Getting Started with HFSS 3D Layout: Slot Fed Patch Antenna

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

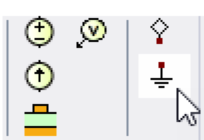
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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 (>) (e.g., "click **HFSS** > **Excitations** > **Assign** > **Wave Port**."
).
  - Labeled keys from the computer keyboard (e.g., "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 **Interface Ground** 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

From the user interface, press **F1** to open the help specific to the active product (design type).

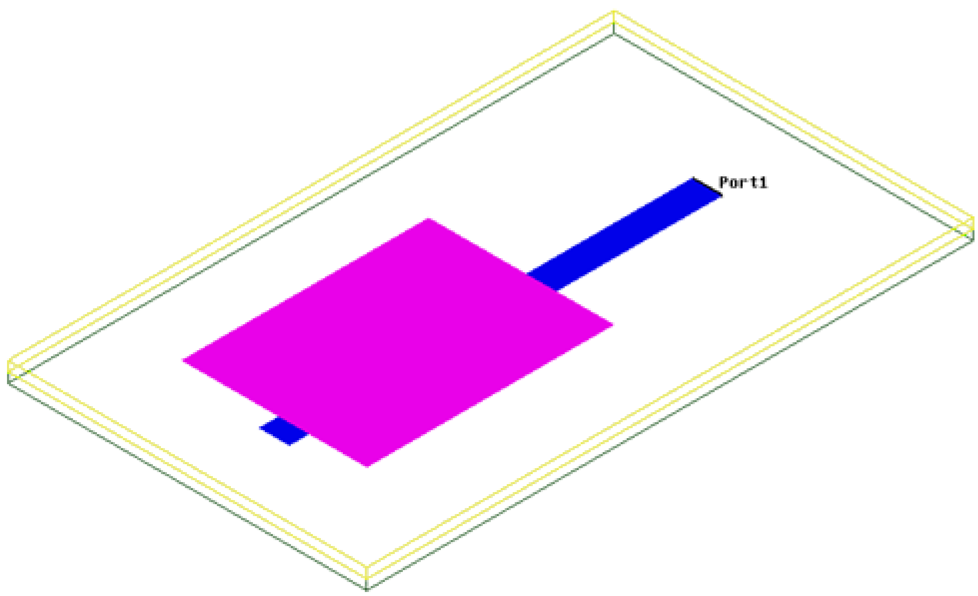
Press **F1** while the cursor is pointing at a menu command or while a particular window tab is open. In this case, the help page associated with the command or open window is displayed automatically.

Table of Contents

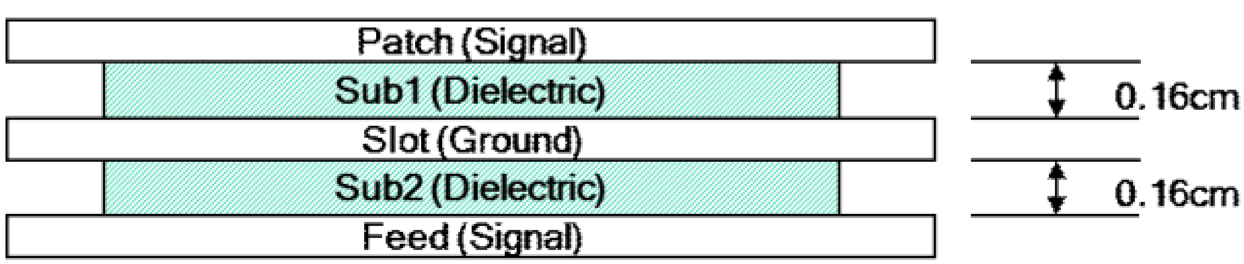
1 - Introduction

Complete the **Getting Started with HFSS 3D Layout: Slot Fed Patch Antenna** guide to create and define a multilayer structure (consisting of three copper and two dielectric layers), define its ports, then set up and analyze the design as a planar EM solution with a frequency sweep. The model pass filter model consists of three copper and two dielectric layers). The user will need to define the layers, draw the model, define the ports, and set up the solution. After solving the model, the user will create a return loss report and add a marker from the trace, view surface currents, and the radiation pattern (2D gain plot and 3D gain pattern).

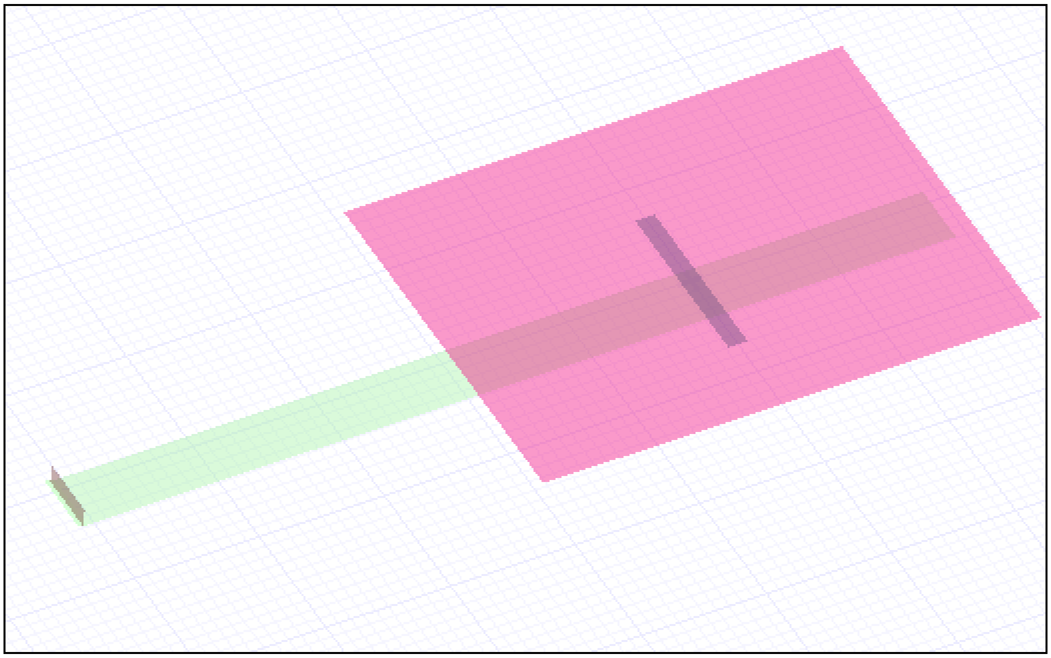
The following figure is an illustration of a typical model representing a slot coupled patch antenna.



The equivalent HFSS 3D layout of the model consists of the layers shown in the following cross-section:

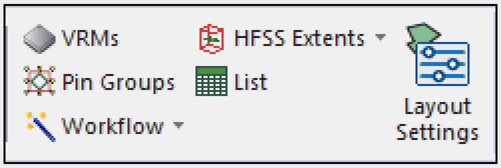


And finally, the following is a view of the HFSS 3D Layout model build:

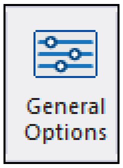


Set General Options

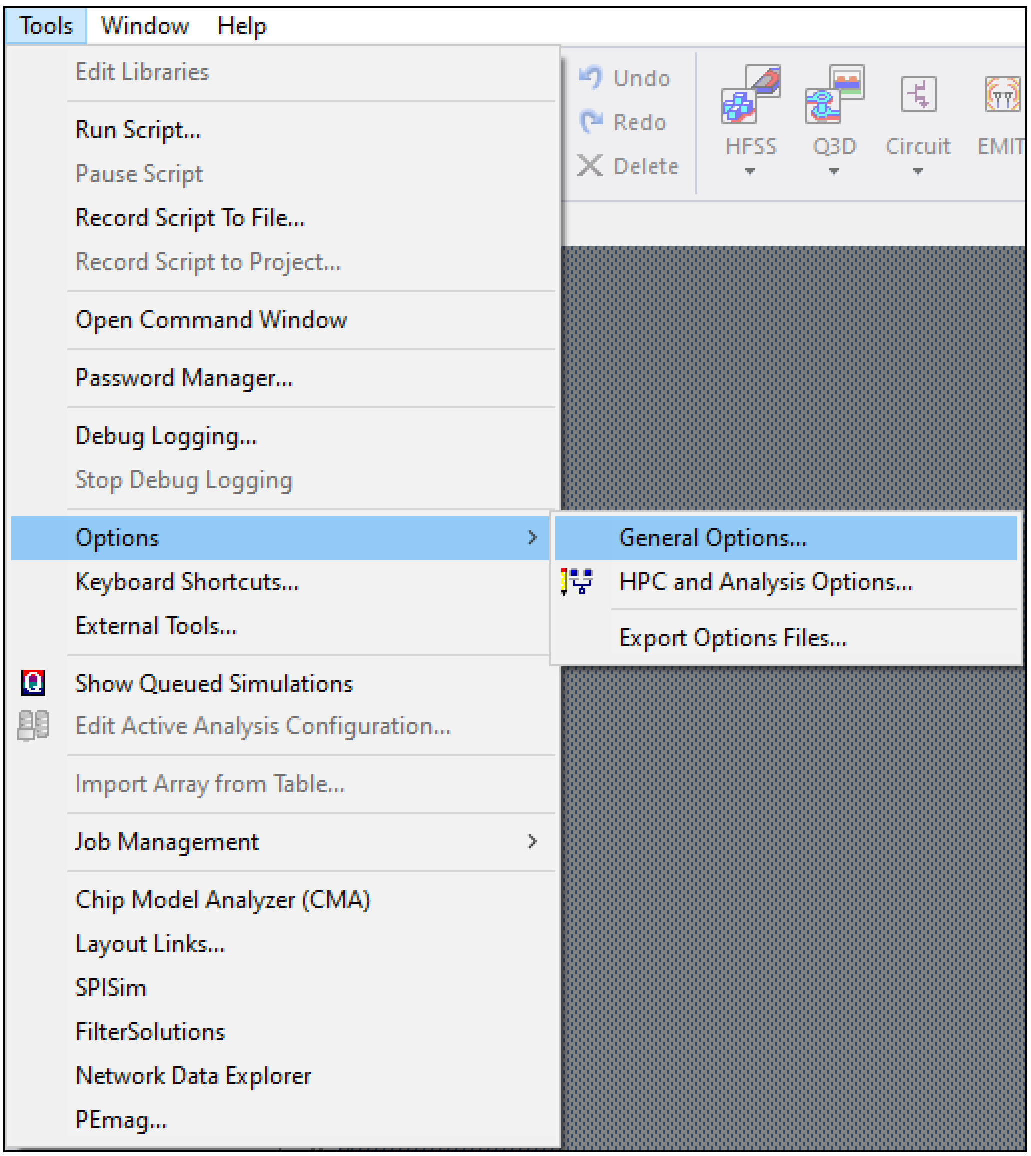
Before creating the bandstop filter, follow these steps to ensure the default unit of length measurement is set to **cm**.

> **Note:**
>
> Define general settings before adding an **HFSS 3D Layout Design** element to a project. The general options control the default settings when the design type is added. To change default settings after a design has been added to the project, from the **Layout** tab, select **Layout Settings**.
>
> 
>

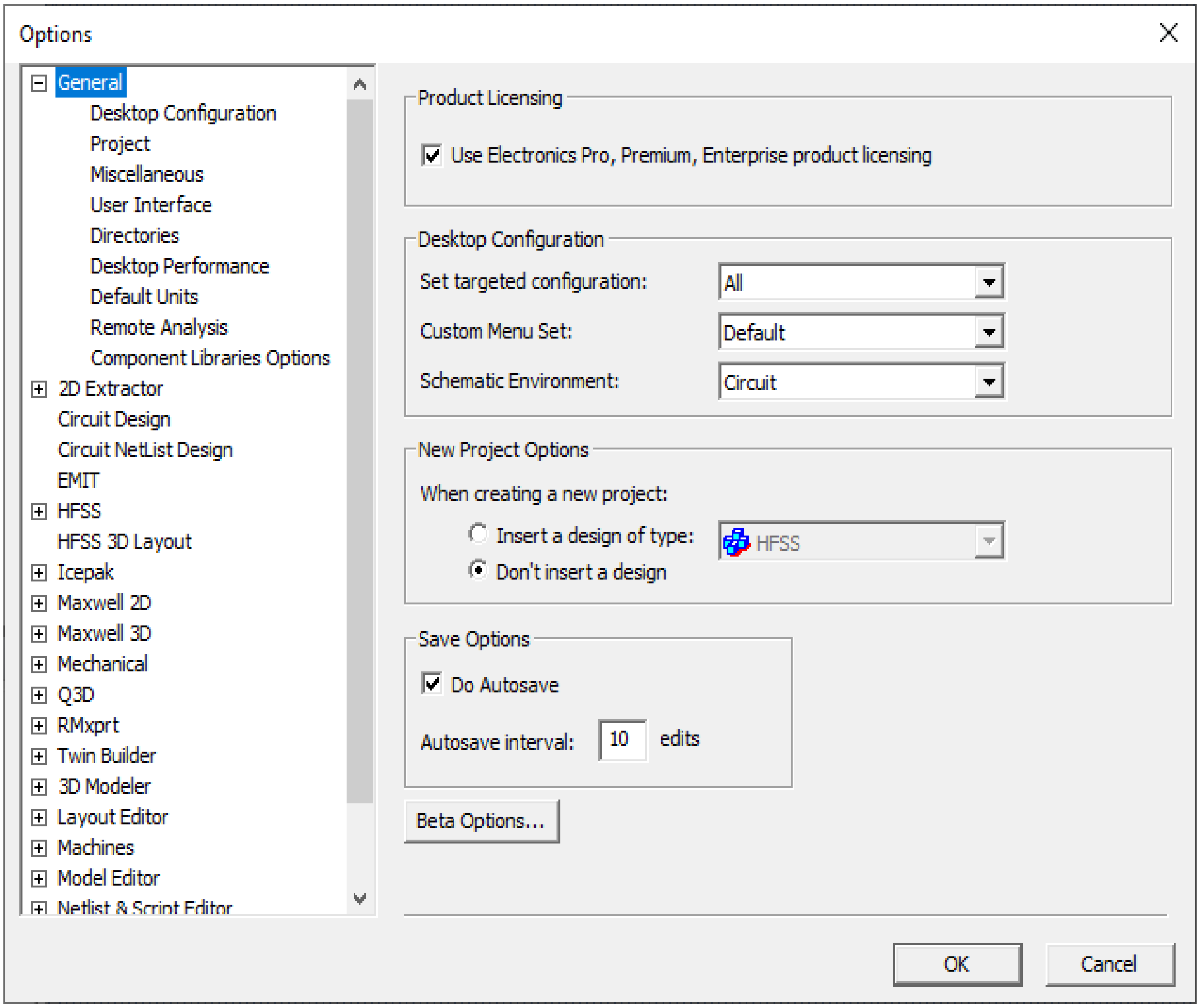
1. Open the **Options** window by doing one of the following:
   - From the **Desktop** ribbon, click **General Options**.



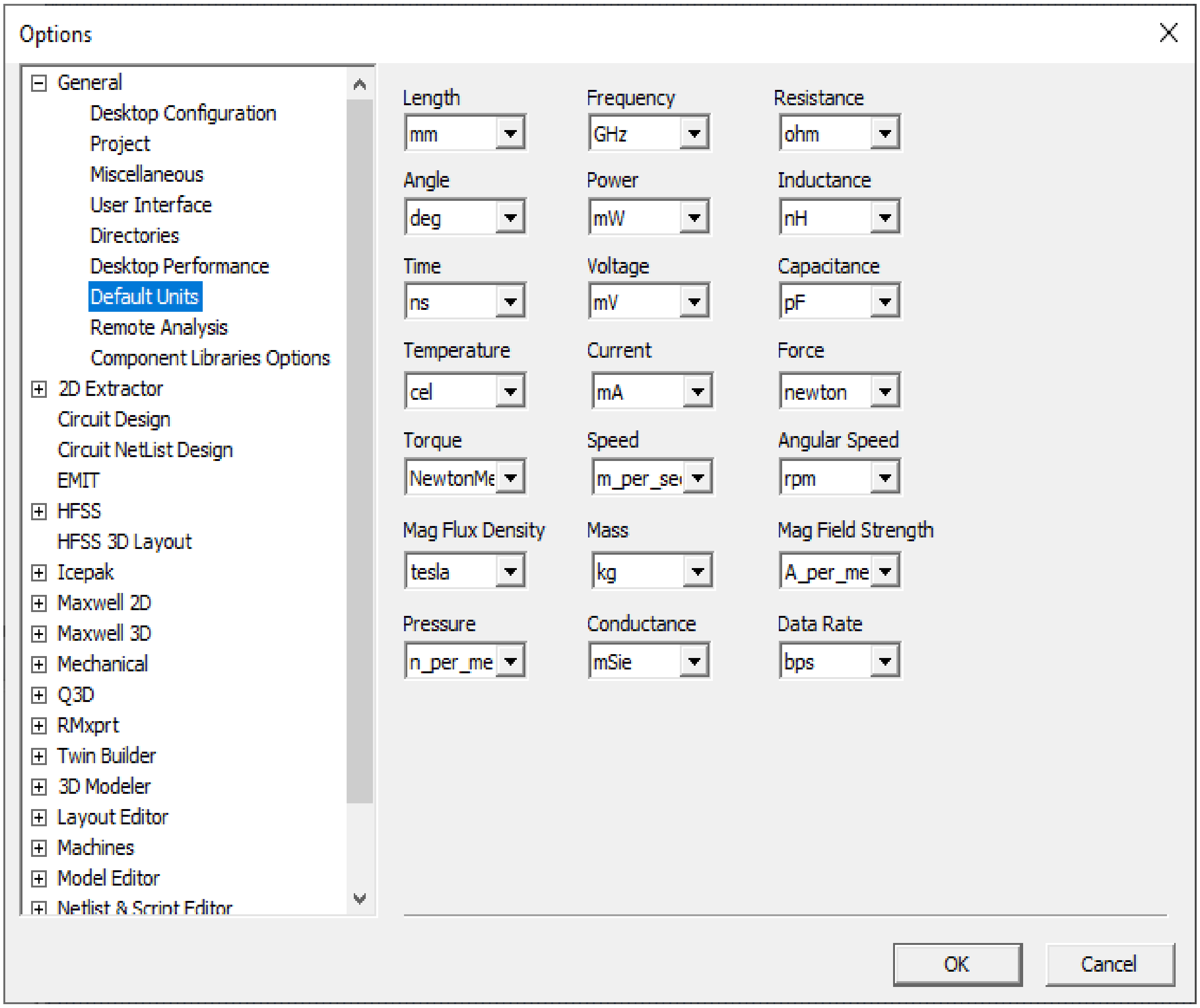
   - From **Tools**, select **Options** > **General Options**.



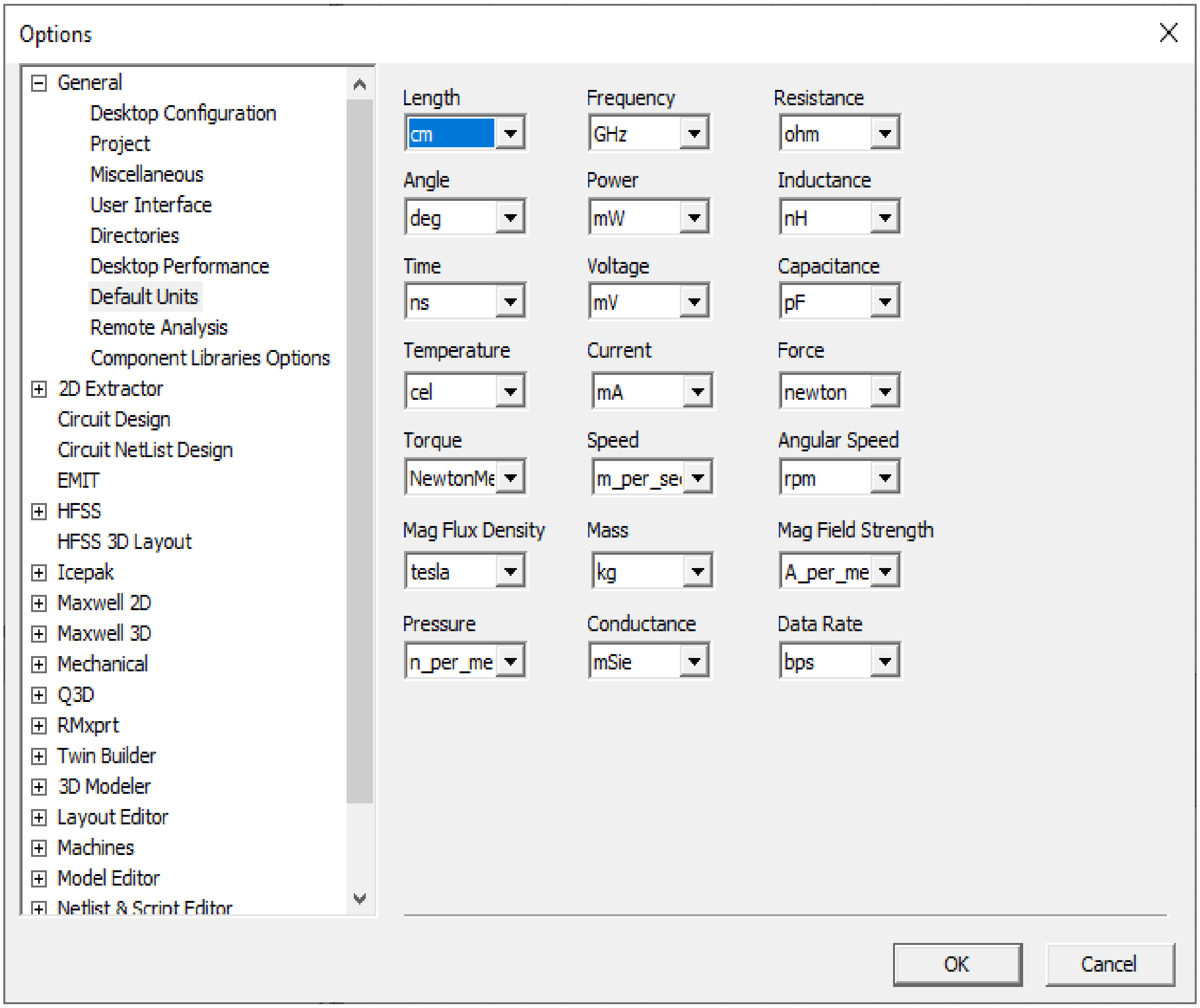
2. If appropriate, expand the **General** group.



3. Select **Default Units**.



4. Select **cm** from the **Length** drop-down menu.

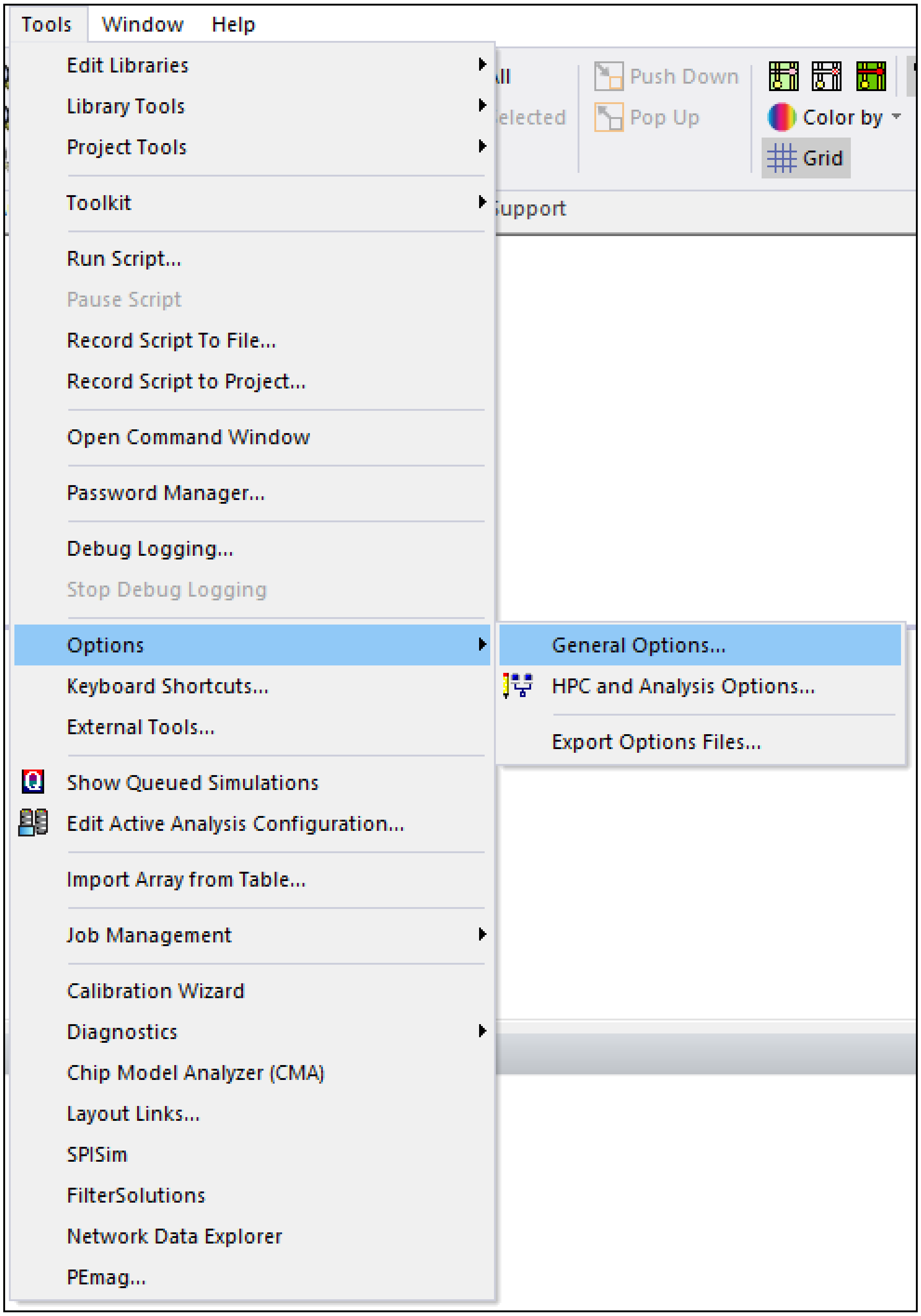


5. Click **OK** to close the **Options** window.

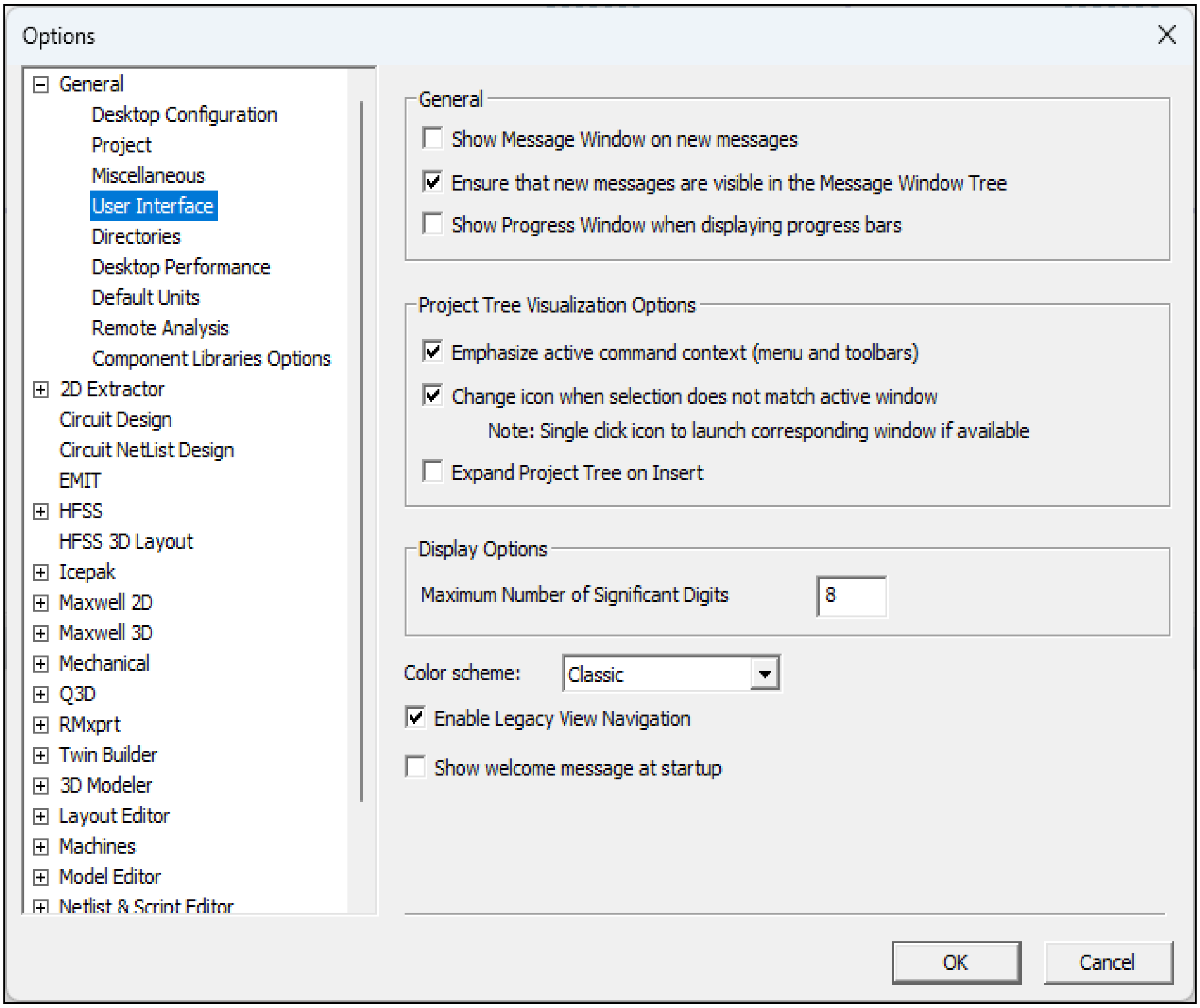
Enabling Legacy View Orientation

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.



