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 2024 R2
July 2024

Conventions Used in this Guide

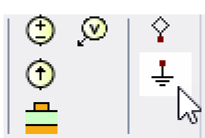
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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

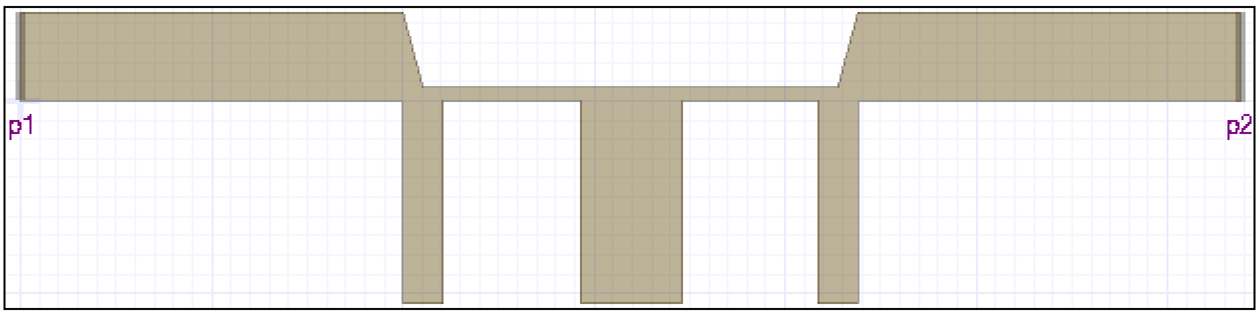
To access help from the user interface, press **F1** to open the chosen help for the active product.

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.

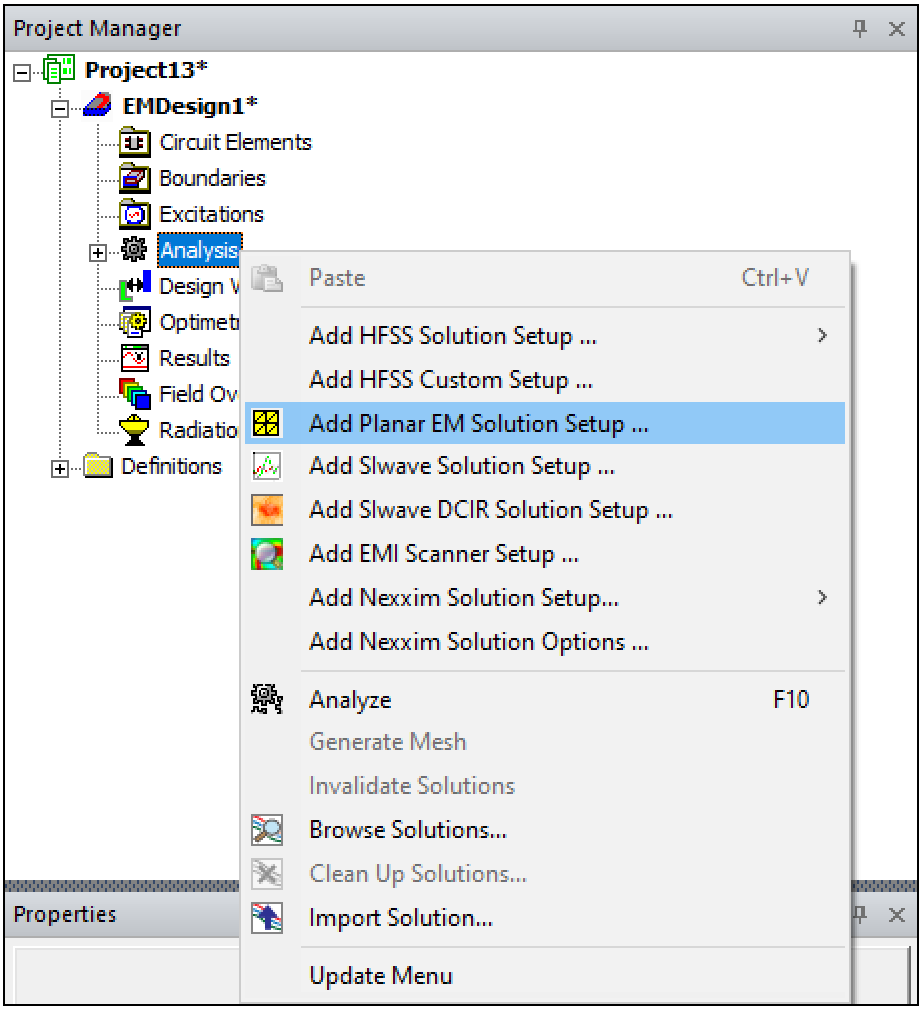
Table of Contents

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 as a planar EM solution. 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.



HFSS 3D Layout offers several design simulators for HFSS, Planar EM, SIwave, EMI, and Nexxim. In this topic, learn about the *PlanarEm Solution Setup only*.



EM Design simulators are the ideal tools for projects that involve full-wave or radiative effects for multilayered structures. For example, draw the physical layout of a patch antenna or a millimeter-wave integrated circuit and simulate the electromagnetic properties to display the following:

- Radiated electric fields
- Basic electromagnetic field quantities
- Characteristic port impedances and propagation constants
- Basic far-field parameters for electromagnetic fields and antennas
- Generalized *S*-parameters, and *S*-parameters renormalized to specific port impedances

For more information see the HFSS 3D Layout Simulator in the main help.

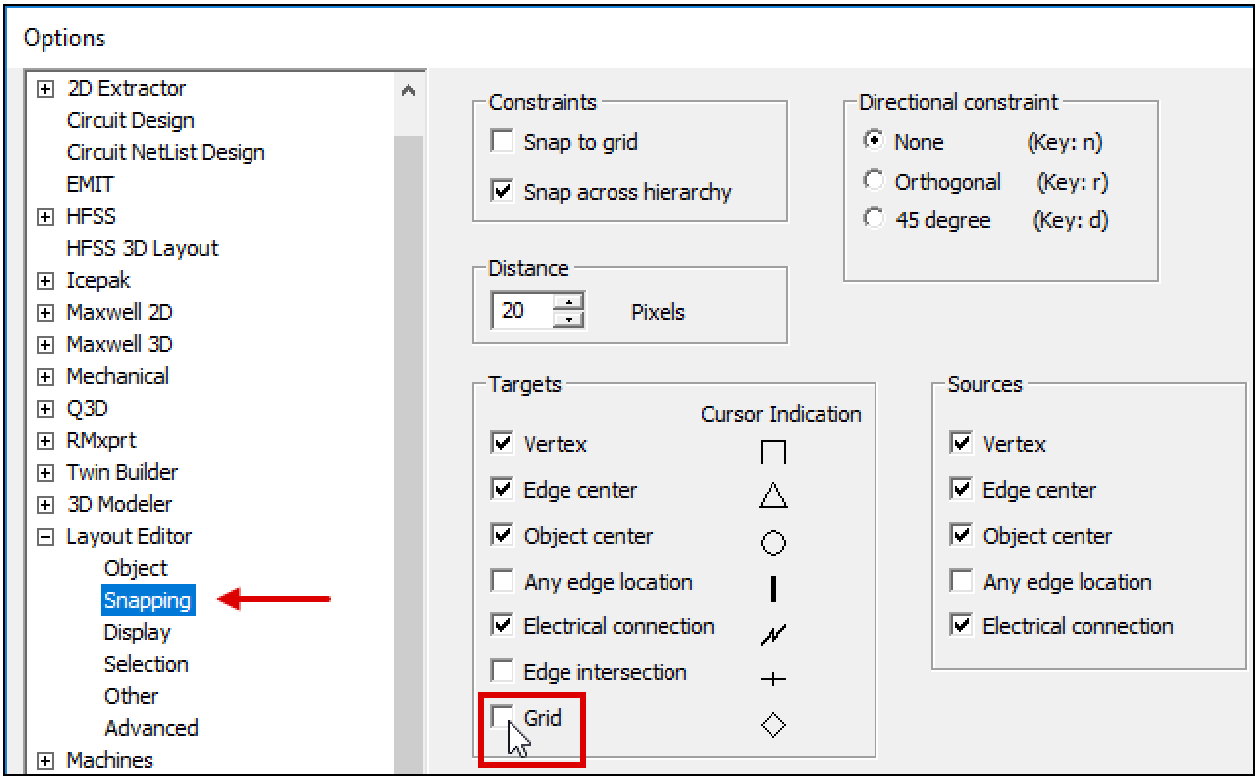
The following topics cover the physical design and EM analysis of a low-pass filter, specifically the following subtopics:

- How to start **Electronics Desktop** and explore the HFSS 3D Layout tools
- How to use the HFSS 3D Layout 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

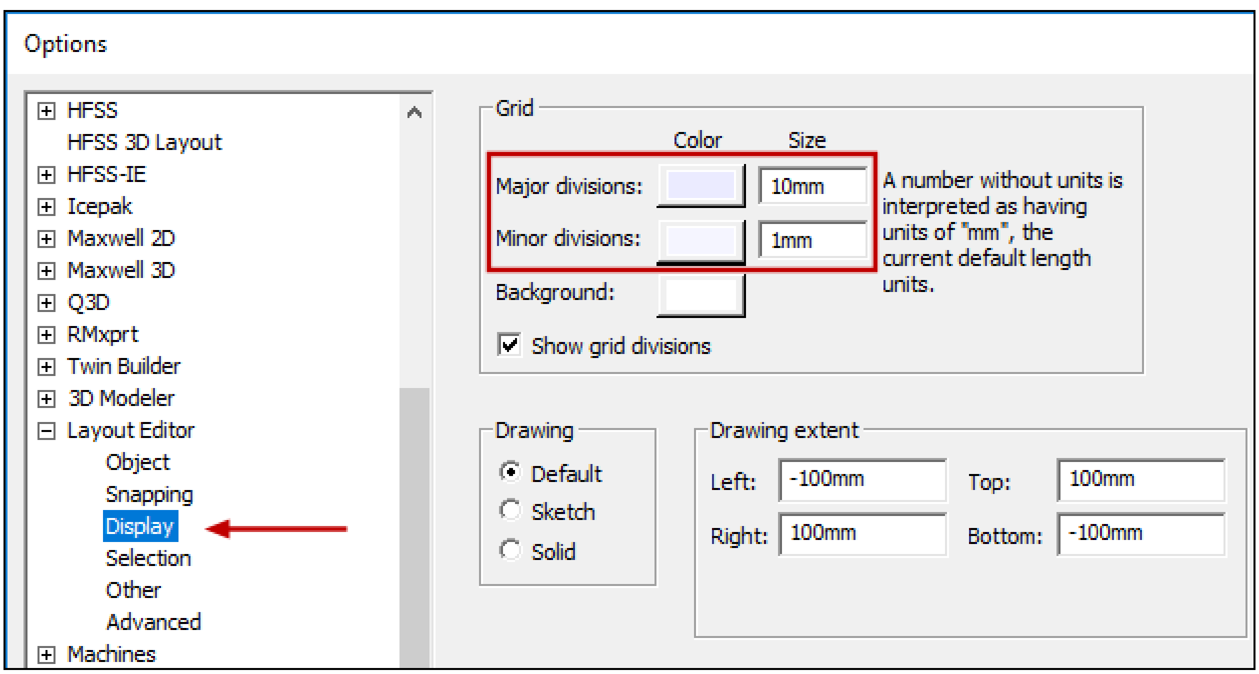
Setting General Options

Before inserting an HFSS 3D Layout Design into the project, ensure the **Electronics Desktop** options are set appropriately 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. Ensure **mm** is selected on 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. Under *Targets*, 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** field, enter **10 mm** for **Major** and **1 mm** for **Minor**, making sure the unit used for each is millimeter (mm), the default setting.

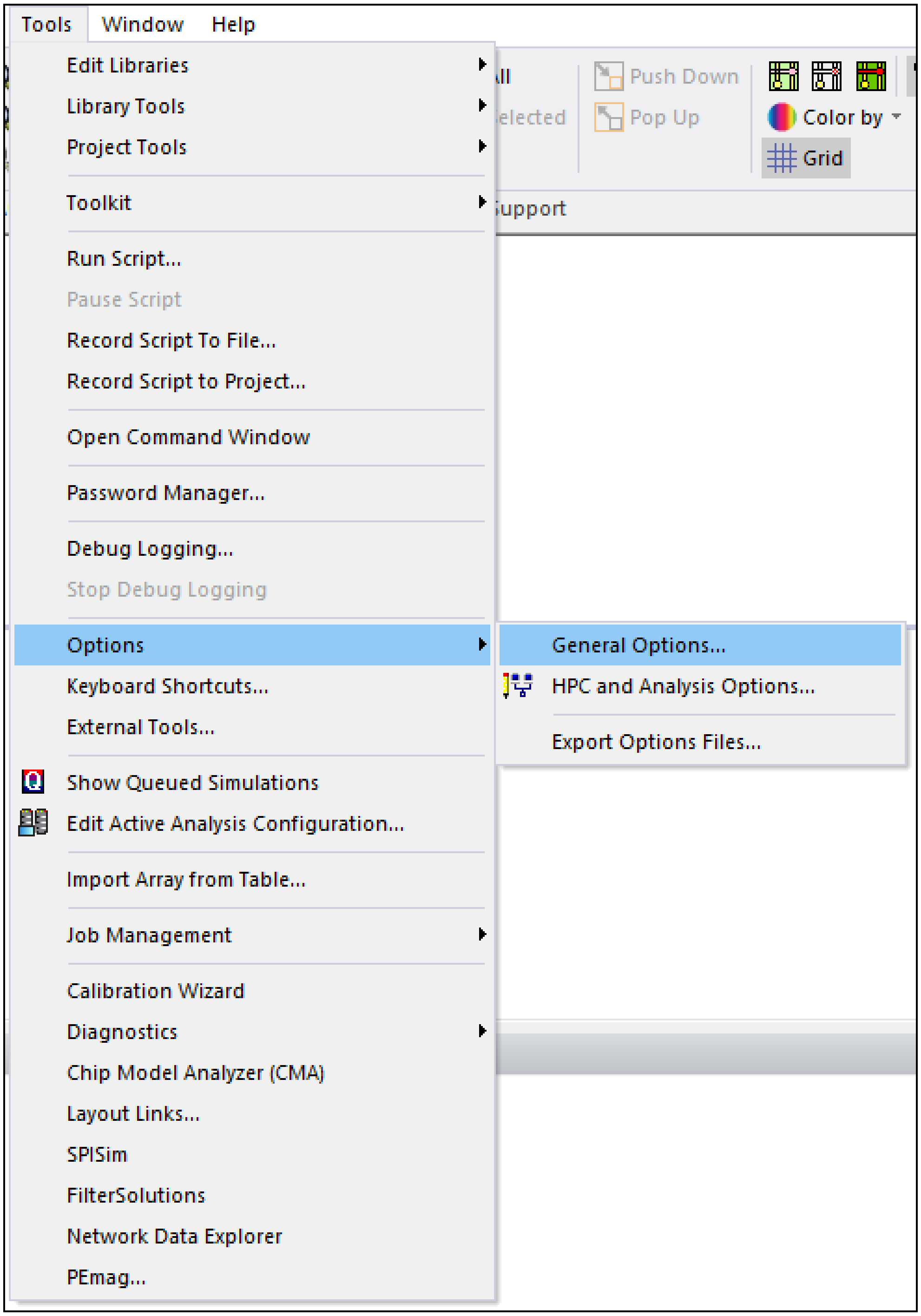


8. Click **OK** to close the **Options** window.

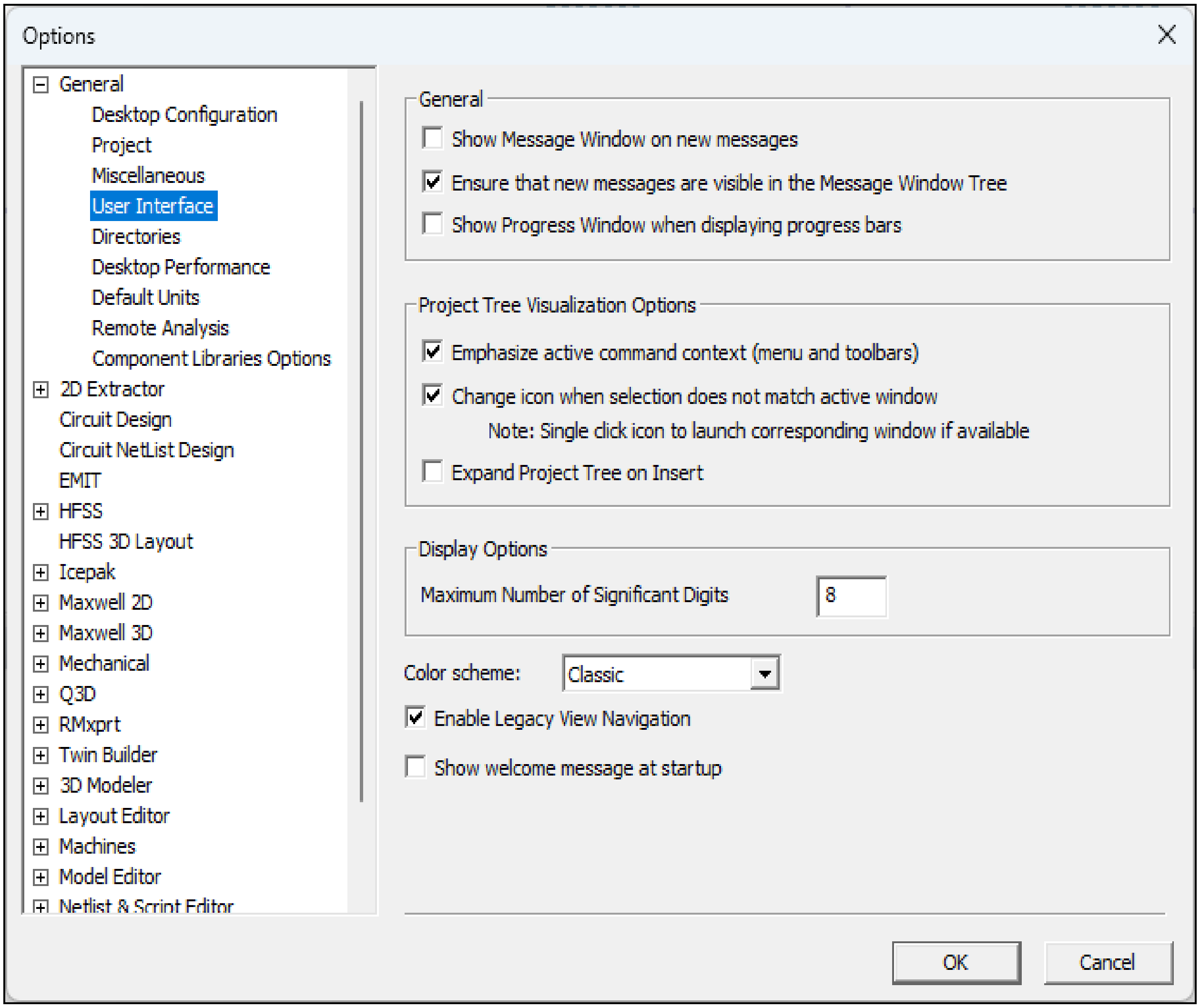
Enabling Legacy View Orientation

The instructions and examples in this guide use the legacy view orientation scheme, rather than the controls introduced in release 2024 R1. Complete these steps to enable the **Legacy View Orientation** and avoid any confusion.

1. Navigate to **Tools** > **Options** > **General Options** to open the **3D UI Options** window.



2. Expand **General** and select **User Interface**.
3. Check the **Enable Legacy View Orientation** box. When the user has completed the **Getting Started Guide**, they should return to **Options** window and uncheck the **Enable Legacy View Orientation** box.



4. Click **OK**.

Create the Model

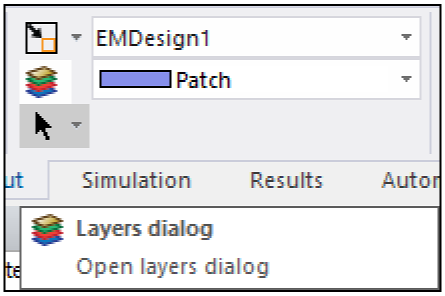
From here, continue to the following topics to define the stackup layers (i.e., topology) of the model, draw the geometry of the filter, and assign the excitation ports.

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

Insert Layers

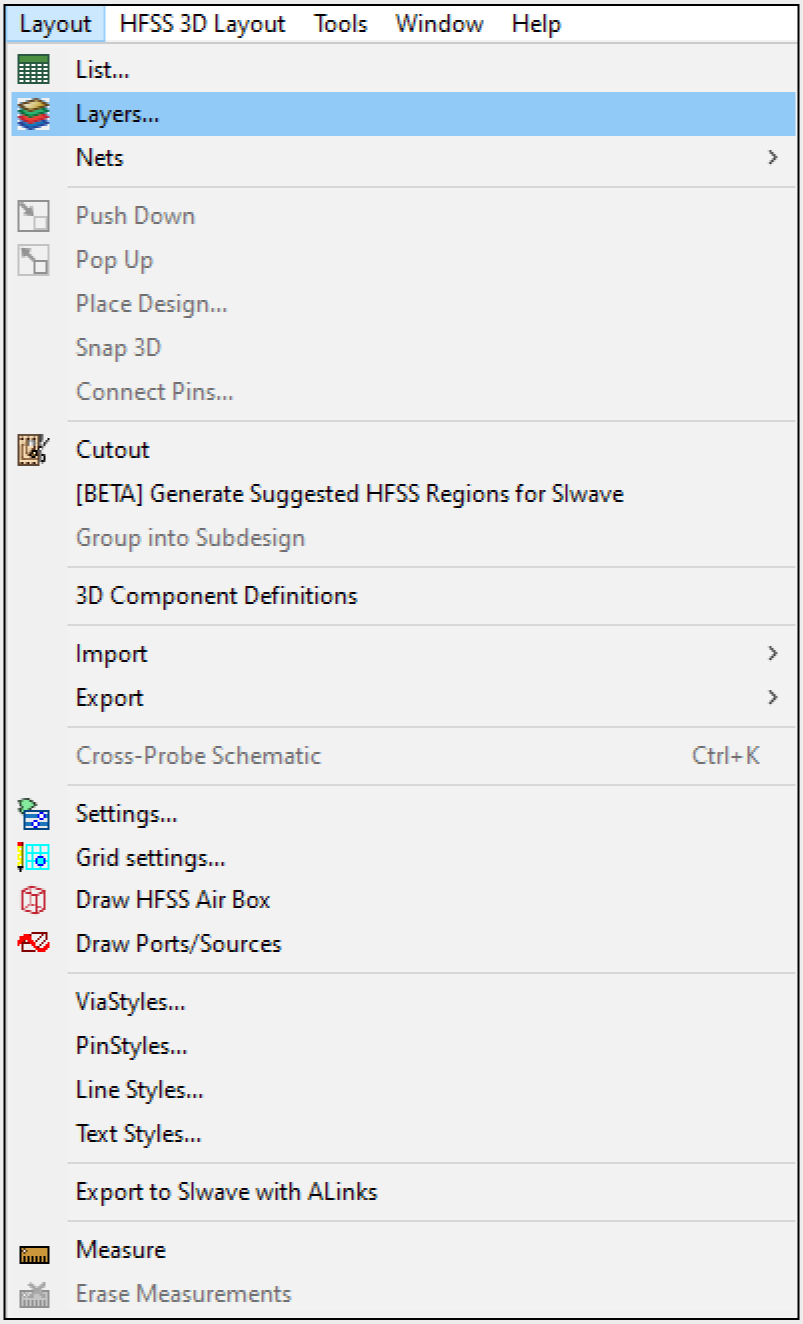
Complete these steps to insert layers in an HFSS 3D Layout design.

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

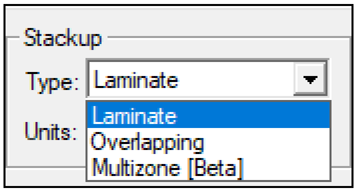


Note:

Alternatively, from **Layout**, select **Layers**.



2. If appropriate, select **Laminate** from the **Stackup** area > **Type** drop-down menu.

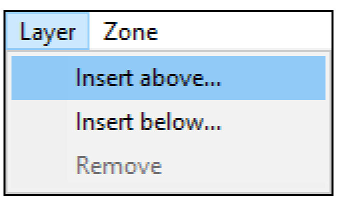


Add a Ground Layer to the Grid Control Table

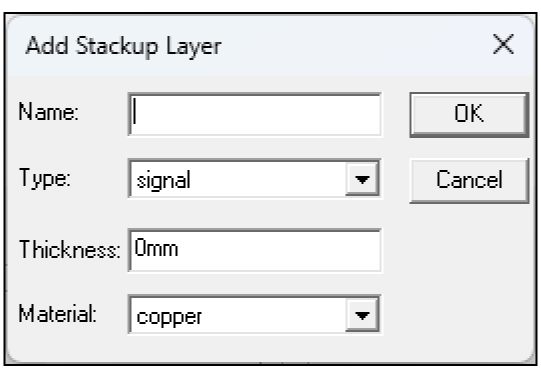
Note:

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 Grid Control Table. After selecting a layer from the table, select the chosen option depending on where you would like the new layer to appear in the table (i.e., above or below).

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



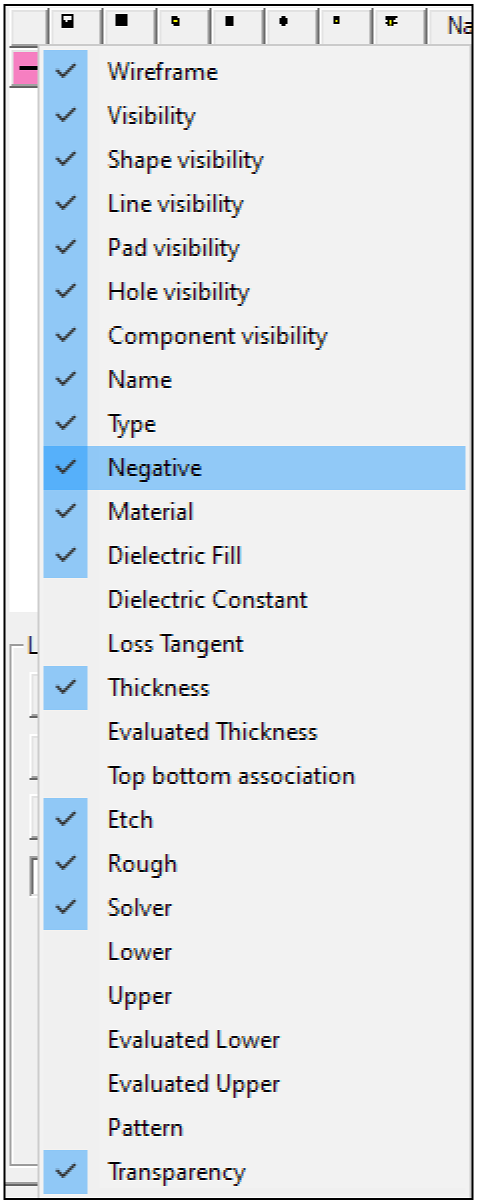
2. In the **Add Stackup Layer** window, do the following:



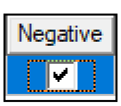
a. Enter **g1** in the **Name** field.
b. Select **signal** from the **Type** drop-down menu.
c. Click **OK** to close the **Add Stackup Layer** window add the new infinite ground layer to the Grid Control Table.

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

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



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



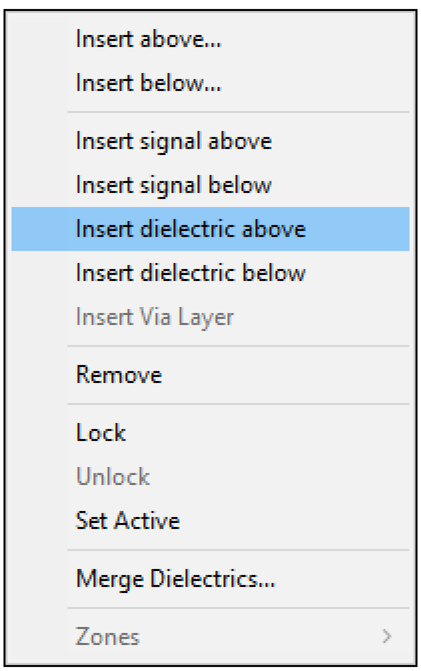
> **Note:**
>
> Checking the box in the **Negative** column identifies the layer as a ground plane layer. Any object drawn on a negative layer becomes a cutout in the ground layer (conductor removed). However, no objects can be drawn from the *g1* layer for this model.

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

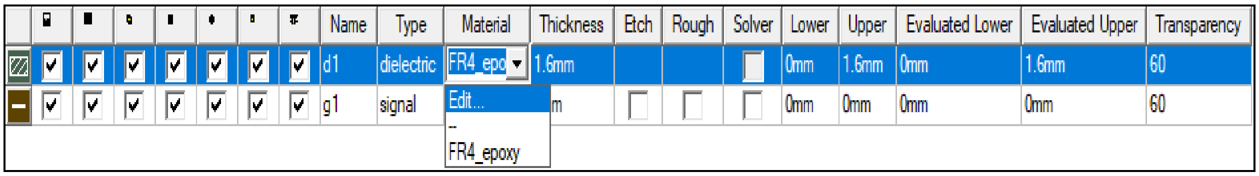
> **Note:**
>
> If the **Rough** box is checked, the surface roughness of the conductors is taken into account when approximating the impedance of the signal traces. However, surface roughness is ignored for ground layers. Removing the check from the box in the Rough column prevents a warning to that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

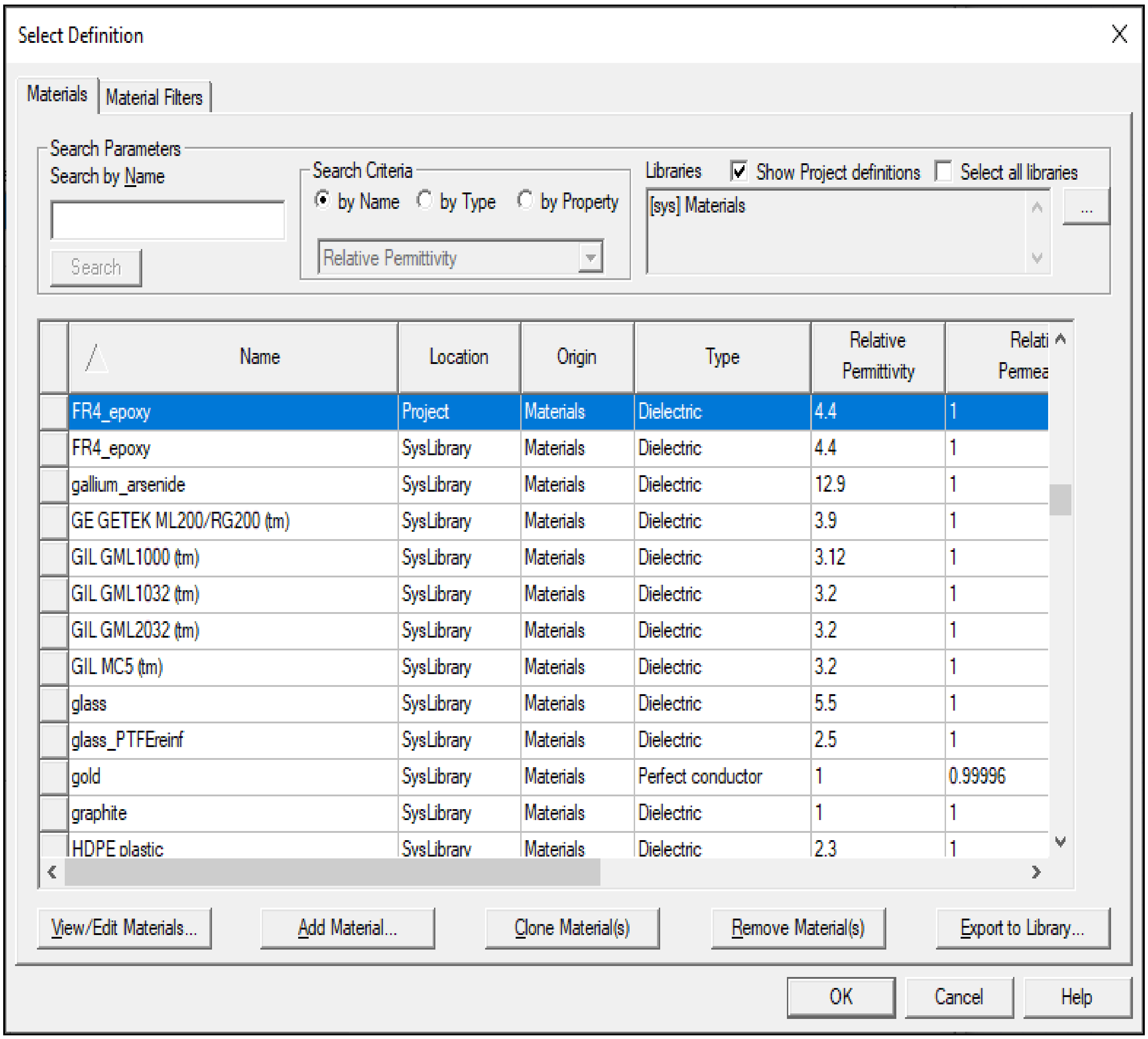
Add a Dielectric Layer to the Layer Table

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

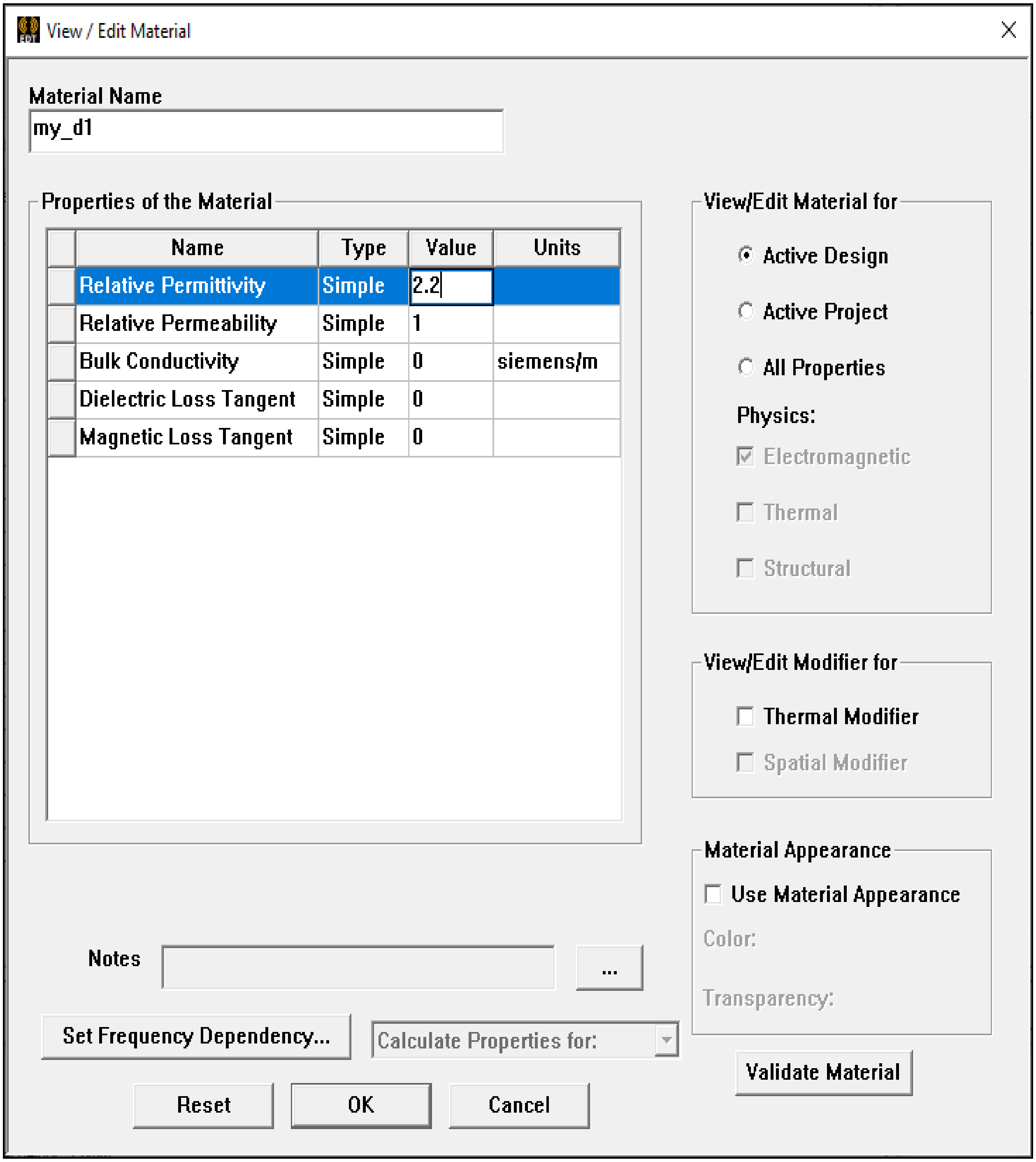


2. In the new **dielectric** row, do the following:
   a. In the **Name** field, replace **dielectric** with **d1**.
   b. Ensure **1.6mm** is entered in the **Thickness** field.
   c. Select **Edit** from the **Material** drop-down menu to open the **Select Definition** window.





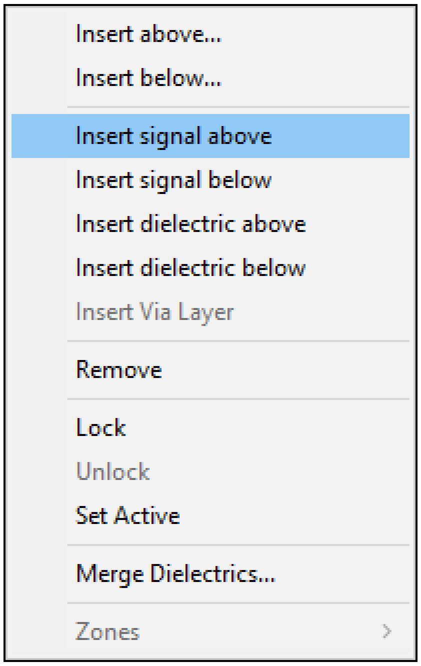
3. In the **Select Definition** window, do the following:
   a. Click **Add Material** to open the **View / Edit Material** window.
   b. In the **Material Name** field, replace **Material1** with **my_d1**.
   c. In the **Relative Permittivity Value** field, replace **1** with **2.2**.
   d. Click **OK**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



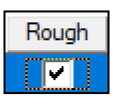
e. Click **OK** to close the **Select Definition** window.

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.

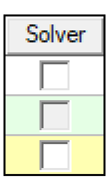


2. In the new **signal** row, do the following:
   a. In the **Name** field, replace **Signal** with **t1**.
   b. Check the box in the **Rough** column.

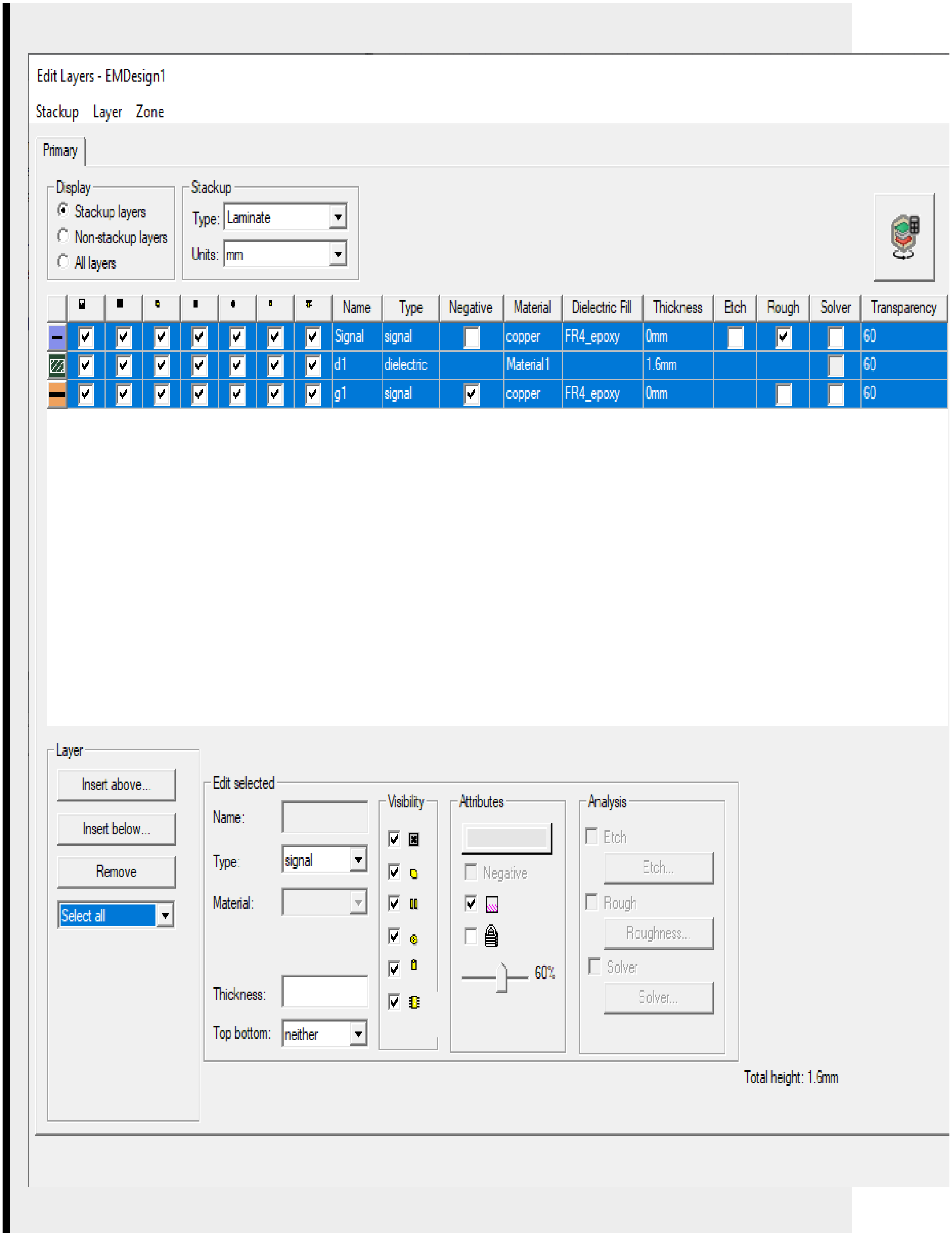


Make Changes to All Layers

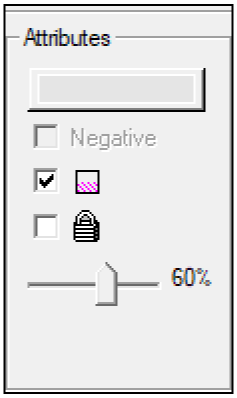
1. Ensure the **Solver** option is deselected for all layers. Default solver options are not overwritten.



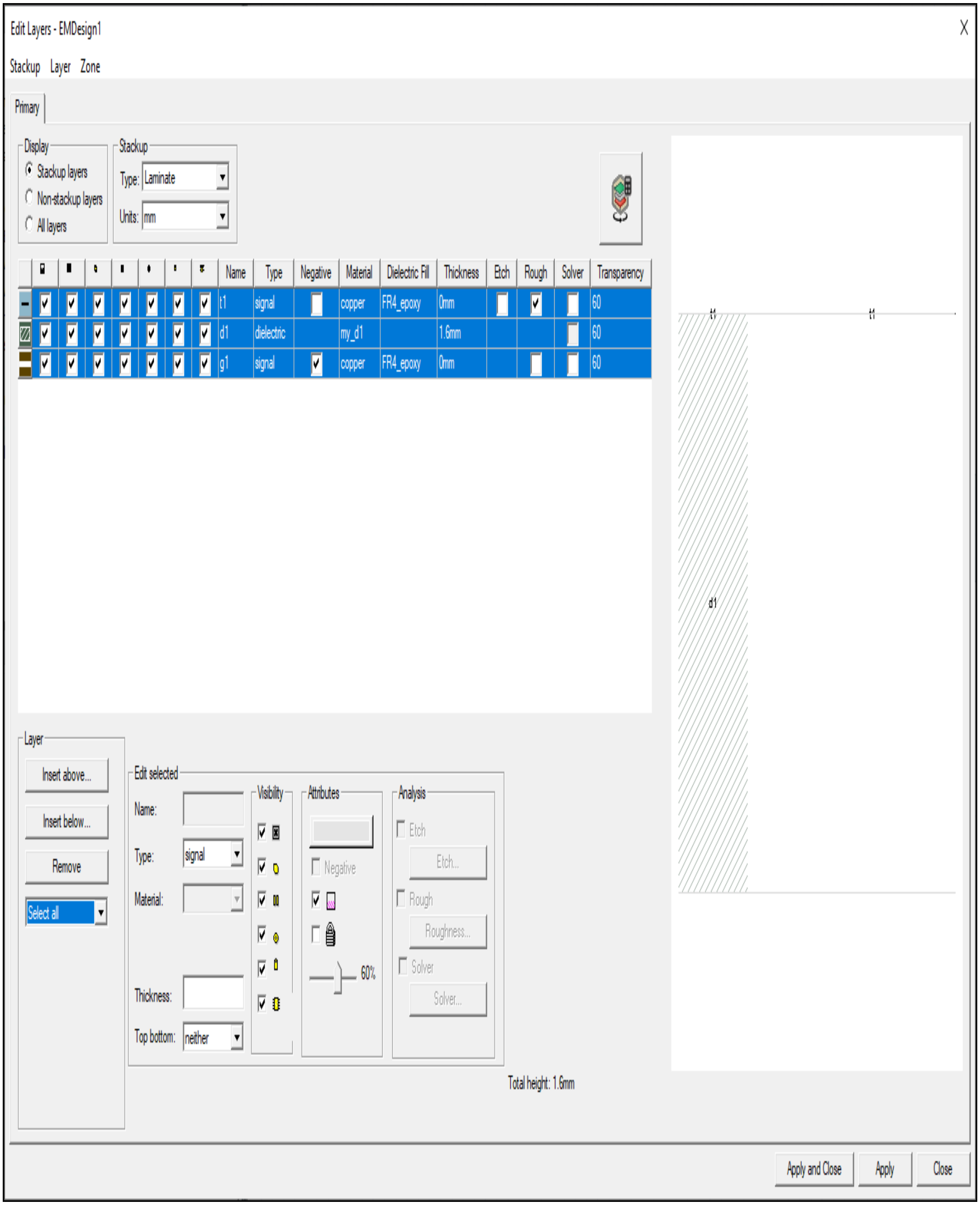
> **Note:**
>
> By default, **Select all** is chosen in the **Layer** area drop-down menu, but if the rows in the layers table are not highlighted, the layers are not actually selected. To complete step 4, click from the **Layer** area drop-down menu and choose **Select All** again. The rows will immediately be highlighted.


Edit Layers - EMDesign1
Stackup Layer Zone
Primary
Display
Stackup layers
Non-stackup layers
All layers
Stackup
Type: Laminate
Units: mm
Name Type Negative Material Dielectric Fill Thickness Etch Rough Solver Transparency
Signal signal copper FR4_epoxy 0mm 60
d1 dielectric Material1 1.6mm 60
g1 signal copper FR4_epoxy 0mm 60
Layer
Insert above...
Insert below...
Remove
Select all
Edit selected
Name:
Type: signal
Material:
Thickness:
Top bottom: neither
Visibility
Attributes
Negative
60%
Analysis
Etch
Etch...
Rough
Roughness...
Solver
Solver...
Total height: 1.6mm

2. From the **Layer** area, choose **Select all** from the drop-down menu.
3. 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 (wire frame).



4. 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 from the top of the list, followed by the **d1** layer in the middle, and the **g1** layer from the bottom.

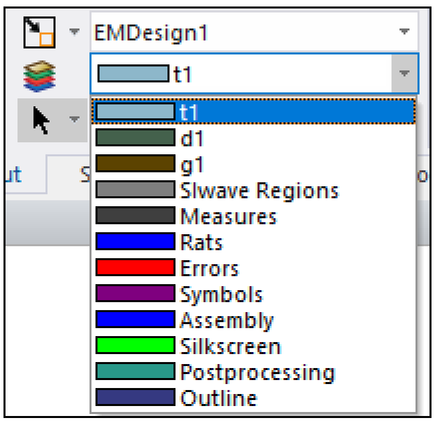
5. Click **Apply and Close** to apply the layer definitions and close the **Edit Layers** window.

Continue to Draw the Model.

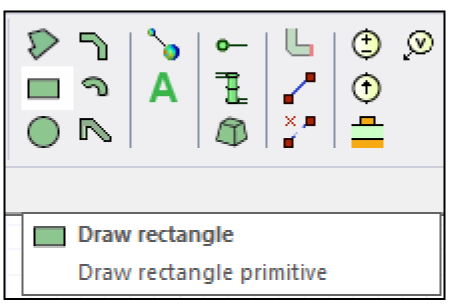
Draw the Model

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

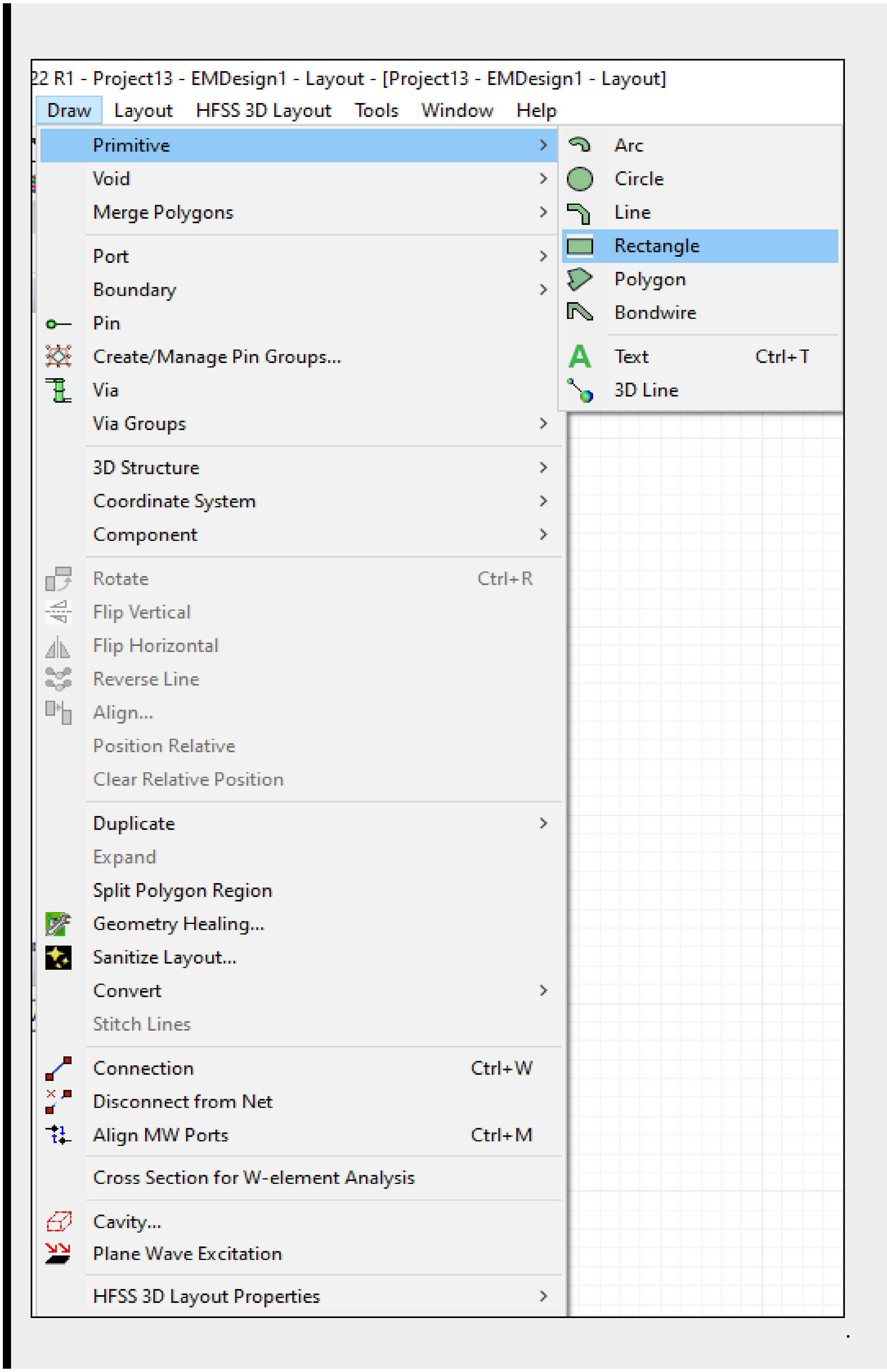
1. From the **Layout** tab, select **t1** from the **Active Layer** drop-down menu:



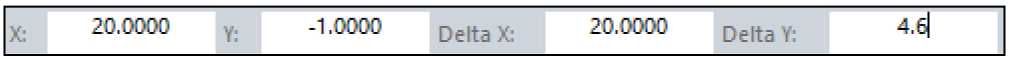
2. From the **Layout** tab, click **Draw rectangle**.



> **Note:**
>
> Alternatively, navigate to **Draw** > **Primitive** > **Rectangle**.


