Лабораторная работа №1

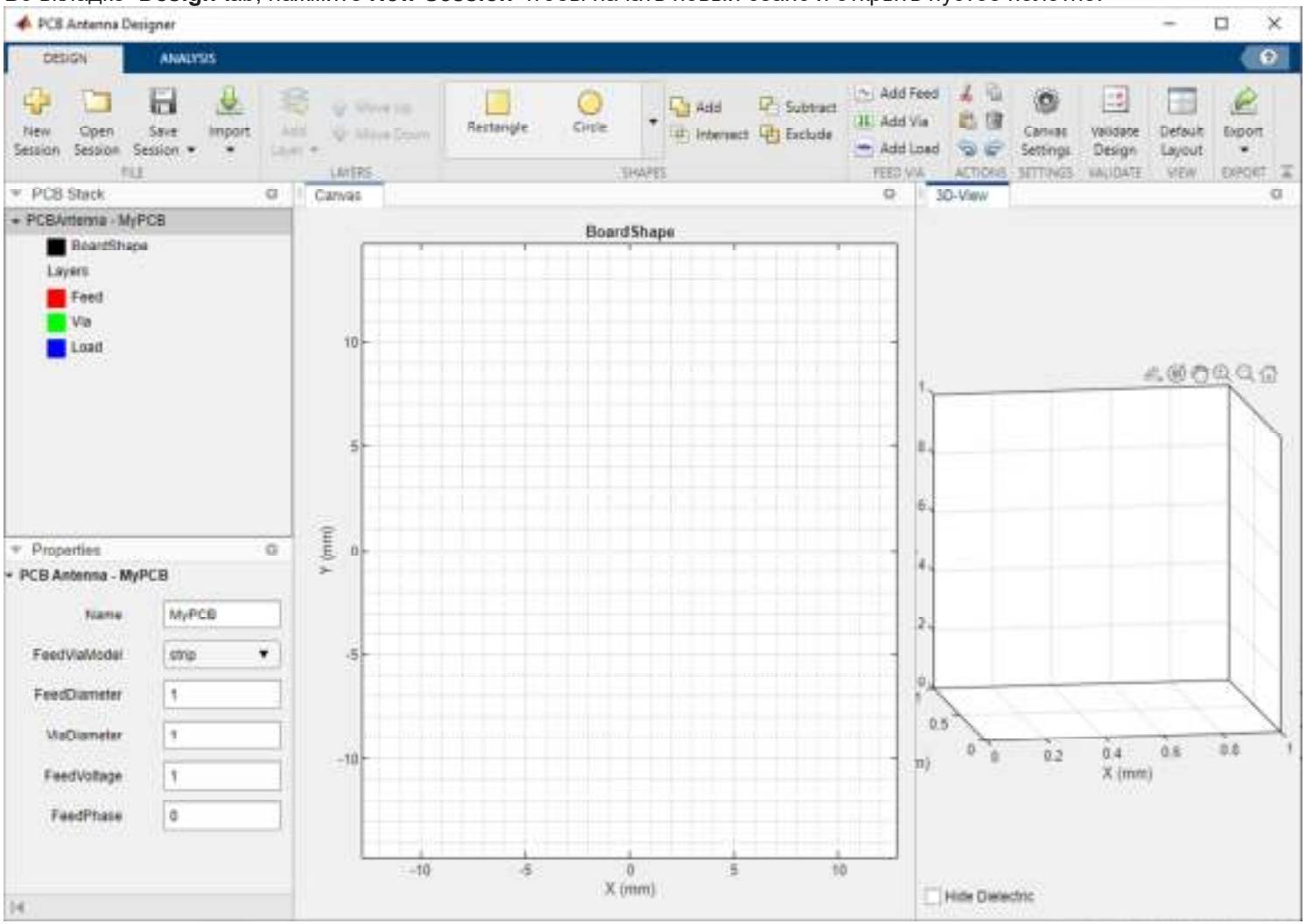
Разработка антенны на печатной плате в среде MATLAB

Откройте New Session

Введите данную команду с командной строке, чтобы запустить **PCB Antenna Designer** app.

```
pcbAntennaDesigner
```

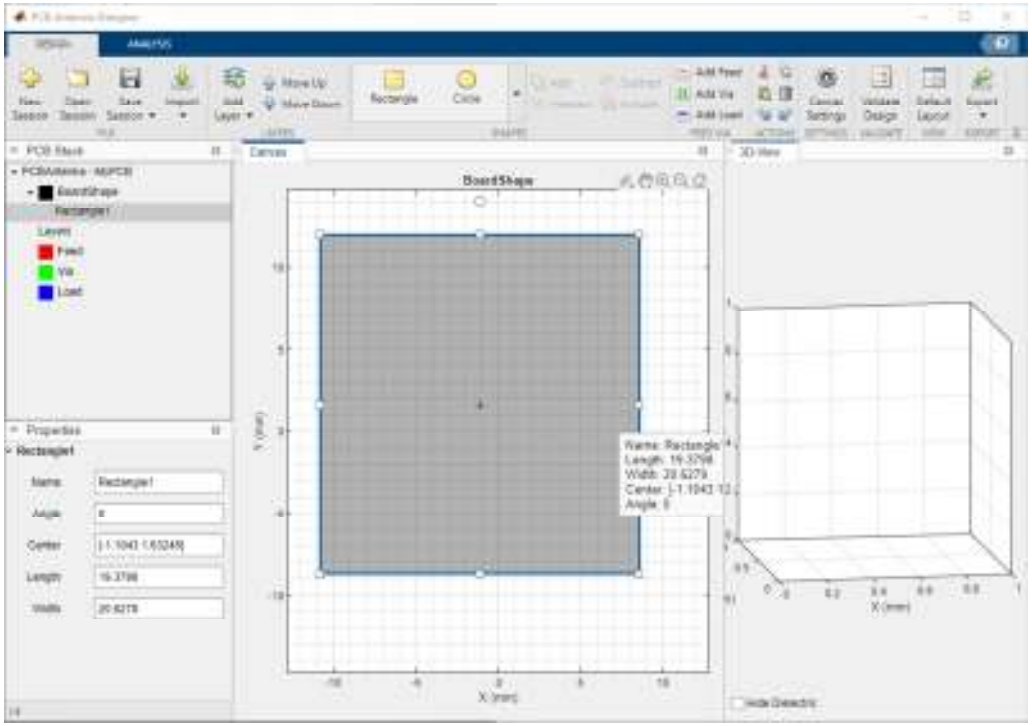
Во вкладке **Design** tab, нажмите **New Session** чтобы начать новый сеанс и открыть пустое полотно.



The default units for the canvas are millimeters (mm). The app uses global units. You can change the units by clicking the **Canvas Settings** button.

Define Board Shape

To design a PCB stack, you must first define a board shape. Select **Rectangle** from the `Shapes` section on the toolbar. Drag the shape on the canvas to create a rectangle.

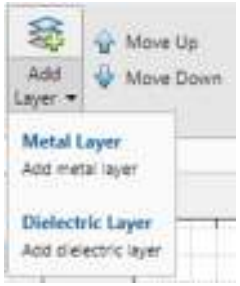


Change the properties of `Rectangle1` in the **Properties** pane to the following:

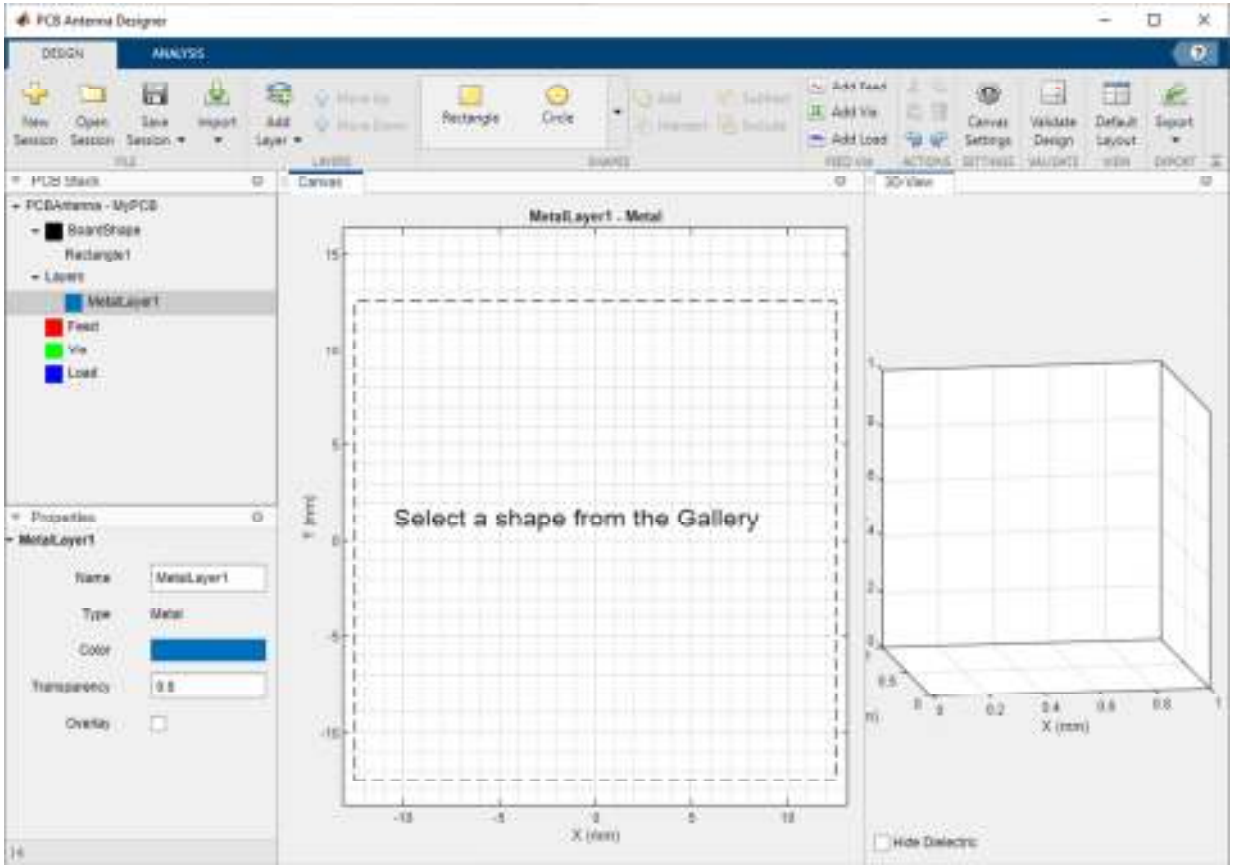
- **Center** — `[0,0]`
- **Length** — `25`
- **Width** — `25`

Add Ground Plane

Click **Add Layer** on the toolbar and then select **Metal Layer** to add a metal layer.



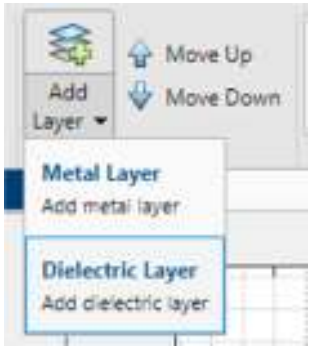
The board shape is represented as a dotted line on the canvas.

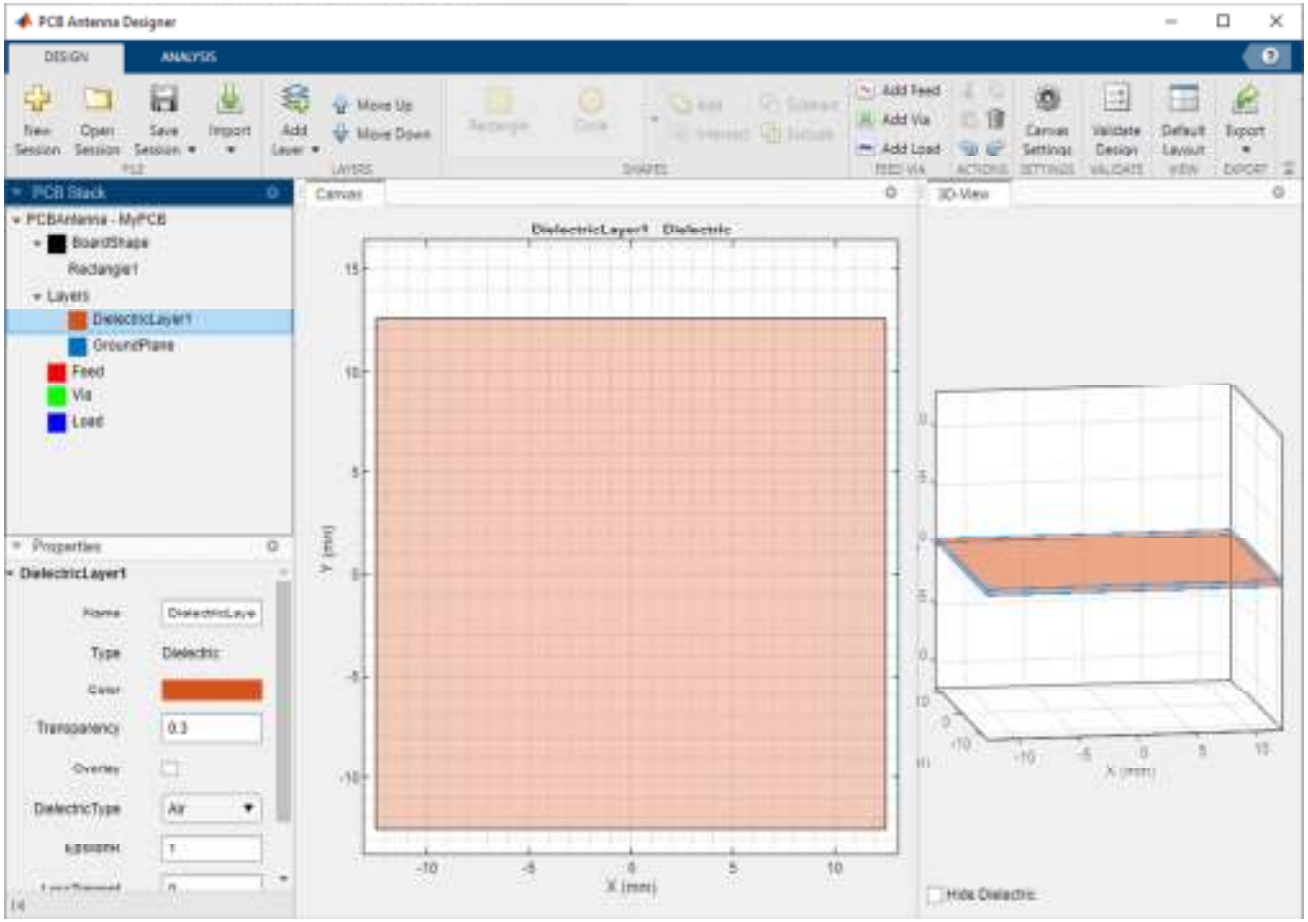


Set the **Name** of the metal layer to `GroundPlane`.

Add Dielectric Layer

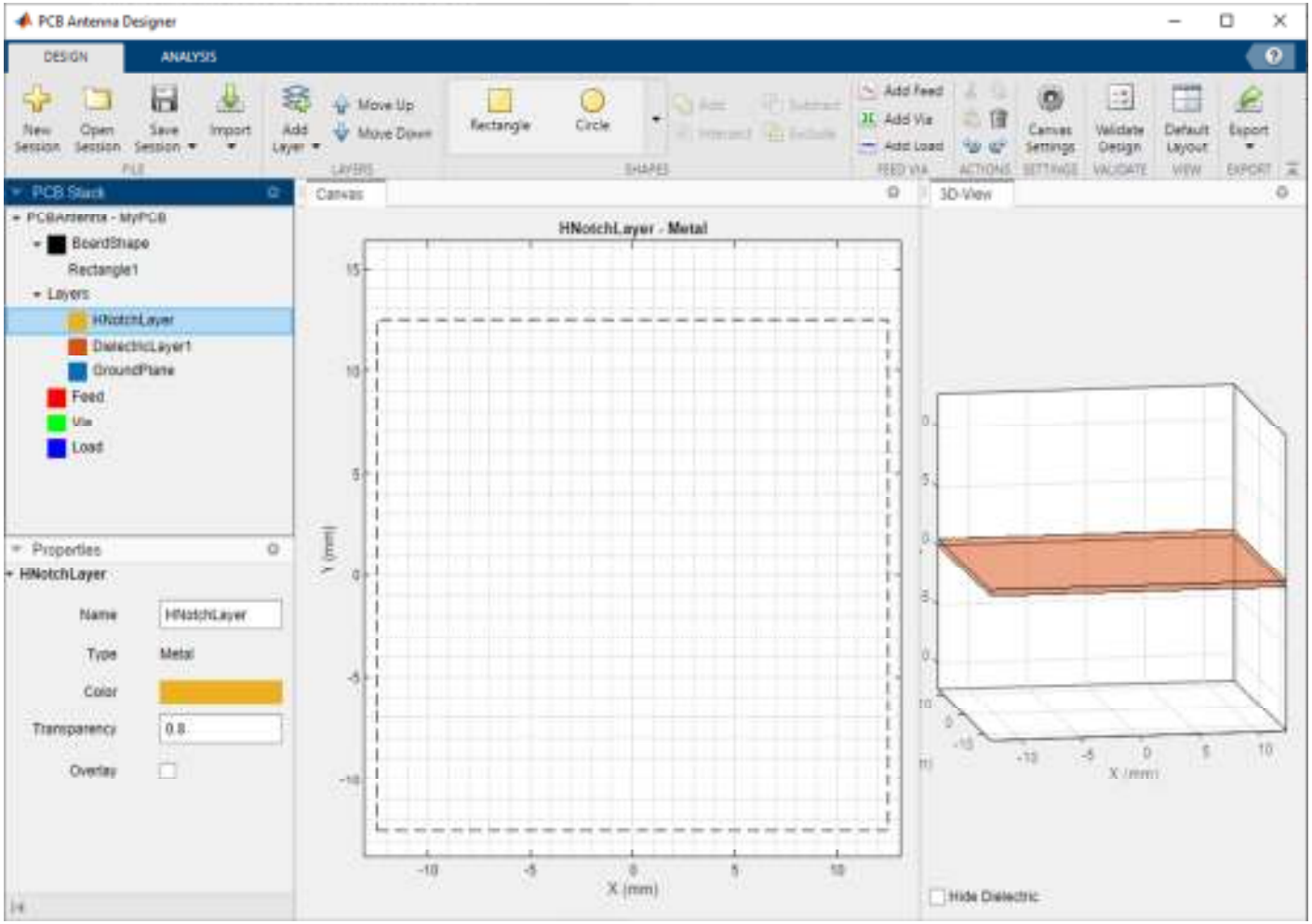
Click **Add Layer** on the toolbar and then select **Dielectric Layer** to add a dielectric layer `DielectricLayer1`.