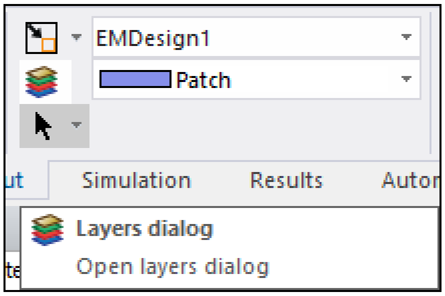
4. Click **OK**.

Continue to Insert Layers.

Inserting Layers

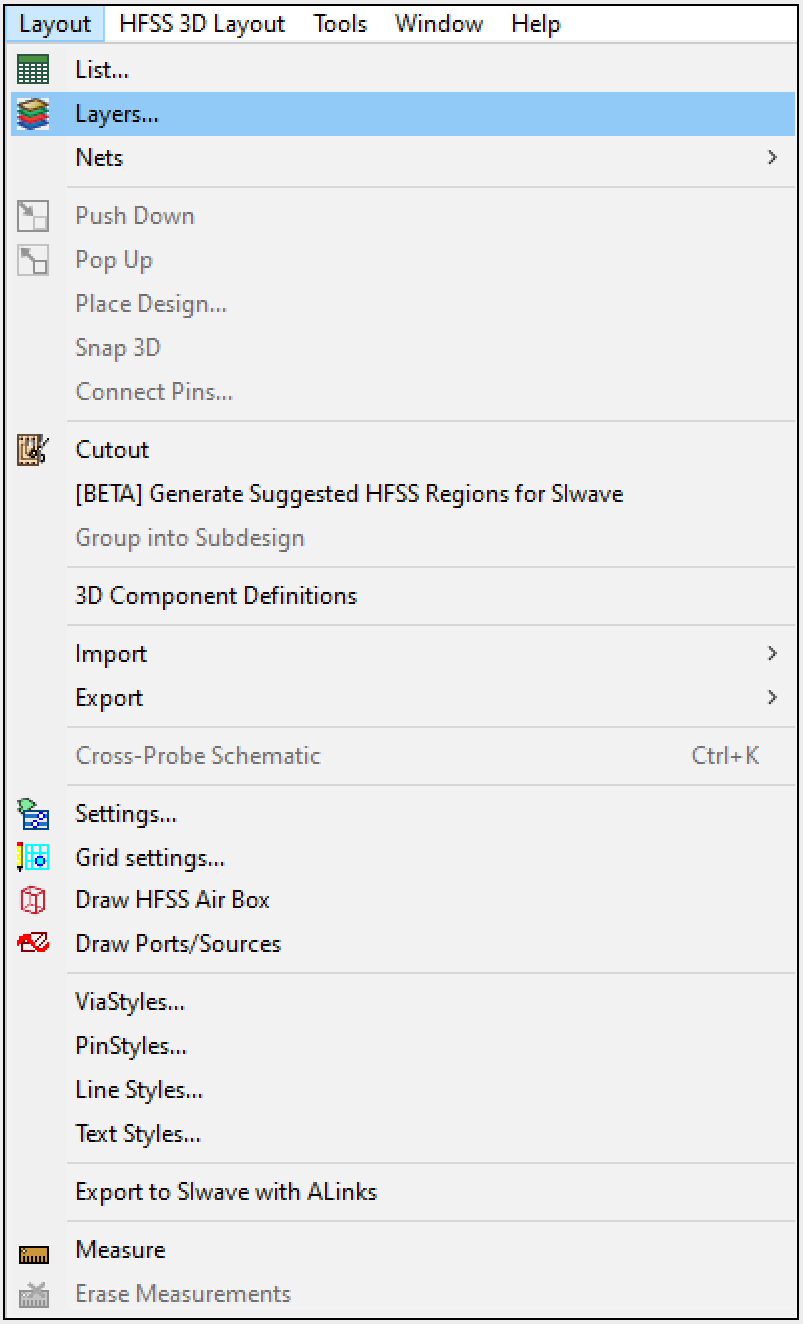
Complete these steps to insert layers in an HFSS 3D Layout design.

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

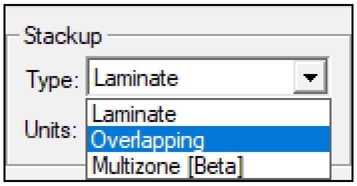


Note:

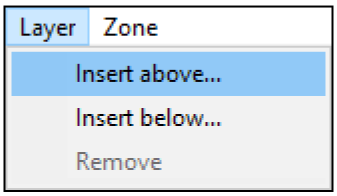
Alternatively, from **Layout**, select **Layers**.



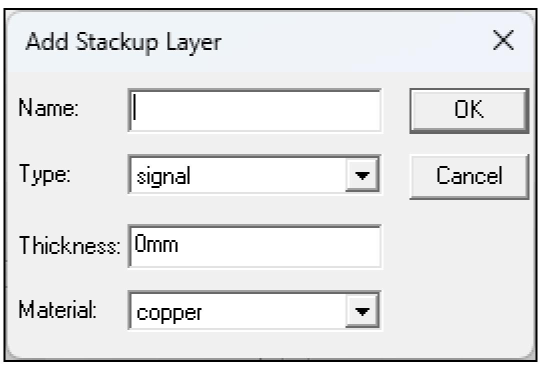
2. In the **Edit Layers** window > **Stackup** area, select **Overlapping**from the **Type** drop-down menu.



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



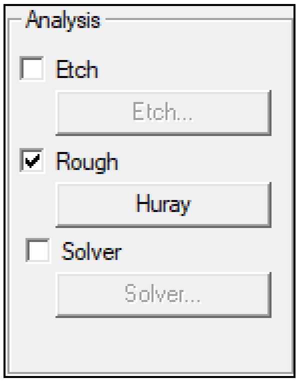
4. In the **Add Stackup Layer** window, do the following:



   a. Enter **Feed** in the **Name** field.
   b. Ensure **signal** is selected from the **Type** drop-down menu.
   c. Click **OK** to close the **Add Stackup Layer** window add the new layer to the table in the **Edit Layers** window.

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

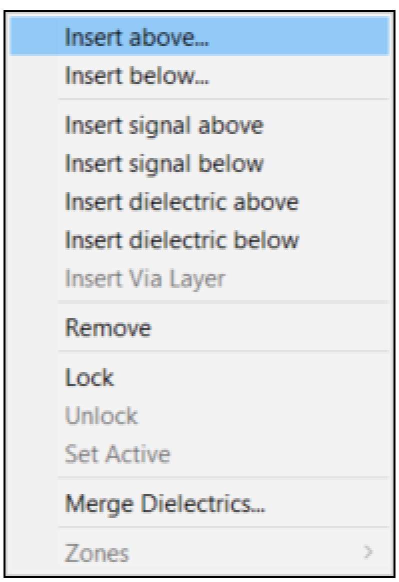
5. Check the **Rough** box in the **Analysis** area.



Note:

The **Rough** attribute indicates the surface roughness of the conductors is taken into account when approximating the impedance of the signal traces.

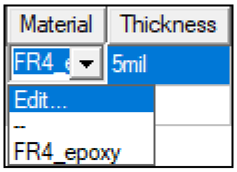
6. Right-click the **Feed** row and select **Insert above** to open a new **Add Stackup Layer** window.



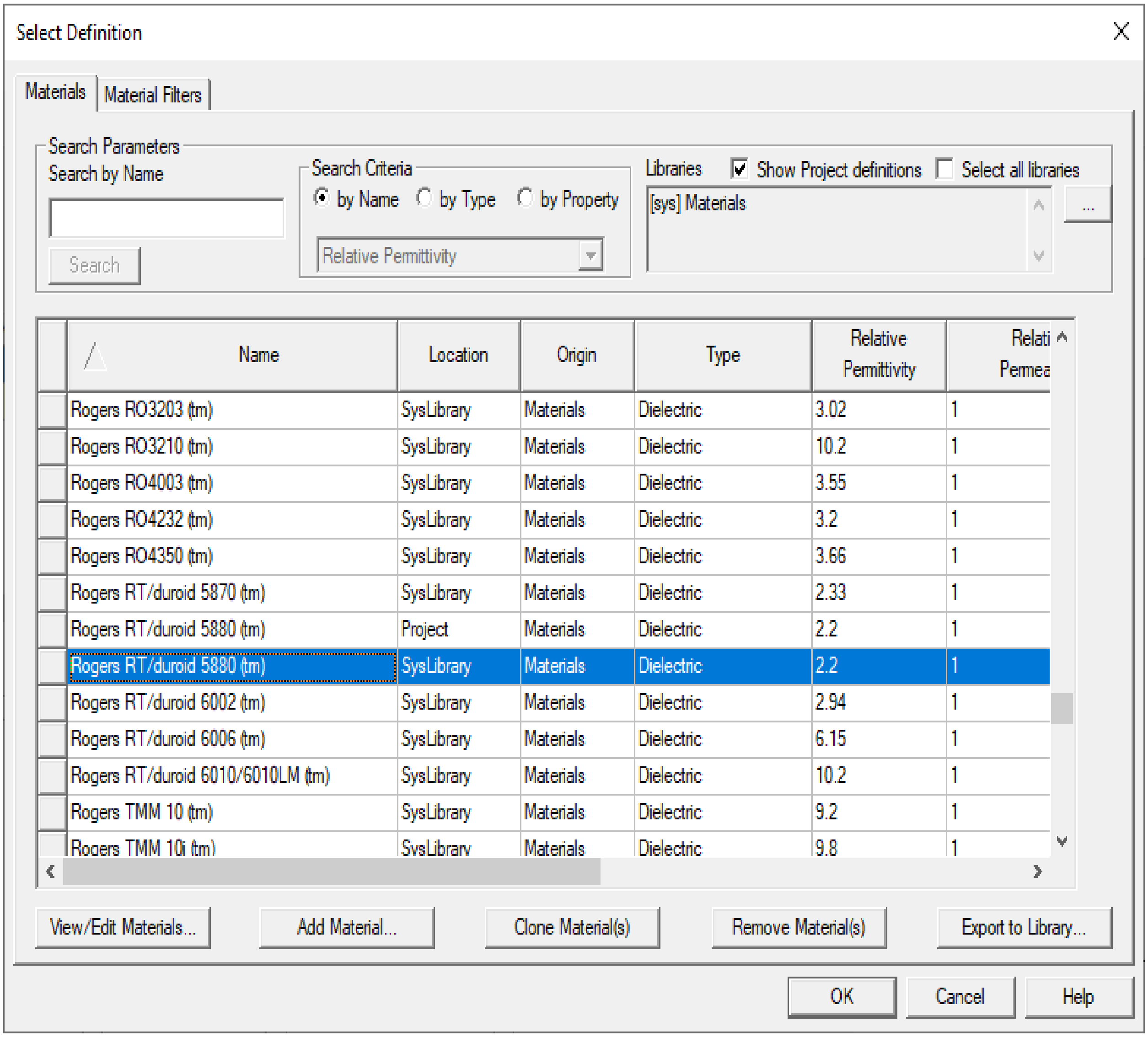
7. In the **Add Stackup Layer** window, do the following:
   a. Enter **Sub2** in the **Name** field.
   b. Select **dielectric** from the **Type** drop-down menu.
   c. Enter **0.16cm** in the **Thickness** field.

   d. Click **OK** to close the **Add Stackup Layer** window add the new layer to the Grid Control Table.

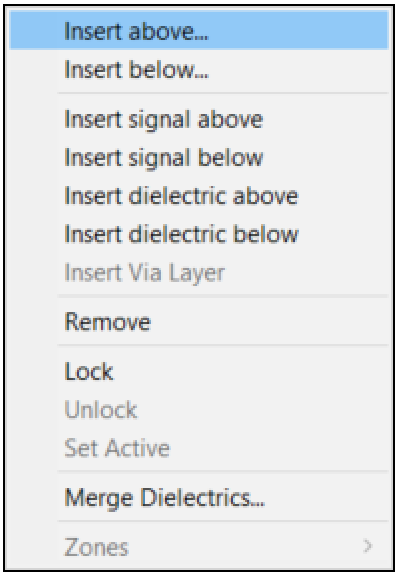
8. Select **Edit** from the **Material** drop-down menu to open the **Select Definition** window.



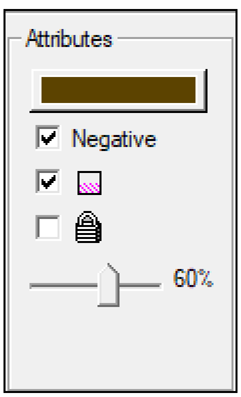
9. From the **Select Definition** window, select **Rogers RT/duroid 5880 (tm)** from the list of library materials. Then click **OK** to close the **Select Definition** window.



10. Right-click the **Sub2** row and select **Insert above** to open a new **Add Stackup Layer** window.

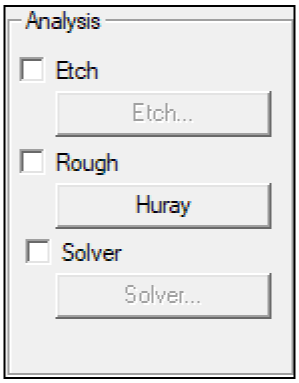


11. In the **Add Stackup Layer** window, do the following:
    a. Enter **Slot** in the **Name** field.
    b. Select **signal** from the **Type** drop-down menu.
    c. Click **OK** to close the **Add Stackup Layer** window add the new signal layer to the table.
12. Check the **Negative** box in the **Attributes** area.



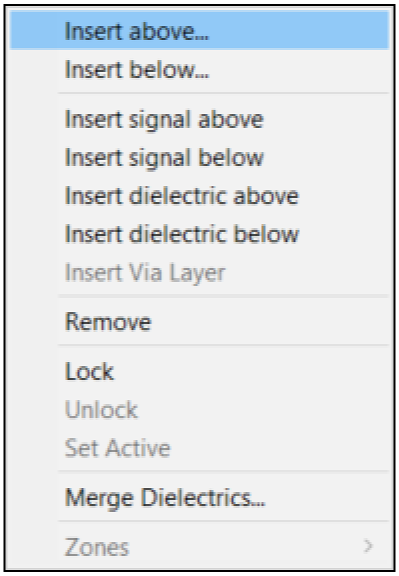
> **Note:**
>
> The **Negative** attribute indicates the conductor is present everywhere except where an object is drawn. The layer is an infinite ground layer, and the conductor is removed in the area of a drawn shape.

13. Ensure the **Rough** box in the **Analysis** area is **not** checked.

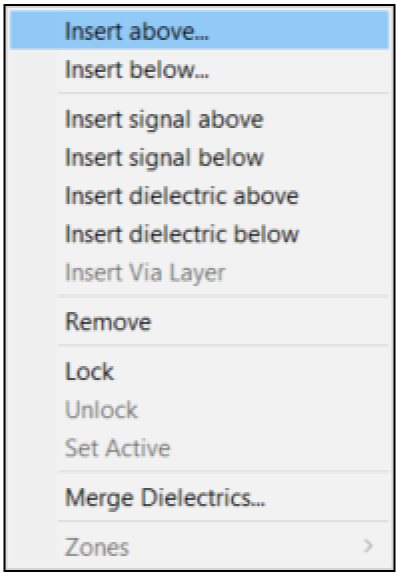


> **Note:**
>
> The **Rough** attribute indicates the surface roughness of the conductors is taken into account when approximating the impedance of the signal traces, but surface roughness is ignored for ground layers. Deactivating the **Rough** attribute prevents a warning to that effect from appearing during model validation and solution setup.

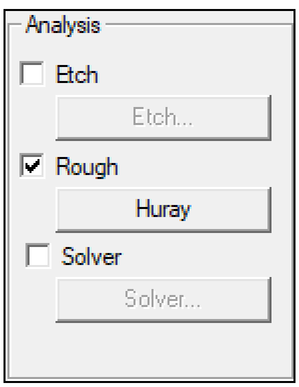
14. Right-click the **Slot** row and select **Insert above** to open a new **Add Stackup Layer** window.



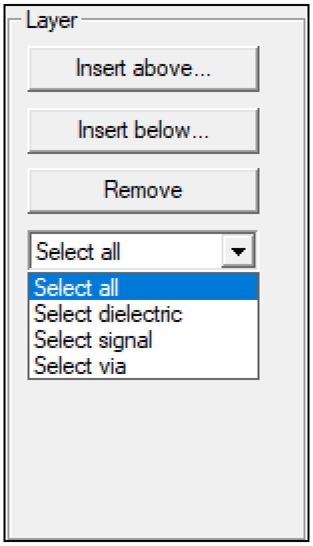
15. In the **Add Stackup Layer** window, do the following:
    a. Enter **Sub1** in the **Name** field.
    b. Select **dielectric** from the **Type** drop-down menu.
    c. Enter **0.16cm** in the **Thickness** field.
    d. Select **Rogers RT/duroid 5880 (tm)** from the **Material** drop-down menu.
    e. Click **OK** to close the **Add Stackup Layer** window add the new dielectric layer to the table.
16. Right-click the **Sub1** row and select **Insert above** to open a new **Add Stackup Layer** window.



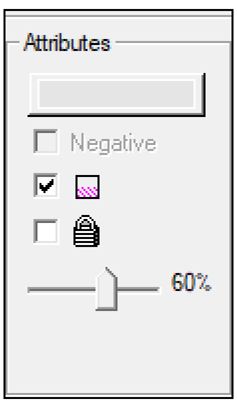
17. In the **Add Stackup Layer** window, do the following:
    a. Enter **Patch** in the **Name** field.
    b. Select **signal** from the **Type** drop-down menu.
    c. Click **OK** to close the **Add Stackup Layer** window add the new dielectric layer to the table.
18. Check the **Rough** box in the **Analysis** area.



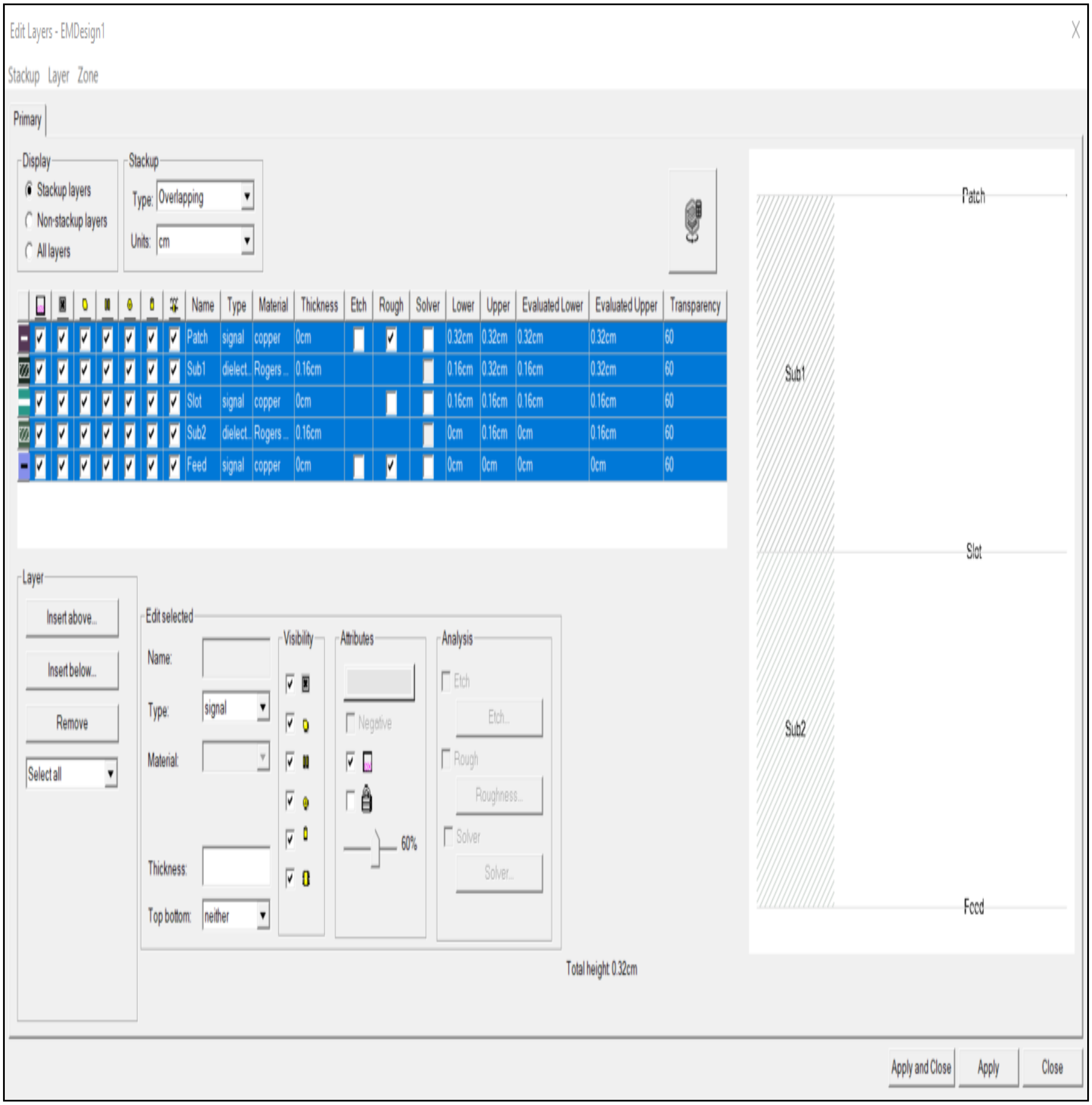
19. From the **Layer** area, choose **Select all** from the drop-down menu.



20. 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).



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



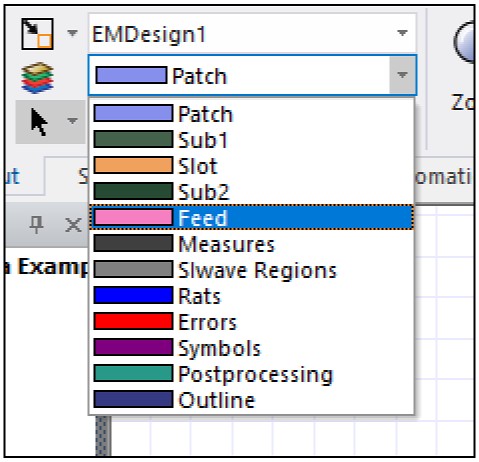
22. Click **Apply and Close**.

Continue to Draw Feed Line.

Drawing a Feed Line

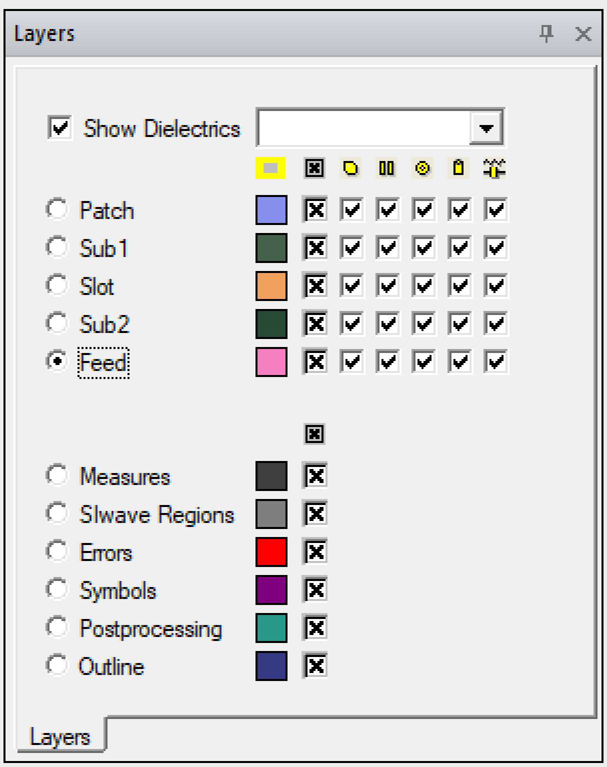
Complete these steps to set the chosen working layer and draw the first object (i.e., a feed line).