22 R1 - Project13 - EMDesign1 - Layout - [Project13 - EMDesign1 - Layout]
Draw
Layout
HFSS 3D Layout
Tools
Window
Help
Primitive
Void
Merge Polygons
Port
Boundary
Pin
Create/Manage Pin Groups...
Via
Via Groups
3D Structure
Coordinate System
Component
Rotate Ctrl+R
Flip Vertical
Flip Horizontal
Reverse Line
Align...
Position Relative
Clear Relative Position
Duplicate
Expand
Split Polygon Region
Geometry Healing...
Sanitize Layout...
Convert
Stitch Lines
Connection Ctrl+W
Disconnect from Net
Align MW Ports Ctrl+M
Cross Section for W-element Analysis
Cavity...
Plane Wave Excitation
HFSS 3D Layout Properties
Arc
Circle
Line
Rectangle
Polygon
Bondwire
Text Ctrl+T
3D Line

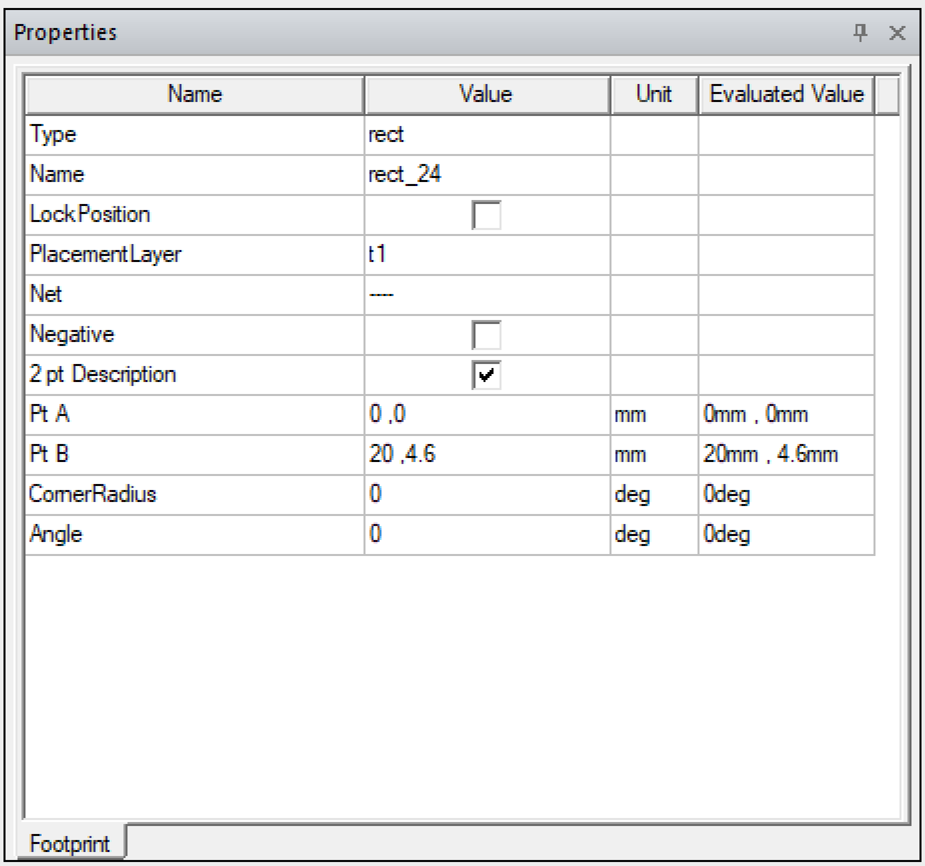
3. Do **not click**+**drag** in the **Layout Editor**. Instead, move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Click inside the field, delete the coordinates already present, and enter **0**.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.
5. Either press **Tab** until the cursor moves to the **Delta X** coordinate field or move the cursor to the field, click inside it, and enter **20**.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field, Then type **4.6** in the field and press **Enter** to complete the shape.



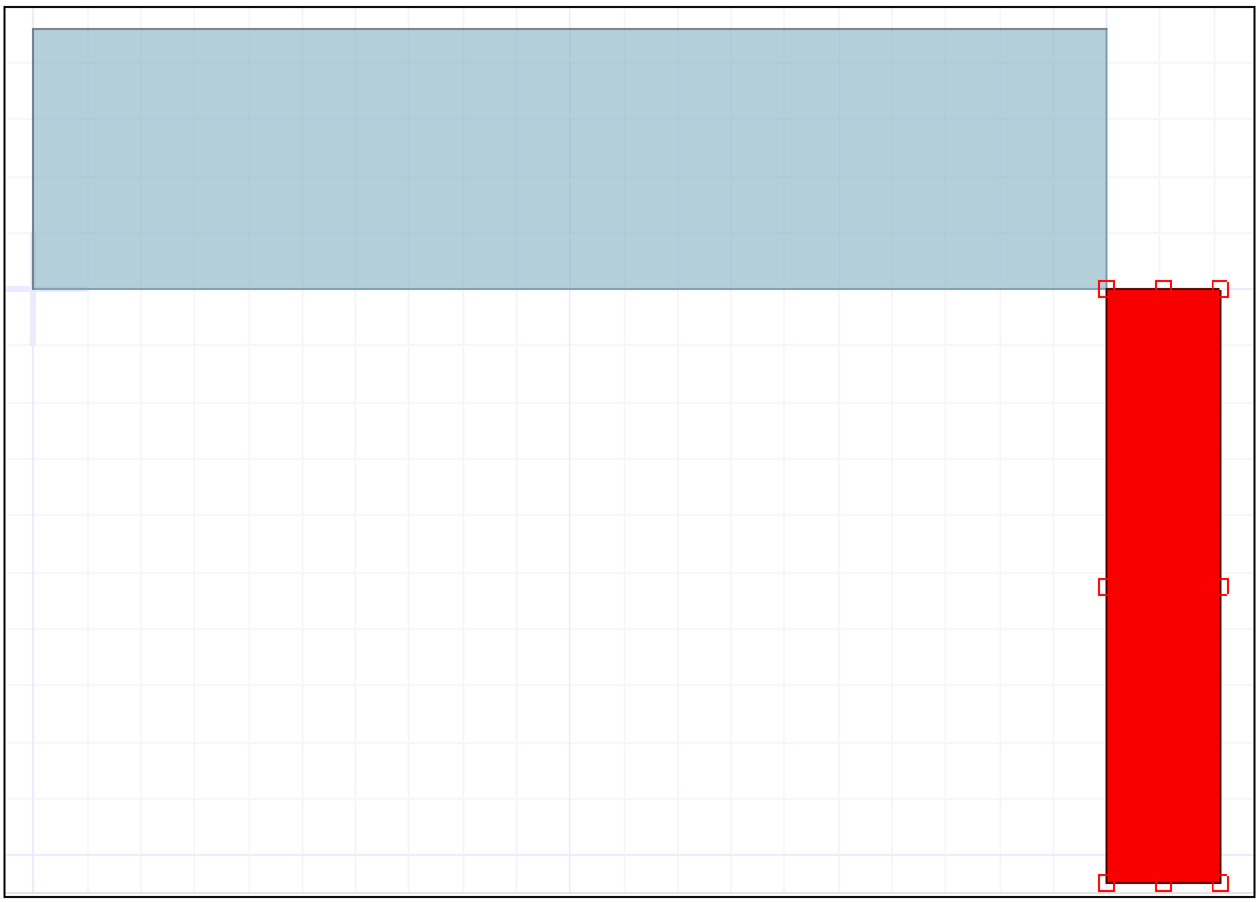
Note:

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. 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. In 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.



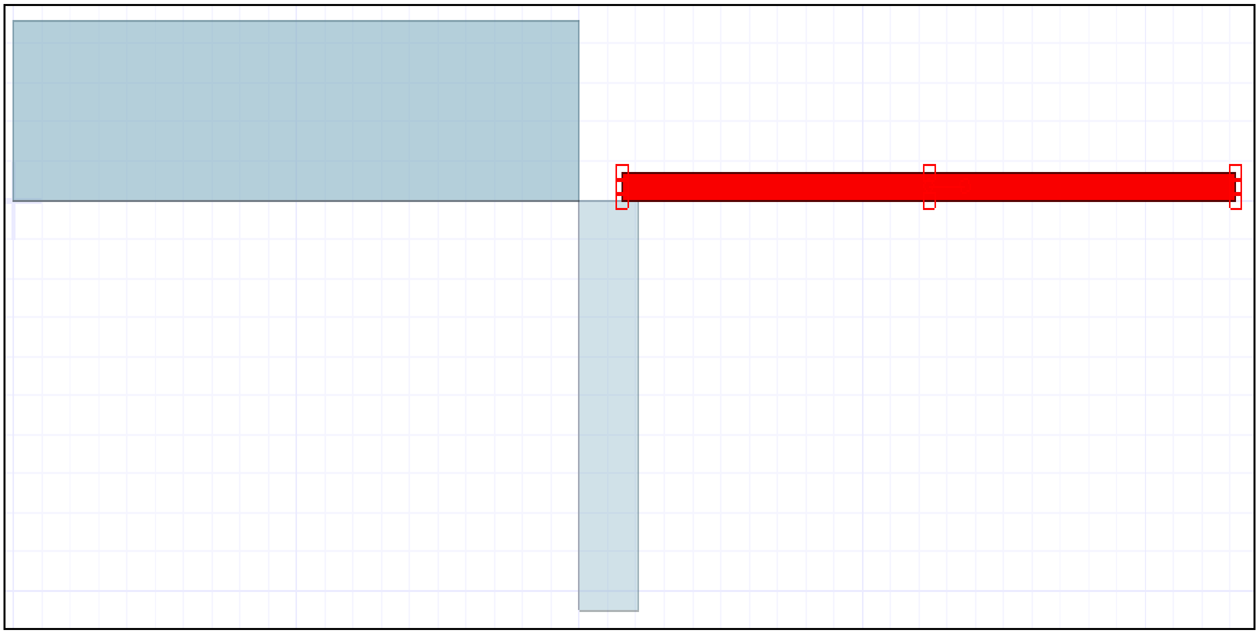
7. Create a second rectangle by doing the following:
   a. From the **Layout** tab, click **Draw rectangle**.
   b. Hover over the lower-right corner of the first rectangle until the cursor turns into a square when the snap point is found. Then click to snap the first corner of the second rectangle to that point. This effectively designates the **X**, **Y** coordinates for the second rectangle, as if this position were **20**, **0**.
   c. Click inside the **Delta X** coordinate field, delete the coordinates already present, and enter **2.1**. Then press **Tab**.
   d. Type **-10.5** in the **Delta Y** coordinate field and press **Enter**.



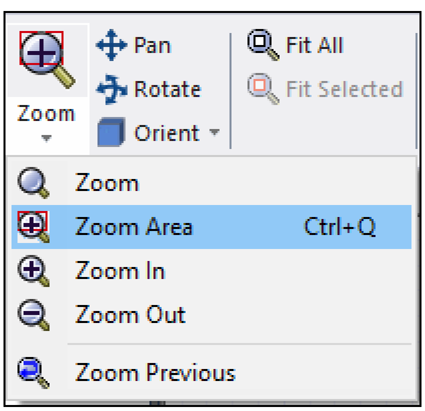
8. Create a third rectangle by doing the following:
   a. From the **Layout** tab, click **Draw rectangle**.
   b. Either hover in the **Layout Editor** and press **Tab** to move the cursor to the **X** coordinate field, or move the cursor to the field and click inside it. Enter **21.05** in the field, then press **Tab**.
   c. Type **0** in the **Y** coordinate field, then press **Enter**.

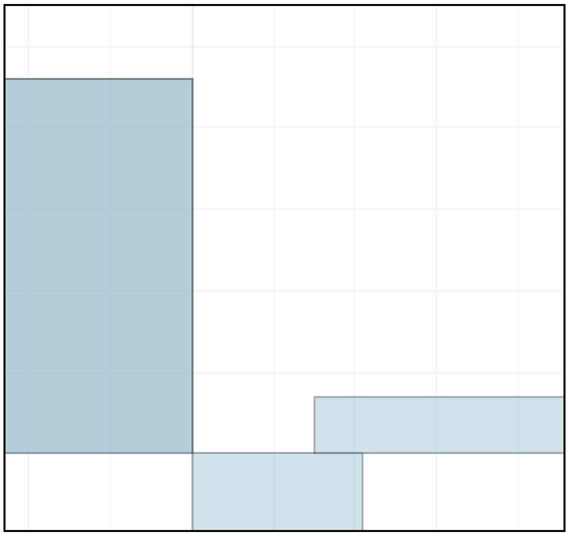
d. Either press **Tab** until the cursor moves to the **Delta X** coordinate field, or move the cursor to the field and click inside it. Enter **21.7** in the field, then press **Tab**.

e. Type **0.7** in the **Delta Y** coordinate field, then press **Enter**.



9. Click anywhere else in the **Layout Editor** to clear the current selection.
10. **Zoom In** from 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, by doing one of the following:
    - Spin the mouse wheel to **Zoom In**/**Out**.
    - Press **Ctrl**+**D**.
    - From the **Layout** tab, click **Fit All**.
    - From **View**, select **Fit All**.
    - From the Layout ribbon, select **Zoom** > **Zoom Area**. Then **click+drag** the mouse to define an area.



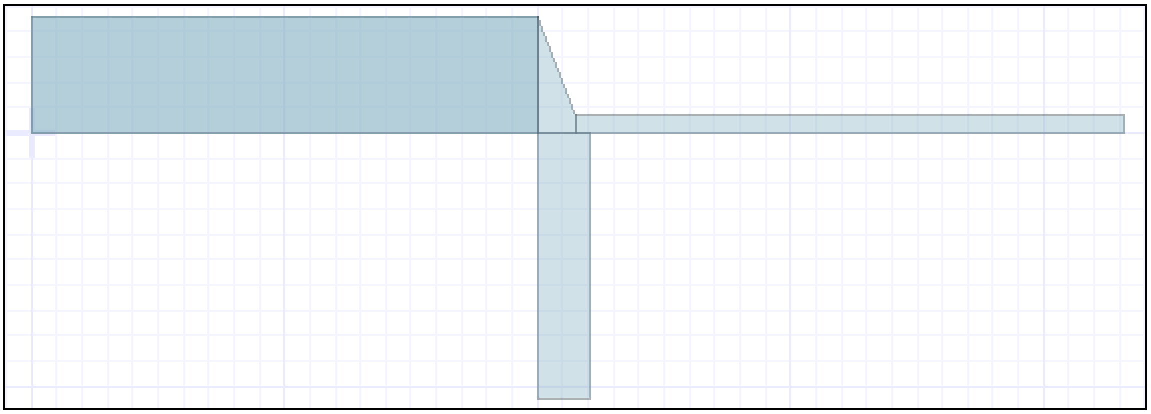


11. Create a polygon by doing the following:

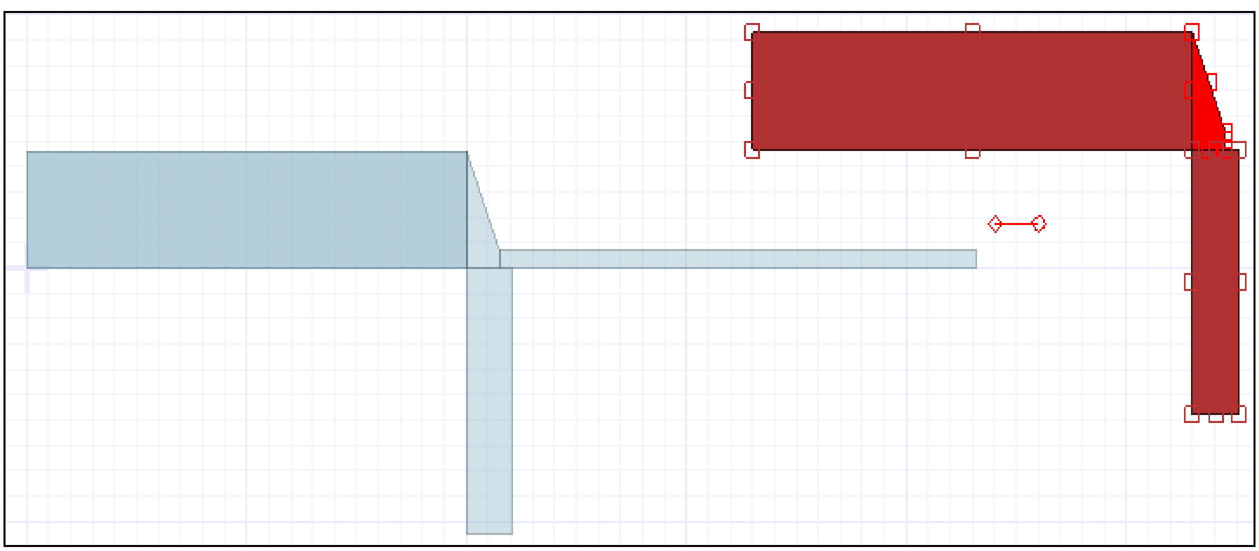
> **Note:**
>
> During step c, the cursor may become a triangle, since the snapping point corresponds to the midpoint between the second rectangle's top edge and the end point of two of the third rectangle's edges.

   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**)

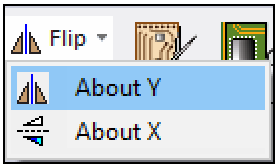
12. Press **Ctrl+D** to fit the drawing in the **Layout Editor** and clear the current selection.



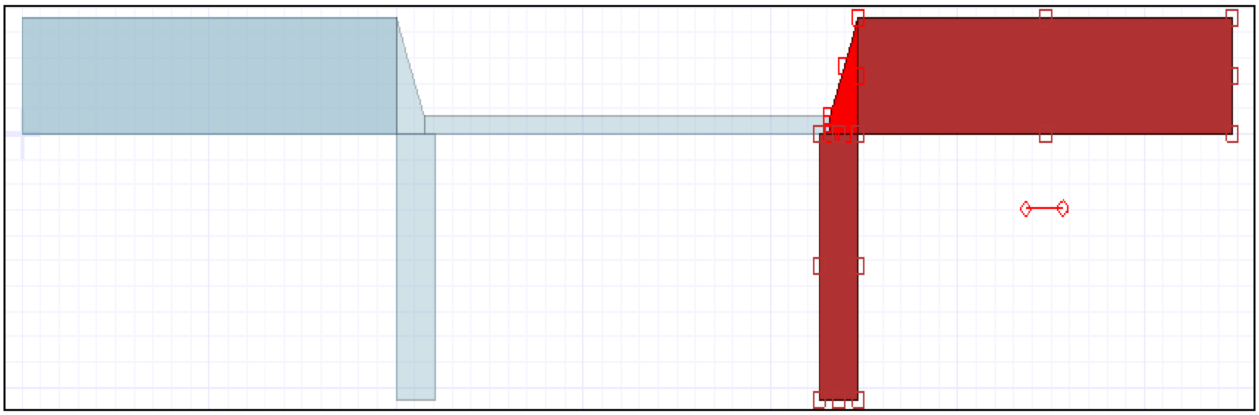
13. Create duplicate copies of the first two rectangles and the new polygon by doing the following:
    a. Press **Ctrl+A** to select all the objects.
    b. While holding down **Ctrl**, click the third rectangle to deselect it. The first two rectangles and the polygon should still be selected.
    c. 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.



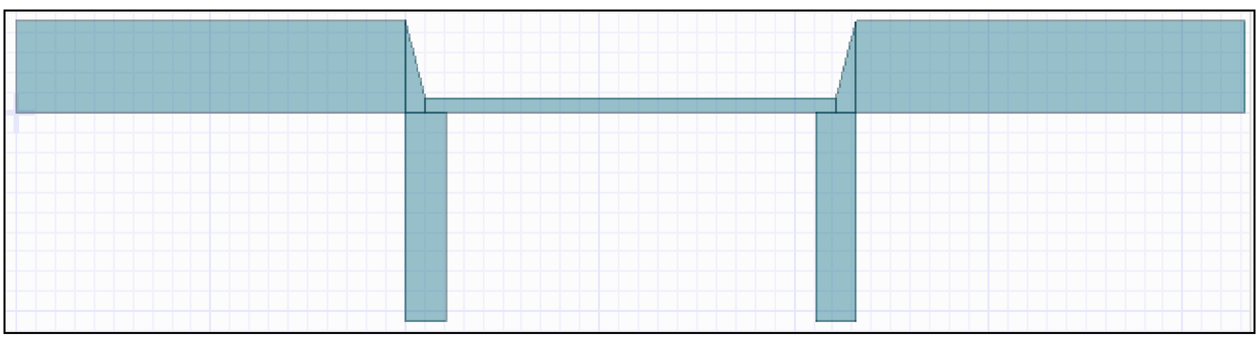
14. From the **Layout** tab, select **Flip** > **About Y** (or, from **Draw**, select **Flip Horizontal**).



15. **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.



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



17. Create a fourth and final rectangle by doing the following:
    a. From the **Layout** tab, click **Draw rectangle**.
    b. Either hover in the **Layout Editor** and press **Tab** to move the cursor to the **X** coordinate field, or move the cursor to the field and click inside it. Enter **29.3** in the **X** coordinate field, then press **Tab**.
    c. Type **0** in the **Y** coordinate field, then press **Enter**.

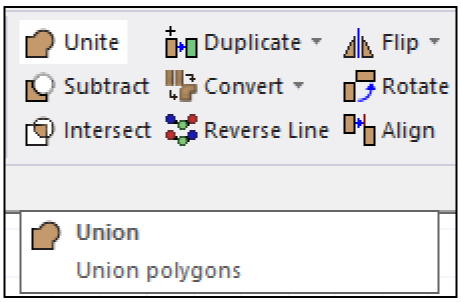
d. Either press **Tab** until the cursor moves to the **Delta X** coordinate field or move the cursor to the field and click inside it. Enter **5.3** in the field, then press **Tab**.

e. Type **-10.5** in the **Delta Y** coordinate field, press **Enter**.

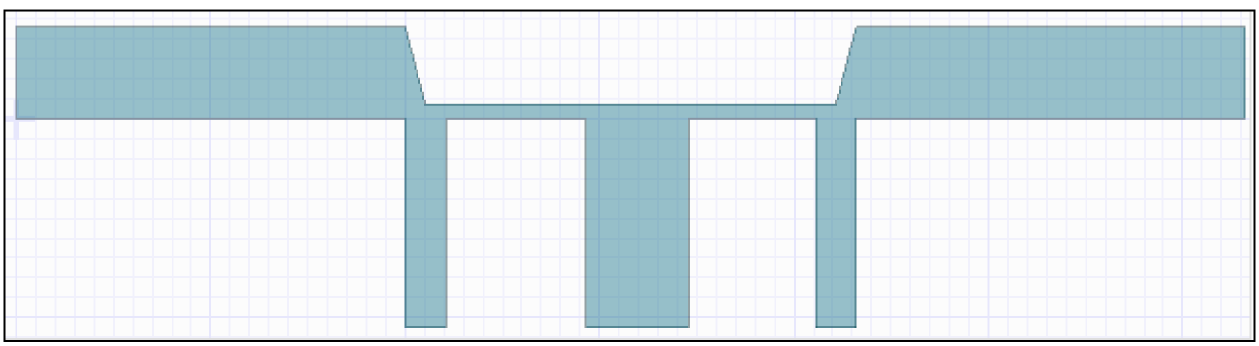
18. Press **Ctrl+A** to select all objects.

> **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.

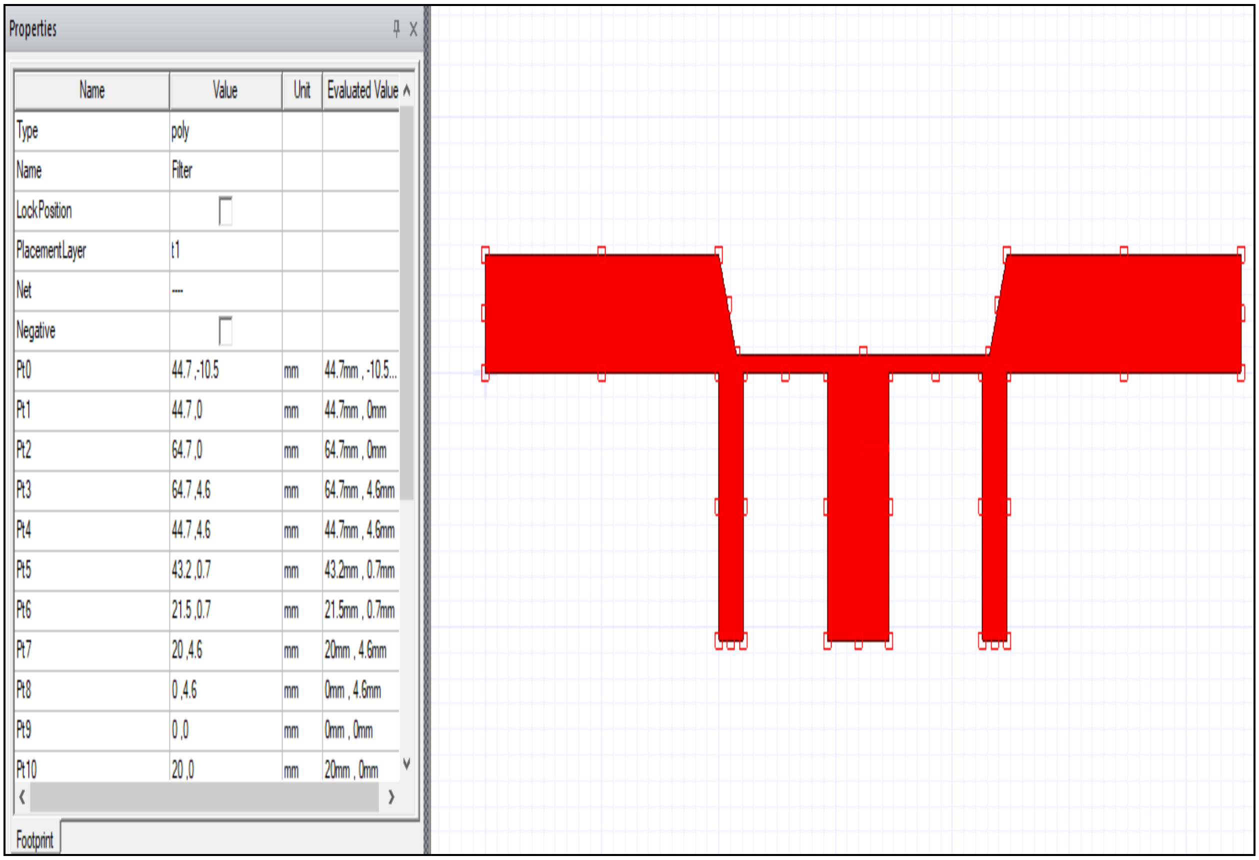
19. From the **Layout** tab, select **Unite** to form all the rectangles and polygons into a single object.



20. The design should match the following figure.



21. 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.



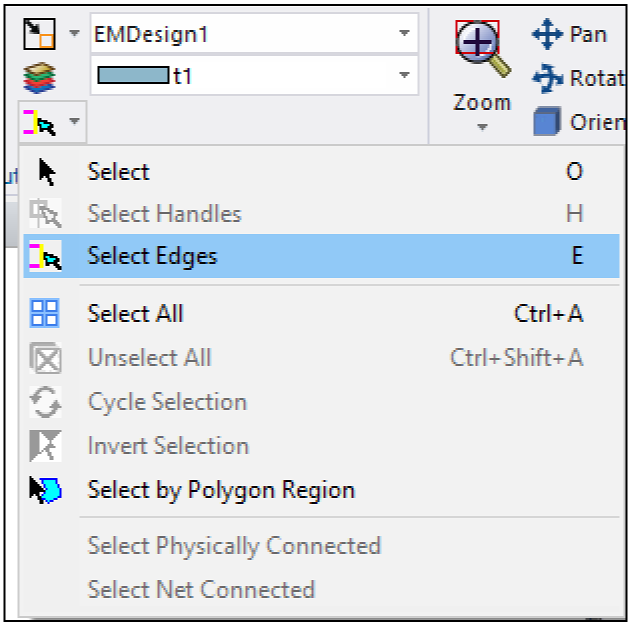
22. **Save** the design, either by navigating to **File** > **Save** or clicking the **Save** button on any of the ribbons.

Continue to Assign the Ports.

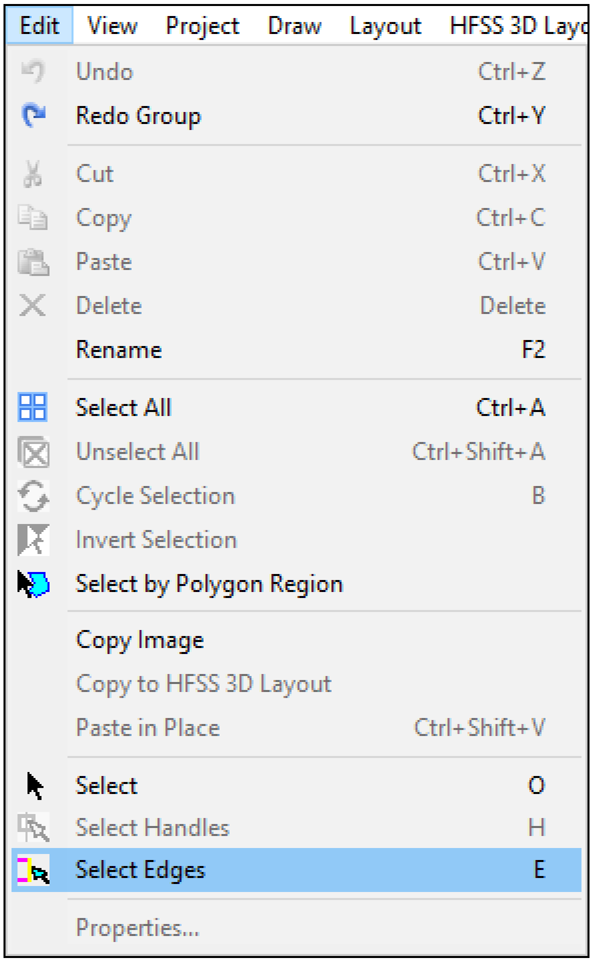
Create Edge Ports

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

1. To create the first port (i.e., ***Port1***), do any of the following:
   - Press **E** to enter **Select Edges** mode.
   - From the **Layout** tab, click **Select edges** > **Select Edges**.

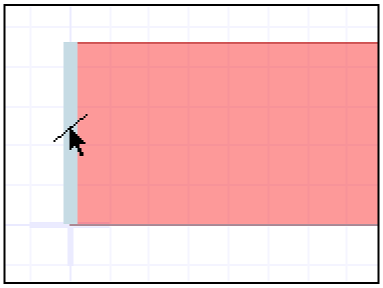


- From **Edit**, click **Select Edges**.

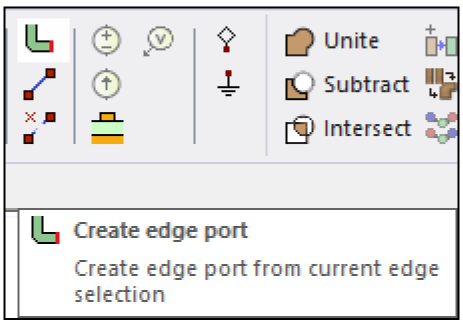


> **Note:**
>
> Once **Select Edges** is chosen, the cursor changes: a diagonal line crosses the tip of the arrow. Refer to the following step.

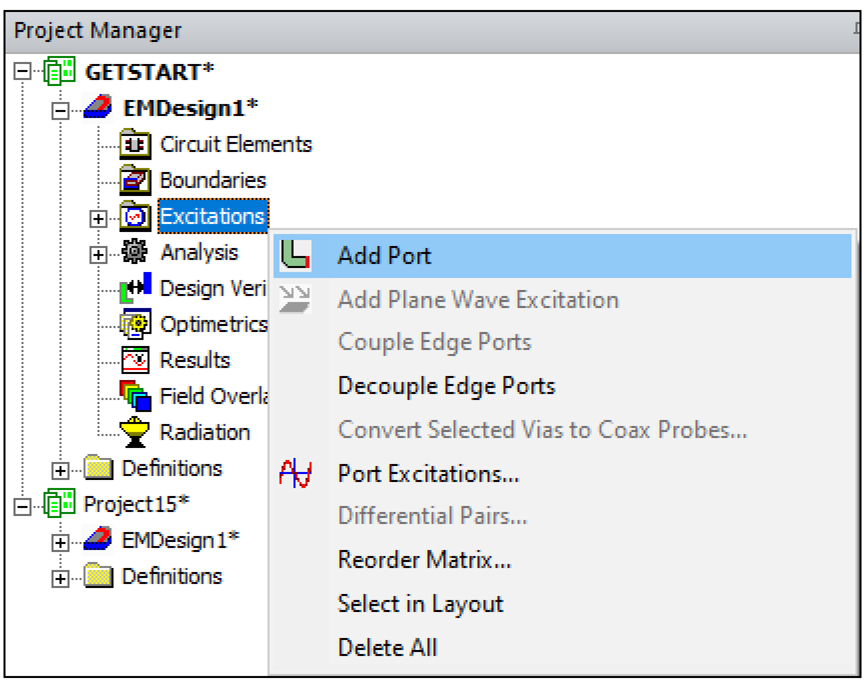
2. Click the left side of the leftmost rectangle to select it.



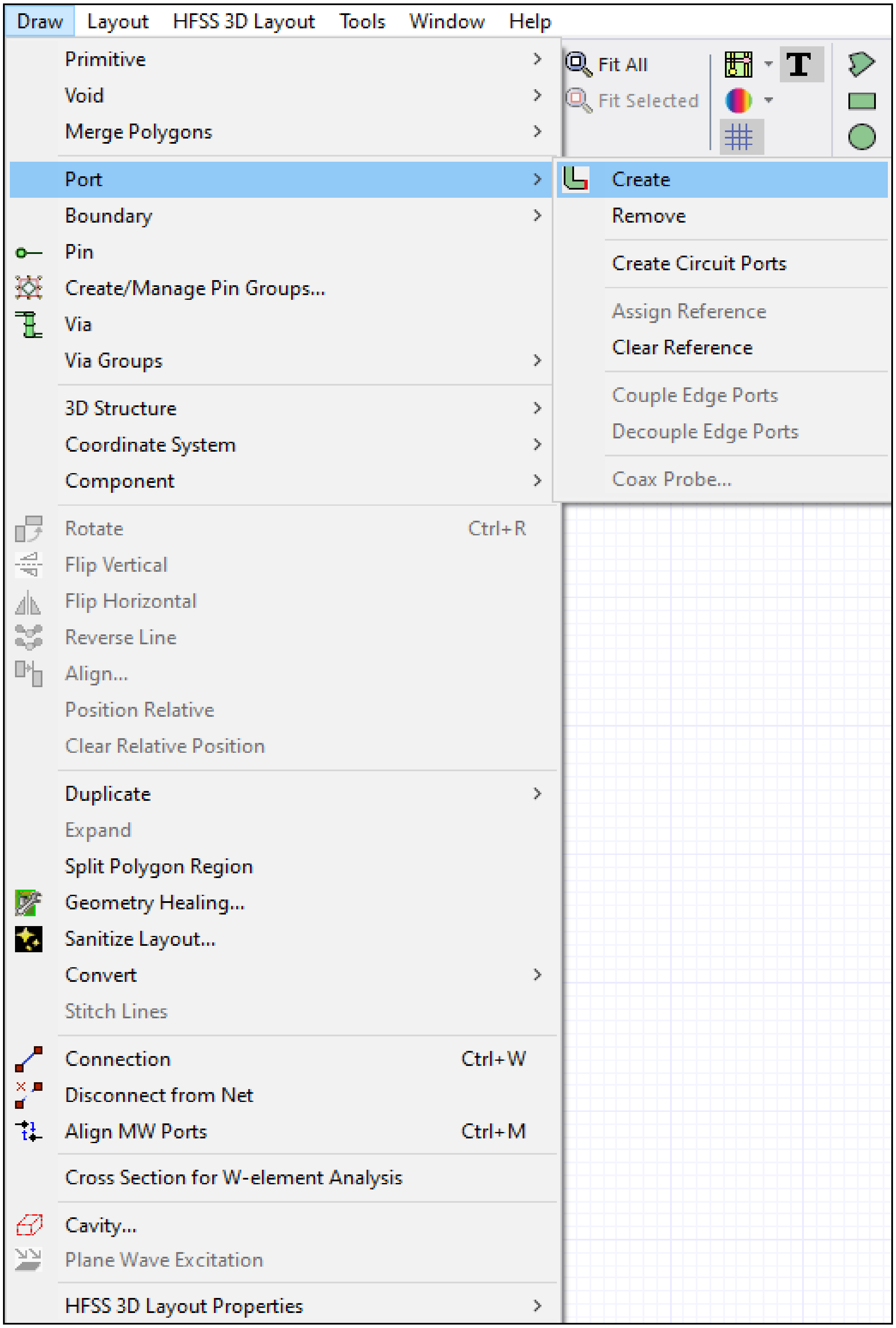
3. To add the first port to the leftmost rectangle, do one of the following:
   - From the **Layout** tab, select **Create edge port**.



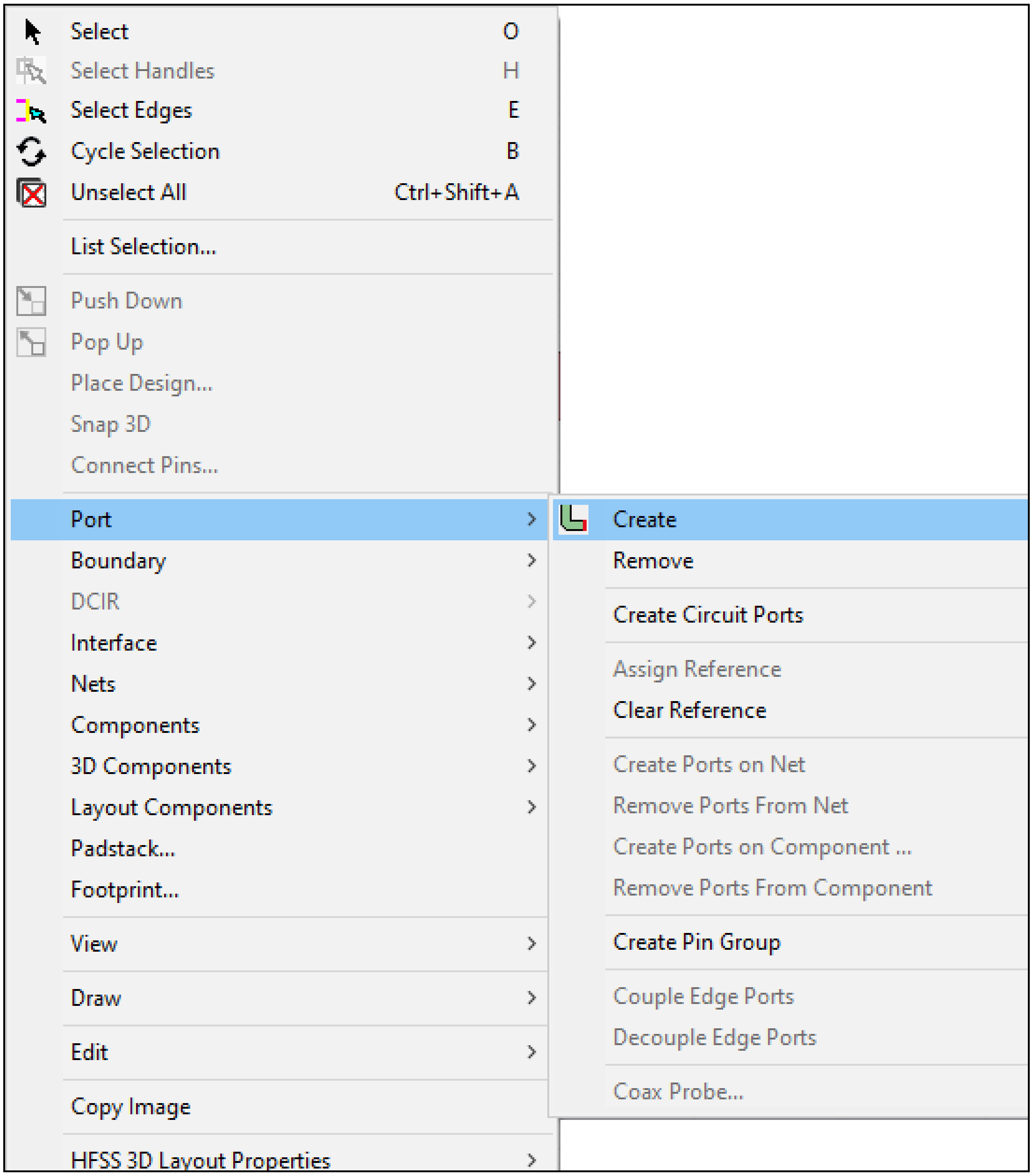
   - From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Excitations** and select **Add Port**.



- From **Draw**, select **Port** > **Create**.

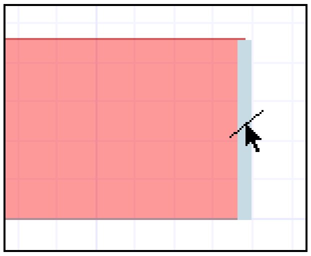


- Right-click in the **Layout Editor** and select **Port** > **Create**.

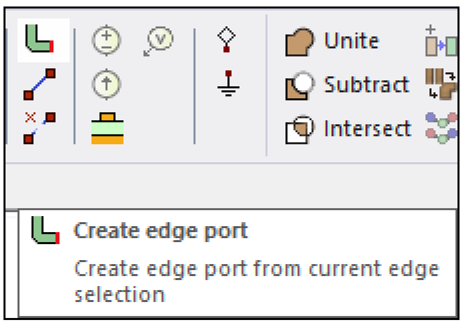


> **Note:**
>
> Once a port is created, it appears in the **Project Manager** > **Project Tree** > [***active design folder***] > **Excitations** folder.

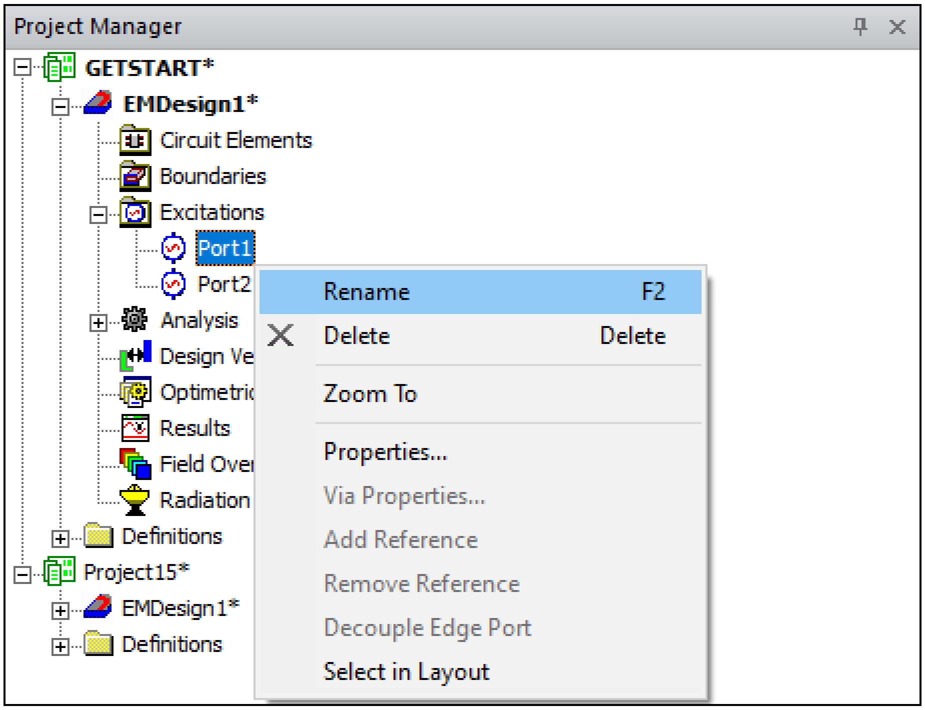
4. Create the second port (i.e., ***Port2***) by doing the following:
   a. If appropriate, press **E** to re-enter **Select Edges** mode.
   b. Click the right edge of the rightmost rectangle to select it.



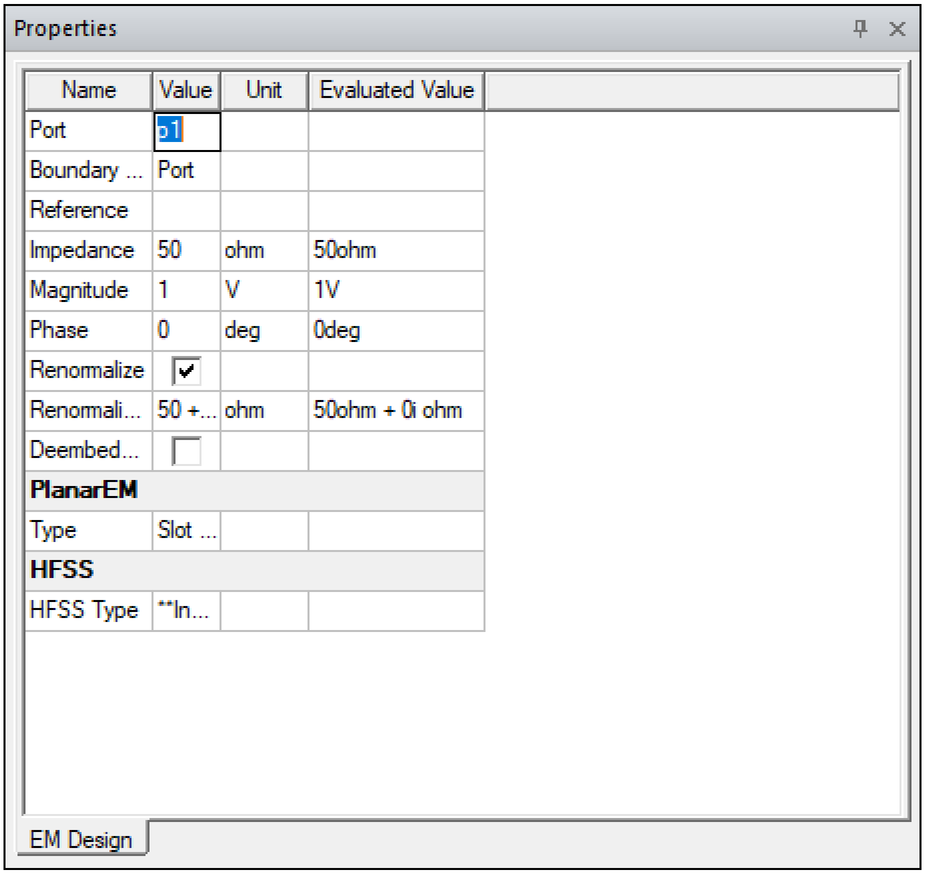
   c. From the **Layout** tab, select **Create edge port**.



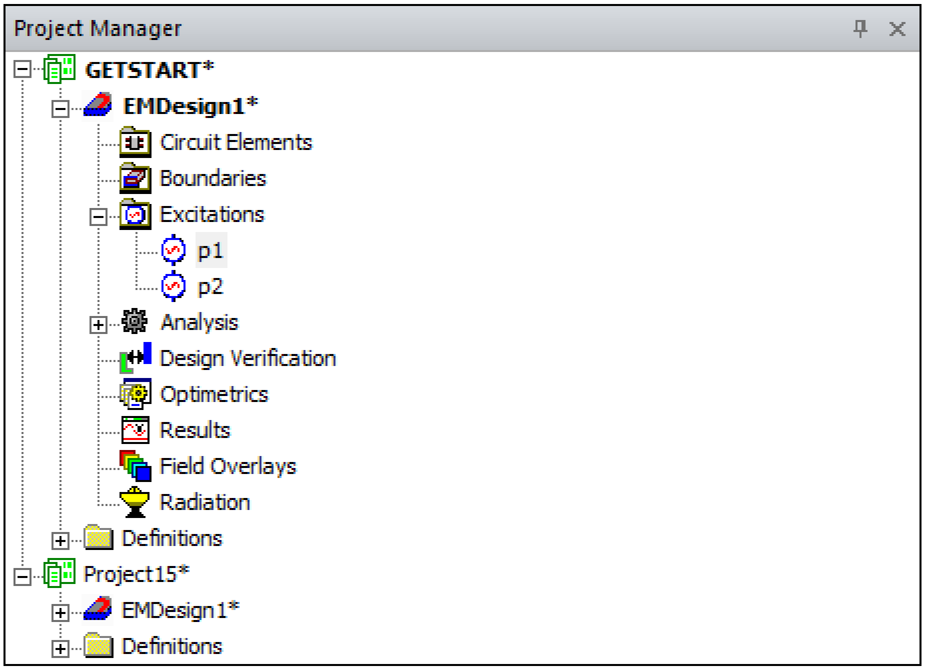
5. Rename *Port 1* by doing either of the following:
   - From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Excitations** folder. Then right-click **Port1**, select **Rename**, and replace **Port1** with **p1**.



- From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Excitations** folder. Then click **Port1**. From the **Properties** window > **Port Value** field, replace **Port1** with **p1**, then press **Enter**.



6. Repeat step 5 to rename **Port2** to **p2**.



Continue to Set Up a Planar EM Analysis.

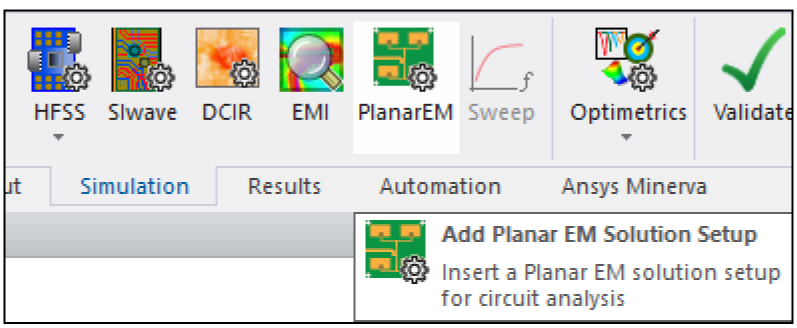
2 - Set Up Solution and Analyze

This chapter contains the following topics:

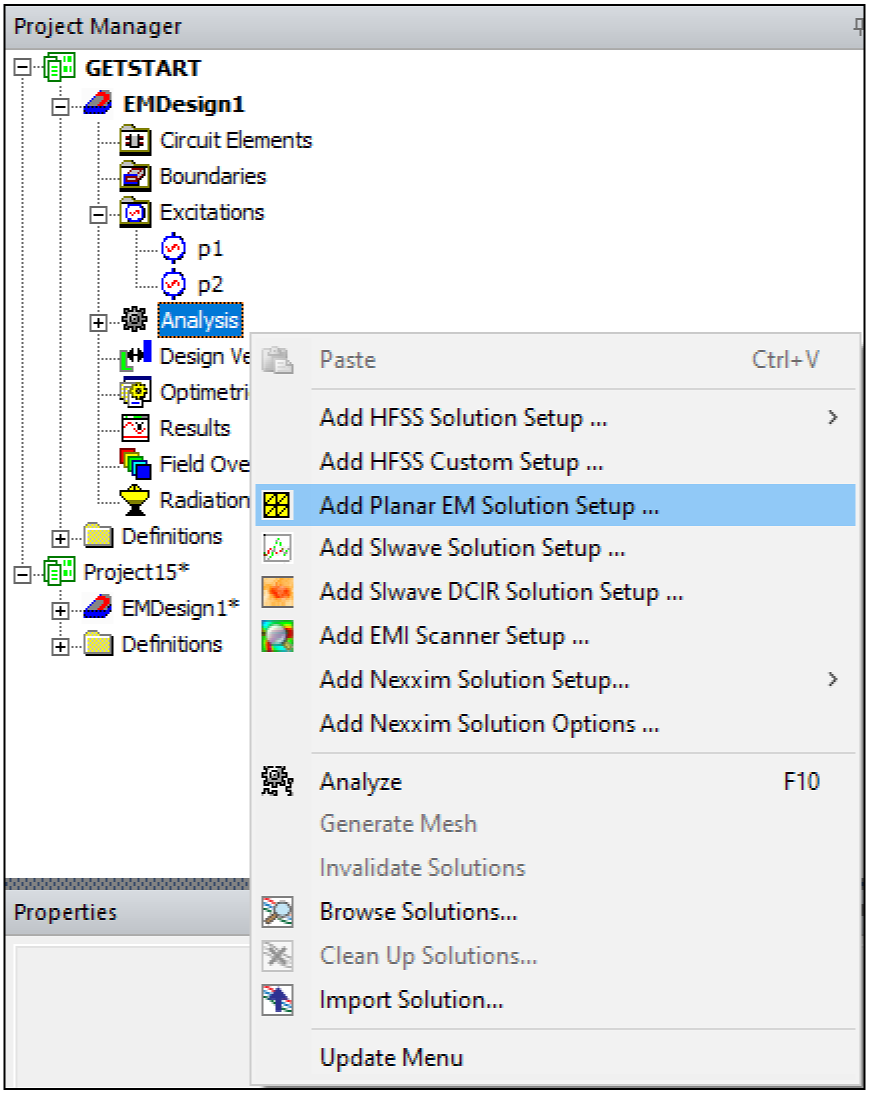
Set Up a Planar EM Analysis

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

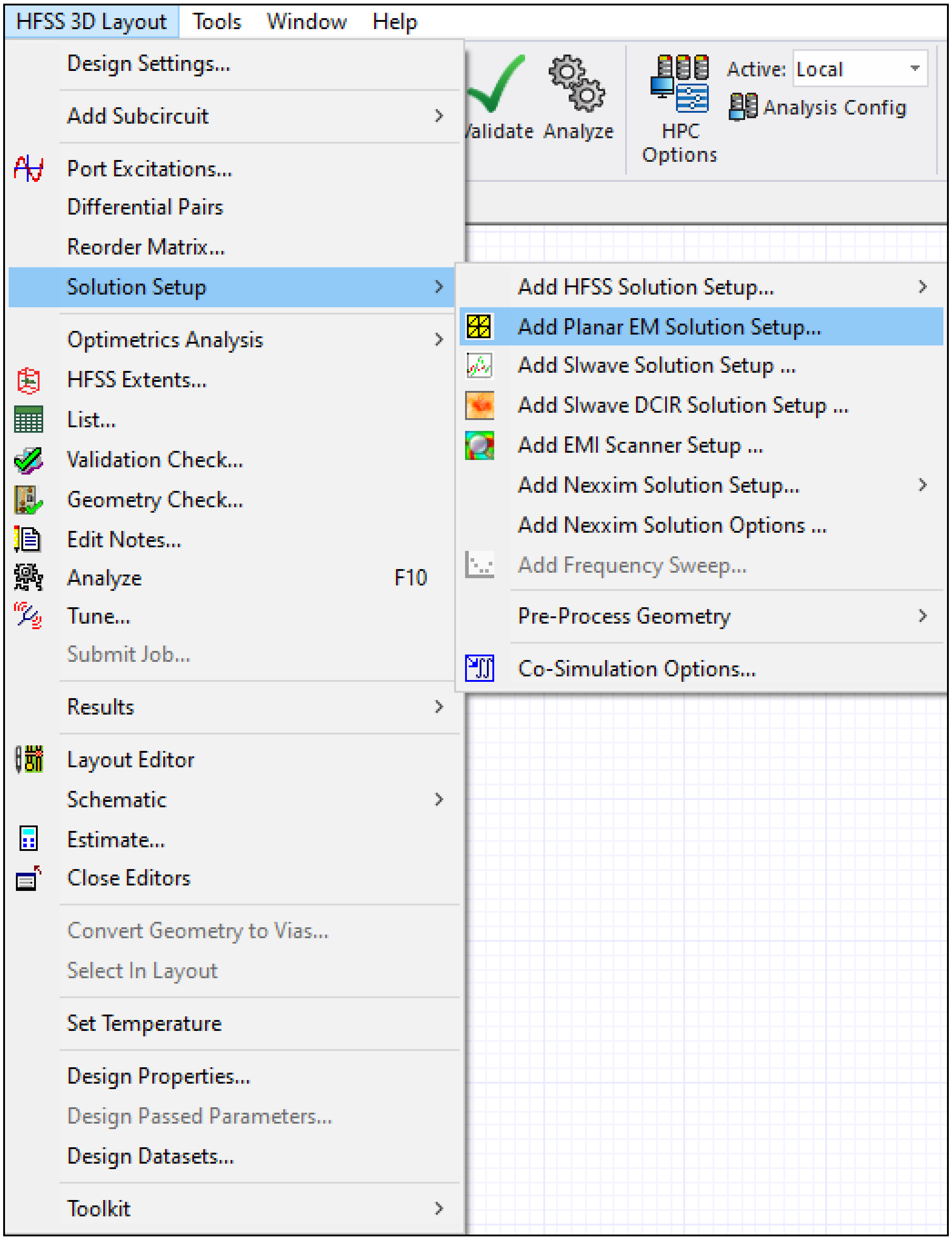
1. Open the **PlanarEMSetup** window by doing one of the following:
   - From the **Simulation** ribbon tab, click **PlanarEM** (**Add Planar EM Solution Setup**).