Add Metal Patch Layer

Click **Add Layer** on the toolbar and then select **Metal Layer** to add a metal layer. Set the name of this layer to `HNotchLayer`.



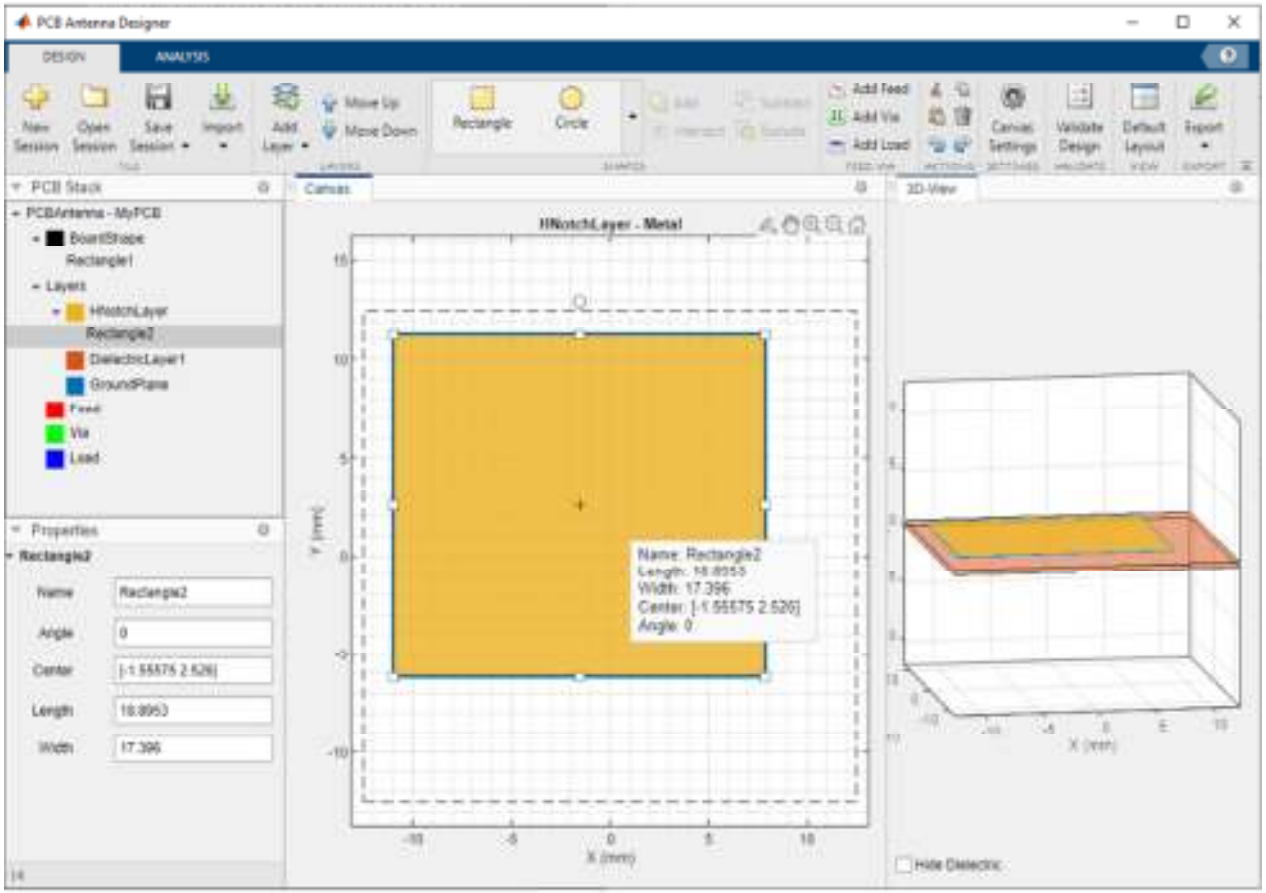
Move Layers

You can move the layers using the **Move Up** and **Move Down** buttons on the toolbar.

This example shows how to design an unit element for a 1-by-2 H-notch linear array. You design the H-notch unit element by subtracting the top and bottom notches from the substrate.

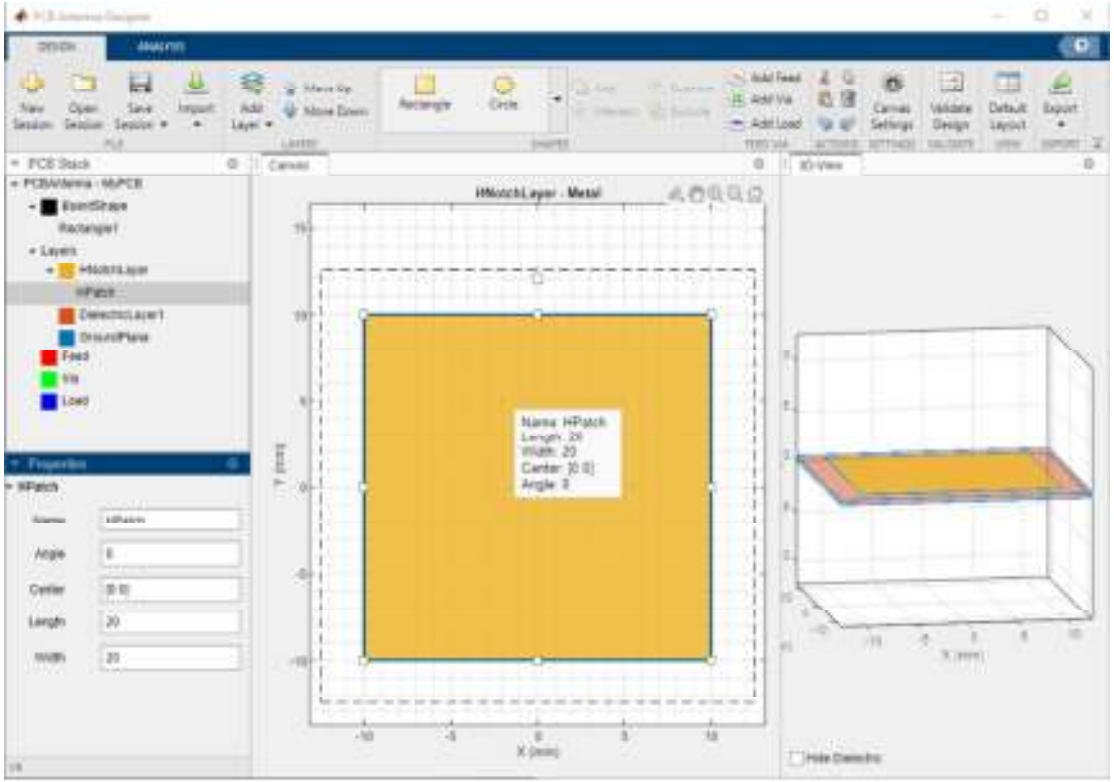
Create H-Notch

Select the HNotchLayer on the PCB Stack navigation tree and then select Rectangle from the Shapes section on the toolbar. Drag the shape onto the canvas to create a rectangular patch.



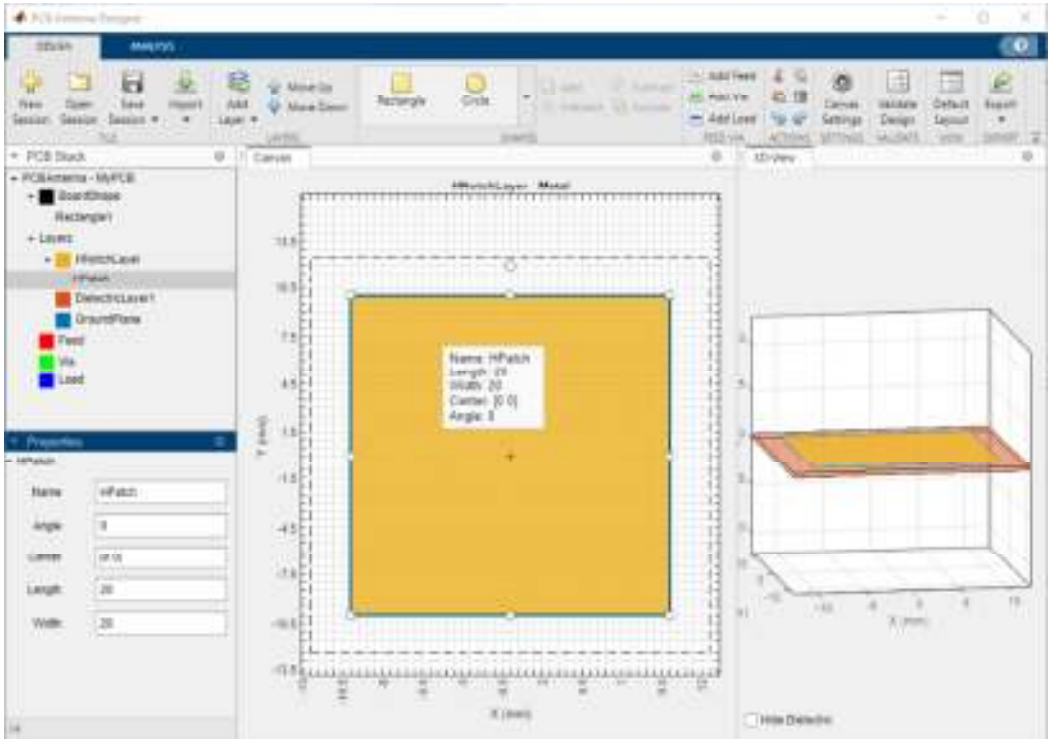
Set the properties of Rectangle2 to the following:

Name — HPatch
Center — [0,0]
Length — 20
Width — 20



Click the Canvas Settings button and select Snap to grid to allow easier and more accurate drawing. Then set the Grid Size to 0.5 mm.

The app displays the drawn shape.



Create Top Notch

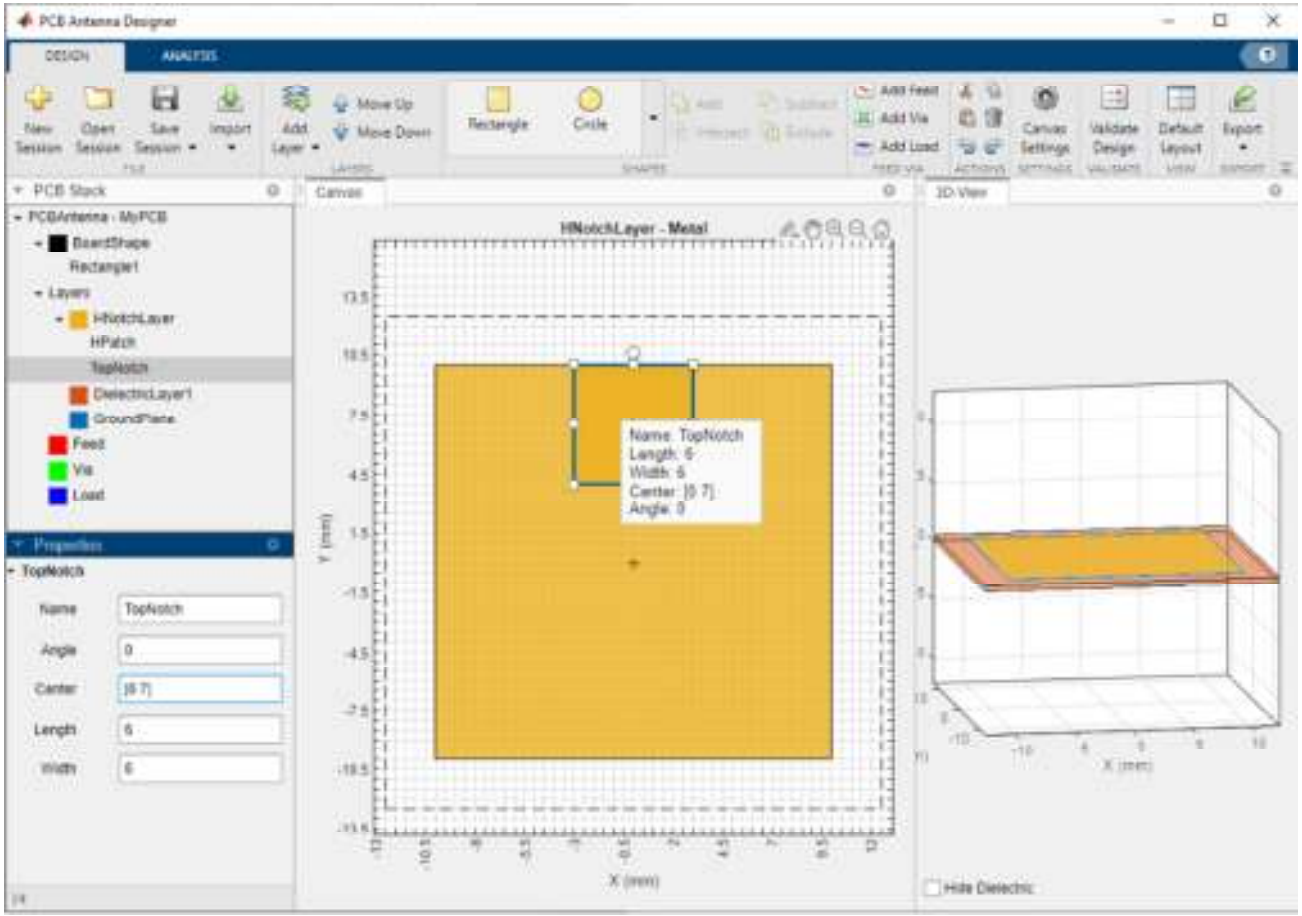
Select Rectangle from the Shapes section and drag the shape to the top of the shape to create a top notch in the patch layer. Set the properties of Rectangle3 to the following:

Name — TopNotch

Center — [0,7]

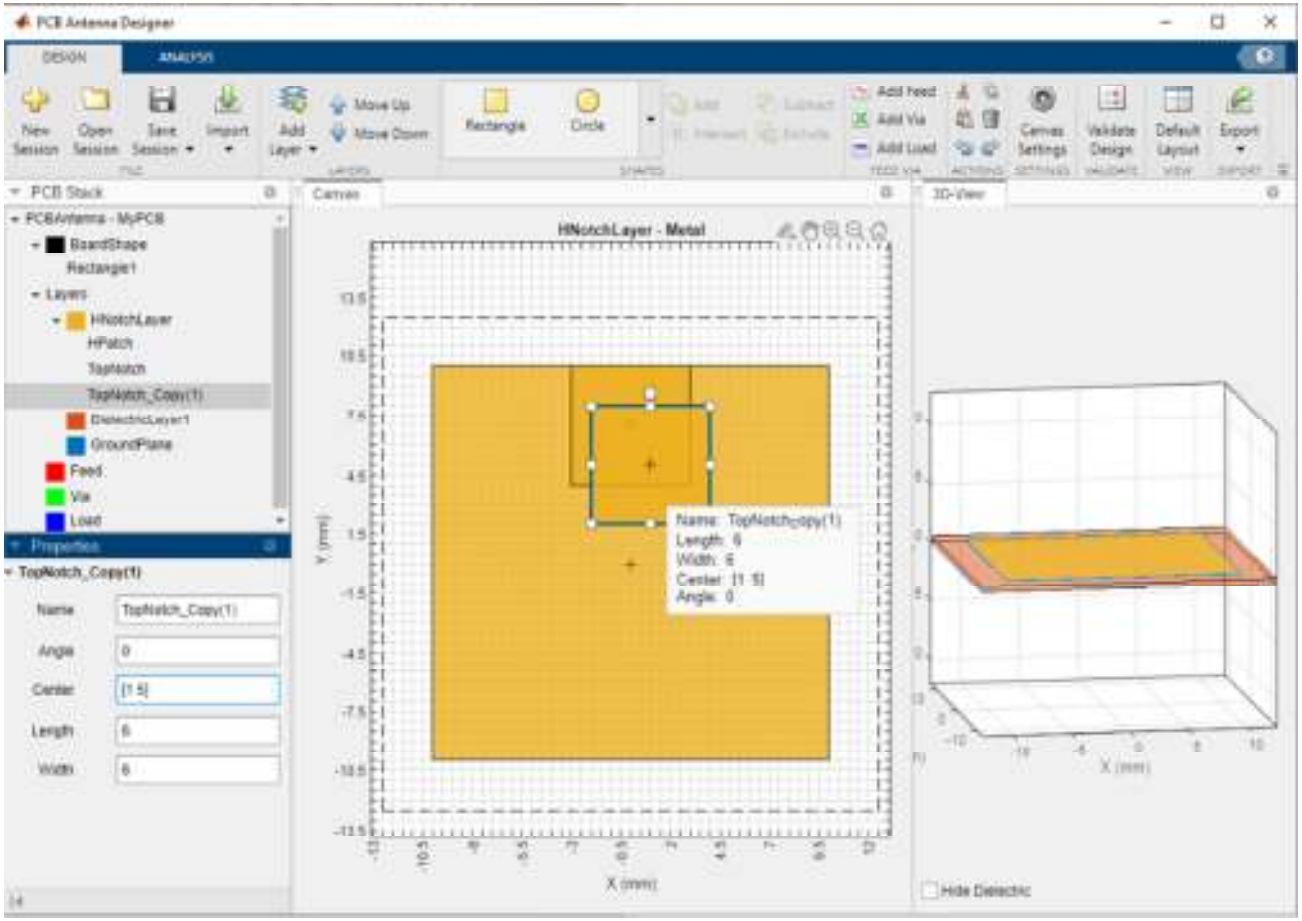
Length — 6

Width — 6


PCB Antenna Designer
DESIGN
ANALYSIS
New Session
Open Session
Save Session
Import
Add Layer
Move Up
Move Down
Rectangle
Circle
Add Feed
Add Via
Add Load
Canvas Settings
Validate Design
Default Layout
Export
PCB Stack
PCBAntenna - MyPCB
BoardShape
Rectangle1
Layers
HNotchLayer
HPatch
TopNotch
DielectricLayer1
GroundPlane
Feed
Via
Load
Properties
TopNotch
Name
TopNotch
Angle
0
Center
[0 7]
Length
6
Width
6
Canvas
HNotchLayer - Metal
Name: TopNotch
Length: 6
Width: 6
Center: [0 7]
Angle: 0
Y (mm)
X (mm)
3D View
Hide Dielectric

Create Bottom Notch

Select TopNotch from the canvas and then copy the layer by clicking Copy in the Actions section. Click Paste to paste a copy of TopNotch. Doing so creates a rectangle TopNotch_Copy_1 with identical dimensions to the TopNotch layer.