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

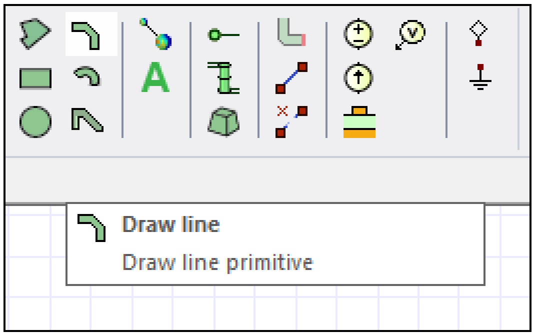


Note:

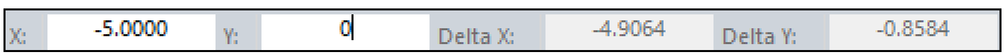
Alternatively, select **Feed** in the **Layers** window.



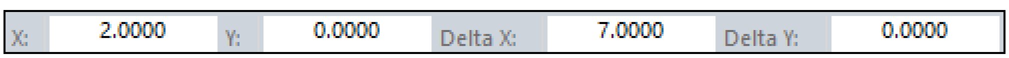
2. From the **Layout** tab, click **Draw line**.



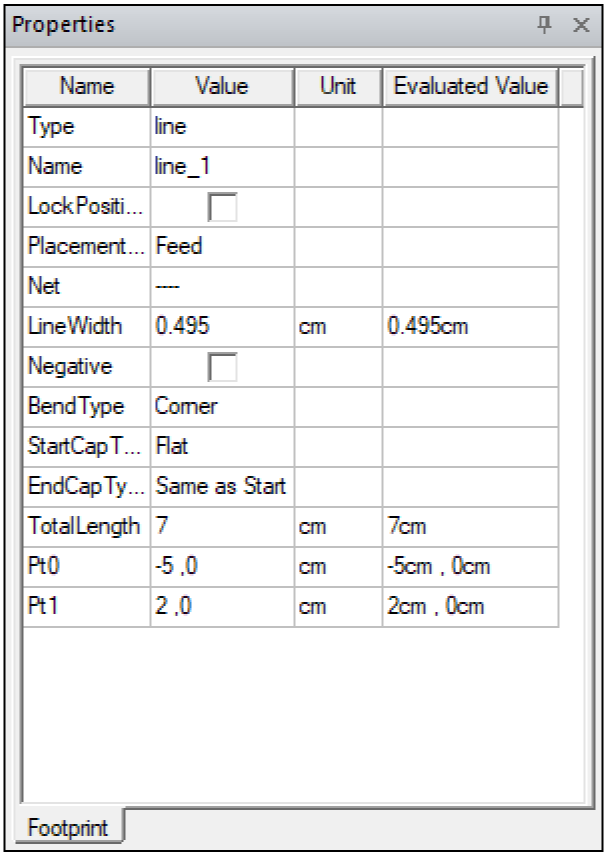
3. Do not **click+drag** in the **Layout Editor**. Instead, move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Type **-5** in the field.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.



5. Press **Tab** until the cursor moves to the **Delta X** coordinate field. Type **7** in the field.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **0** in the field and press **Enter** to complete the ground plane.



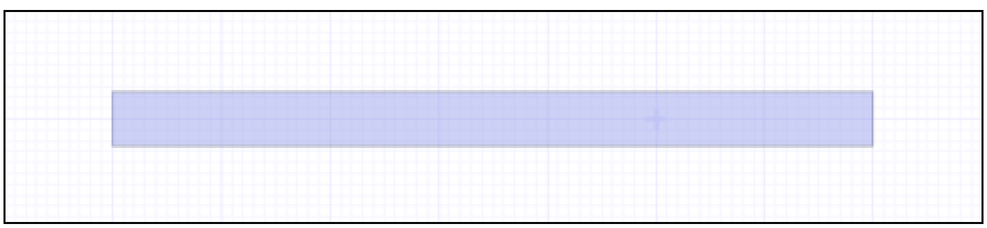
7. Press **Enter** again to finish the line.
8. Select the new object to populate the **Properties** window. Then make the following changes:
   a. Enter **0.495** in the **LineWidth** row **Value** field.
   b. Select **cm** from the **Unit** drop-down menu in the **LineWidth** row.
   c. Ensure **Flat** is selected from the **StartCapType** drop-down menu.



Properties

Name	Value	Unit	Evaluated Value
Type	line		
Name	line_1		
LockPositi...			
Placement...	Feed		
Net	----		
LineWidth	0.495	cm	0.495cm
Negative			
BendType	Corner		
StartCapT...	Flat		
EndCapTy...	Same as Start		
TotalLength	7	cm	7cm
Pt0	-5 ,0	cm	-5cm , 0cm
Pt1	2 ,0	cm	2cm , 0cm

Footprint

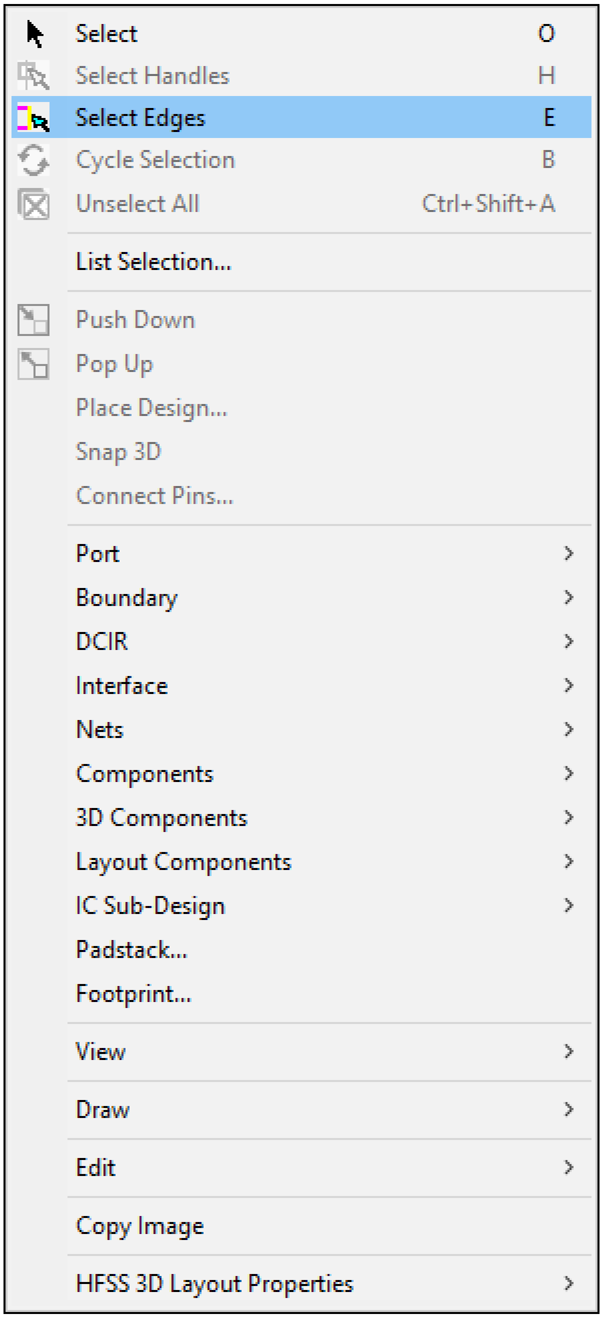


Continue to [Add Excitation (Port)](#).

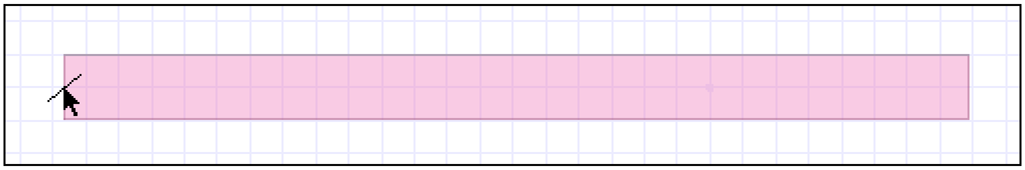
Adding an Excitation (Port)

Complete these steps to create an edge port.

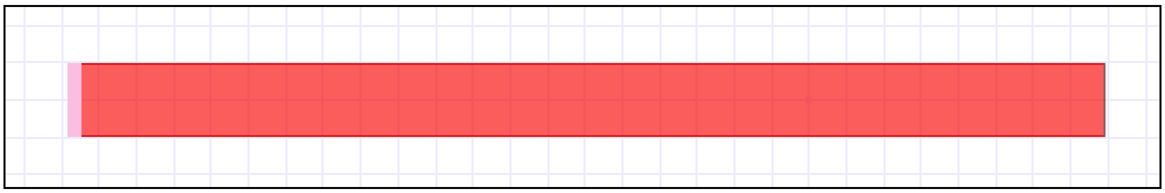
1. Right-click in the **Layout Editor** and choose **Select Edges**.



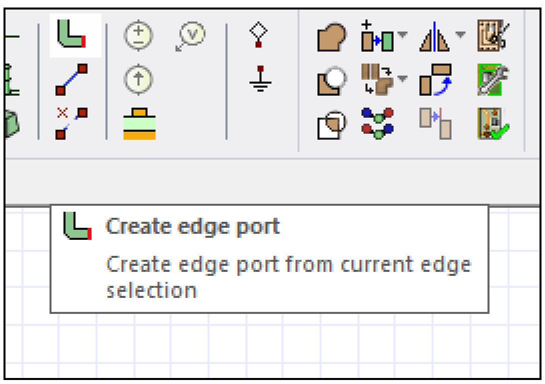
The cursor appearance changes to indicate it is in edge-selection mode.



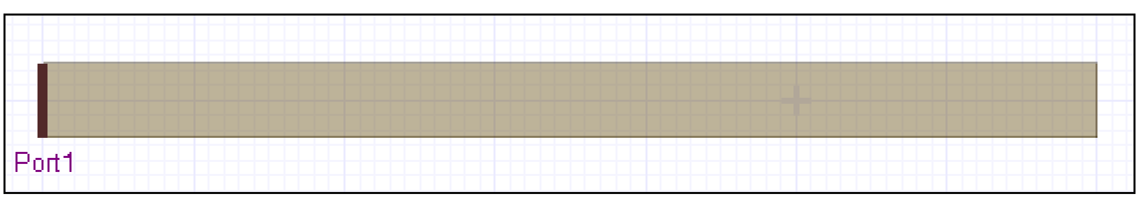
2. Click the left edge of the model to select it.



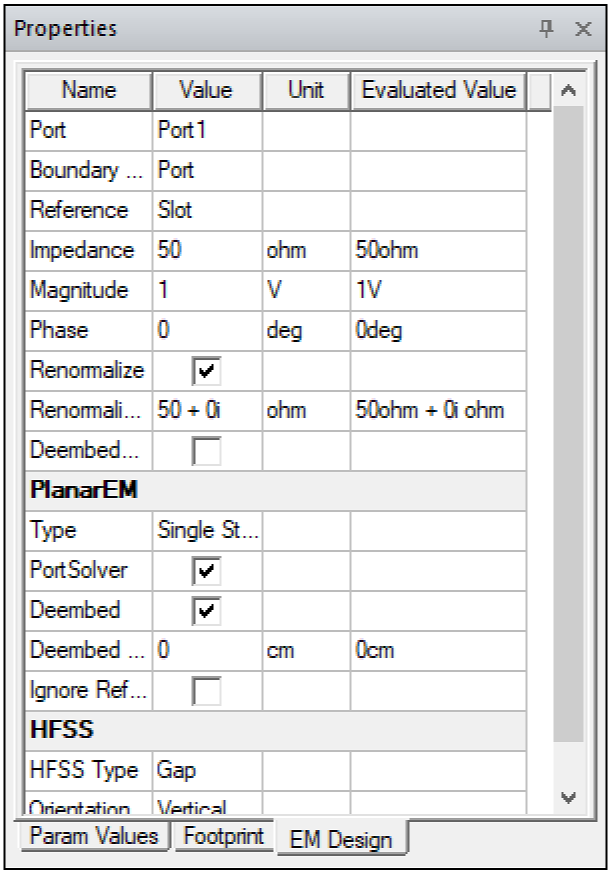
3. From the **Layout** ribbon tab, click **Create edge port**.



The port is added to the drawing in the **Layout Editor** (default name, *Port*n) and in the **Project Manager** window (i.e., from the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Excitations**). Do **not** deselect the port.



4. From the **Properties** window, select the **EM Design** tab.



Note:

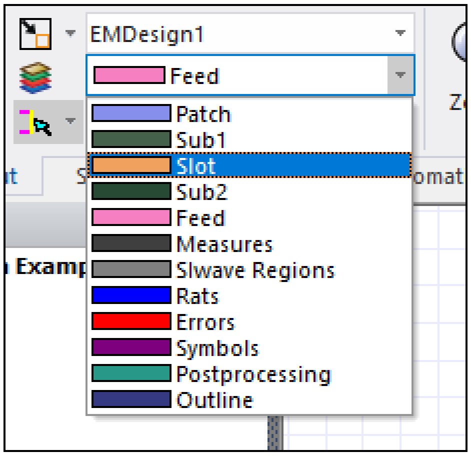
There are PlanarEM-, HFSS-, and HFSS-PI-specific port properties. For an EM Design, run either a Planar EM simulation or an HFSS simulation. Specific properties can be assigned to the port depending on the solver used (i.e., select **Gap**, **Wave**, or **Circuit** port for an HFSS simulation.)

Continue to Draw Slot.

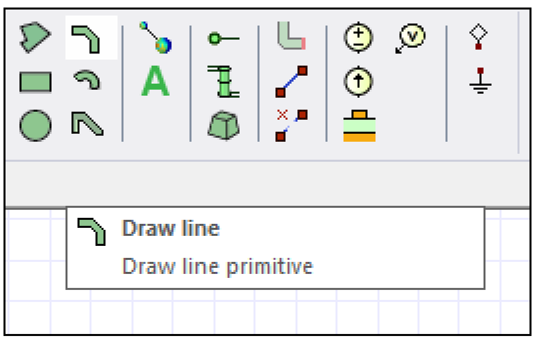
Creating a Draw Slot

Complete these steps to create the second object (i.e., a draw slot).

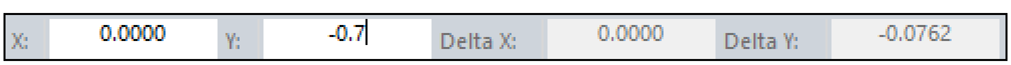
1. From the **Layout** tab, select **Slot** from the **Active Layer** drop-down menu.



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

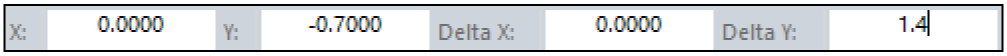


3. Press **Tab** to move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Enter **0** in the field.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **-0.7** in the field and press **Enter**.

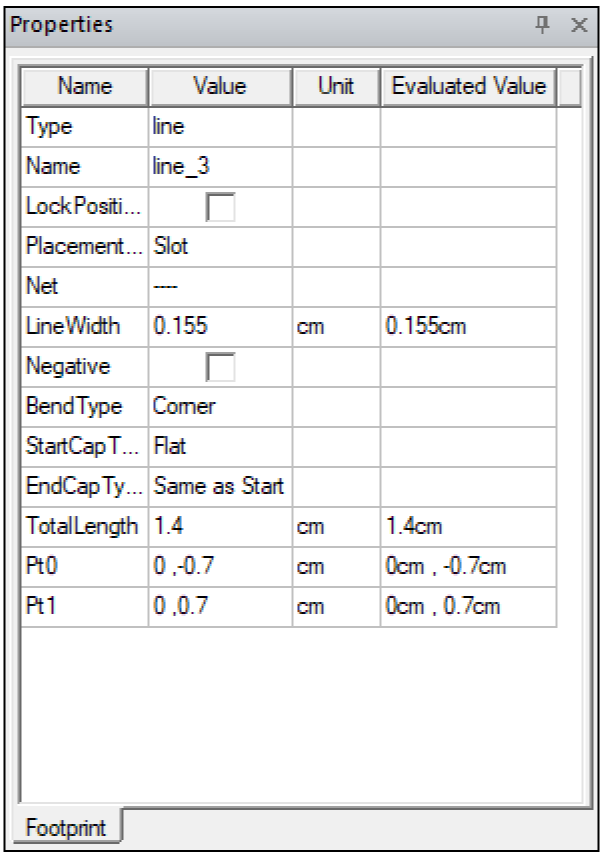


5. Either press **Tab** until the cursor moves to the **Delta X** coordinate field or move the cursor to the field, click inside it, and enter **0**.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field, Then type **1.4** in the field and

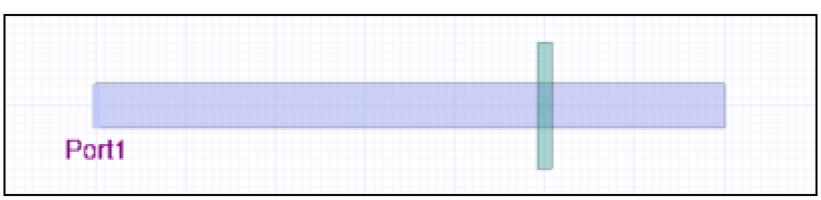
press **Enter** to complete the ground plane.



7. Press **Enter** again to finish the line.
8. Select the new object to populate the **Properties** window. Then make the following changes:
   a. Enter **0.155** in the **LineWidth** row **Value** field.
   b. Select **cm** from the **Unit** drop-down menu in the **LineWidth** row.
   c. Ensure **Flat** is selected from the **StartCapType** drop-down menu.



Name	Value	Unit	Evaluated Value
Type	line		
Name	line_3		
LockPositi...			
Placement...	Slot		
Net	----		
LineWidth	0.155	cm	0.155cm
Negative			
BendType	Corner		
StartCapT...	Flat		
EndCapTy...	Same as Start		
TotalLength	1.4	cm	1.4cm
Pt0	0 ,-0.7	cm	0cm , -0.7cm
Pt1	0 ,0.7	cm	0cm , 0.7cm

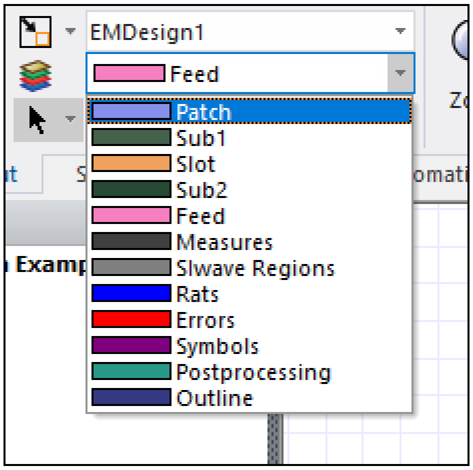


Continue to Draw Patch.

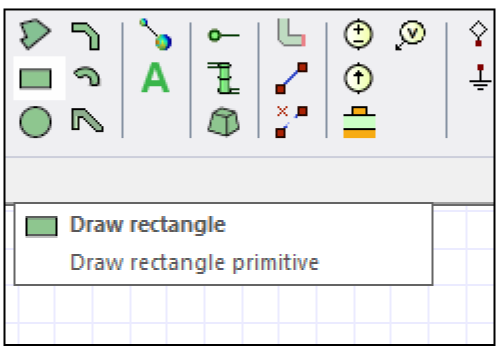
Creating a Draw Patch

Complete these steps to create the third object (i.e., a draw patch).

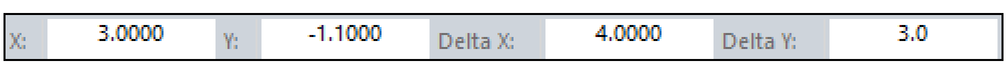
1. From the **Layout** tab, select **Patch** from the **Active Layer** drop-down menu.



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

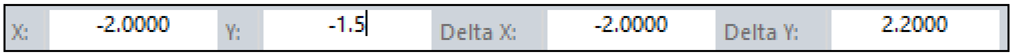


3. Press **Tab** to move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Enter **-2.0** in the field.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **-1.5** in the field and press **Enter**.

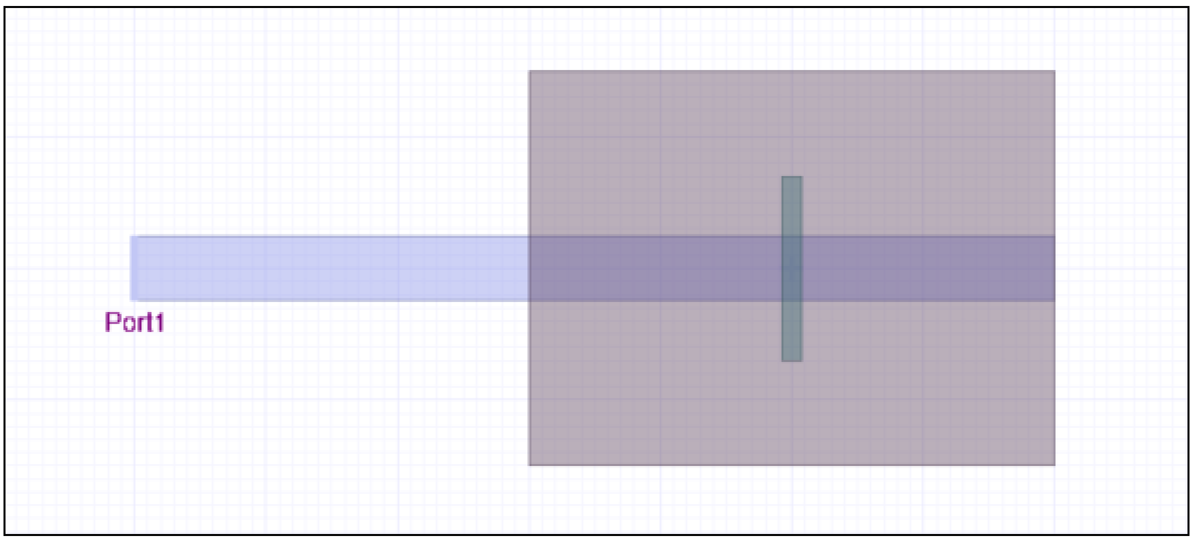


5. Either press **Tab** until the cursor moves to the **Delta X** coordinate field or move the cursor to the field, click inside it, and enter **4.0**.

6. Press **Tab** to move the cursor to the **Delta Y** coordinate field, Then type **3.0** in the field and press **Enter** to complete the ground plane.



7. Press **Enter** again to finish the line.



Continue to Set Up Solution and Analyze.

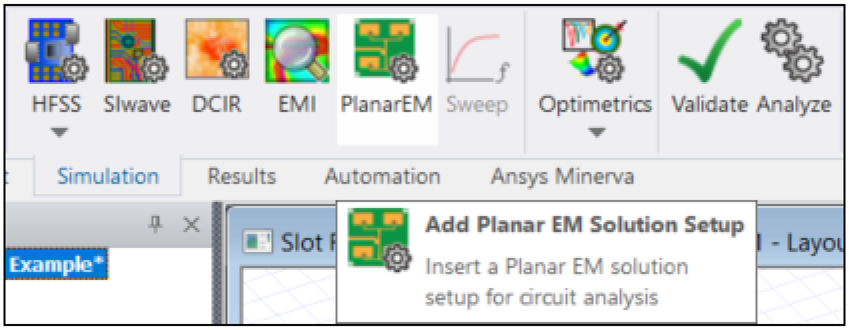
2 - Set Up Solution and Analyze

This chapter contains the following topics:

Setting Up and Solving a Planar EM Analysis

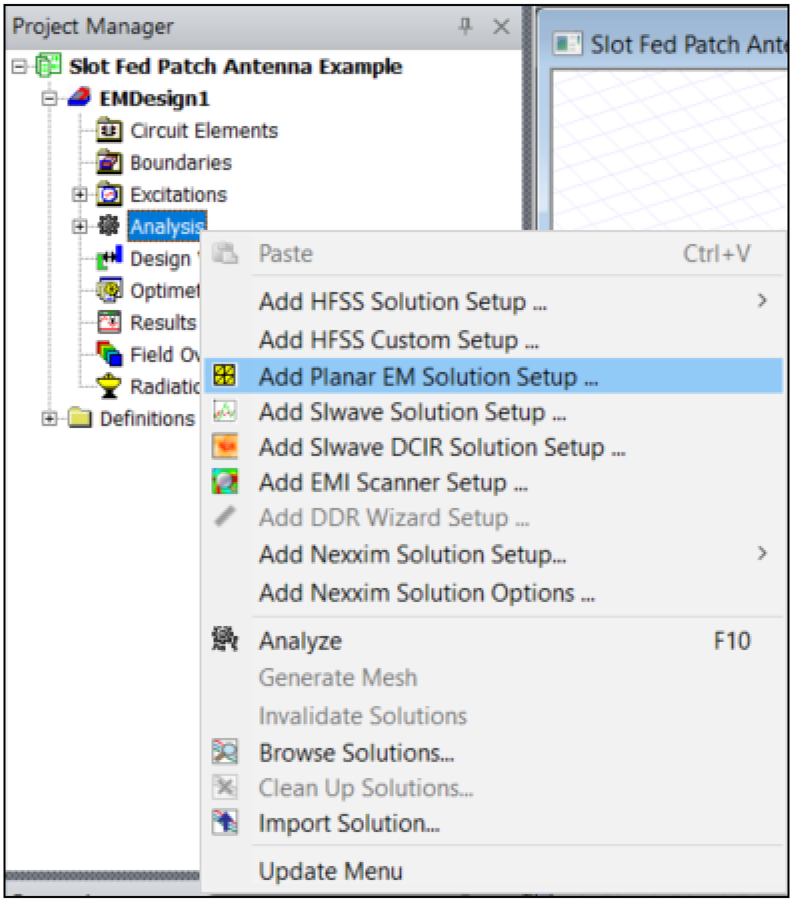
Complete these steps to add a Planar EM analysis to the project, then validate and solve it.

