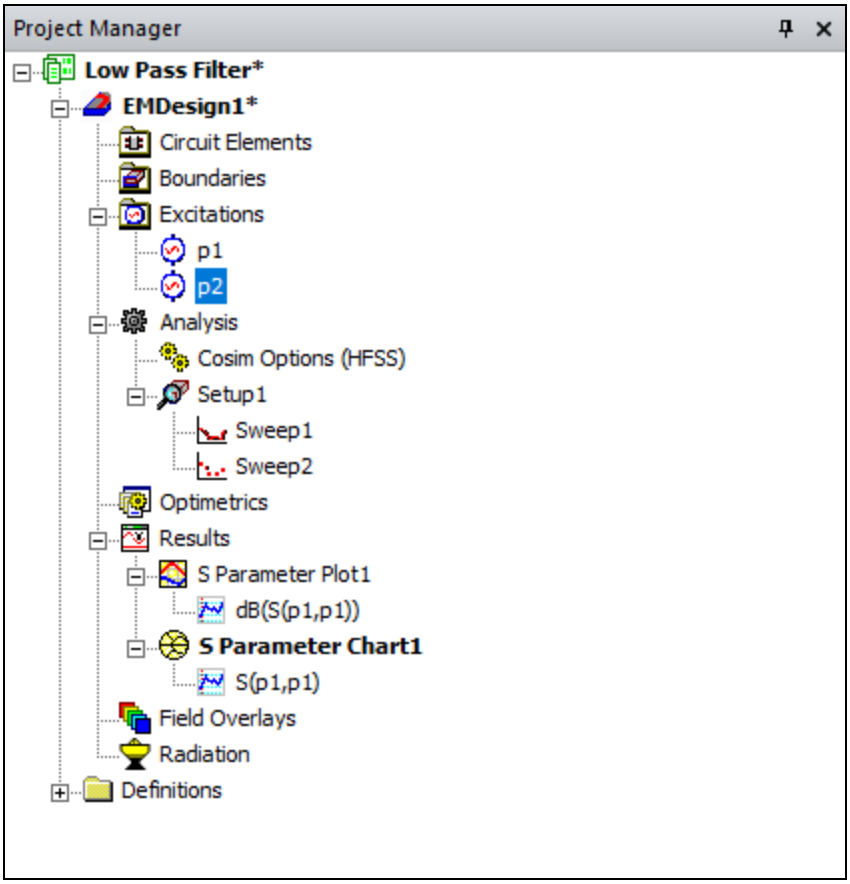
Continue to Revise An Excitation.

Revise An Excitation

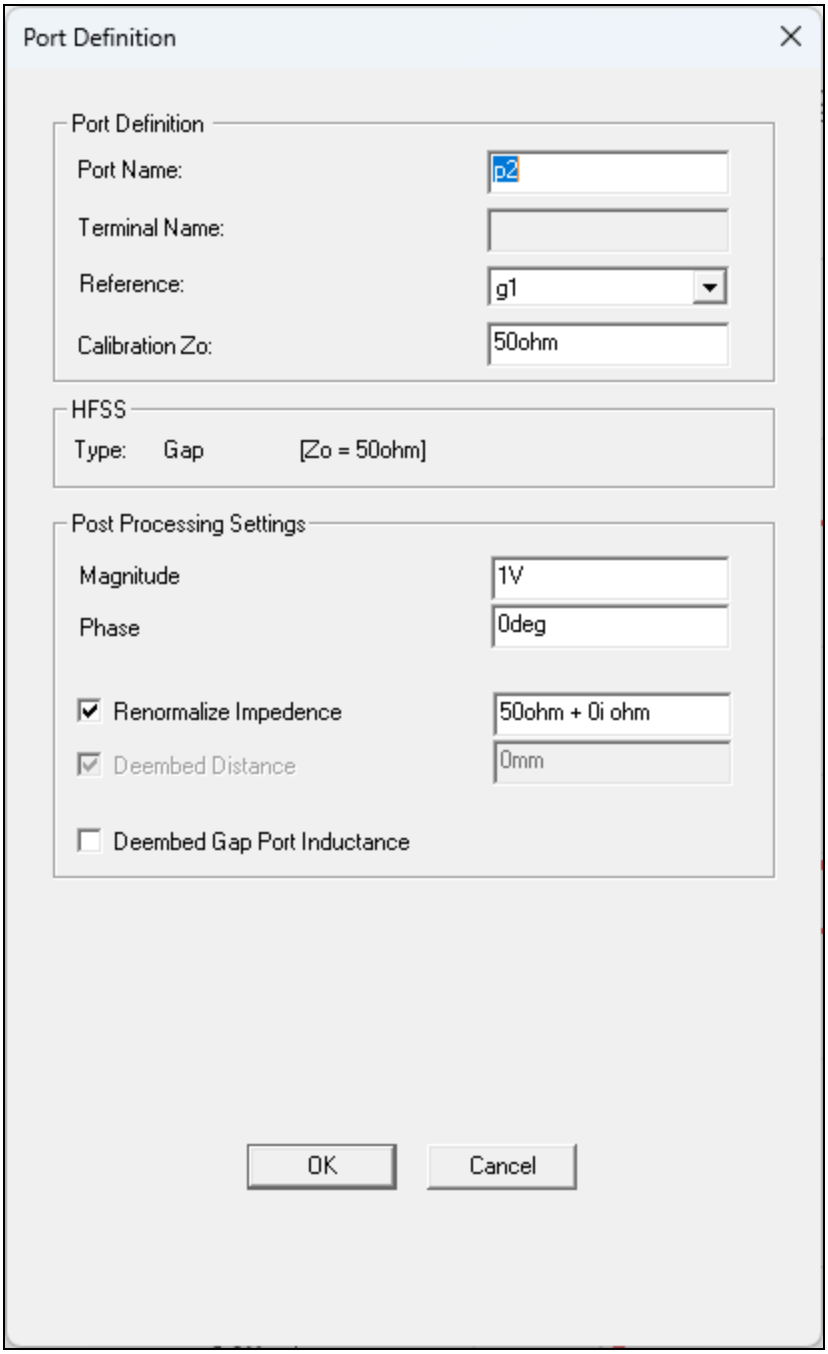
Before creating an overlay of the current results and animating it, alter the edge port definition for port **p2**. The default configuration applies an identical **1** volt magnitude excitation at 0 degrees phase angle to both of the added ports (**p1** and **p2**). Follow these steps to set the **p2** excitation to

0 volts, essentially making **p1** the input of the low pass filter and **p2** the output, for the purpose of the overlay.

1. From the **Project Manager** window, double-click **p2** to open the **Edge Port Definition** window.



The **Edge Port Definition** window provides the following configurable settings:



Port Definition area:

- **Port Name** specifies the port being defined
- **Terminal** is the name of the terminal (automatic, unfillable field)
- **Reference** is the **Port Solver** (**Reference**) used by the port

- **Calibration Zo** is the impedance (equivalent to the Full Port Impedance of the port (in Ohms))

HFSS area:

- **Type** of port (automatic, unfillable field)

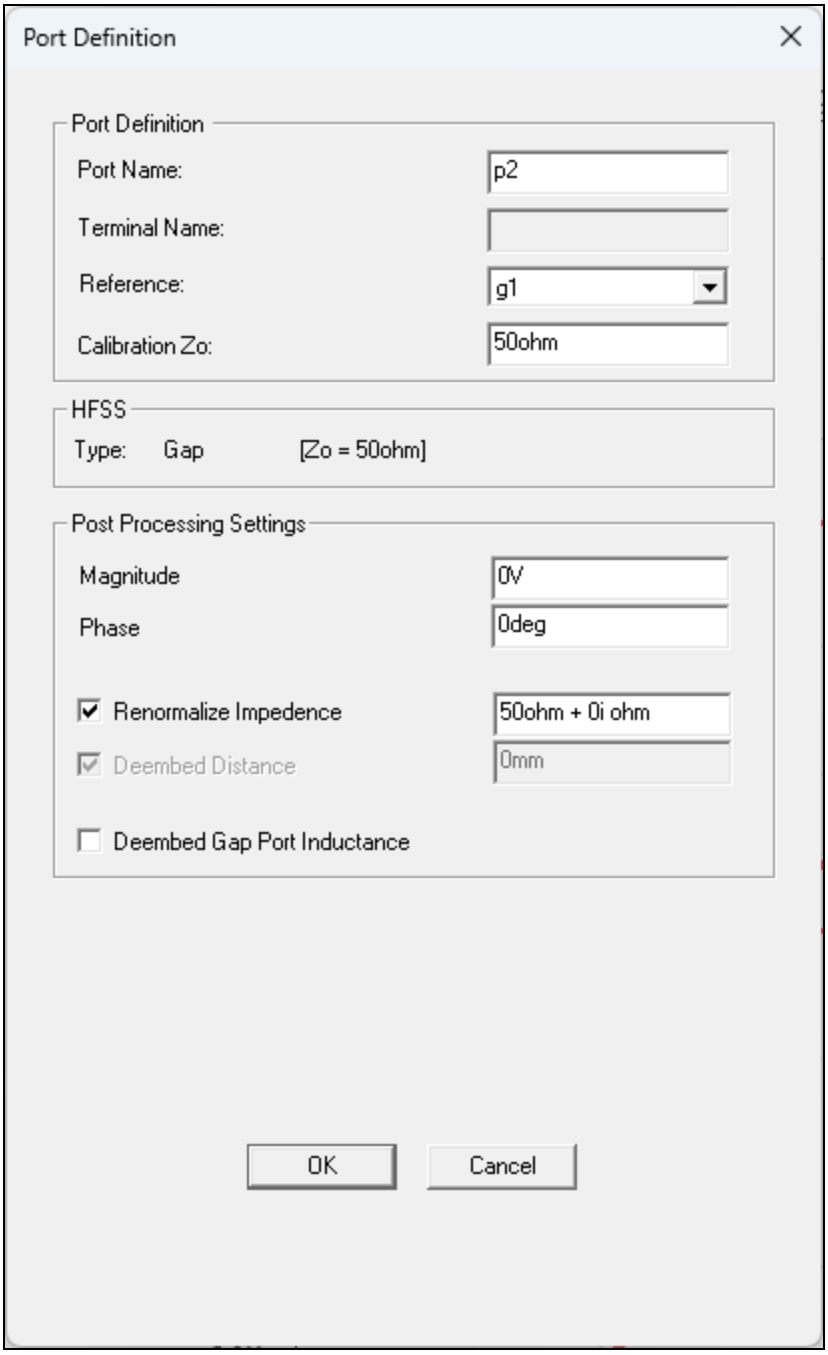
Post Processing Settings area:

- **Magnitude** is expressed in Volts (V)
- **Phase** is expressed in degrees (deg)
- **Renormalize Impedance** (check to use) is expressed in ohms

> **Note:**
>
> Even if **Renormalize Impedance** is checked, renormalization is ignored if it is set to **0**. De-embedding is still honored. However, a warning message appears for all ports with a **0** post-processing renormalization impedance: *Zero impedance on port '*<PortName>*' is ignored; renormalization is skipped for this port*.