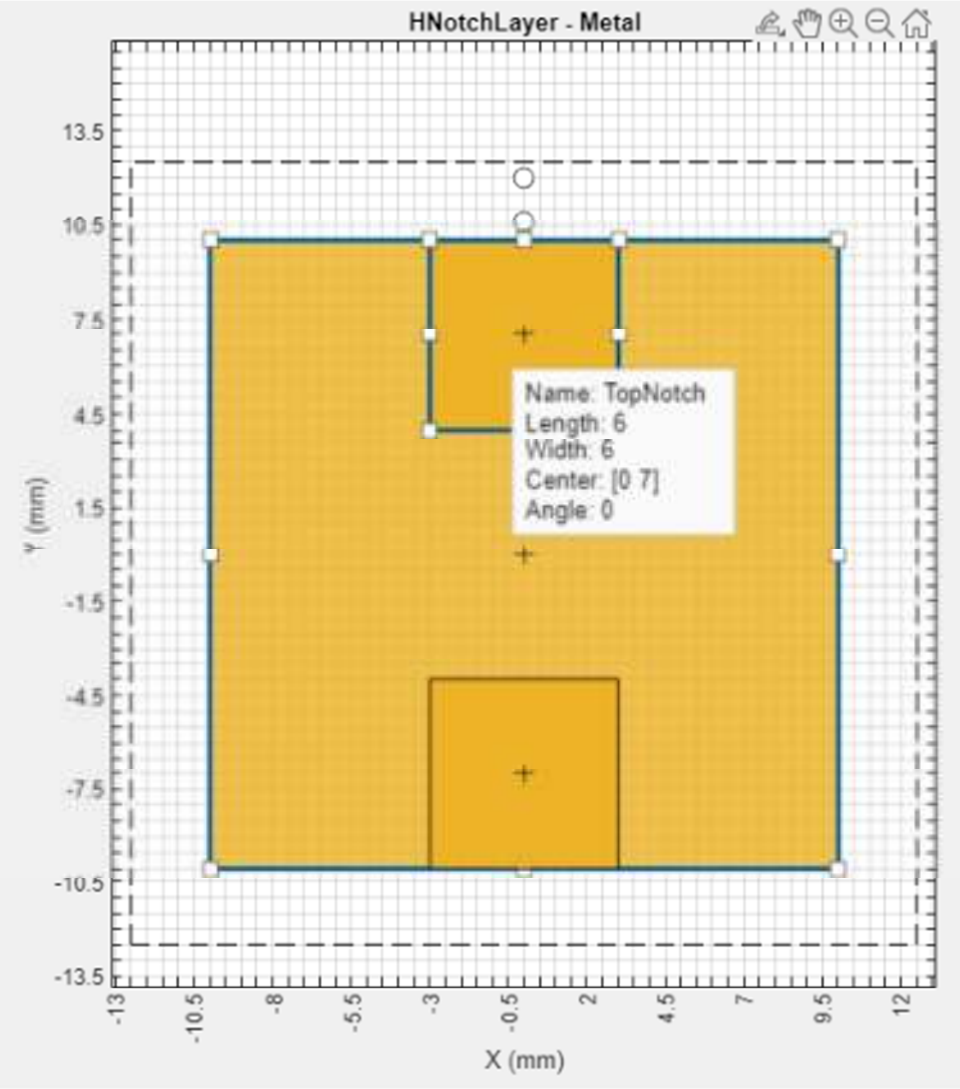
Set the properties of TopNotch_Copy_1 to the following:

Name — BottomNotch

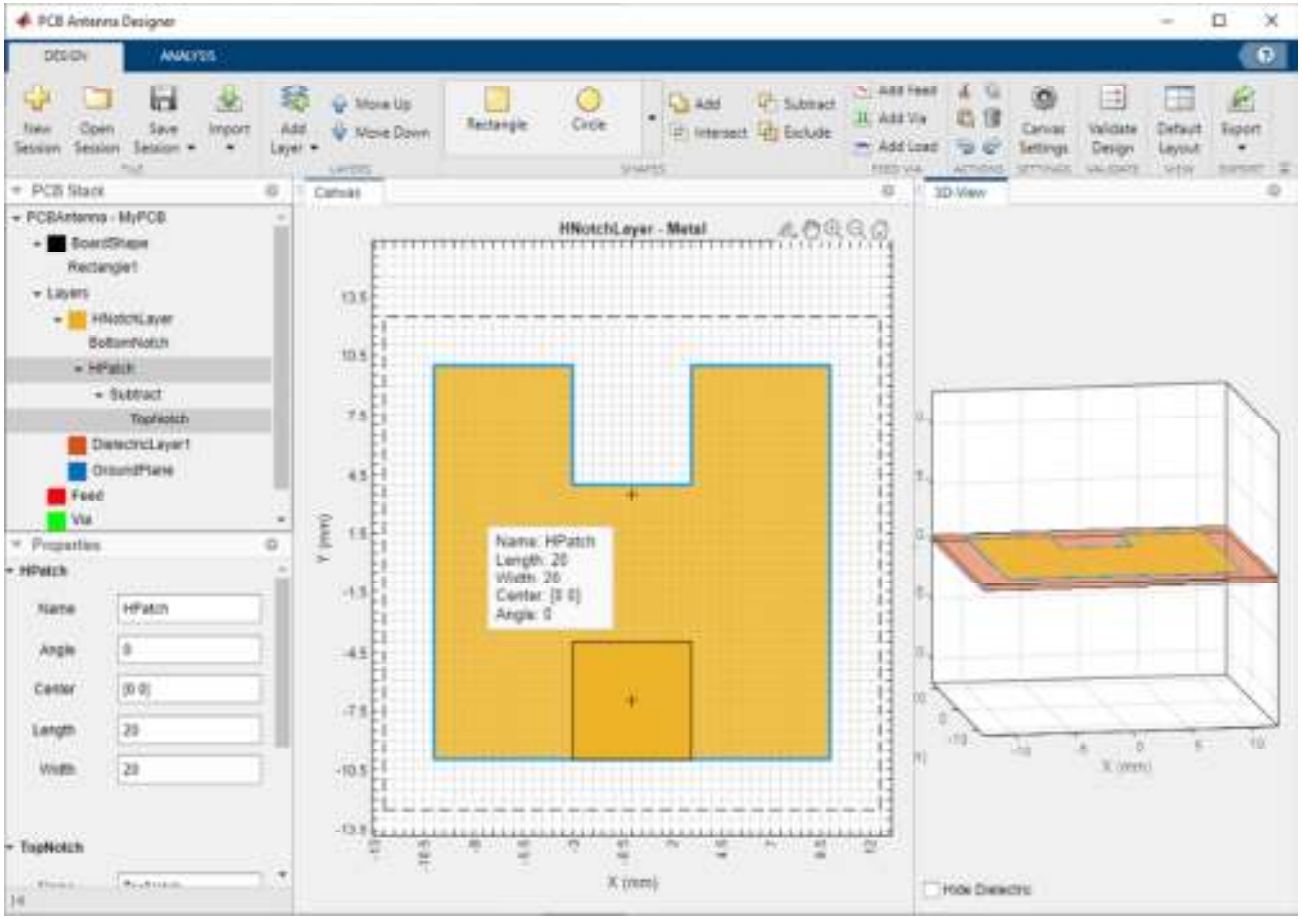
Center — [0,-7]

Subtract Top and Bottom Notch Layers from Metal Patch

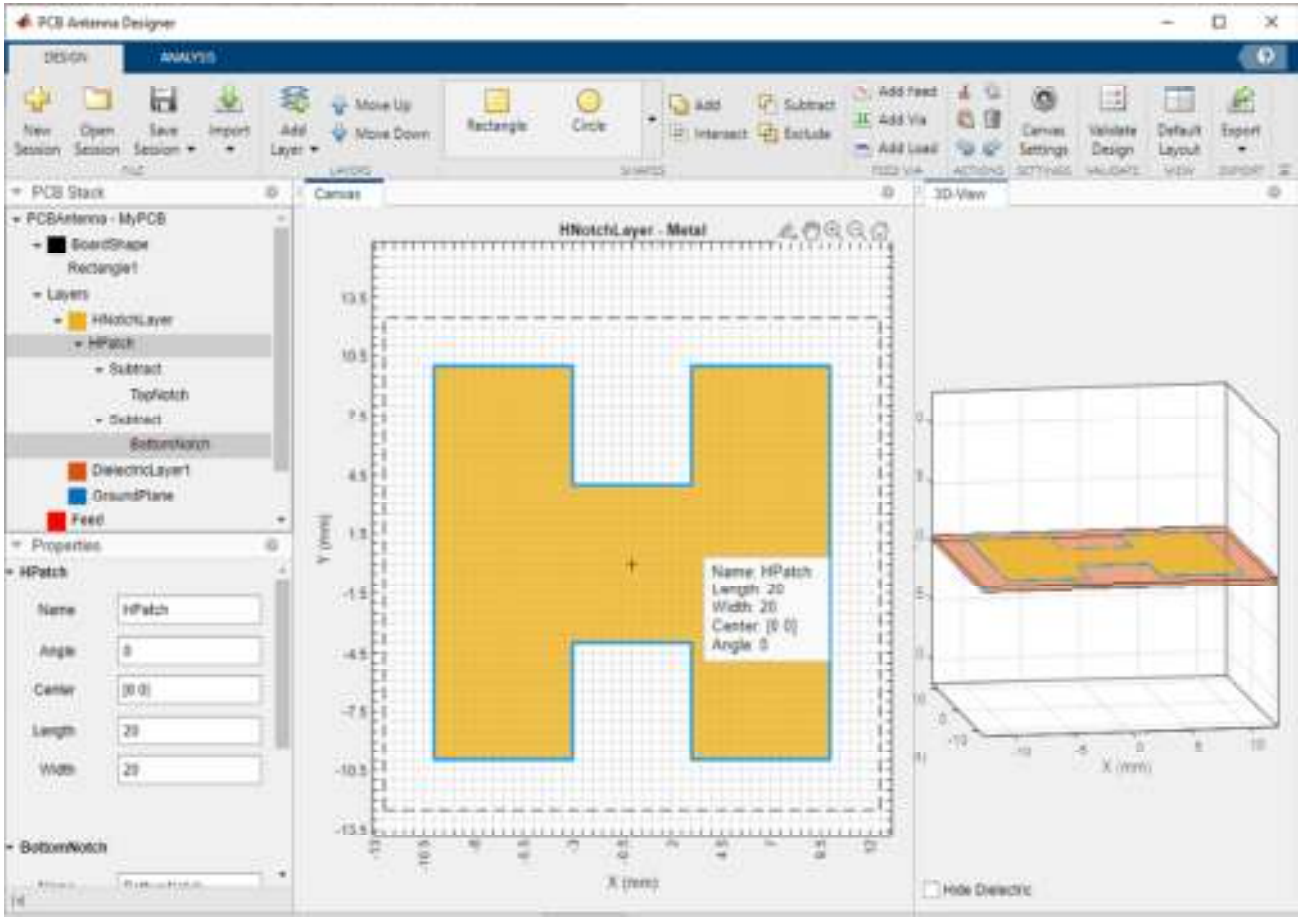
Select HPatch and TopNotch and then click Subtract in the Shapes section of the toolbar to remove both rectangles.



The app updates the HPatch layer.

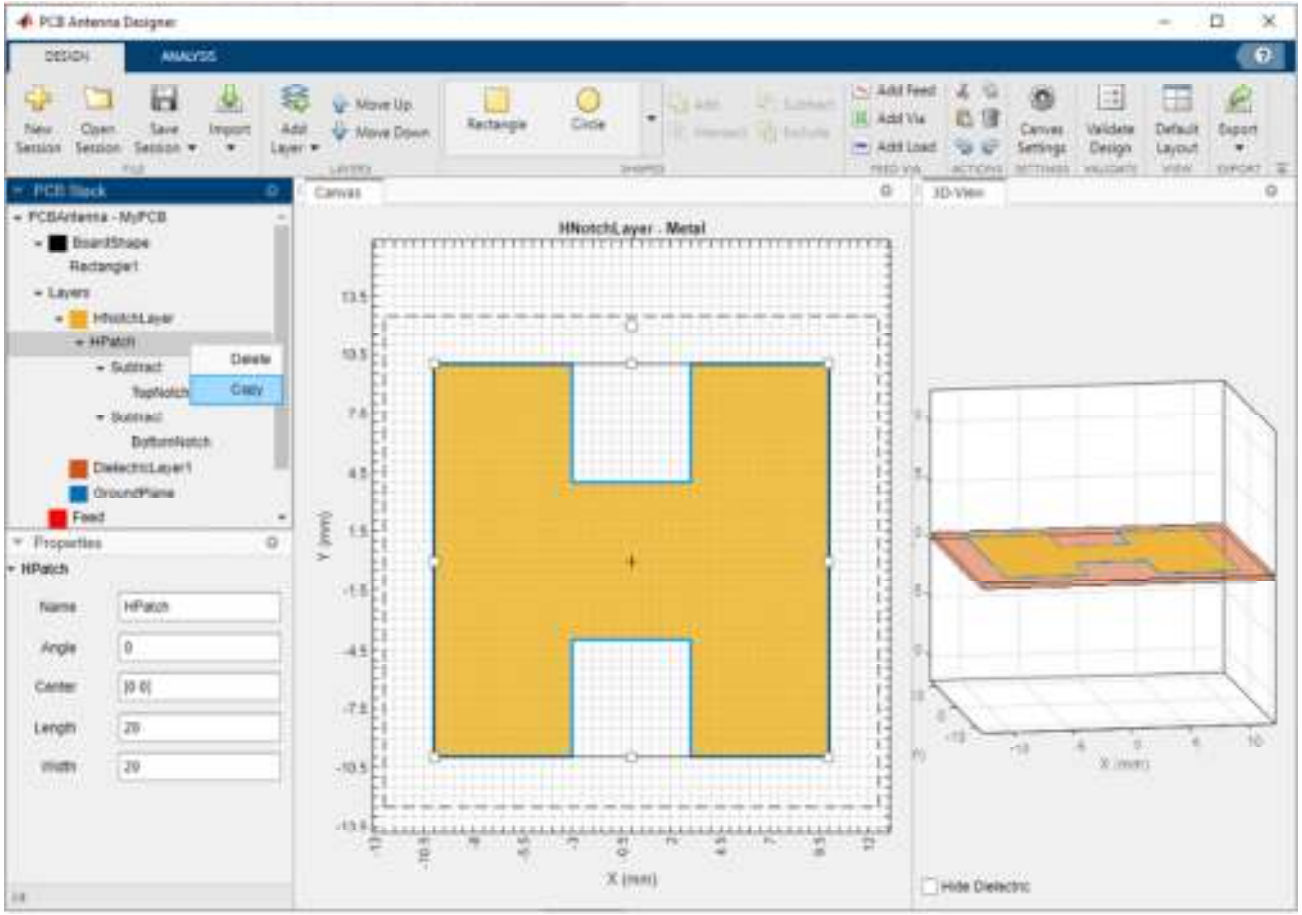


Select the HPatch and BottomNotch and click Subtract in the Shapes section of the toolbar to remove both rectangles.