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

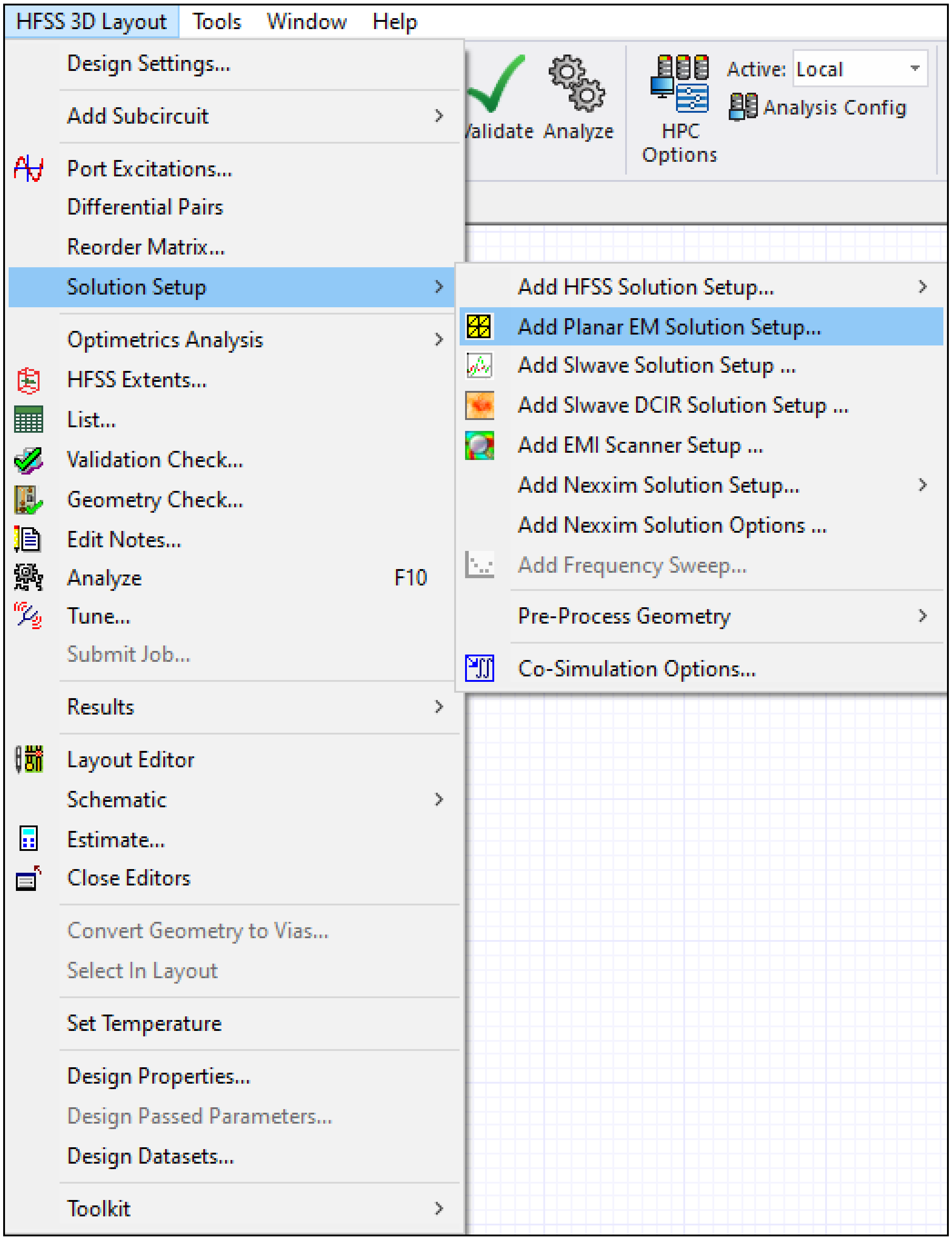


    - Right-click **Analysis** in the **Project Manager** window and click **Add Planar EM**

Solution Setup.

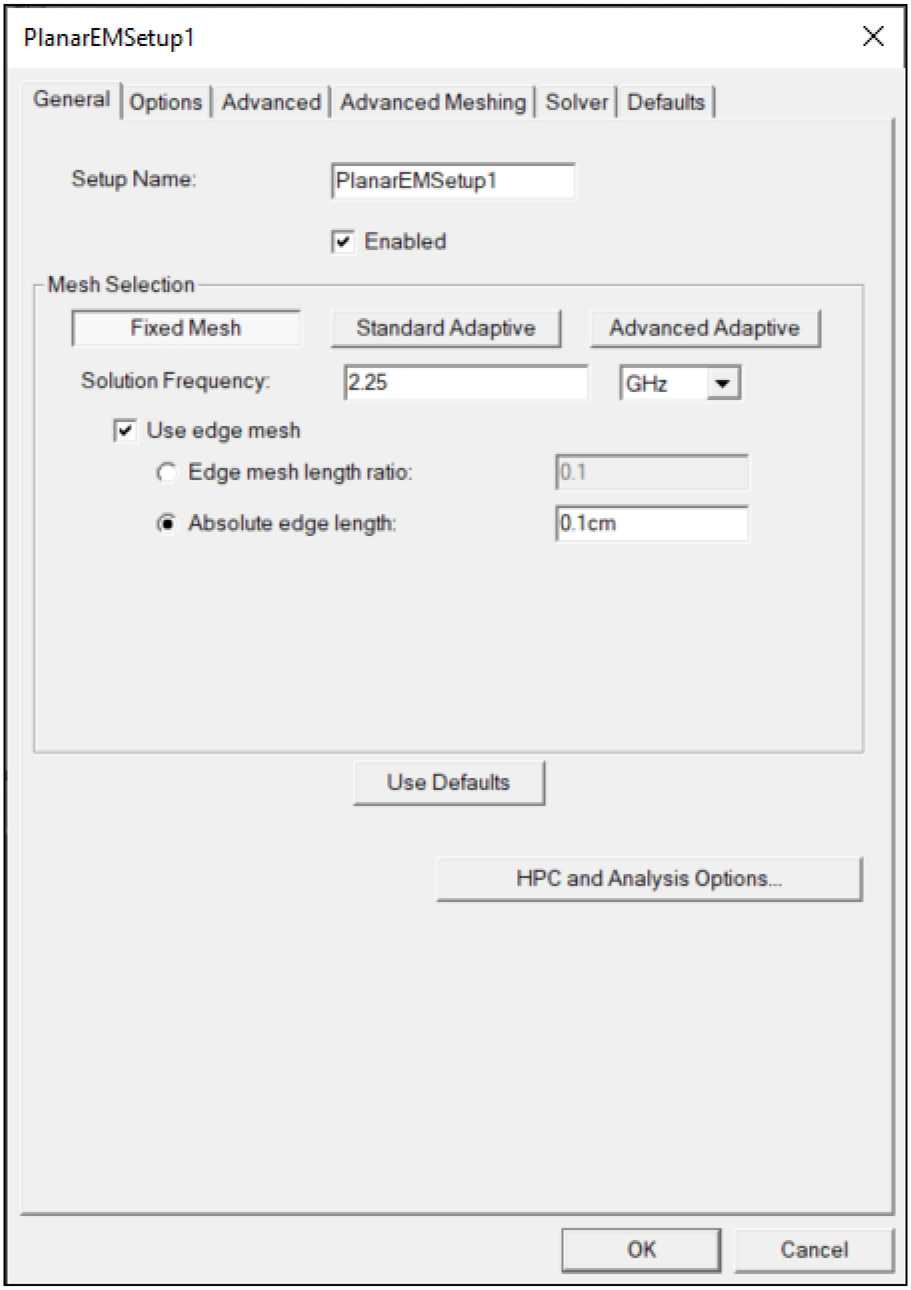


- From **HFSS 3D Layout**, select **Solution Setup** > **Add Planar EM Solution Setup**.

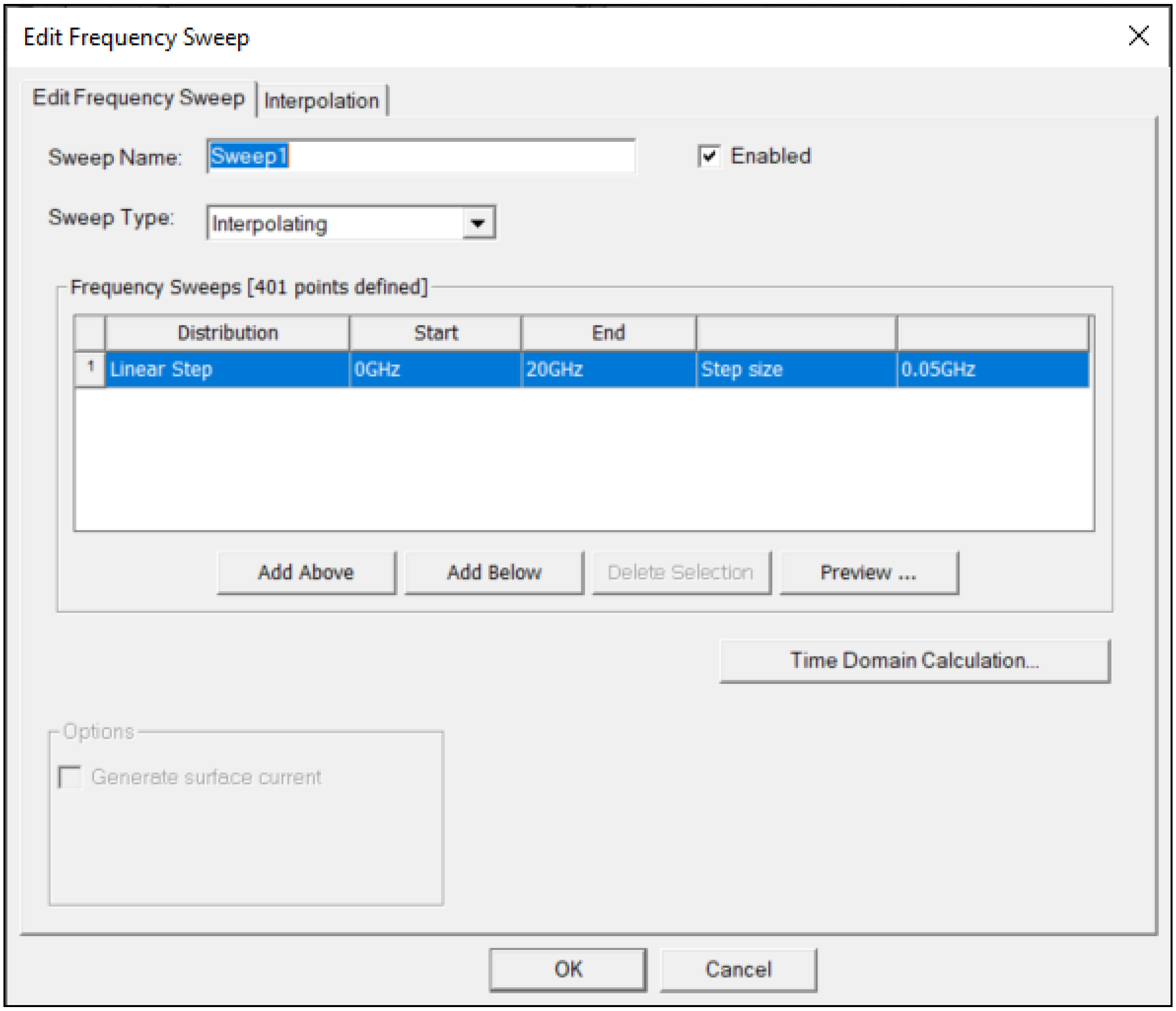


2. From the **PlanarEMSetup** window > **Mesh Selection** area, do the following:
   - Ensure **Fixed Mesh** is selected.
   - Enter **2.5** in the **Solution Frequency** field.

- Check the **Use edge mesh** box.
- Select **Absolute edge length** and enter **0.1cm** in the field.

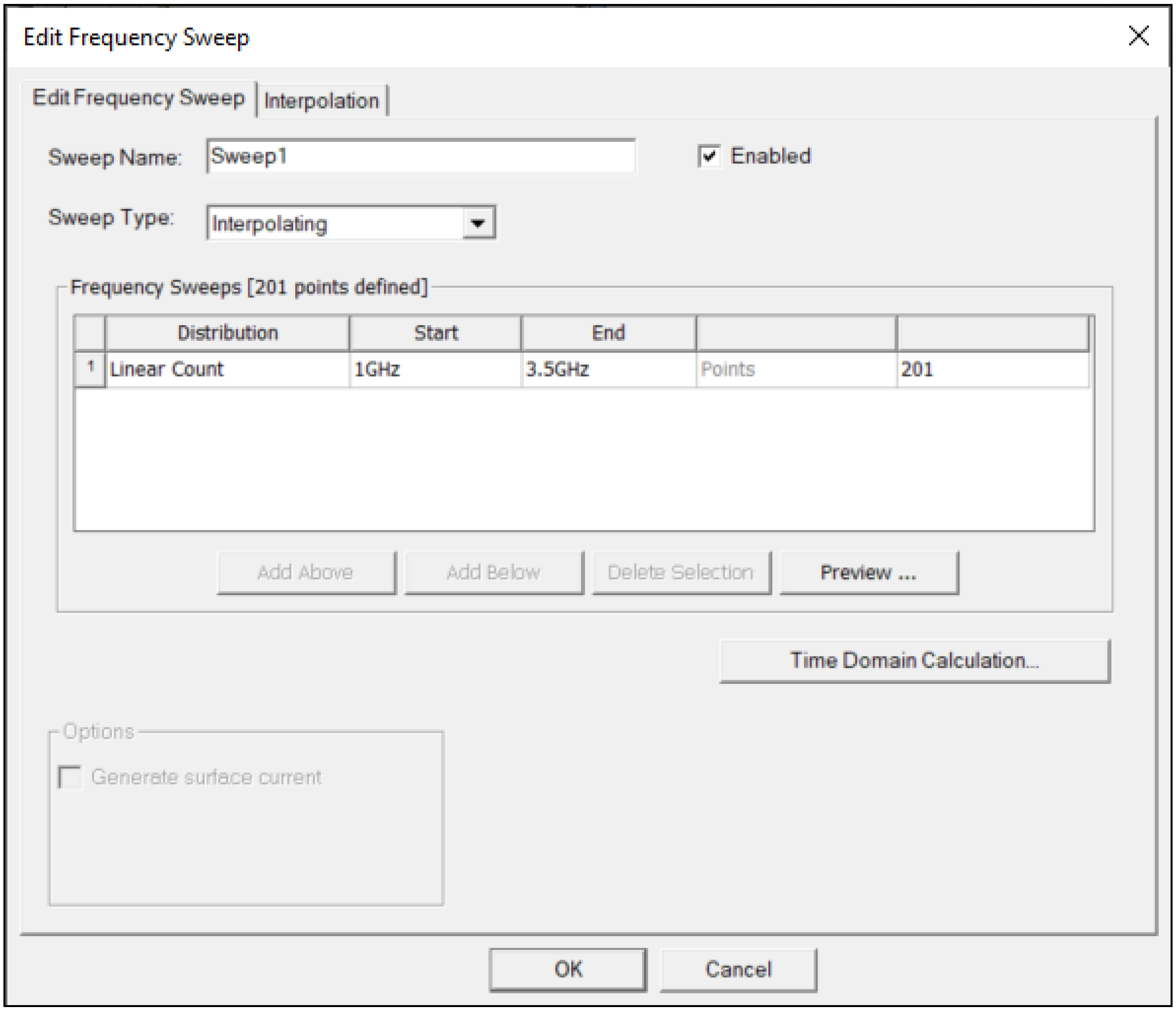


3. Click **OK** to close the **PlanarEM Setup** window and open the **Edit Frequency Sweep** window.

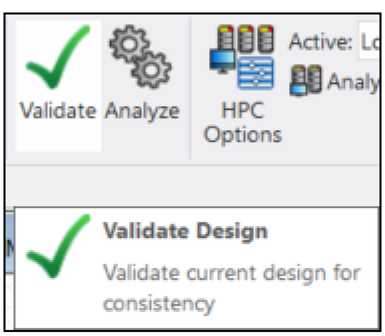


4. Ensure **Interpolating** is selected from the **Sweep Type** drop-down menu.
5. Select **Linear Count** from the **Distribution** drop-down menu in the first row of the **Frequency Sweeps** table.
6. Enter the following parameters in the first row of the **Frequency Sweeps** table:

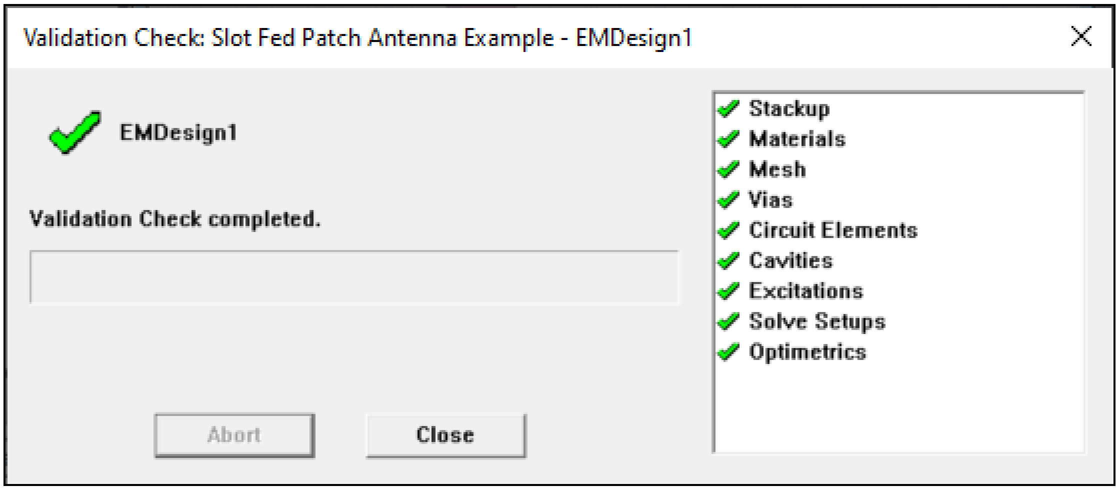
- Enter **1** (GHz) in the **Start** column.
- Enter **3.5** (GHz) in the **Stop** column.
- Enter **201** in the **Points** field.



7. Click **OK** to finalize the interpolating sweep and close the **Edit Frequency Sweep** window.
8. From the **Simulation** ribbon tab, click **Validate** to open the **Validation Check** window.

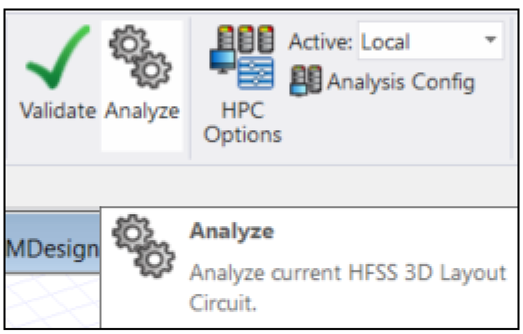


9. When the validation check is complete, click **Close**.

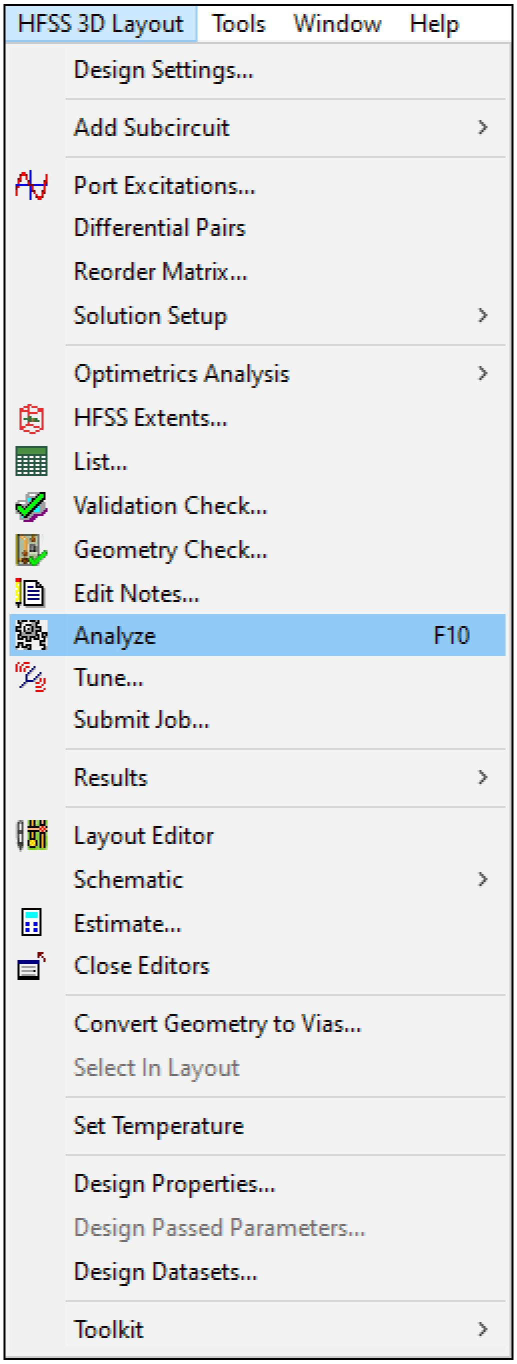


10. To run the Planar EM analysis, do one of the following:

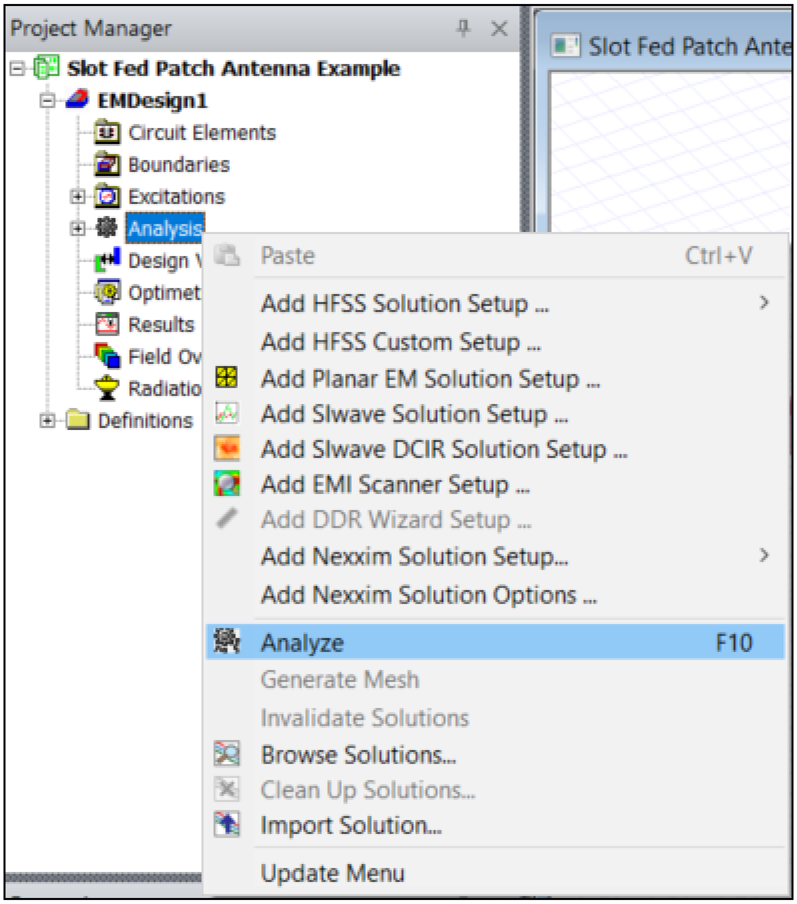
    - From the **Simulation** tab, click**Analyze**.



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

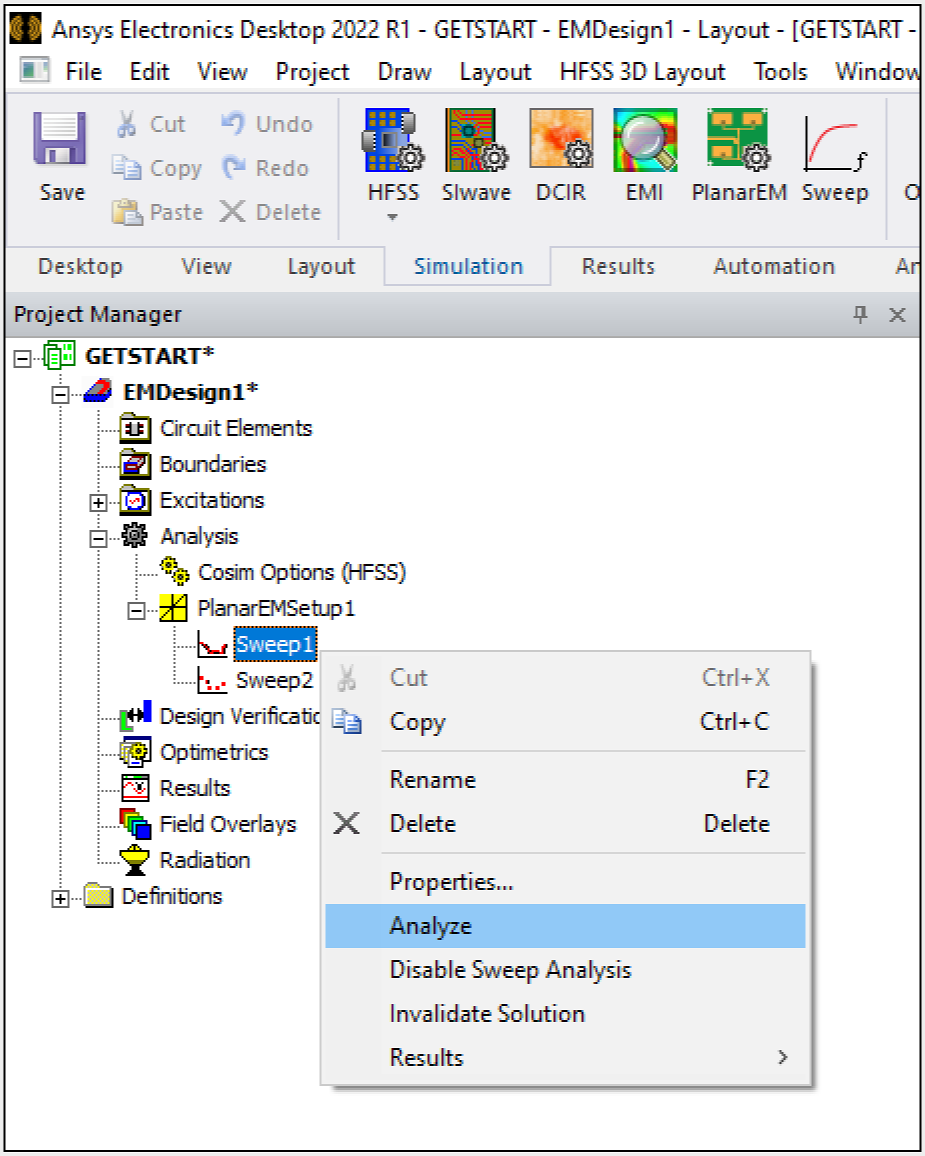


- From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]**. Then right-click **Analysis** and select **Analyze**.


Project Manager
Slot Fed Patch Antenna Example
EMDesign1
Circuit Elements
Boundaries
Excitations
Analysis
Results
Definitions
Slot Fed Patch Ante
Paste Ctrl+V
Add HFSS Solution Setup ...
Add HFSS Custom Setup ...
Add Planar EM Solution Setup ...
Add SIwave Solution Setup ...
Add SIwave DCIR Solution Setup ...
Add EMI Scanner Setup ...
Add DDR Wizard 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**.