- **Deembed Distance (PEM Only)** (check to use) is expressed in millimeters (mm)
- **Deembed Gap Port Inductance** (check to use) specifies the inductance of the port, which is calculated and cached as part of the solution data for subsequent use. During post-processing a list of the ports to deembed is used to calibrate the network data.

2. From the **Post Processing Settings**, enter **0V** in the **Magnitude** field:

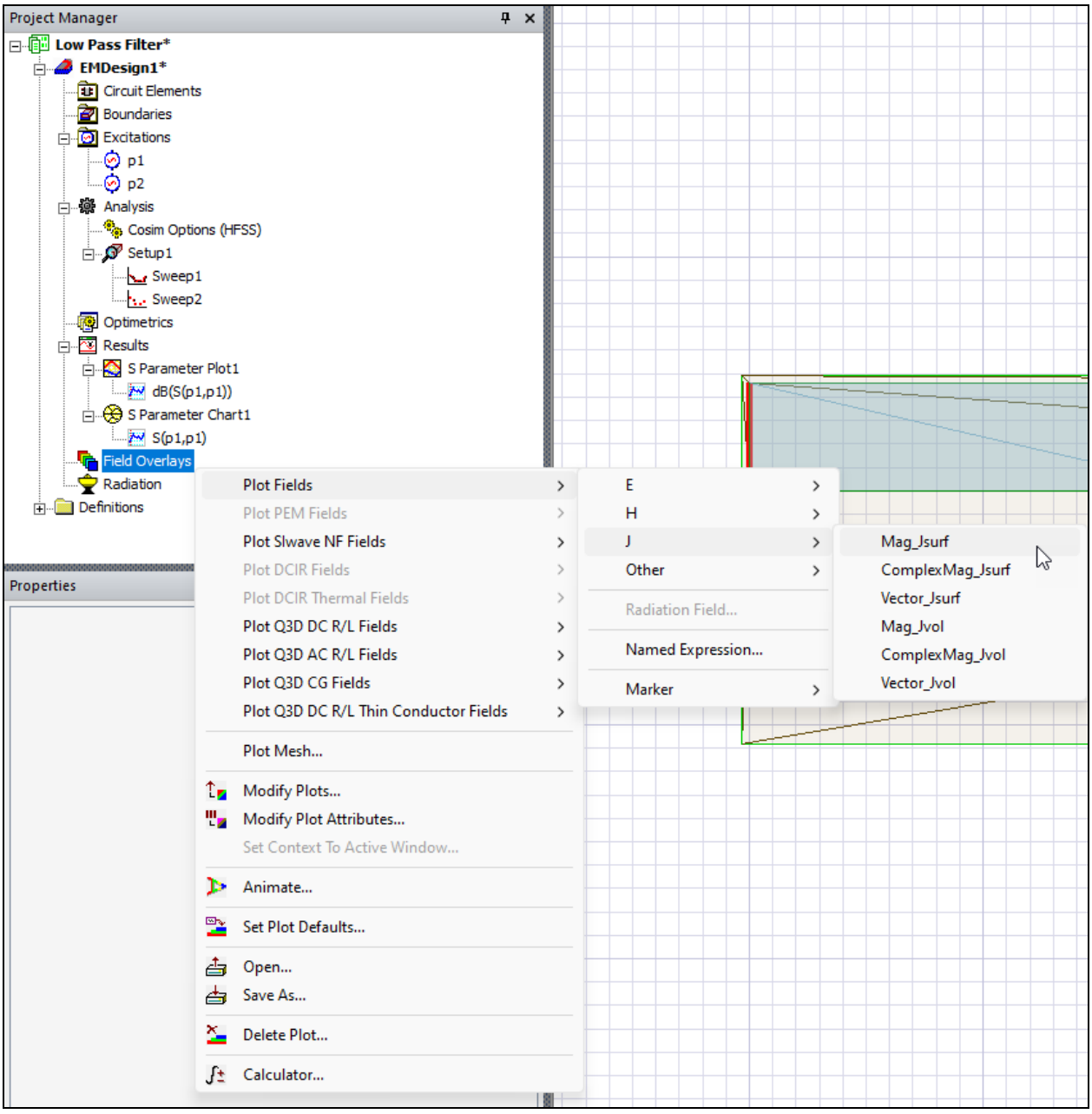


Continue to Overlay Current Results.

Use Field Overlays

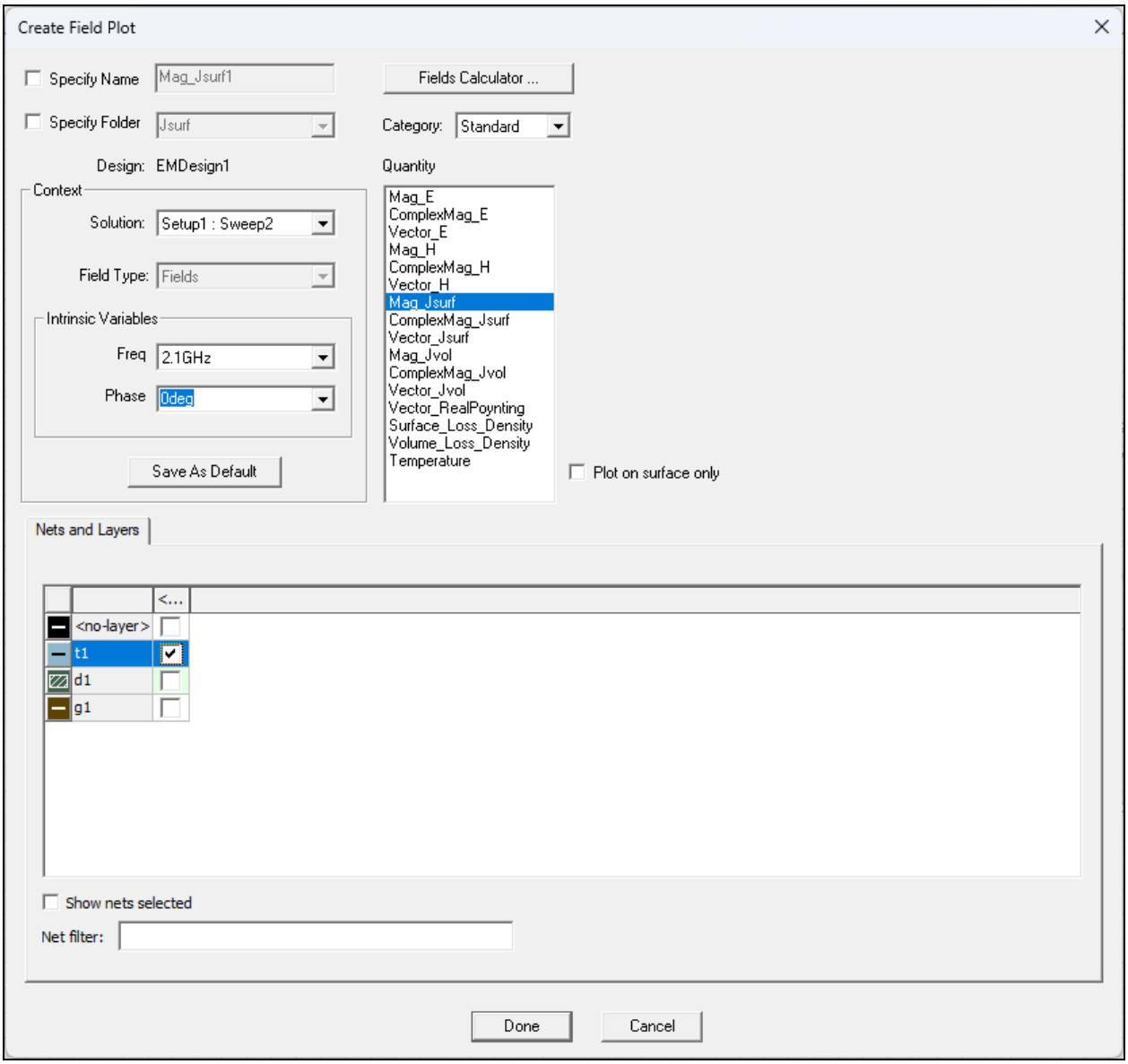
Follow these steps to overlay the magnitude of the surface current results from the trace layer on the low pass filter.

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

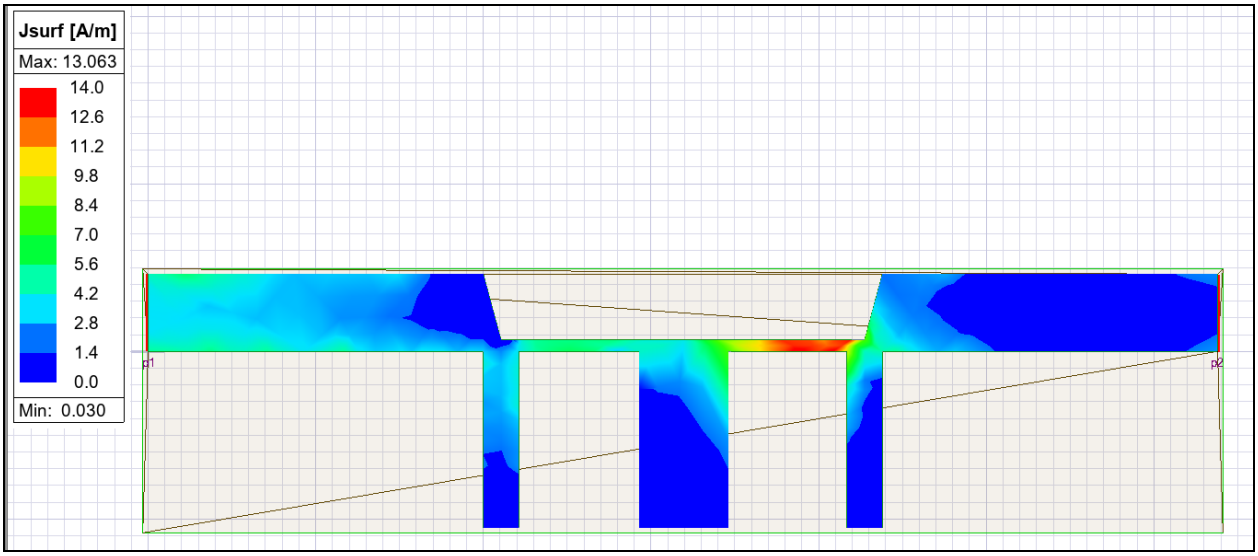


> **Note:**
>
> The only **Solution** available is **Setup1 : Sweep2**, since only the discrete sweep type (i.e., **Sweep2**) has the option to output currents.