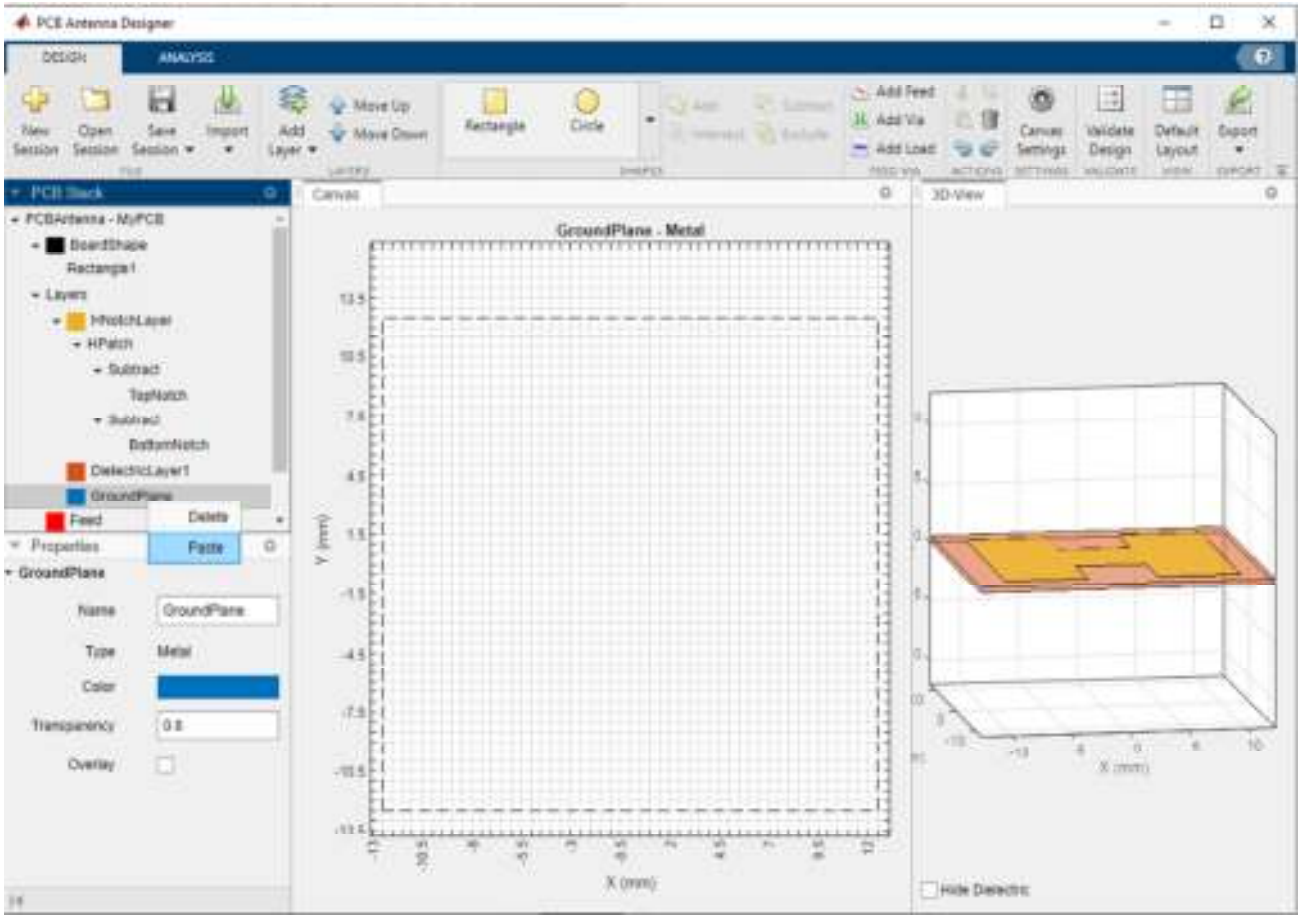


Set Dimensions for Ground Plane

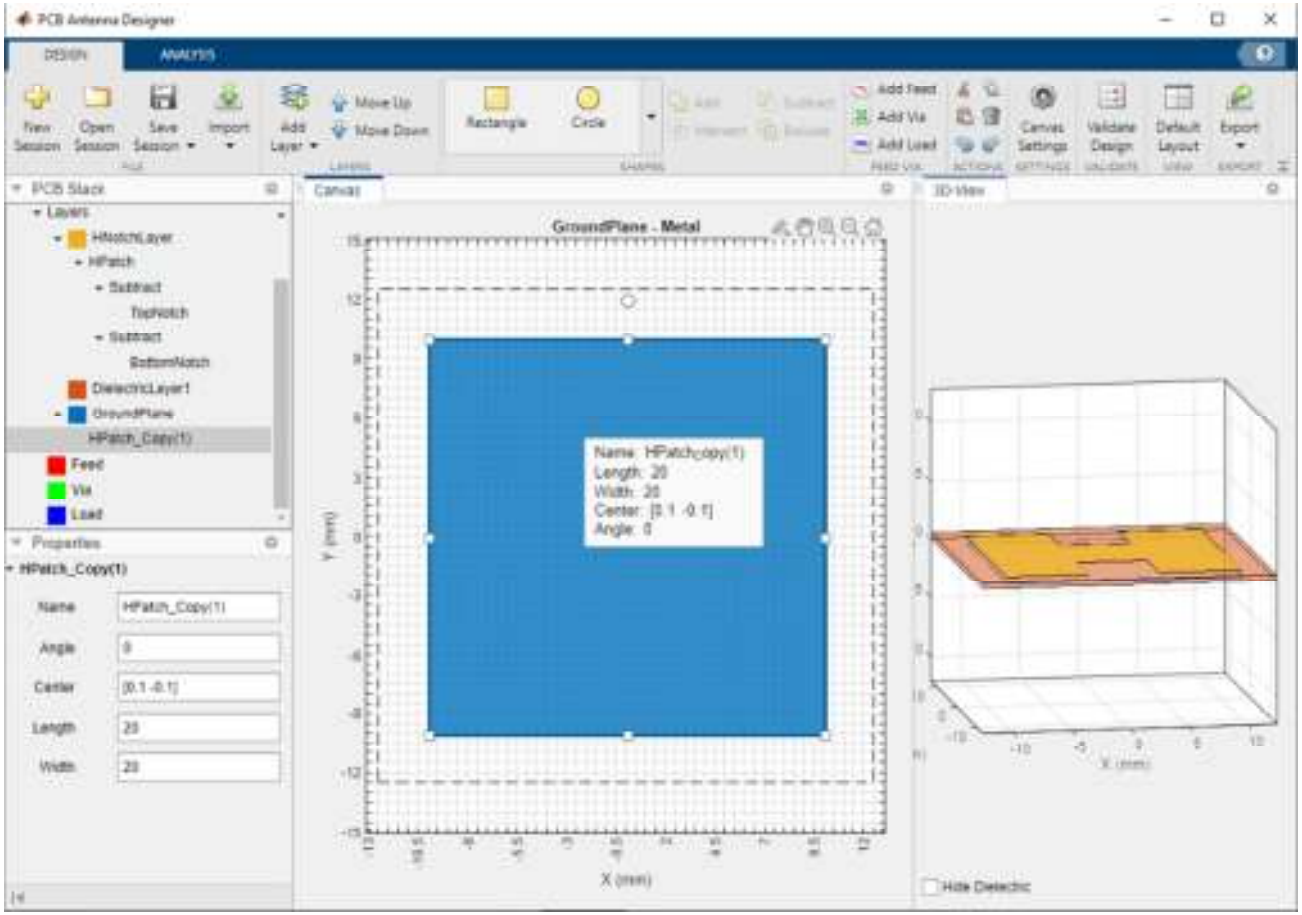
The ground plane must be same size as the HNotchLayer. To make the ground plane the same size as HPatch, right-click HPatch and select Copy.



Right-click the GroundPlane and click Paste on the toolbar to paste the rectangle in the GroundPlane.



The app copies the rectangle to the ground plane layer with the name HPatch_Copy(1). HPatch_Copy(1) and HPatch have the same dimensions.



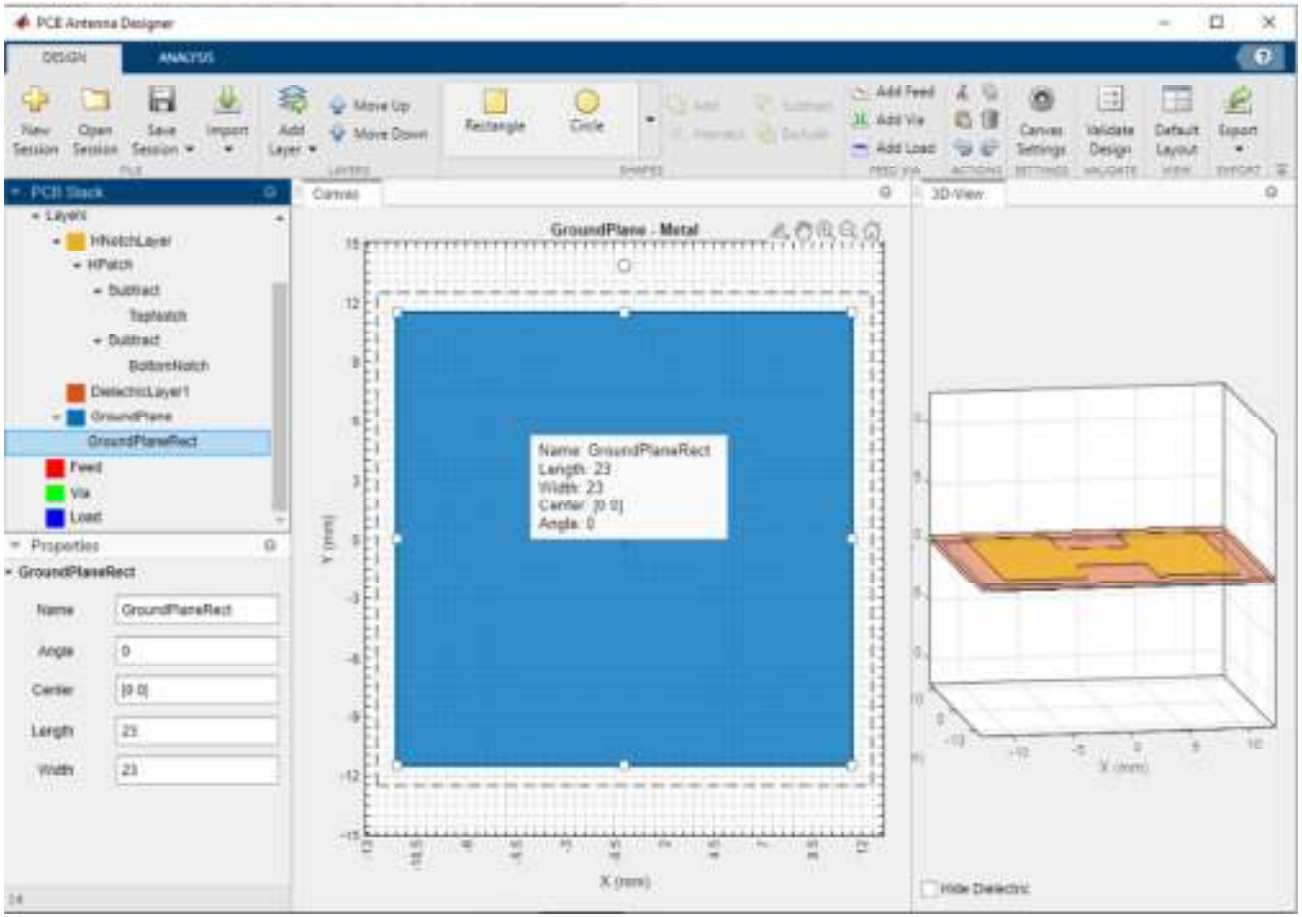
Set the properties of HPatch_Copy(1) to the following:

Name — GroundPlaneRect

Center — [0,7]

Length — 23

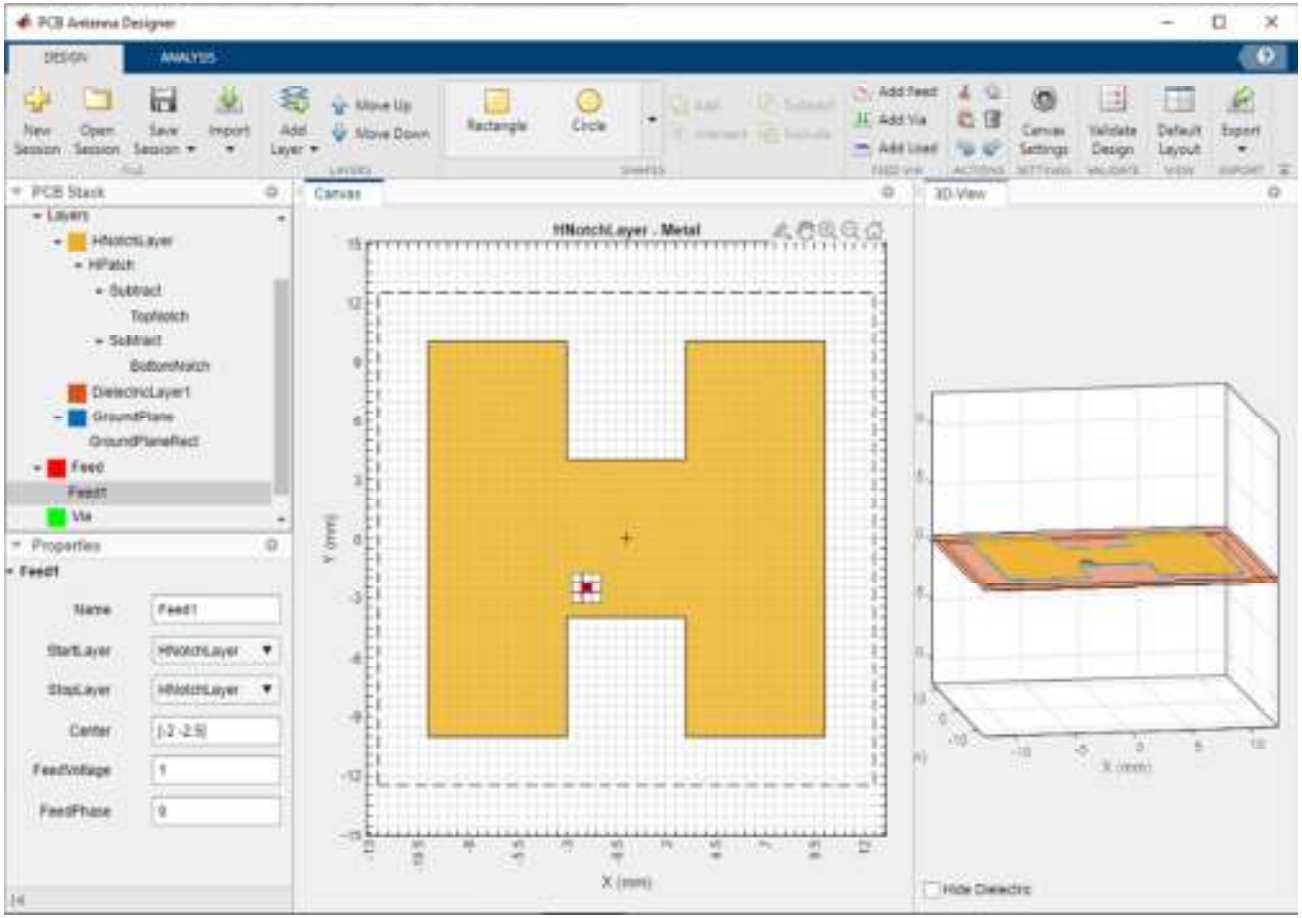
Width — 23



Add Feed

Select the HNotchLayer metal layer on the PCB Stack navigation tree and then click Add Feed in the Feed Via section on the toolbar. You can add a feed only to a metal layer.

As the PCB Stack navigation tree shows, the app adds the feed to HNotchLayer as Feed1.

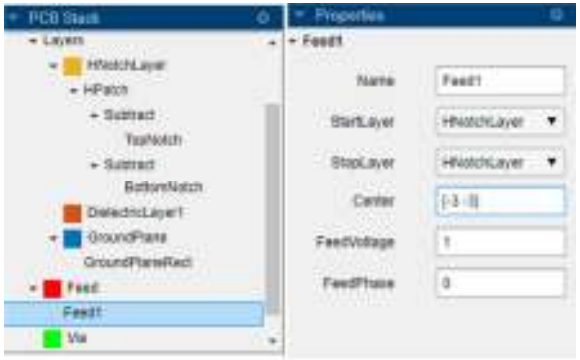


Change Properties of Feed

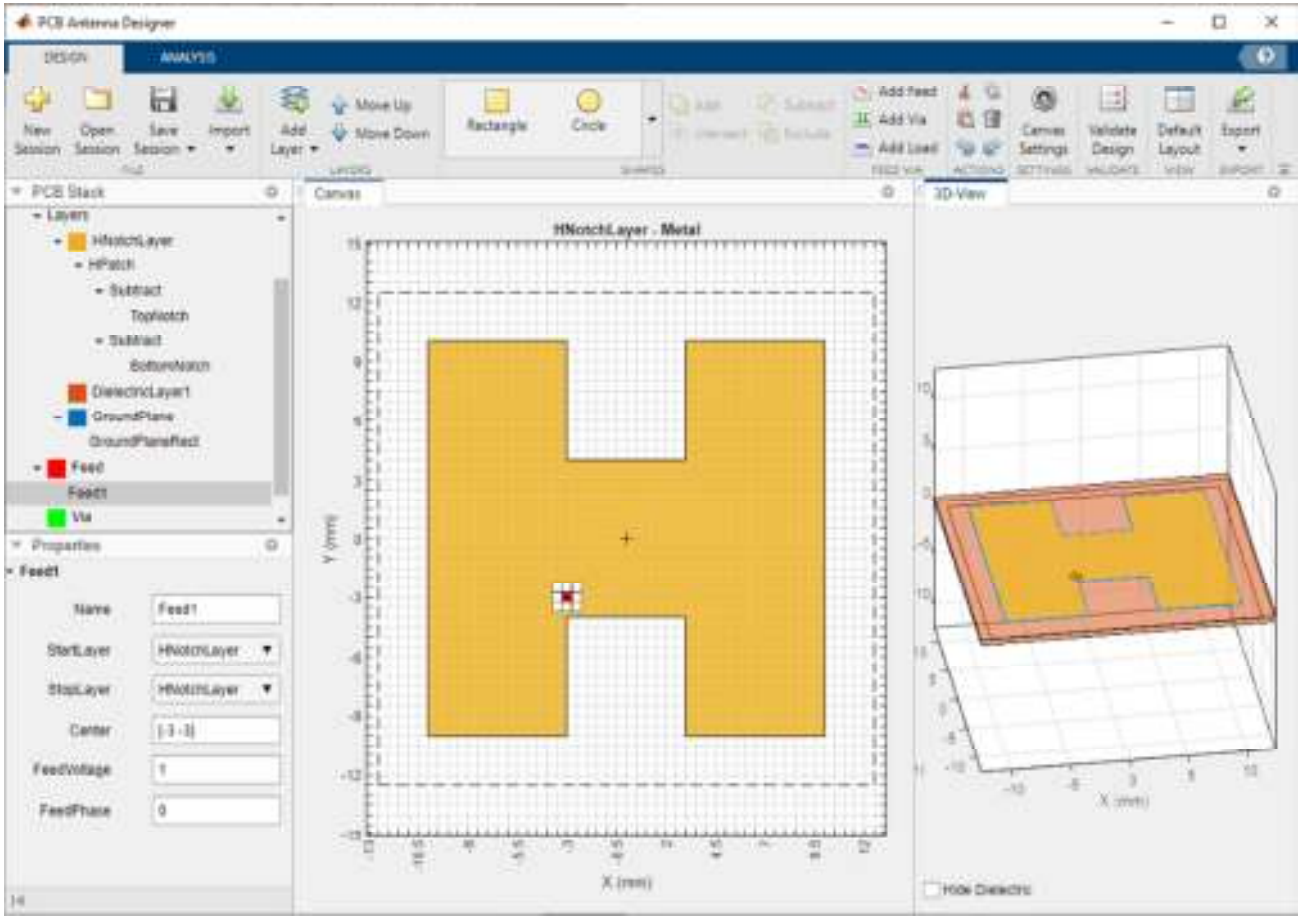
Select the parent Feed node on the PCB Stack navigation tree, and then set the FeedDiameter to 1 and FeedViaModel to Strip. FeedDiameter is a global property.