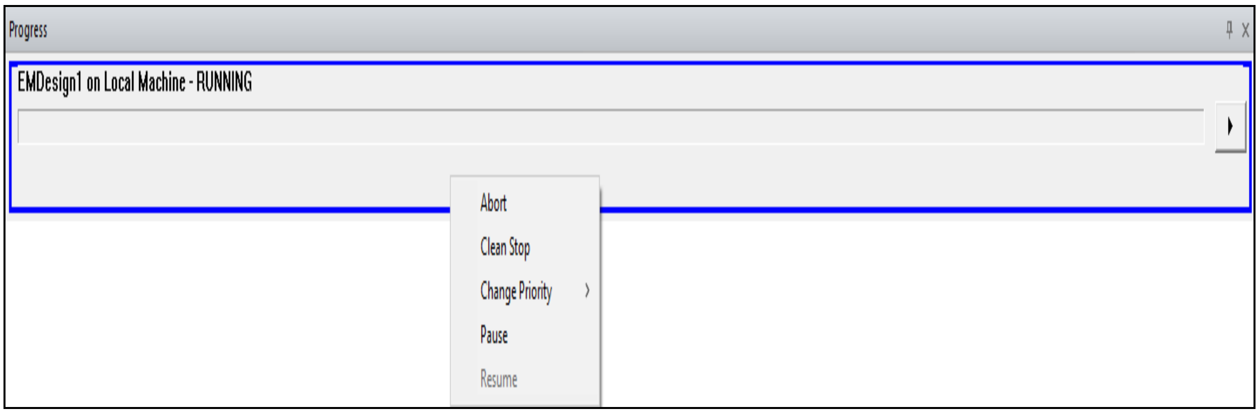


View Progress

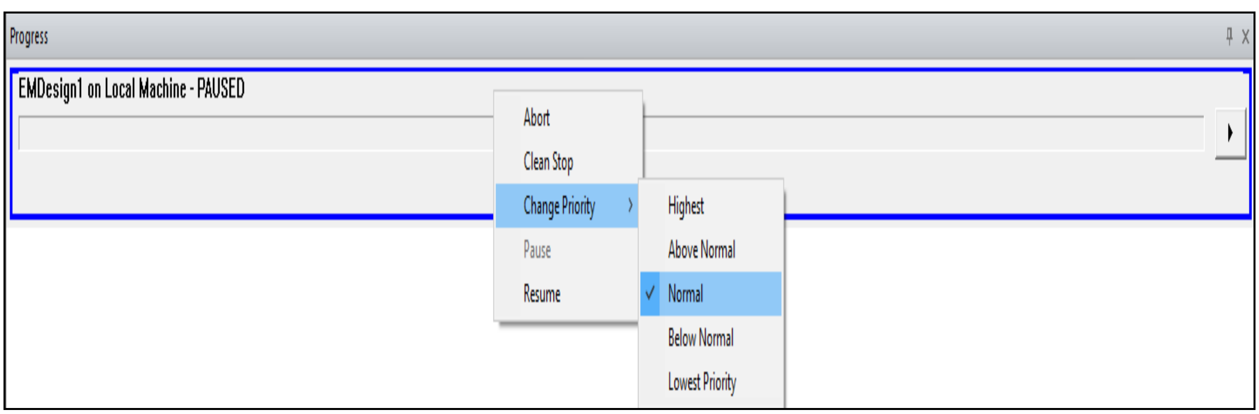
From the **Electronics Desktop** status bar, select **Show Progress** to expand the **Progress** window.

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.

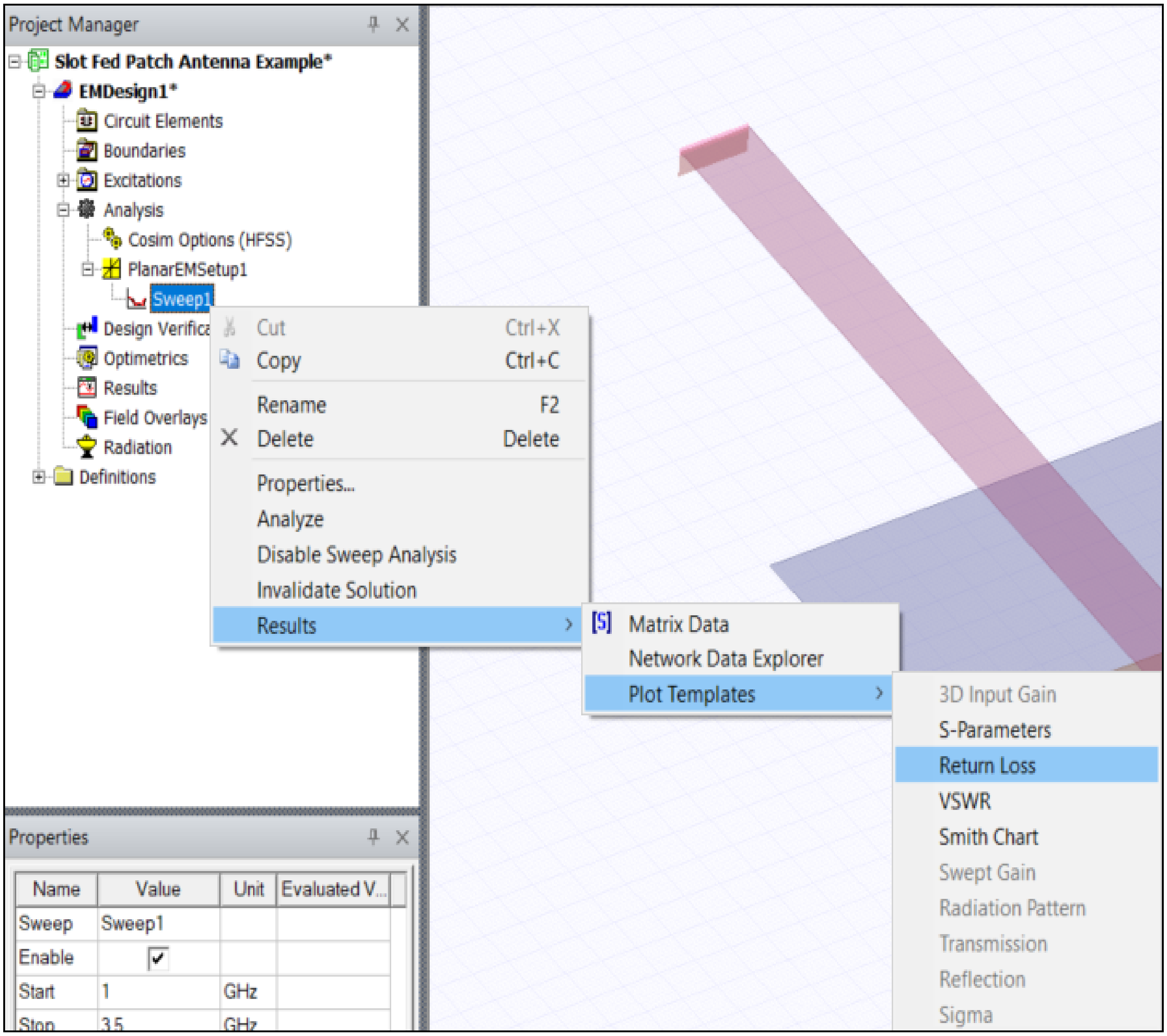


Continue to **Create Return Loss Report**.

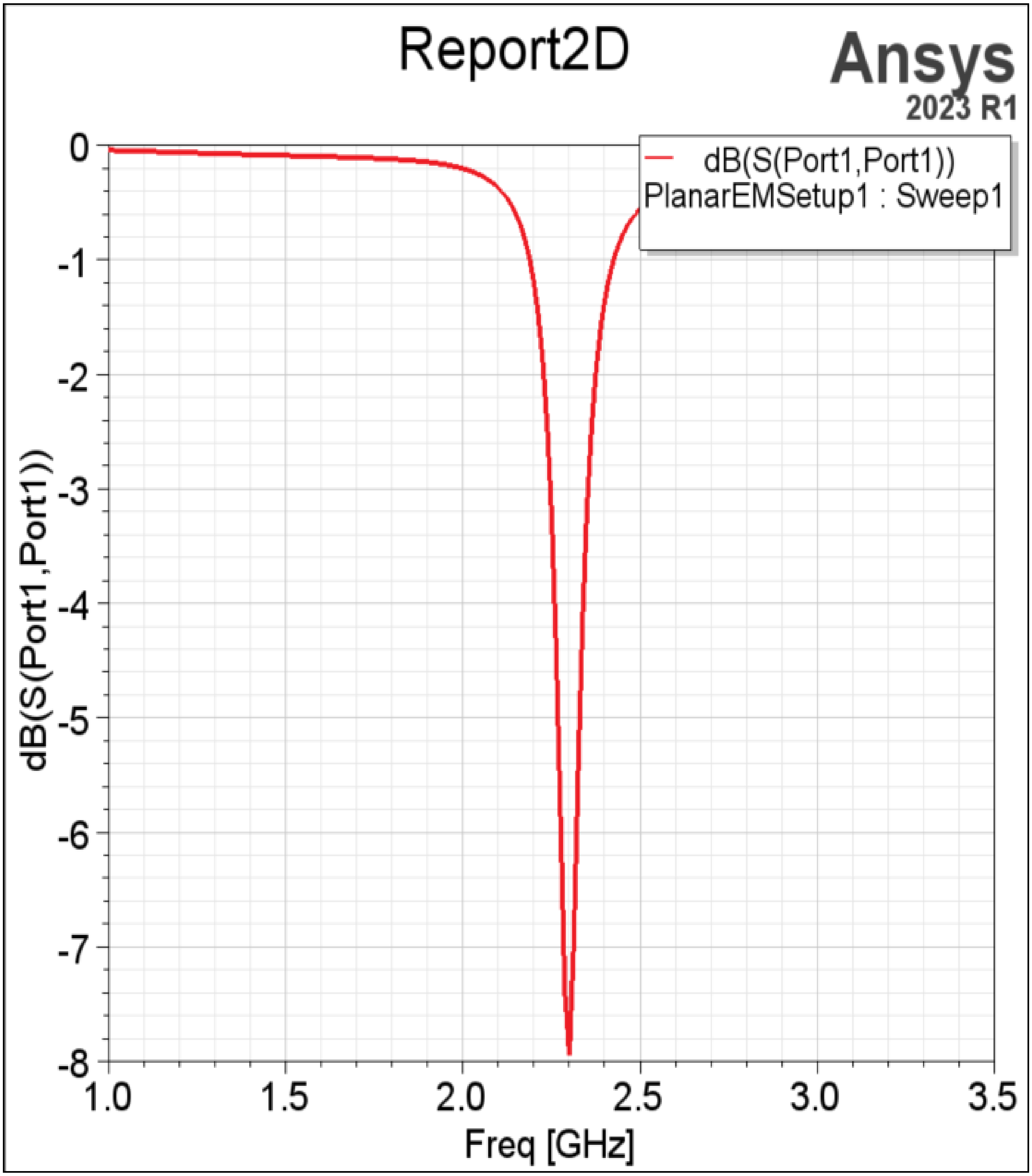
Creating a Return Loss Report

Complete these steps to create a return loss report.

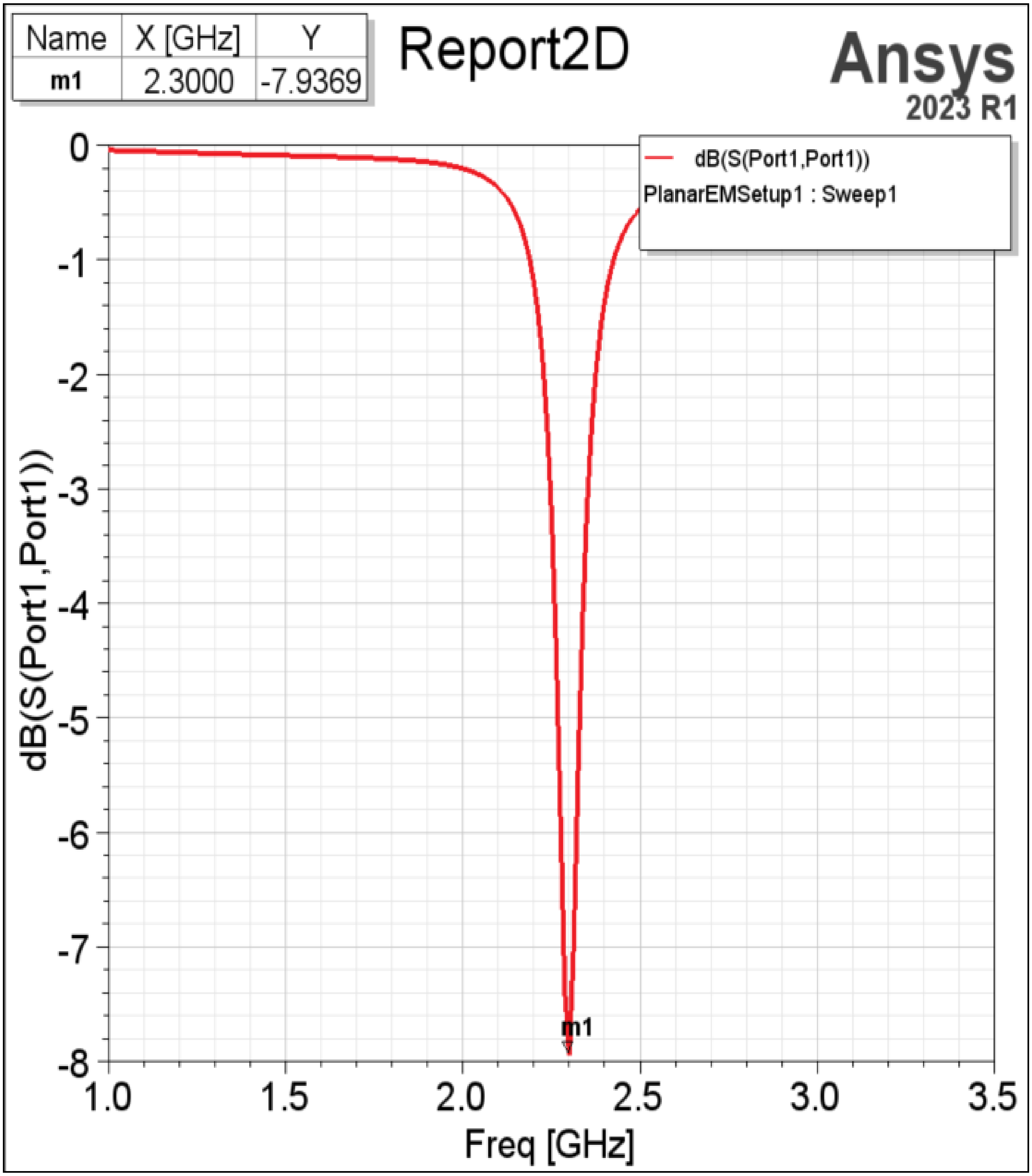
1. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Analysis** > **[*chosen setup (e.g., PlanarEMSetup1*)]**. Then right-click the chosen sweep (e.g., **Sweep1**) and select **Results** > **Plot Templates** > **Return Loss** to open a **Report2D** window with an S Parameter plot (e.g., **dB[S(Port1,Port1)]**).



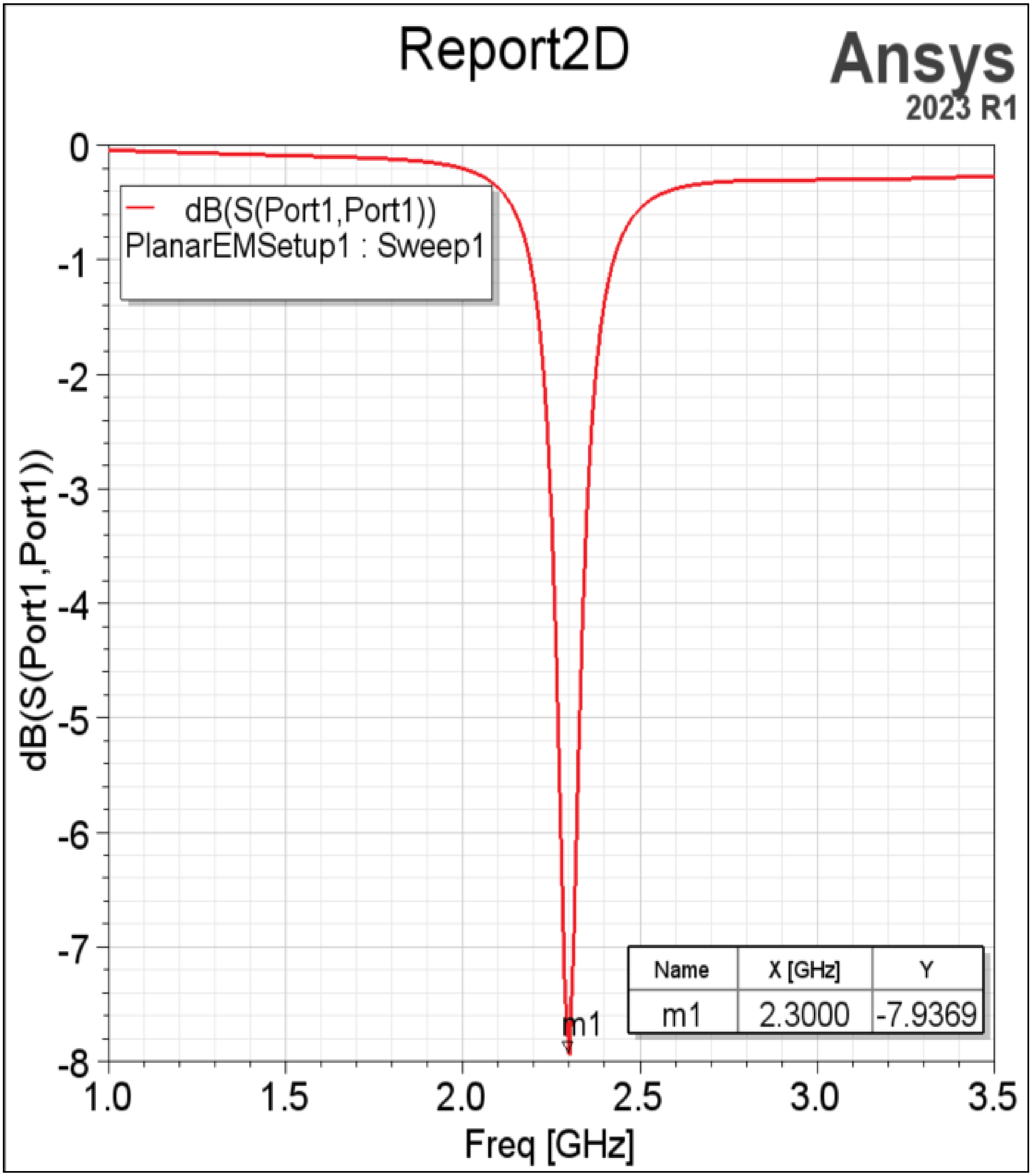
2. Click anywhere from the trace (i.e., the red line) to select it. The line becomes thicker when selected.



3. While the trace is selected, right-click anywhere within the **Report2D** window. Then select **Marker** > **Add Minimum**. An arrow will appear, indicating the lowest position from the line, along with a legend giving the marker's exact X,Y coordinates.



4. Drag the legends to suitable locations and click anywhere outside of the trace to deselect it.

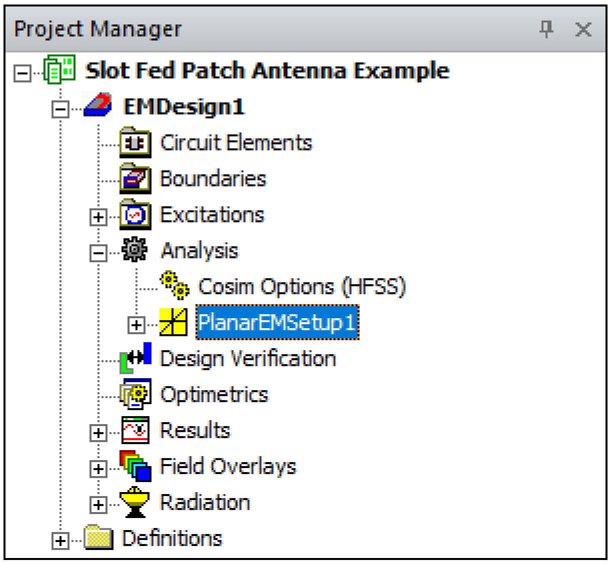


Continue to [Add and Analyze a Discrete Sweep](#).

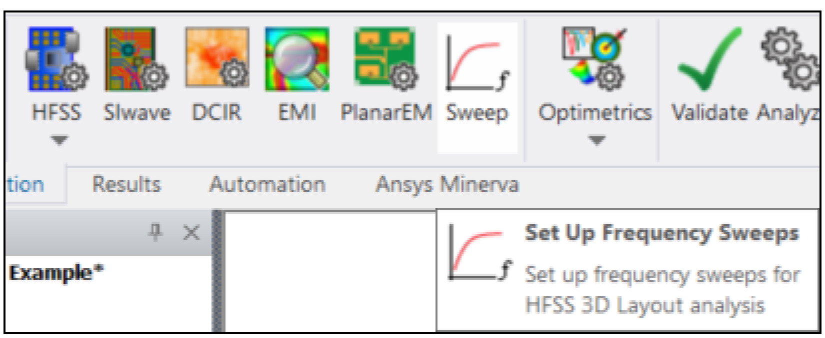
Adding and Analyzing a Discrete Sweep

Complete these steps to add a new discrete frequency sweep, targeting the frequency at which the minimum return loss occurred, and save the fields so a current overlay can be created and animated after the sweep is analyzed.

1. From the **Project Manager** window, select the chosen analysis (e.g., **PlanarEMSetup1**).



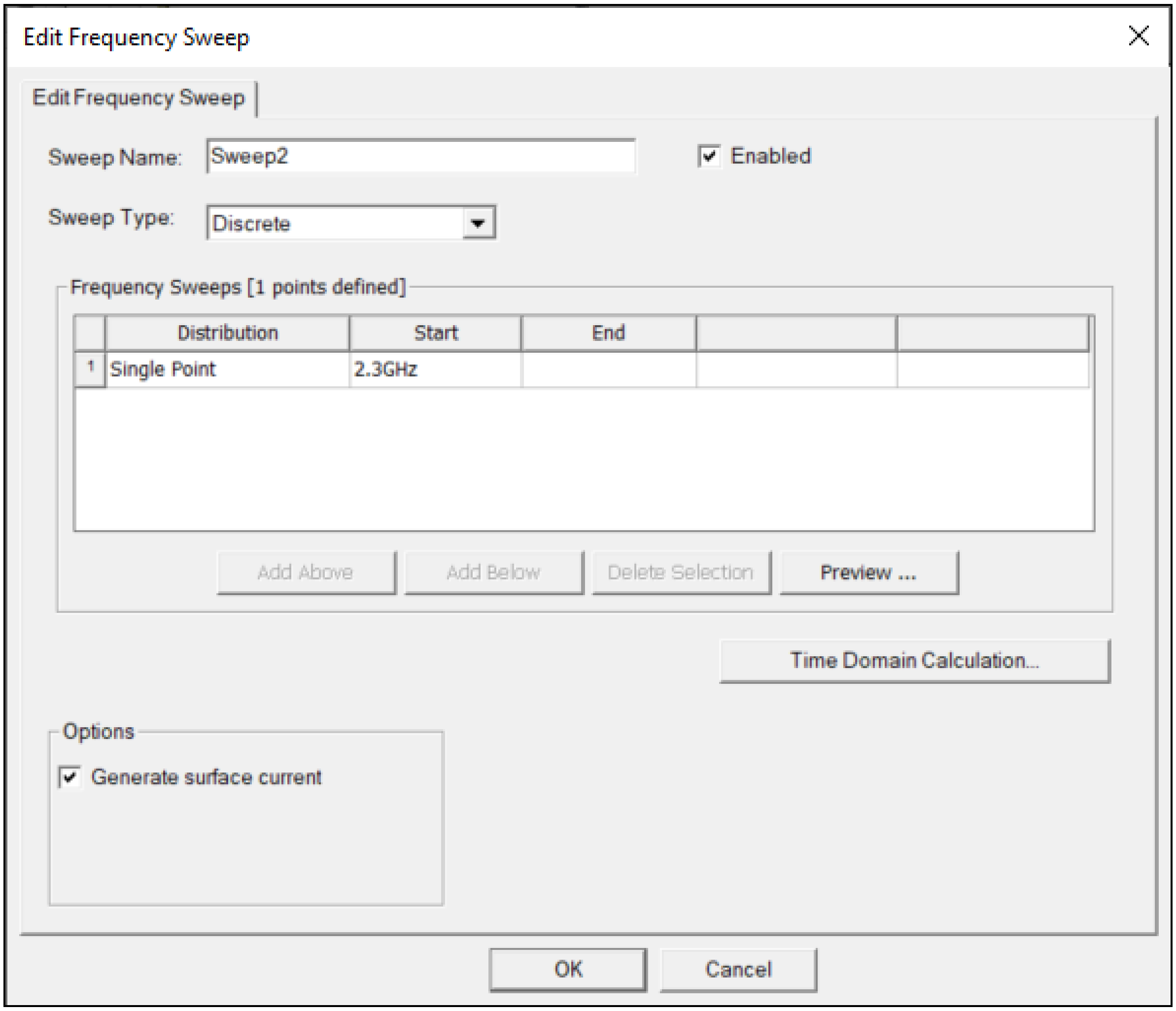
2. From the **Simulation** ribbon tab, click **Sweep** to open the **Edit Frequency Sweep** window.



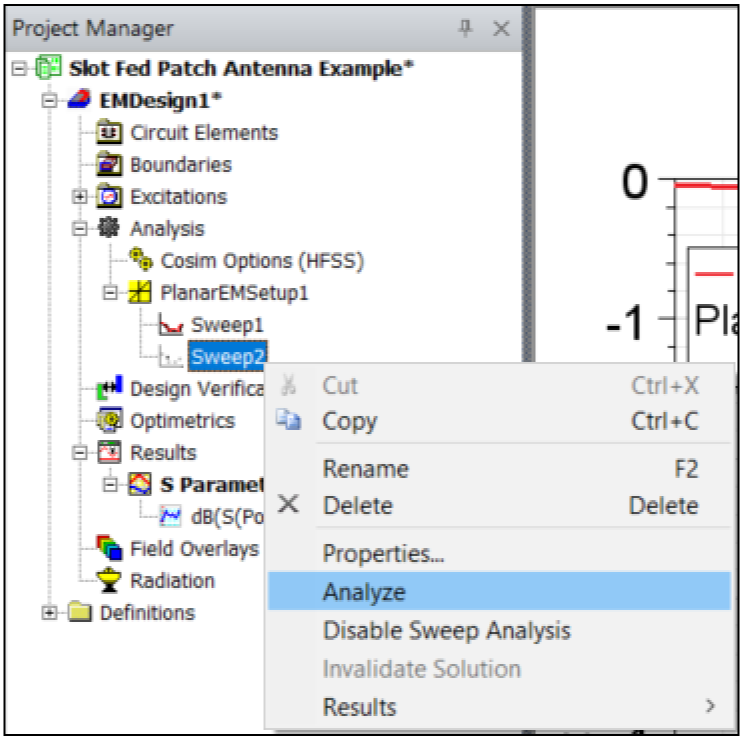
3. From the **Edit Frequency Sweep** window, do the following:
   a. Select **Discrete** from the **Sweep Type** drop-down menu.
   b. From the **Frequency Sweeps** area, select **Single Point** from the **Distribution** drop-down menu.
   c. Enter **2.3** in the **Start** field.

d. Check the **Generate surface current** box.