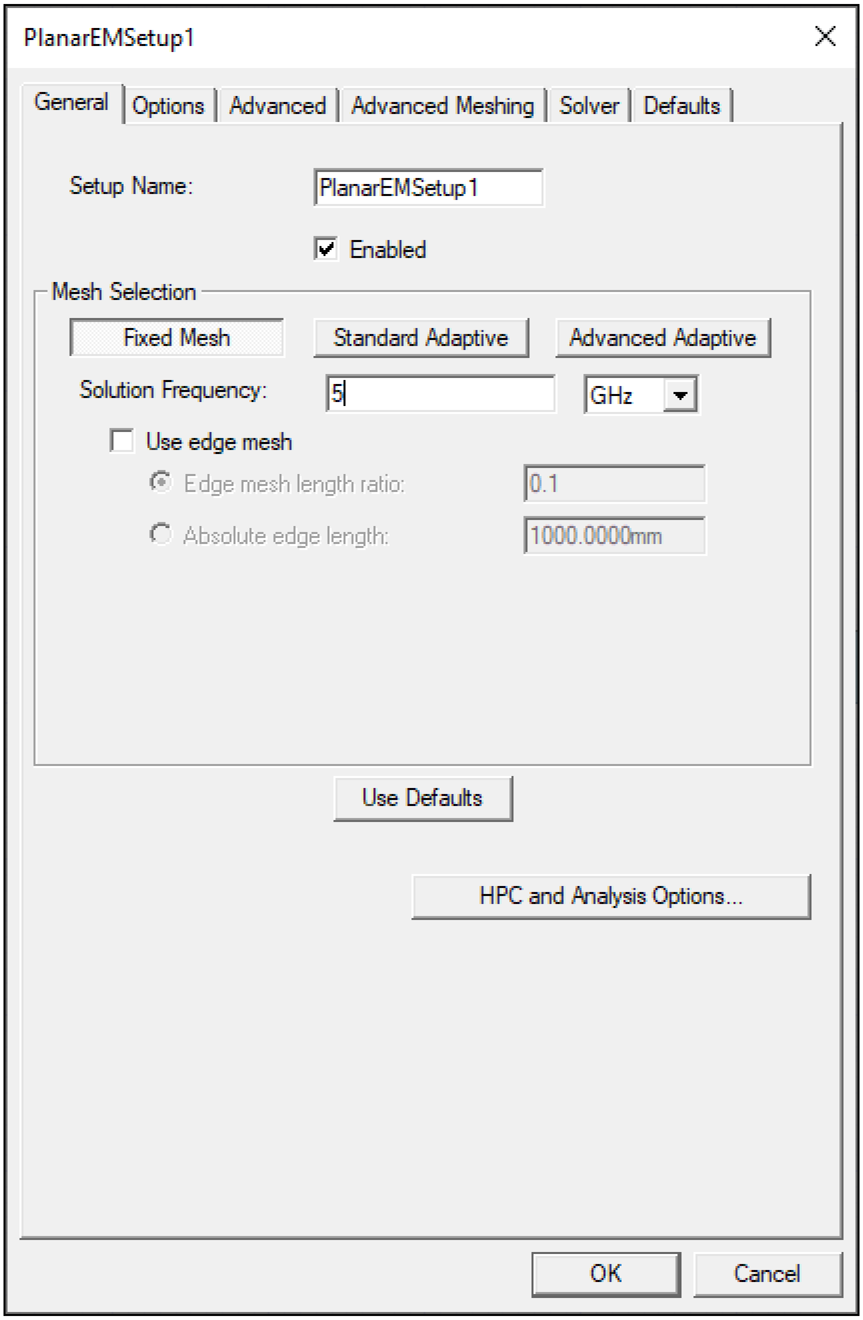
- Right-click **Analysis** in the **Project Manager** window and click **Add Planar EM Solution Setup** .



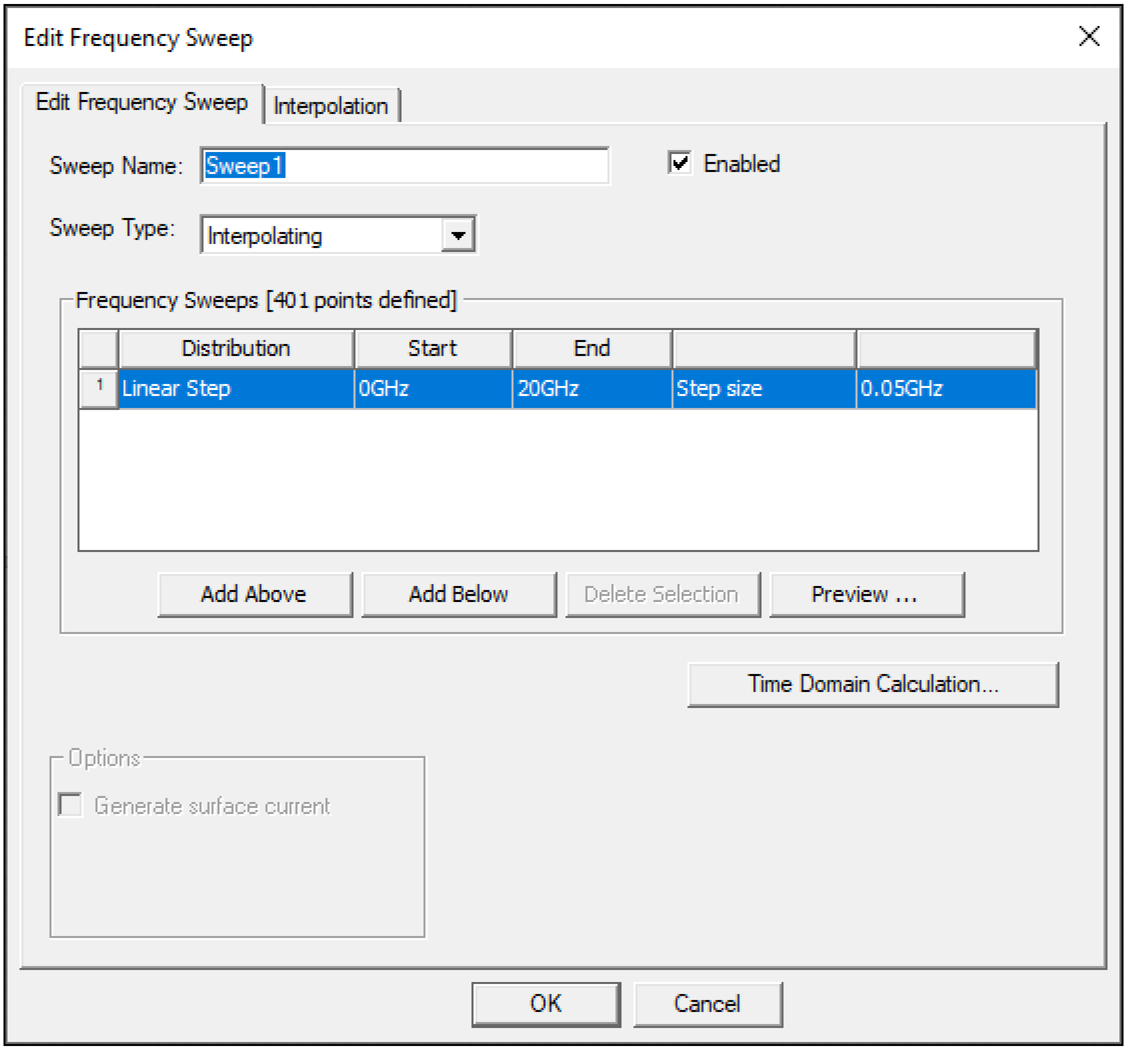
- From **HFSS 3D Layout**, select **Solution Setup** > **Add Planar EM Solution Setup**.



2. From the **PlanarEM Setup** window > **Mesh Selection** area, do the following:
   - Ensure **Fixed Mesh** is selected.
   - Enter **5** in the **Solution Frequency** field.



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



Continue to **Set Up Frequency Sweeps** to define the frequency sweep.

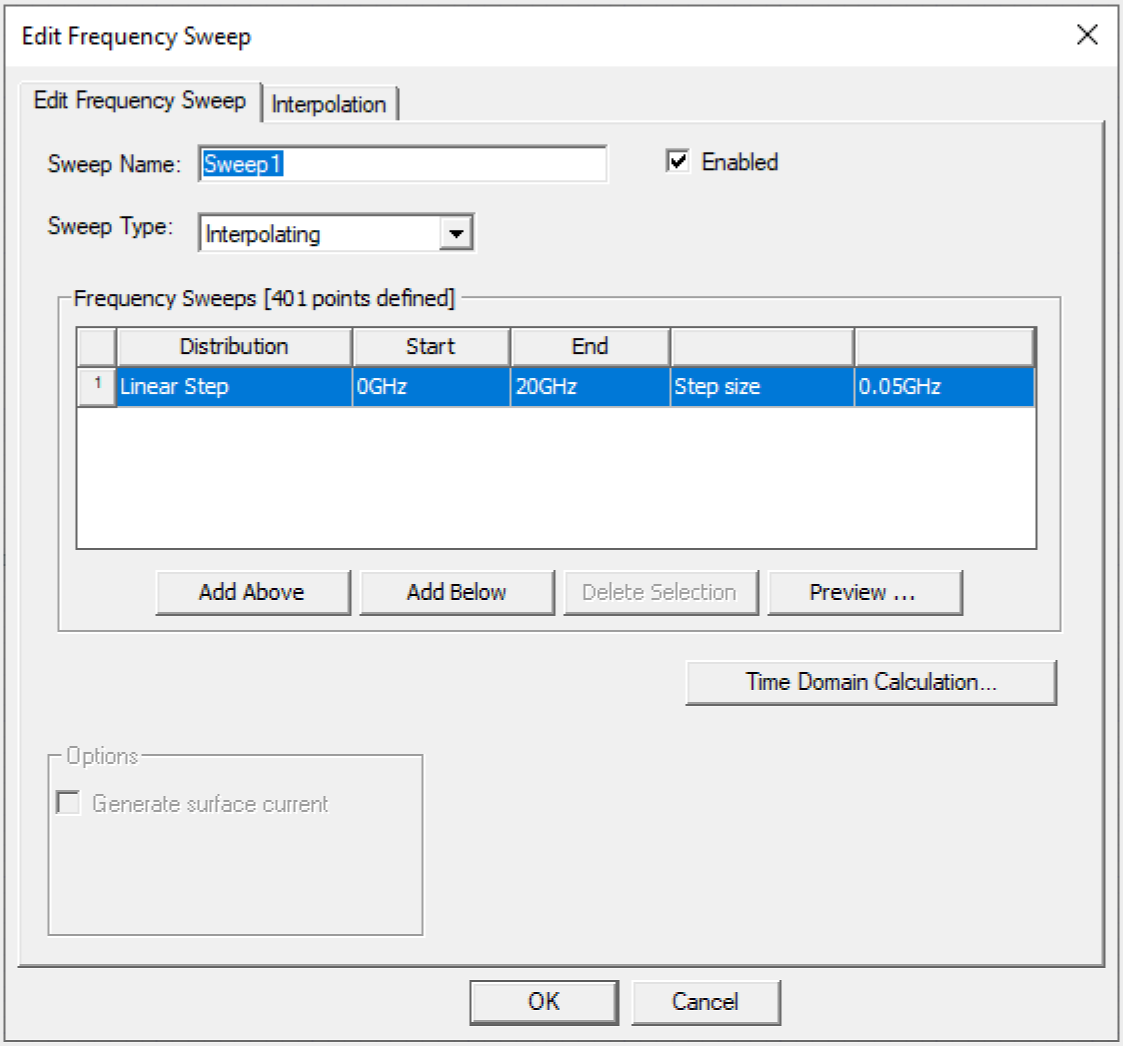
> **Note:**
>
> For more information on setting up an HFSS simulation, see **Add HFSS Solution Setup** in the product Help.

Set Up Frequency Sweeps

Complete these steps to add, set up, and define either an interpolating or discrete frequency sweep.

Note:

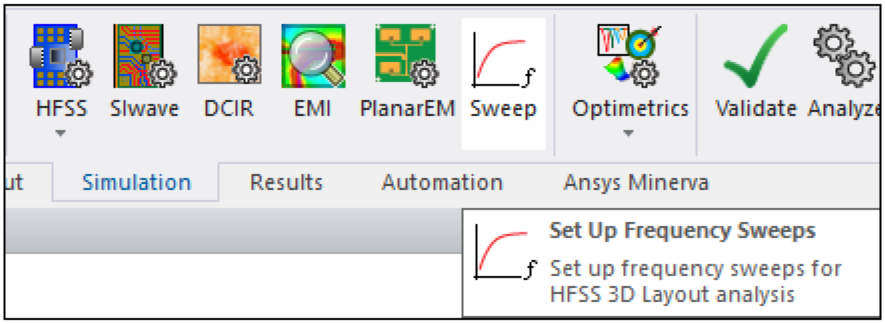
If this topic is reached in succession after completing the steps in **Set Up a Planar EM Analysis**, the **Edit Frequency Sweep** window should still be open. If the **Edit Frequency Sweep** window is open, skip to step 2.



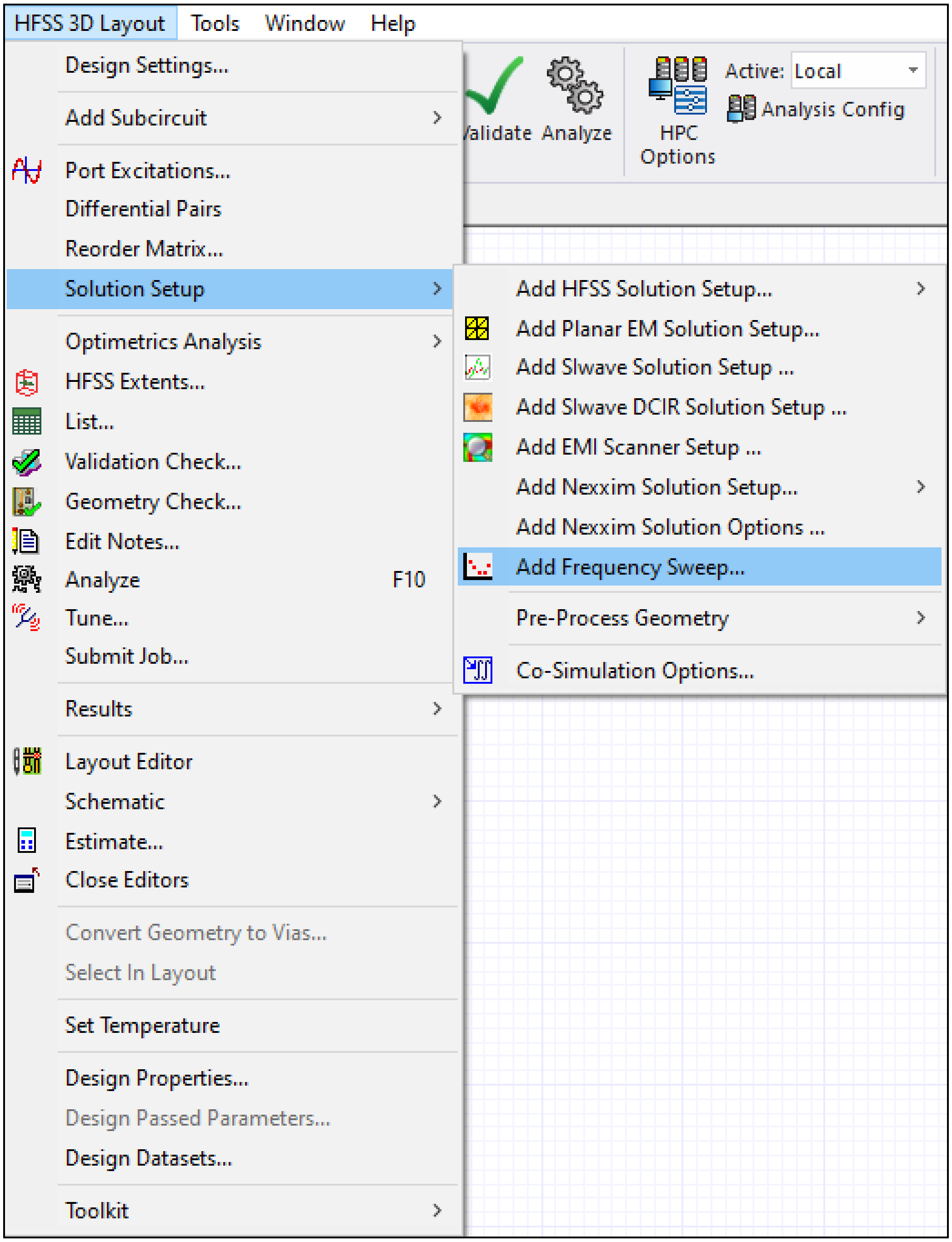
Add an Interpolating Frequency Sweep

1. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder to view the analysis setup (e.g., **PlanarEmSetup1**). Then add an interpolating frequency sweep associated with a Planar EM solution setup by doing one of the following:

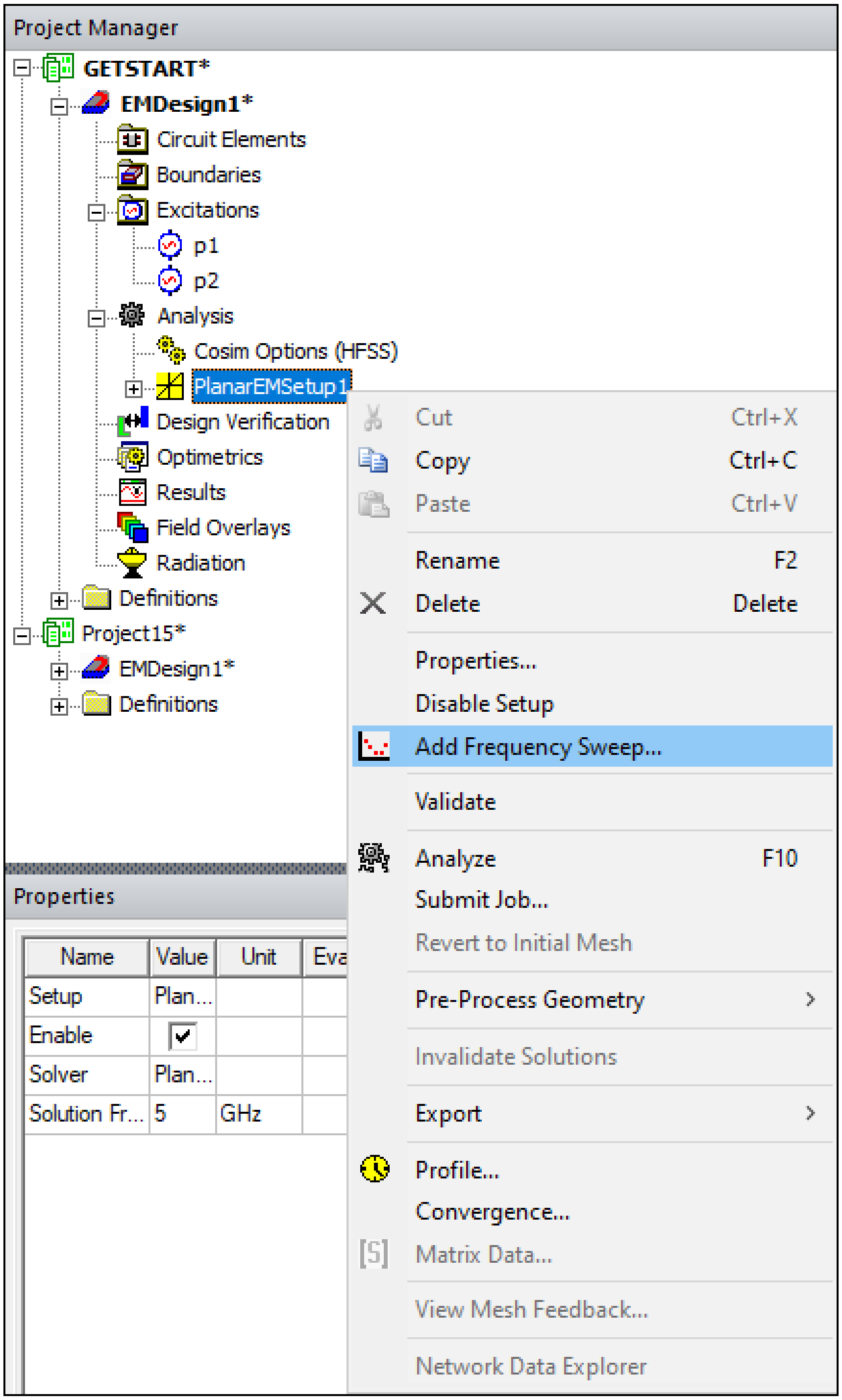
- Create a solution setup and the **Edit Frequency Sweep** window automatically appears (see **Set Up a Planar EM Analysis**).
- Select the analysis setup (e.g., **PlanarEmSetup1**). From the **Simulation** tab, click **Sweep** (**Set Up Frequency Sweeps**) from the ribbon.



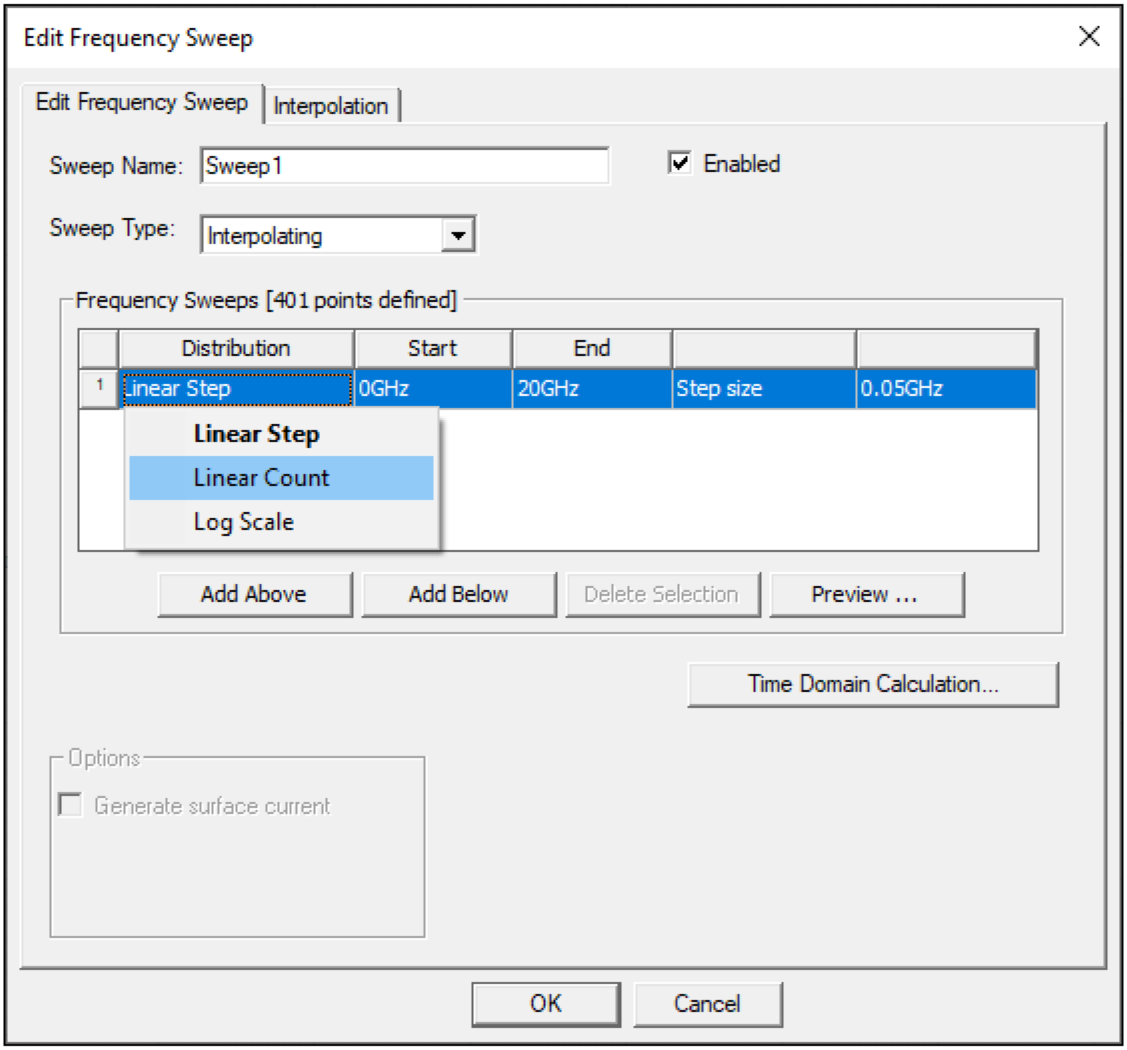
- Select the analysis setup (e.g., **PlanarEM Setup1**). From **HFSS 3D Layout**, select **Solution Setup** > **Add Frequency Sweep**.



- Right-click the analysis setup (e.g., **PlanarEMSetup1**) and select **Add Frequency Sweep**.



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

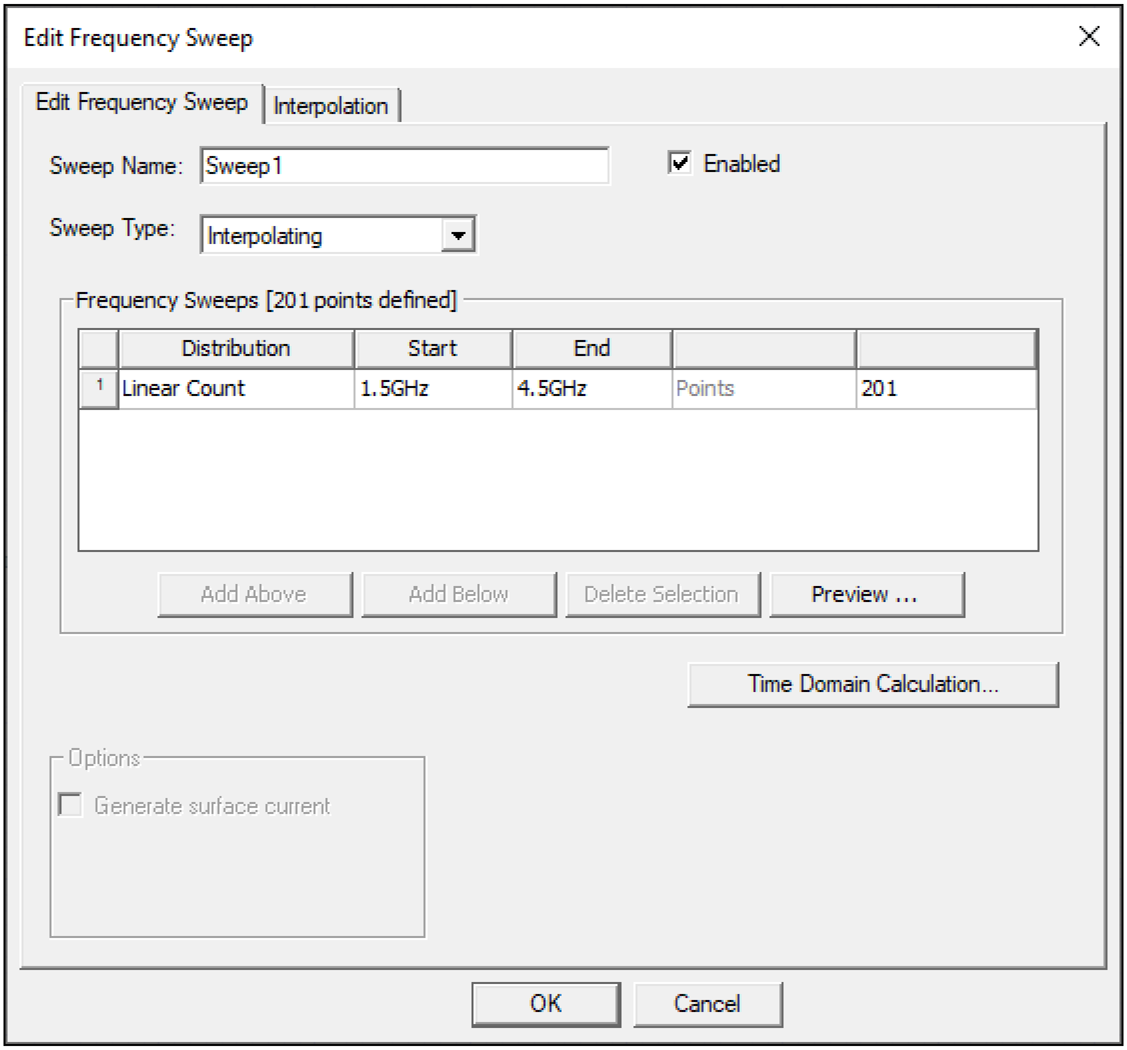


4. Enter the following parameters in the first row of the **Frequency Sweeps** table:

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

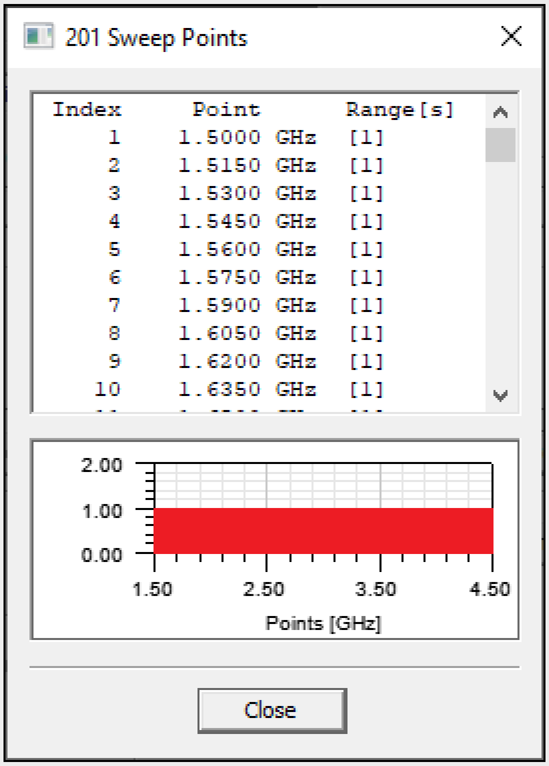
Note:

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.


Edit Frequency Sweep
Edit Frequency Sweep
Interpolation
Sweep Name:
Sweep1
Enabled
Sweep Type:
Interpolating
Frequency Sweeps [201 points defined]
Distribution
Start
End
1
Linear Count
1.5GHz
4.5GHz
Points
201
Add Above
Add Below
Delete Selection
Preview ...
Time Domain Calculation...
Options
Generate surface current
OK
Cancel

Note:

From the **Edit Frequency Sweep** window, click **Preview** to display the **Sweep Points** window.

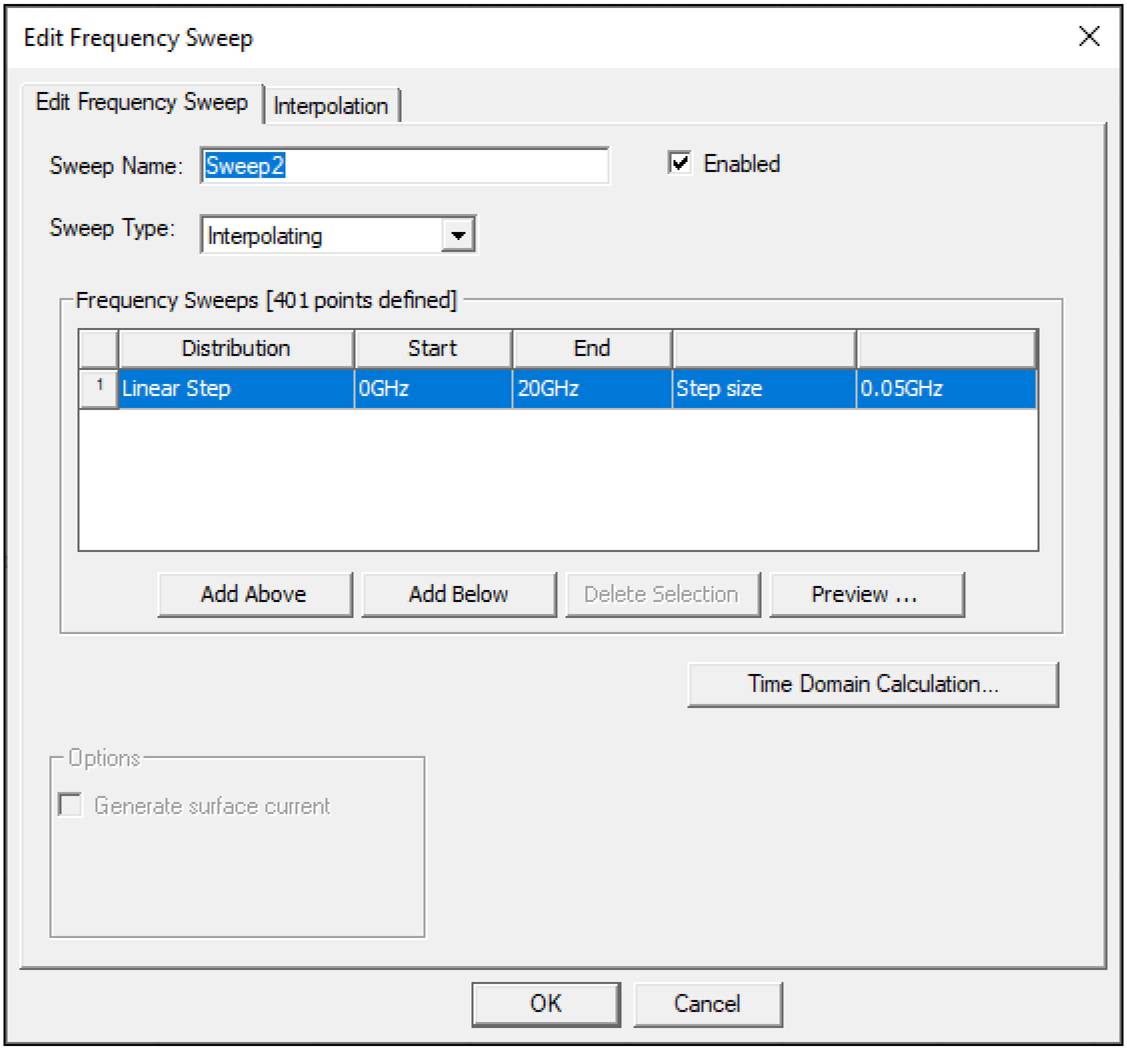


Click **Close** to return to the **Edit Frequency Sweep** window.

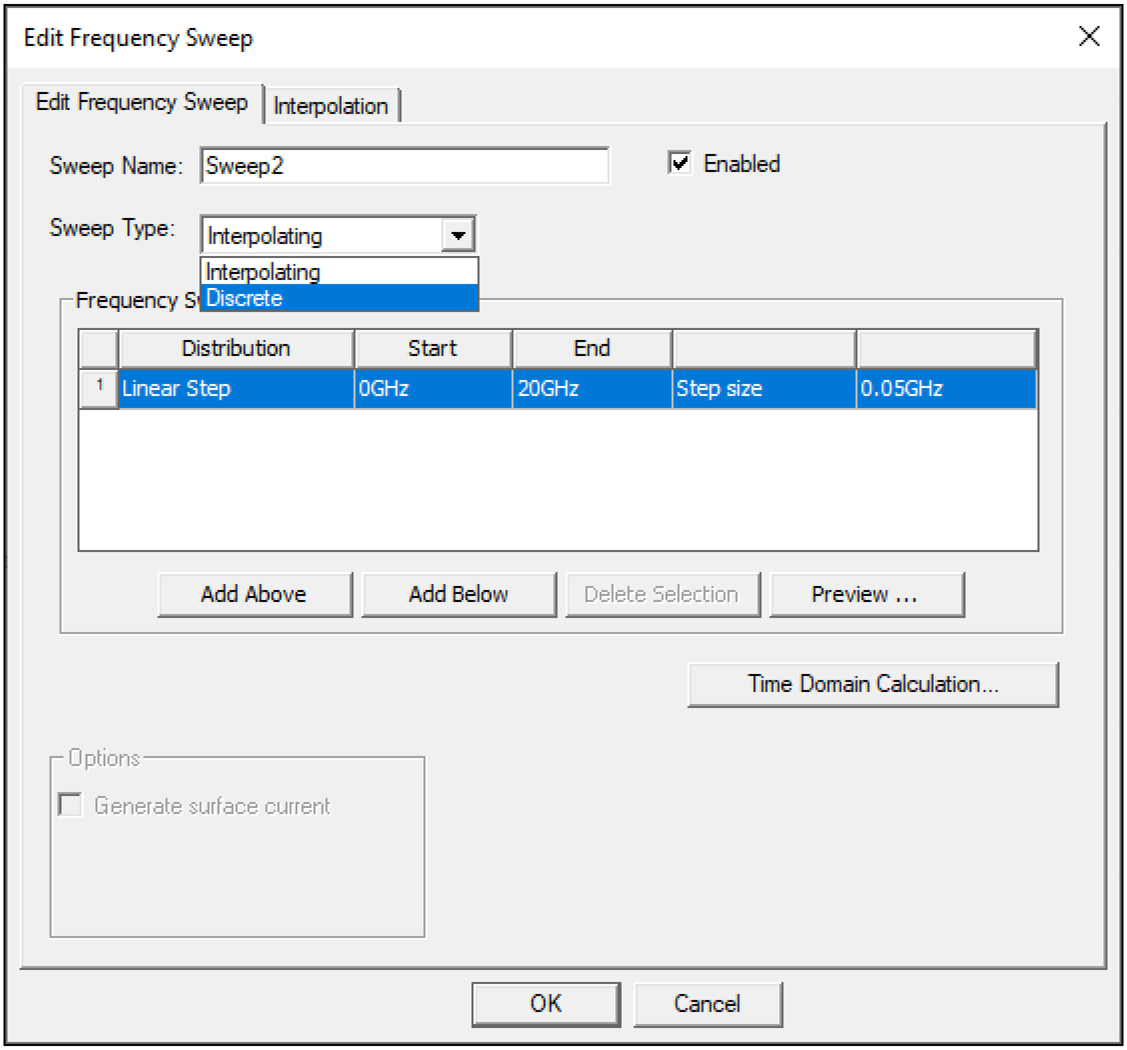
5. Click **OK** to add the interpolating sweep and close the **Edit Frequency Sweep** window.

Add a Discrete Frequency Sweep

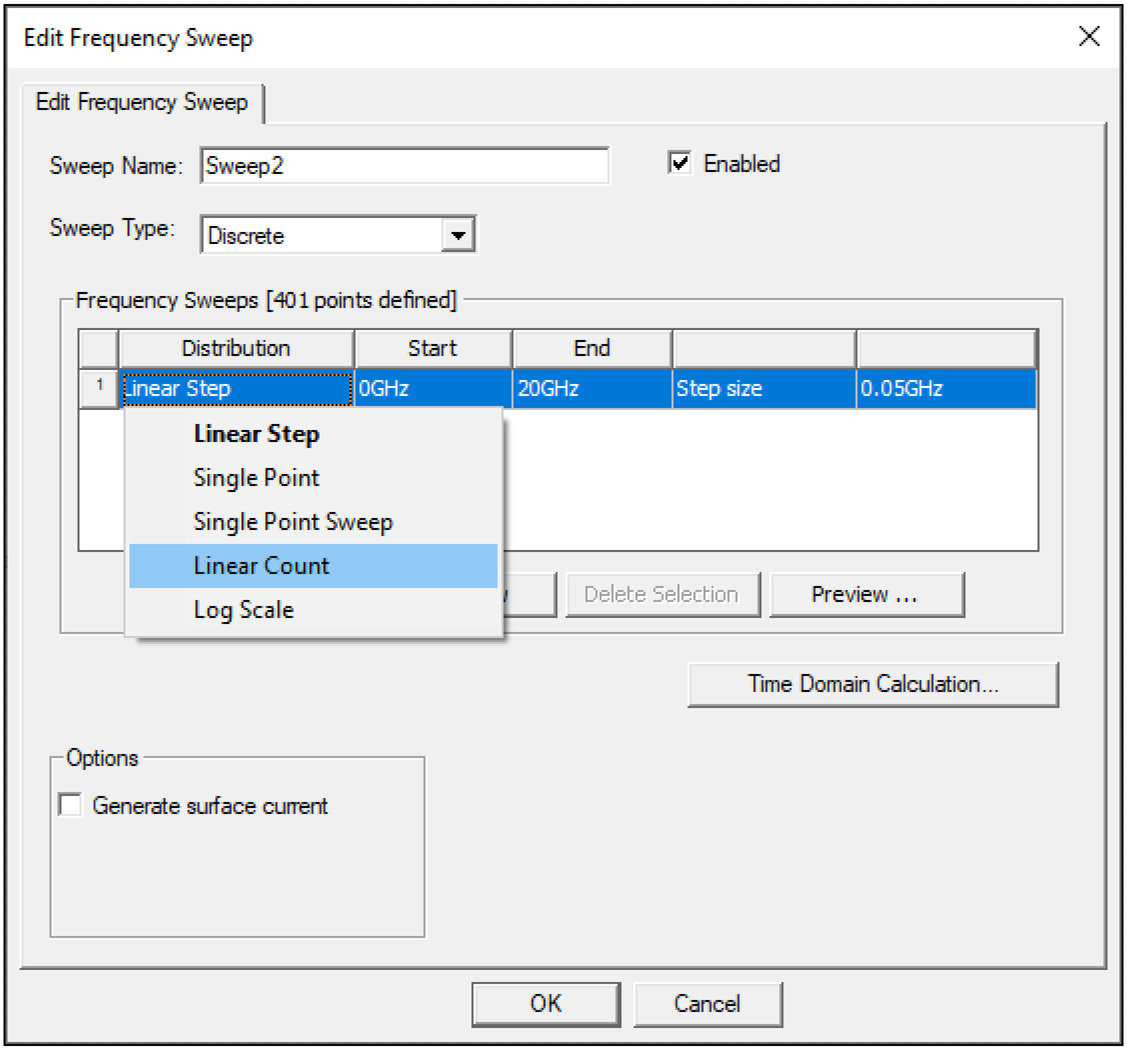
1. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click the analysis setup (e.g., **PlanarEMSetup1**) and select **Add Frequency Sweep** to open the **Edit Frequency Sweep** window.



2. Select **Discrete** from the **Sweep Type** drop-down menu.



3. Select **Linear Count** from the **Distribution** drop-down menu in the first row of the **Frequency Sweeps** table.



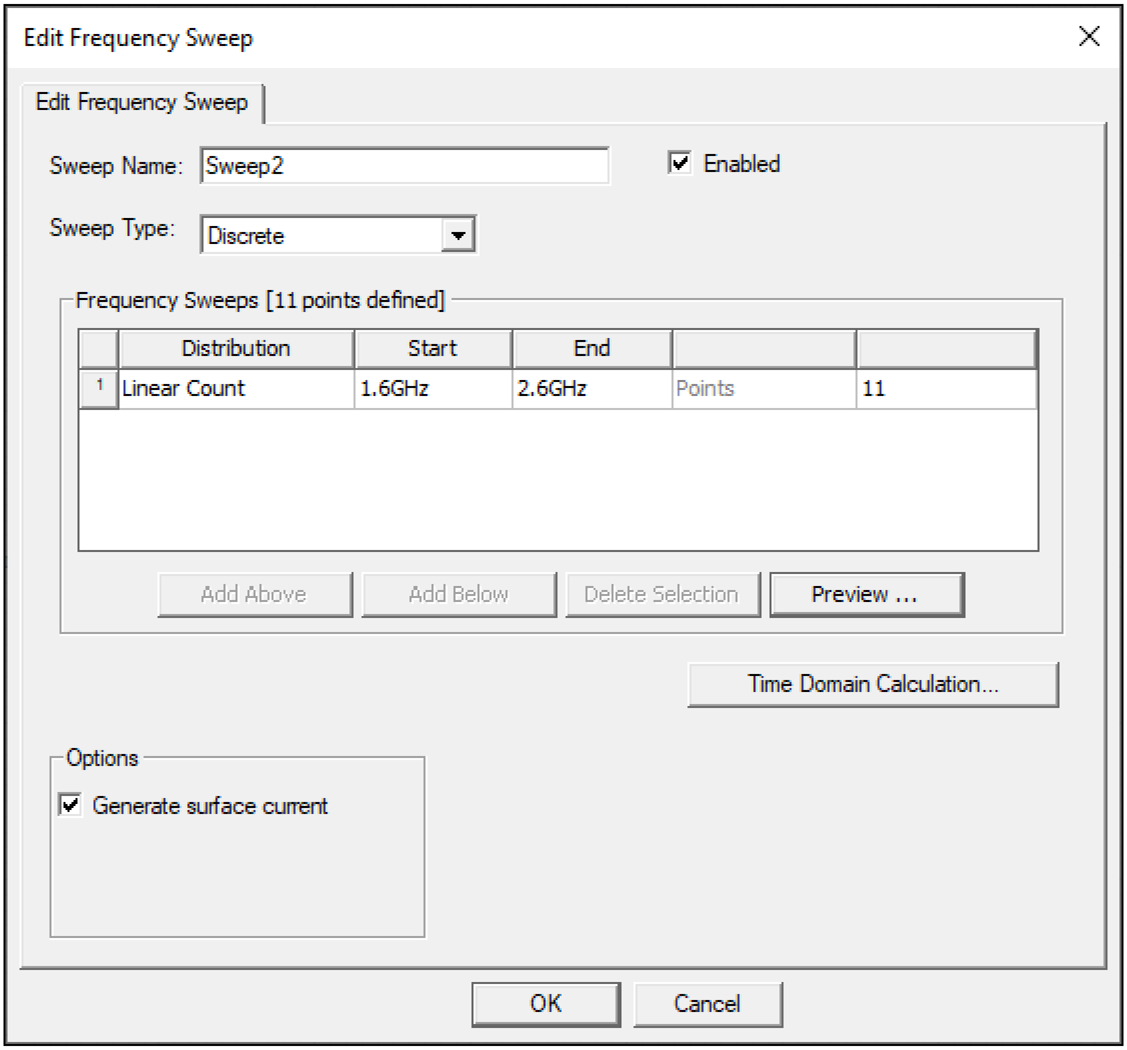
4. Enter the following parameters in the first row of the **Frequency Sweeps** table:

- In the **Start** column, enter **1.6** (GHz).
- In the **Stop** column, enter **2.6** (GHz).
- Enter **11** in the **Points** field.

Note:

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

5. From the **Options** area, check the **Generate surface current** box.



6. Click **OK** to finalize the discrete sweep and close the **Edit Frequency Sweep** window.

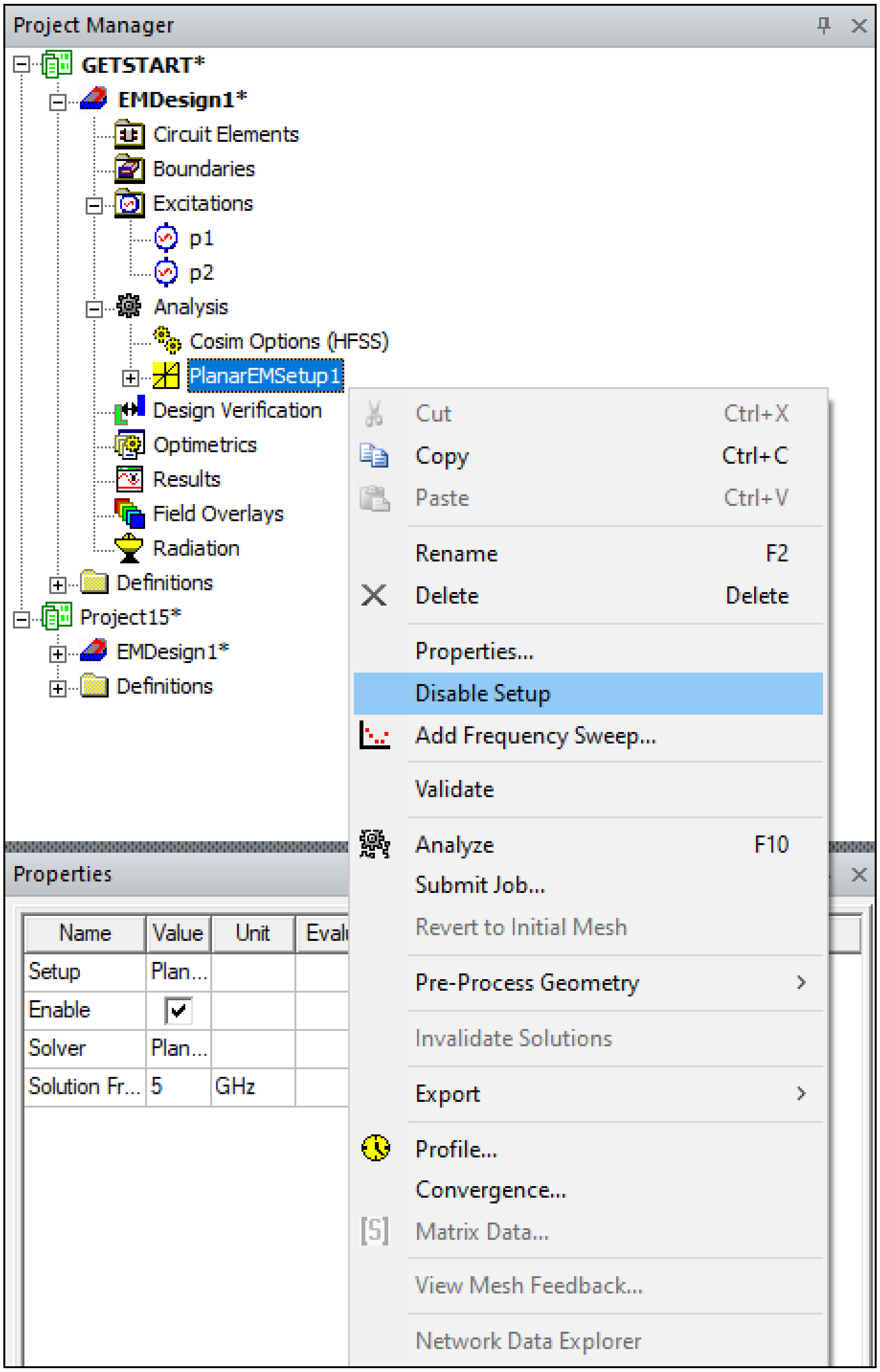
Continue to Explore Disabling Sweeps and Setups.