2. From the **Intrinsic Variables** area, select **2.1GHz** from the **Freq** (i.e., frequency) drop-down menu. **2.1 GHz** most closely corresponds to the point of minimum return loss from the S Parameter plot.
3. Under the **Nets and Layers** tab, check the **t1** box.



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

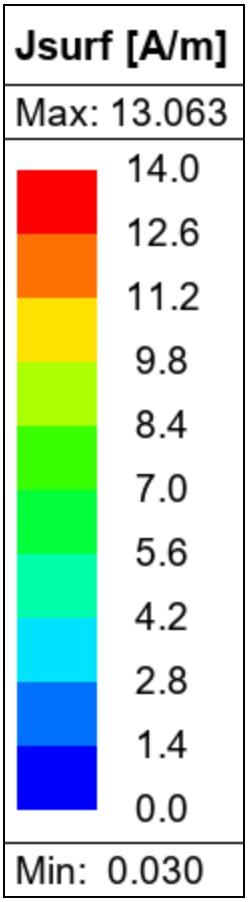


Continue to Modify and Animated Current Overlay.

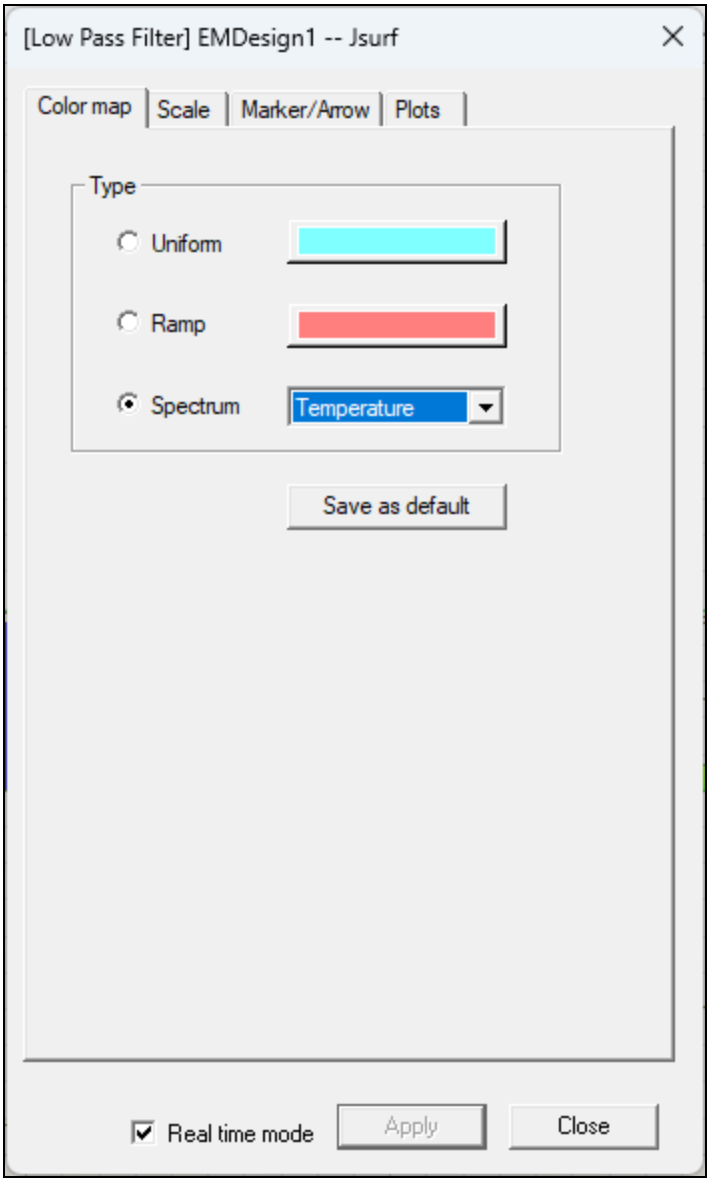
Modify and Animate the Current Overlay

Follow these steps to animate the current plot (i.e., **Jsurf**).

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



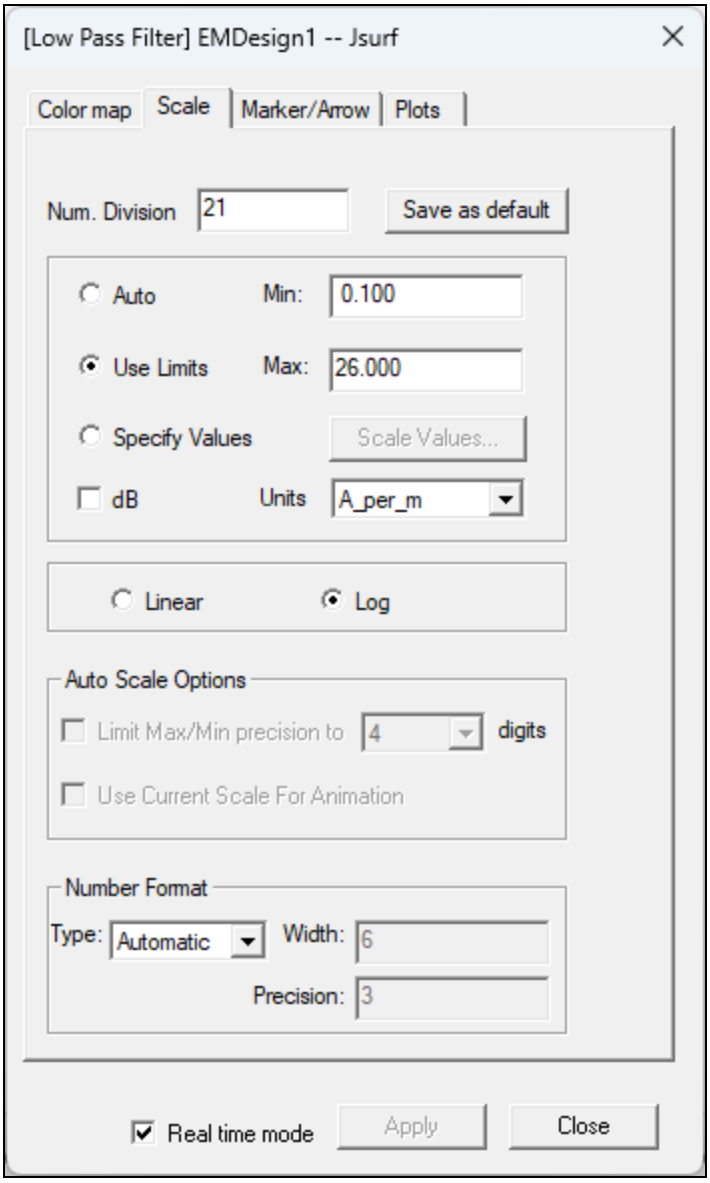
2. From the **Color map** tab, select **Temperature** from the **Spectrum** drop-down menu.



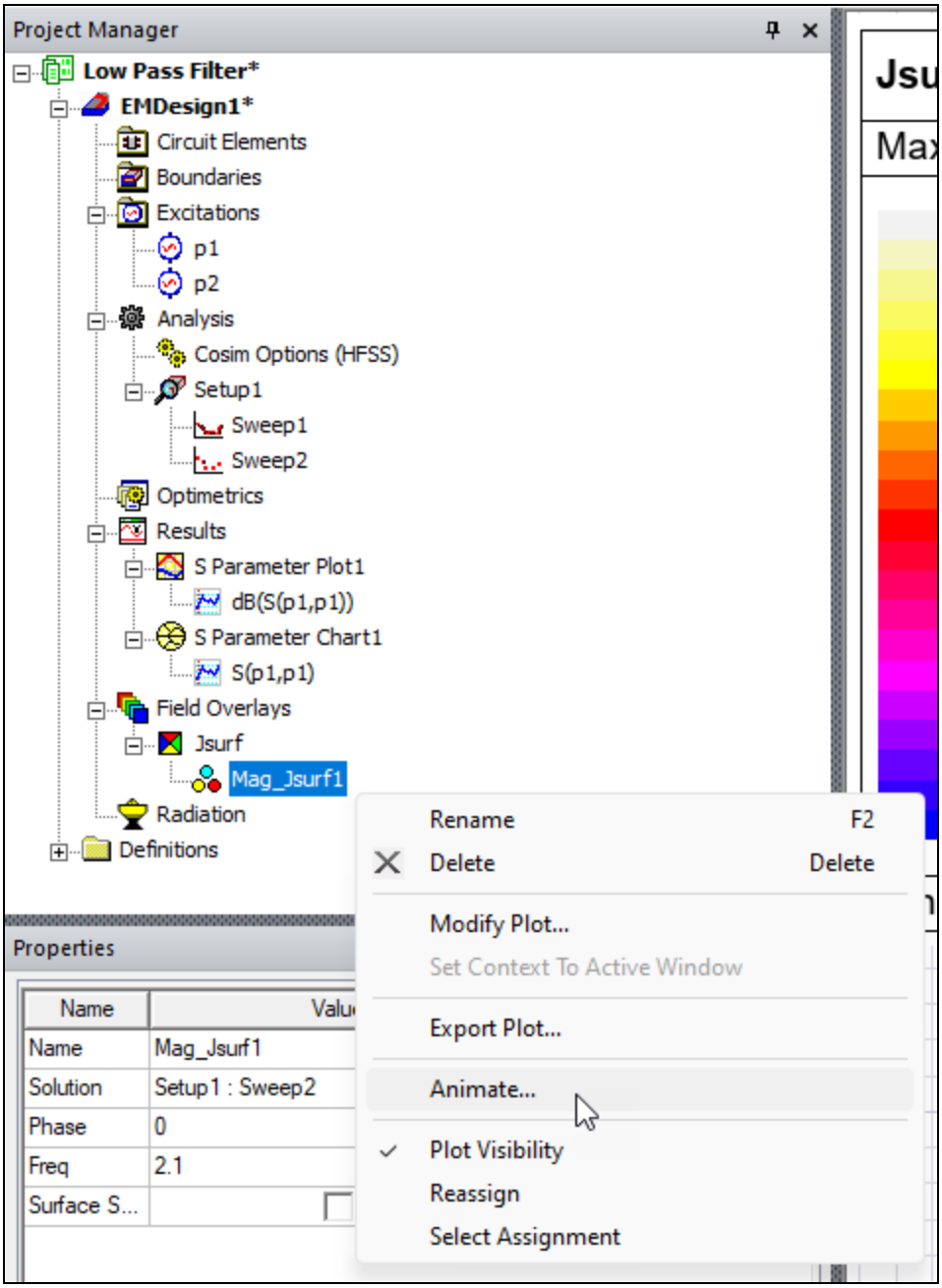
> **Note:**
>
> From this tab, the colors and scale of the plot legend can be modified, if necessary.

3. Select the **Scale** tab and do the following:
   a. Enter **21** in the **Num. Division** field.
   b. Select **Use Limits**.

c. Enter **0.100** in the **Min** field.
d. Enter **26.000** in the **Max** field.
e. Select **Log** to produce a logarithmic scale.
f. Click **Apply** to save changes.