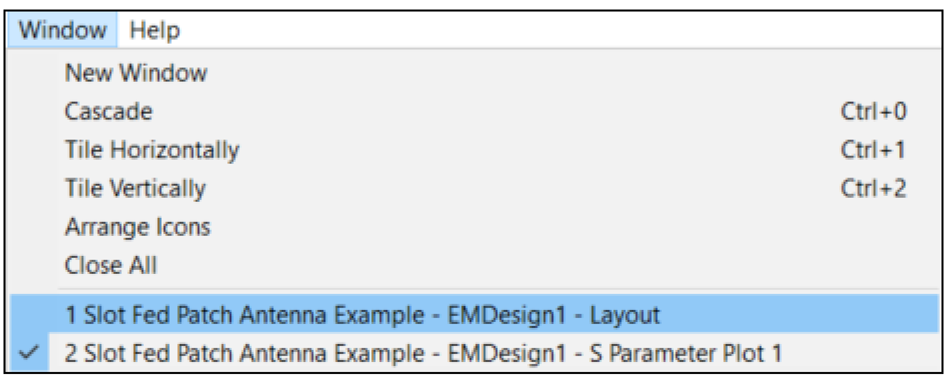
e. Click **OK** to close the **Edit Frequency Sweep** window.



4. From the **Project Manager** window, right-click the chosen sweep (e.g., **Sweep2**) and select **Analyze**.



5. From **Window**, select **Layout Editor**.

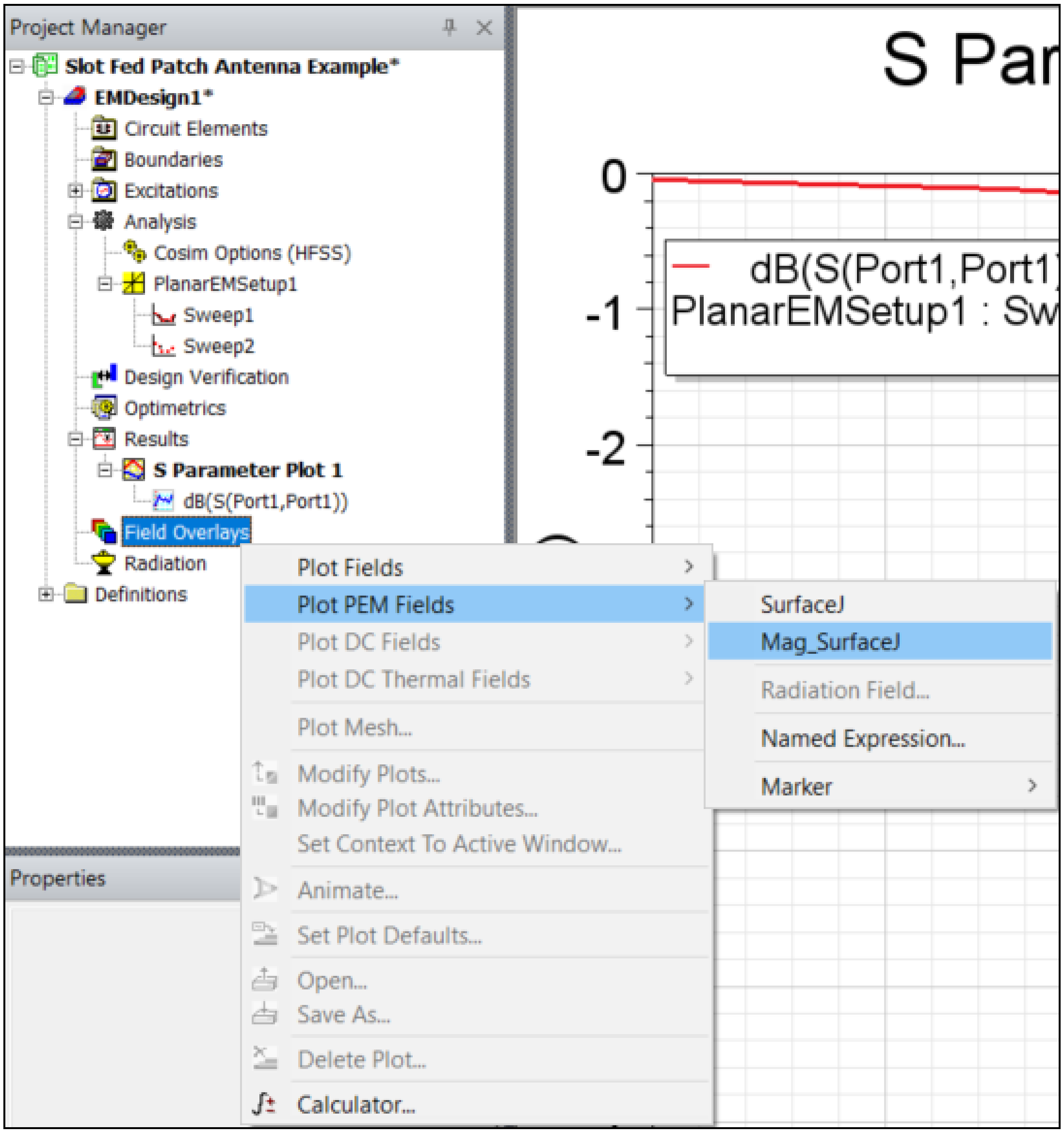


Continue to View Surface Currents.

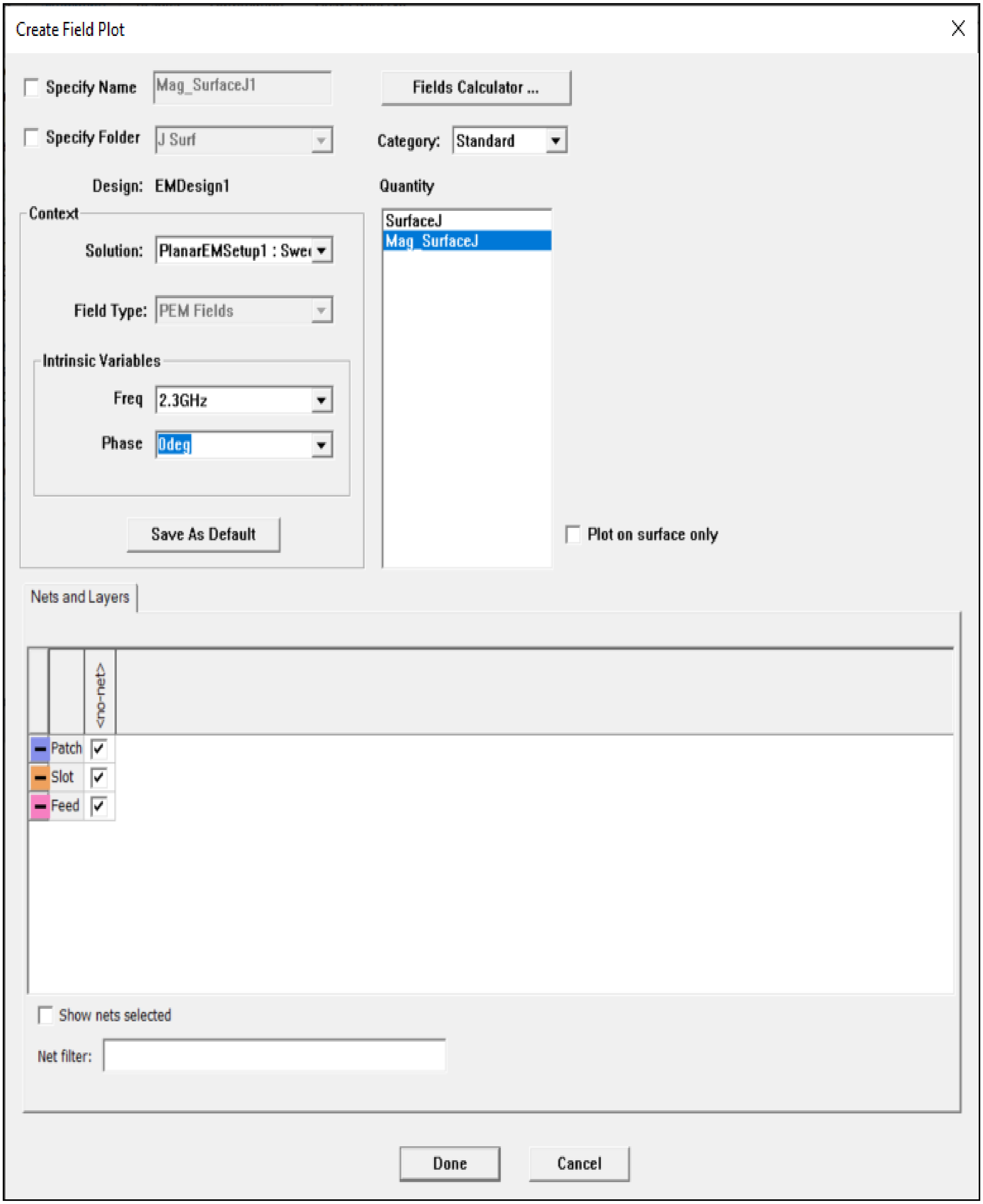
Viewing Surface Currents

After the discrete sweep analysis, complete these steps to add and view a current density overlay.

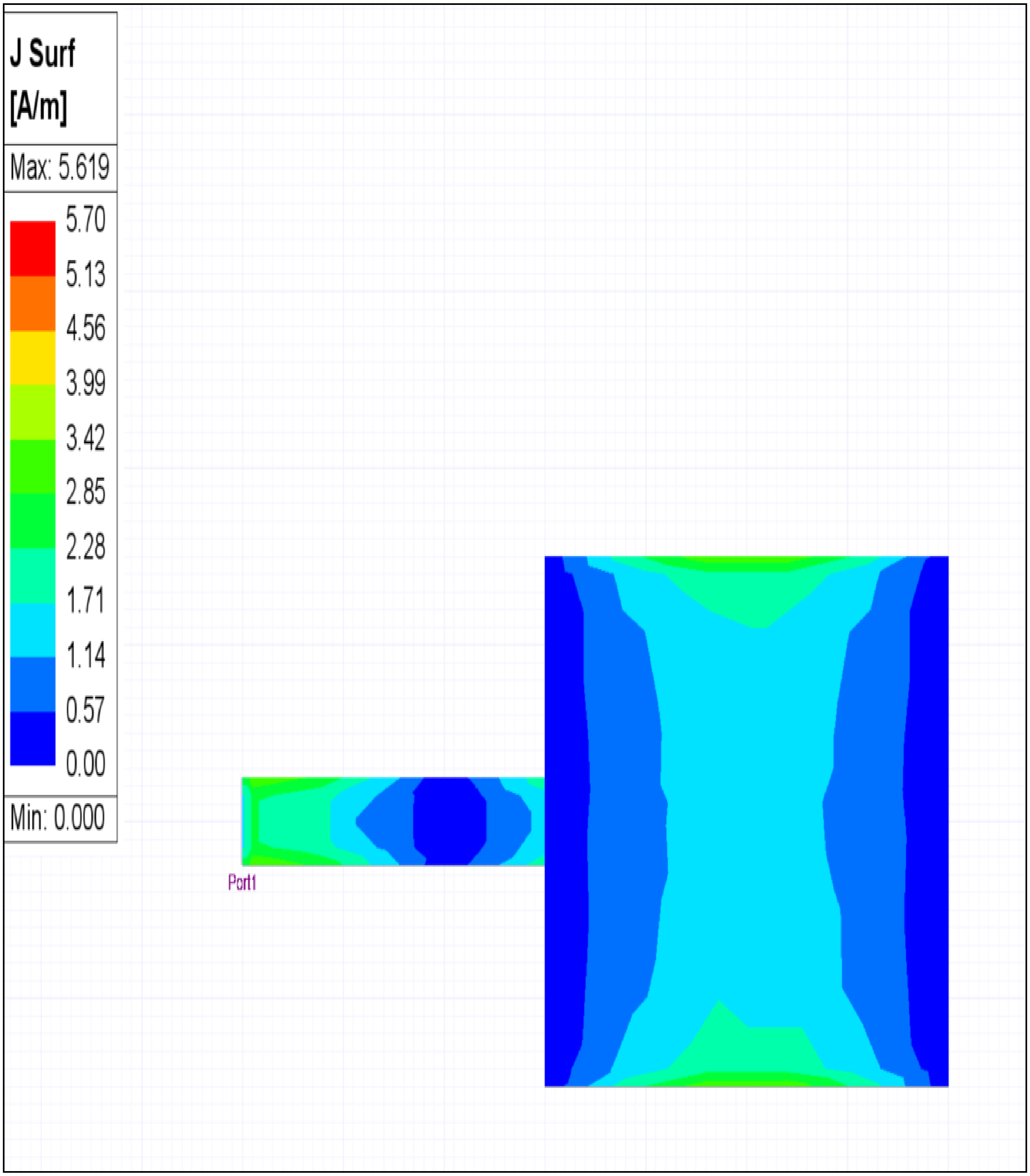
1. In the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Field Overlays** and select **Plot PEM Fields** > **Mag_SurfaceJ** to open the **Create Field Plot** window.



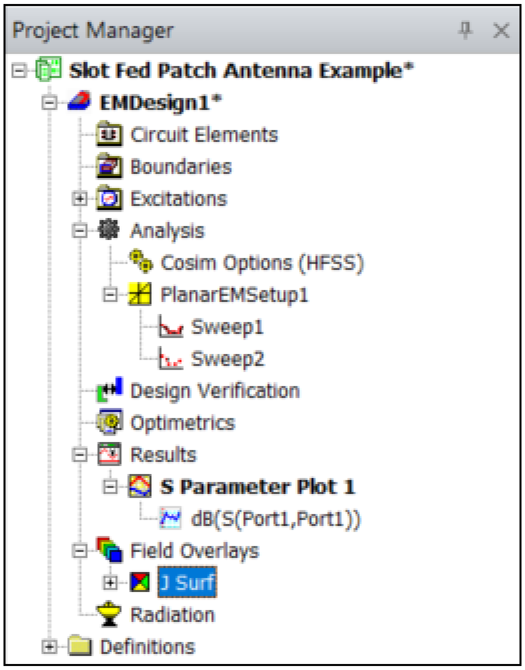
2. Under the **Nets and Layers** tab, click the blank column heading above the layer names to simultaneously select all three conducting layers (e.g., **Patch**, **Slot**, and **Feed** layers.


Create Field Plot
Specify Name
Mag_SurfaceJ1
Fields Calculator ...
Specify Folder
J Surf
Category:
Standard
Design:
EMDesign1
Quantity
Context
SurfaceJ
Mag_SurfaceJ
Solution:
PlanarEMSetup1 : Swe
Field Type:
PEM Fields
Intrinsic Variables
Freq
2.3GHz
Phase
0deg
Save As Default
Plot on surface only
Nets and Layers
<no-net>
Patch
Slot
Feed
Show nets selected
Net filter:
Done
Cancel

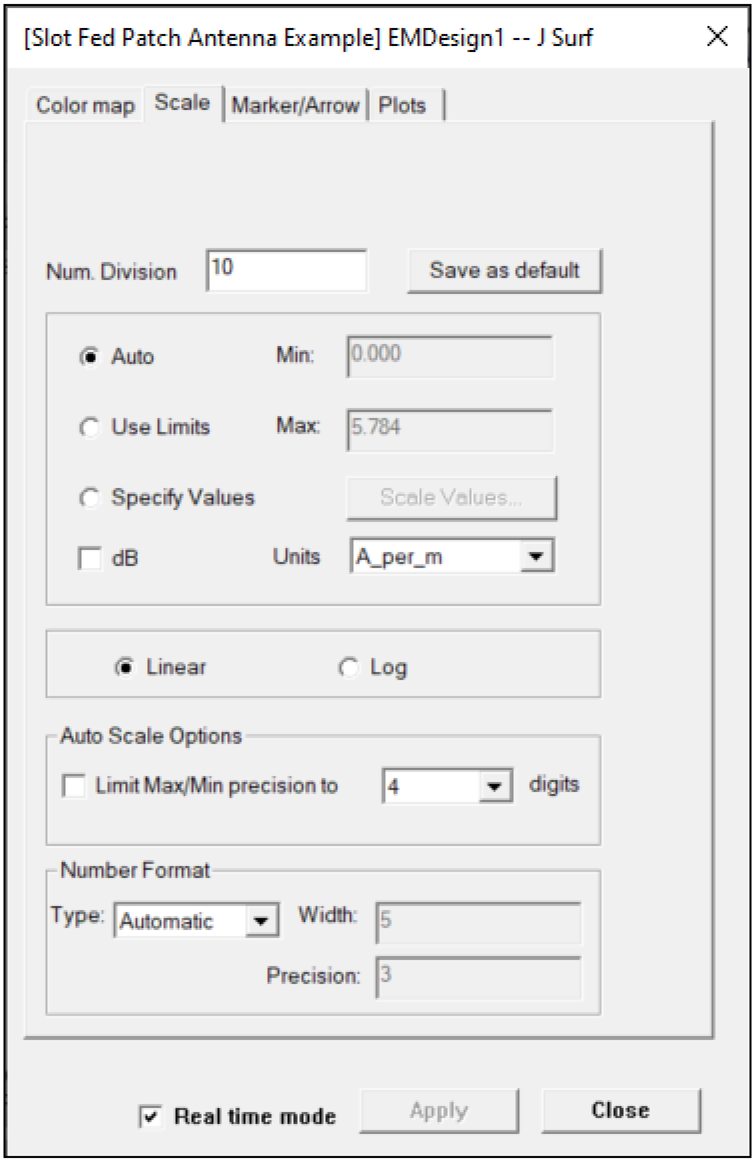
3. Click **Done** to generate the surface current density overlay.



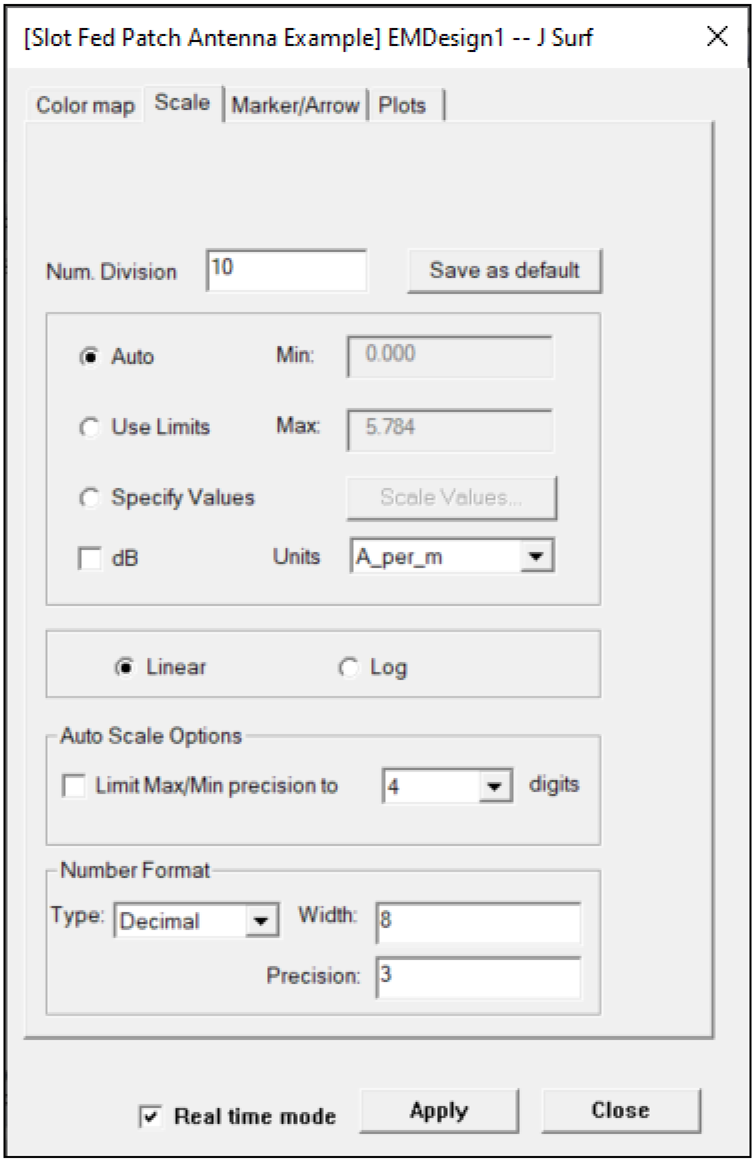
4. Within the **Layout Editor**, double-click the *J Surf* plot legend to open the legend settings window.


Project Manager
Slot Fed Patch Antenna Example*
EMDesign1*
Circuit Elements
Boundaries
Excitations
Analysis
Cosim Options (HFSS)
PlanarEMSetup1
Sweep1
Sweep2
Design Verification
Optimetrics
Results
S Parameter Plot 1
dB(S(Port1,Port1))
Field Overlays
J Surf
Radiation
Definitions

5. Click the **Scale** tab.

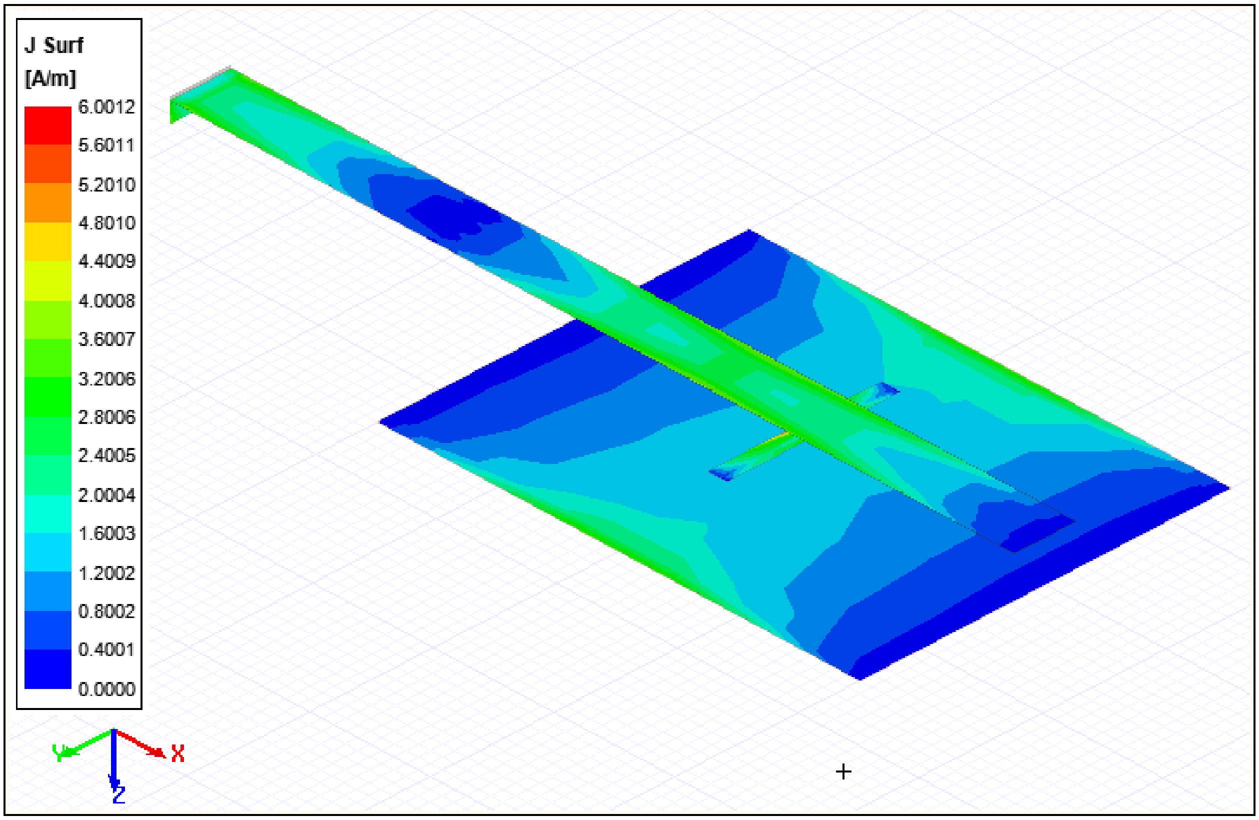


6. Within the **Number Format** area, make the following changes:
   a. Select **Decimal** from the **Type** drop-down menu.
   b. Enter **8** in the **Width** field.