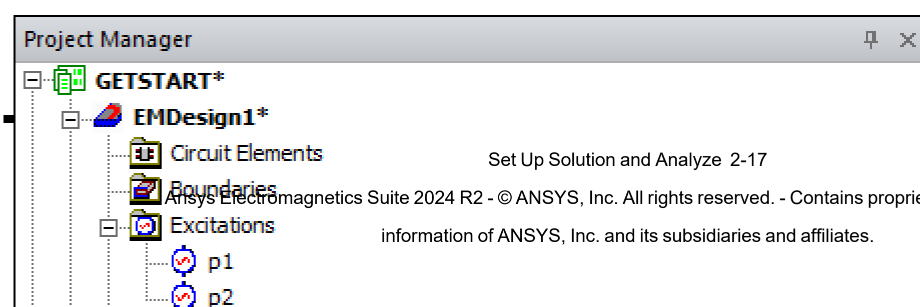
Deactivate/Activate Setups and Frequency Sweeps

Follow these instructions to activate or deactivate setup definitions and frequency sweeps.

Deactivate or Enable a Setup Definition

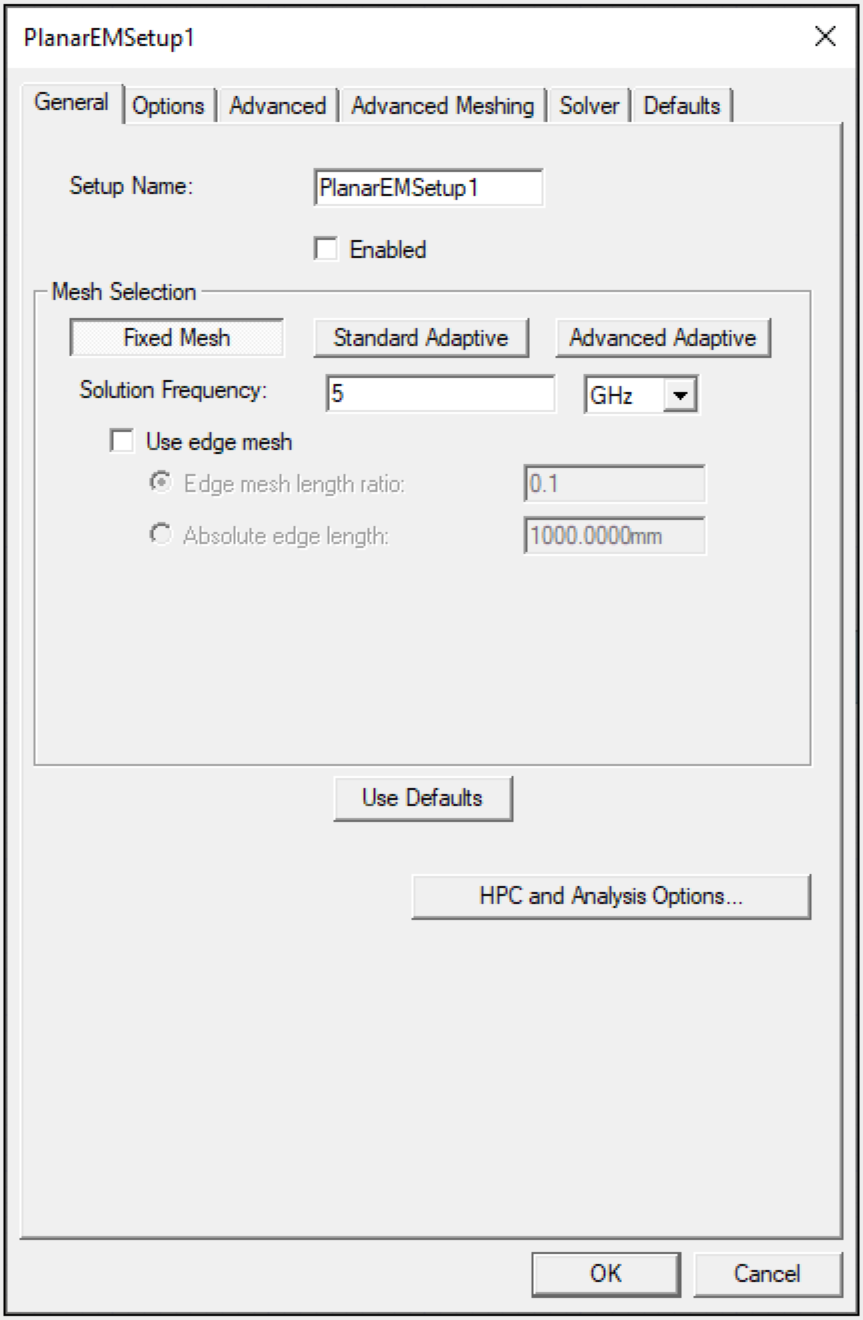
From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click the setup (e.g., **PlanarEmSetup1**) and select **Disable Setup** or **Enable Setup**. Only one option will be available, depending from the current status of the setup (e.g., if the setup is currently deactivated, only **Enable Setup** will appear in the shortcut menu).





Note:

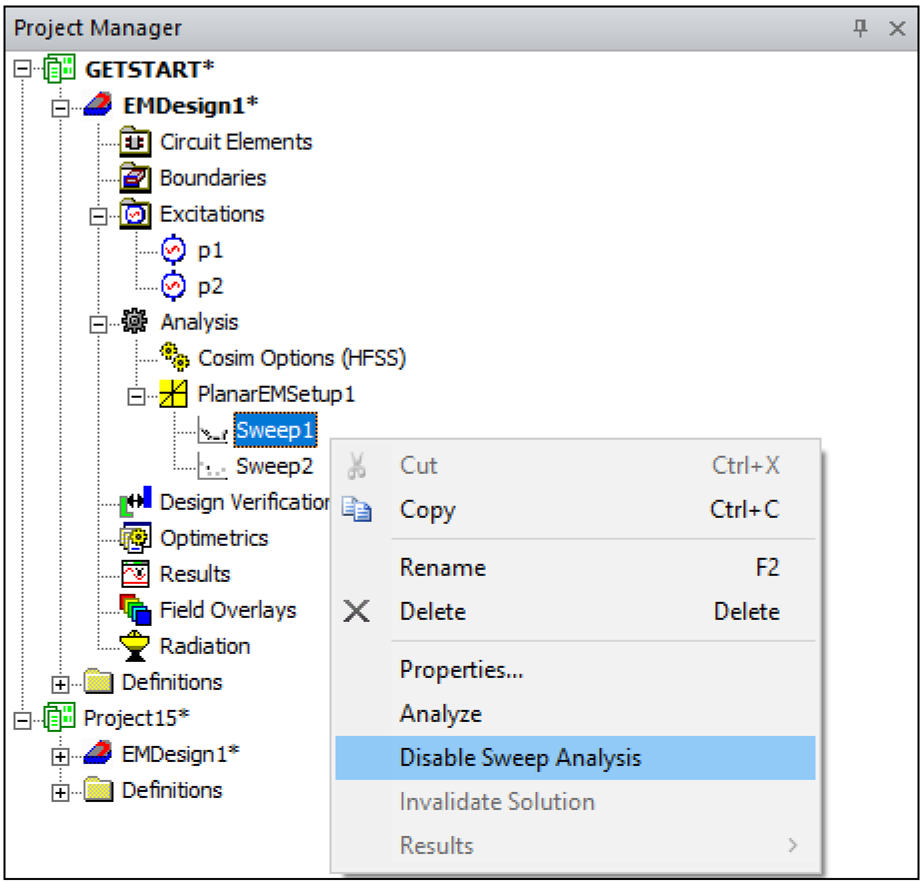
Alternatively, deactivate/activate an analysis setup by either double-clicking the setup (i.e., located at the **Project Manager**window > **Project Tree** > [***active design folder***] > **Analysis** folder directory) or right-clicking the setup and selecting **Properties** to open the **PlanarEMSetup** window. Beneath the **Setup Name** field, uncheck or check the **Enabled** box and click **OK** to commit the change and close the window.

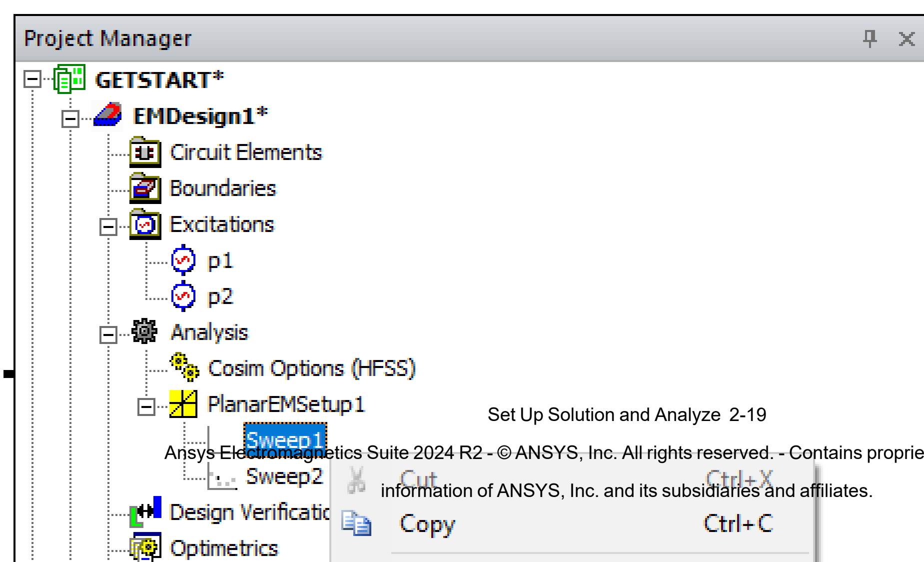


When an analysis setup is deactivated, any sweep associated with it has no effect, even if an associated sweep is enabled.

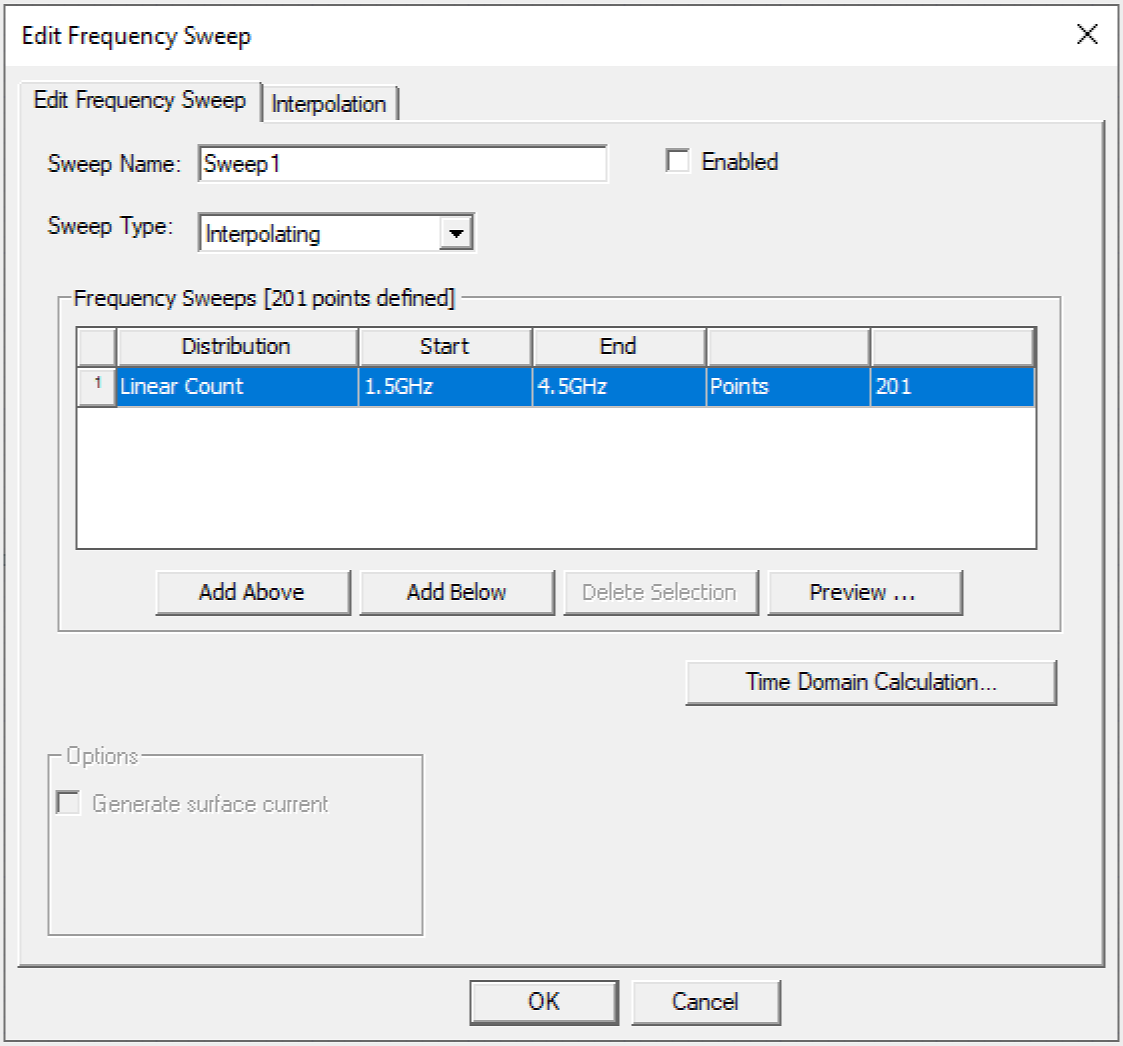
Deactivate or Activate a Sweep Definition

From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder > setup (e.g., **PlanarEMSetup1**). Then right-click the sweep (e.g., **Sweep1**) and select **Disable Sweep Analysis** or **Enable Sweep Analysis**. Only one option will be available, depending from the current status of the setup (e.g., if the sweep is currently deactivated, only **Enable Sweep Analysis** will appear in the shortcut menu).





Note: Alternatively, deactivate/activate a sweep analysis by either double-clicking the sweep (i.e., located at the **Project Manager**window > **Project Tree** > **[*active design folder*]** > **Analysis** folder > setup (e.g., PlanarEMSetup1) directory) or right-clicking the sweep and select **Properties** to open the **Edit Frequency Sweep** window. Uncheck or check the **Enabled** box and click **OK** to commit the change and close the window.



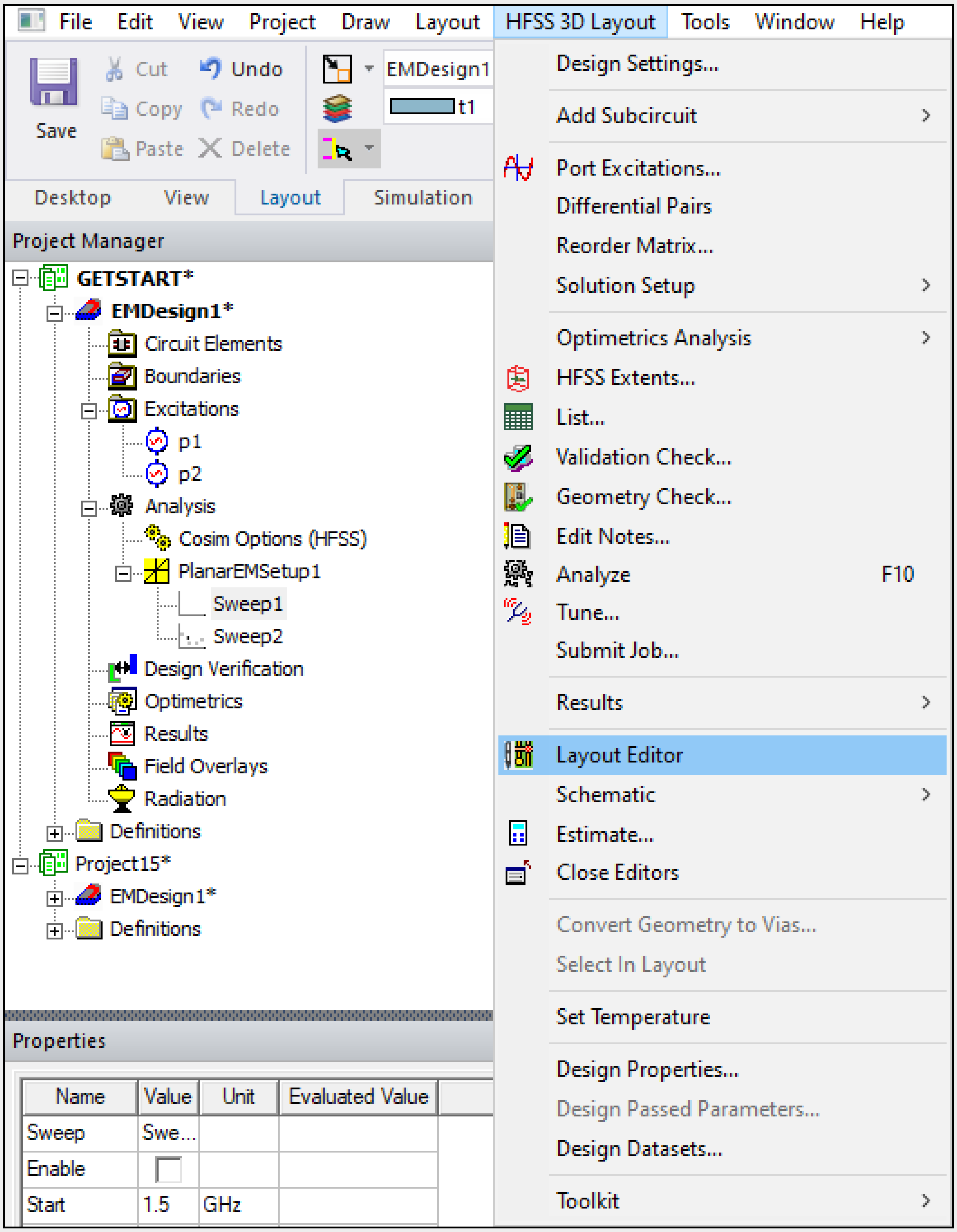
Continue to View the Mesh.

View the Mesh

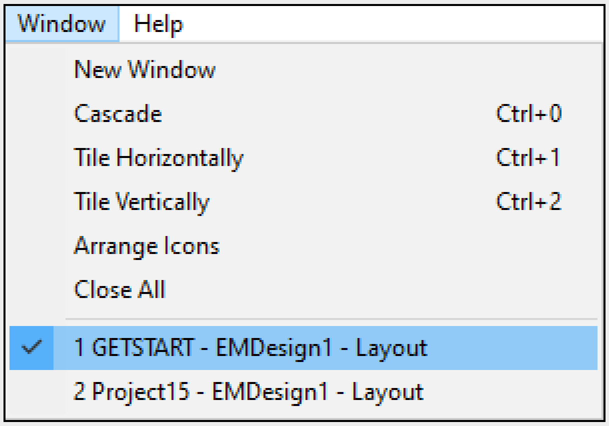
Follow these instructions to view the active design in a number of planar or 3D views.

Note:

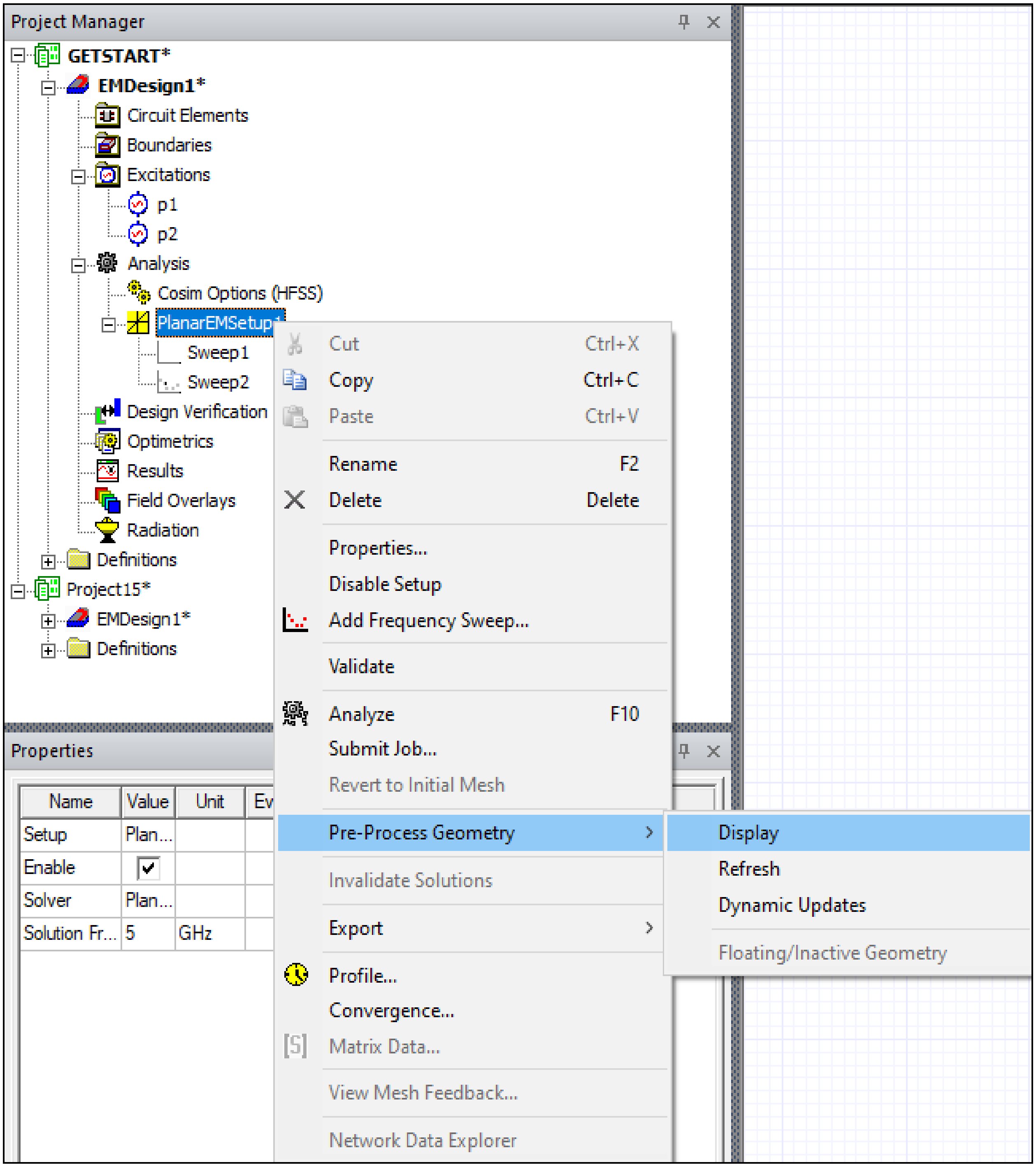
If the **Layout Editor** is closed or another window is dominant, navigate to **HFSS 3D Layout** and select **Layout Editor**.



If the **Layout Editor** is still open but no longer dominant, navigate to **Window** and select the active design project from the available options.

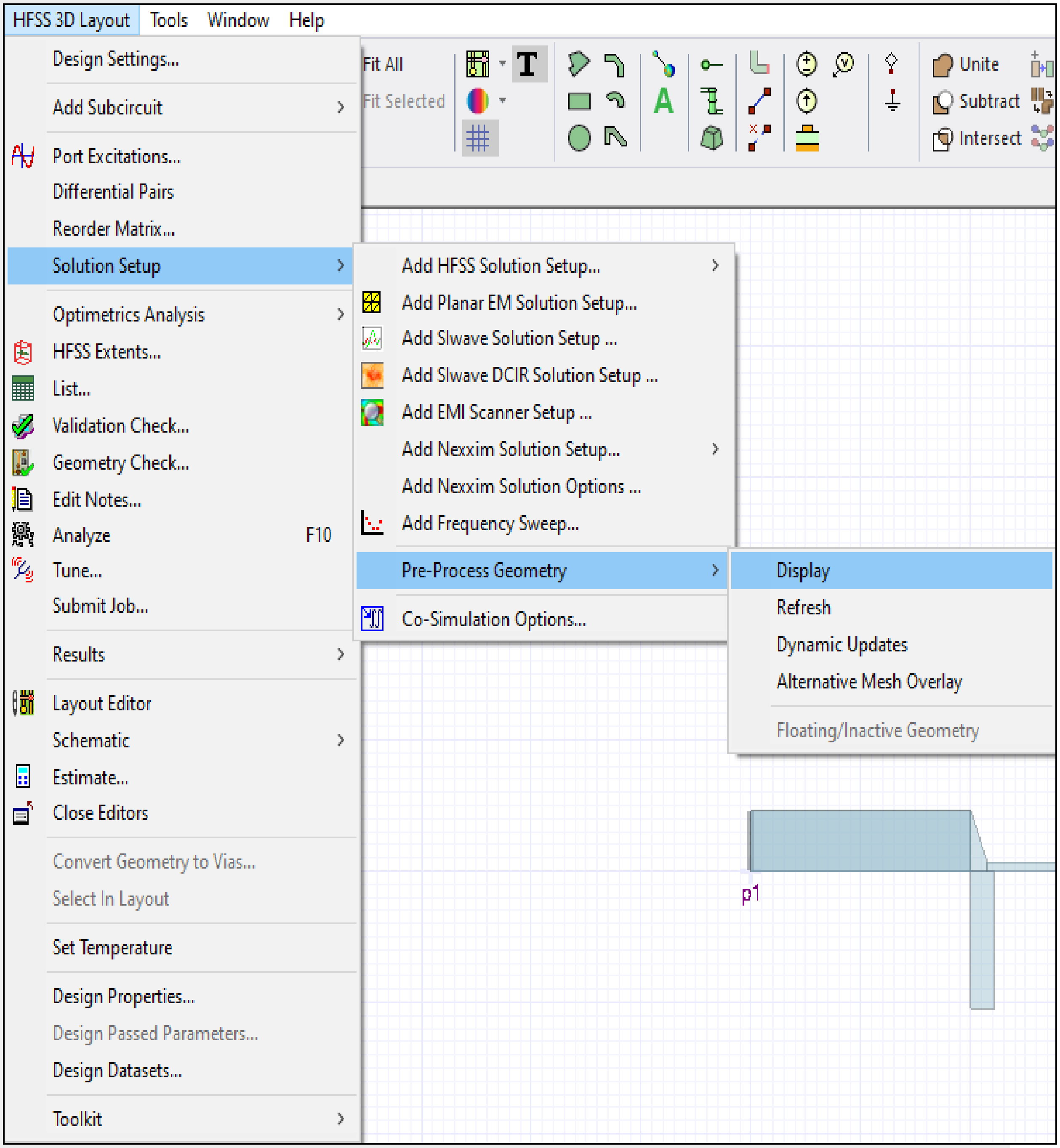


From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click the setup (e.g., **PlanarEmSetup1**) and select **Pre-Process Geometry** > **Display**.

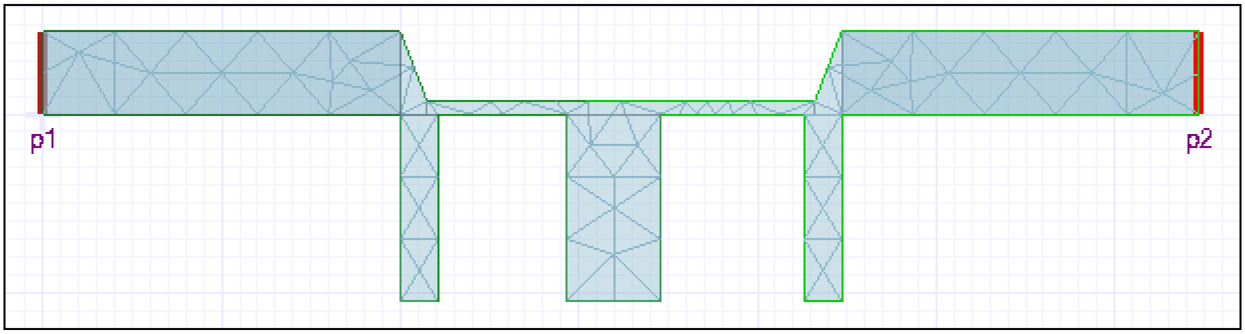

Project Manager
GETSTART*
EMDesign1*
Circuit Elements
Boundaries
Excitations
p1
p2
Analysis
Cosim Options (HFSS)
PlanarEMSetup1
Sweep1
Sweep2
Design Verification
Optimetrics
Results
Field Overlays
Radiation
Definitions
Project15*
EMDesign1*
Definitions
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Rename F2
Delete Delete
Properties...
Disable Setup
Add Frequency Sweep...
Validate
Analyze F10
Submit Job...
Revert to Initial Mesh
Pre-Process Geometry
Display
Refresh
Dynamic Updates
Floating/Inactive Geometry
Invalidate Solutions
Export
Profile...
Convergence...
Matrix Data...
View Mesh Feedback...
Network Data Explorer
Properties
Name
Value
Unit
Ev
Setup
Plan...
Enable
Solver
Plan...
Solution Fr...
5
GHz

Note:

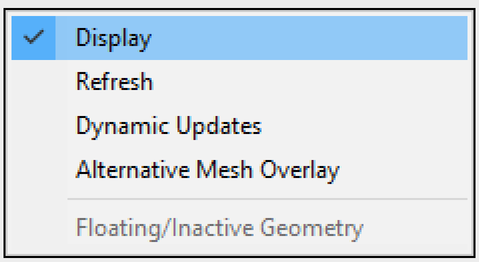
Alternatively, left-click the setup (e.g., **PlanarEmSetup1**) and navigate to **HFSS 3D Layout** > **Solution Setup** > **Pre-Process Geometry** > **Display**.



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



The mesh does not display on a layer that is not visible, and if there are self-intersecting mesh edges, they will appear highlighted in yellow.

> **Note:**
>
> If this topic has been reached in succession after completing the steps in **Explore Disabling Sweeps and Setups**, leave the mesh visible on the model. However, the mesh can be deactivated by repeating either method described in this topic to view the mesh. A check mark appears adjacent to **Display** when the mesh is visible and disappears when the mesh is hidden.
>
> 
>

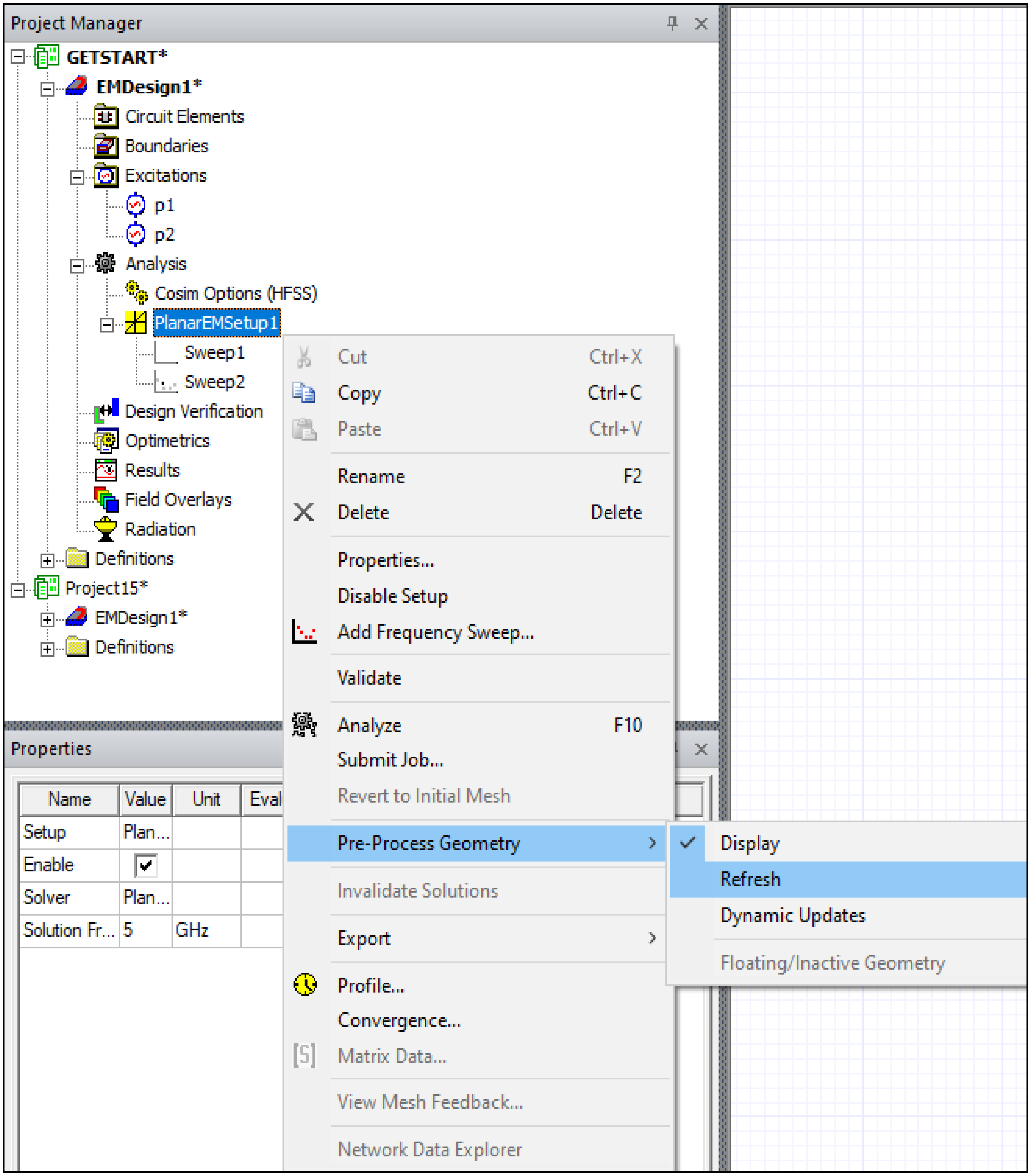
Continue to Explore Dynamic Mesh Updates.

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 an object is altered. The following instructions explain how to refresh the mesh manually or enable **Dynamic Mesh** to update the mesh in real time.

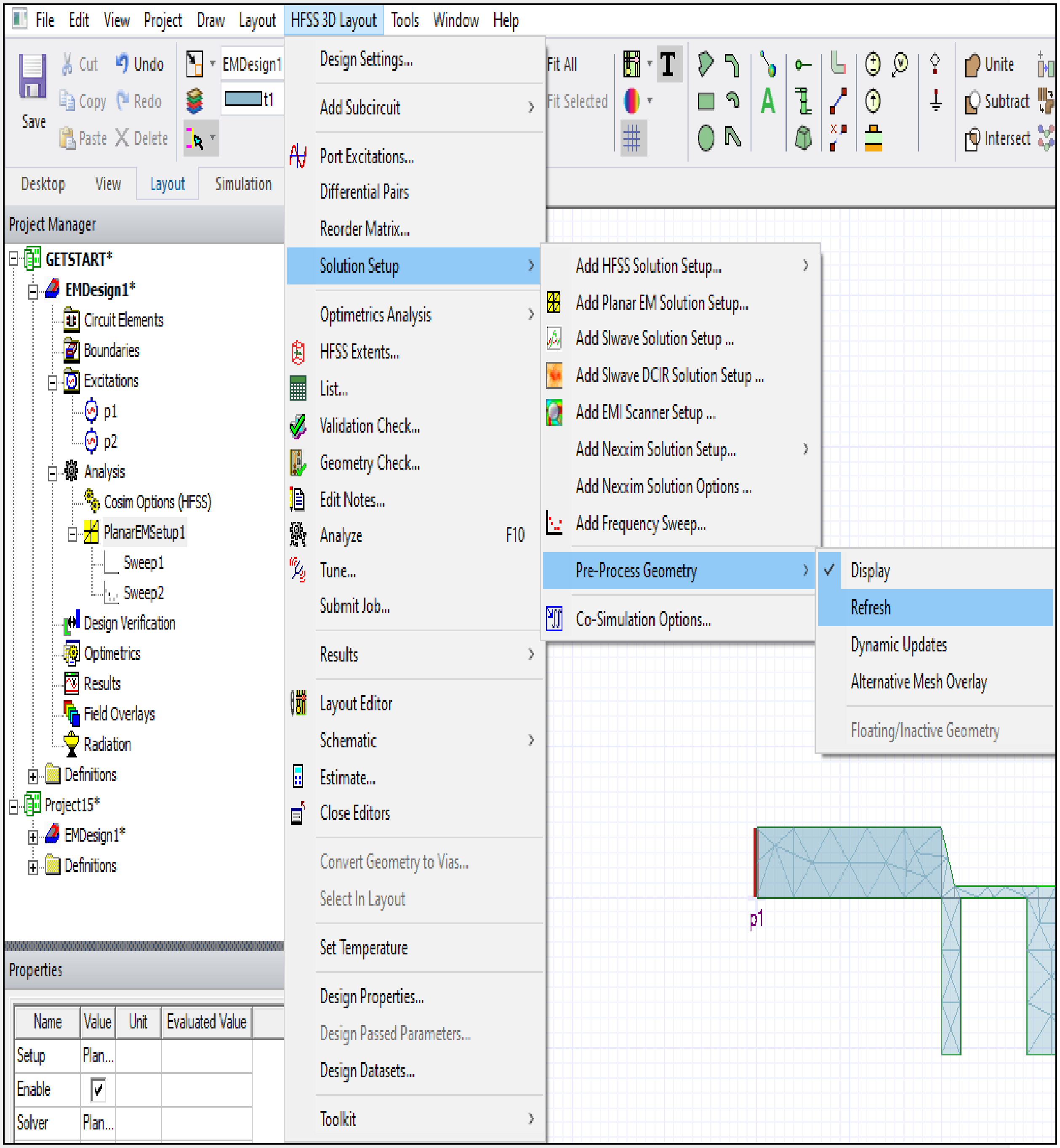
Update the Mesh Manually

From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click the setup (e.g., **PlanarEmSetup1**) and select **Pre-Process Geometry** > **Refresh**.


Project Manager
GETSTART*
EMDesign1*
Circuit Elements
Boundaries
Excitations
p1
p2
Analysis
Cosim Options (HFSS)
PlanarEMSetup1
Sweep1
Sweep2
Design Verification
Optimetrics
Results
Field Overlays
Radiation
Definitions
Project15*
EMDesign1*
Definitions
Properties
Name
Value
Unit
Eval
Setup
Plan...
Enable
Solver
Plan...
Solution Fr...
5
GHz
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Rename F2
Delete Delete
Properties...
Disable Setup
Add Frequency Sweep...
Validate
Analyze F10
Submit Job...
Revert to Initial Mesh
Pre-Process Geometry
Invalidate Solutions
Export
Profile...
Convergence...
Matrix Data...
View Mesh Feedback...
Network Data Explorer
Display
Refresh
Dynamic Updates
Floating/Inactive Geometry

Note:

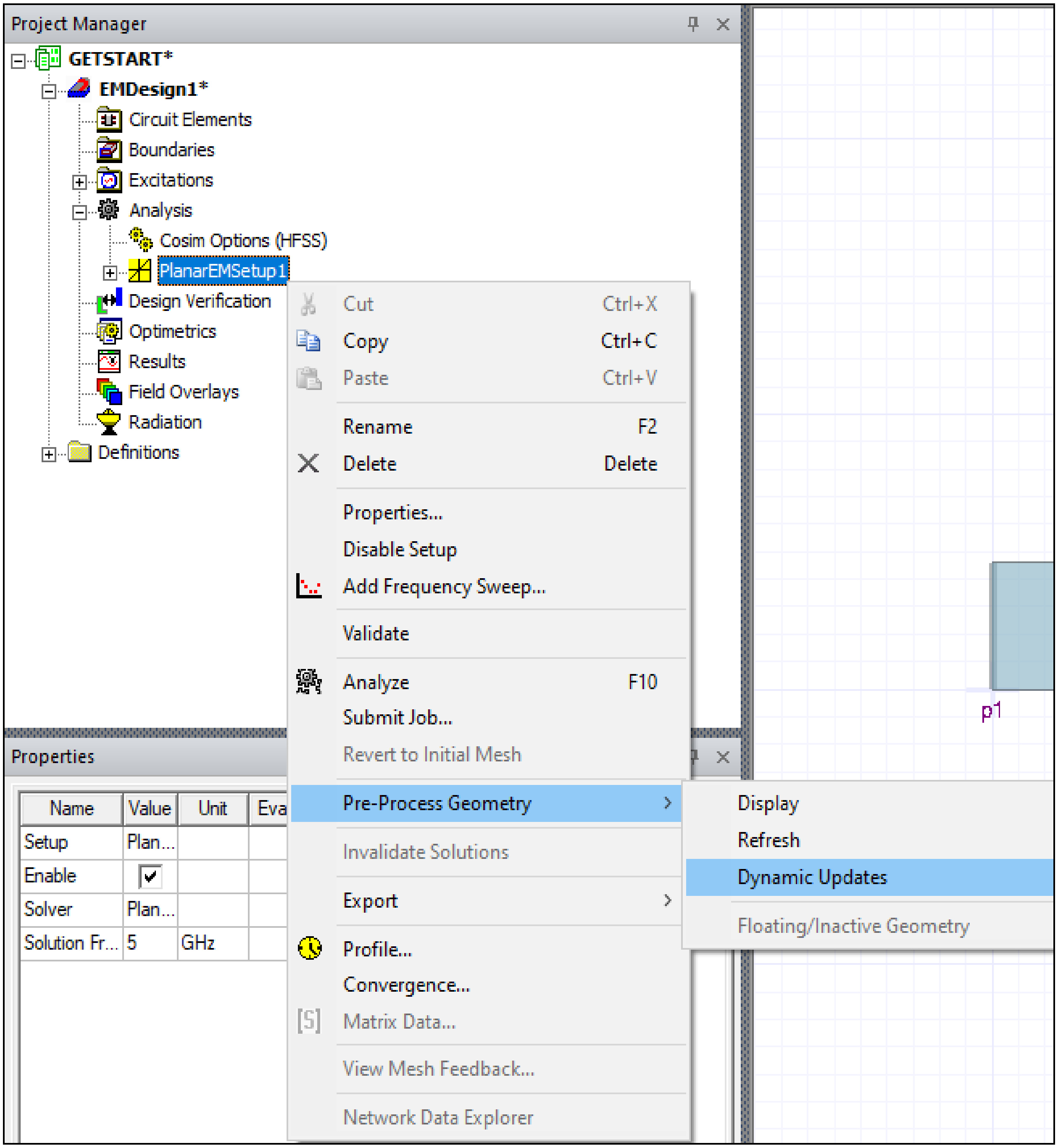
Alternatively, left-click the setup (e.g., **PlanarEmSetup1**) and navigate to **HFSS 3D Layout** > **Solution Setup** > **Pre-Process Geometry** > **Refresh**.



> **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.

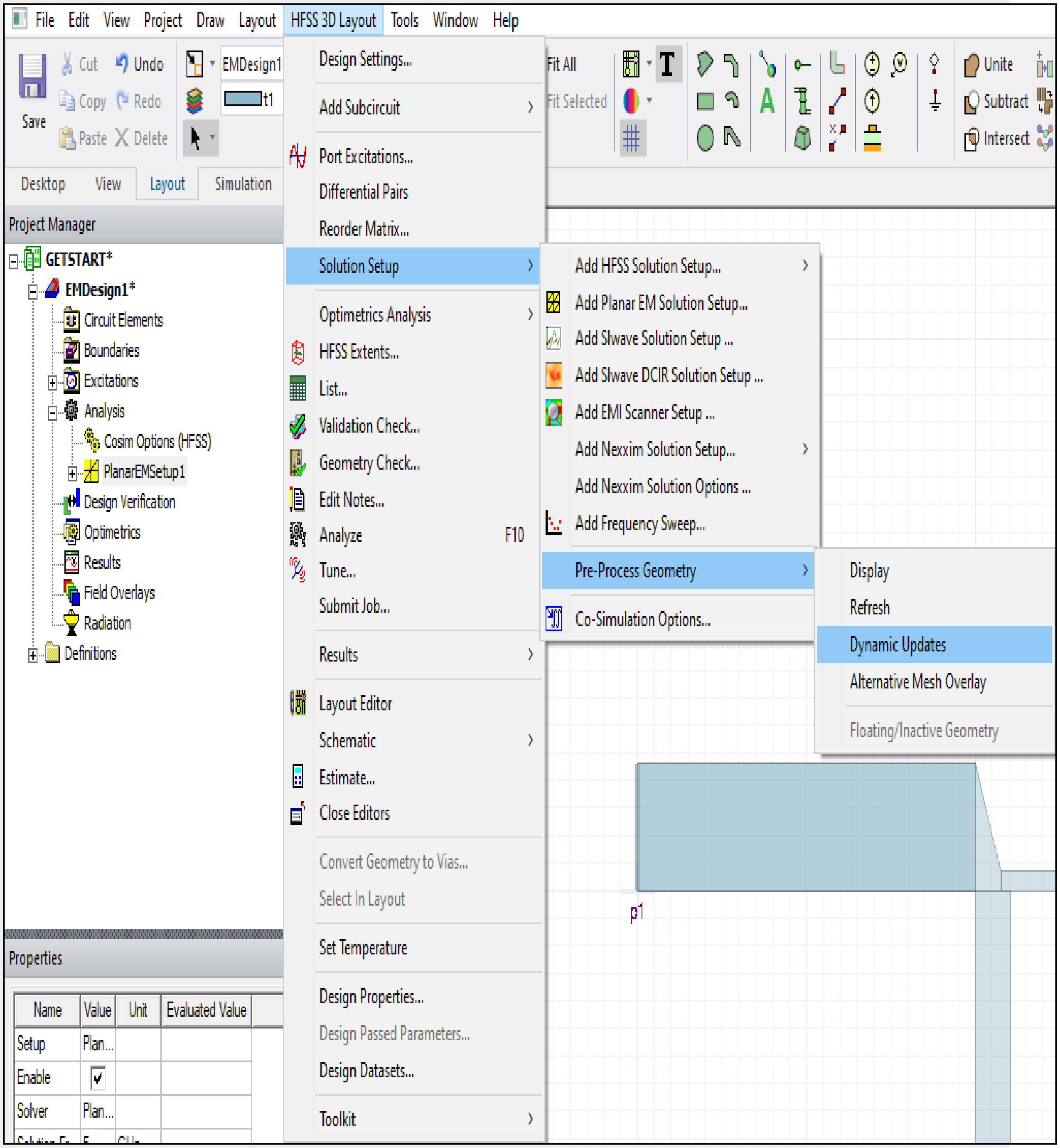
Enable Dynamic Updates

From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder. Then right-click the setup (e.g., **PlanarEMSetup1**) and select **Pre-Process Geometry** > **Dynamic Updates**.


Project Manager
GETSTART*
EMDesign1*
Circuit Elements
Boundaries
Excitations
Analysis
Cosim Options (HFSS)
PlanarEMSetup1
Design Verification
Optimetrics
Results
Field Overlays
Radiation
Definitions
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Rename F2
Delete Delete
Properties...
Disable Setup
Add Frequency Sweep...
Validate
Analyze F10
Submit Job...
Revert to Initial Mesh
Pre-Process Geometry
Invalidate Solutions
Export
Profile...
Convergence...
Matrix Data...
View Mesh Feedback...
Network Data Explorer
Display
Refresh
Dynamic Updates
Floating/Inactive Geometry
Properties
Name
Value
Unit
Eva
Setup
Plan...
Enable
Solver
Plan...
Solution Fr...
5
GHz
p1

Note:

Alternatively, left-click the setup (e.g., **PlanarEmSetup1**) and navigate to **HFSS 3D Layout** > **Solution Setup** > **Pre-Process Geometry** > **Dynamic Updates**.

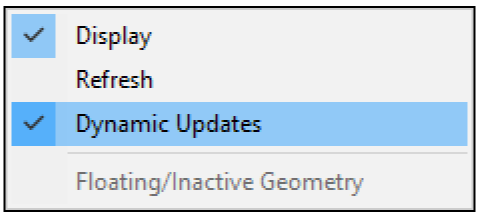


Reset the Mesh After Using Undo or Redo

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**, as previously described, to reset the mesh.

Deactivate Dynamic Updates

Deactivate dynamic updates by repeating either method described in this topic to enable dynamic updates. A check mark appears adjacent to **Dynamic Updates** when dynamic updates are enabled and disappears when dynamic updates is deactivated.



> **Important:**
>
> Before continuing, **Save** the design.

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 chosen.

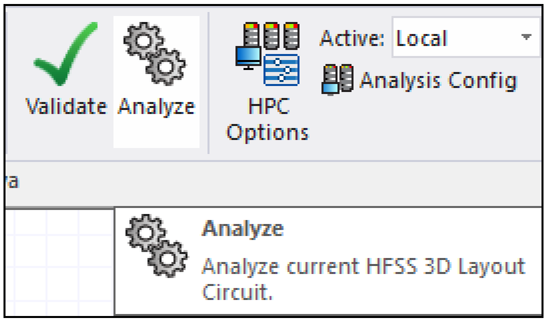
After experimenting, **deactivate the mesh**.

Continue to Run the Analysis.

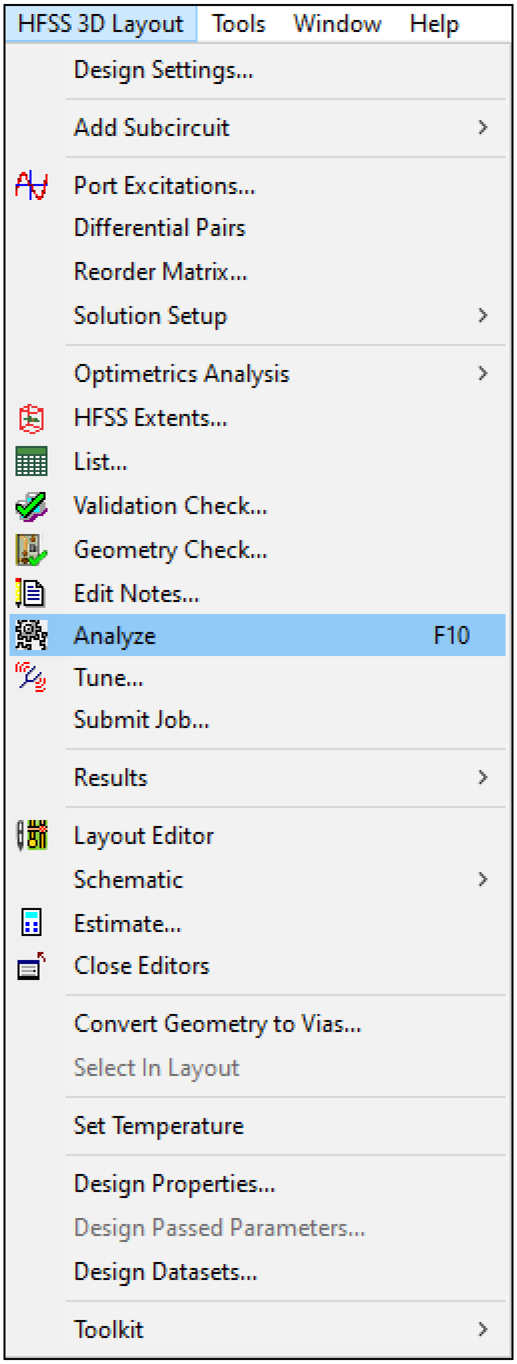
Run Analyses and Choose Analyses Options

To sequentially run all analysis setups and associated frequency sweeps in the active design, do one of the following:

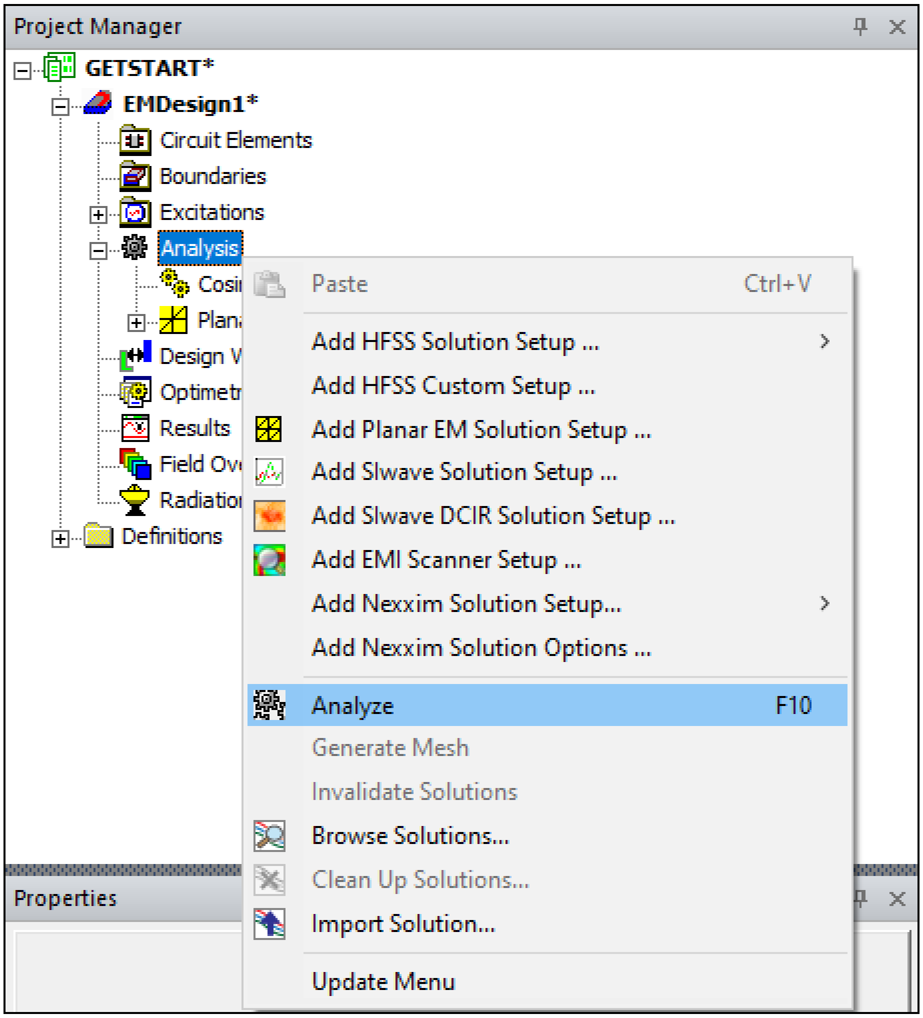
- From the **Simulation** tab, select **Analyze**.