Change Position of Feed

Select Feed1 on the PCB Stack navigation tree. In the Properties pane, set Center to [-3,-3].

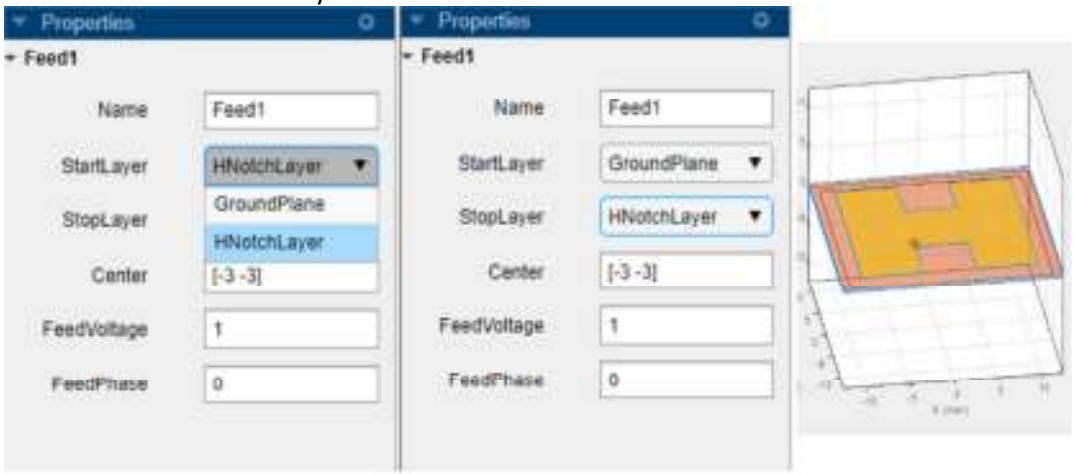


In the 3D-View pane, you can see that the feed is connected only to HNotchLayer.



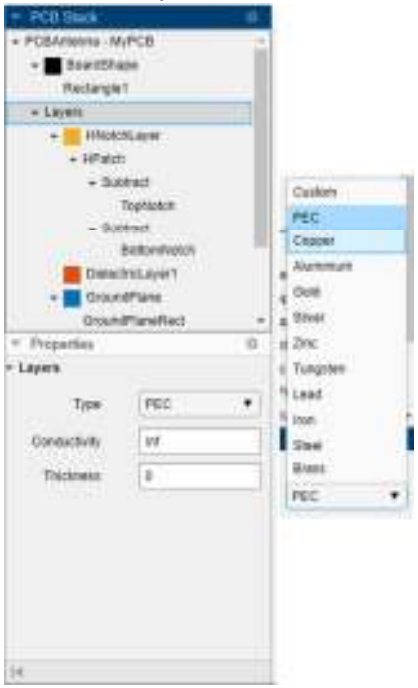
Change Start Layer and Stop Layer

Connect the feed from HNotchLayer to GroundPlane by setting StartLayer to HNotchLayer and StopLayer to GroundPlane. This creates a feed on the GroundPlane which connects to the HNotchLayer.



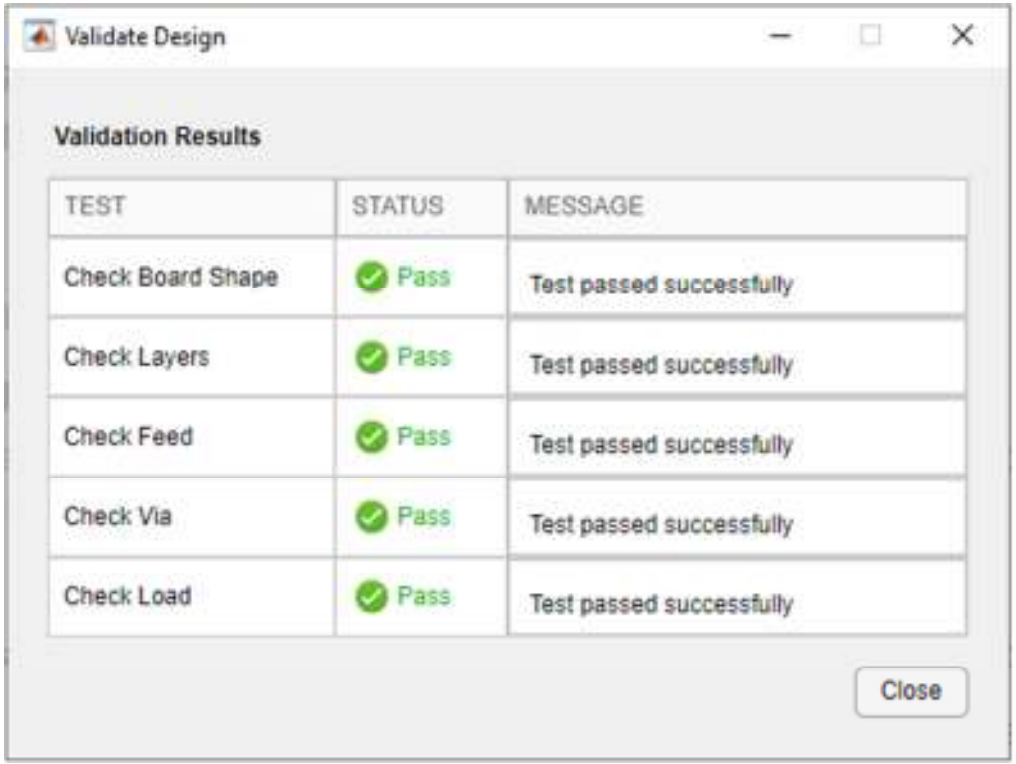
Change Metal Properties

Select the Layers in on the PCB Stack navigation tree and set the Type of the metal layer to Copper in the Properties pane.



Validate Design

Click Validate Design on the toolbar to validate your board shape, layers, feed, via, and load.



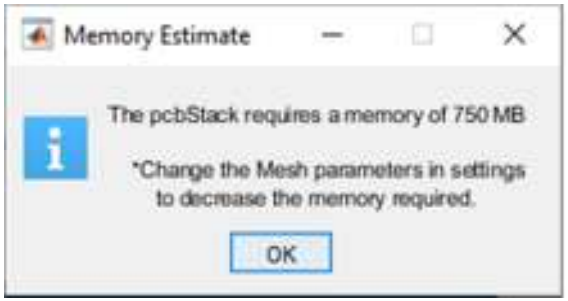
Analyze H-Notch Unit Element

This example shows how to perform vector and scalar frequency analysis on a 1-by-2 H-notch linear array.

In the Analysis tab, set the Center Frequency to 4.65 GHz and Frequency Range to 4:0.1:5 GHz .

Memory Estimate

Click the Memory Estimate button from the Mesh section to estimate the memory you will require to run the analysis.



You will need a memory of 750 MB (RAM) to analyze the antenna design. You can alter the memory requirements by changing the mesh parameters. Select Mesh on the toolbar to alter the mesh settings.

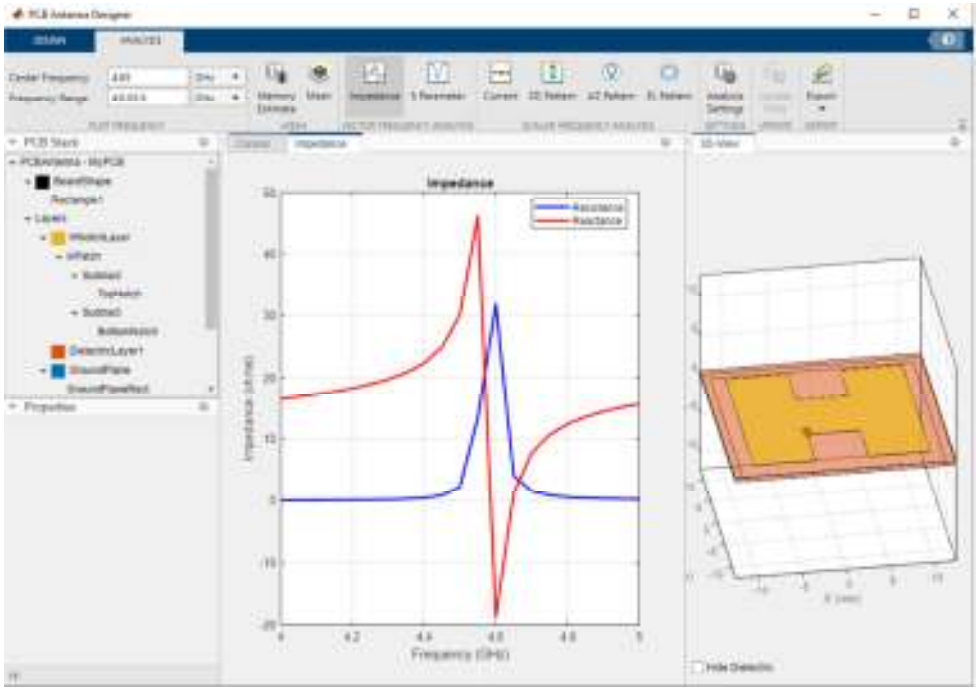
Analysis Settings

Click Analysis Settings from the Analysis section on the toolbar to set the the plot and mesh parameters to the following:
Ref Impedance(Z0) — 50 ohms
Az Range — 0:5:360 deg
El Range — 0:5:360 deg
Meshing Mode — auto

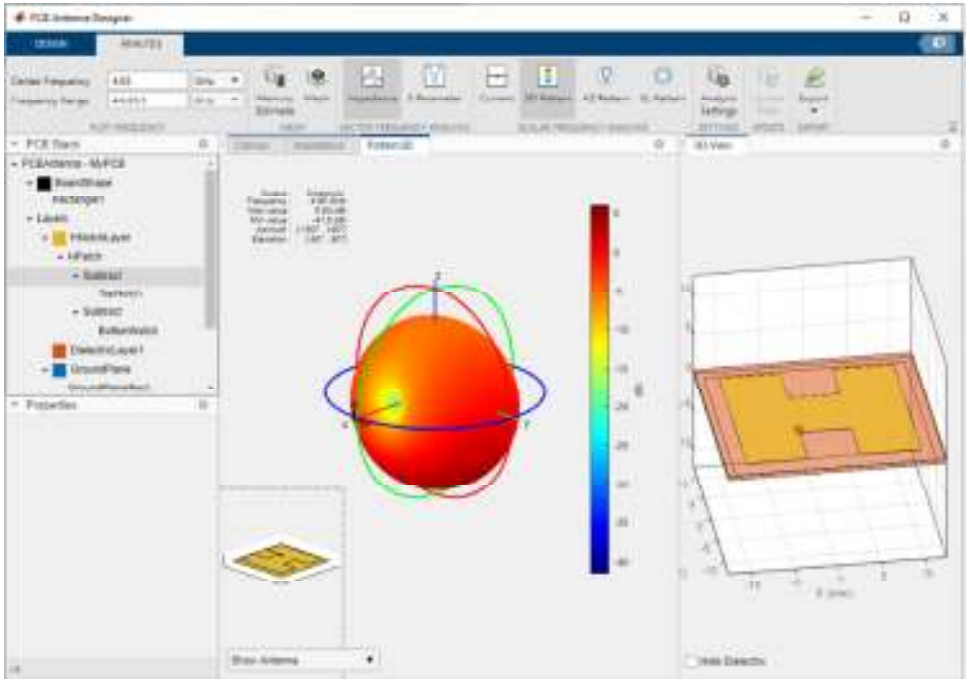
Click OK to save the settings.

Run Analysis

Select Impedance from the Vector Frequency Analysis section on the toolbar to plot the impedance plot. The impedance plot shows that the PCB stack antenna resonates at 4.6 GHz.



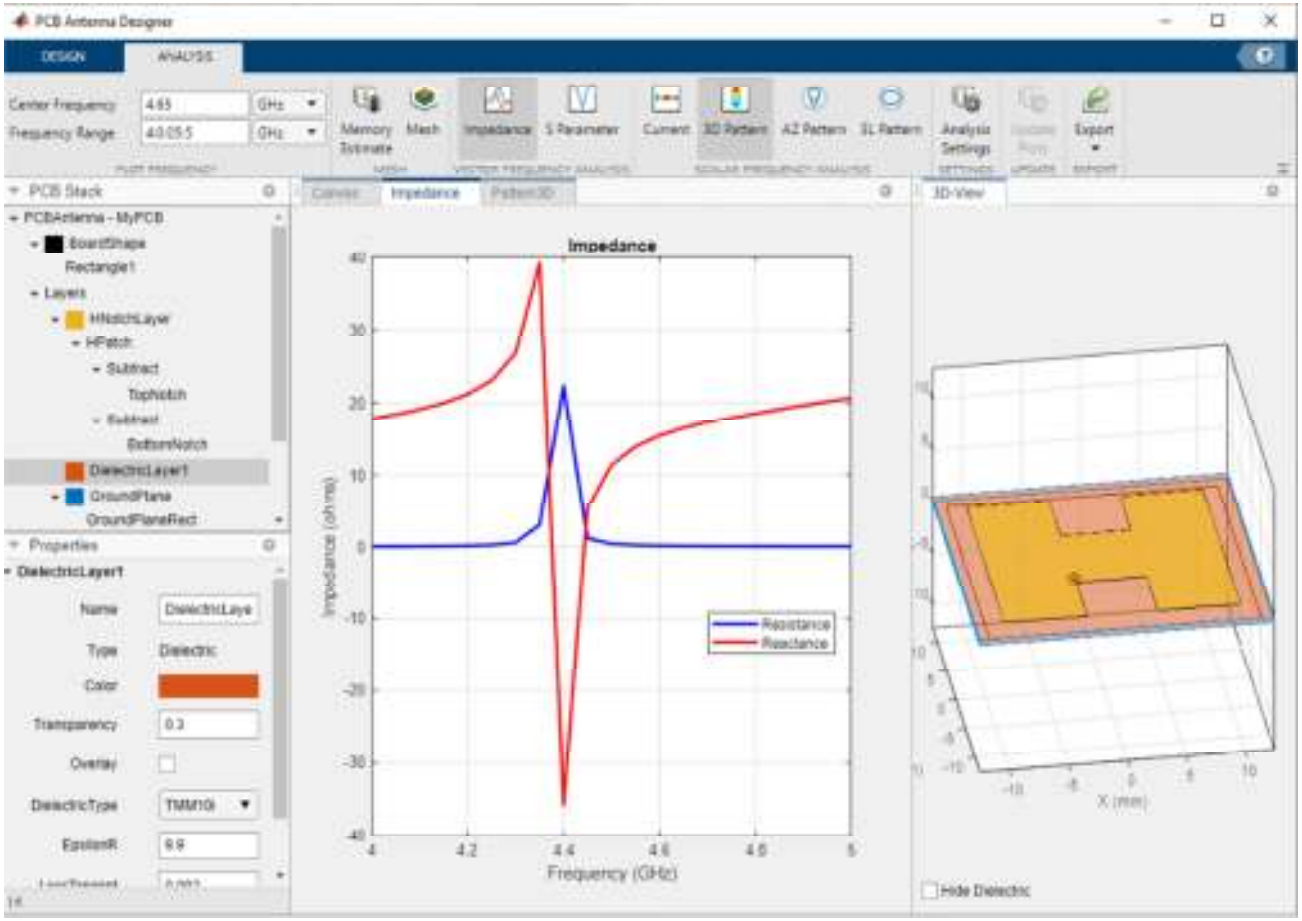
Click 3D Pattern from the Scalar Frequency Analysis section on the toolbar to plot the 3-D far-field radiation pattern of the antenna. The directivity of the H-notch patch unit element is 5.93 dBi.