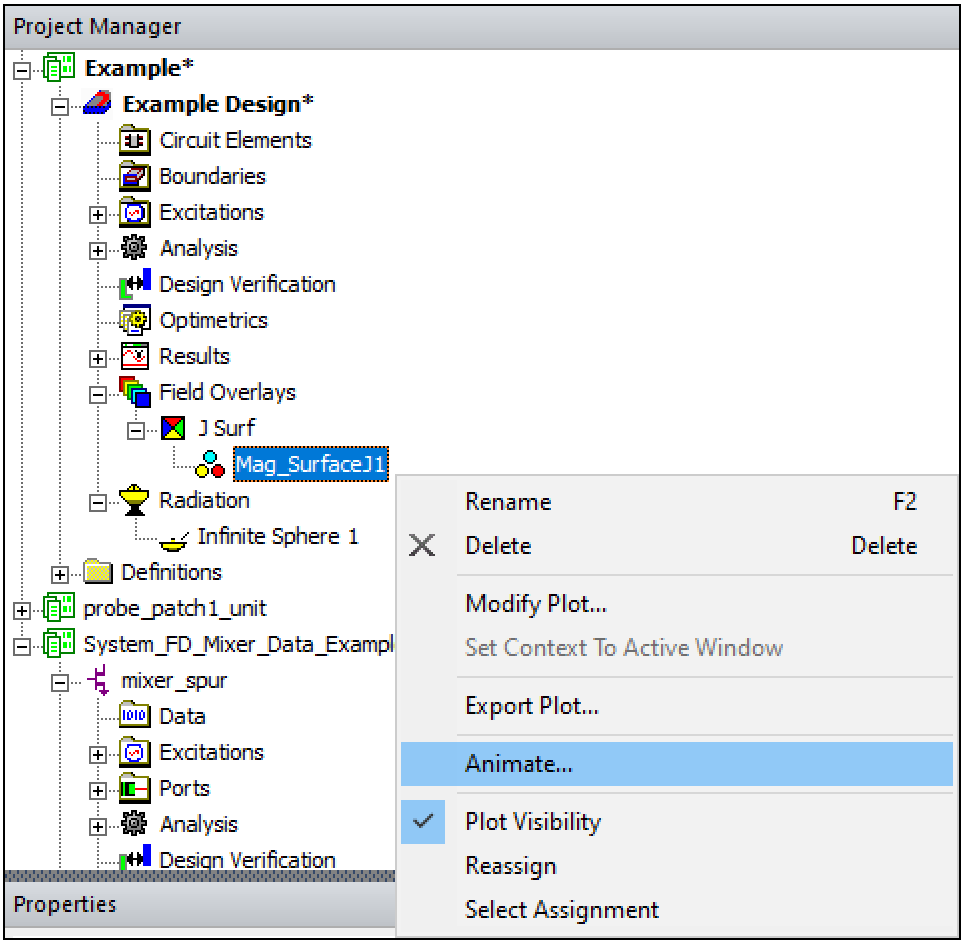


7. Click **Apply**.
8. Click **Close**. The changes will be immediately noticeable within the **Layout Editor**.

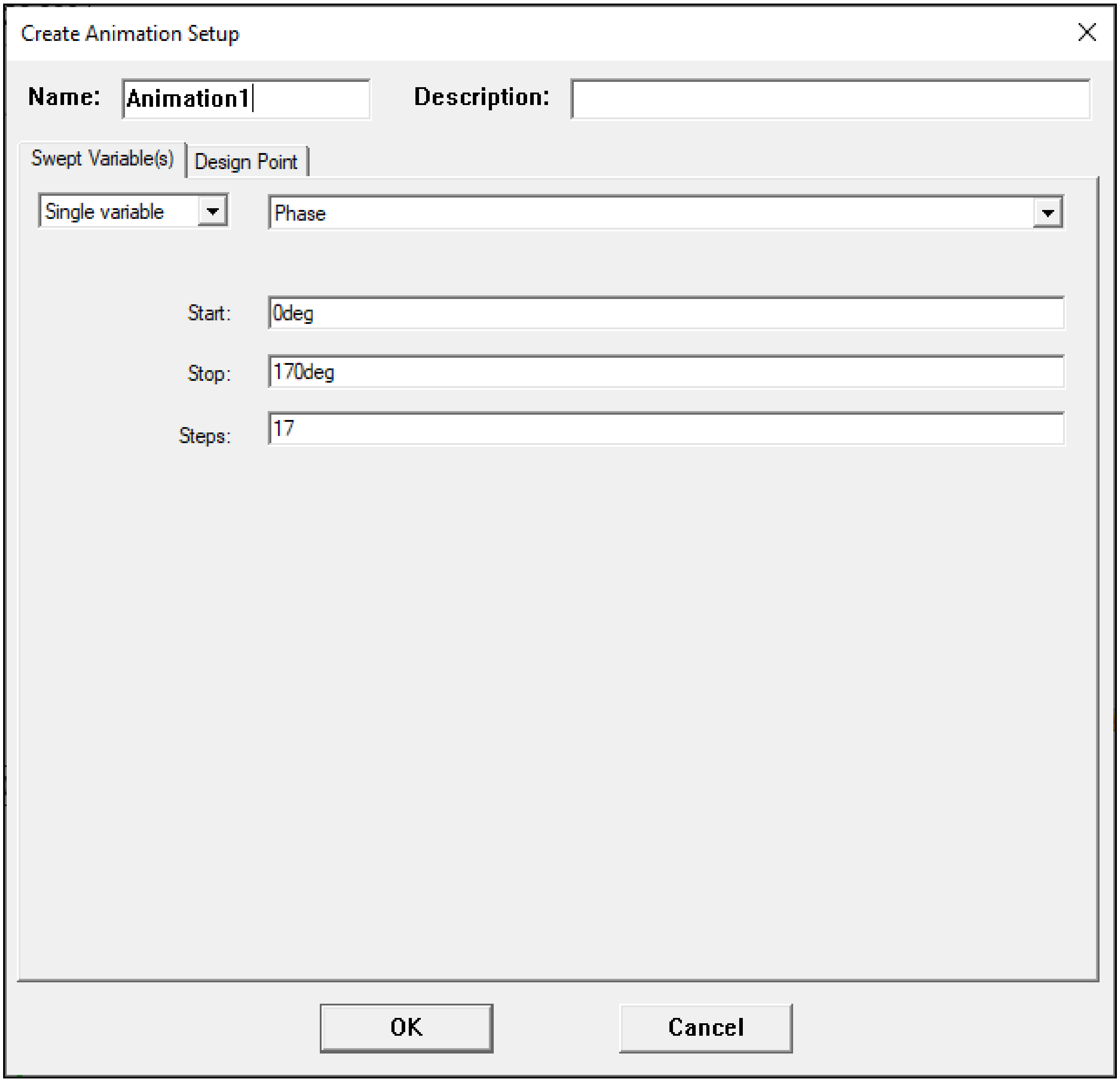
9. Rotate the model to see all three layers, as shown in the following figure.



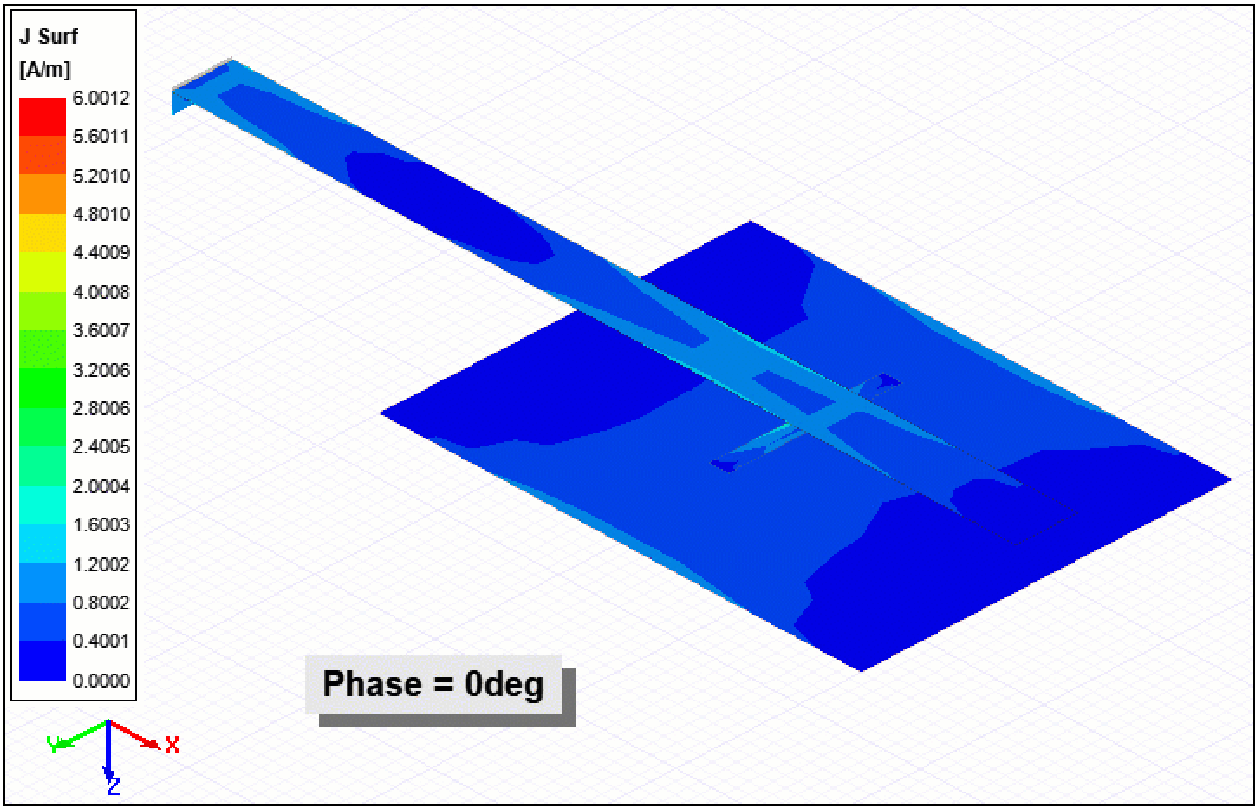
10. From the **Project Manager** window, expand the **Project Tree** > **[*active design folder*]** > **Field Overlays** > **J Surf**. Then right-click **Mag_SurfaceJ1** and select **Animate** to open the **Create Animation Setup** window.



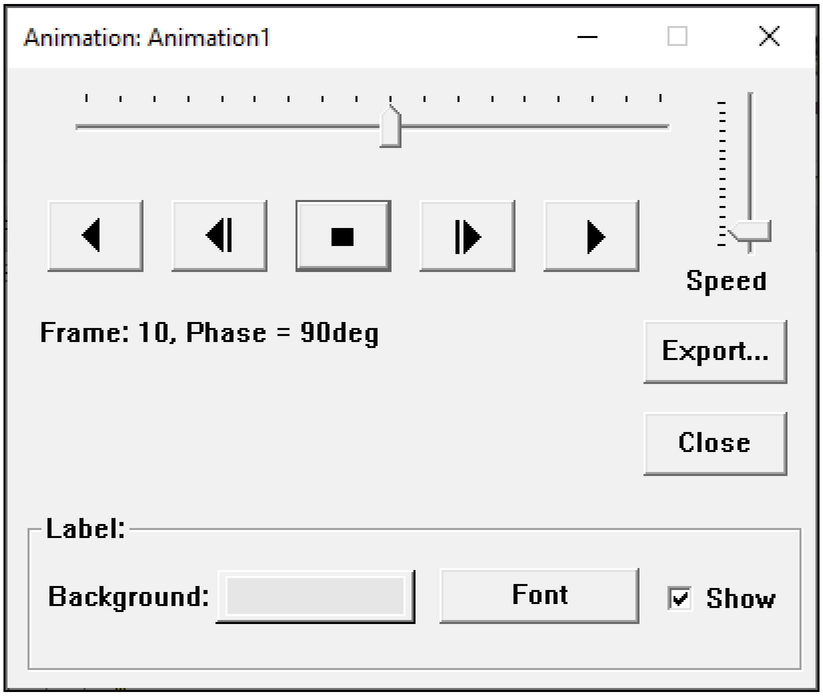
11. Within the **Create Animation Setup** window, it should not be appropriate to change the default settings, which follow:
    a. The left and right drop-down menus are set to **Single variable** and **Phase**, respectively.
    b. **0deg** is typed in the **Start** field.
    c. **170deg** is typed in the **Stop** field.
    d. **17** is typed in the **Steps** field.



12. Click **OK** to close the **Create Animation Setup** window. The **Animation** controls window will open and the animation will start.



13. Use the controls in the **Animation** controls window to **Rewind**, **Reverse**, **Stop**, **Skip Forward**, or **Fast Forward**.



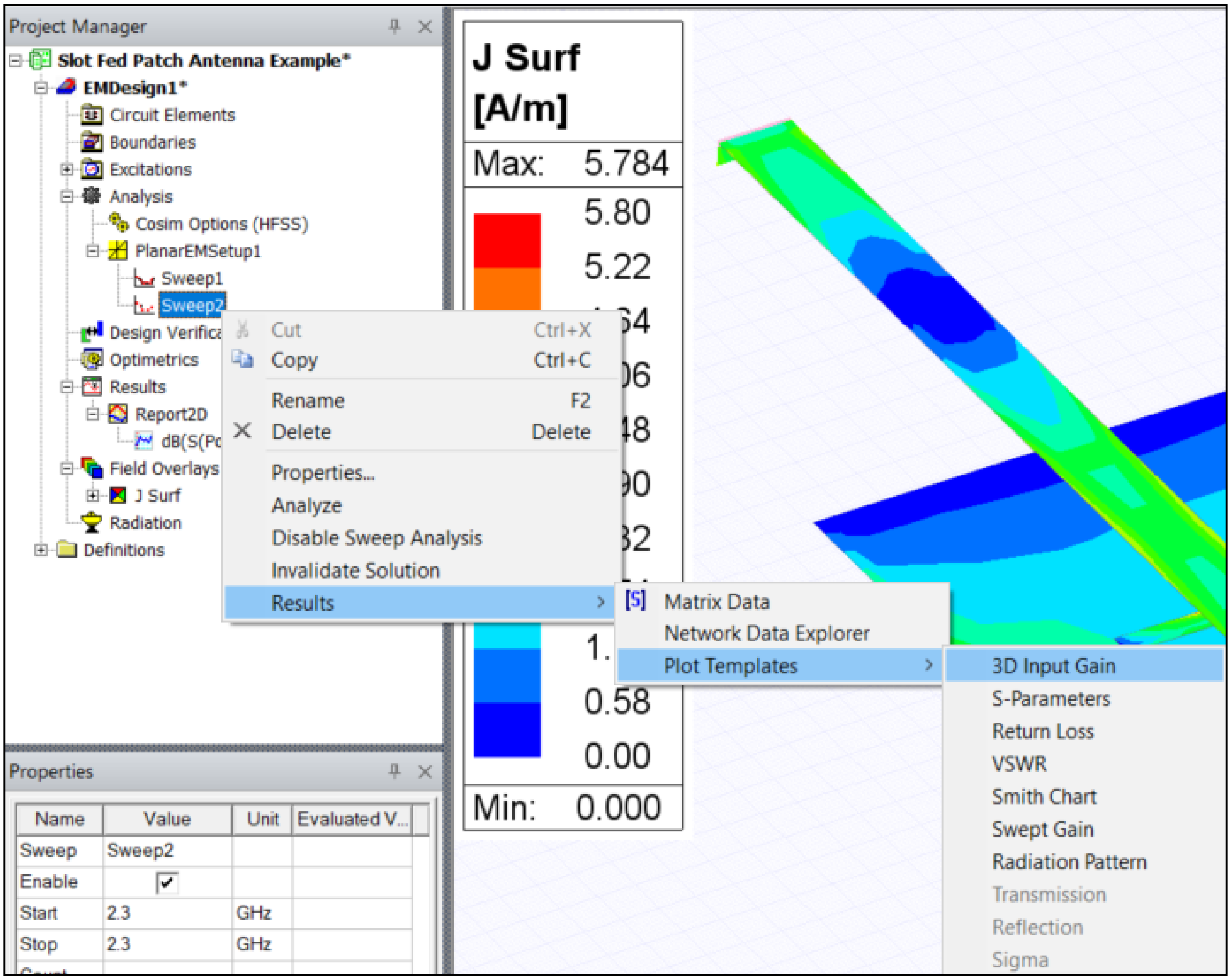
14. To end the animation, click **Close**.

Continue to Create Radiation Pattern.

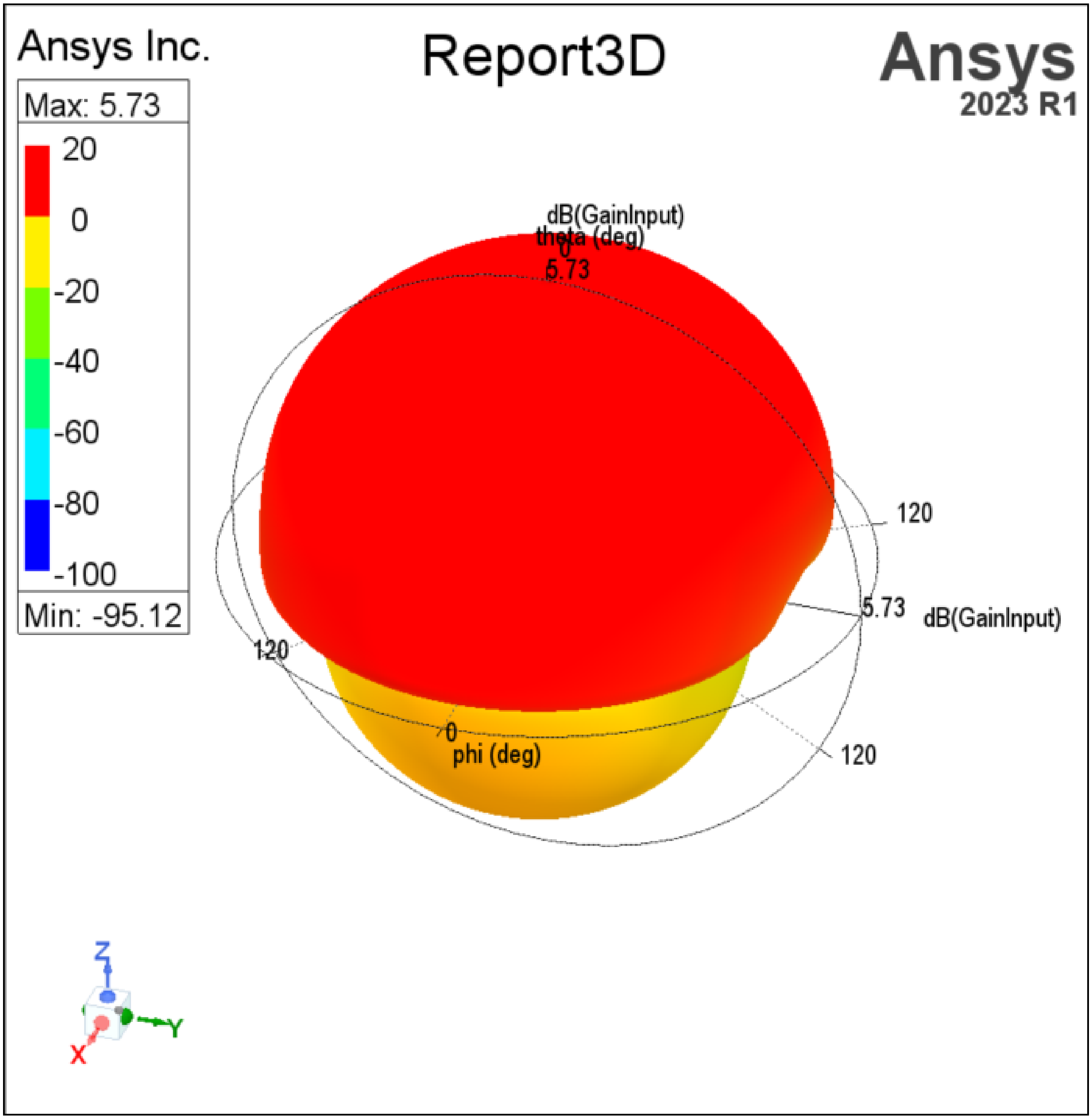
Creating a Radiation Pattern

Complete these steps to create a 3D gain pattern, a 2D polar gain plot, and then review the results. Use predefined templates to generate these plots.

1. From the **Project Manager** window,right-click **Sweep2** and select **Results** > **Plot Templates** > **3D Input Gain**.

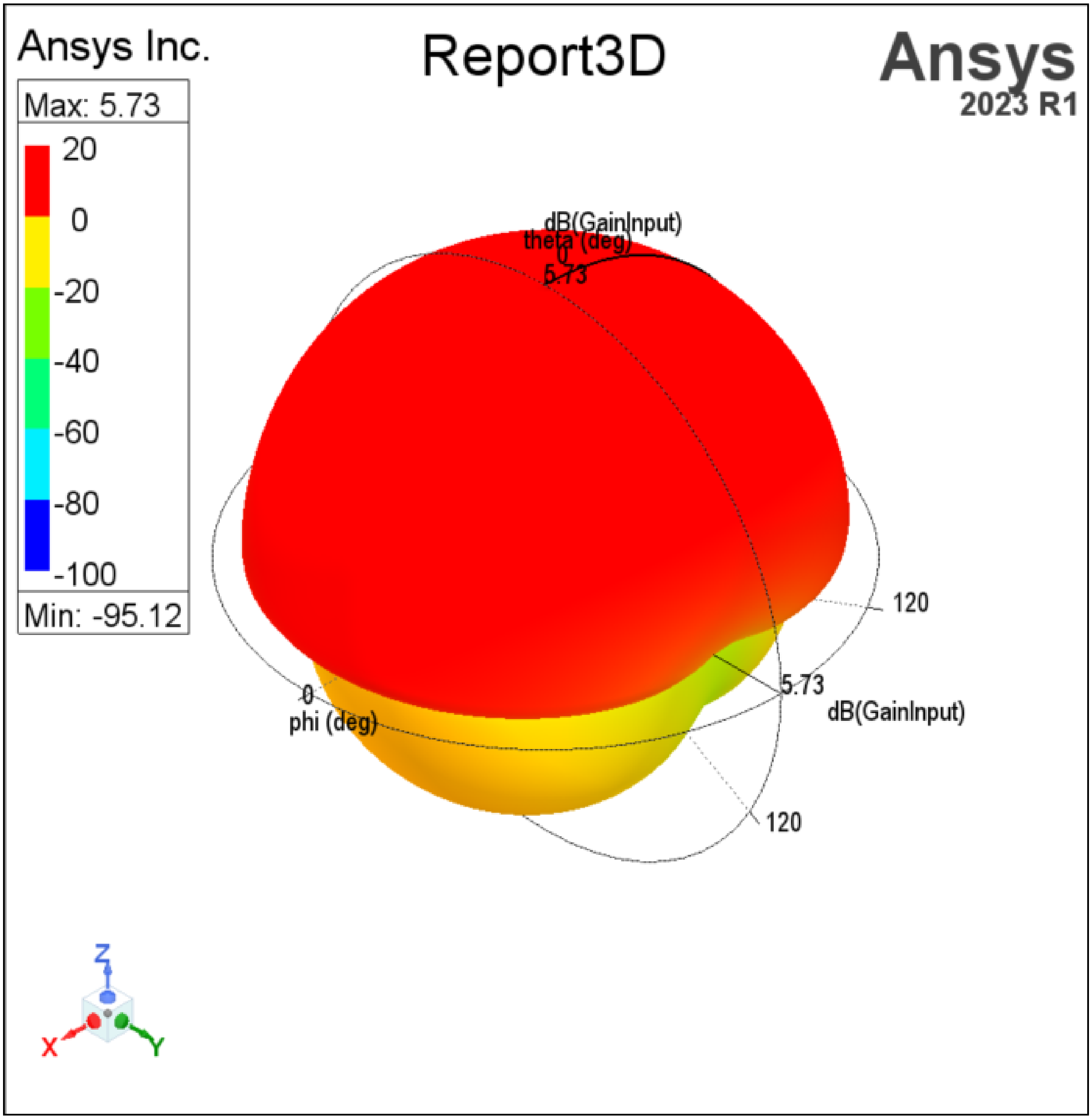


The 3D gain input pattern report appears (e.g., **Report3D**).


Ansys Inc.
Report3D
Ansys
2023 R1
Max: 5.73
20
0
-20
-40
-60
-80
-100
Min: -95.12
dB(GainInput)
theta (deg)
0
5.73
120
5.73
dB(GainInput)
120
0
phi (deg)
120
Z
X
Y

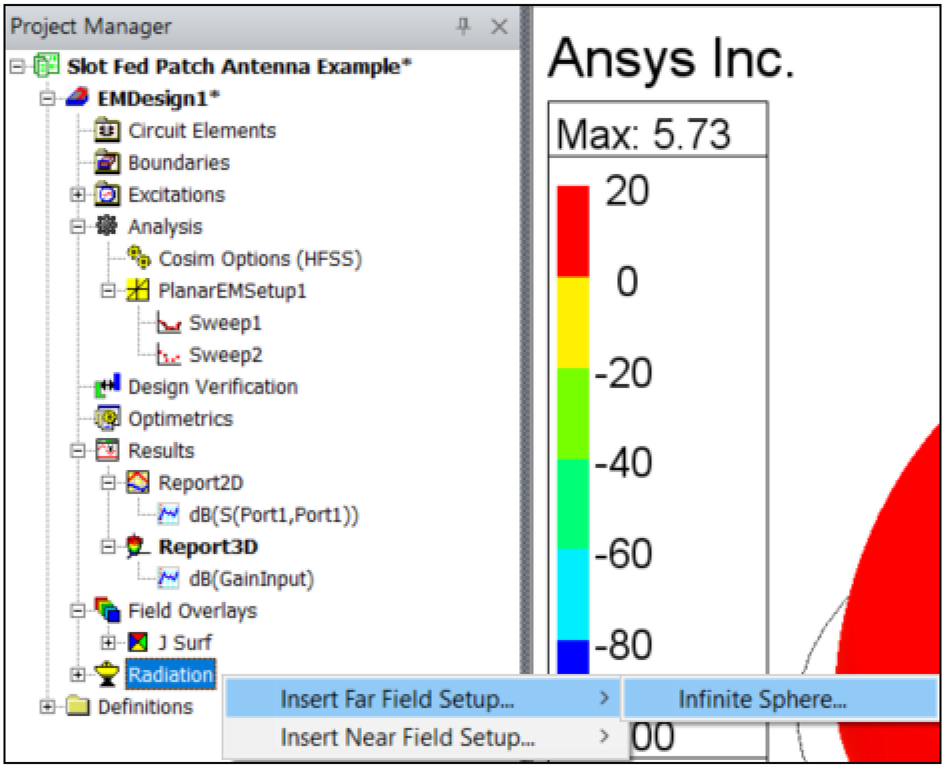
> **Note:**
>
> A default radiation setup is automatically defined (*Infinite Sphere - Default Setup*) (e.g., **Project Manager** window > **Radiation**). This radiation setup is the basis of the 3D Gain Pattern. *Theta* values range from 0 to 180 degrees (in 2 degree increments), and *Phi* values range from -180 to 180 degrees (also in 2 degree increments).

2. Rotate the model to see the visualization from different angles.

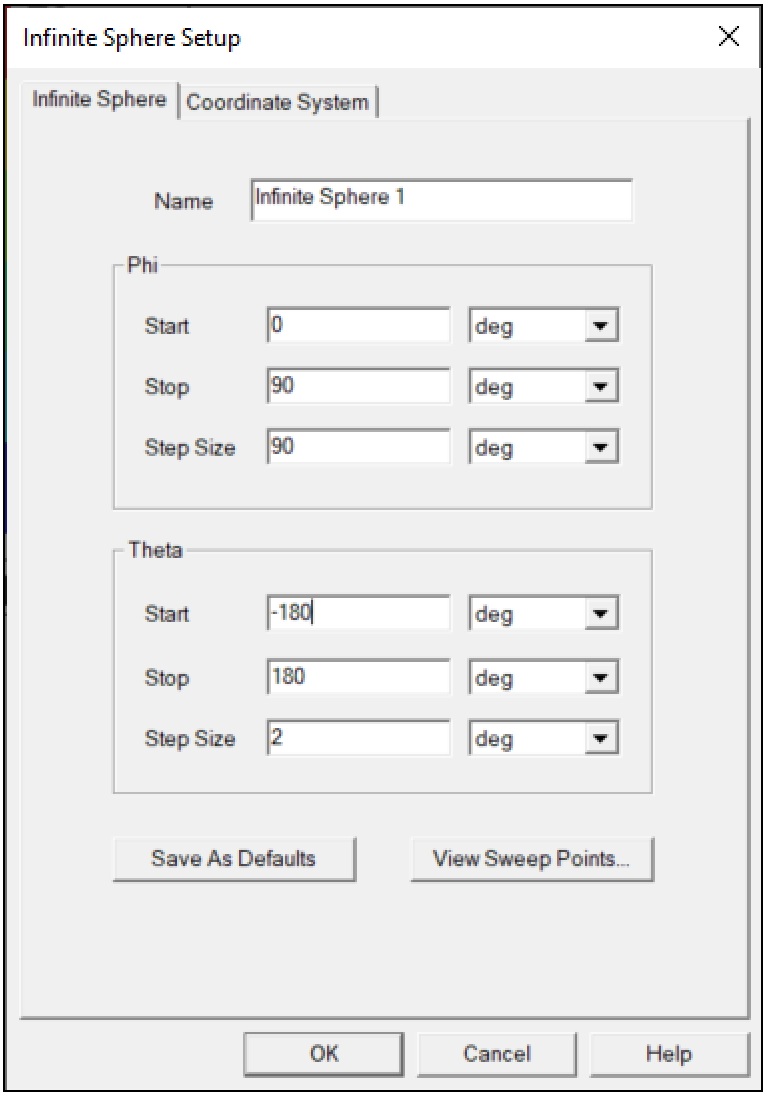


> **Note:**
>
> In the following example, a *Theta* range of -180 to 180 degrees is desirable. Limit *Phi* to two values (0 degrees and 90 degrees), resulting in only two traces.