- From **HFSS 3D Layout**, select **Analyze**.

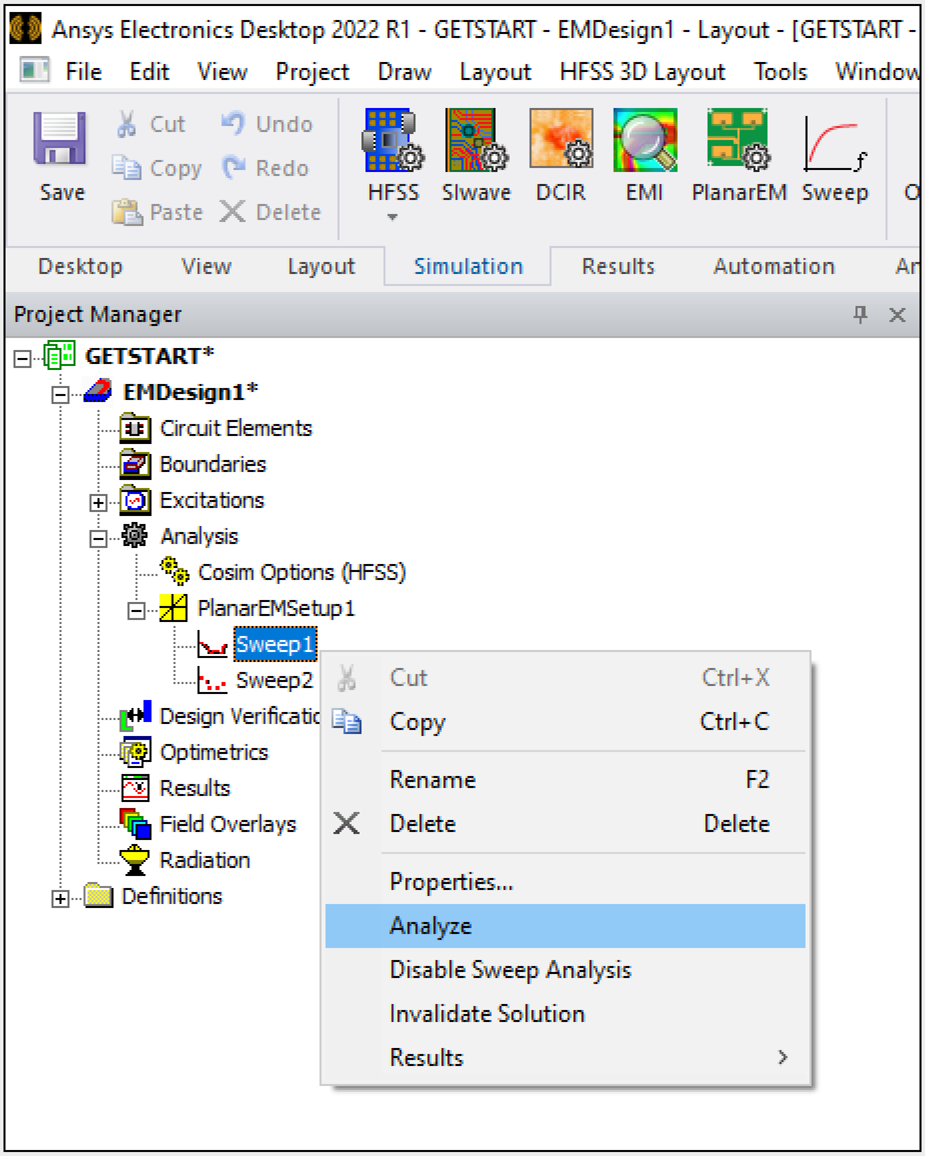


- From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]**. Then right-click **Analysis** and select **Analyze**.


Project Manager
GETSTART*
EMDesign1*
Circuit Elements
Boundaries
Excitations
Analysis
Design V
Results
Definitions
Properties
Paste Ctrl+V
Add HFSS Solution Setup ...
Add HFSS Custom Setup ...
Add Planar EM Solution Setup ...
Add Slwave Solution Setup ...
Add Slwave DCIR Solution Setup ...
Add EMI Scanner Setup ...
Add Nexxim Solution Setup...
Add Nexxim Solution Options ...
Analyze F10
Generate Mesh
Invalidate Solutions
Browse Solutions...
Clean Up Solutions...
Import Solution...
Update Menu

Note:

Alternatively, run a single sweep by expanding the **Project Tree** > **[*active design folder*]** > **Analysis** folder in the **Project Manager** window. Then right-click the chosen sweep and select **Analyze**.

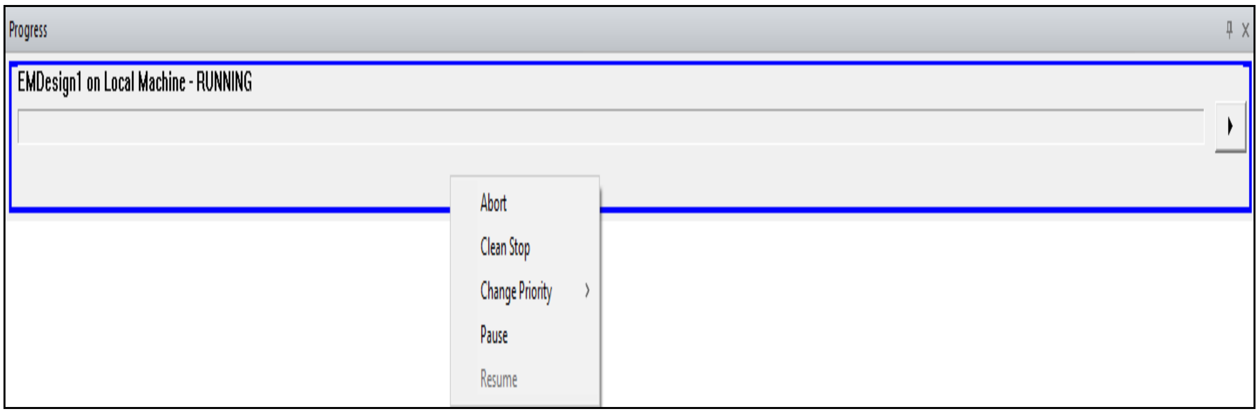


View Progress

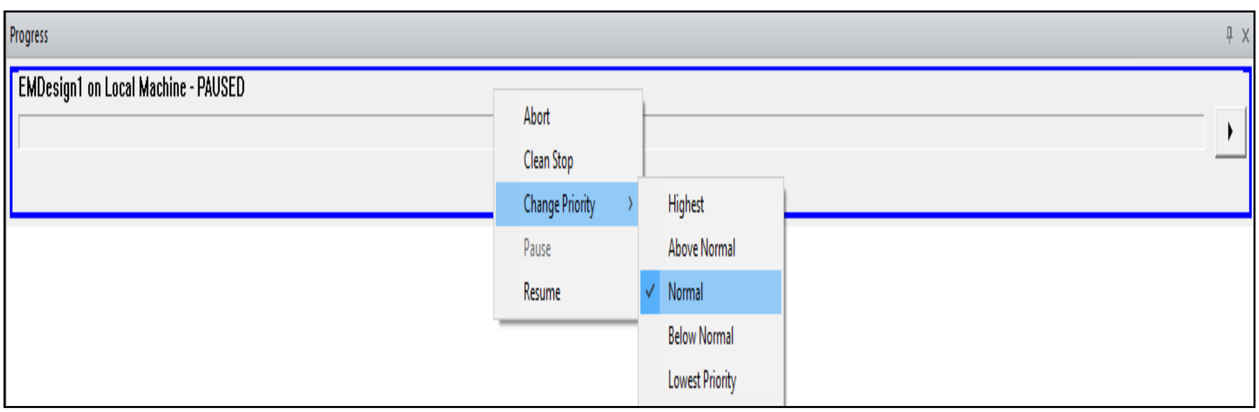
From the **Electronics Desktop** status bar, select **Show Progress** to expand the **Progress** window.

During analysis, perform the following actions, as appropriate:

- Right-click within the **Progress** window and select **Abort**, **Clean Stop**, or **Pause**. **Clean Stop** completes the computation of the current frequency point, then analysis ends.



- Right-click within the **Progress** window and 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.



Animated Demonstration

Continue to View S Matrix Data.

3 - Evaluate the Results (Post-processing)

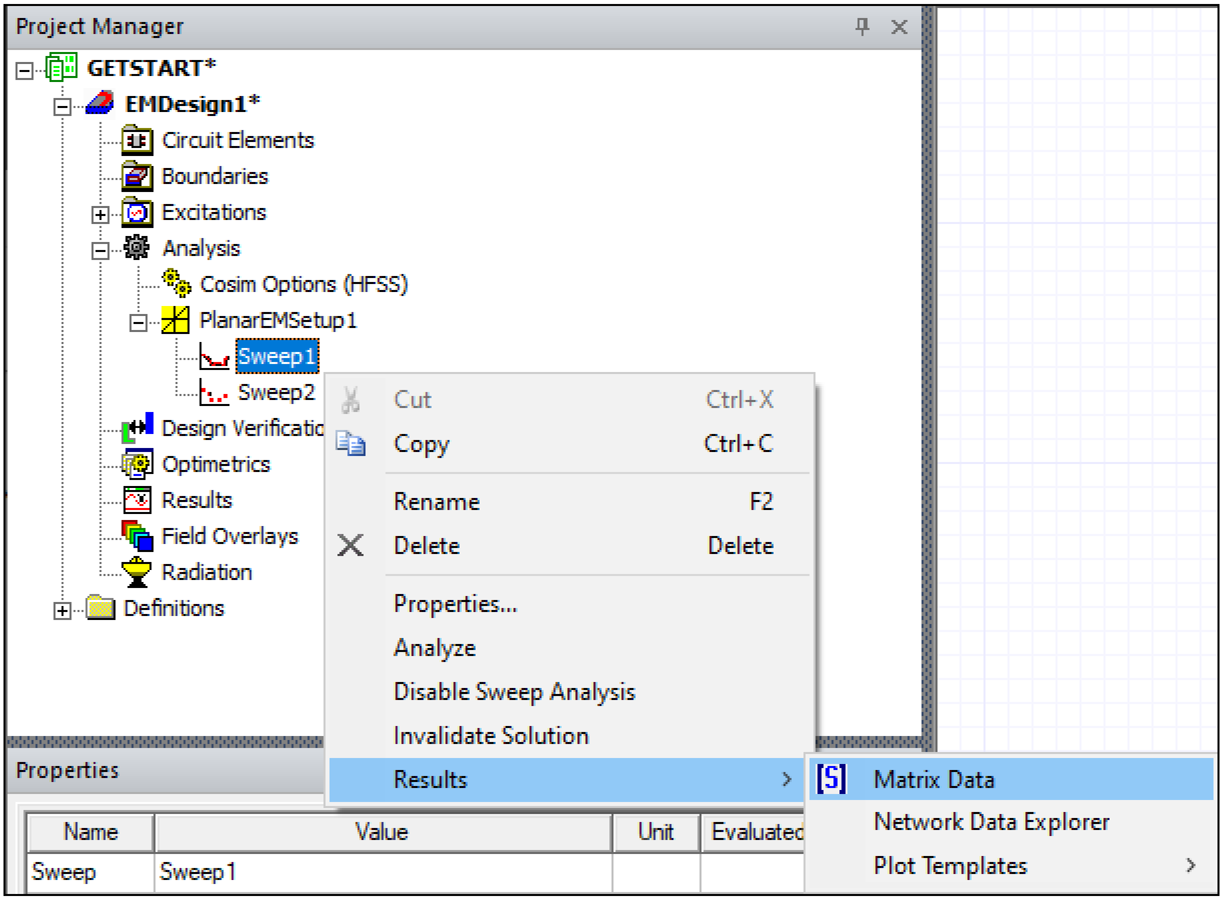
Use the post-processing capabilities of Ansys Electronics Desktop to display the results of a simulation. Also use the export features to save the analysis data (and an equivalent circuit) in various industry-standard file formats.

The sections in this topic include:

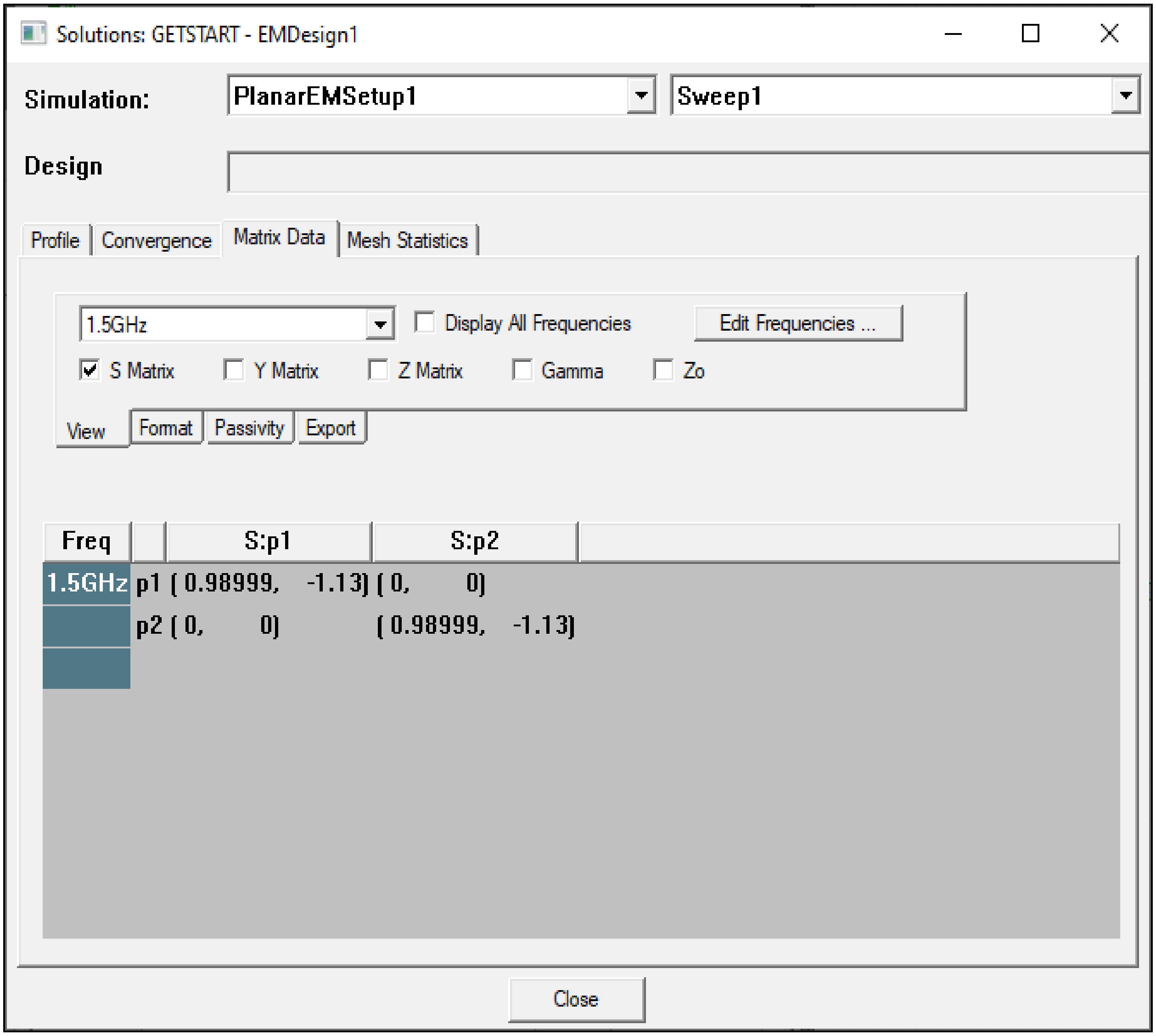
View *S* Matrix Data

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.

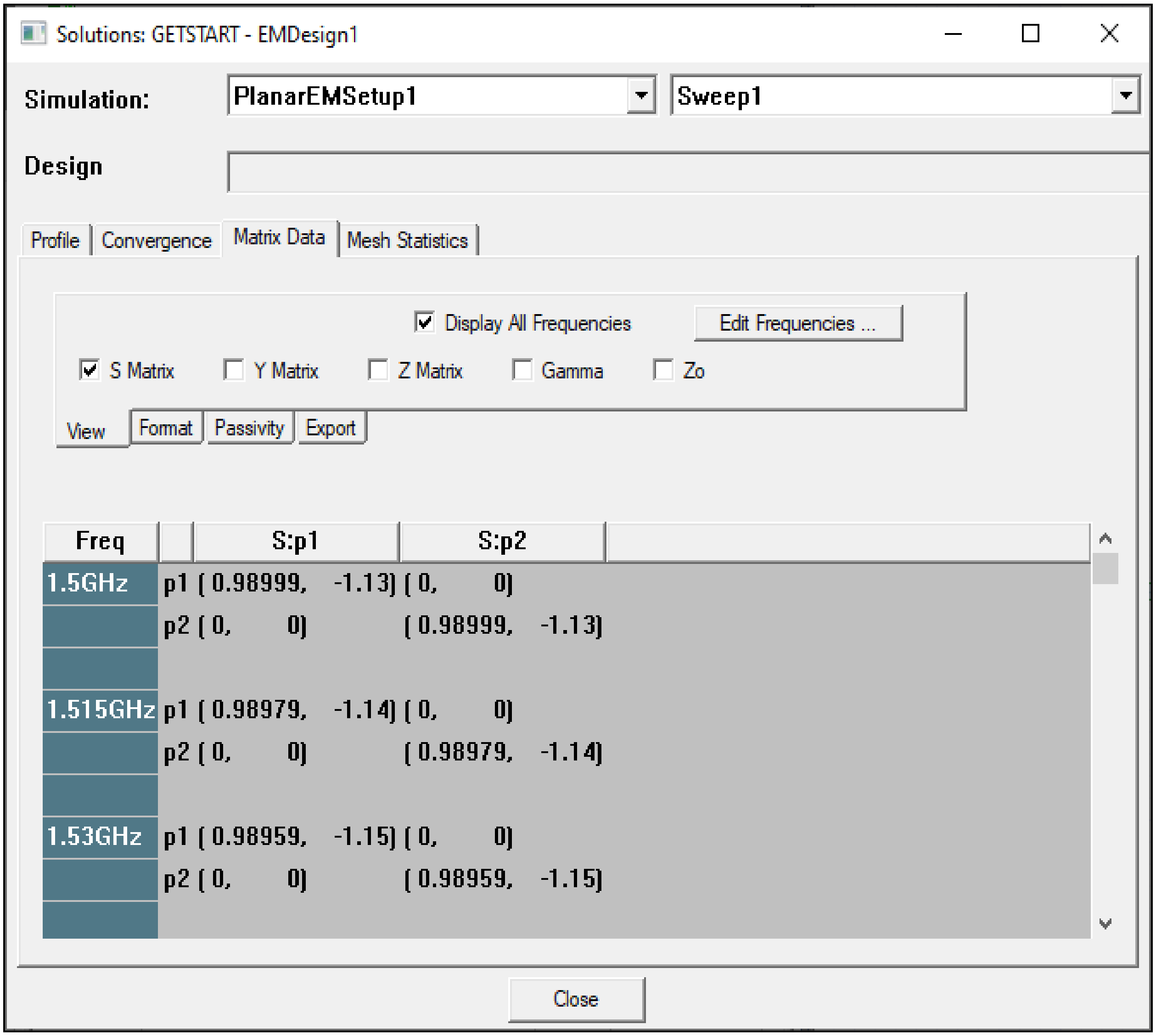
1. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** folder > setup (e.g., **PlanarEMSetup1**). Then right-click the chosen sweep (e.g., **Sweep1**), and select **Results** > **Matrix Data** to open the **Solutions** window.



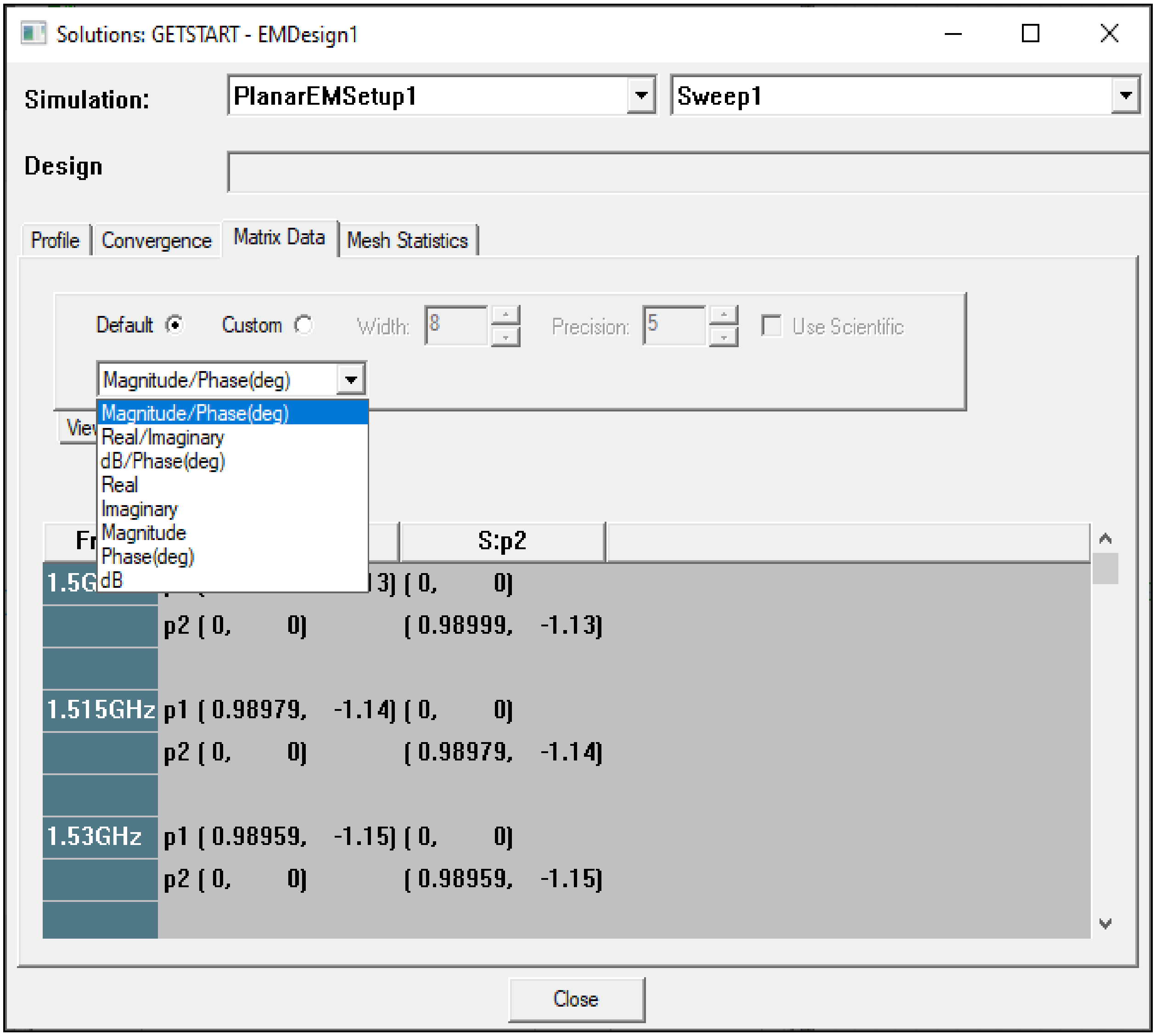
2. From the **Matrix Data** tab, ensure the **S Matrix** box is checked.



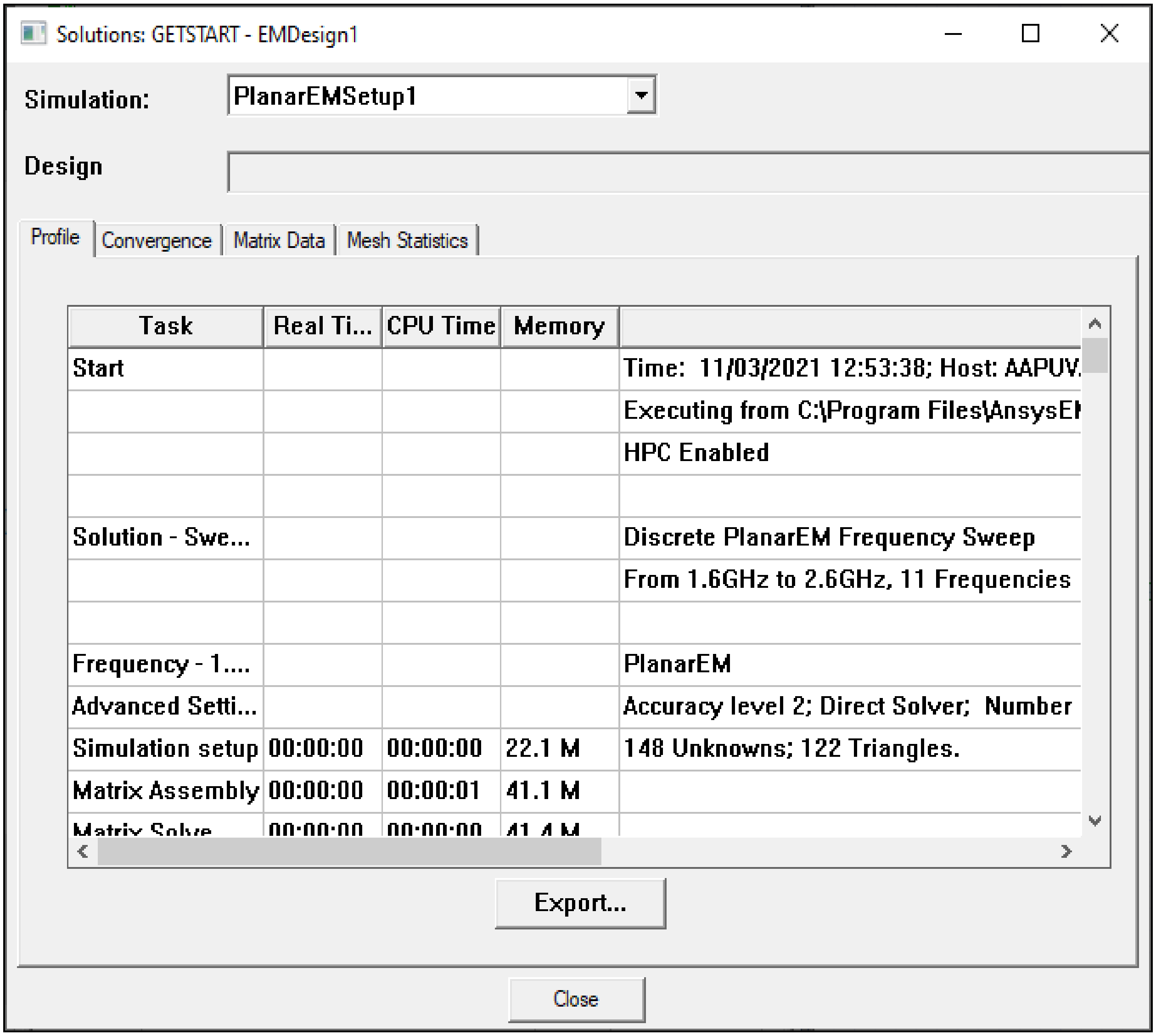
3. Check the **Display All Frequencies** box.



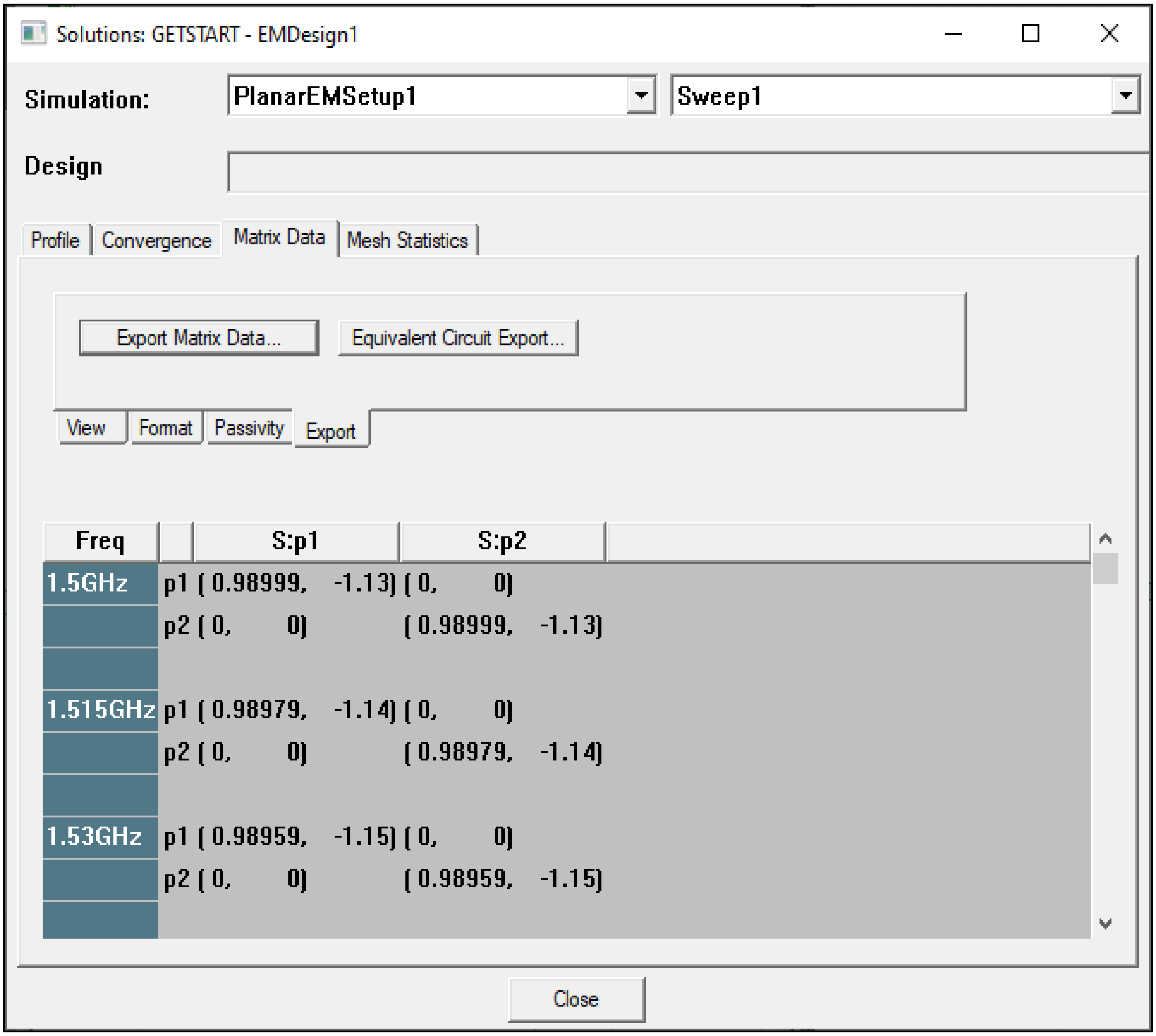
4. Navigate to the *Format* sub tab. Then choose the chosen data format from the drop-down menu (i.e., **Magnitude/Phase(deg)**, **dB/Phase(deg)**, **Real**, **Imaginary**, **Magnitude**, **Phase(deg)**, or **dB**).

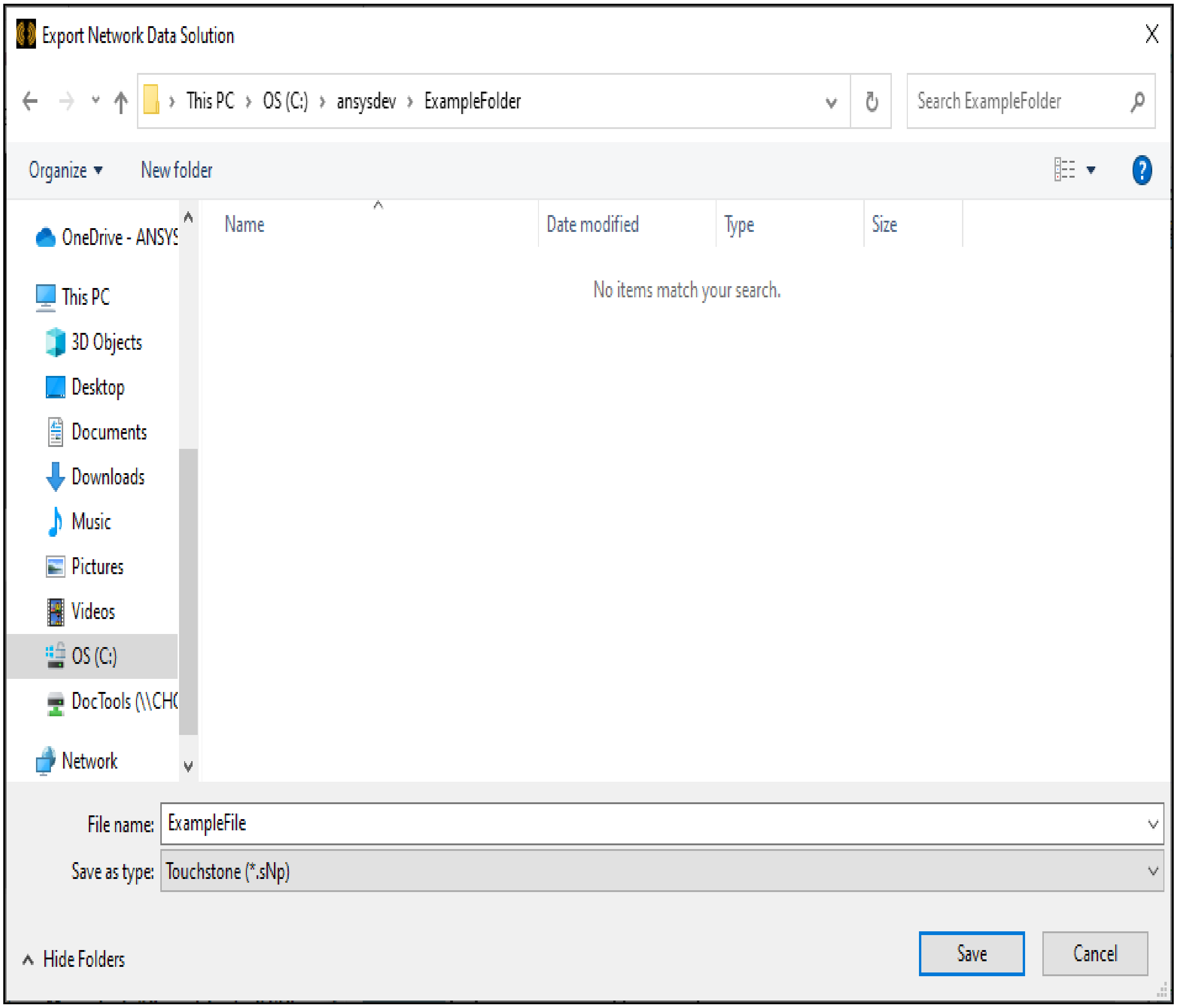


5. To see runtime data pertaining to the analysis, navigate to the **Profile** tab.



6. To export data, do the following:
   a. Navigate to the *Export* sub tab and click **Export Matrix Data** to open an explorer window.


Solutions: GETSTART - EMDesign1
Simulation:
PlanarEMSetup1
Sweep1
Design
Profile
Convergence
Matrix Data
Mesh Statistics
Export Matrix Data...
Equivalent Circuit Export...
View
Format
Passivity
Export
Freq
S:p1
S:p2
1.5GHz
p1 (0.98999, -1.13) (0, 0)
p2 (0, 0) (0.98999, -1.13)
1.515GHz
p1 (0.98979, -1.14) (0, 0)
p2 (0, 0) (0.98979, -1.14)
1.53GHz
p1 (0.98959, -1.15) (0, 0)
p2 (0, 0) (0.98959, -1.15)
Close



b. From the explorer window, do the following:

i. Type a **File name**, or use the default (e.g., <**ProjectName_DesignName**>).

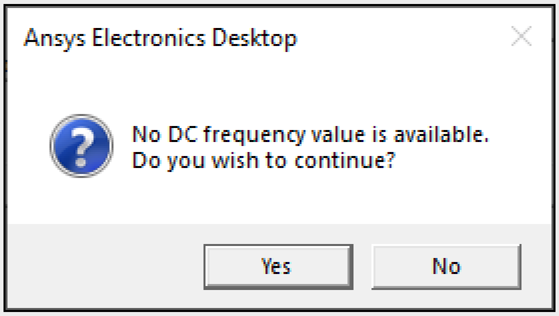
ii. Choose the chosen file format from the **Save as type** drop-down menu (e.g., **Touchstone (*.*sNp**), **Data Table (spreadsheet) (*.*tab**), **Neutral Model Format (*.*nmf**), **MATLAB (*.*m**), or **Citifile (*.*cit**)).

iii. Navigate to the chosen save location. (Default save location is the folder where the model is saved).

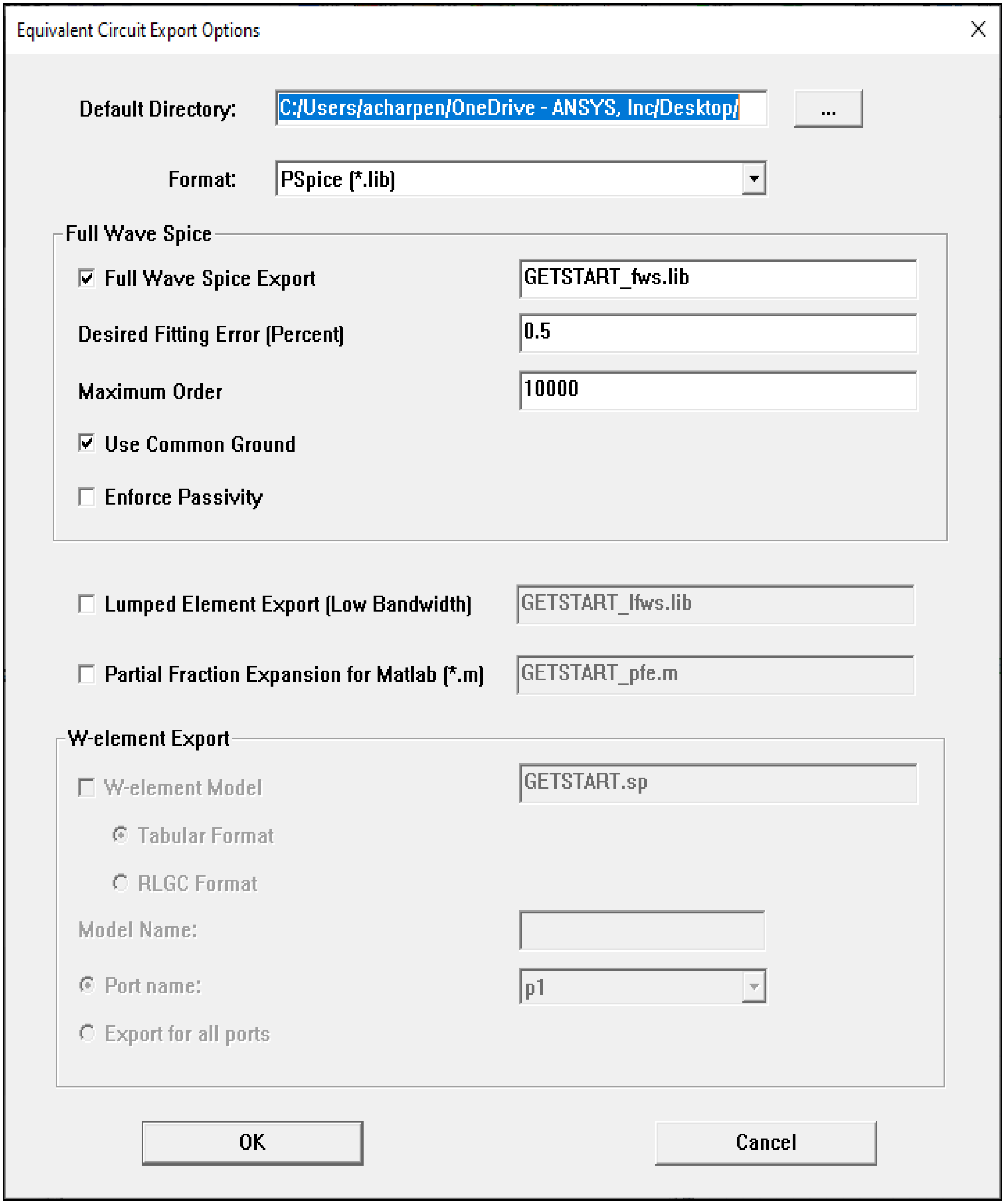
iv. Click **Save** to save changes, close the explorer window, and return to the **Solution** window, or click **Cancel** to close the explorer window without exporting anything.

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

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



   b. Click **Yes** to close the dialog box and open the **Equivalent Circuit Export Options** window.



c. Select the chosen 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.

8. **Close** the **Solutions** window.

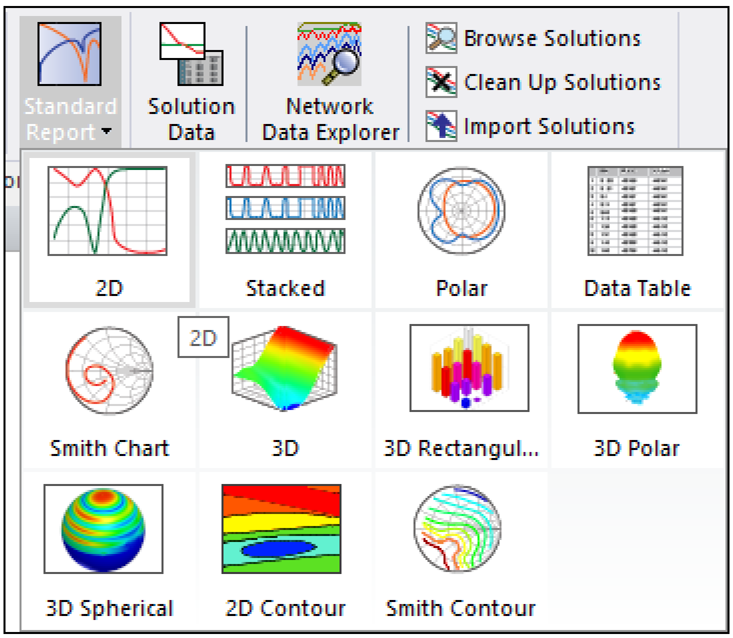
Continue to Plot Return Loss.

Plot Return Loss (a Standard Report)

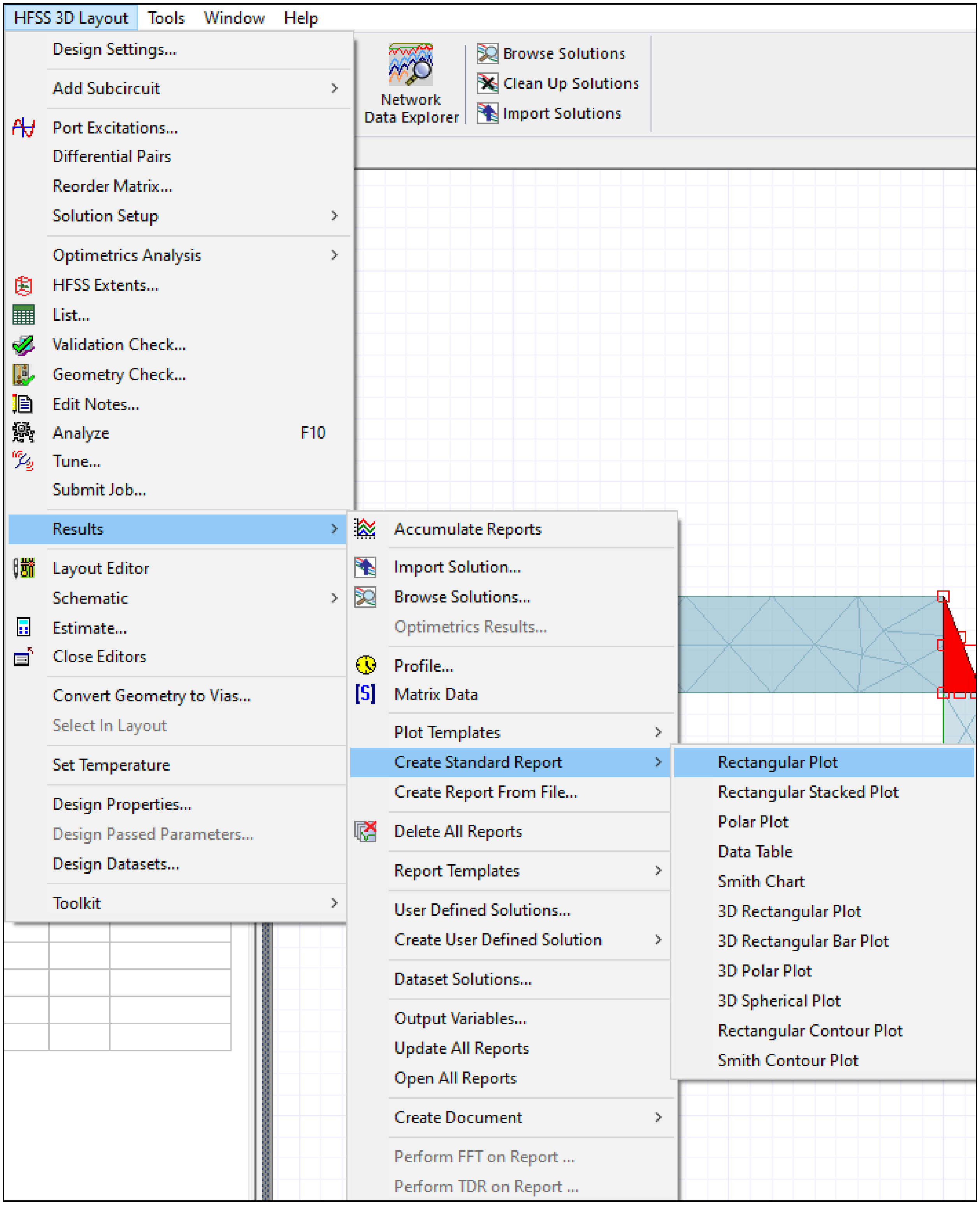
Complete these steps to create a report and plot return loss.

1. Open the **Report** window by doing one of the following:

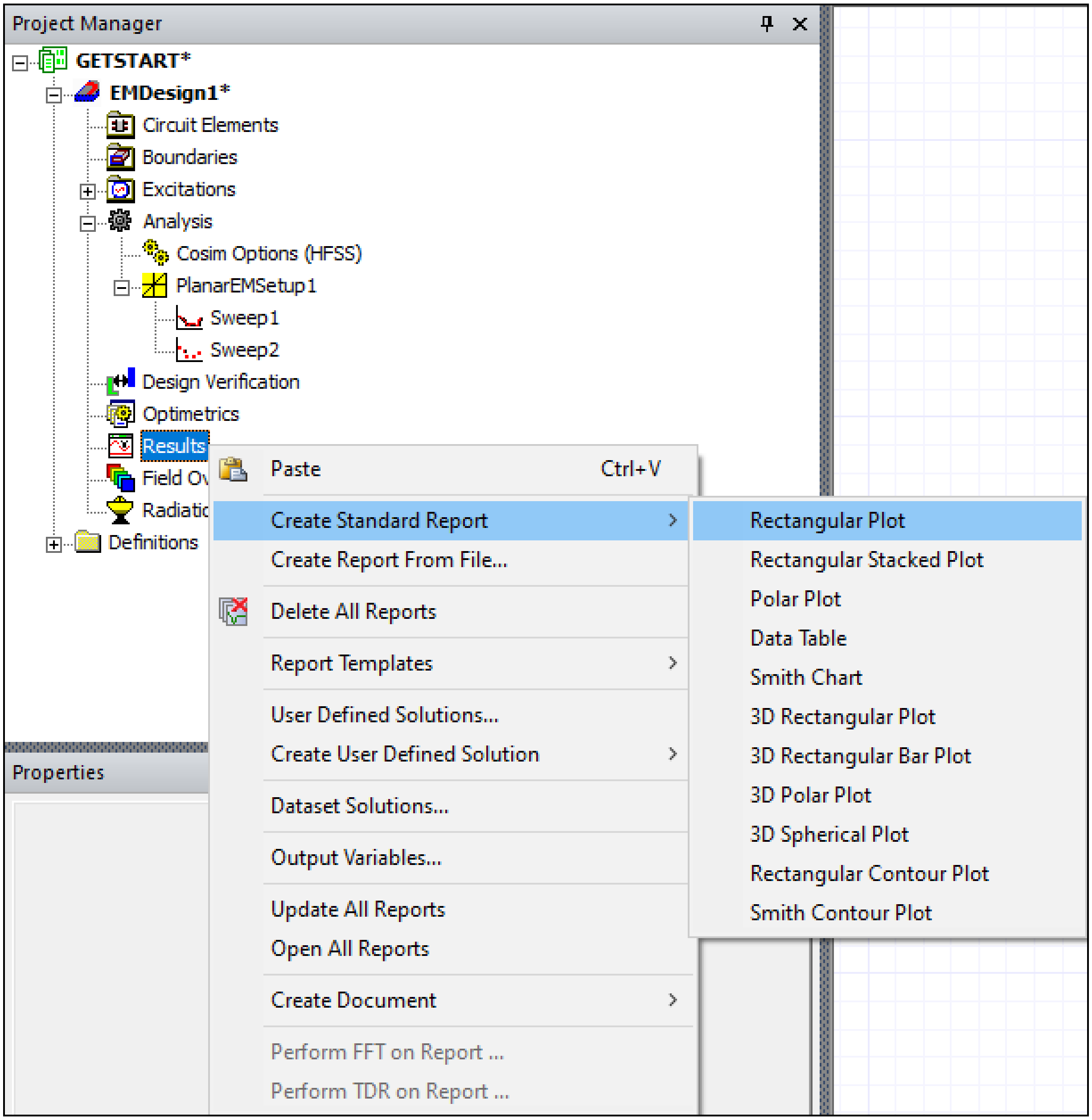
- From the **Results** tab, select **Standard Report** > **2D**.



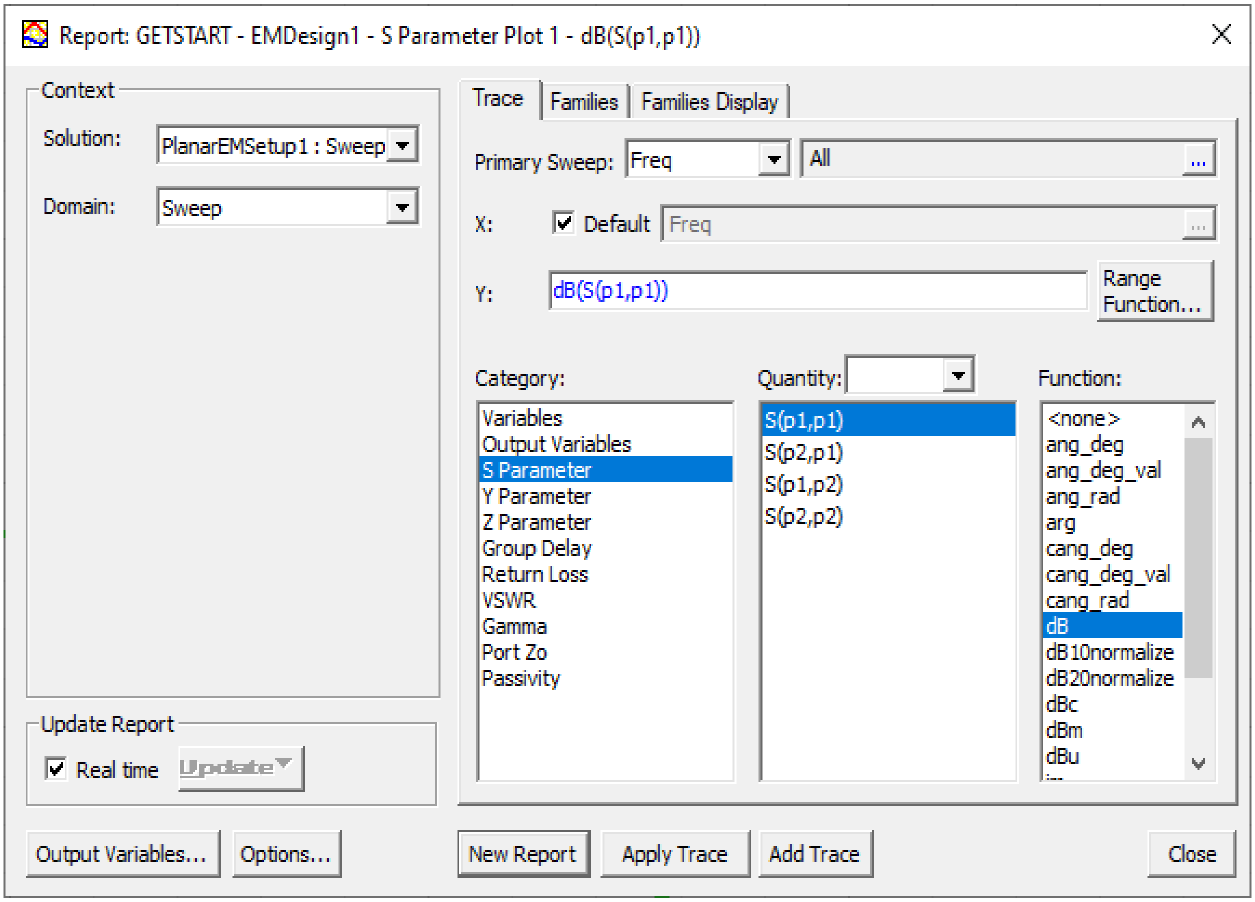
- From **HFSS 3D Layout**, select **Results** > **Create Standard Report** > **Rectangular Plot**.



- From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Results** and select **Create Standard Report** > **Rectangular Plot**.