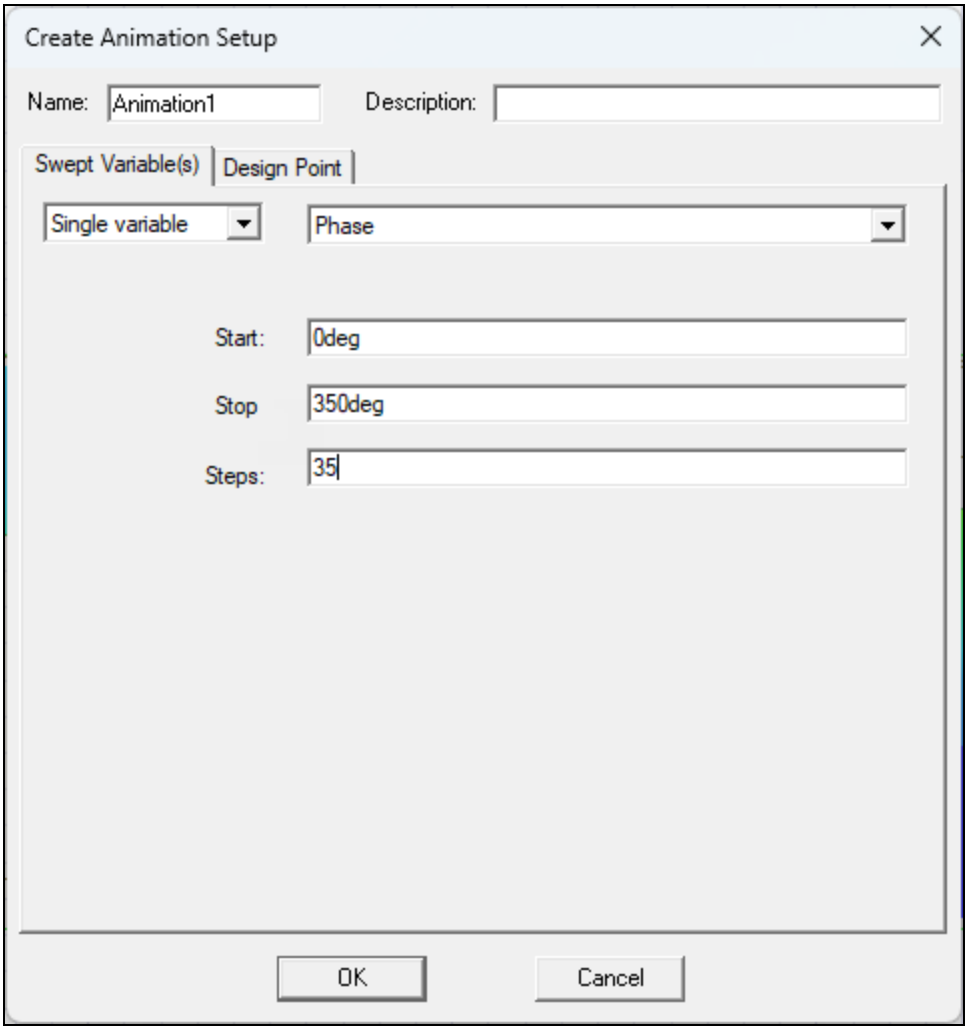
4. **Close** the plot settings window.
5. From the **Project Manager** window, expand **Field Overlays** > **JSurf**. Then right-click **Mag_Jsurf1** and click **Animate** to open the **Create Animation Setup** window.



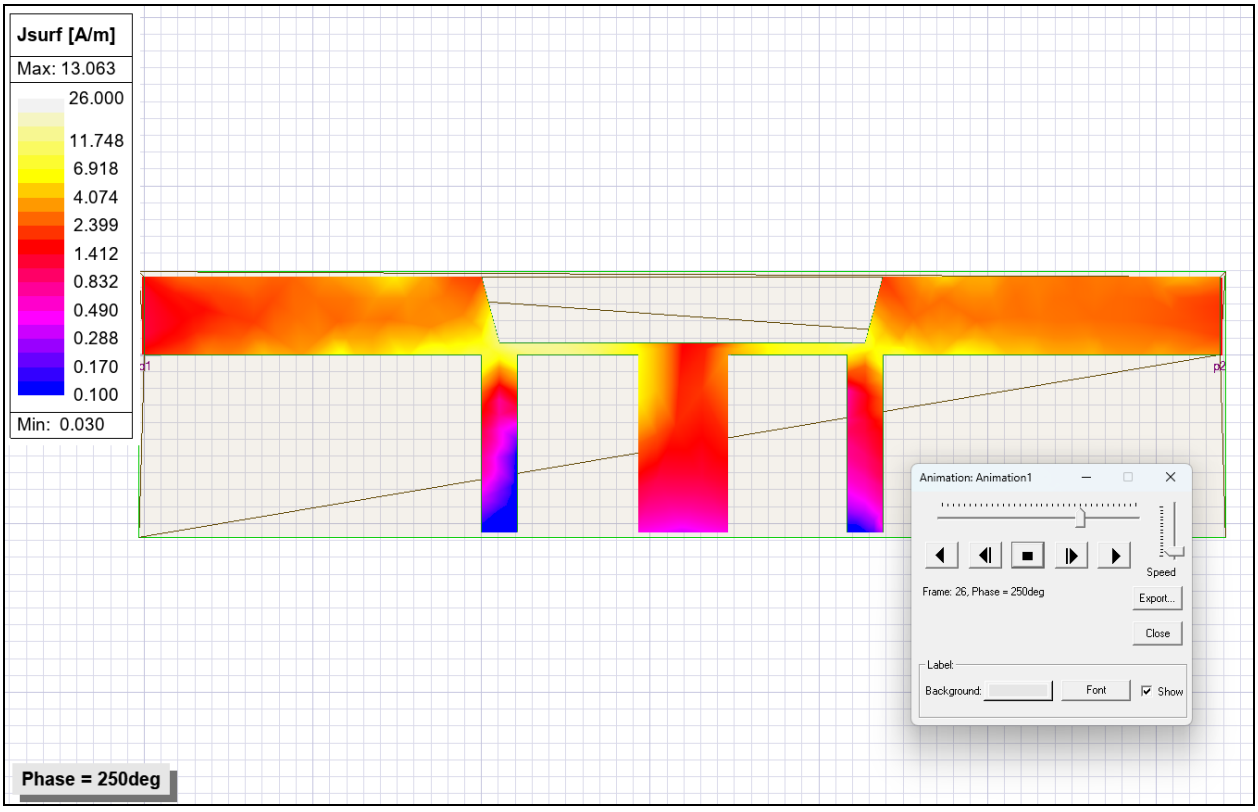
6. From the **Create Animation Setup** tab, do the following:
    a. Ensure **Single variable** and **Phase** are selected from the drop-down menus.
    b. In the **Stop** field, enter **350deg**.
    c. In the **Steps** field, enter **35**.

Note:

These settings result in an animation with 10° (**35deg** / **350deg**) phase increments. A 360° phase (**360deg**) would be identical to the 0° phase, resulting in a redundant animation frame, and is thus omitted.



7. Click **OK** to close the **Create Animation Setup** window, open an animation control panel, and start the animation in the **Layout Editor**.



8. Use the animation controls to reverse, stop, and change the speed of the animation, among other settings.
9. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.
10. From the animation control panel, click **Close** to end the animation.

Continue to Create Far Field Plot.

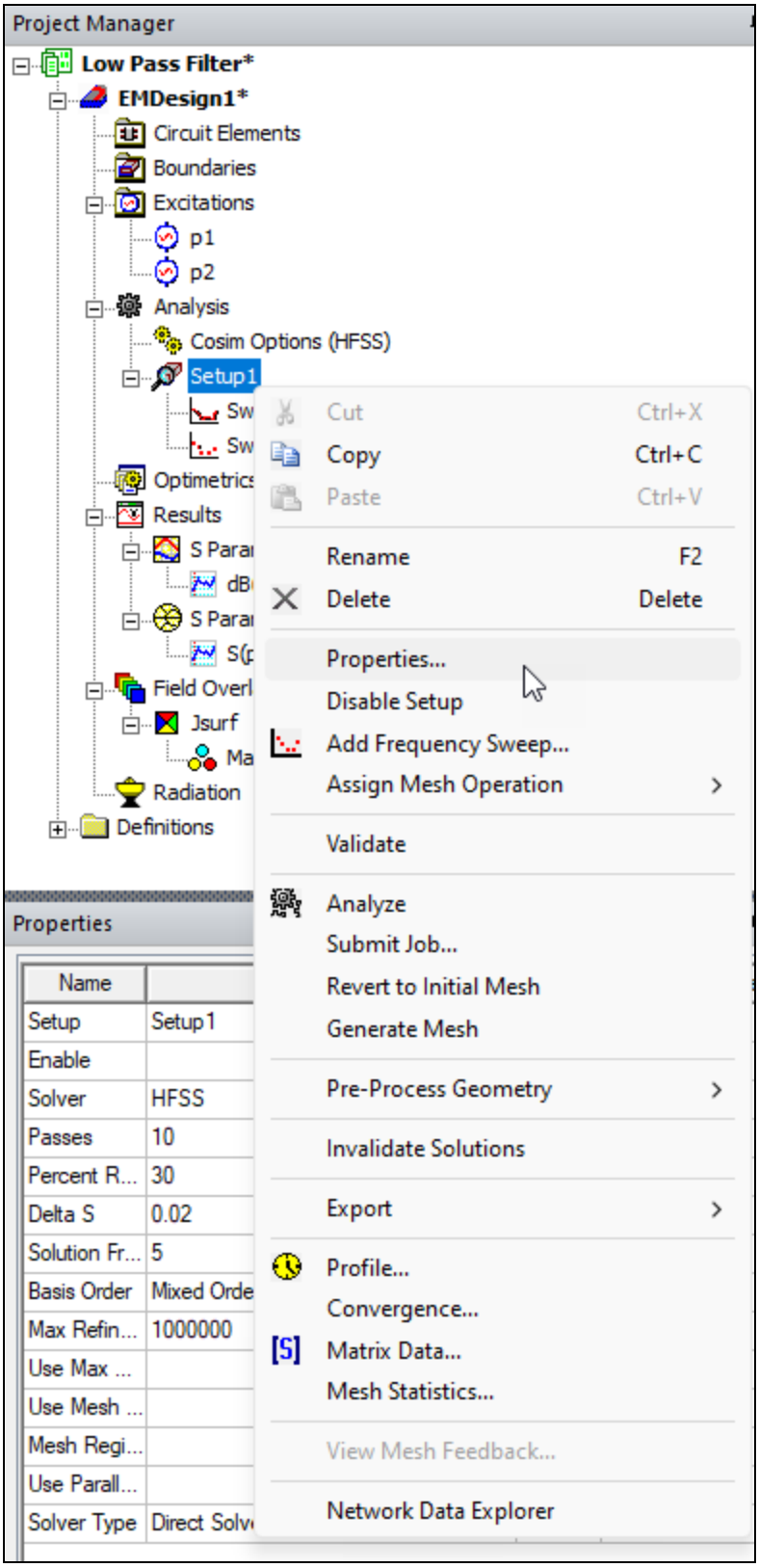
Create Far Field Plot

Far fields are usually not of interest when designing a filter, but the procedure is shown here for reference. Current outputs are required as the basis of far field computations. Follow these steps to save field results.