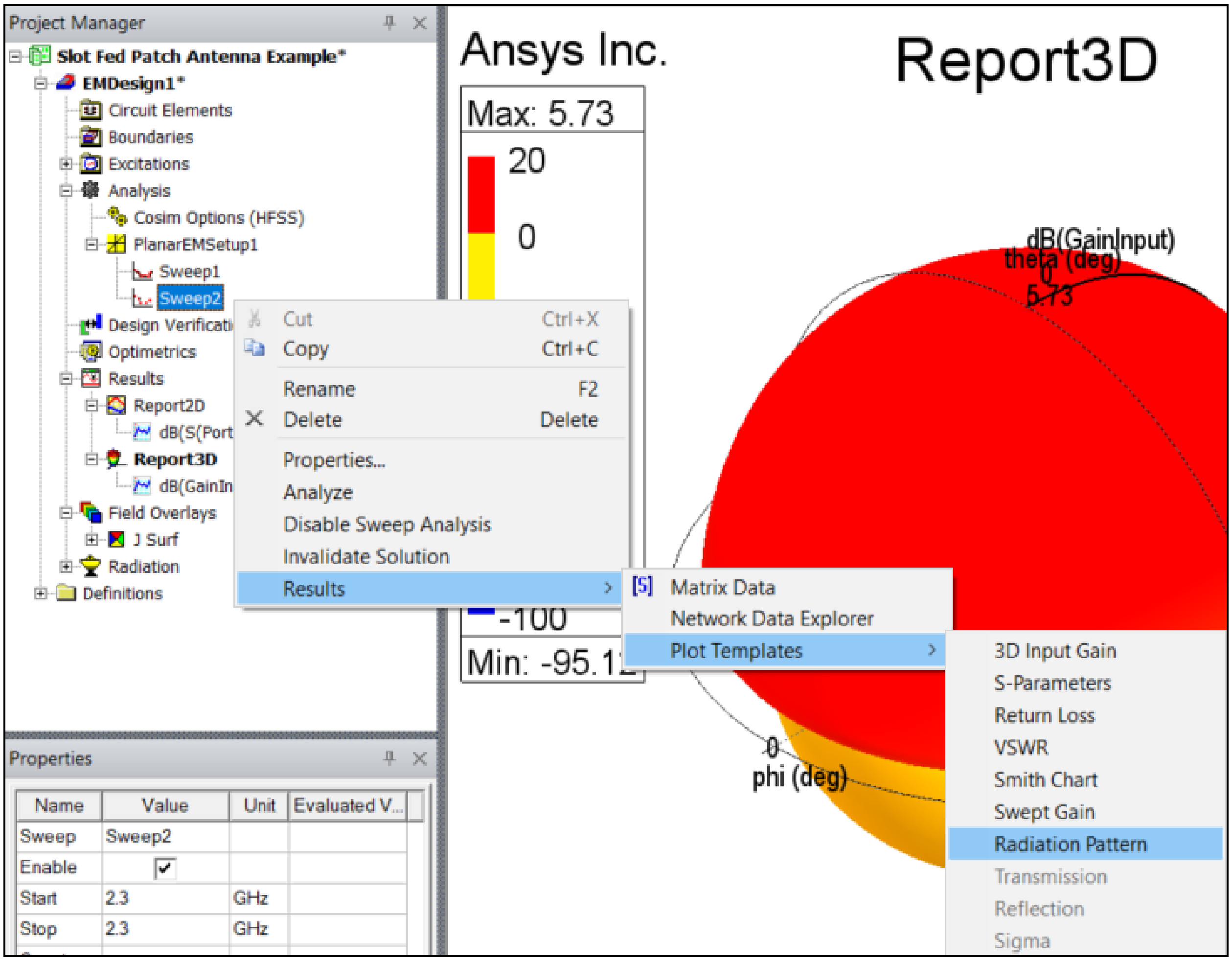
3. From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Radiation** and select **Insert Far Field Setup** > **Infinite Sphere** to open the **Infinite Sphere Setup** window.



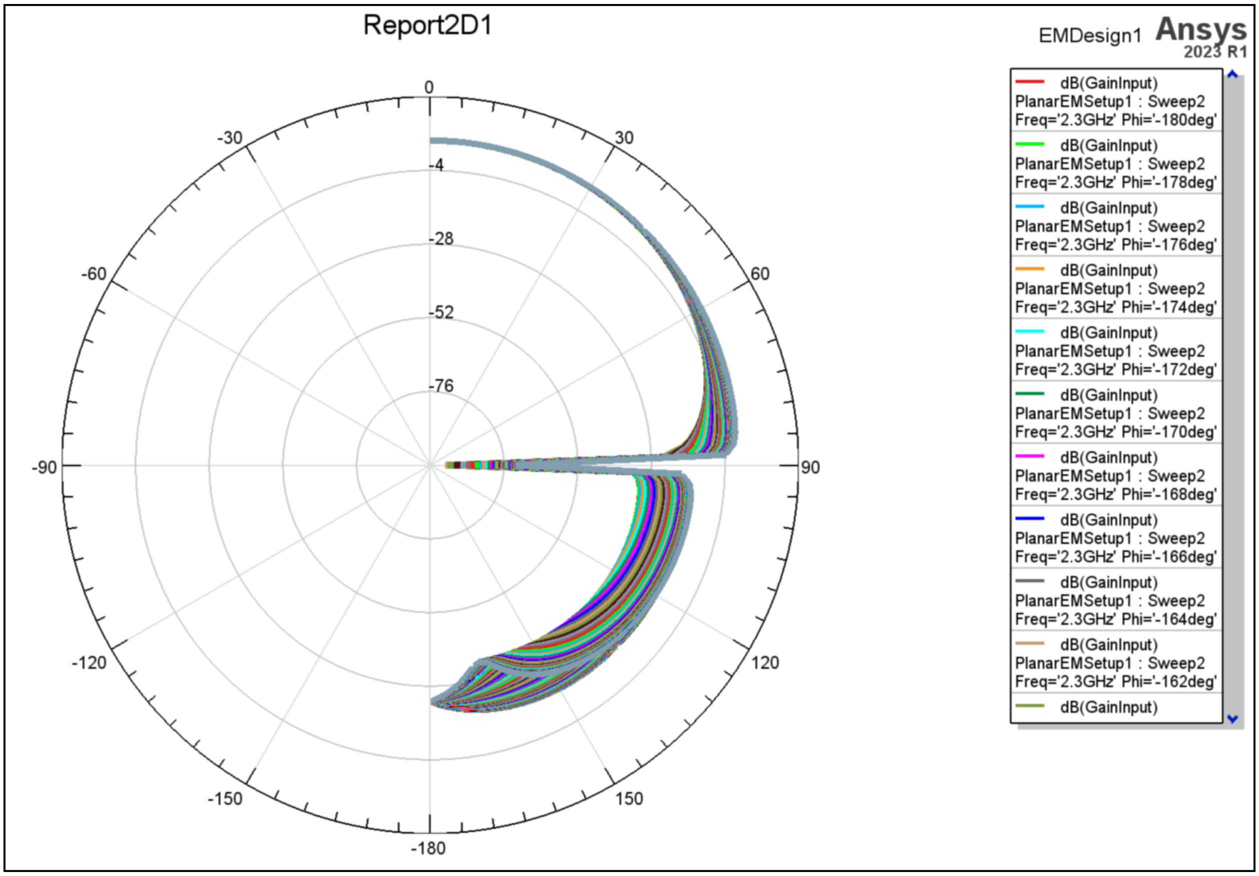
4. From the **Infinite Sphere Setup** window, make the following changes:
   a. From the **Phi** area:
      - Enter **0** in the **Start** field.
      - Enter **90** in the **Stop** field.
      - Enter **90** in the **Step Size** field.
   b. From the **Theta** area:
      - Enter **-180** in the **Start** field.



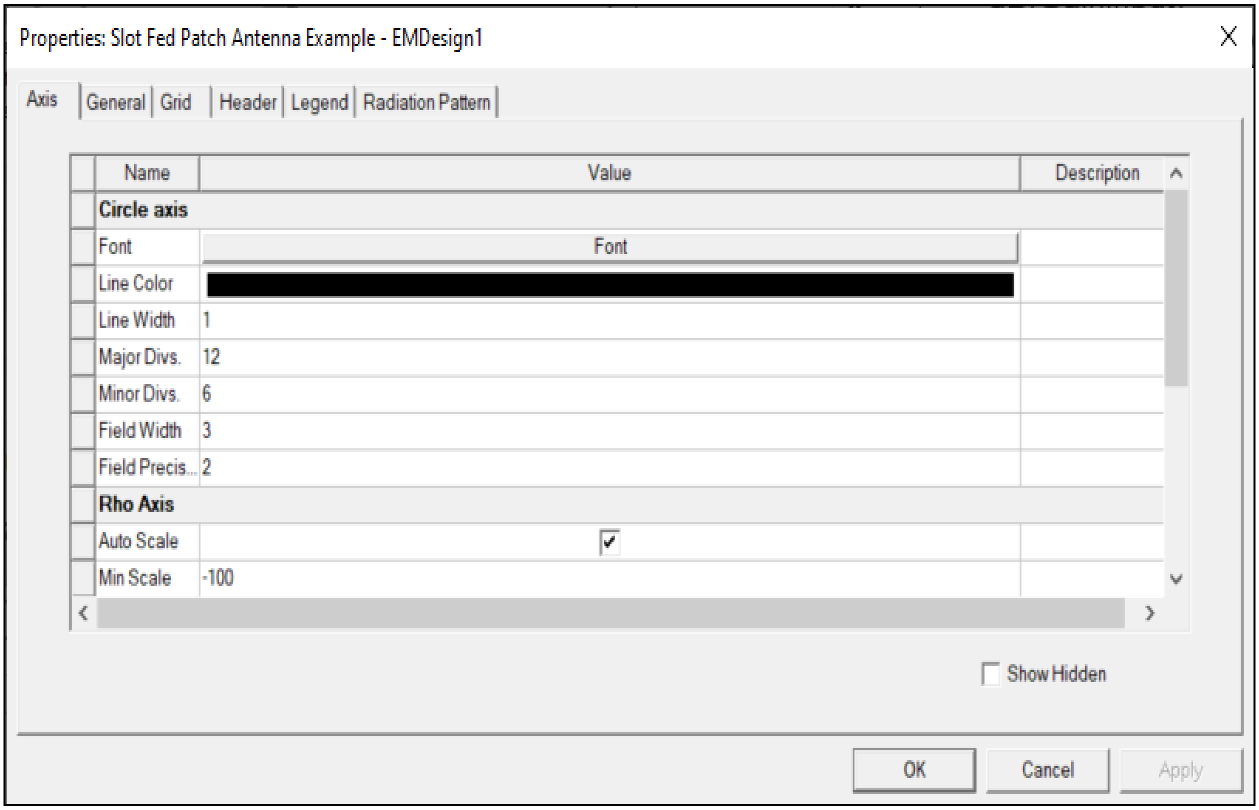
5. Click **OK** to close the **Infinite Sphere Setup** window.
6. From the **Project Manager** window, right-click **Sweep2** and select **Results** > **Plot Templates** > **Radiation Pattern**.



The new radiation pattern report appears (e.g., **Report2D1**).



7. Double-click the Theta axis (i.e., the outer circumference of the plot) to open the **Properties** window.



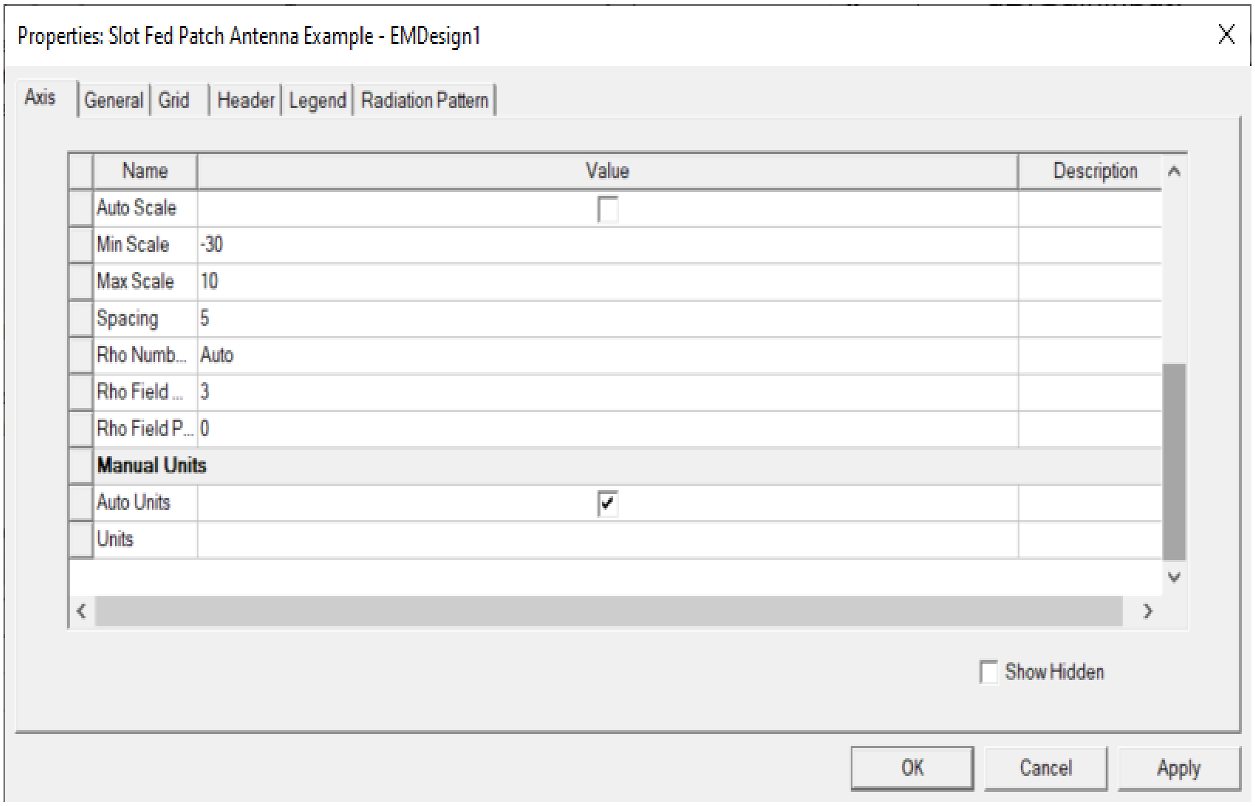
8. From the **Axis** tab, do the following:

    a. Remove the check from the box in the **Auto Scale** field.

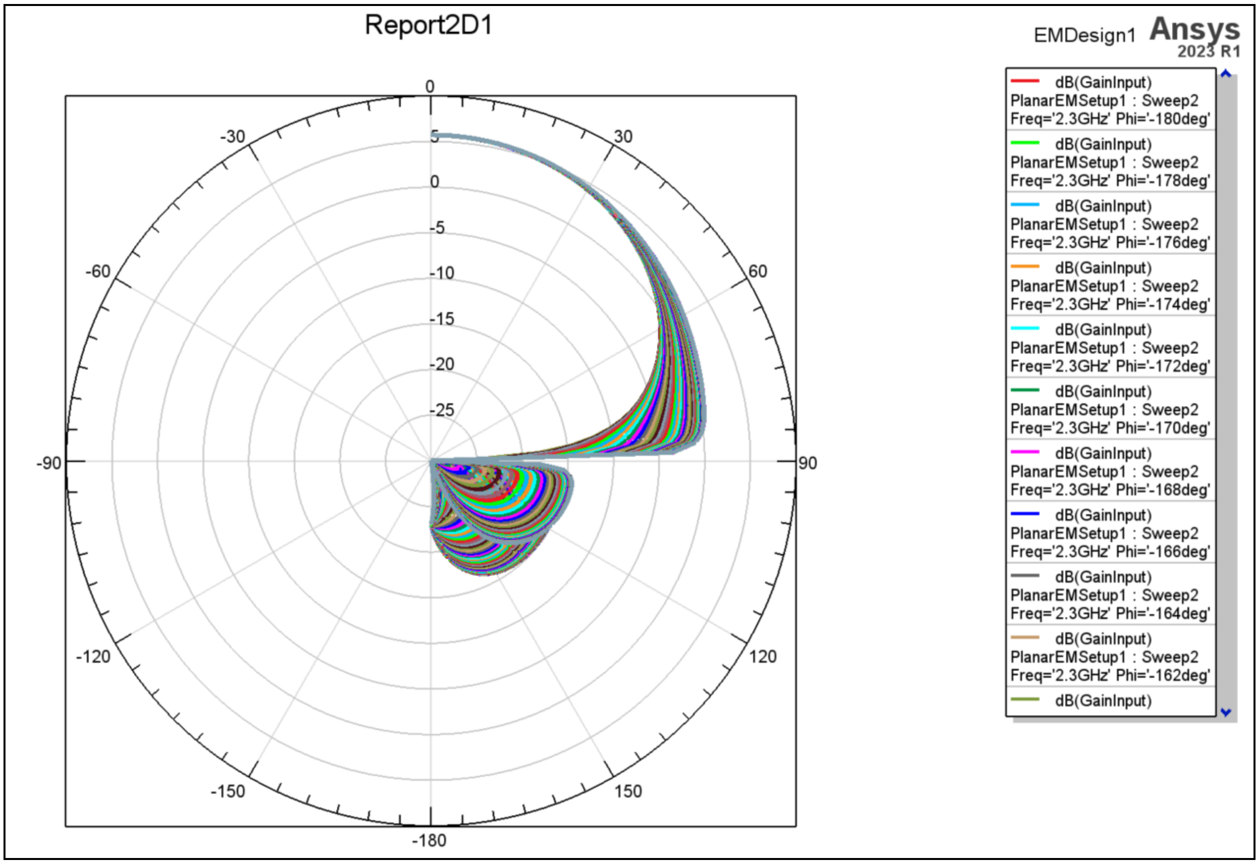
    b. Enter **-30** in the **Min Scale** field.

    c. Enter **10** in the **Max Scale** field.

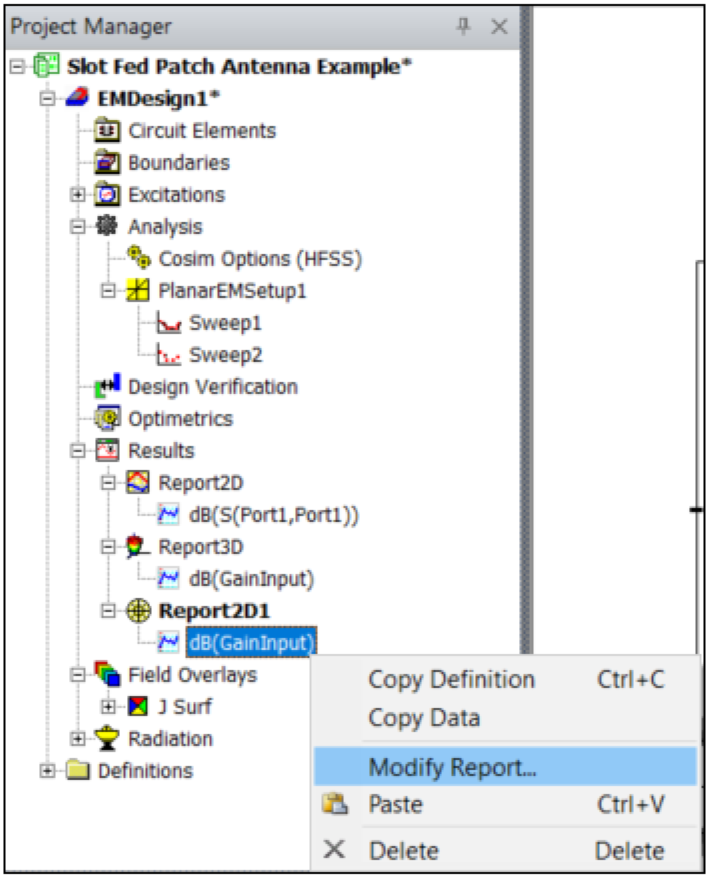
    d. Enter **5** in the **Spacing** field.


Properties: Slot Fed Patch Antenna Example - EMDesign1
Axis
General
Grid
Header
Legend
Radiation Pattern
Name
Value
Description
Auto Scale
Min Scale
-30
Max Scale
10
Spacing
5
Rho Numb...
Auto
Rho Field ...
3
Rho Field P...
0
Manual Units
Auto Units
Units
Show Hidden
OK
Cancel
Apply

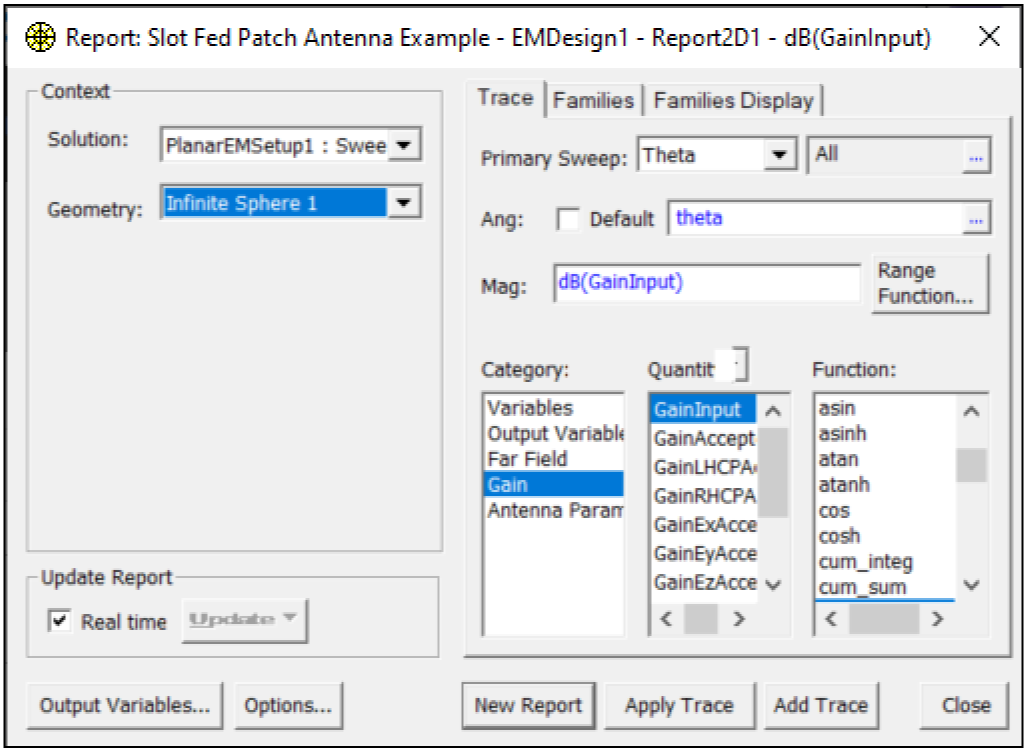
9. Click **Apply**. Then click **OK** to close the window.



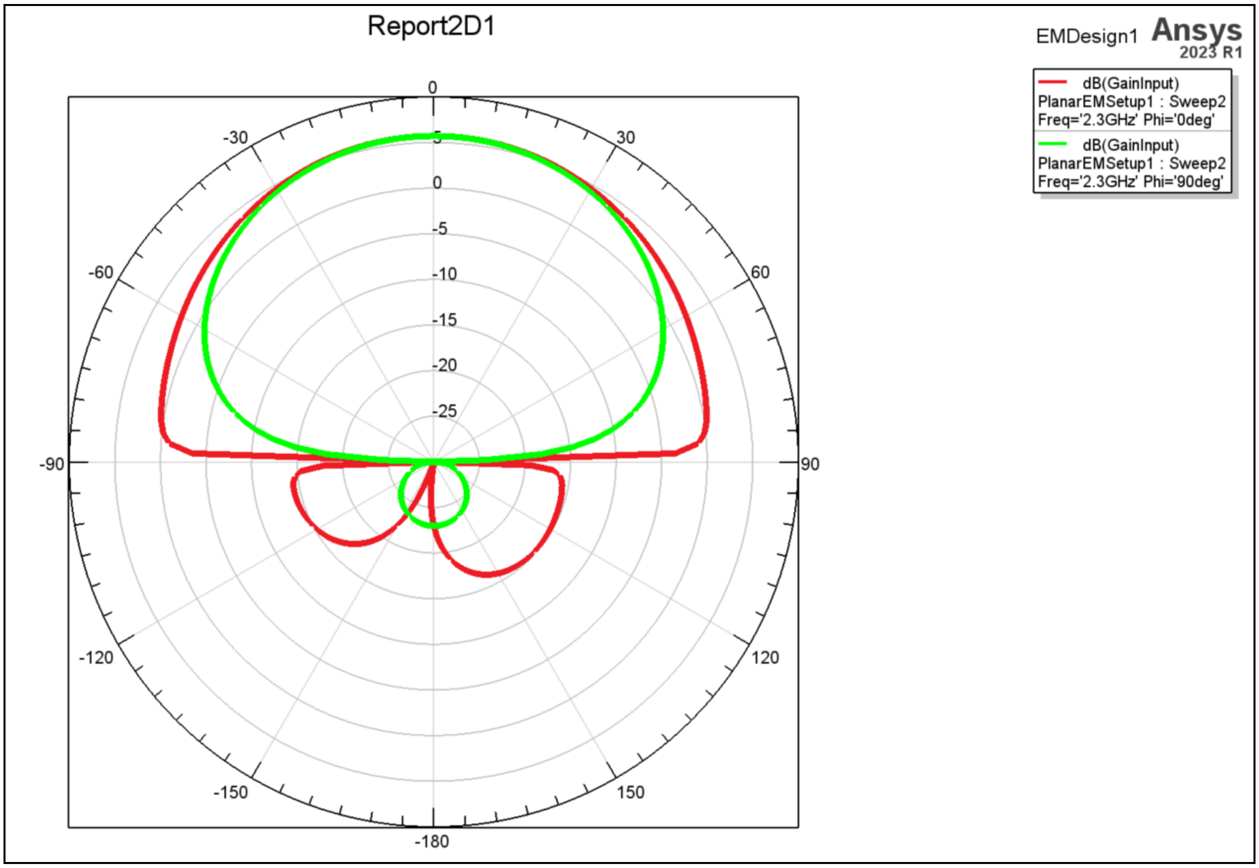
10. If your plot matches the preceding figure, or simply contains more than two traces and covers the range of Theta = 0 through 180 degrees, the plot is most likely based from the default infinite sphere setup. To correct for this, do the following:
    a. From the **Project Manager** window, right-click **dB(GainInput)** (i.e., **Report 2D1** > **dB(GainInput)**) and select **Modify Report** to open the **Report** window.


Project Manager
Slot Fed Patch Antenna Example*
EMDesign1*
Circuit Elements
Boundaries
Excitations
Analysis
Cosim Options (HFSS)
PlanarEMSetup1
Sweep1
Sweep2
Design Verification
Optimetrics
Results
Report2D
dB(S(Port1,Port1))
Report3D
dB(GainInput)
Report2D1
dB(GainInput)
Field Overlays
J Surf
Radiation
Definitions
Copy Definition Ctrl+C
Copy Data
Modify Report...
Paste Ctrl+V
Delete Delete

b. From the **Geometry** drop-down menu, select **Infinite Sphere 1**.



c. Click **Apply Trace** to simultaneously apply the adjustment to the current report.



d. Click **Close**.

Congratulations, the HFSS 3D Layout slot fed patch antenna getting started guide is complete.