Update Plots

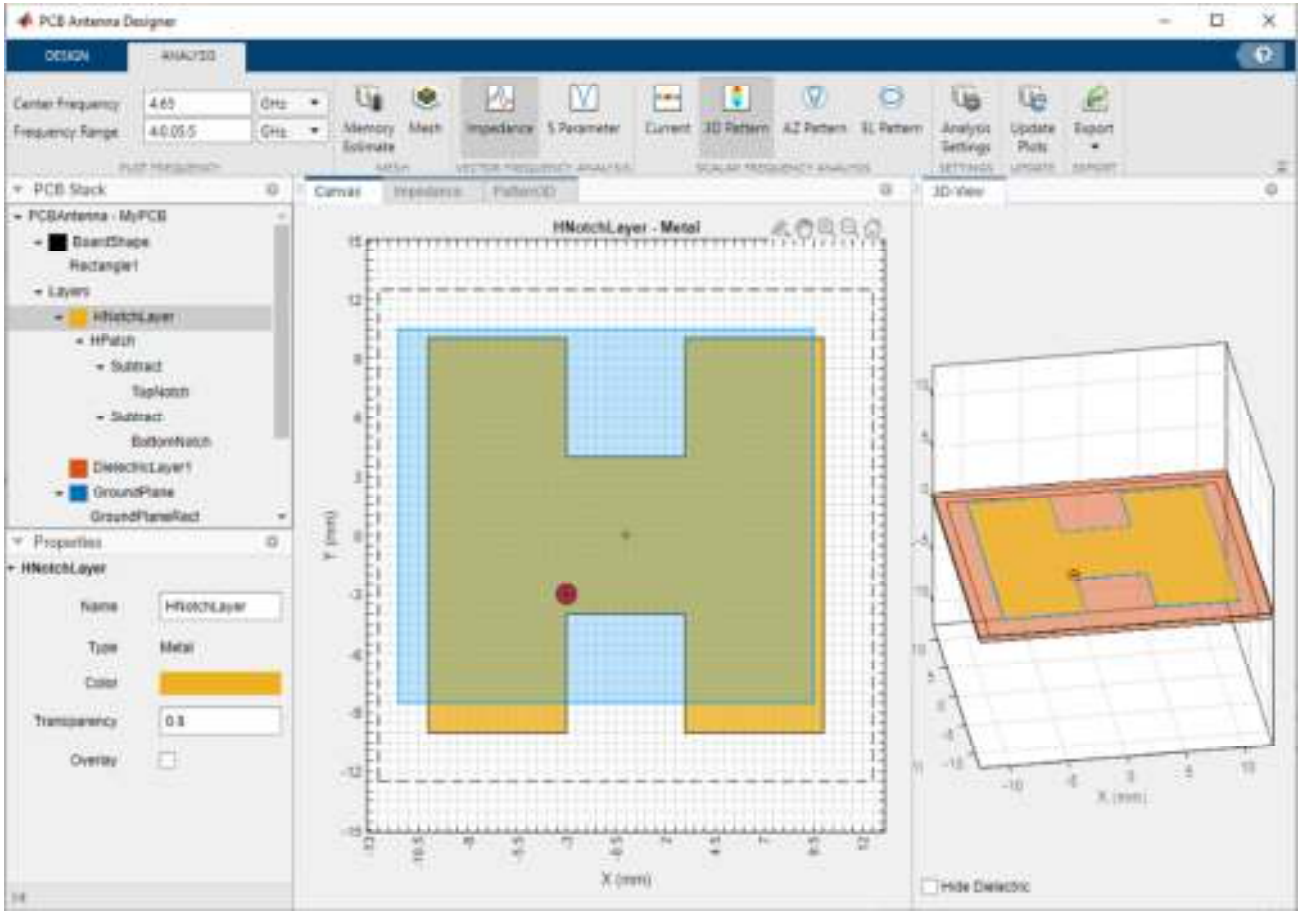
Select DielectricLayer1 from the PCB Stack navigation tree navigation tree and set the DielectricType to TMM10i in the Properties pane.

Click Update Plots from the Update section on the Analysis tab to update the analysis results. The impedance plot now shows that the PCB stack antenna resonates at 4.4 GHz.

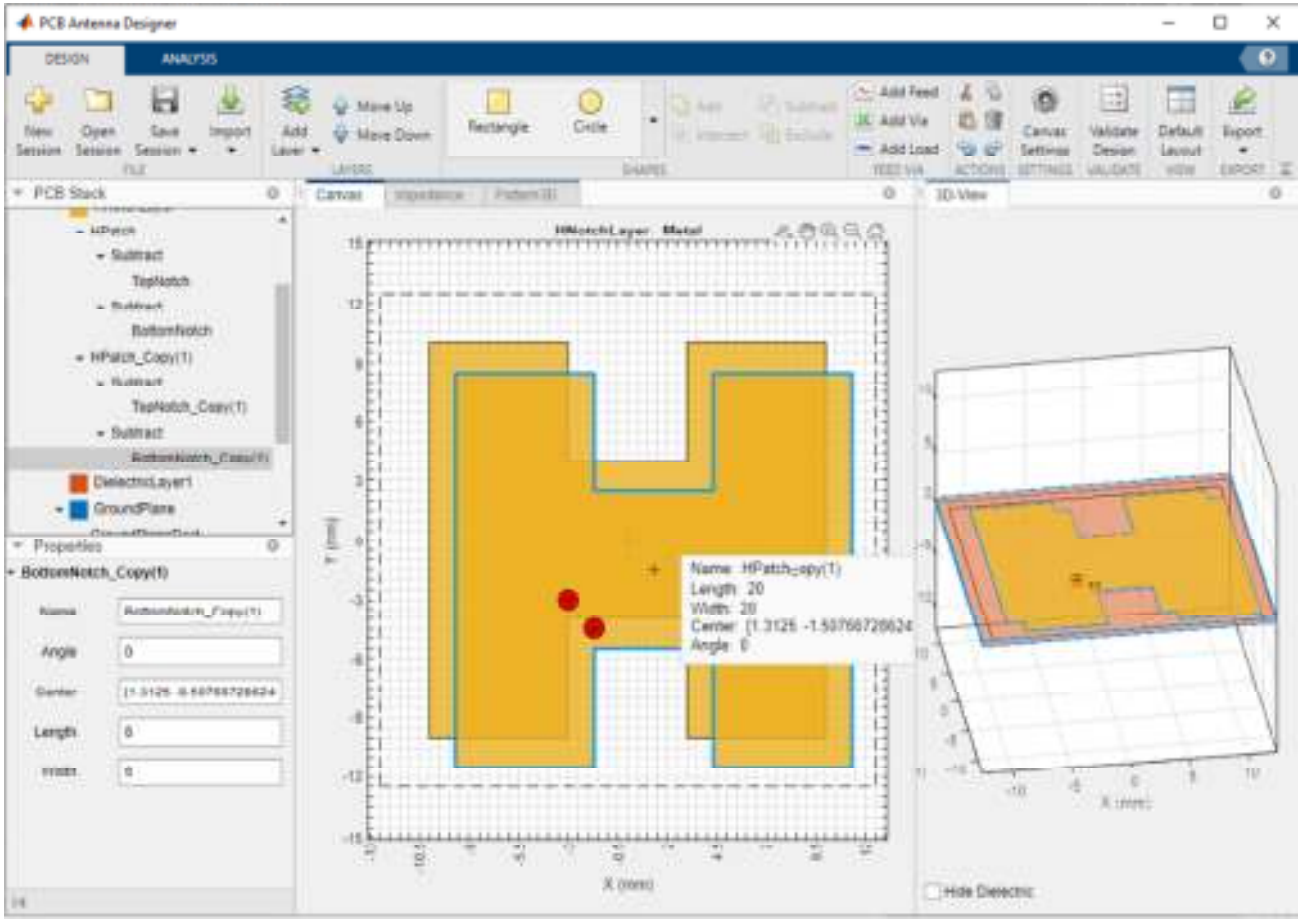


Design 1-by-2 H-Notch Linear Antenna Array

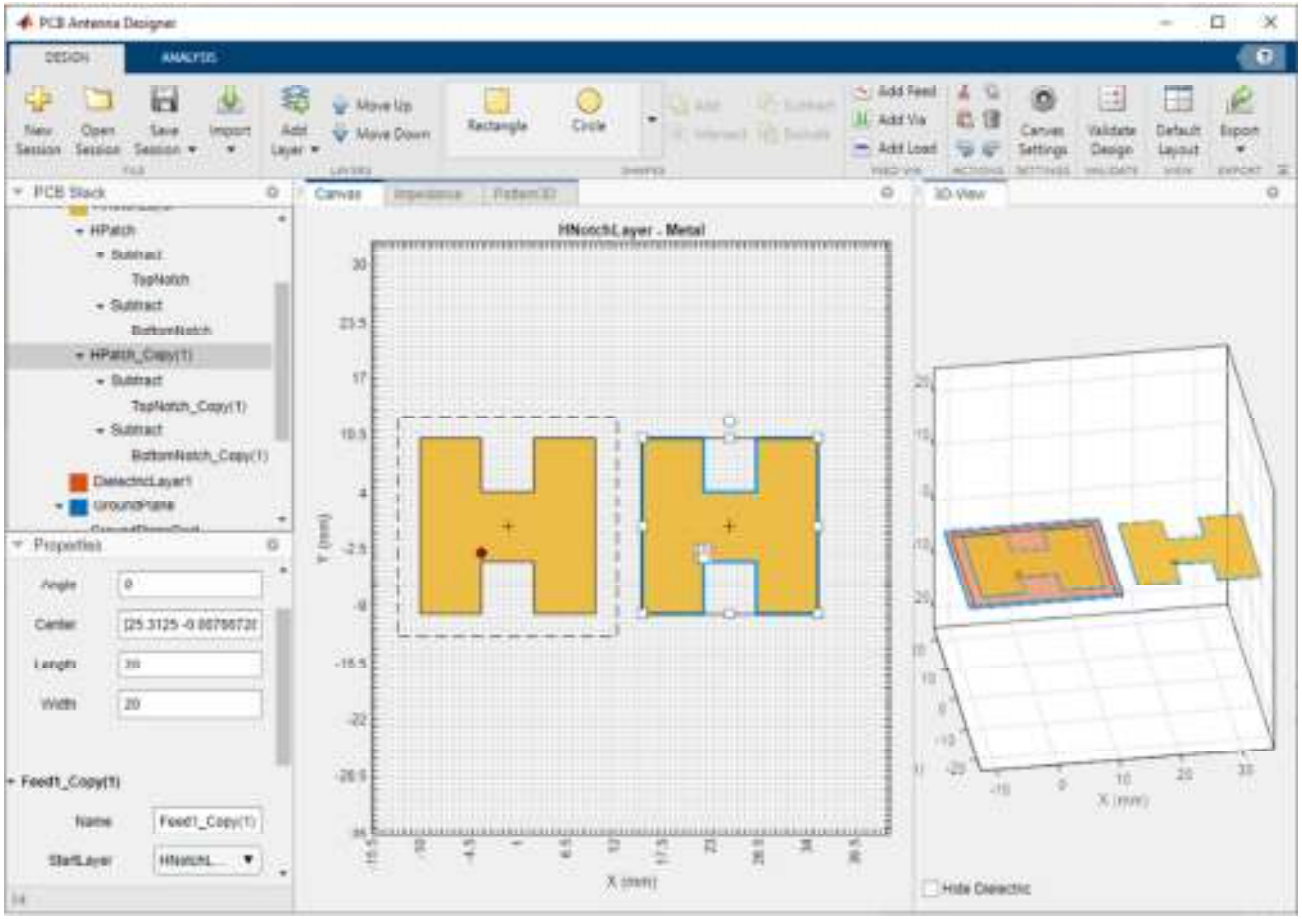
Select HNotchLayer in the PCB Stack pane and drag the axes to create a selection box. Drag the axes until the selection box encloses Feed and HNotchLayer.



Click Copy on the Design tab to copy the layers enclosed in the selection box and click Paste to create the HPatch_Copy(1) layer.

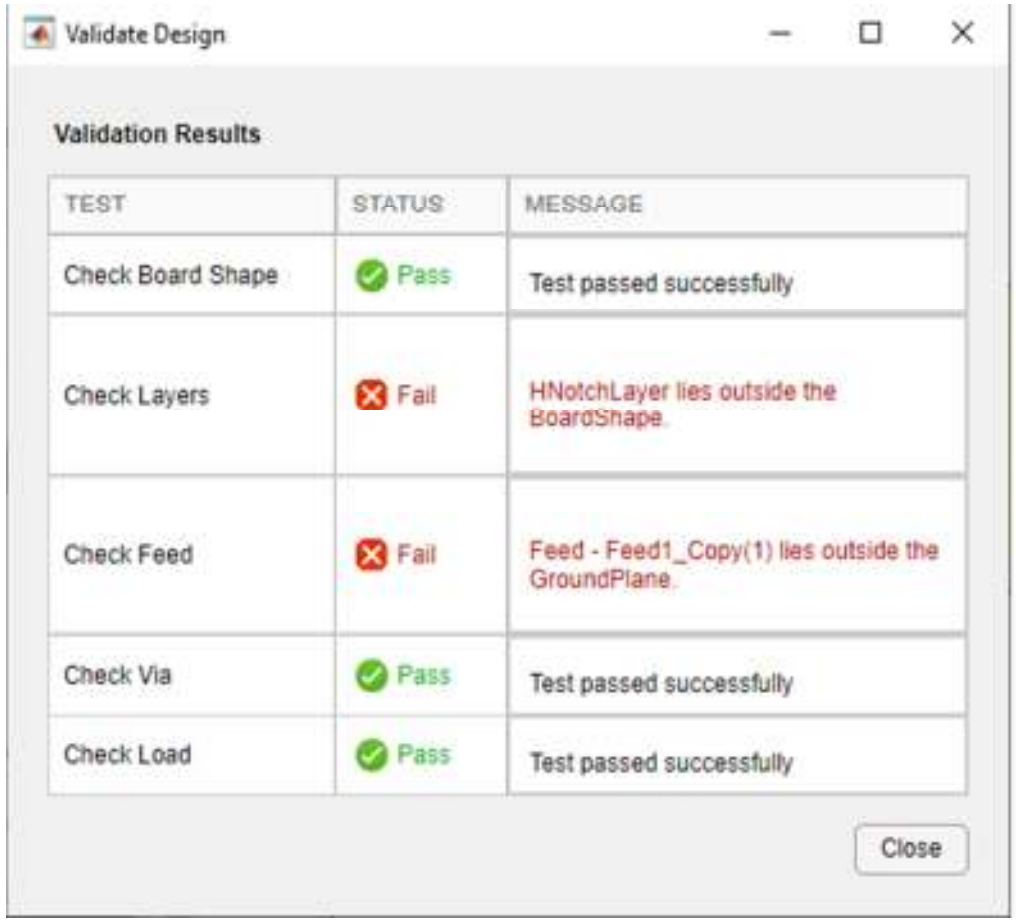


Select the HPatch_Copy(1) layer and drag it to place it to adjacent to the HPatch layer.



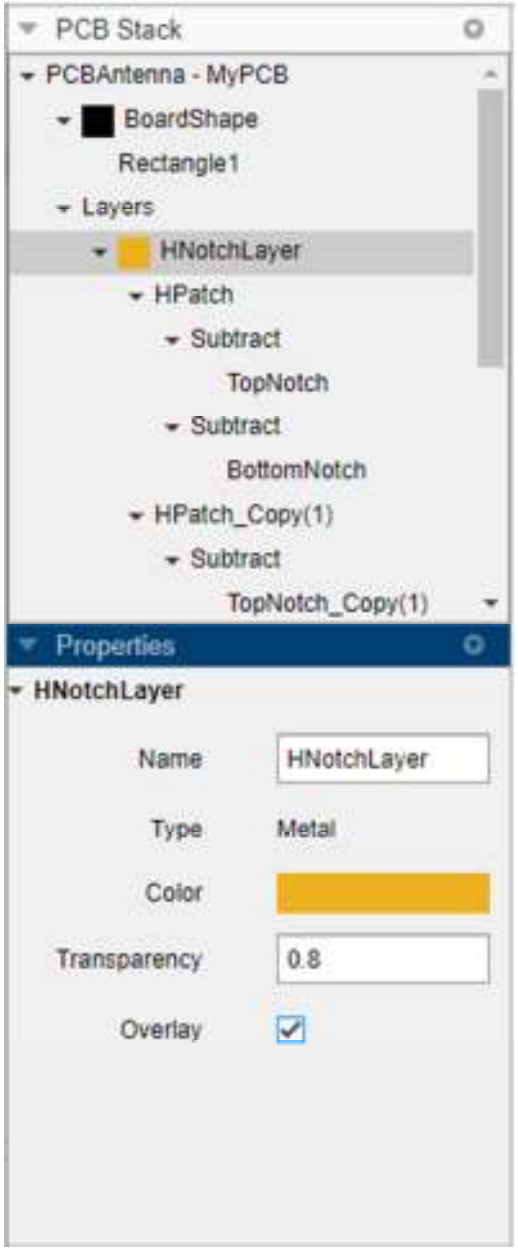
Validate Design for Errors

Click Validate Design on the toolbar to validate your board shape, layers, feed, via, and load. The Check Layers and Check Feed validation failed because HNotchLayer and Feed1_Copy(1) lie outside BoardShape and GroundPlane, respectively.