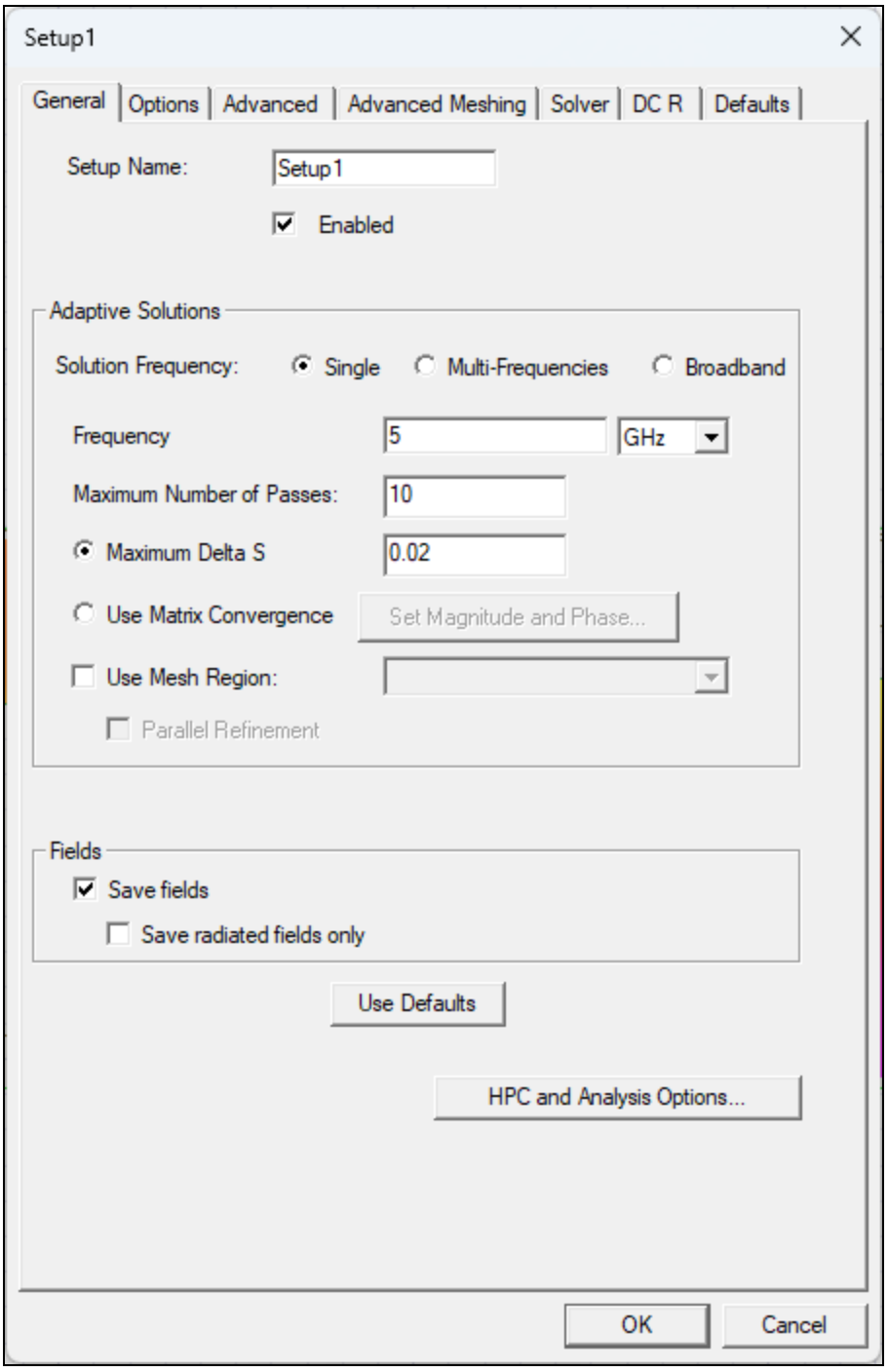
1. Follow these steps to **run a discrete frequency sweep**:
   a. From the **Options** area, ensure the **Save fields** box is checked.
   b. Specify multiple discrete frequency points and have the current and field results available for all specified frequencies.
   c. **Define the frequency at which the far field results** are viewed.

d. **Animate** the resulting plot. The animation should progress from one frequency to the next.

2. Right-click **Setup1** and select **Properties** to open the setup options window.



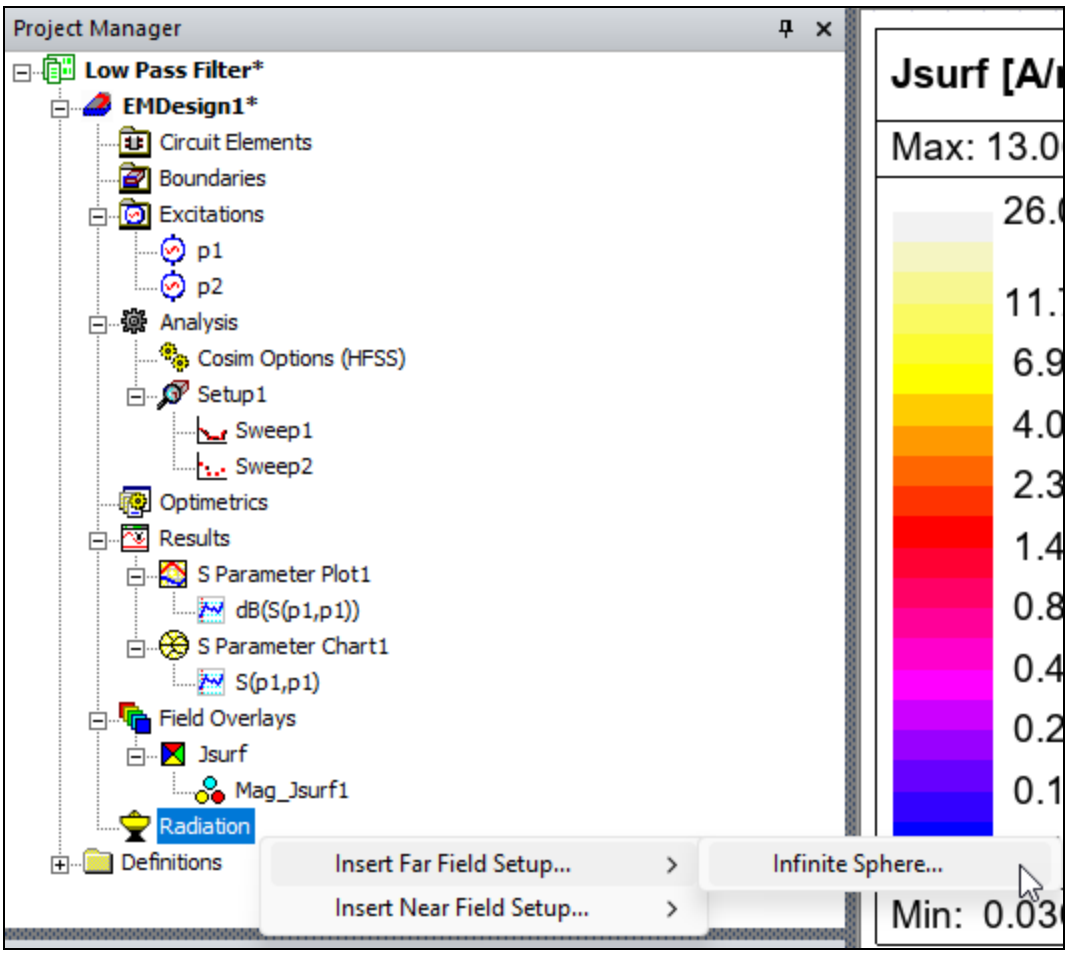
3. From the **Fields** area, check the **Save fields** box.



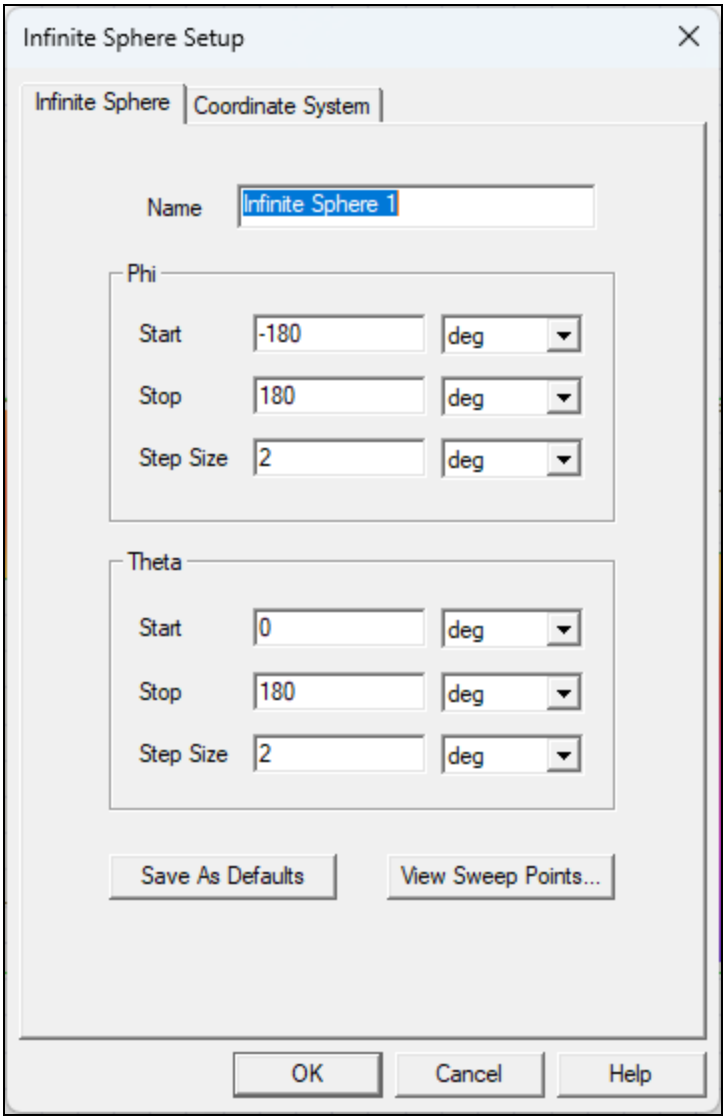
Plot Far Field Results

Follow these steps to set up a far field infinite sphere and plot the results.