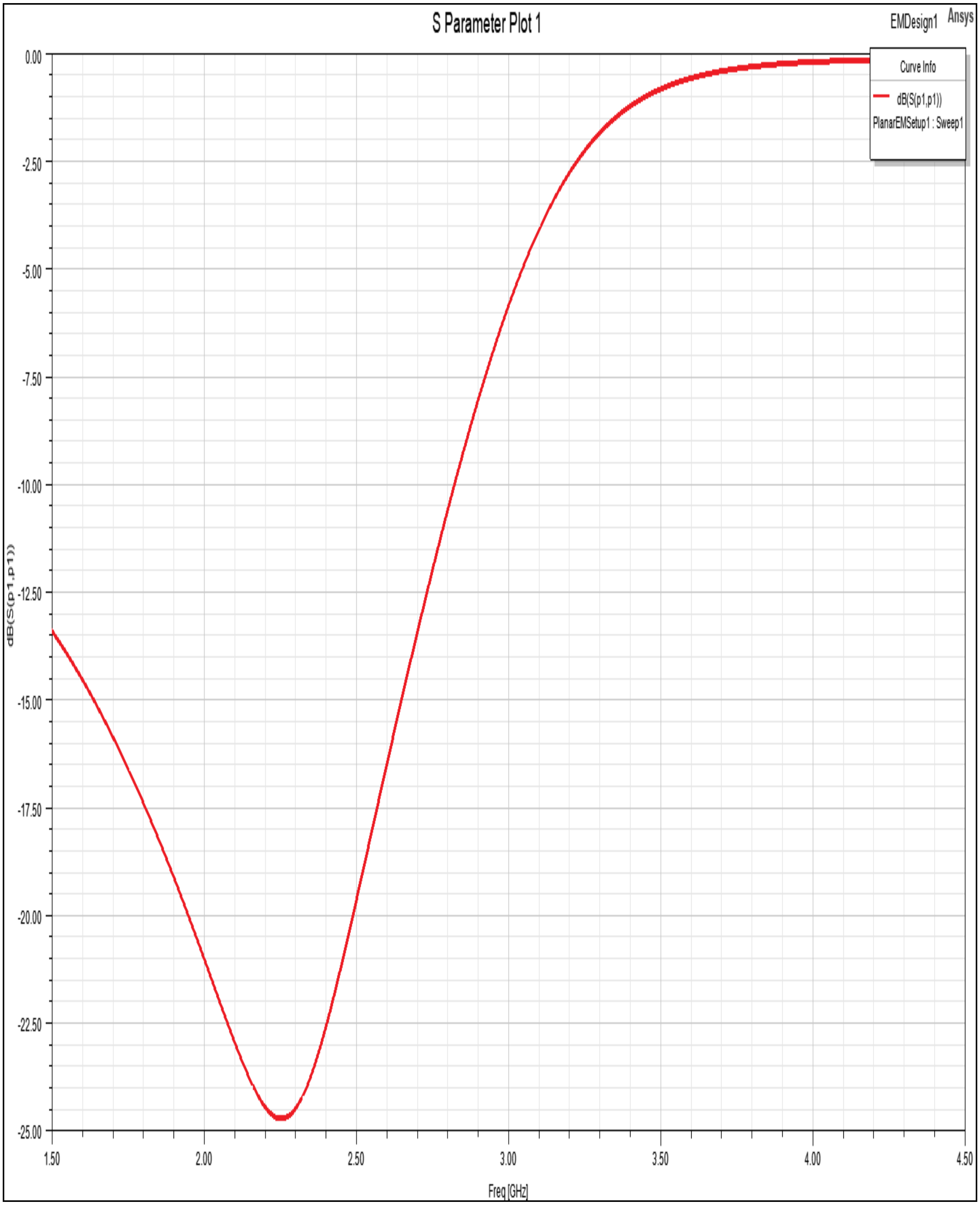
2. The **Report** window opens from the **Trace** tab.



Note:

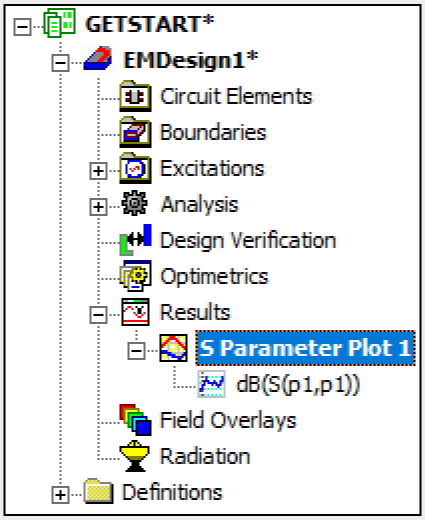
The settings described in Step 3 are the default settings in the **Report** window and it should not be appropriate to change anything.

3. From the **Trace** tab, select the following:
   a. Choose a sweep from the **Solution** drop-down menu (e.g., **PlanarEMSetup1 : Sweep1**).
   b. From the **Category** list, select **S Parameter**.
   c. From the **Quantity** list, select **S(p1,p1)**.
   d. From the **Function** list, select **dB**.
4. Click **New Report** and the return loss plot opens under the **Report** window.
5. **Close** the **Report** window to view the plot.


S Parameter Plot 1
EMDesign1
Ansys
Curve Info
dB(S(p1,p1))
PlanarEMSetup1 : Sweep1
0.00
-2.50
-5.00
-7.50
-10.00
-12.50
-15.00
-17.50
-20.00
-22.50
-25.00
dB(S(p1,p1))
1.50
2.00
2.50
3.00
3.50
4.00
4.50
Freq [GHz]

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 chosen plot).



Note:

For additional details about formatting the plot, refer to **Post Processing and Generating Reports**.

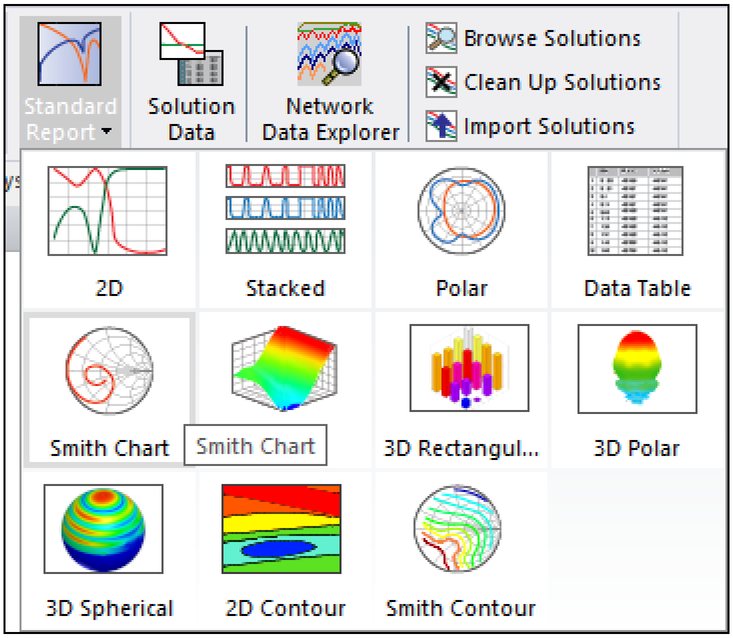
Continue to Plot a Smith Chart.

Plot a Smith Chart

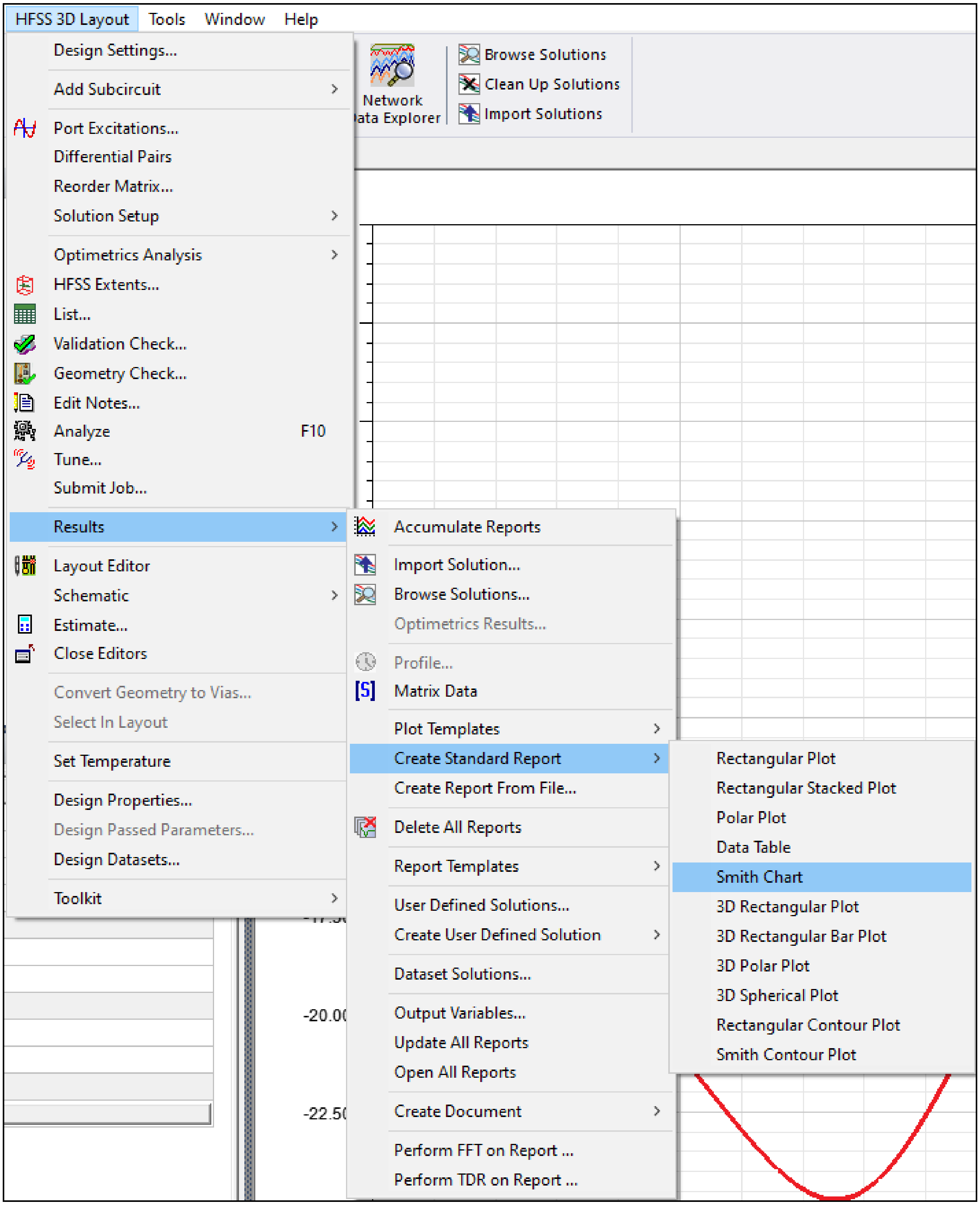
A Smith chart is a convenient means of simultaneously displaying multiple parameters (e.g., impedance, reflection coefficients, scattering parameters, constant gain contours, et cetera). It is a useful graphical aid for electronics engineers working in the radio frequency disciplines. Complete these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

1. Open the **Report** window by doing one of the following:

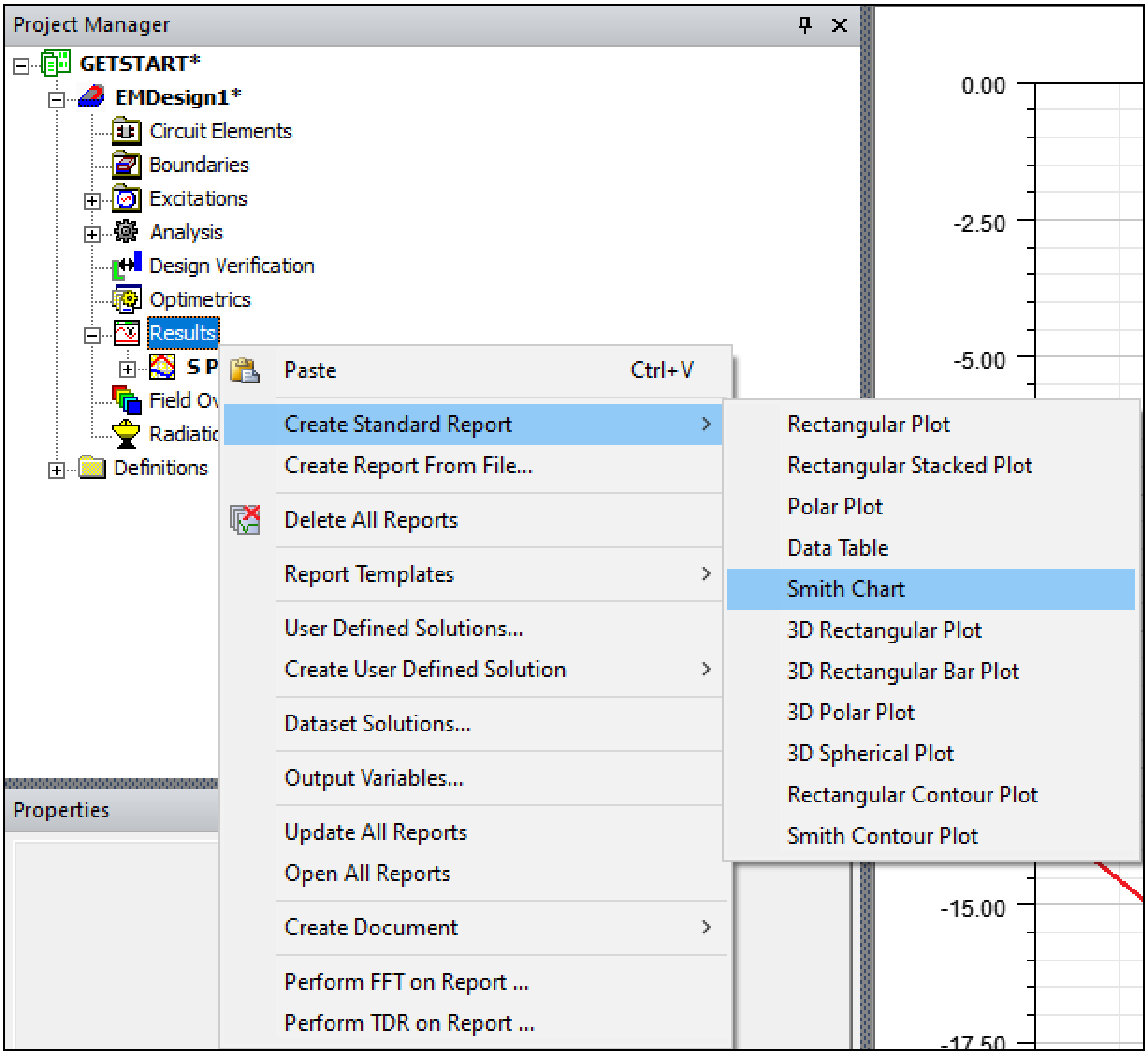
- From the **Results** tab, select **Standard Report** > **Smith Chart**.



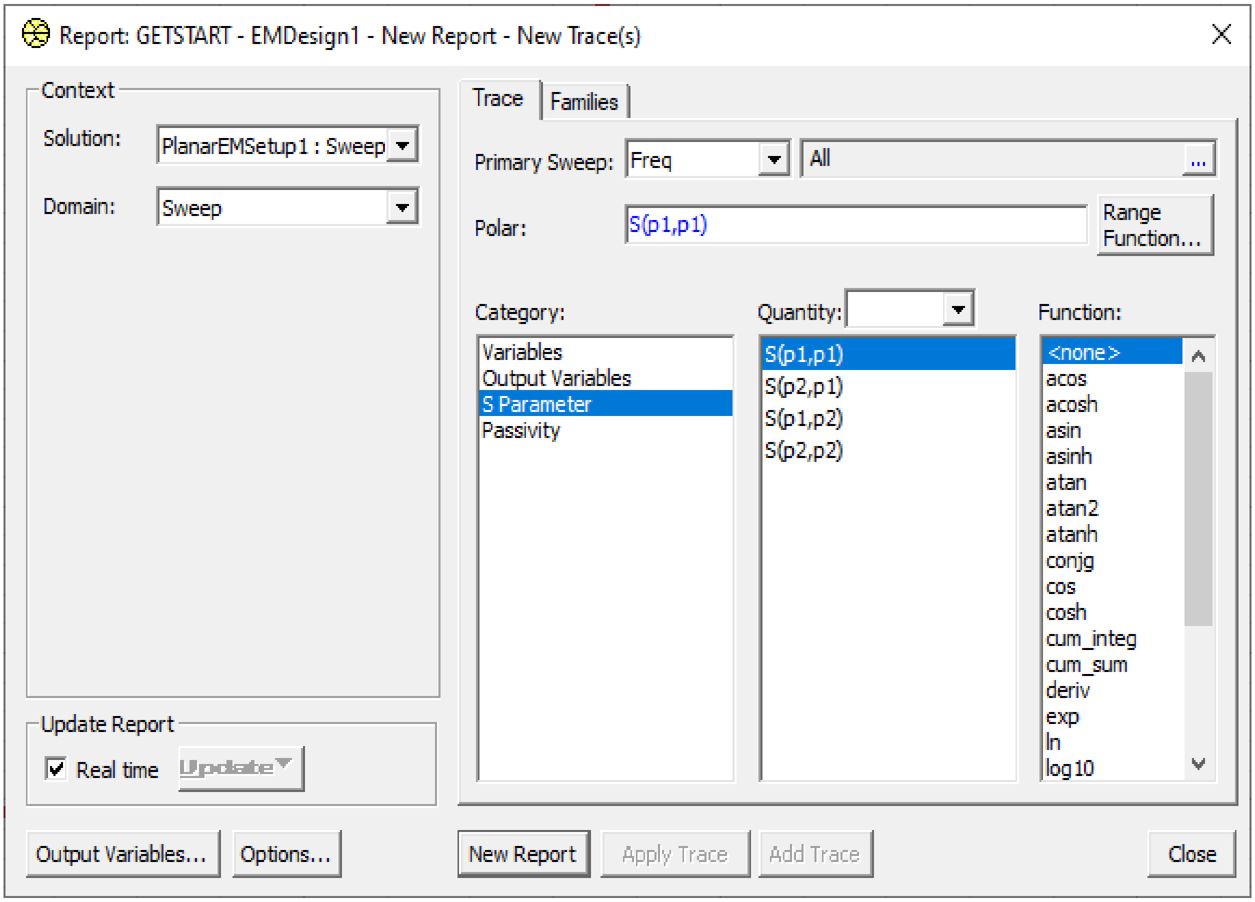
- From **HFSS 3D Layout**, select **Results** > **Create Standard Report** > **Smith Chart**.



- From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Results** and select **Create Standard Report** > **Smith Chart**.

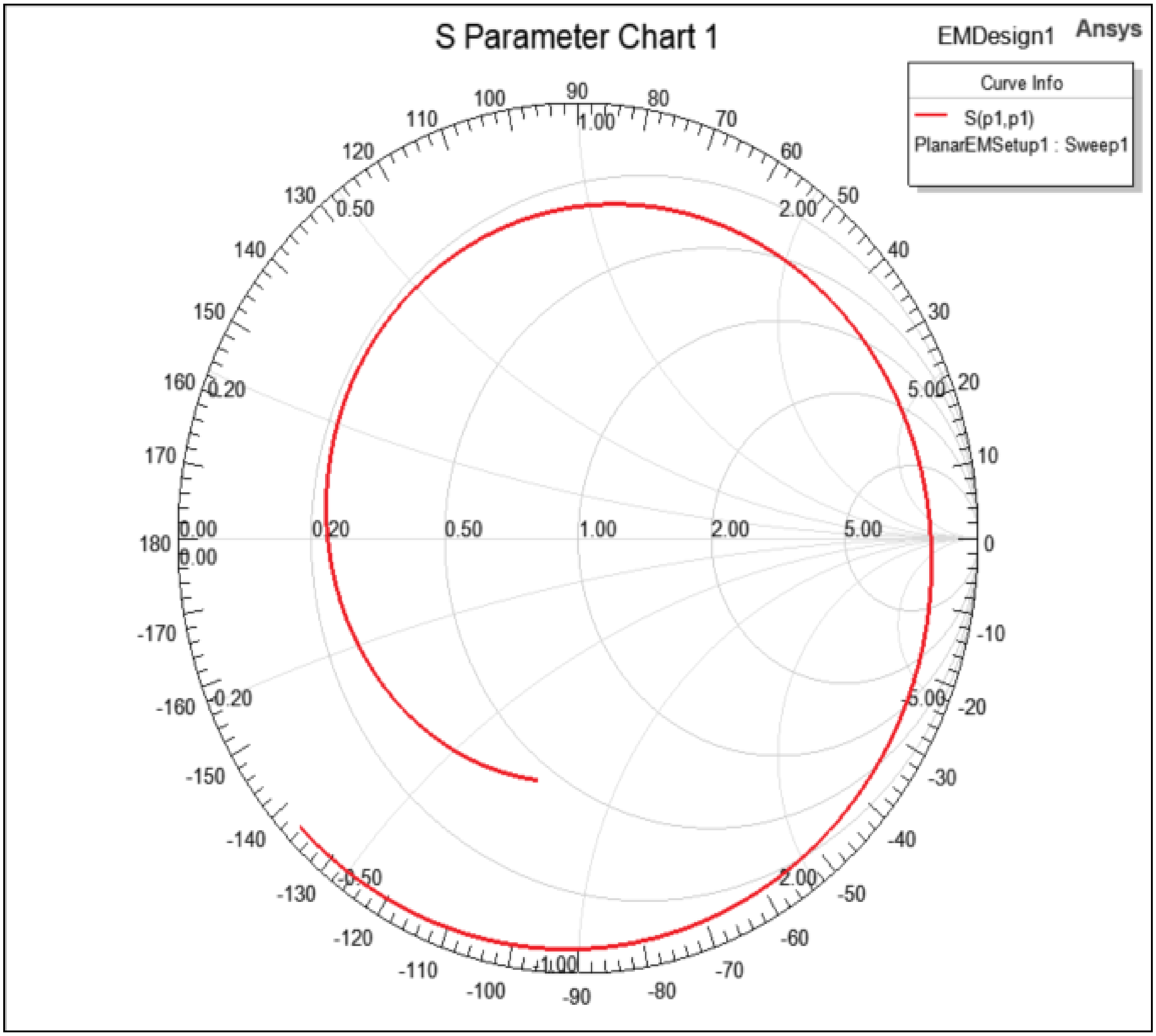


2. The **Report** window opens from the **Trace** tab.



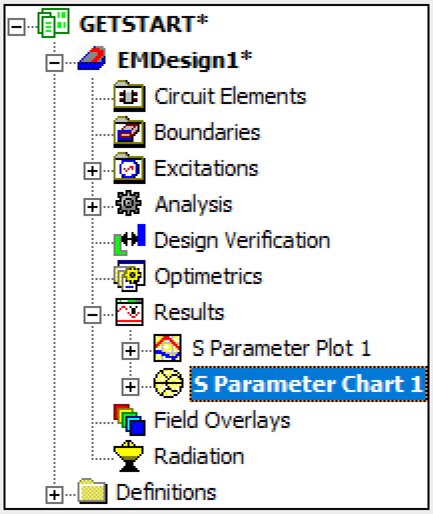
> **Note:**
>
> The settings described in Step 3 are the default settings in the **Report** window and it should not be appropriate to change anything.

3. From the **Trace** tab, select the following:
   a. Choose a sweep from the **Solution** drop-down menu (e.g., **Planar EM Setup 1 : Sweep 1**).
   b. From the **Category** list, select **S Parameter**.
   c. From the **Quantity** list, select **S(p1,p1)**.
   d. From the **Function** list, select **<none>**.
4. Click **New Report** and the return loss plot opens under the **Report** window.
5. **Close** the **Report** window to view the plot.


S Parameter Chart 1
EMDesign1
Ansys
Curve Info
S(p1,p1)
PlanarEMSetup1 : Sweep1
90
80
70
60
50
40
30
20
10
0
-10
-20
-30
-40
-50
-60
-70
-80
-90
-100
-110
-120
-130
-140
-150
-160
-170
180
170
160
150
140
130
120
110
100
0.00
0.20
0.50
1.00
2.00
5.00
-0.20
-0.50
-1.00
-2.00
-5.00

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 chosen plot).



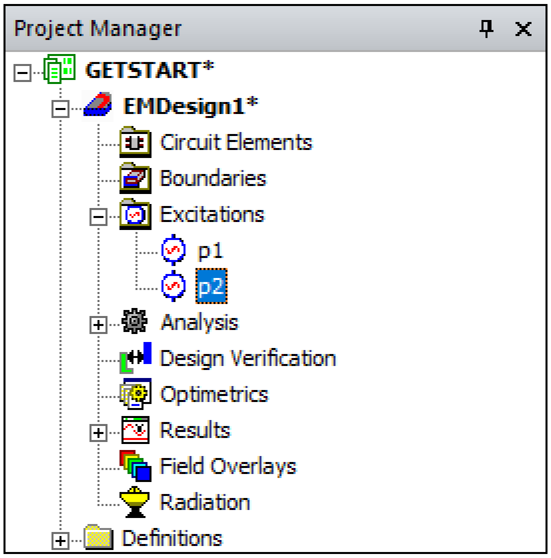
Continue to Revise An Excitation.

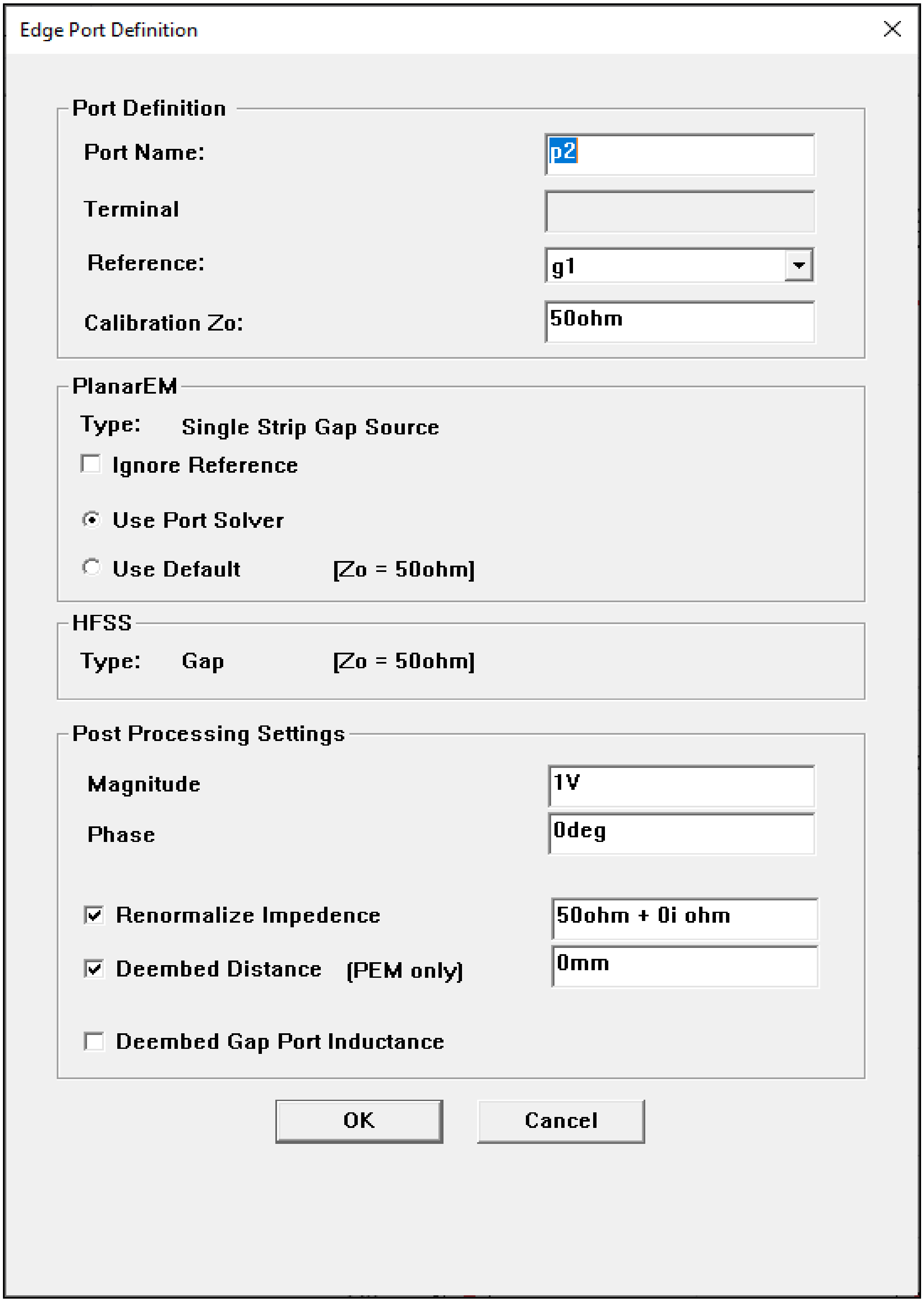
Revise An Excitation

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*). Now 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.

To revise the *p2* excitation:

1. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Excitations** folder. Then double-click **p2** to open the **Edge Port Definition** window.


Project Manager
GETSTART*
EMDesign1*
Circuit Elements
Boundaries
Excitations
p1
p2
Analysis
Design Verification
Optimetrics
Results
Field Overlays
Radiation
Definitions



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

Port Definition area:

- **Port Name** specify the port being defined
- **Terminal** the name of the terminal (automatic, unfillable field)
- **Reference** the **Port Solver** (**Reference**) used by the port
- **Calibration Zo** the impedance (equivalent to the Full Port Impedance of the port (in Ohms))

PlanarEM area

- **Type** of port (automatic, unfillable field)
- **Ignore Reference** check to ignore the **Port Solver** (**Reference**)
- **Use Port Solver** select to calculate the characteristic impedance and propagation constant of the port. The gap source is automatically calibrated for greater accuracy
- **Use Default** select if the default setting (expressed in 0hms) is used, instead of the **Port Solver** (**Reference**)

HFSS area

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

Post Processing Settings area

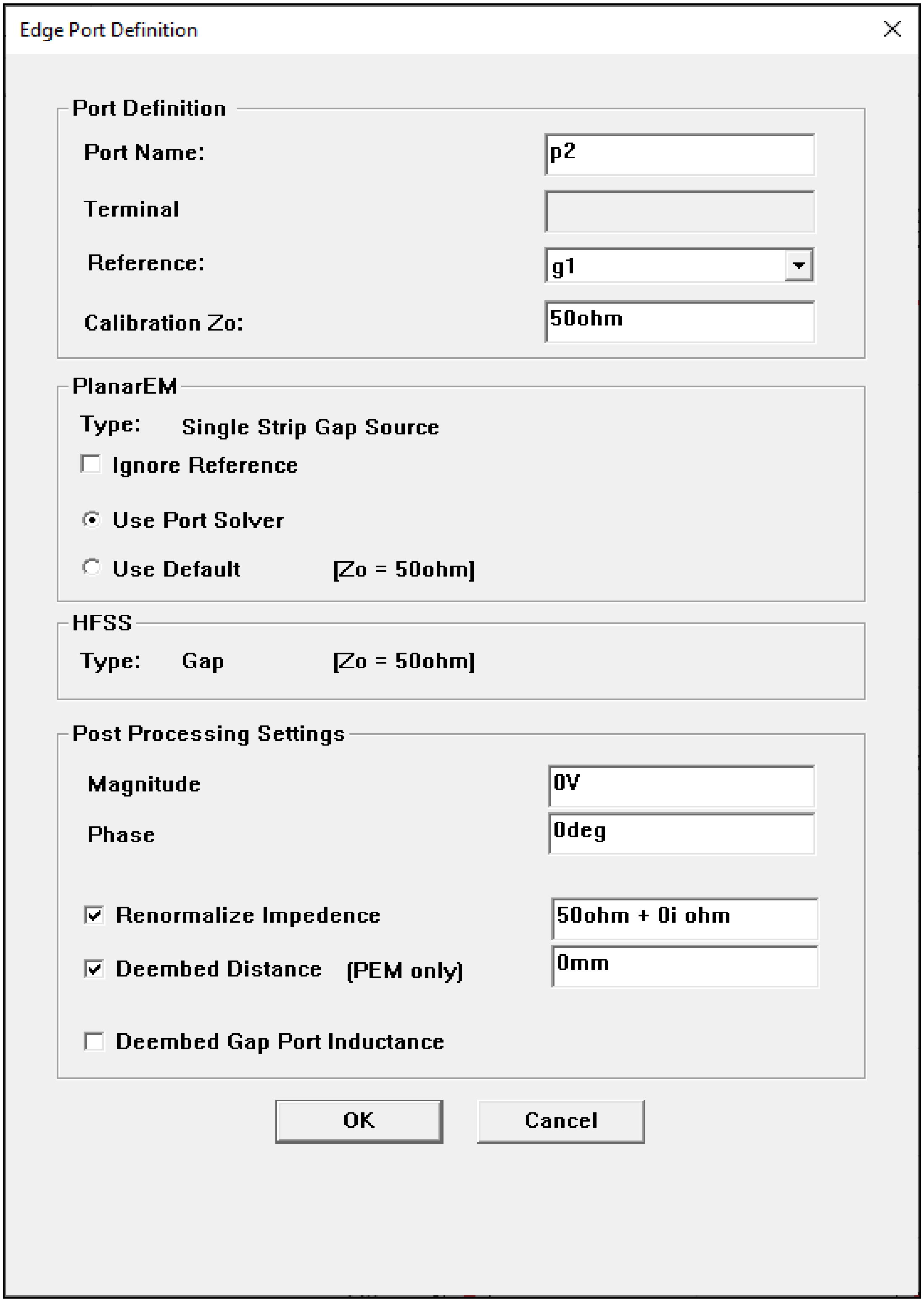
- **Magnitude** expressed in Volts (V)
- **Phase** expressed in degrees (deg)
- **Renormalize Impedance** check to use, expressed in ohms

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

- **Deembed Distance (PEM Only)** check to use, expressed in millimeters (mm)
- **Deembed Gap Port Inductance** check to specify 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.

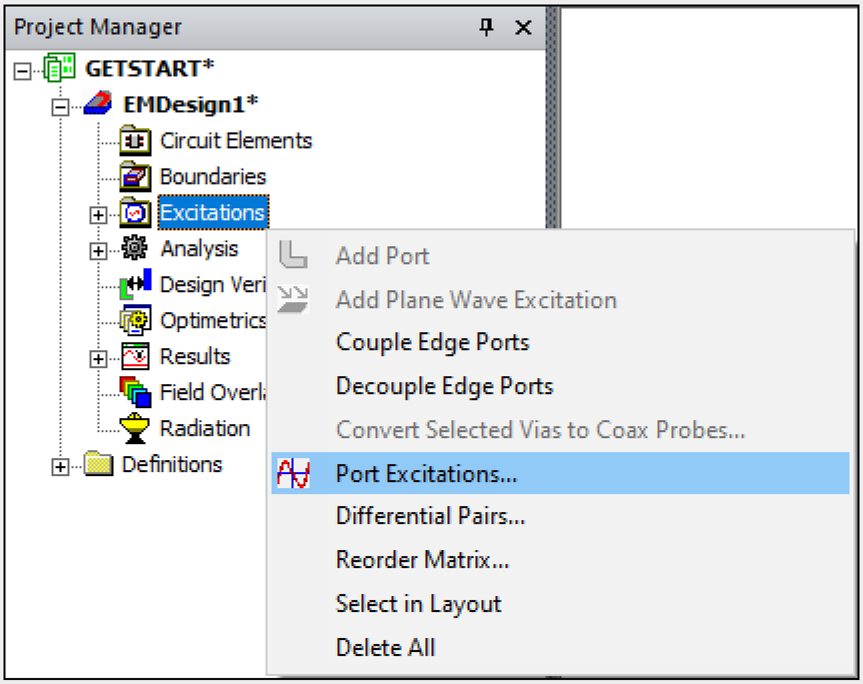
> **Note:**
>
> For more information, refer to **Deembedding**.

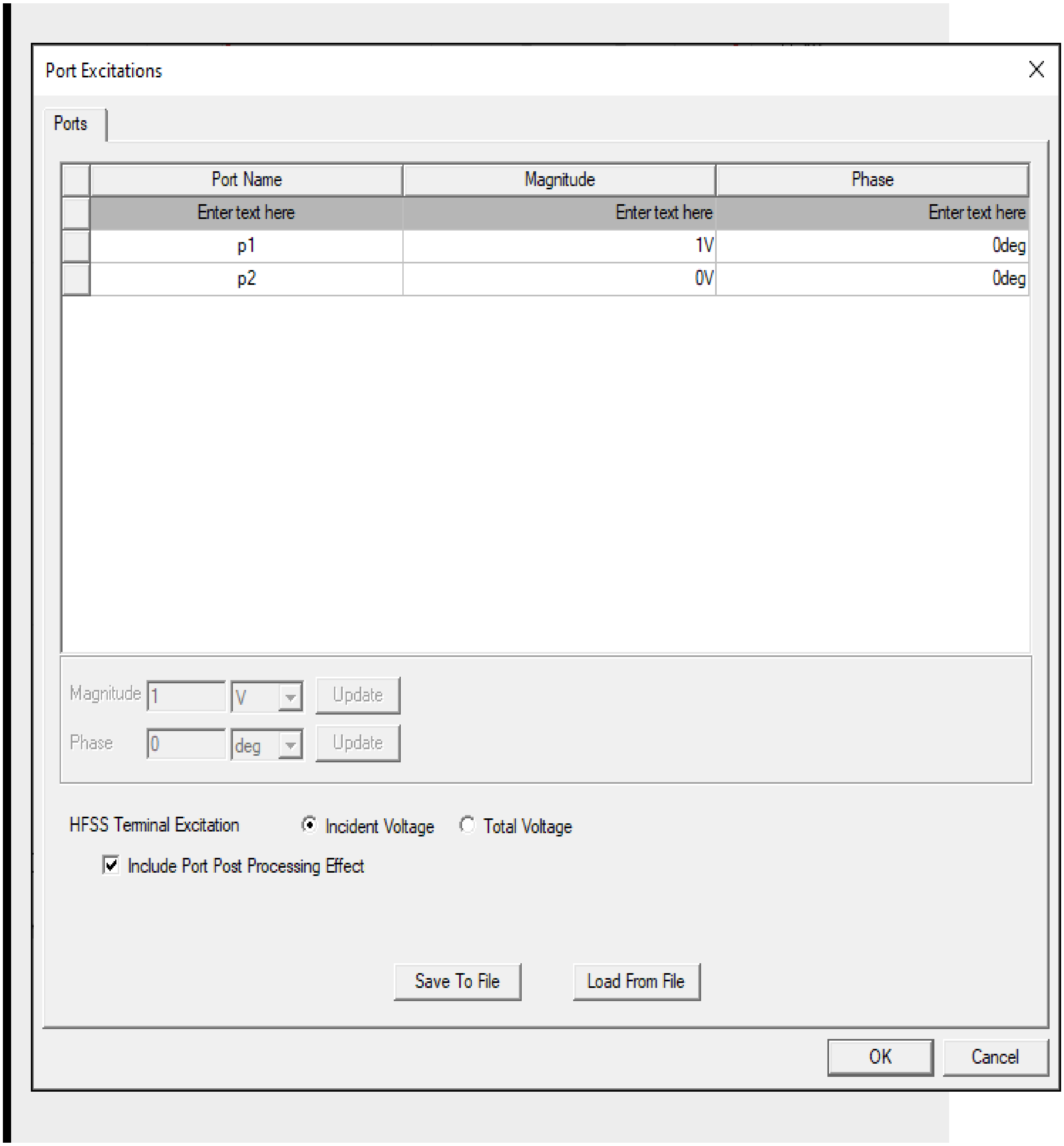
2. From the **Post Processing Settings** area, enter the following:
   a. In the **Magnitude** field, enter **0V**.
   b. In the **Phase** field, enter **0deg**.


Edge Port Definition
Port Definition
Port Name:
p2
Terminal
Reference:
g1
Calibration Zo:
50ohm
PlanarEM
Type: Single Strip Gap Source
Ignore Reference
Use Port Solver
Use Default [Zo = 50ohm]
HFSS
Type: Gap [Zo = 50ohm]
Post Processing Settings
Magnitude
0V
Phase
0deg
Renormalize Impedence
50ohm + 0i ohm
Deembed Distance (PEM only)
0mm
Deembed Gap Port Inductance
OK
Cancel

3. Click **OK** to save changes and close the **Edge Port Definition** window. Repeat for additional ports.

Note: Alternatively, use the **Port Excitations** window to make changes to multiple ports within the same window. To access the **Port Excitations** window, from the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Excitations** and select **Port Excitations** to open the **Port Excitations** window.



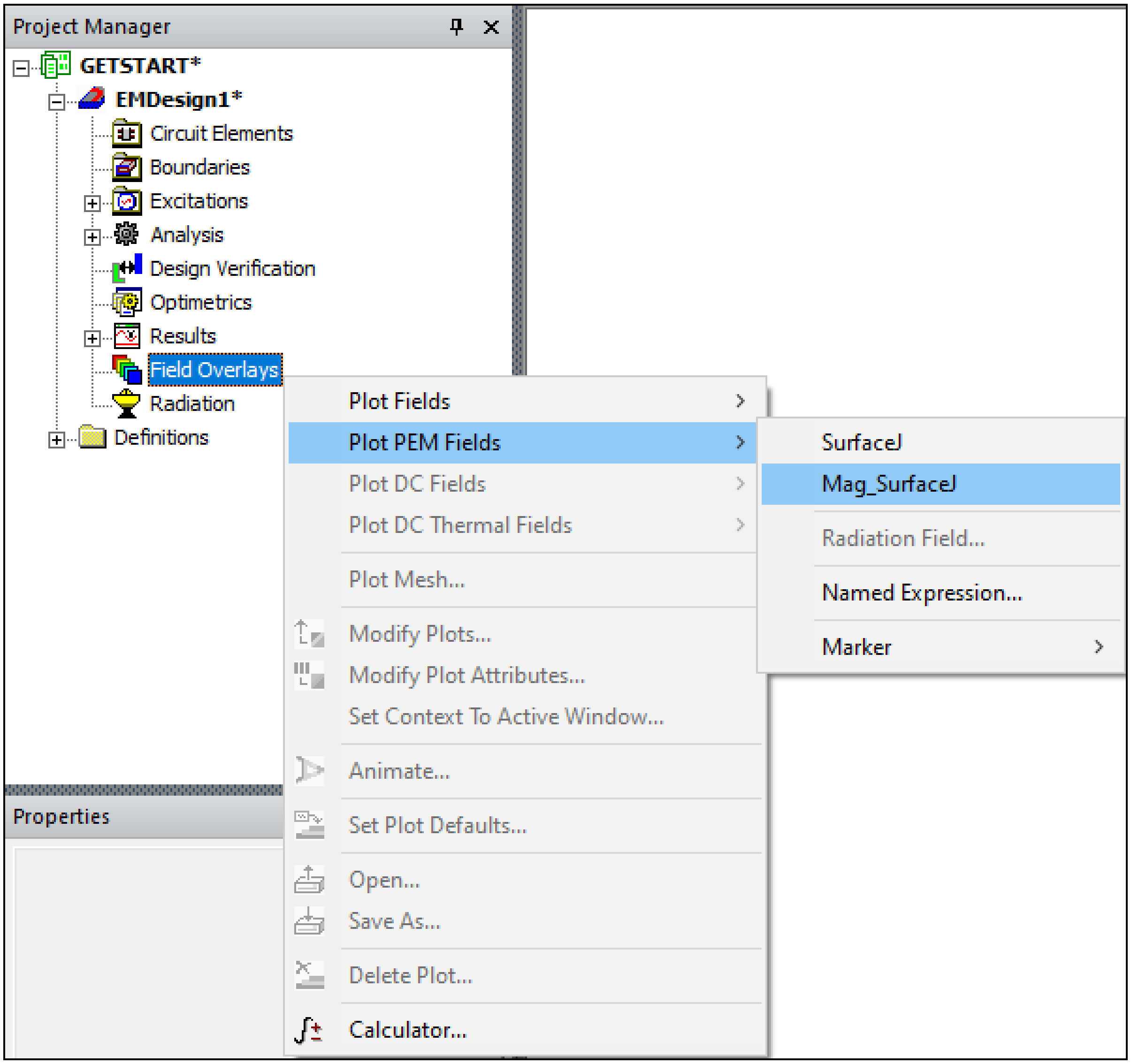


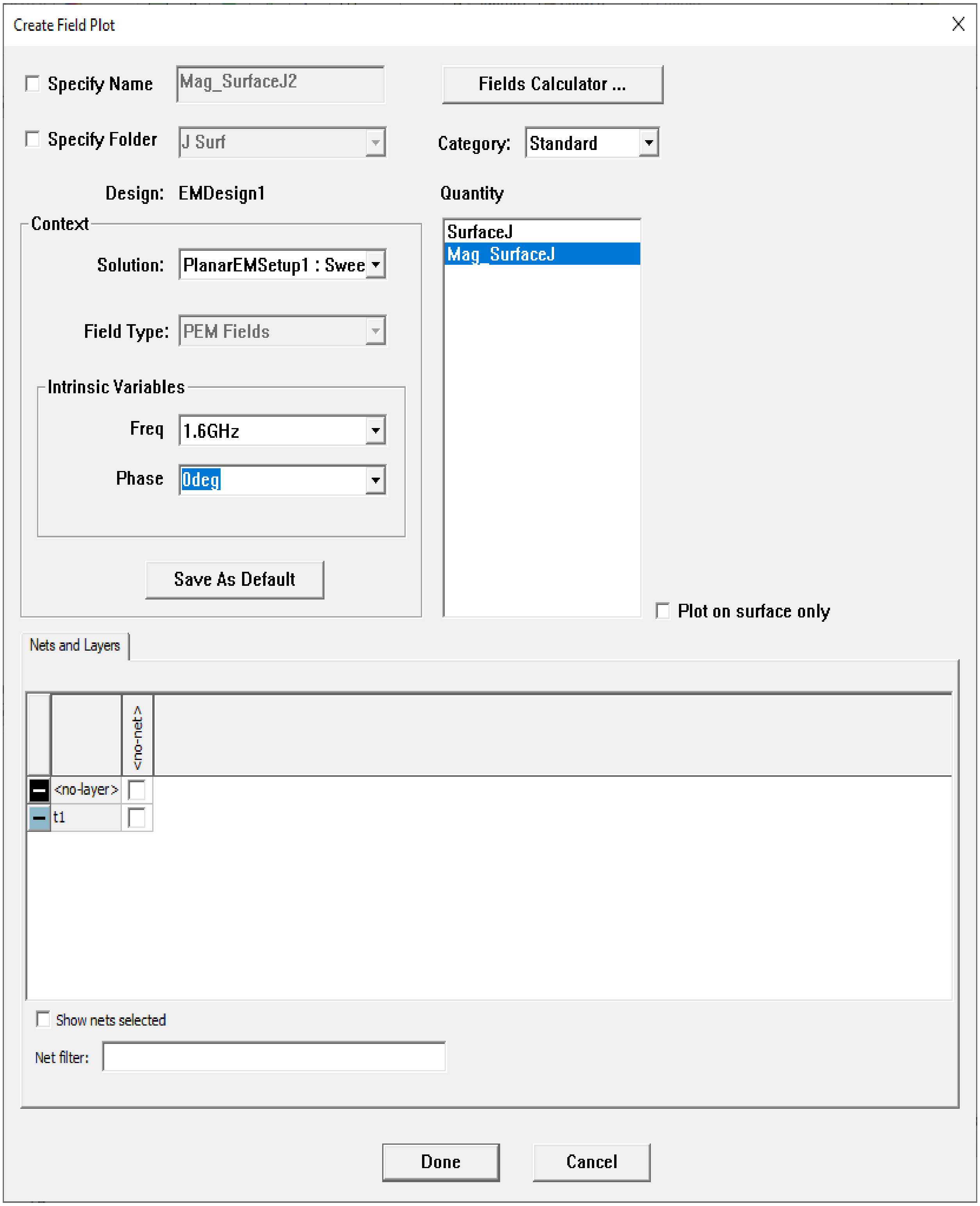
Continue to Overlay Current Results.

Use Field Overlays

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

1. From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Field Overlays** and select **Plot PEM Fields** > **Mag_SurfaceJ** to open the **Create Field Plot** window.

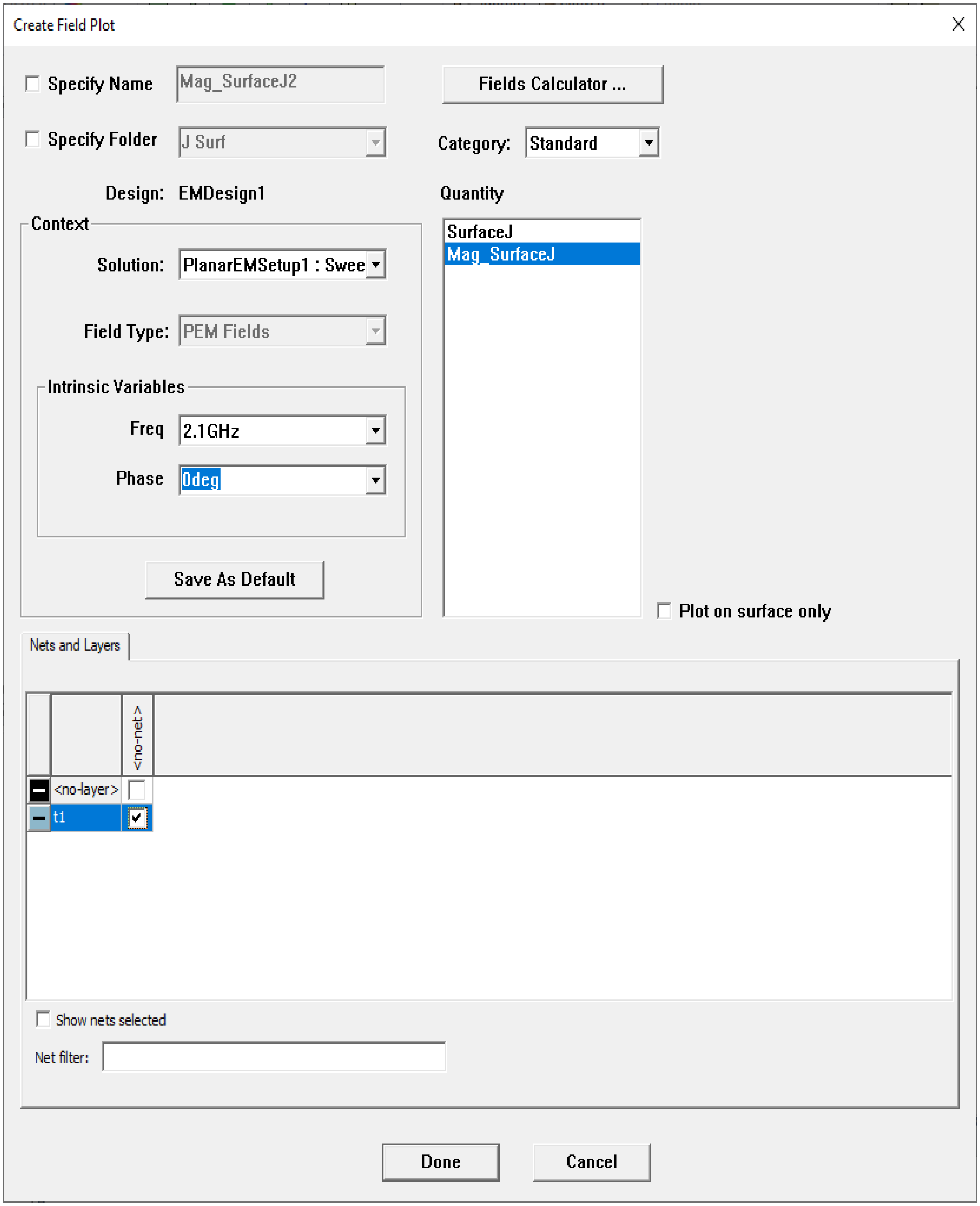



Create Field Plot
Specify Name
Mag_SurfaceJ2
Fields Calculator ...
Specify Folder
J Surf
Category:
Standard
Design:
EMDesign1
Quantity
Context
SurfaceJ
Mag_SurfaceJ
Solution:
PlanarEMSetup1 : Swee
Field Type:
PEM Fields
Intrinsic Variables
Freq
1.6GHz
Phase
0deg
Save As Default
Plot on surface only
Nets and Layers
<no-net>
<no-layer>
t1
Show nets selected
Net filter:
Done
Cancel

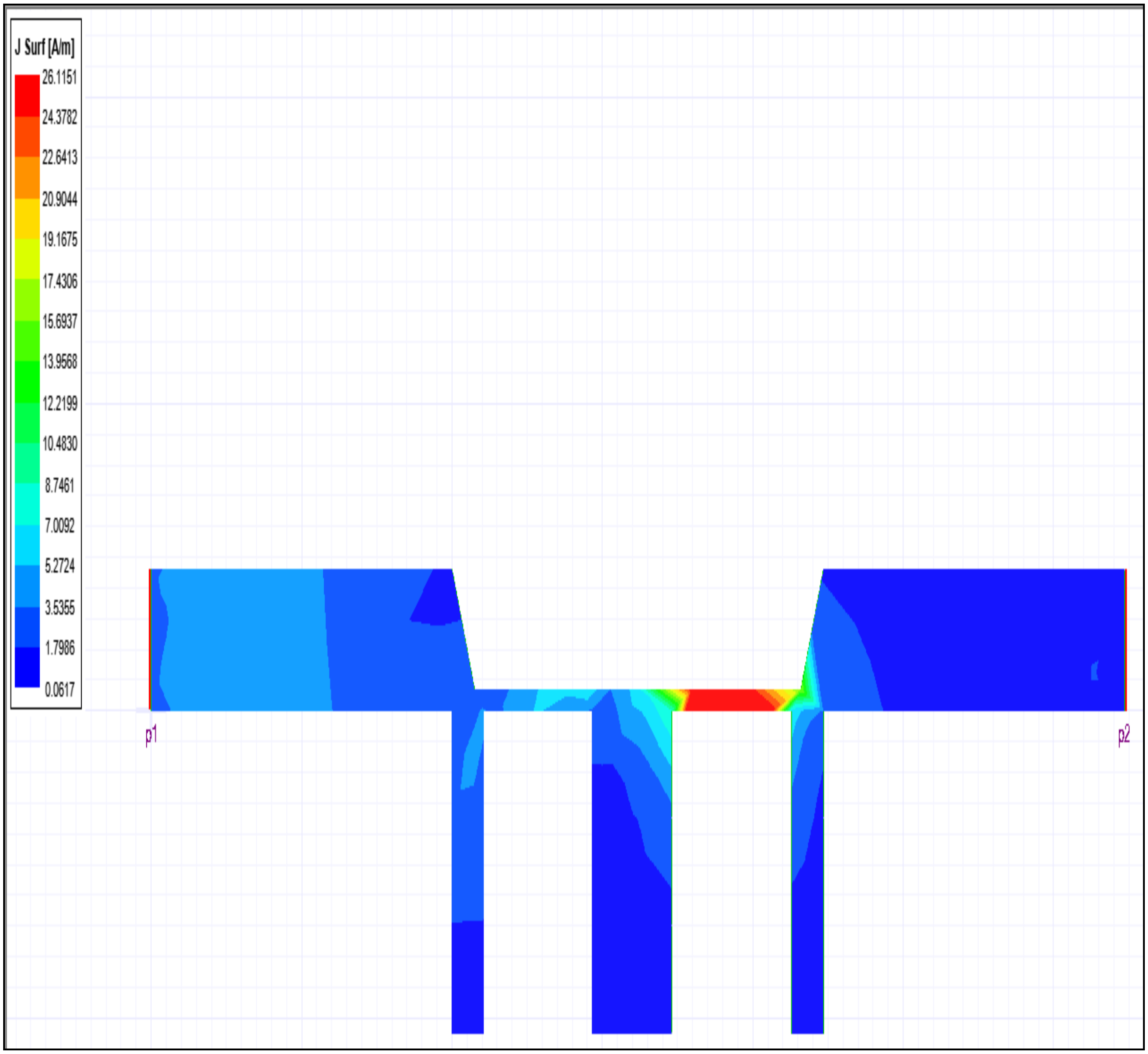
> **Note:**
>
> The only **Solution** available is **PlanarEMSetup1:Sweep2**, since only the discrete sweep type has the option to output currents.