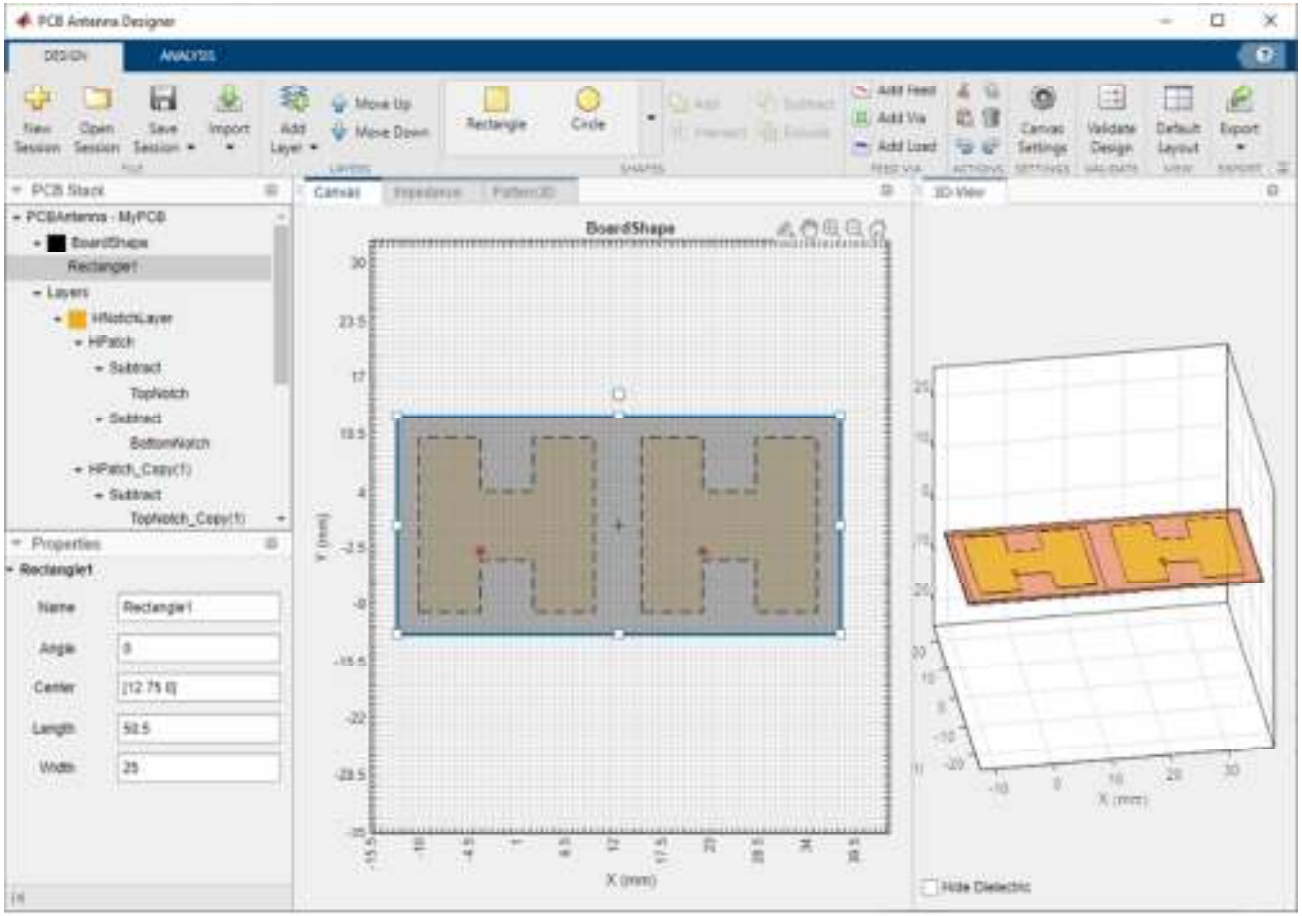


Overlay Layers

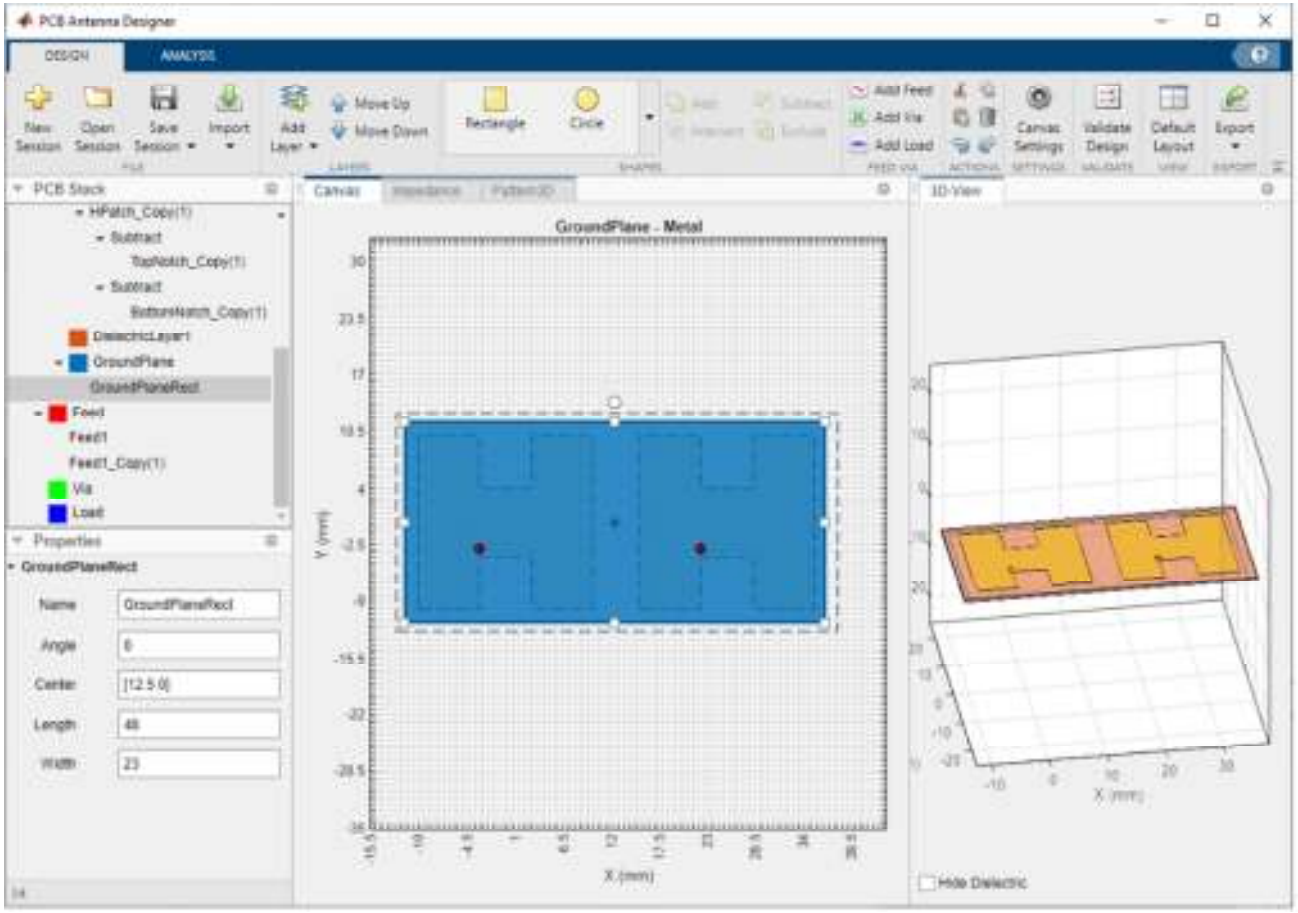
Select HNotchLayer and then select the Overlay check box in the Properties pane to create an image of HNotchLayer in the canvas.



Select Rectangle1 under BoardShape on the canvas and use the marker on the right to change the shape of BoardShape such that it encloses the HNotch image.



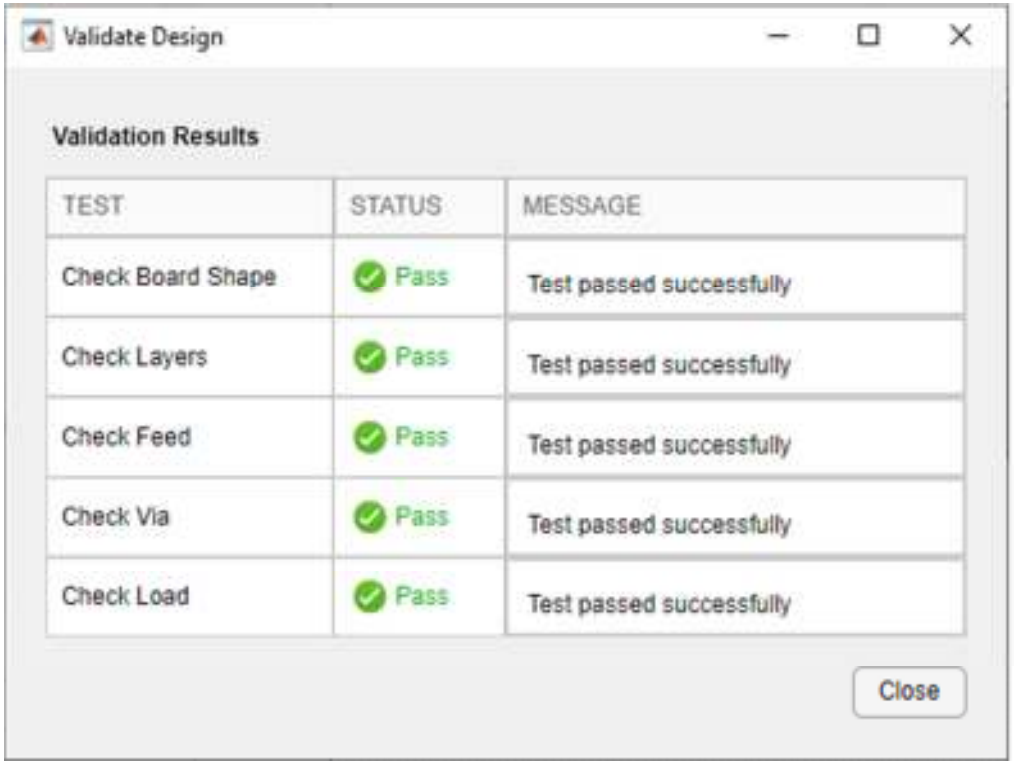
Repeat the procedure with the GroundPlane layer.



The BoardShape now encloses the HNotchLayer. You can change the layer dimensions using the Properties pane.

Validate Design for Errors

Click Validate Design to check for issues in the design. The design is validated with no errors.

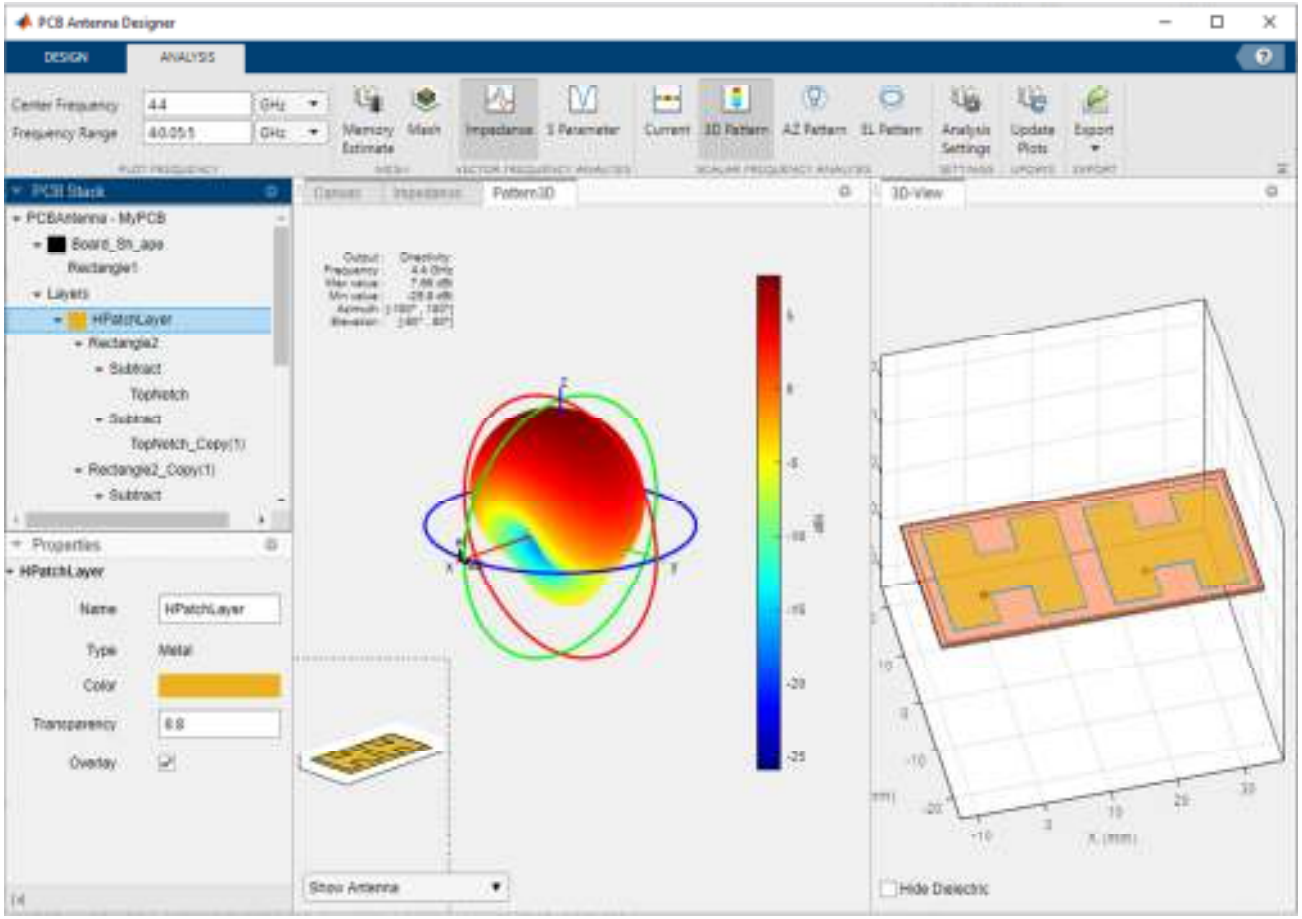


Run Analysis

Click Update Plots in the Analysis tab to update the analysis results. This updates all the analysis plots in the app for the design of a 1-by-2 H-notch linear array.

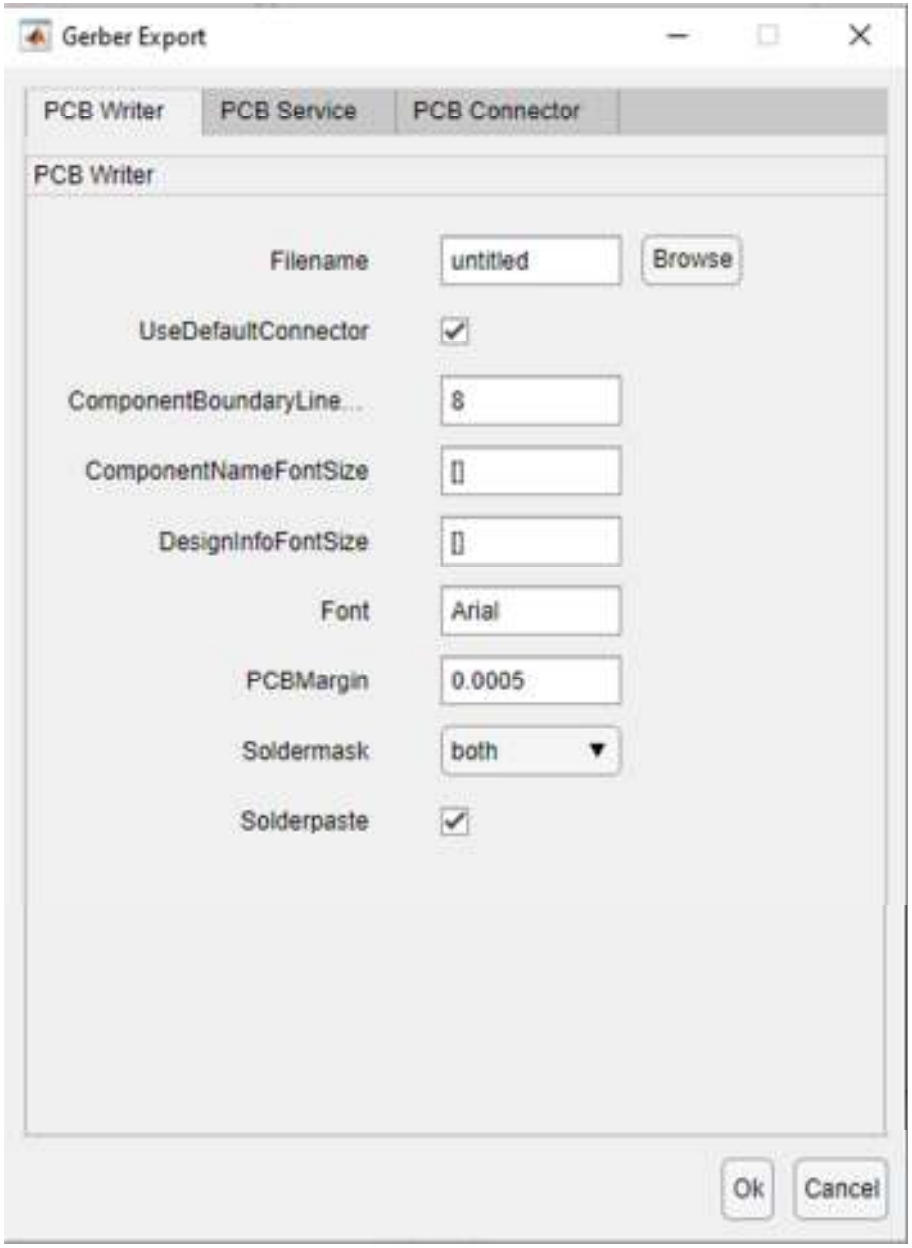
3-D Pattern

The pattern for the antenna is more directional with a higher gain compared to the the H-notch patch unit element. The directivity of the 1-by-2 H-notch linear array is 7.66 dBi.