1. From the **Project Manager** window, right-click **Radiation** and select **Insert Far Field Setup** > **Infinite Sphere**.

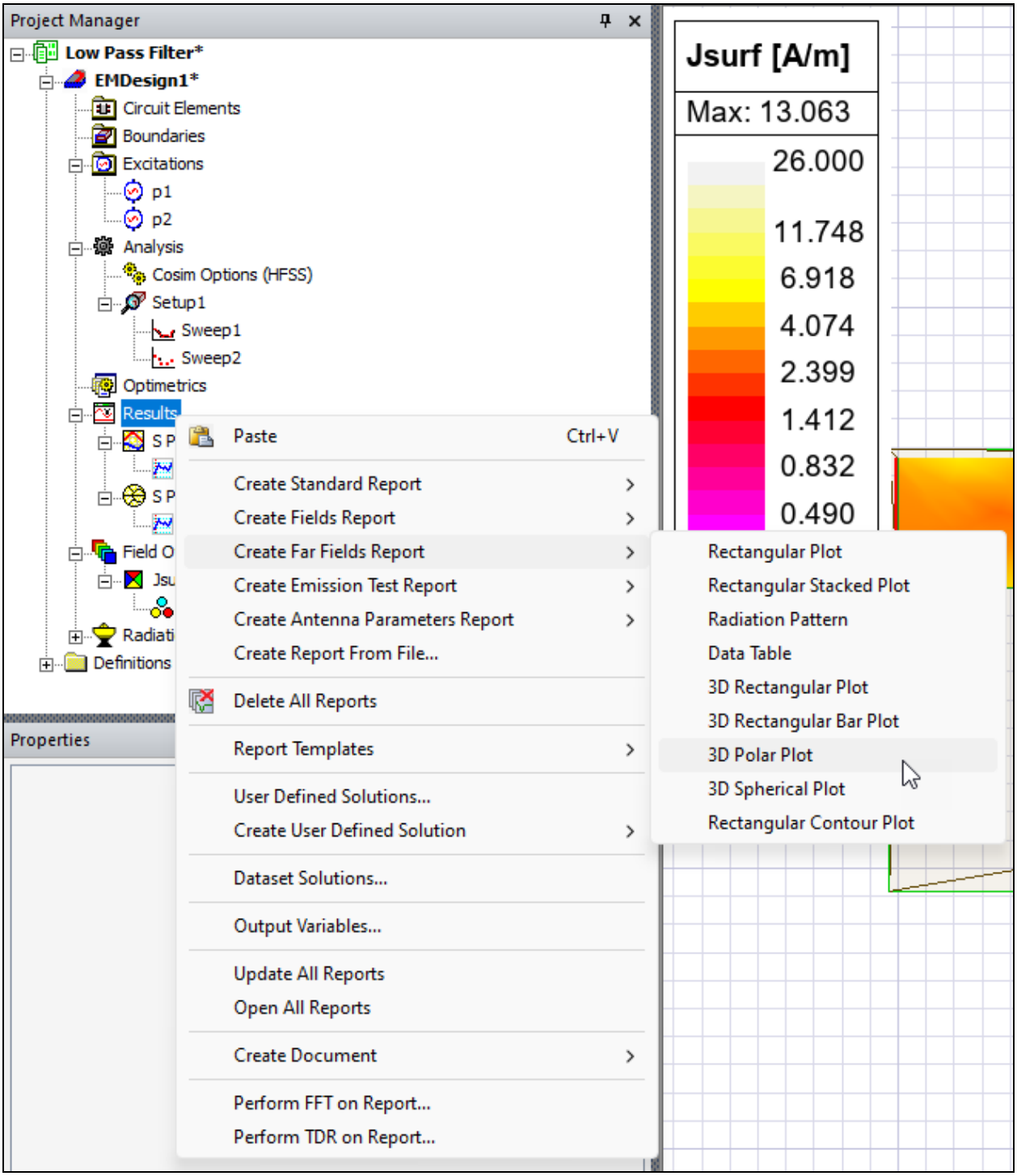


The **Infinite Sphere Setup** window opens.



2. Make **no** changes to the default configuration and click **OK** to close the **Infinite Sphere** window and add the far field setup to the design.
3. Right-click **Results** and select **Create Far Fields Report** > **3D Polar Plot** to open the

Report window.

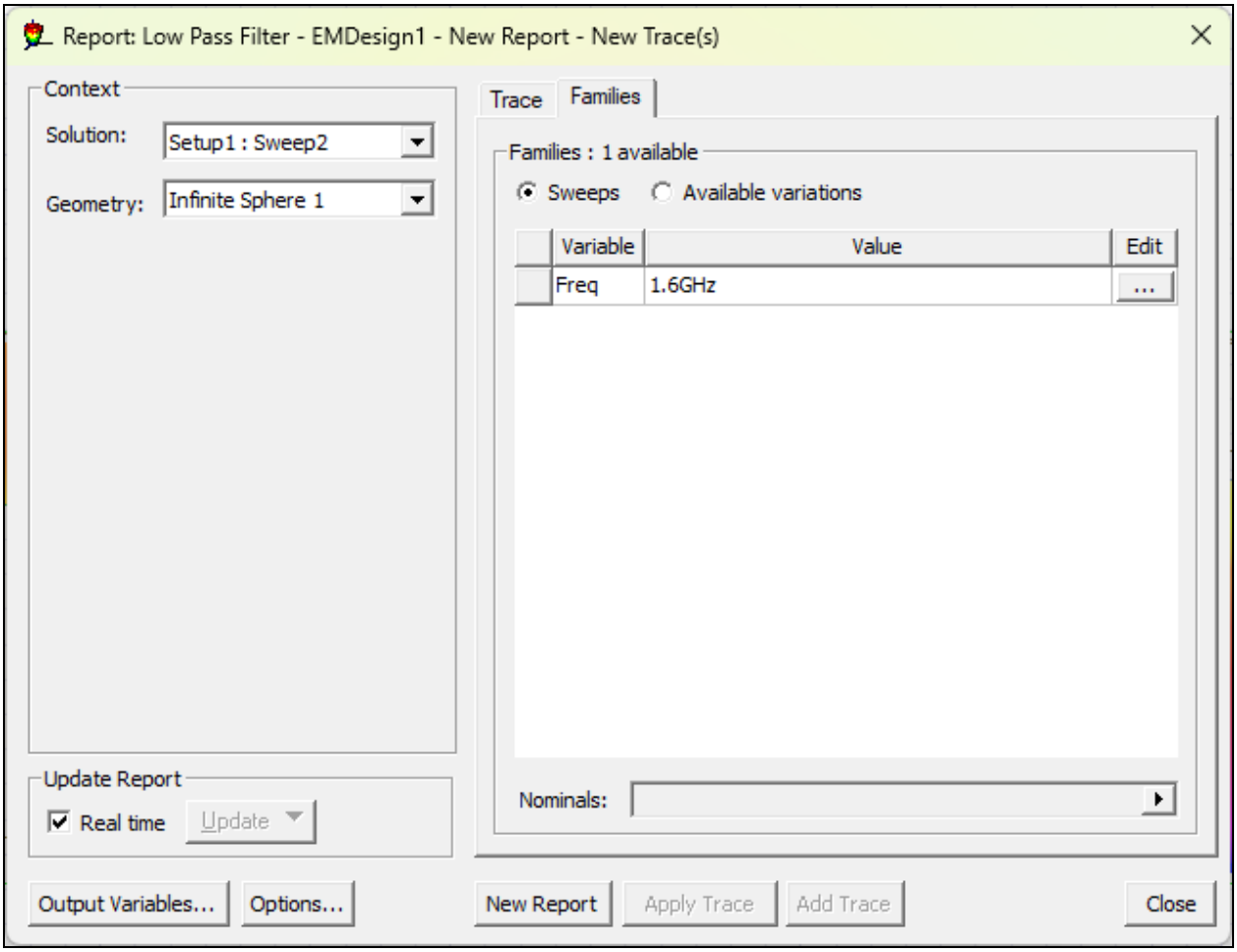


Note:

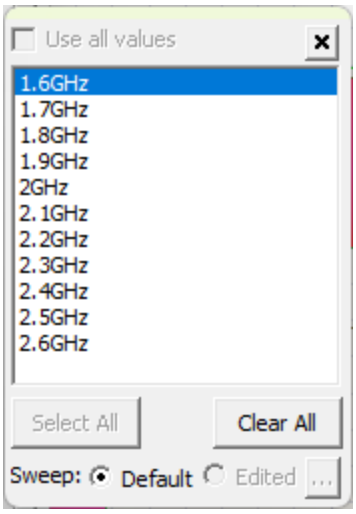
In Step 4, the **Solution** and **Geometry** drop-down menus pre-select the only available choices that provide far field results (i.e., **Setup1 : Sweep2** is selected from the **Solution** drop-down menu, and **Infinite Sphere 1** is selected from the **Geometry** drop-down menu).

4. Select or ensure the following options are highlighted in the **Category**, **Quantity**, and **Function** lists:
    - **Category**: **rE**
    - **Quantity**: **rETotal**
    - **Function**: **<none>**

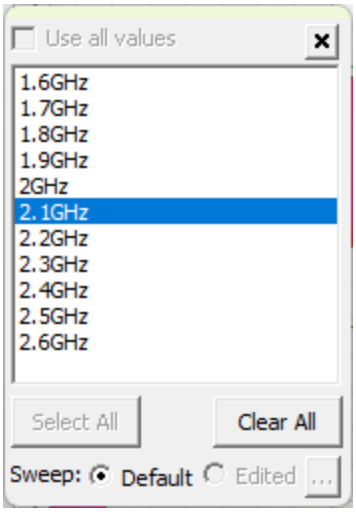
5. Navigate to the **Families** tab.