2. From the **Intrinsic Variables** area, select **2.1GHz** from the **Freq** (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 and close the **Create Field Plot** window.
5. If appropriate, refocus from the overlay. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Field Overlays** folder. Then double-click **J Surf**.



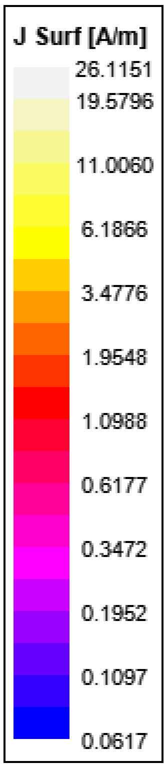
Animated Demonstration

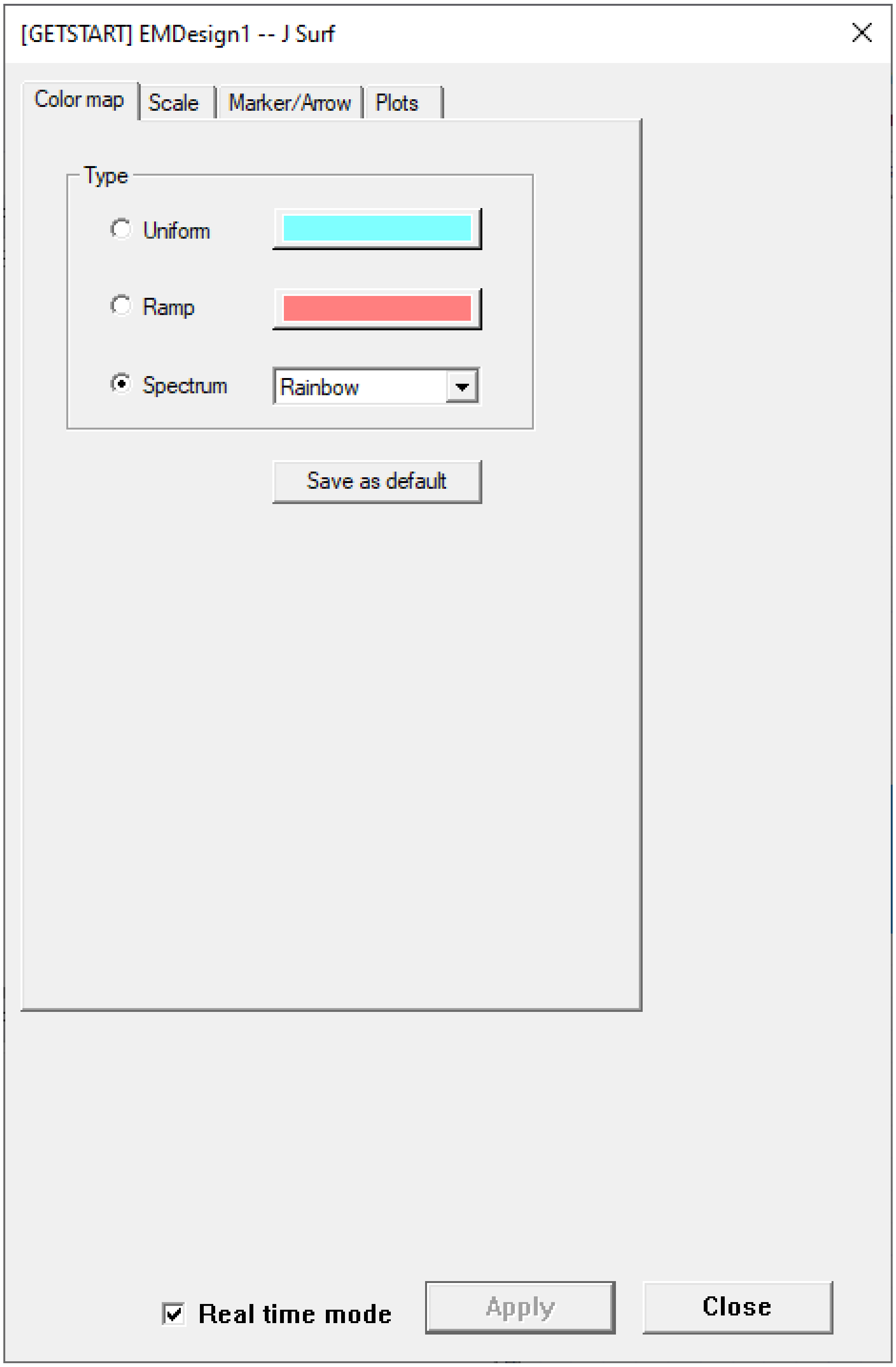
Continue to Modify and Animated Current Overlay.

Modify and Animate the Current Overlay

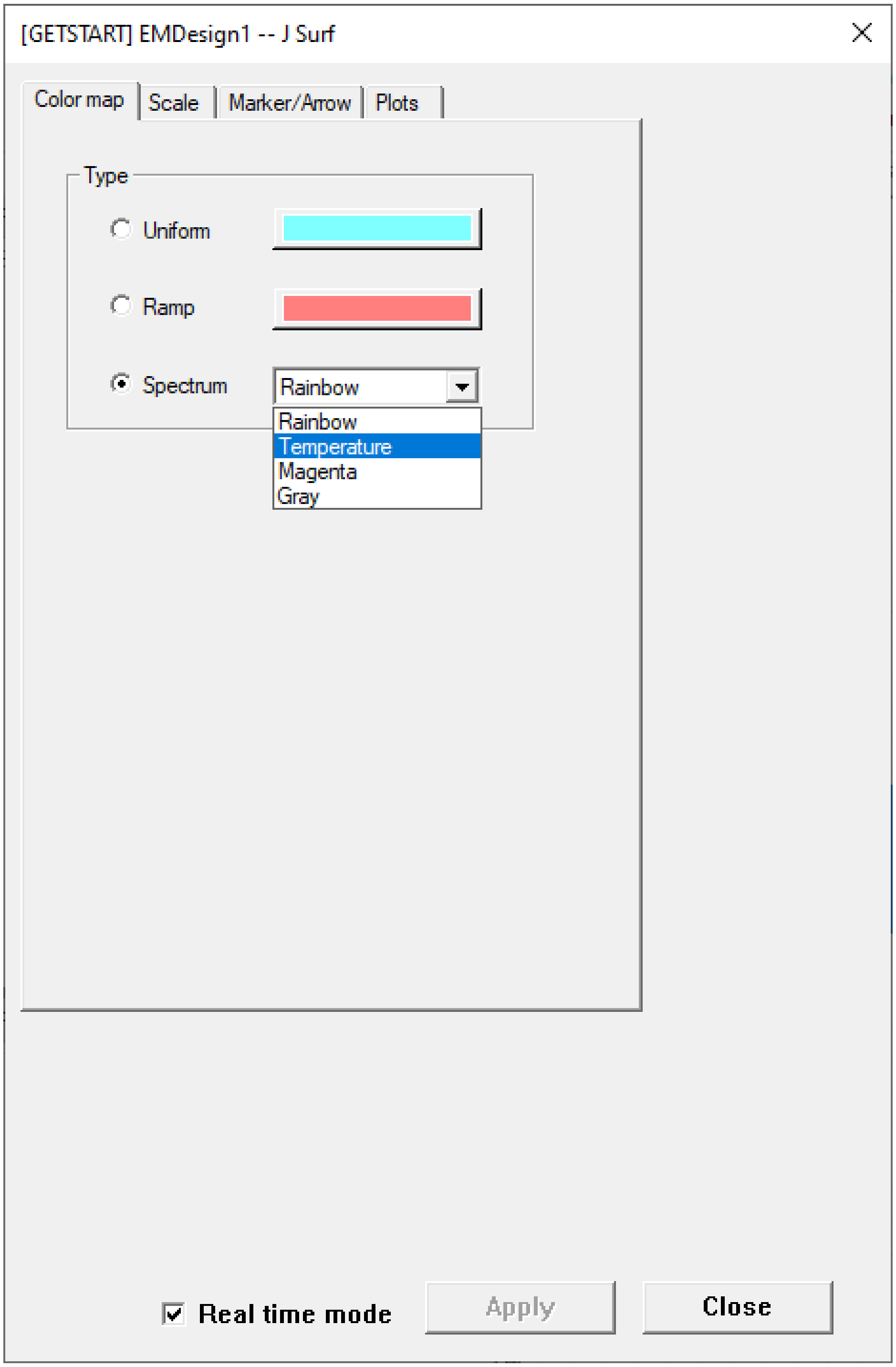
Complete these steps to modify a plot legend (colors and scale) and animate the current plot (i.e., **J Surf**):

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

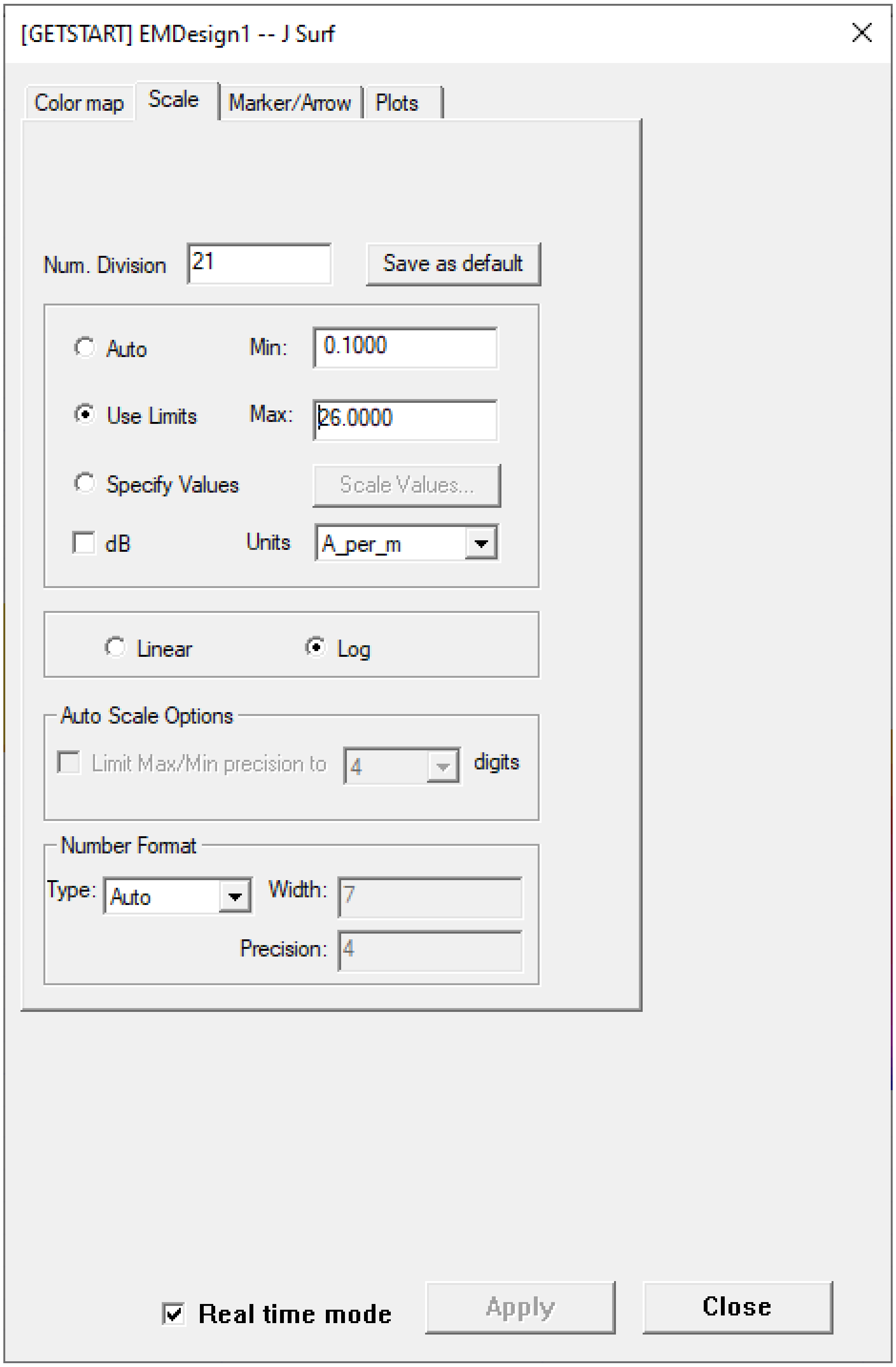



[GETSTART] EMDesign1 -- J Surf
Color map
Scale
Marker/Arrow
Plots
Type
Uniform
Ramp
Spectrum
Rainbow
Save as default
Real time mode
Apply
Close

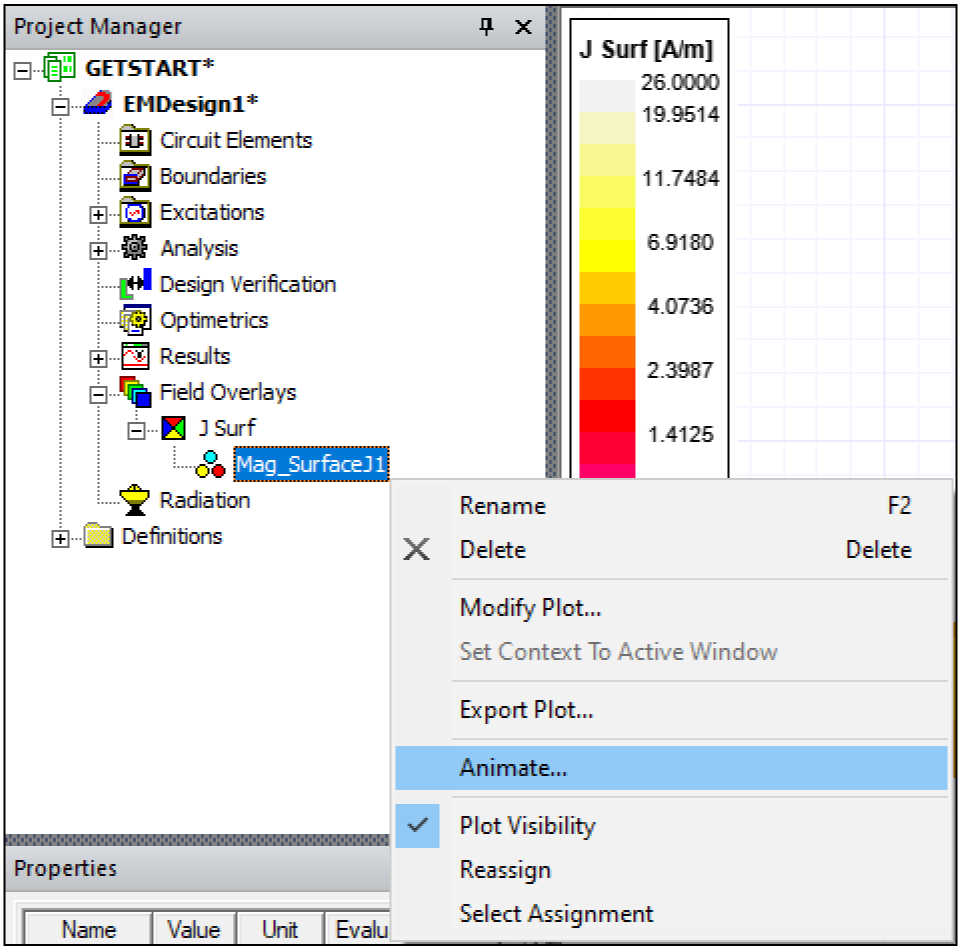
2. From the **Color map** tab, select **Temperature** from the **Spectrum** drop-down menu.

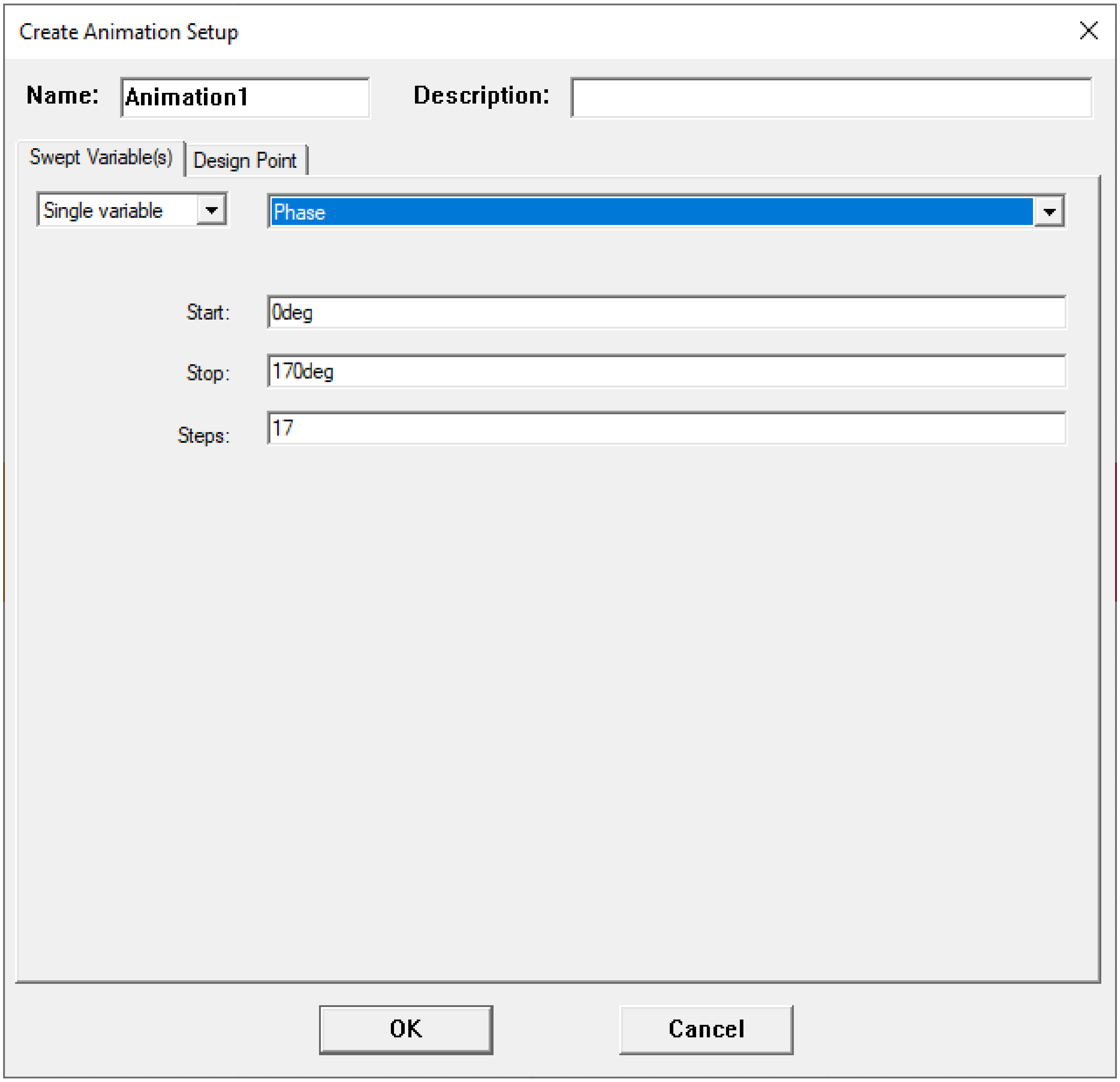


3. Select the **Scale** tab and do the following:
   a. In the **Num. Division** field, enter **21**.
   b. Select **Use Limits**.
   c. In the **Min** field, enter **0.1**.
   d. In the **Max** field, enter **26.0**.
   e. Select **Log** (to produce a logarithmic scale).



4. **Close** the plot settings window.
5. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Field Overlays** > **J Surf**. Then right-click **Mag_SurfaceJ1** and click **Animate** to open the **Create Animation Setup** window.

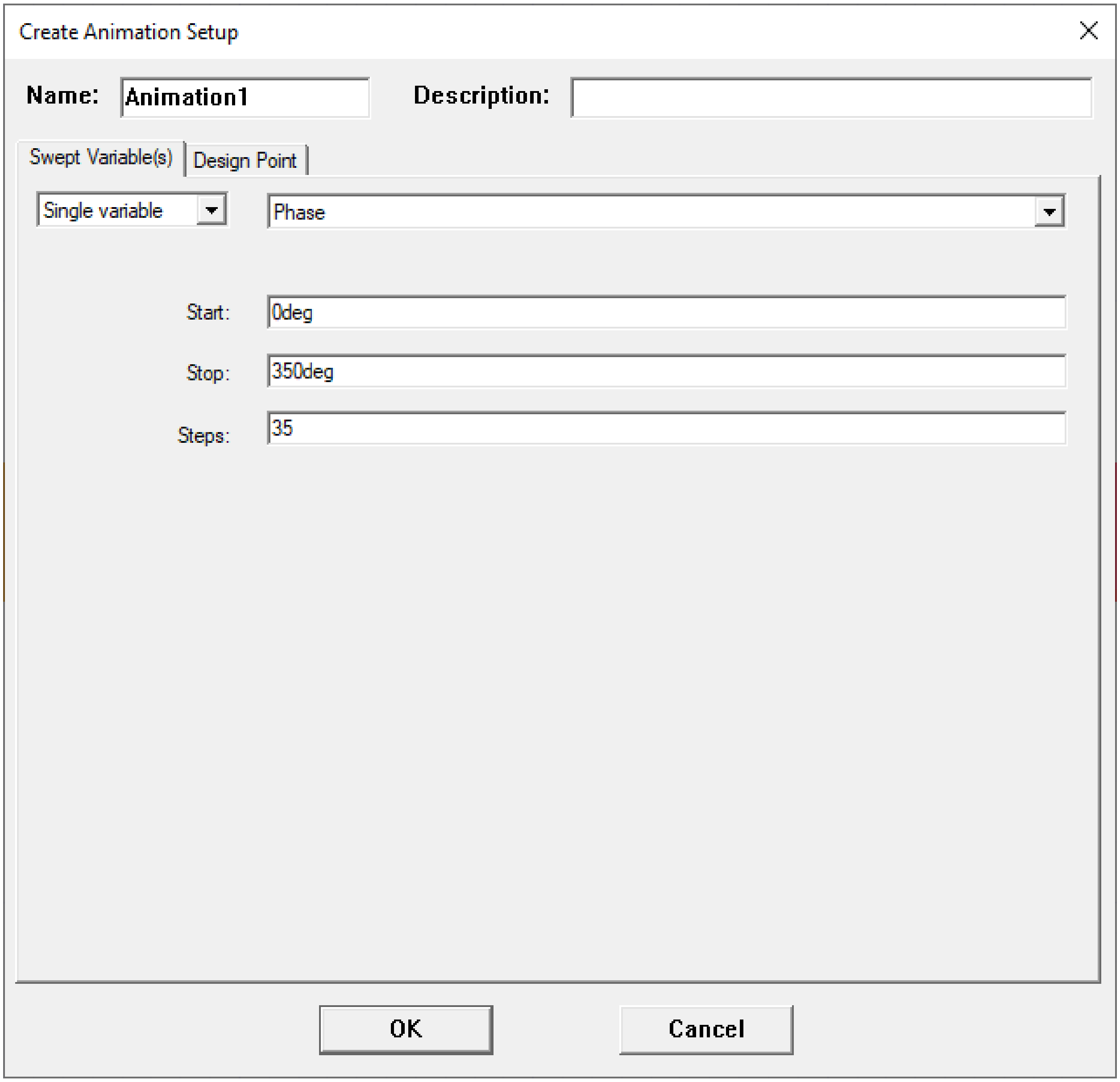

Project Manager
GETSTART*
EMDesign1*
Circuit Elements
Boundaries
Excitations
Analysis
Design Verification
Optimetrics
Results
Field Overlays
J Surf
Mag_SurfaceJ1
Radiation
Definitions
Properties
Name
Value
Unit
Evalu
J Surf [A/m]
26.0000
19.9514
11.7484
6.9180
4.0736
2.3987
1.4125
Rename F2
Delete Delete
Modify Plot...
Set Context To Active Window
Export Plot...
Animate...
Plot Visibility
Reassign
Select Assignment



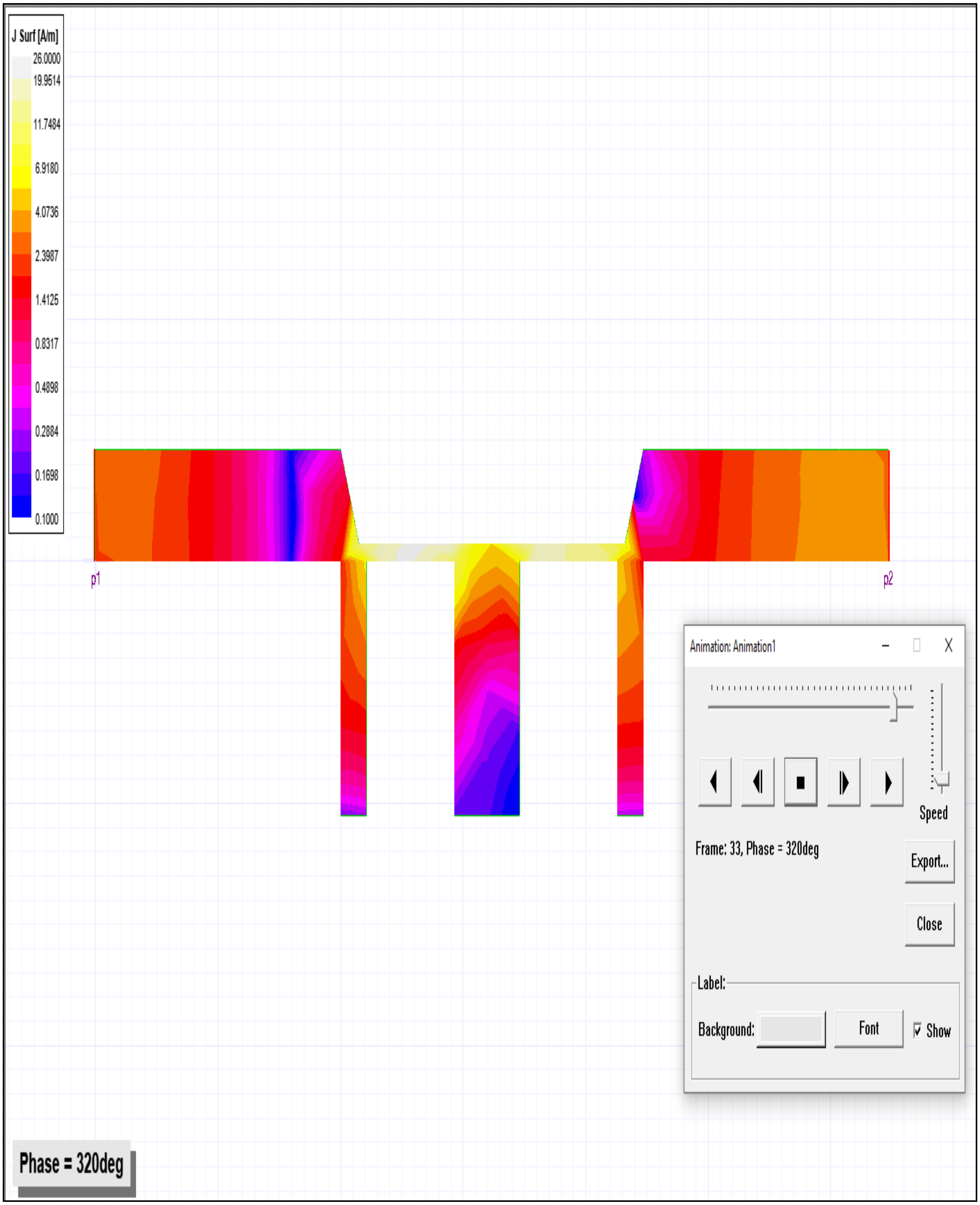
6. From the Swept Variables tab, do the following:
    a. Ensure **Single variable** and **Phase** are selected from the drop-down menus.
    b. In the **Stop** field, enter **350deg**.
    c. In the **Steps** field, enter **35**.

Note:

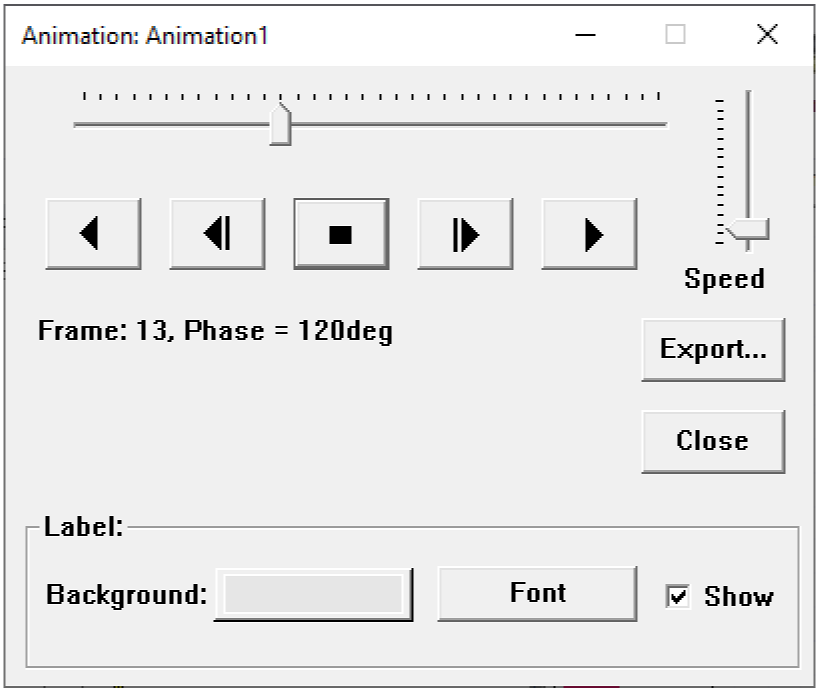
These settings result in an animation with 10° (**35deg** / **350deg**) phase increments. A 360° phase (**360deg**) would be identical to the 0° phase, resulting in a redundant animation frame, and is thus omitted.



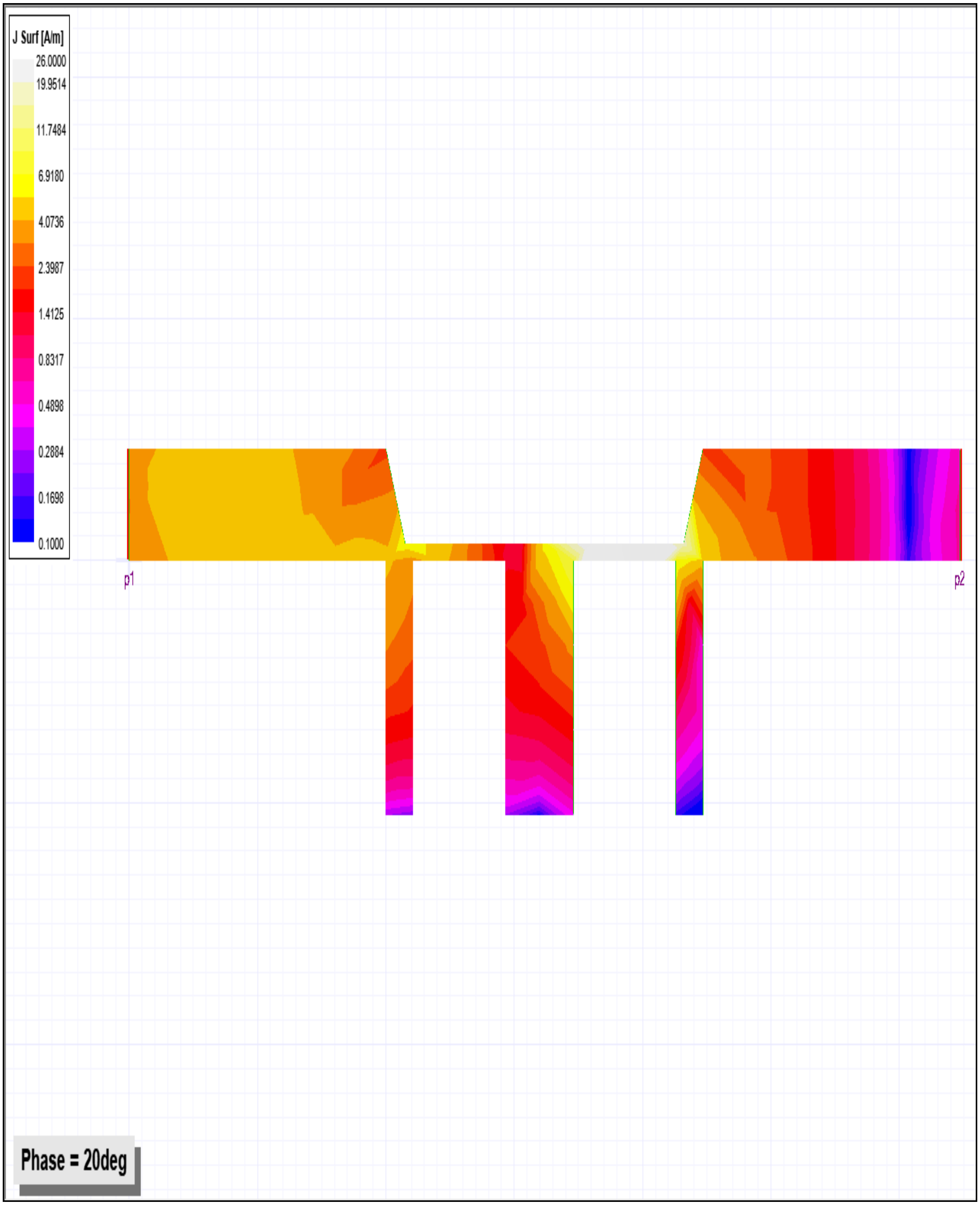
7. Click **OK** to close the **Create Animation Setup** window to open an animation control panel and start the animation in the **Layout Editor**.


J Surf [A/m]
26.0000
19.9514
11.7484
6.9180
4.0736
2.3987
1.4125
0.8317
0.4898
0.2884
0.1698
0.1000
p1
p2
Animation: Animation1
Speed
Frame: 33, Phase = 320deg
Export...
Close
Label:
Background:
Font
Show
Phase = 320deg

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.



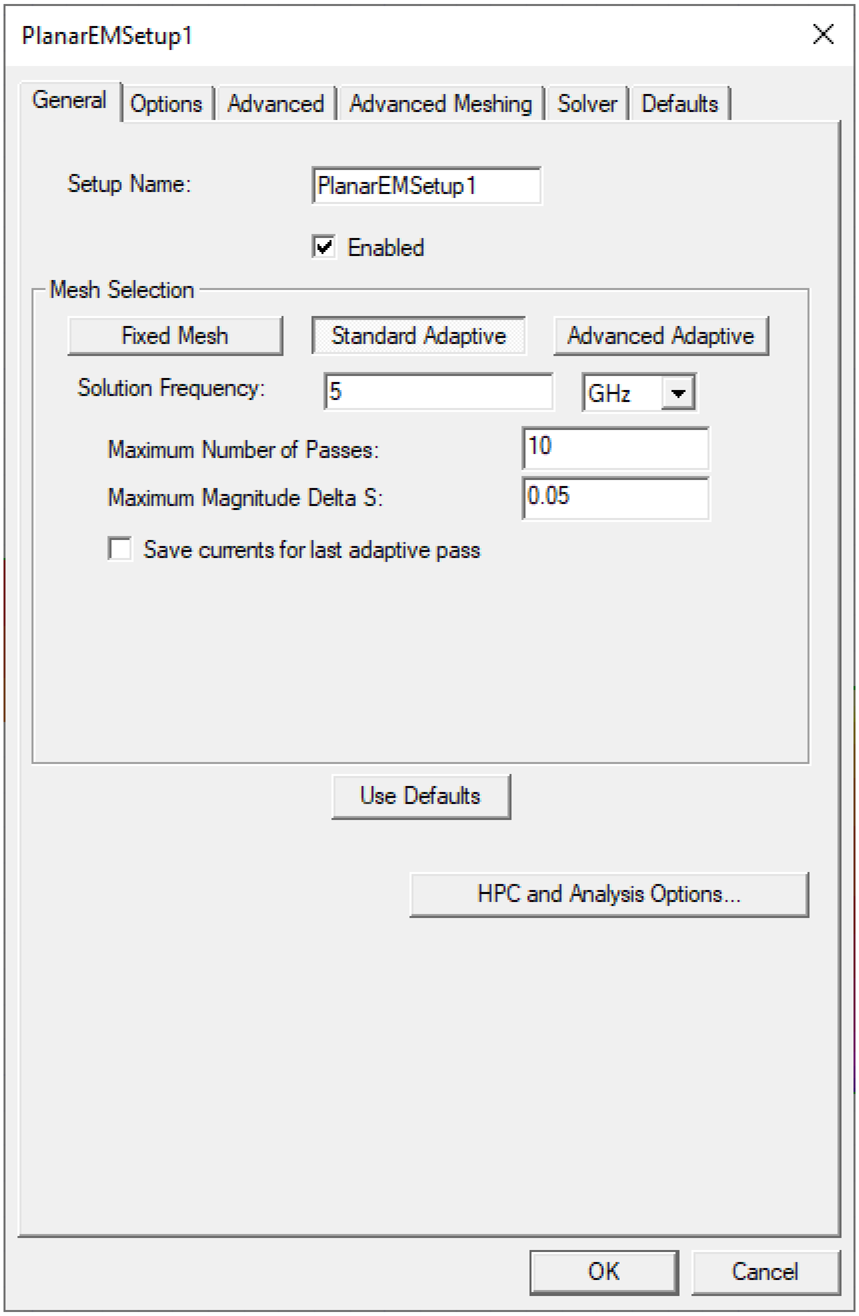
10. From the animation control panel, click **Close** to end the animation.

Continue to Create Far Field Plot.

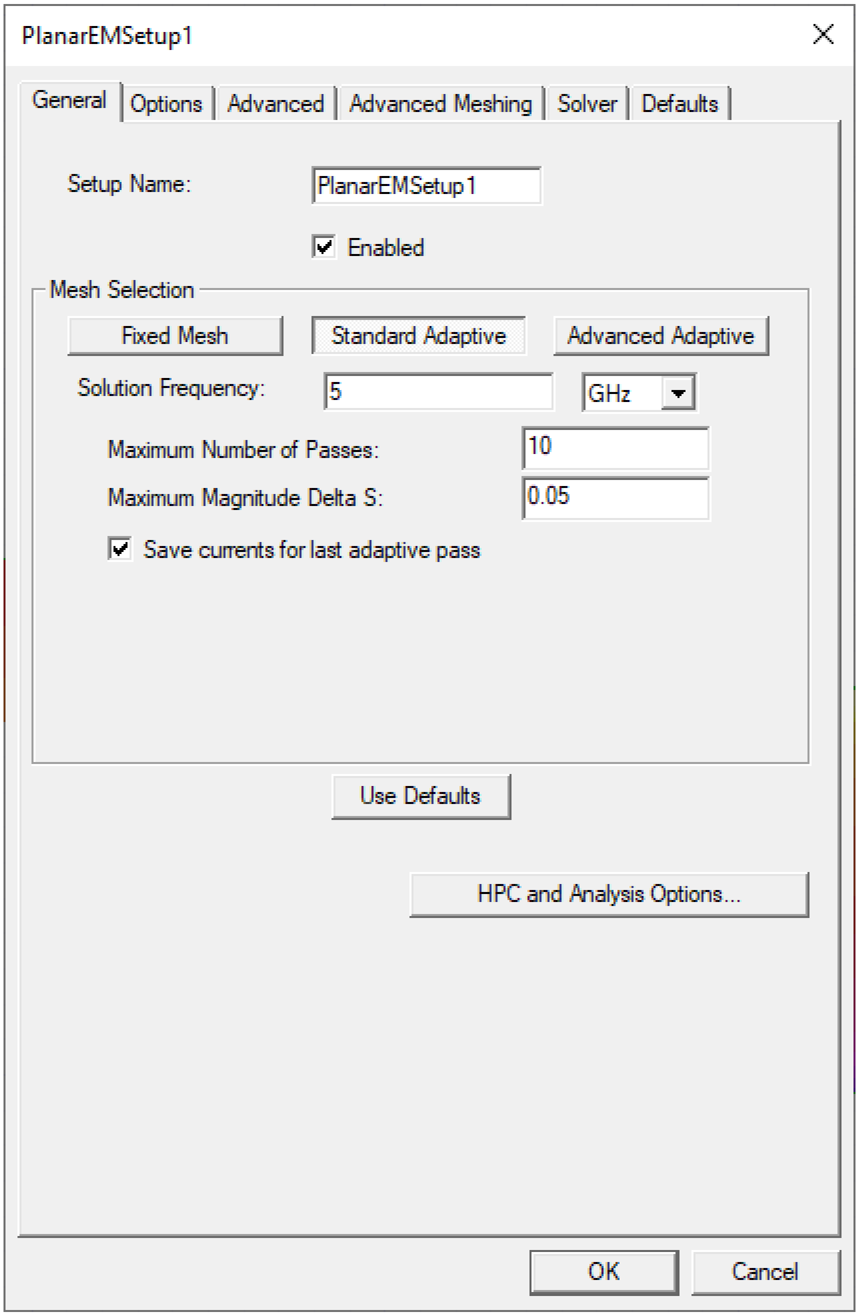
Create Far Field Plot

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. To save results, and current and field results available to plot, satisfy one of the following conditions, then complete these steps.

- **Run a discrete frequency sweep**, as follows:
  - From the **Options** area, check the **Generate surface current** box.
  - Specify multiple discrete frequency points and have the current and field results bavailable for all specified frequencies.
  - **Define the frequency at which the far field results** are viewed.
  - **Animate** the resulting plot. The animation should progress from one frequency to the next.
- **Set up a Planar EM analysis**, as follows:
  a. From the **Planar EM Setup** window > **Mesh Selection** area, choose **Standard Adaptive** or **Advanced Adaptive**.



b. Check the **Save currents for last adaptive pass** box. The current and field results are only available for the last adaptive pass of the specified solution frequency.



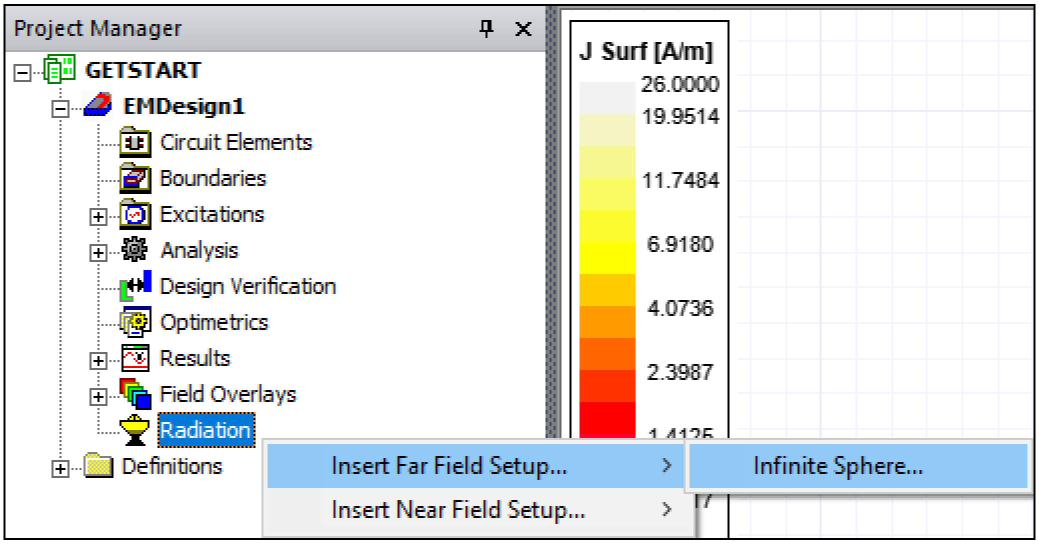
c. Click **OK** to close the **PlanarEMSetup1** window and save changes.

Plot Far Field Results

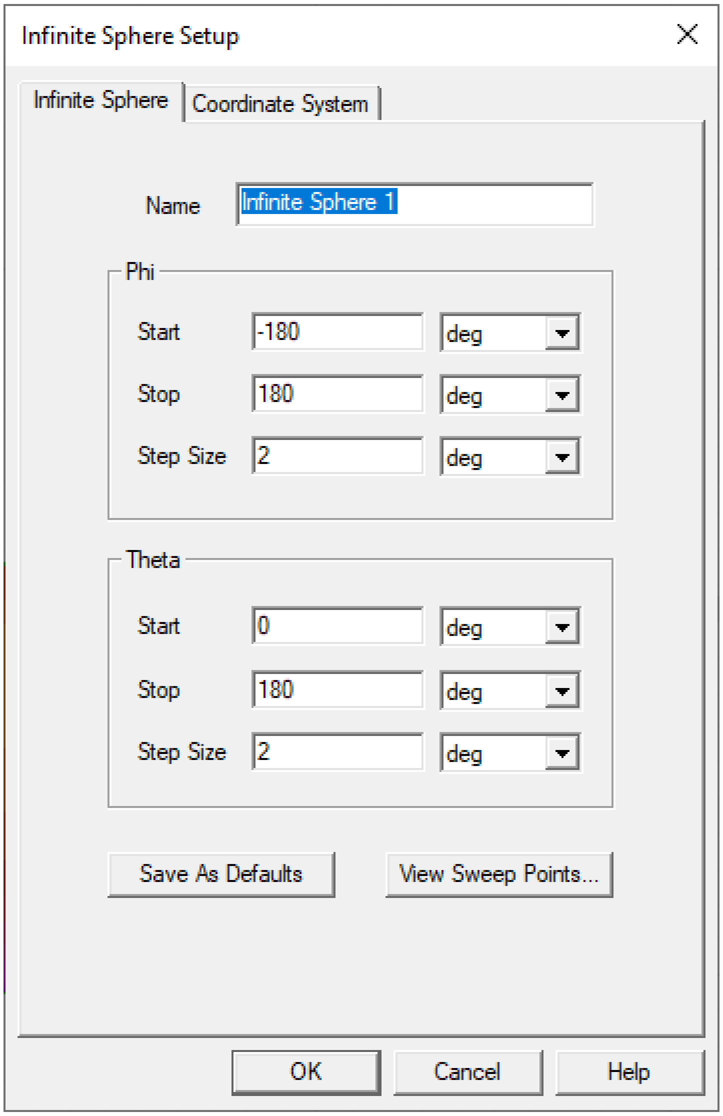
Complete these steps to set up a far field infinite sphere and plot the results.

1. From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Radiation** and select **Insert Far Field Setup...** > **Infinite**

Sphere..

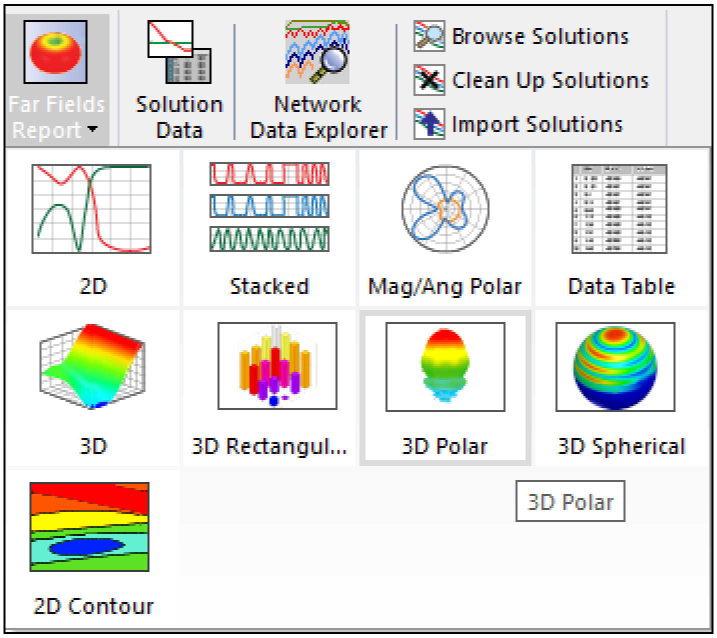


The **Infinite Sphere Setup** window opens.

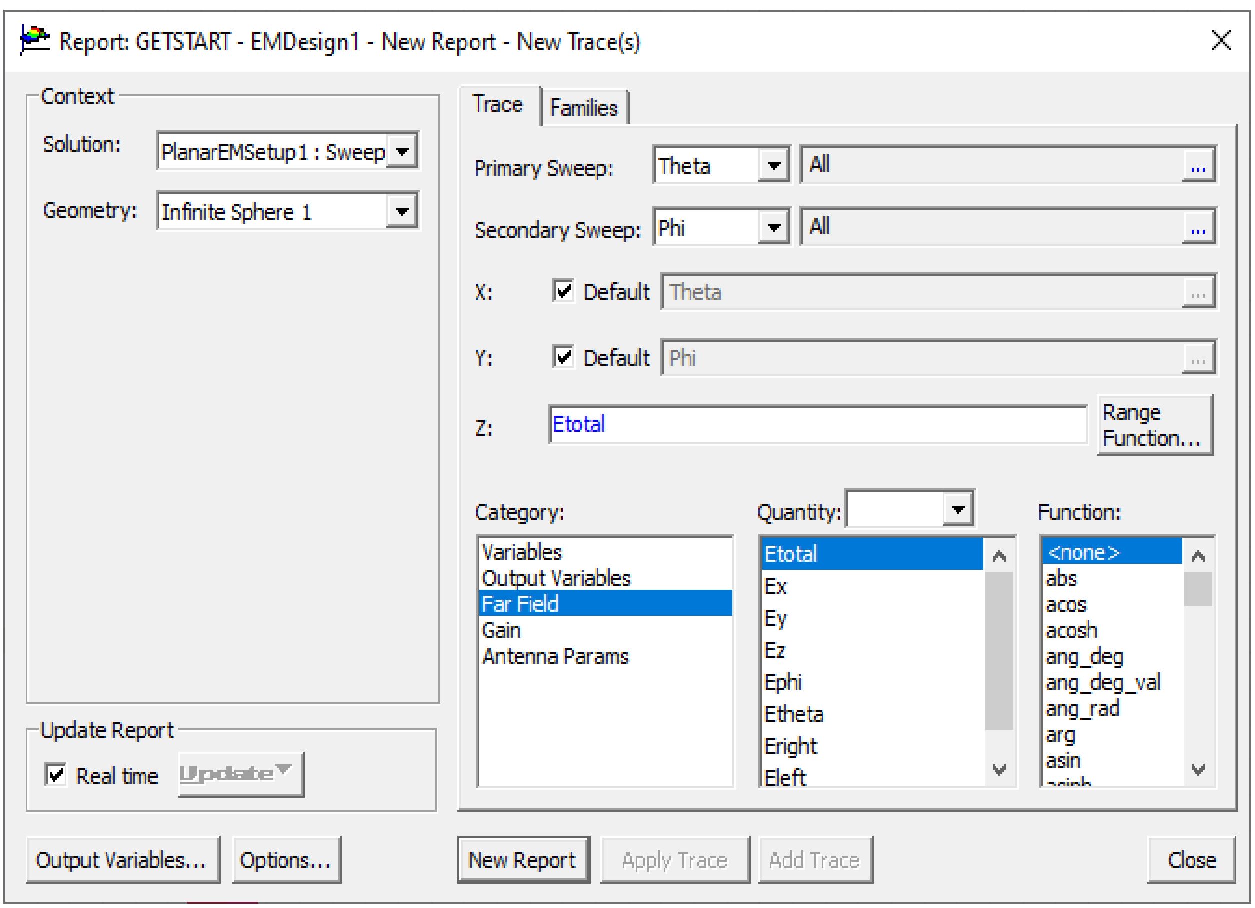


2. Make **no** changes to the default configuration and click **OK** to close the **Infinite Sphere** window and add the far field setup to the design.