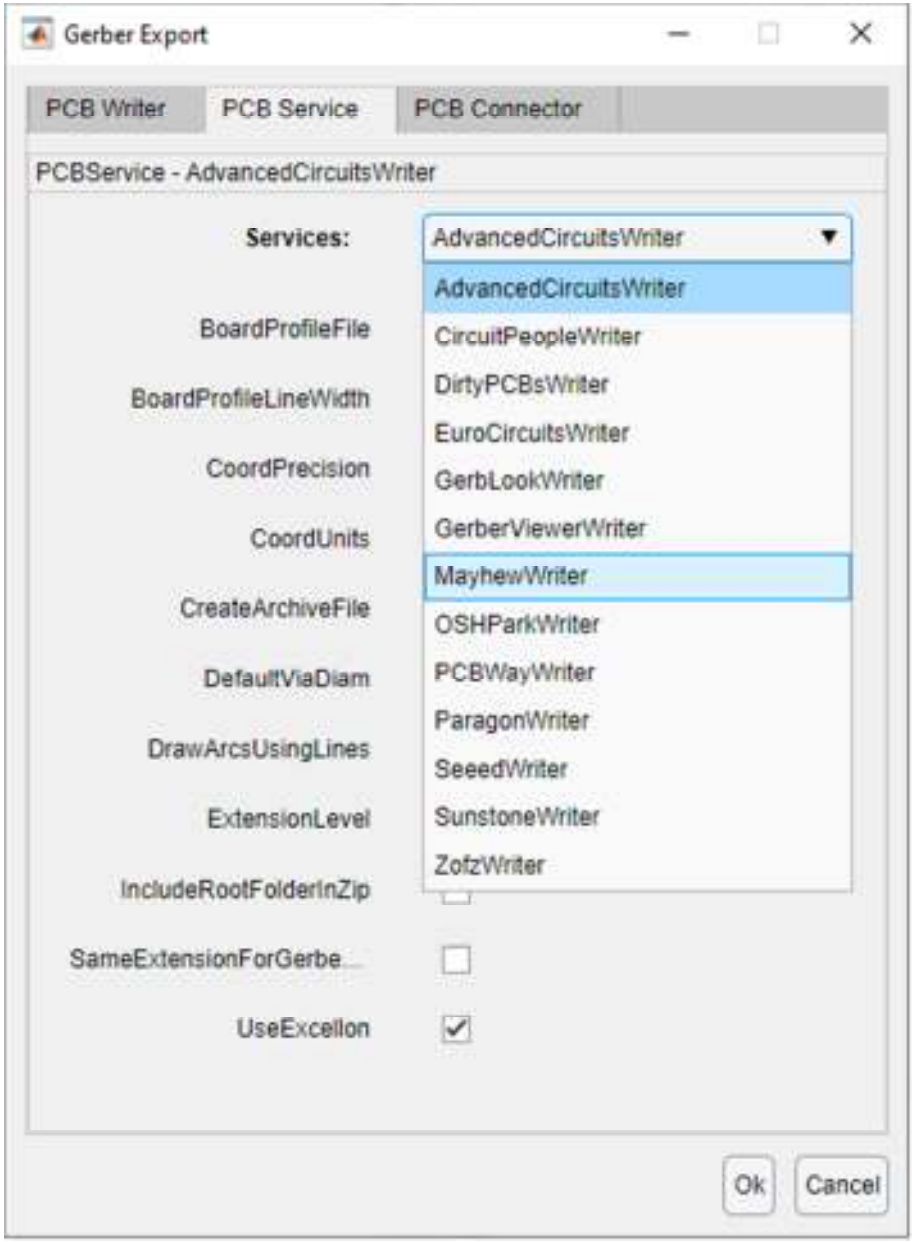


Export Your Design to Gerber Files

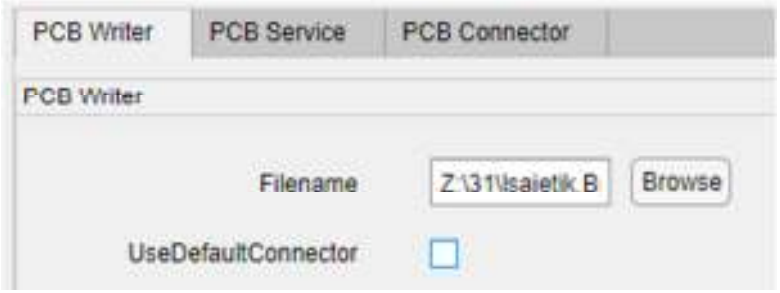
In the Analysis tab, click Export and then select Export as Gerber File. In the Gerber Export dialog box, select Browse in the PCB Writer tab to select the directory in which you want the Gerber files to be stored. Clear the UseDefaultConnector check box.



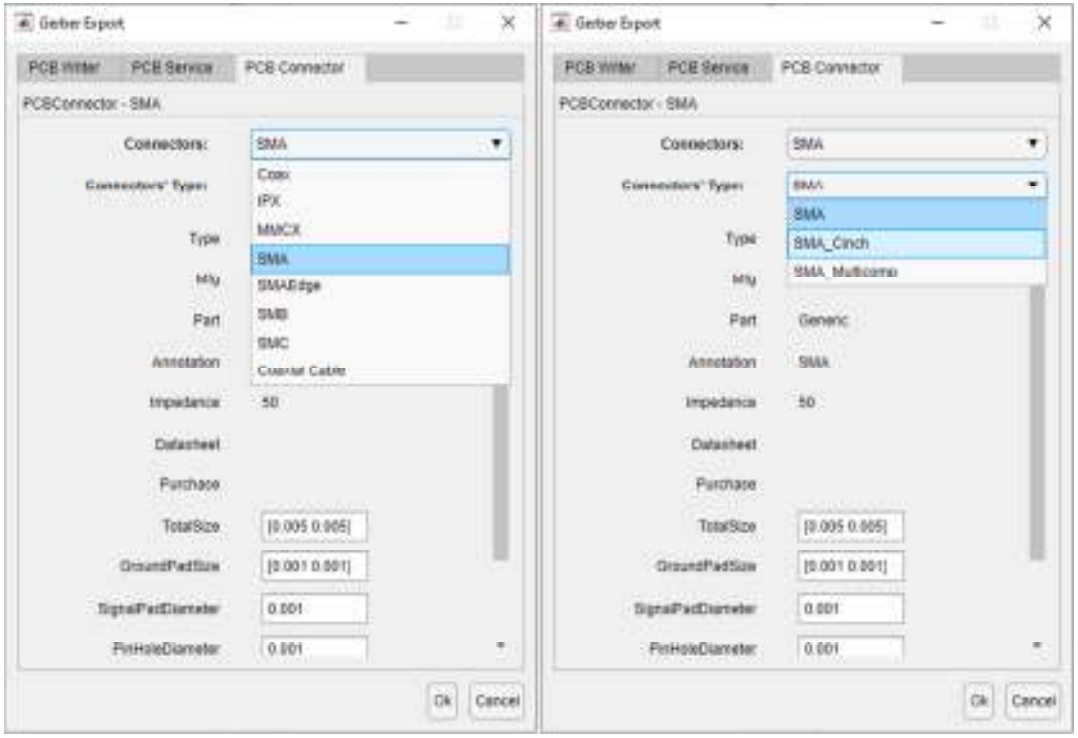
In the PCB Services tab, under Services, select MayhewWriter. The PCB writer uses the selected service to create the Gerber files.



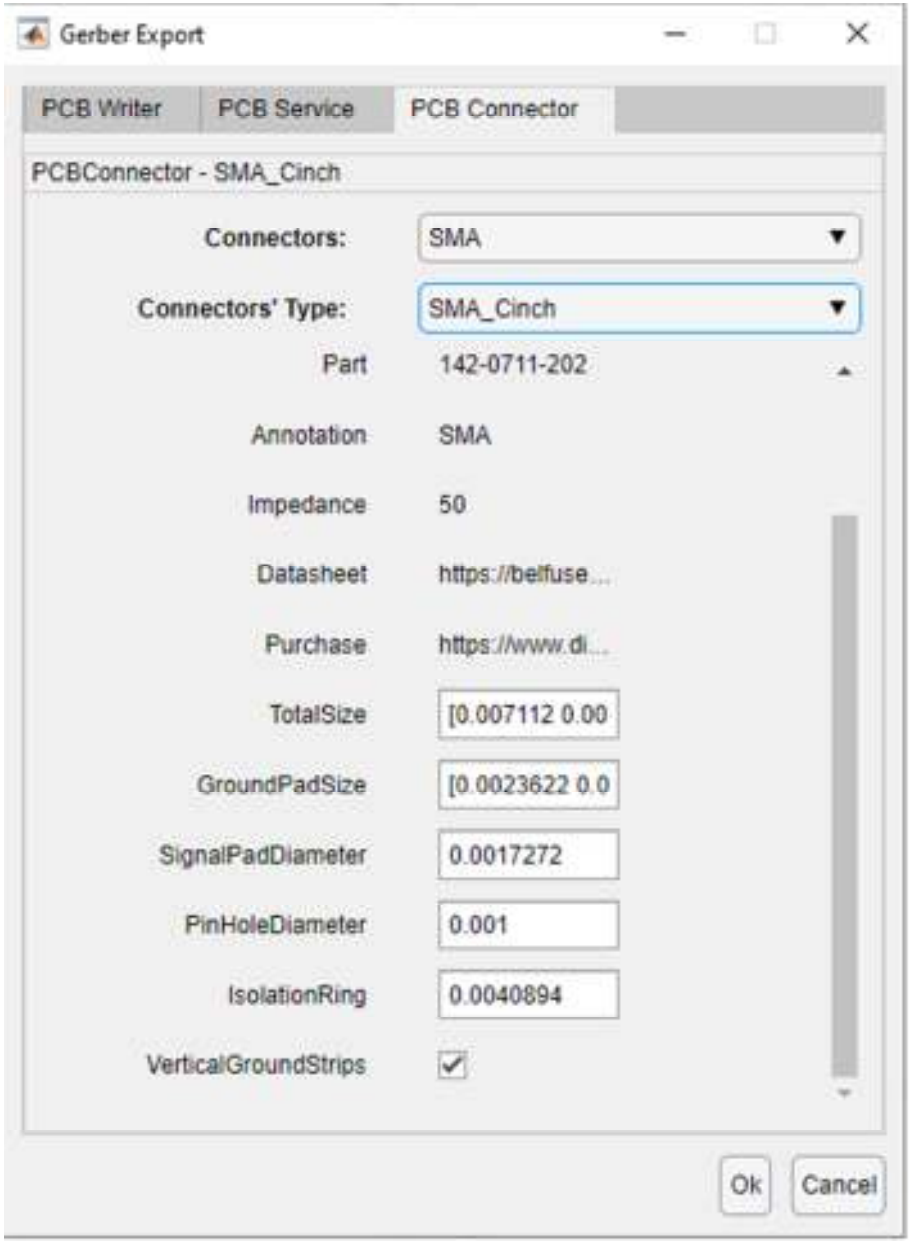
In the PCB Connector tab, clear the UseDefaultConnector check box.



In the PCB Connector tab, set Connectors to SMA and Connectors' Type to SMA_Cinch.



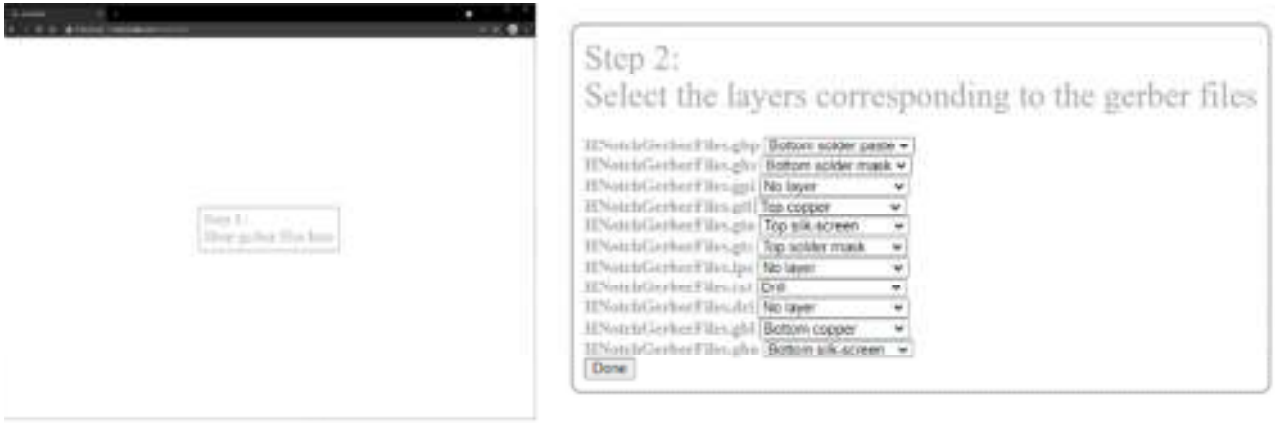
The PinHoleDiameter is of the same value as the FeedDiameter. Click OK to generate the Gerber files.



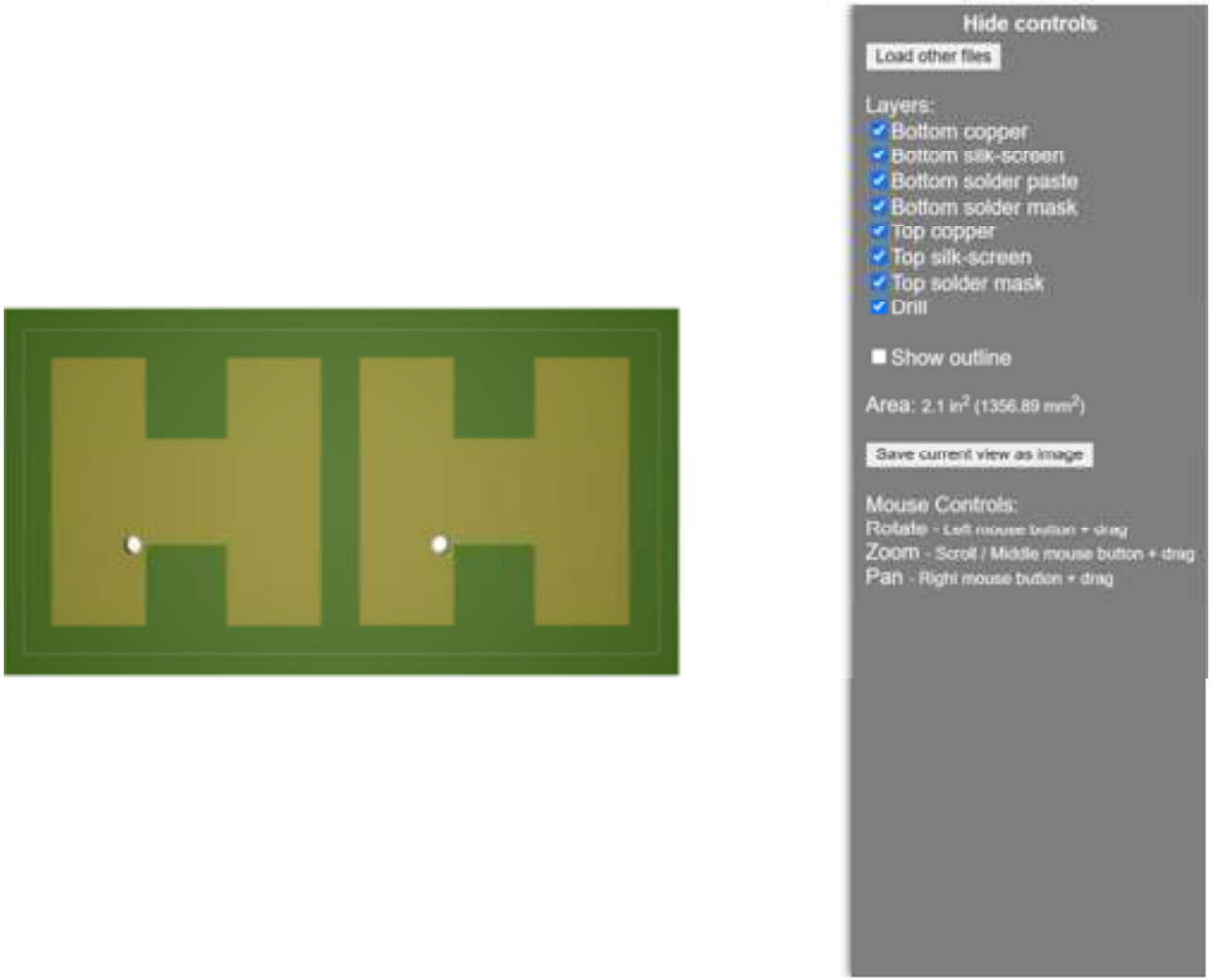
The generated Gerber files are stored in the directory you selected in the PCB Writer tab.

Name	Type	Size
HNotchArray.dri	DRI File	1 KB
HNotchArray.gbl	GBL File	2 KB
HNotchArray.gbo	GBO File	184 KB
HNotchArray.gbp	GBP File	2 KB
HNotchArray.gbs	GBS File	1 KB
HNotchArray.gpi	GPI File	2 KB
HNotchArray.gtl	GTL File	1 KB
HNotchArray.gto	GTO File	1 KB
HNotchArray.gts	GTS File	1 KB
HNotchArray.ipc	IPC File	1 KB
HNotchArray	Text Document	1 KB

Open mayhewlabs.com/webGerber from your browser. Drag and drop the Gerber files from the directory and select Done.



The Gerber files are displayed in the Gerber viewer. You can use the Gerber files to fabricate of your 1-by-2 H-notch linear array.



You can also export your design to the MATLAB® workspace or MATLAB script by selecting Export to MATLAB Workspace or Export to MATLAB Script under Export in the Design or Analysis