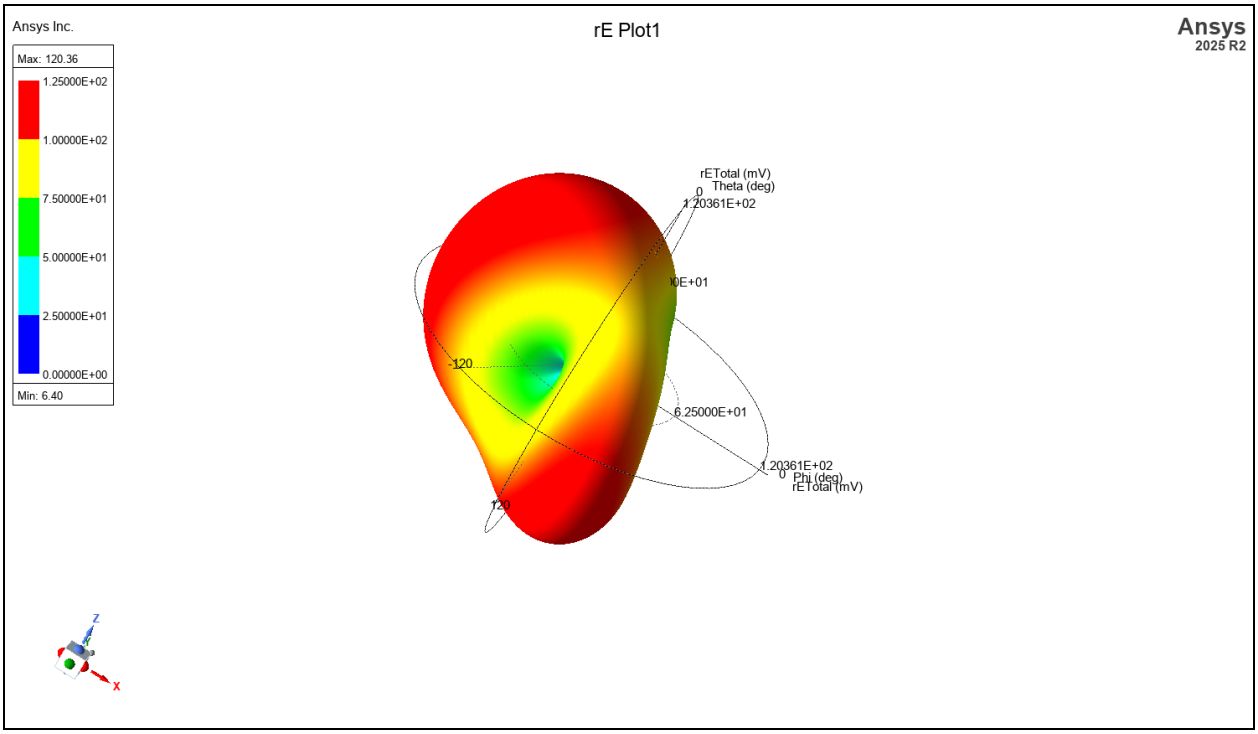
6. From the **Families** tab > **Freq** row, click the button in the **Edit** column to open a value list.



7. Click **2.1GHz**, which corresponds closest to the point of minimum return loss. Then click outside of the value list to close it.



8. From the **Report** window, click **New Report** and the far field plot opens under the **Report** window. **Close** the **Report** window to access the plot.
9. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** the far field plot using the standard **Layout Editor** controls.



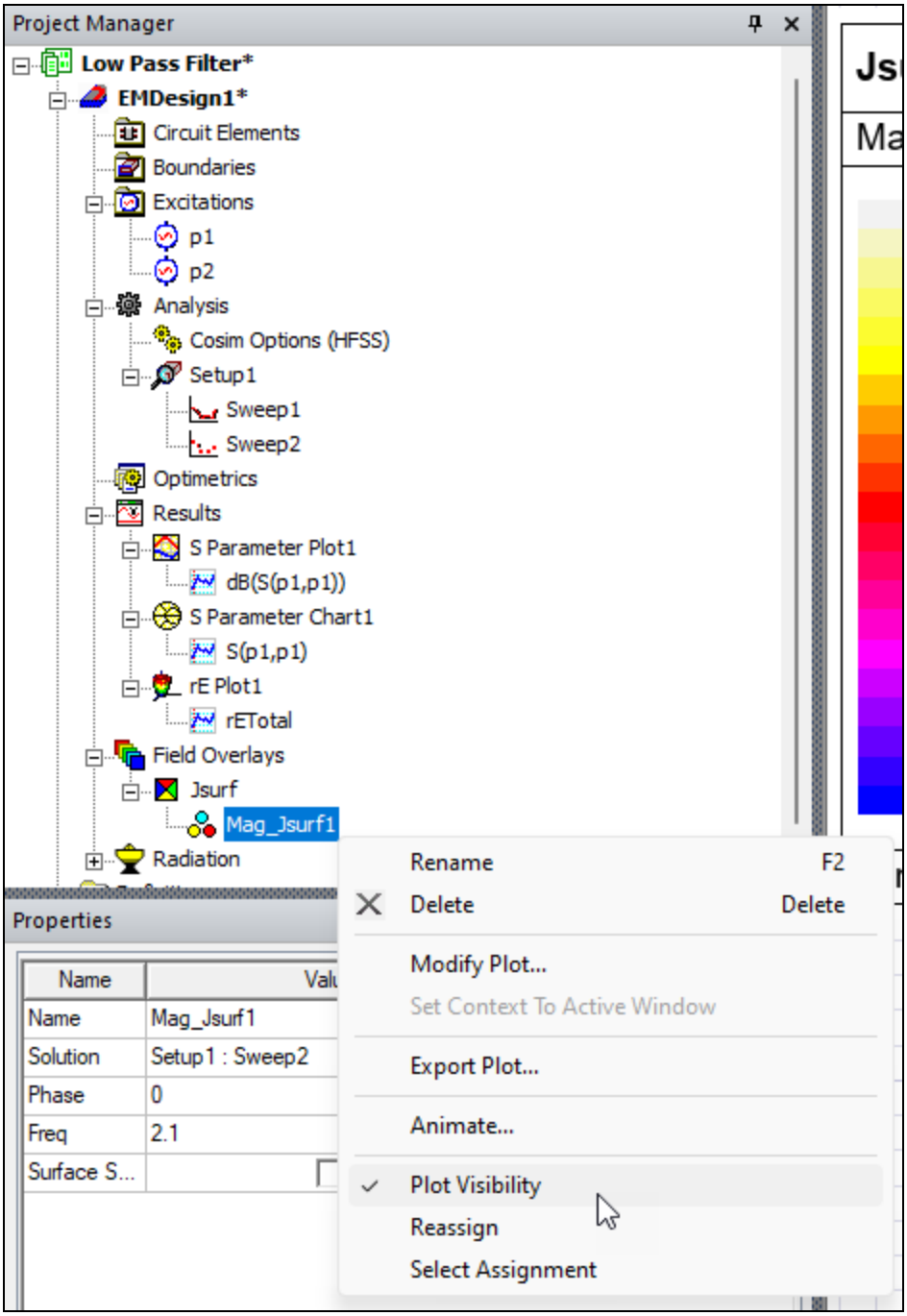
Continue to Overlay Far Field Plot on Model Geometry.

Overlay a Far Field Plot on Model Geometry

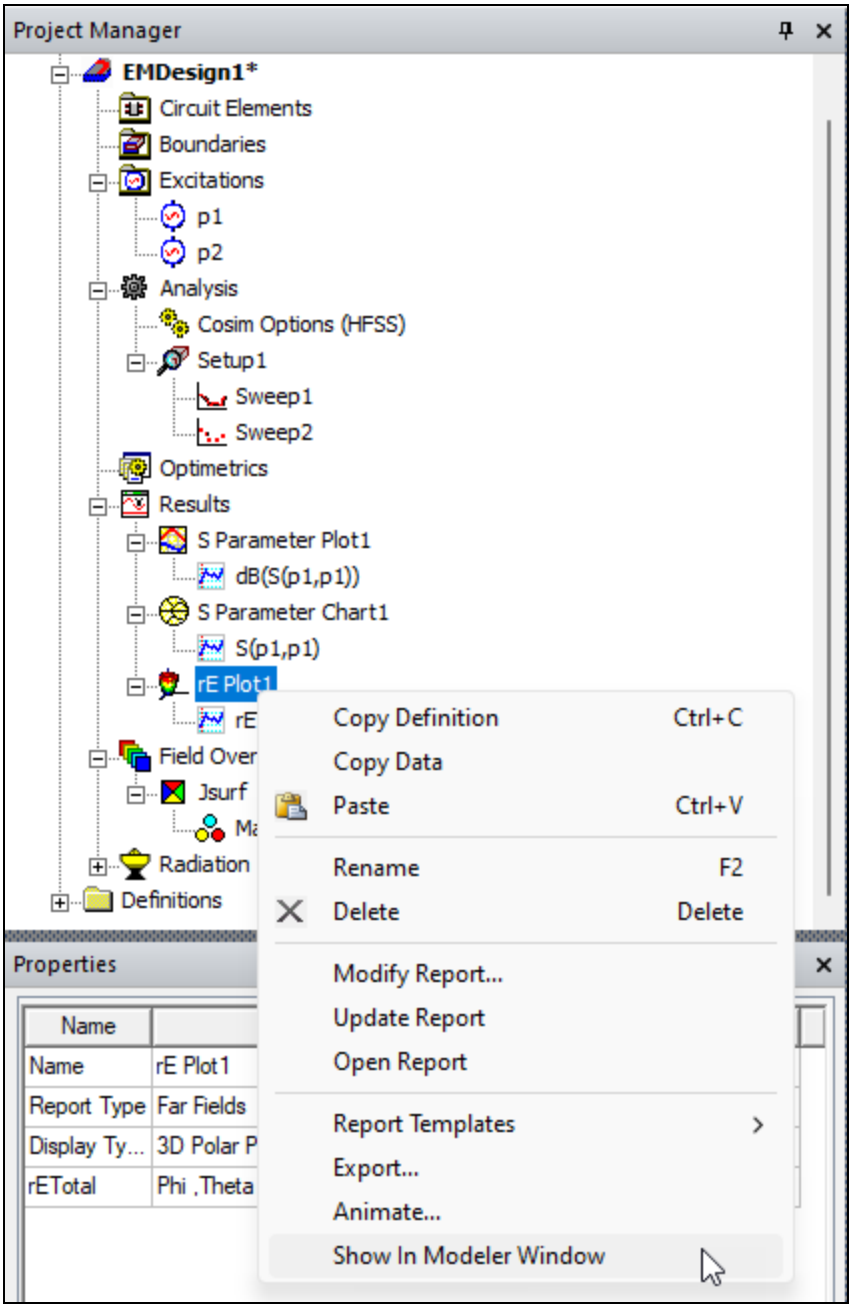
To better see how the far field pattern relates to the low pass filter geometry, follow these steps to overlay the pattern on the model in the **Layout Editor**.

> **Note:**
>
> If necessary, refocus on the field overlay by expanding the **Project Manager** window > **Project Tree** > **[*active design folder*]** > **Field Overlays**. Then double-click **Jsurf**.