3. From the **Results** ribbon, select **Far Fields Report** > **3D Polar**.



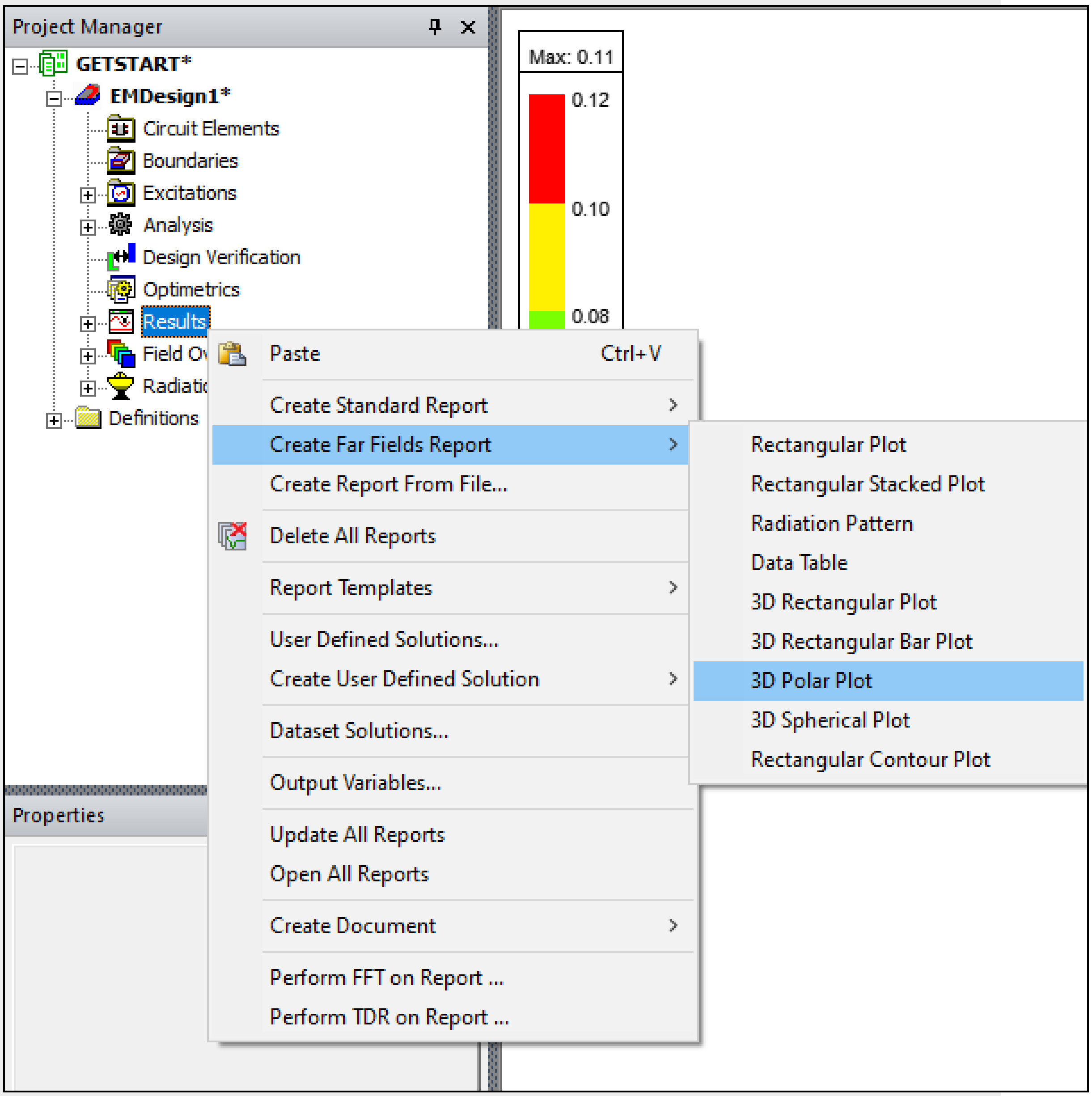
The **Report** window opens.



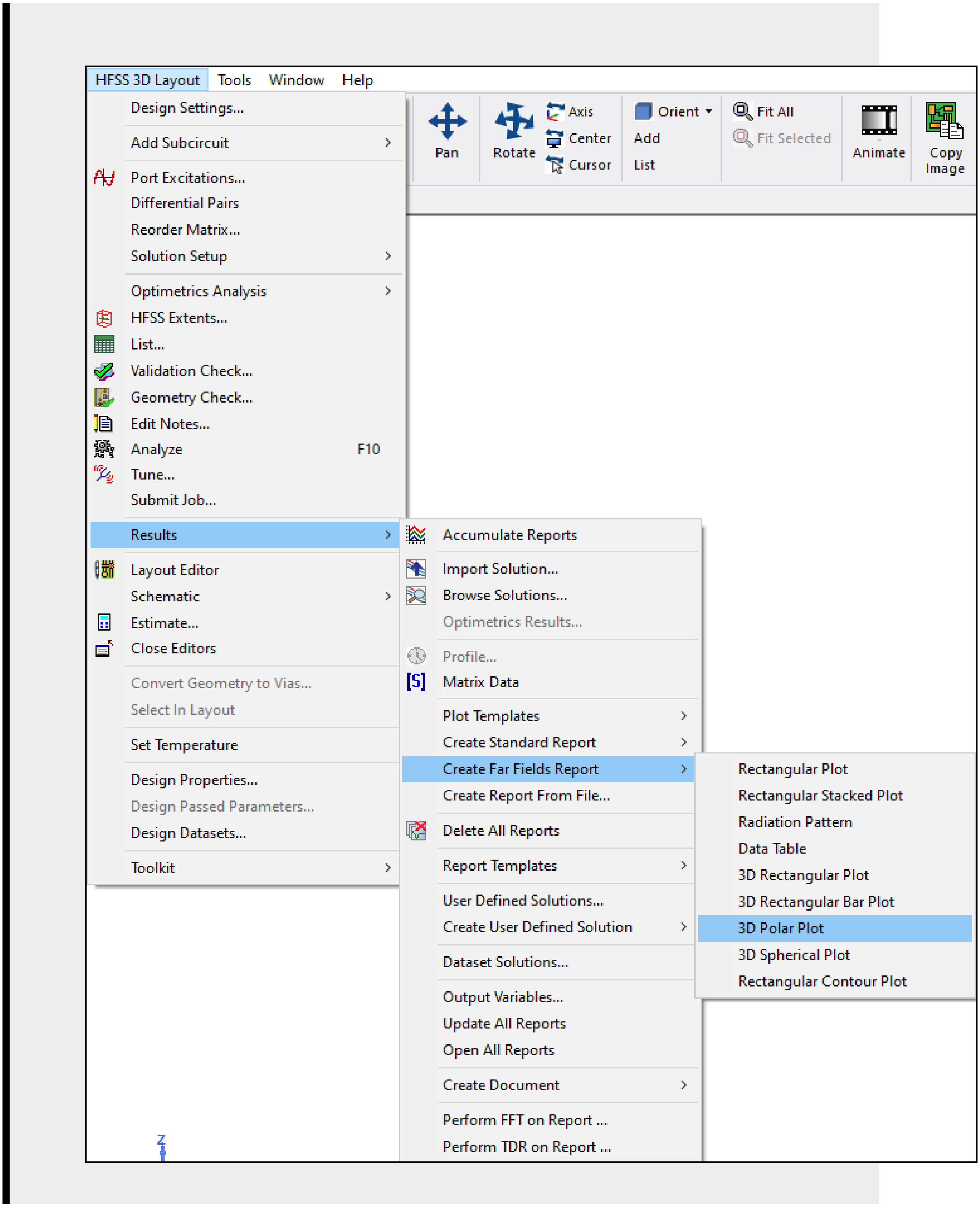
Note:

Alternatively, open the **Report** window by doing one of the following:

- From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Results** and select **Create Far Fields Report** > **3D Polar Plot**.

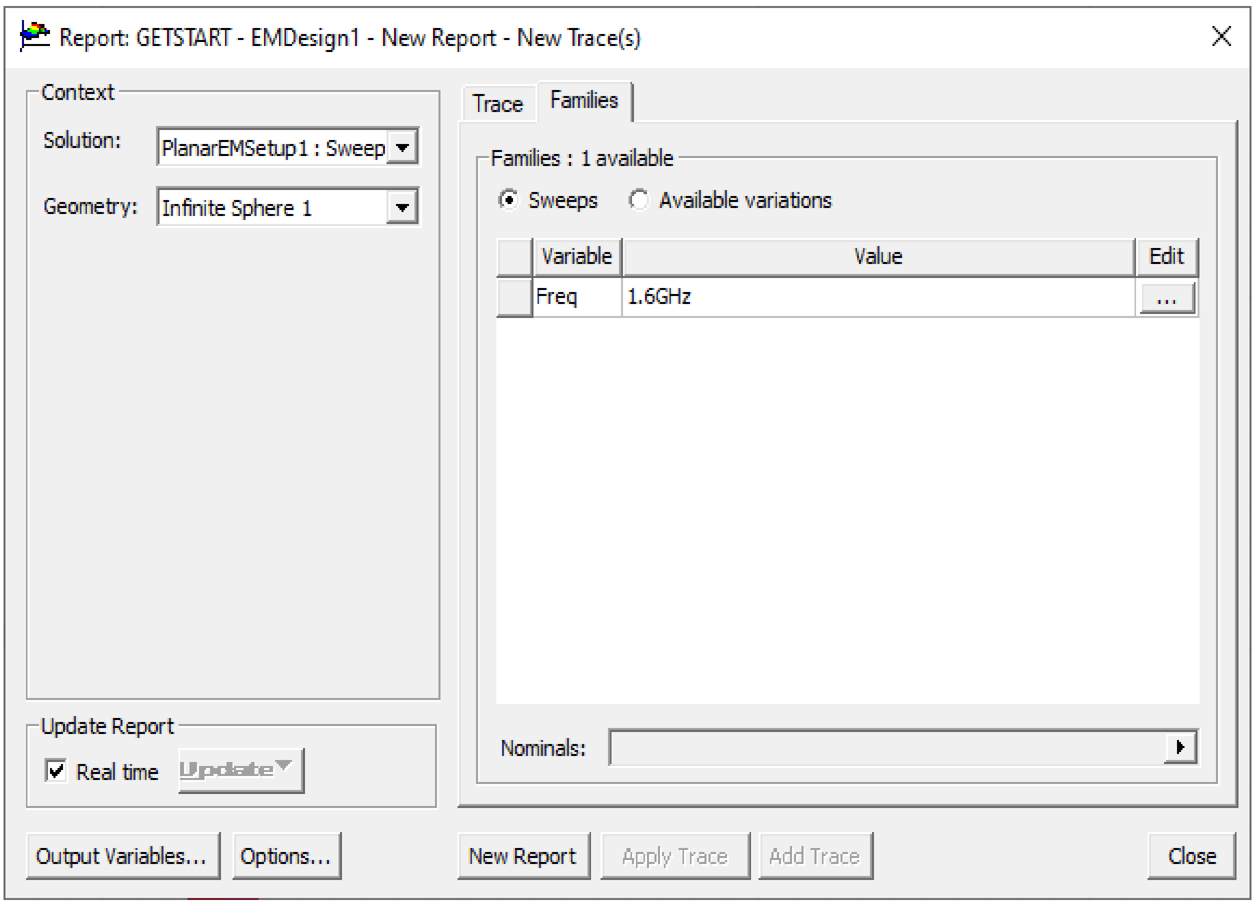


- From **HFSS 3D Layout**, select **Results** > **Create Far Fields Report** > **3D Polar Plot**.

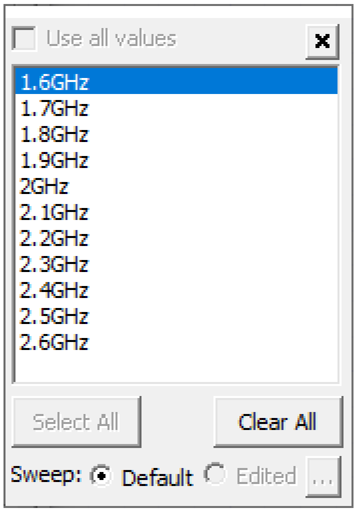

HFSS 3D Layout
Tools
Window
Help
Design Settings...
Add Subcircuit
Port Excitations...
Differential Pairs
Reorder Matrix...
Solution Setup
Optimetrics Analysis
HFSS Extents...
List...
Validation Check...
Geometry Check...
Edit Notes...
Analyze
F10
Tune...
Submit Job...
Results
Layout Editor
Schematic
Estimate...
Close Editors
Convert Geometry to Vias...
Select In Layout
Set Temperature
Design Properties...
Design Passed Parameters...
Design Datasets...
Toolkit
Pan
Rotate
Axis
Center
Cursor
Orient
Add
List
Fit All
Fit Selected
Animate
Copy Image
Accumulate Reports
Import Solution...
Browse Solutions...
Optimetrics Results...
Profile...
Matrix Data
Plot Templates
Create Standard Report
Create Far Fields Report
Create Report From File...
Delete All Reports
Report Templates
User Defined Solutions...
Create User Defined Solution
Dataset Solutions...
Output Variables...
Update All Reports
Open All Reports
Create Document
Perform FFT on Report ...
Perform TDR on Report ...
Rectangular Plot
Rectangular Stacked Plot
Radiation Pattern
Data Table
3D Rectangular Plot
3D Rectangular Bar Plot
3D Polar Plot
3D Spherical Plot
Rectangular Contour Plot
Z

> **Note:**
>
> In Step 4, the **Solution** and **Geometry** drop-down menus pre-select the only available choices that provide far field results (i.e., **PlanarEMSetup1: Sweep2** is selected in the **Solution** drop-down menu, and **Infinite Sphere 1** is selected in the **Geometry** drop-down menu) and the lists in the **Trace** tab pre-select the chosen choices (i.e., **Far Field** is selected in the **Category** list, **Etotal** is selected in the **Quantity** list, and **<none>** is selected in the **Function** list).

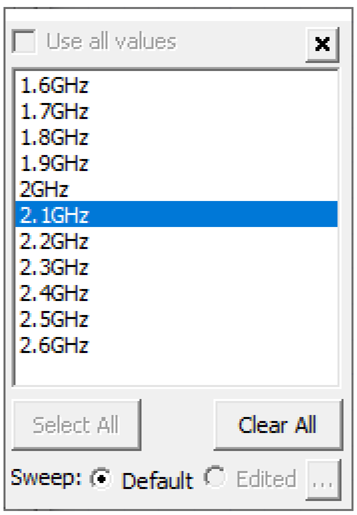
4. Select the **Families** tab. From the **Families** tab > **Freq** row, click the **[...]** button in the **Edit** column.



A value list opens.



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



6. From the **Report** window, click **New Report** and the far field plot opens under the **Report**

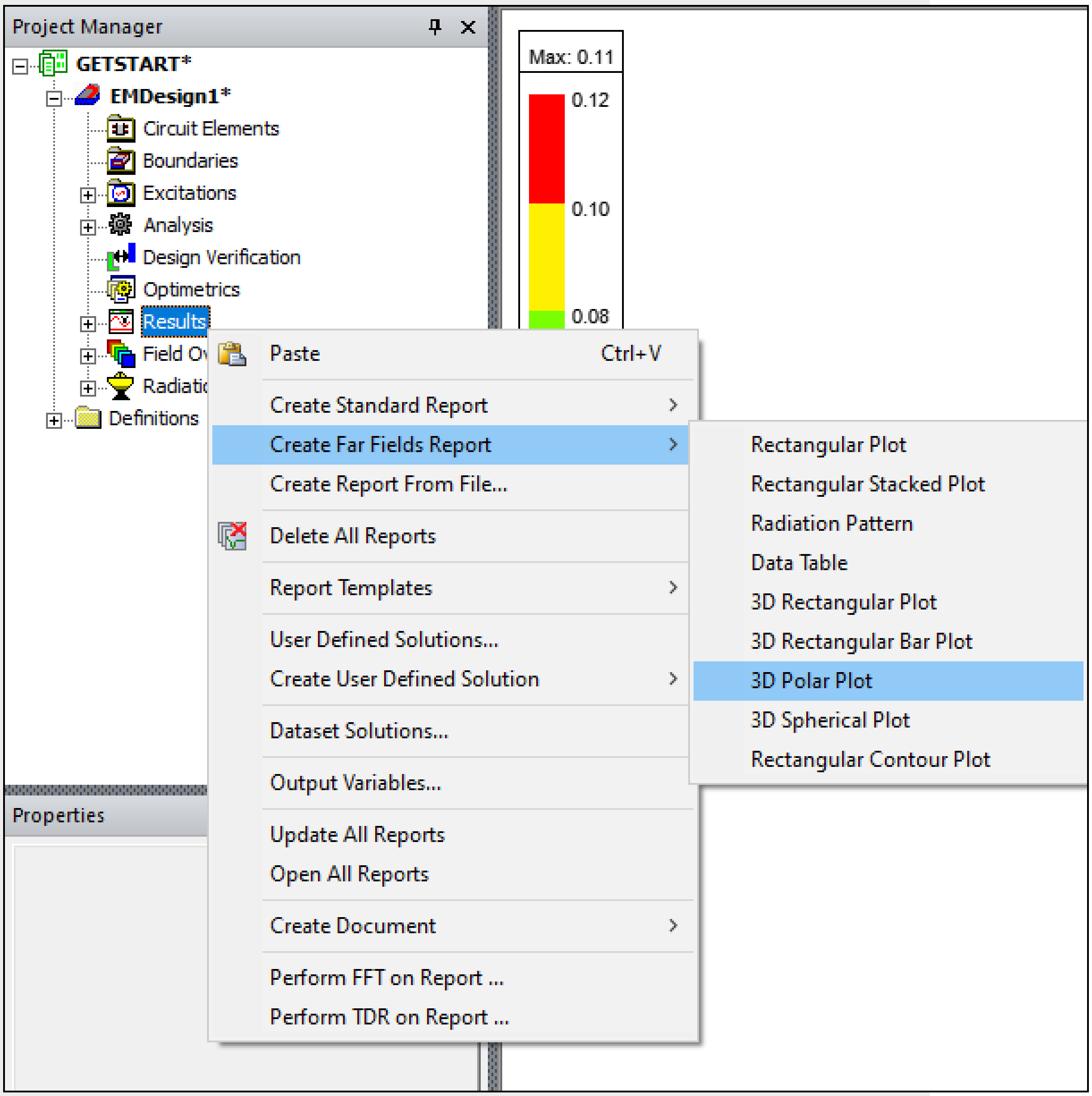
window. **Close** the **Report** window to access the plot.

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

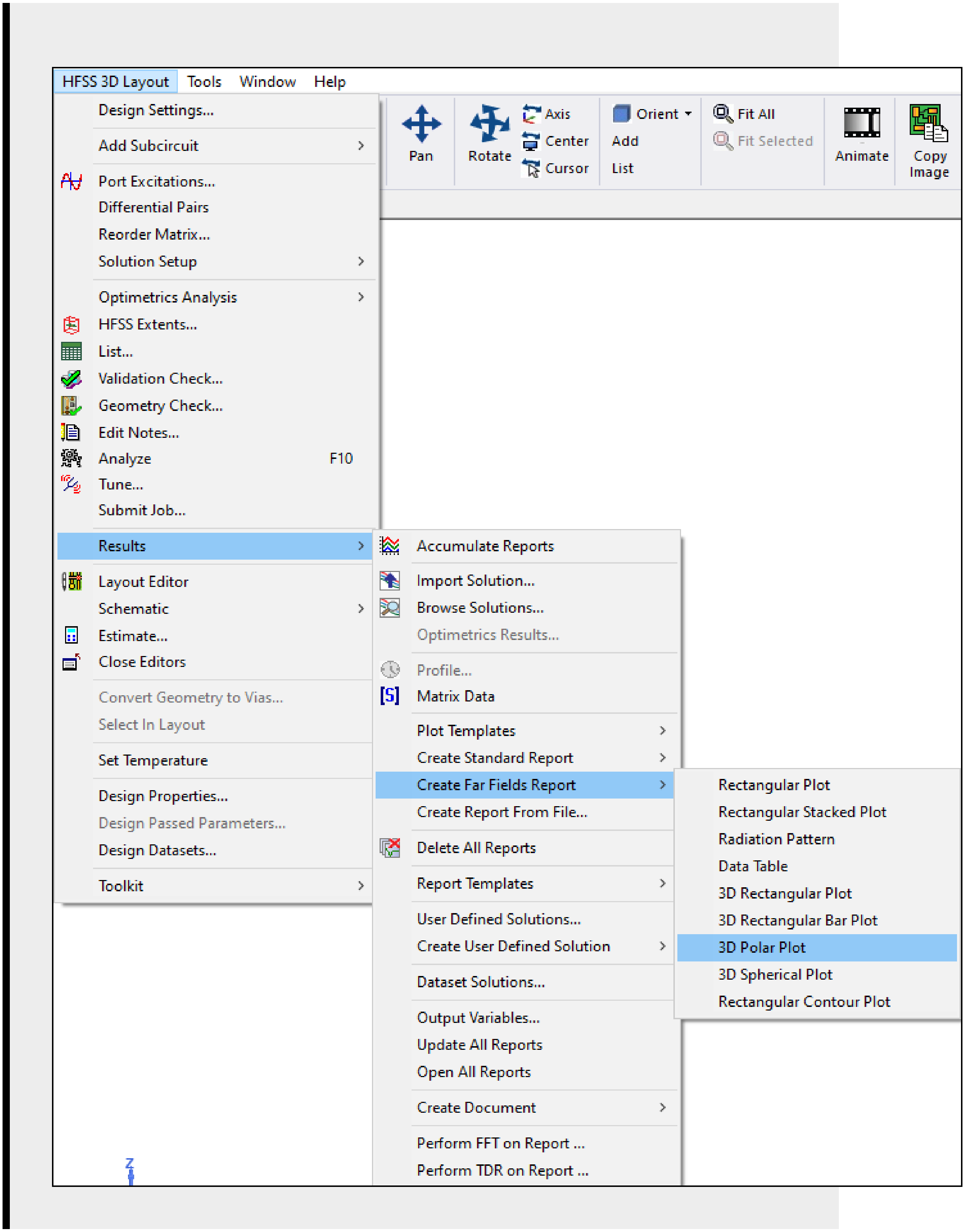
Note:

Alternatively, open the **Report** window by doing one of the following:

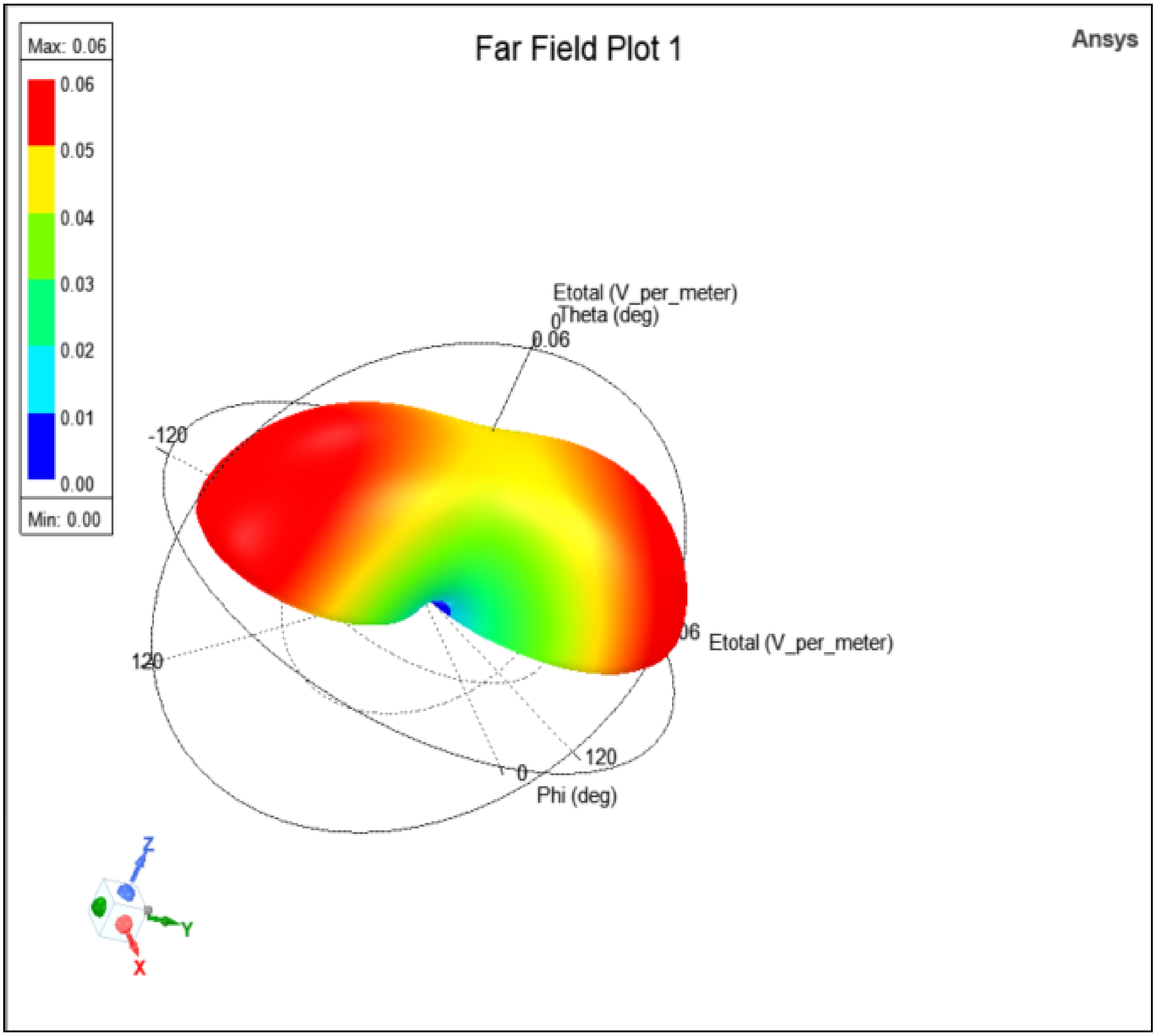
- From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Results** and select **Create Far Fields Report** > **3D Polar Plot**.



- From **HFSS 3D Layout**, select **Results** > **Create Far Fields Report** > **3D Polar Plot**.


HFSS 3D Layout
Tools
Window
Help
Design Settings...
Add Subcircuit
Port Excitations...
Differential Pairs
Reorder Matrix...
Solution Setup
Optimetrics Analysis
HFSS Extents...
List...
Validation Check...
Geometry Check...
Edit Notes...
Analyze F10
Tune...
Submit Job...
Results
Layout Editor
Schematic
Estimate...
Close Editors
Convert Geometry to Vias...
Select In Layout
Set Temperature
Design Properties...
Design Passed Parameters...
Design Datasets...
Toolkit
Pan
Rotate
Axis
Center
Cursor
Orient
Add
List
Fit All
Fit Selected
Animate
Copy Image
Accumulate Reports
Import Solution...
Browse Solutions...
Optimetrics Results...
Profile...
Matrix Data
Plot Templates
Create Standard Report
Create Far Fields Report
Create Report From File...
Delete All Reports
Report Templates
User Defined Solutions...
Create User Defined Solution
Dataset Solutions...
Output Variables...
Update All Reports
Open All Reports
Create Document
Perform FFT on Report ...
Perform TDR on Report ...
Rectangular Plot
Rectangular Stacked Plot
Radiation Pattern
Data Table
3D Rectangular Plot
3D Rectangular Bar Plot
3D Polar Plot
3D Spherical Plot
Rectangular Contour Plot

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

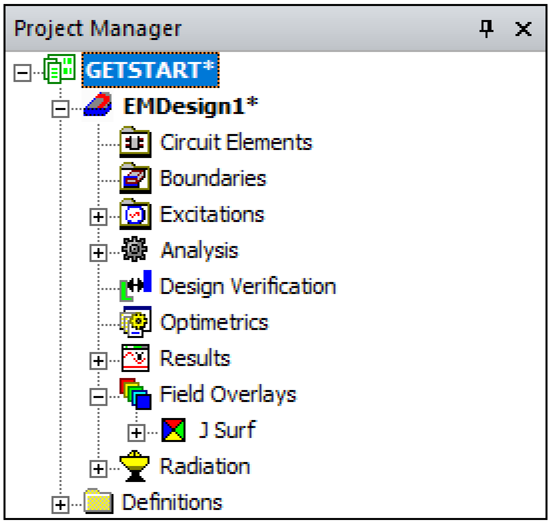


Continue to Overlay Far Field Plot on Model Geometry.

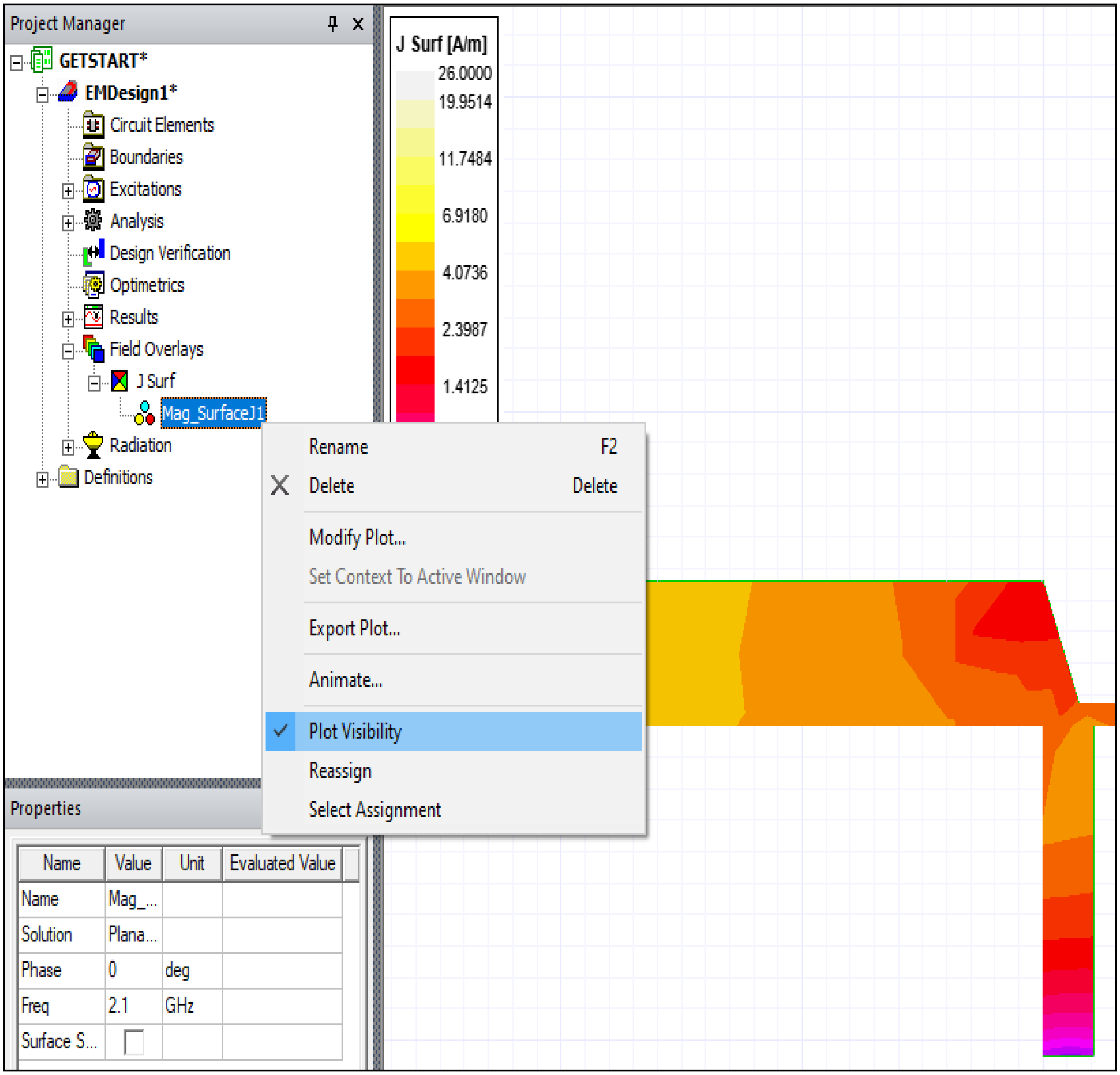
Overlay a Far Field Plot on Model Geometry

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**.

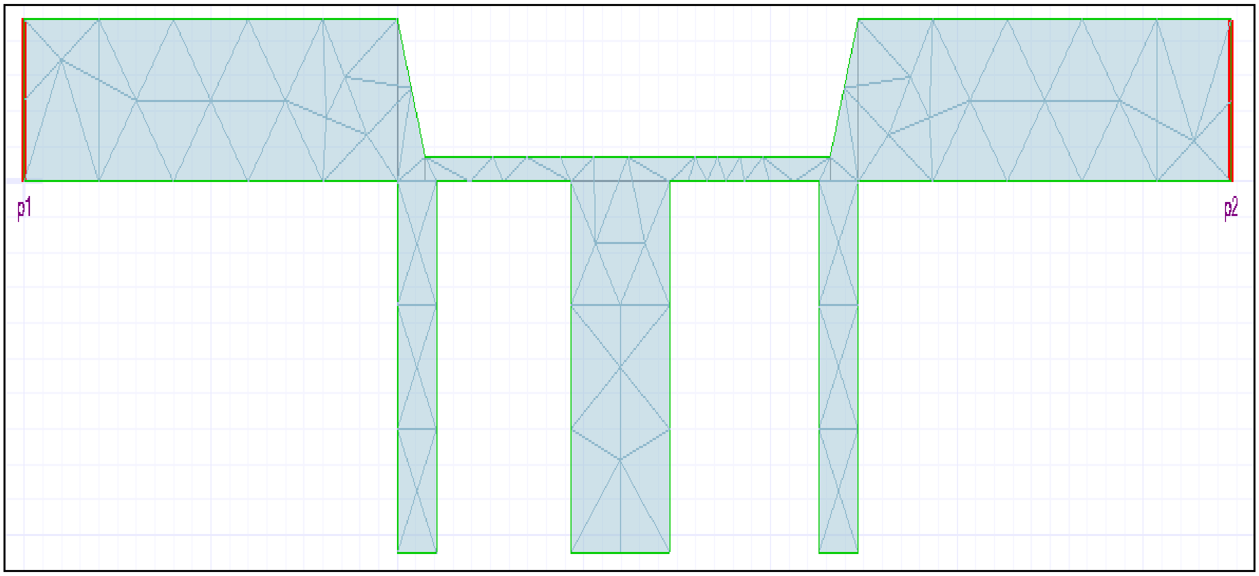
1. To refocus from the field overlay in the **Layout Editor**, expand the **Project Manager** window > **Project Tree** > **[*active design folder*]** > **Field Overlays**. Then double-click **J Surf**.



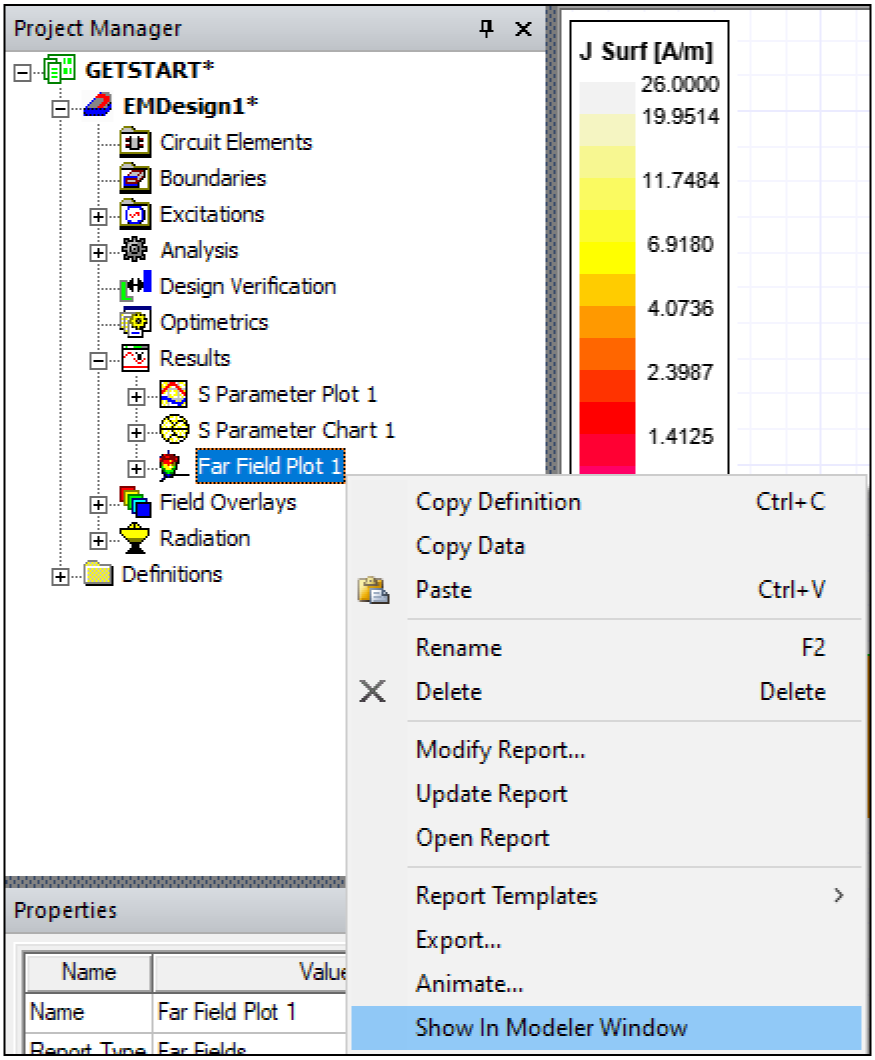
2. Expand **J Surf**, then right-click **Mag_SurfaceJ1** and select **Plot Visibility** to remove the check and the overlay from the design shown in the **Layout Editor**.



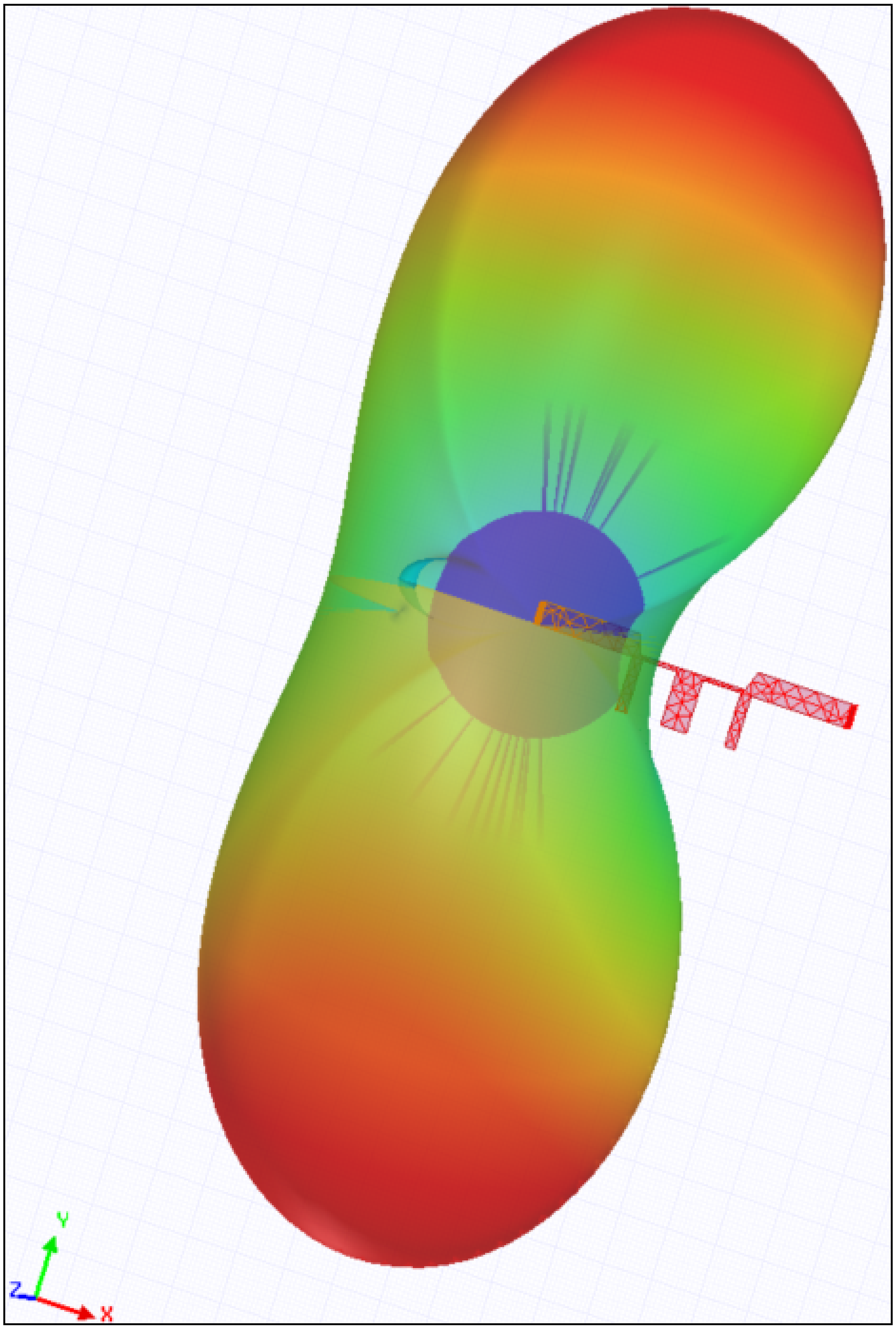
The J Surf legend and color contour map are no longer visibile. Making the surface current results invisible avoids confusion about the values and units of the far field plot overlay.



3. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Results**. Then right-click **Far Field Plot 1** and select **Show In Modeler Window**.



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

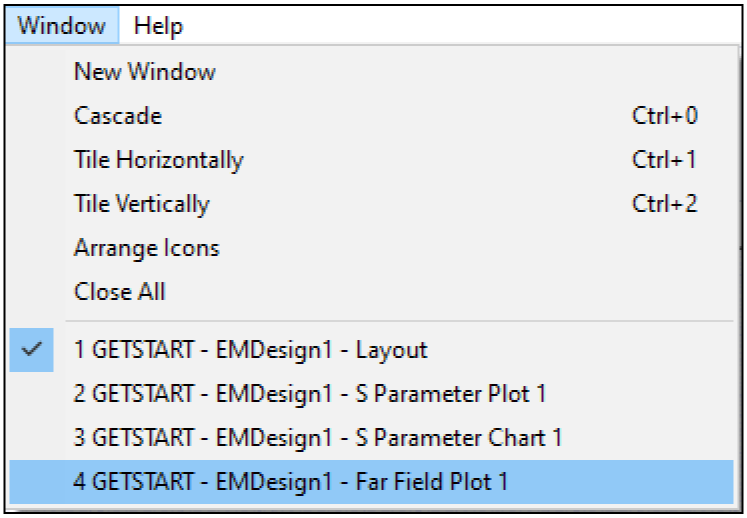


Continue to **Frequency Animated Far Field Plot**.

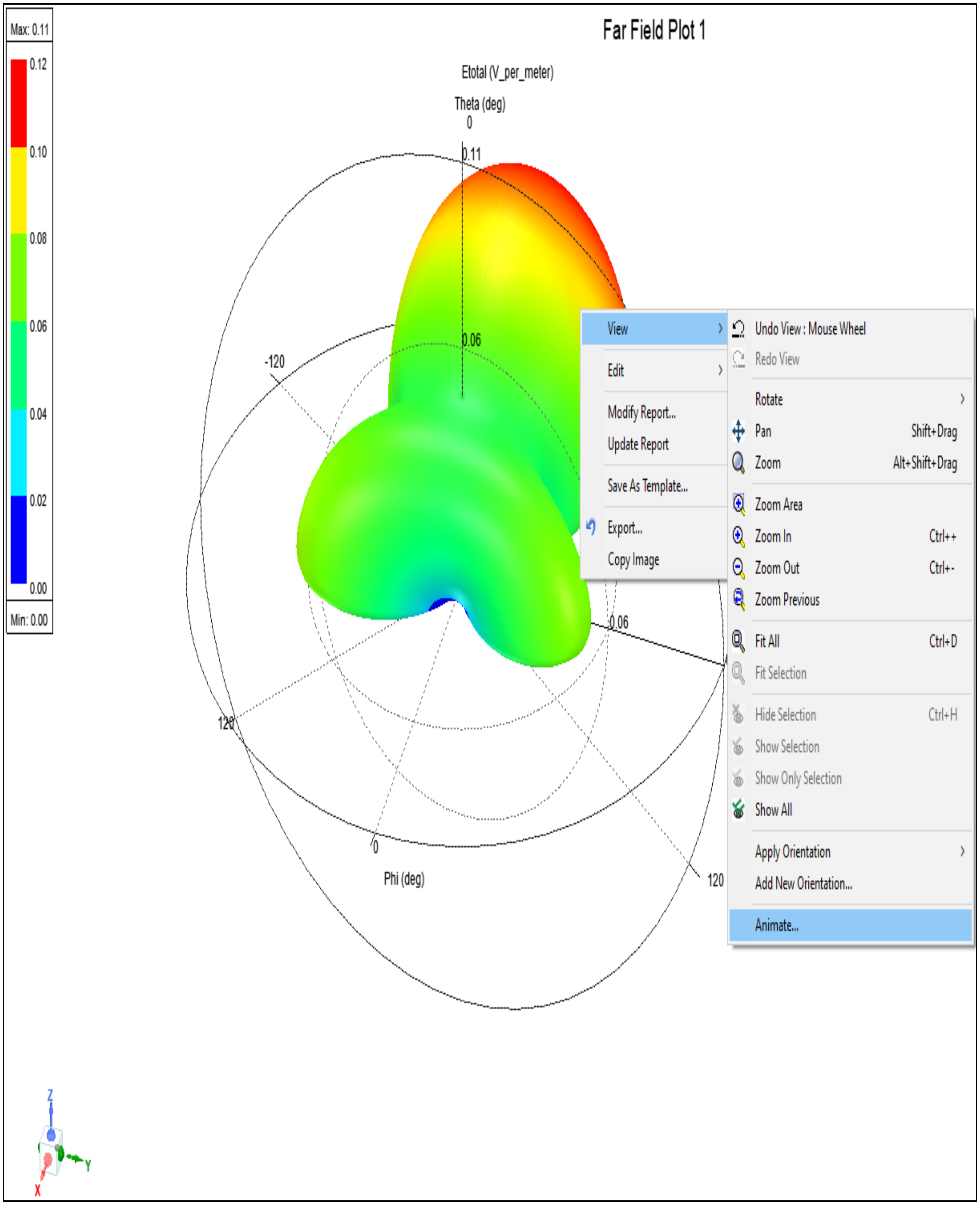
Animate a Field Plot

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

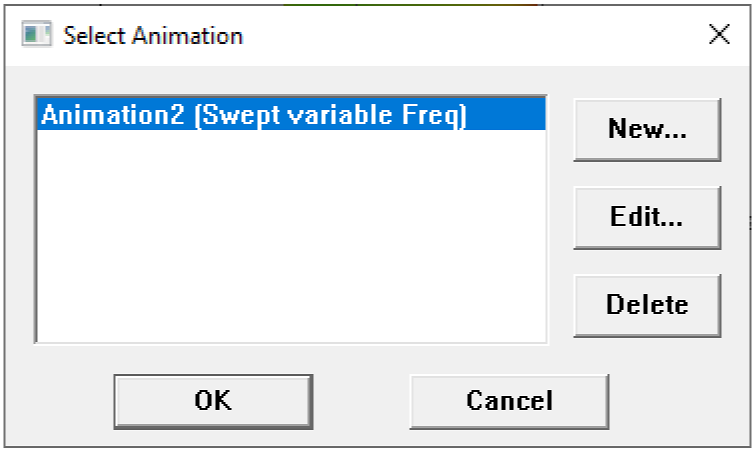
1. From **Window**, select **Far Field Plot 1** to refocus from the far field plot in the **View** tab.



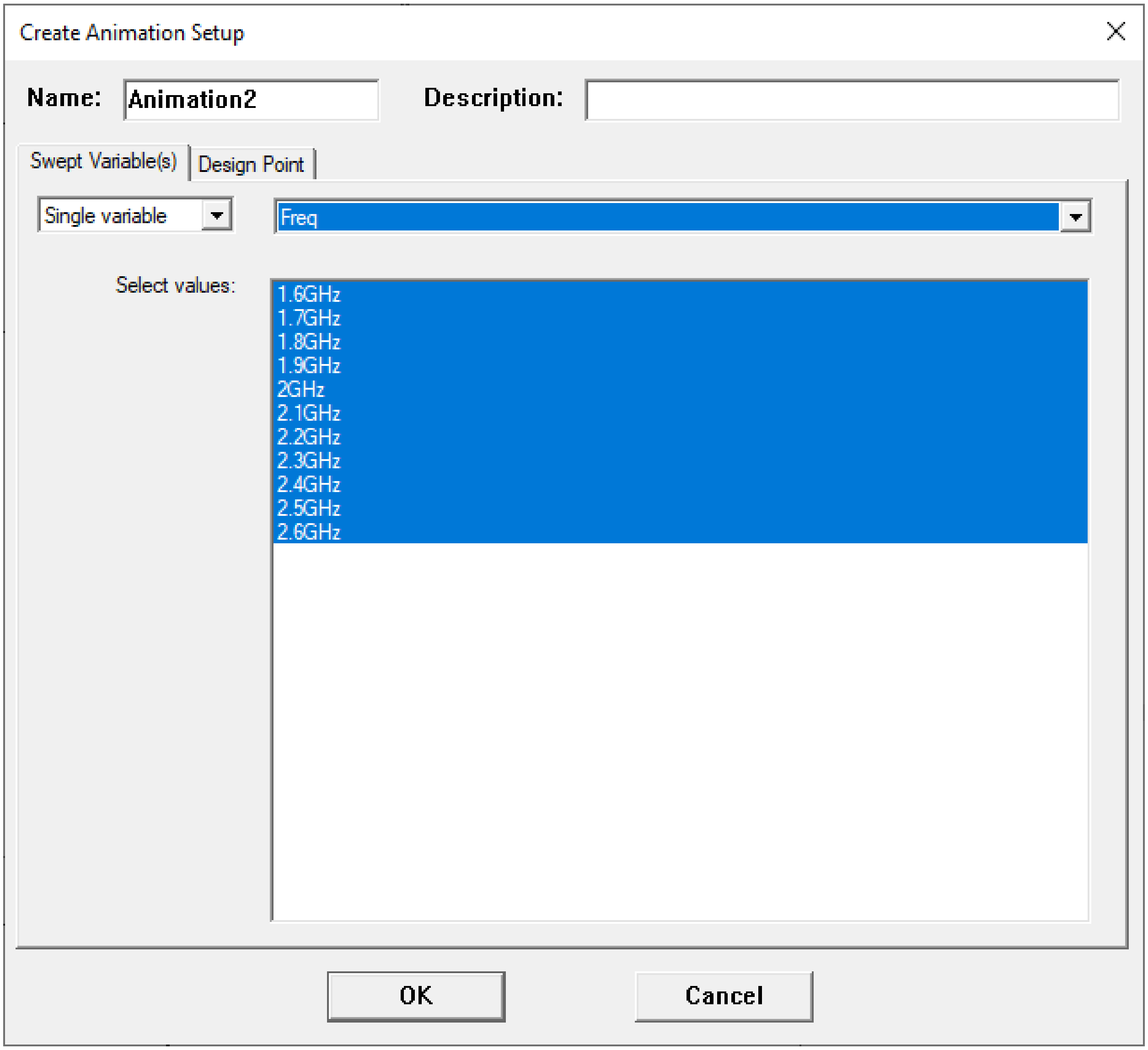
2. Right-click in the **Far Field Plot 1** and select **View** > **Animate** to open the **Select Animation** window.


Far Field Plot 1
Max: 0.11
0.12
0.10
0.08
0.06
0.04
0.02
0.00
Min: 0.00
Etotal (V_per_meter)
Theta (deg)
0
0.11
0.06
-120
120
0
120
0.06
Phi (deg)
View
Edit
Modify Report...
Update Report
Save As Template...
Export...
Copy Image
Undo View : Mouse Wheel
Redo View
Rotate
Pan Shift+Drag
Zoom Alt+Shift+Drag
Zoom Area
Zoom In Ctrl++
Zoom Out Ctrl+-
Zoom Previous
Fit All Ctrl+D
Fit Selection
Hide Selection Ctrl+H
Show Selection
Show Only Selection
Show All
Apply Orientation
Add New Orientation...
Animate...
Z
Y
X

3. Click **Edit** to open the **Create Animation Setup** window.



4. Ensure the settings in the **Create Animation Setup** window match the following example.



5. Click **OK** to close the **Create Animation Setup** window and start the animation in the **View** tab. Simultaneously, an animation control panel opens.
6. Use the animation controls to **Reverse**, **Stop**, and change the speed of the animation, as chosen.

   style="border:1px solid #000000"

7. If appropriate, **click+drag** the frequency legend to a more desirable location.
8. From the animation control panel, click **Close** to end the animation.
9. **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.