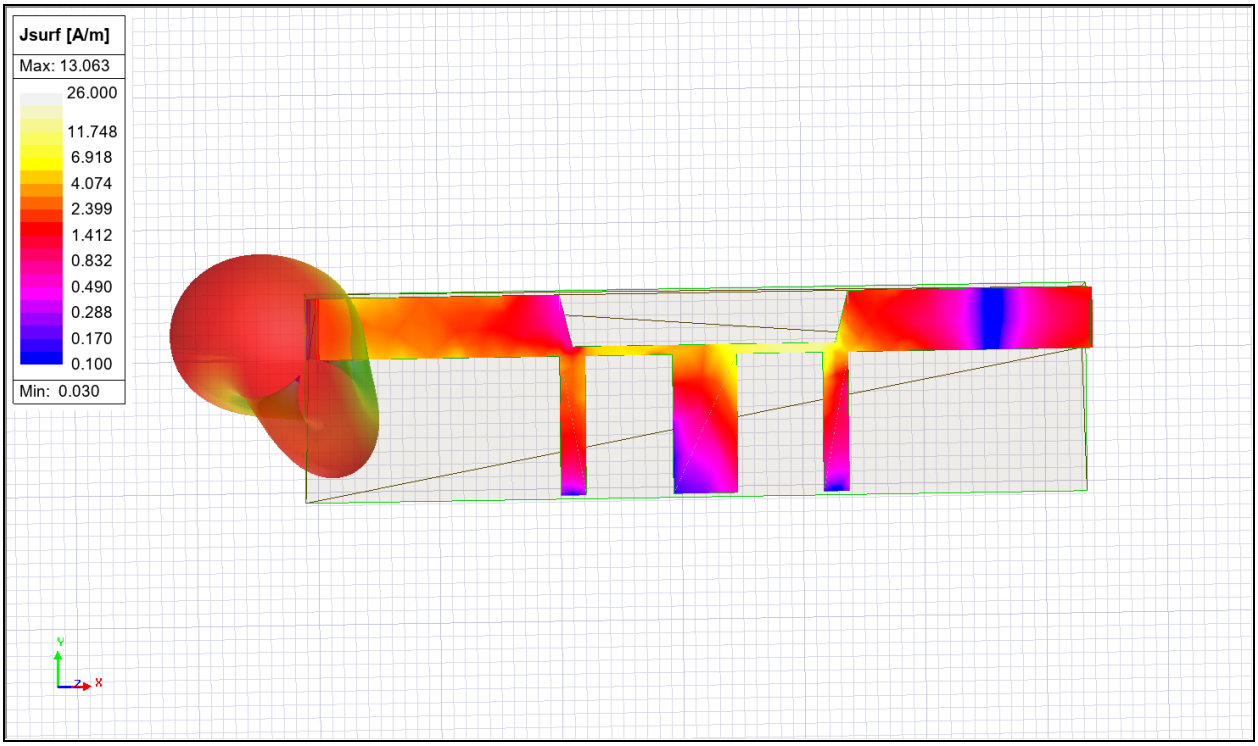
1. Avoid confusion about the values and units of the far field plot overlay by deactivating the legend and color contour map. From the **Project Manager** window, right-click **Mag_ Jsurf1** and select **Plot Visibility** to remove the check. The overlay disappears from the design in the **Layout Editor**.



2. From the **Project Manager** window, right-click **rE Plot1** and select **Show In Modeler Window** to overlay the far field plot on the design.



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



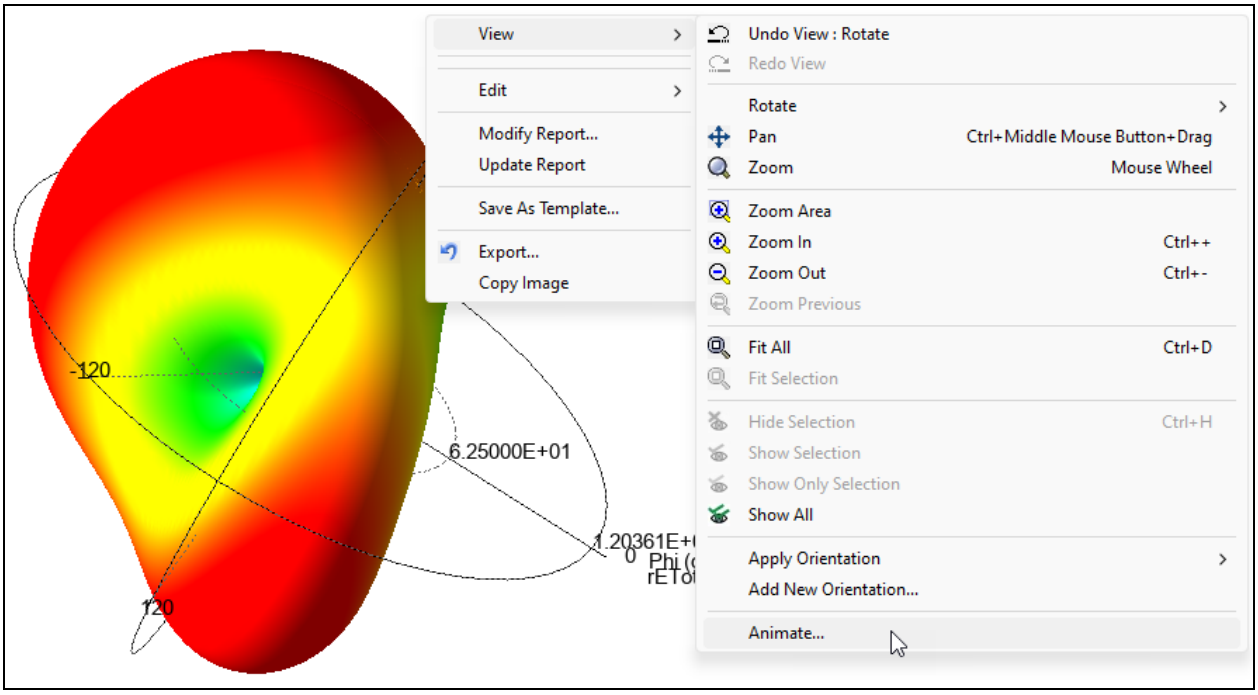
Continue to **Frequency Animated Far Field Plot**.

Animate a Field Plot

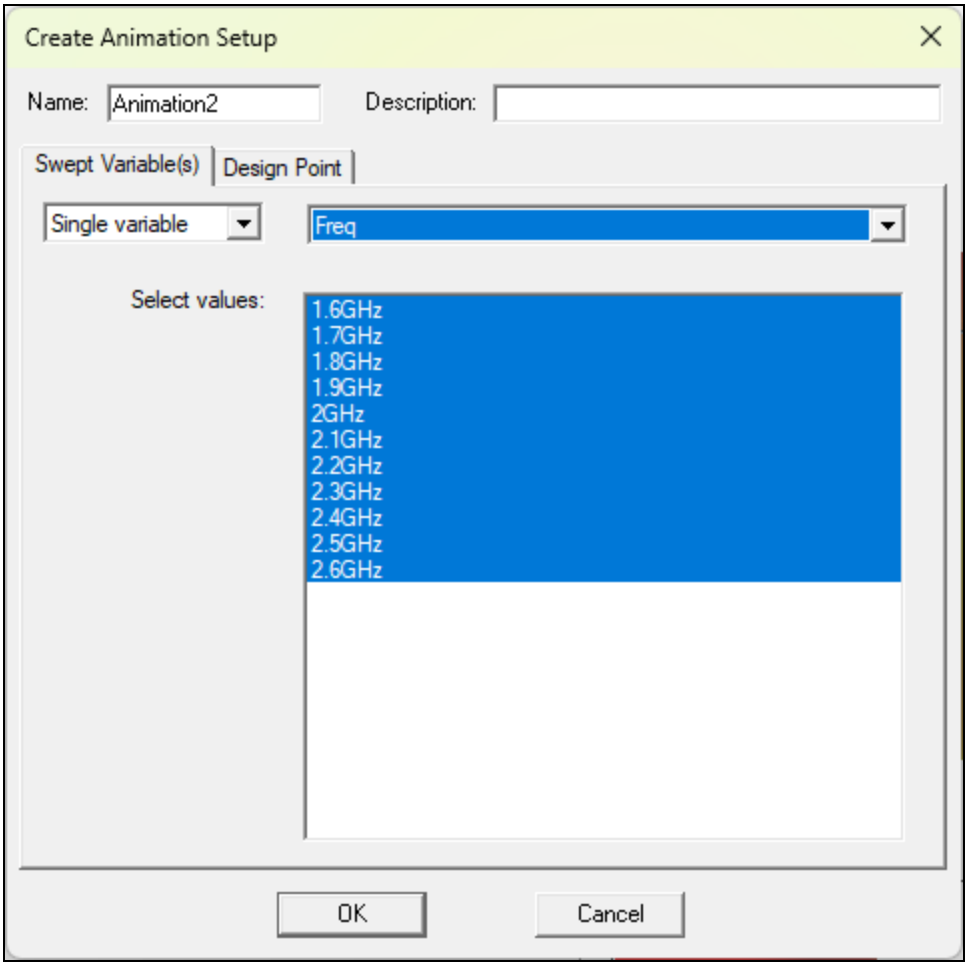
Follow these steps to animate the far field plot to see how fields vary with frequency.

> **Note:**
>
> If necessary, refocus on the far field plot. From the **Project Manager** window, double-click **rE Plot1**.

1. Right-click in the **Far Field Plot 1** and select **View** > **Animate** to open the **Create Animation Setup** window.

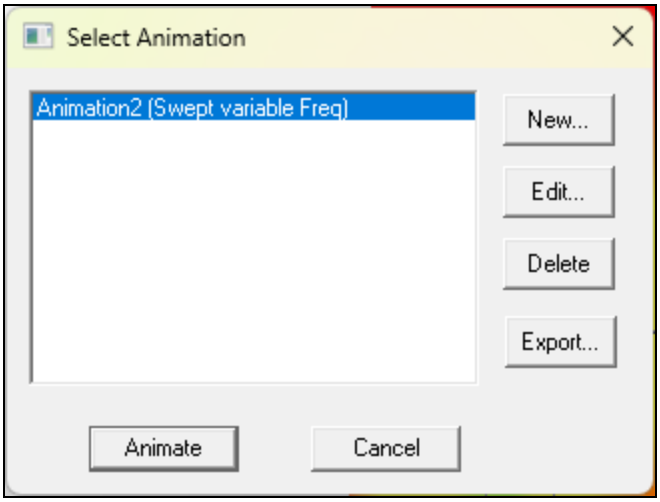


2. Ensure the settings in the **Create Animation Setup** window match the following image.



> **Note:**
>
> These settings result in an animation with 10° (**35deg** / **350deg**) phase increments. A 360° phase (**360deg**) would be identical to the 0° phase, resulting in a redundant animation frame, and is thus omitted.

3. Click **OK** to close the **Create Animation Setup** window and open the **Select Animation** window.



4. Click **OK** to close the **Create Animation Setup** window and start the animation in the **View** tab. Simultaneously, an animation control panel opens.
5. Use the animation controls to **Reverse**, **Stop**, and change the speed of the animation, as needed.
6. If applicable, **click+drag** the frequency legend to a more desirable location.
7. From the animation control panel, click **Close** to end the animation.
8. **Save** the design, either by navigating to **File** > **Save** or clicking the **Save** button on any of the ribbons.

Congratulations, the HFSS 3D Layout low pass filter getting started guide is complete.