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

fians:	Reitarge1	_
Arghi	9	
Center	10.01	
Length	25	
with 5	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.

	😧 Move Up
Add Layer 👻	Wove Down
Metal L Add met	ayer ai layer
Dielect	ric Layer



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.

a Settings	2		×
Carves Setings			
thisp to grid	2		
010 Sce	9.5		
Unite	in	٠	
		(Dk)	Cantal

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



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].

PCO State	0	 Evoportion 	10
+ Layers		+ Feedt	
 HRotchLayer HPatch 		Name	Feed1
+ Subtract Yes/Kolub		StartLoyer	Headdelayer +
+ Suttract		StopLayer	HNotchLayer +
BattoreNatch		Center	[+3]
GroundPlana GroundPla		FeedVodage	1
- E feet		FeedPtuse	0
Fest	1	second lines	
Ma Na	1.4		

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.

alidation Results		
TEST	STATUS	MESSAGE
Check Board Shape	📀 Pass	Test passed successfully
Check Layers	📀 Pass	Test passed successfully
Check Feed	📀 Pass	Test passed successfully
Check Via	O Pass	Test passed successfully
Check Load	🔗 Pass	Test passed successfully

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.

A PCB Antenna	Designer														1	×
DESIGN	4942/315	1														0
Center Frequency	4.65	10Hz		0	۲		V	-		(Q)	0	06	18	B		
Frequency Range	40.15	GHL	٠	Meniory Estimate	Math	Incodence	S.Paranteler	Control	30 Padem	AZ Feltern	IL Fettern	Analysis Settings	32	Toport		
	NOT PERSONNEL		1	N/CS	ĸ	102104 1020	DICY MINOSE		SCALAR PROC	NUTRY ANAD	95	MITTINGS.	UPDATE.	DADAL		- 1

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.

M	emory Estimate	-	Ø	×
i	The pobStack requined the Me to decrease the terms of terms	ires a me sh param ie memor	mory of 7 eters in se y required	50 MB attings L
	0			

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 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

Plot Settingo		
Port:		
Refimpedance(20)	50	ohms
3D Pattern:		
Az Range	0:5:360	699
El Range	0.5.350	deg
Mesh Settings		
Meshing Mode	auto 🔻	1
Max Edge Length		m
Min Edge Length		m

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.

4 PC2 1-1-1-2	night -		+ p x
- Constant	Annation ((8)
Carles Frequency Frequency Freque	449		
- PCR flam		The Parent Father &	OF AllVest D
PERATANG 40 PERATANG 40 PERATANG PERATANGANANG PERATANGANANG PERATANGANANG PERATANGANANG PERATANGANANG PERATANGANANGANANG PERATANGANANGANANGANANGANANGANANGANANGANAN	PCB cri 1 NAM National Technology Rec	itiliti itiliti	
 Dense Oracional	telerikasi Manuf Pana Reserved		
		Bas Arized +	Owners

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.

alidation Results		
TEST	STATUS	MESSAGE
Check Board Shape	🔗 Pass	Test passed successfully
Check Layers	🔀 Fail	HNotchLayer lies outside the BoardShape.
Check Feed	🔀 Fail	Feed - Feed1_Copy(1) lies outside the GroundPlane.
Check Via	🥝 Pass	Test passed successfully
Check Load	📀 Pass	Test passed successfully

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.

Validation Results			
TEST	STATUS	MESSAGE	
Check Board Shape	🖉 Pass	Test passed successfully	
Check Layers	O Pass	Test passed successfully	
Check Feed	O Pass	Test passed successfully	
Check Via	O Pass	Test passed successfully	
Check Load	🔗 Pass	Test passed successfully	

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.

Gerber Expo	rt		-		×
PCB Writer	PCB Service	PCB Connector			
PCB Writer					
	Filename	untitled	Browse	6	
Use	efaultConnector				
Component	BoundaryLine	8]		
Compone	ntNameFontSize	0]		
De	signInfoFontSize	D]		
	Font	Arial]]		
	PCBMargin	0.0005]		
	Soldermask	both 🔻)		
	Solderpaste				
			1		
				Ok	Canc

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

Gerber Expo	JT.		-		~
PC8 Writer	PCB Service	PCB Connector			
PCBService -	AdvancedCircuitsW	/riter			
Services:		AdvancedCircuit	tsWriter	3	•
Board	BoardProfileFile ProfileLineWidth CoordPrecision	AdvancedCircuit CircuitPeopleWr DirtyPCBsWriter EuroCircuitsWriter GerbLookWriter	tsWriter riter r ter		
CoordUnits CreateArchiveFile DefaultViaDiam DrawArcsUsingLines ExtensionLevel		GerberViewerWriter			
		OSHParkWriter	3		-
		PCBWayWriter ParagonWriter			
		SeeedWriter			
		SunstoneWriter			
Include	eRootFolderInZip	LUIZVVINEI			_
SameExter	sionForGerbe				
	UseExcellon	$[\mathbf{v}]$			
				<u>_</u>	

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

PCB Writer	PCB Service	PCB Connector	
PCB Writer			
	Filename	Z \31\Isaietik B	Browse

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

CENTRAL PCE Service	PC8 Connector		PCB WWW	PCE Service	PCB Connector	
CBConnector - SMA			PCBConnecto	(SMA		
Connectors:	SMA	1.0		Connectors:	SMA	
Controlors' Type:	Cosx		Gim	ecdors' Type:	8844	
Tope	MACK			Type	SMA	
Min	SMA.			100	SMA, Muttomp	
Fart	SMAC Spe			Part	Generic	
Annetation	SMC Cventel Cable			Annetation	SMA	
Impedance	50			Impedance	50	
Defasteet				Dutasheet		
Putchase				Purchase		
TotalSize	10.005 0.0051			TotalSize	[0.005.0.005]	
OrsundPedflux	10.001.0.001	1.77		GroundPadSize	[0.001 0.001]	
SignaPerDianater	0.001		91	paPatDurster	0.001	
PirtHaleDiameter	0.001	53	3	PisHoleDiameter	0.001	
		1923 (1928)				177117223

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

	n		-	- 1-1-	^
PCB Writer	PCB Service	PCB Connector			
PCBConnector	r - SMA_Cinch				
Connectors:		SMA			۲
Con	nectors' Type:	SMA_Cinch			•
	Part	142-0711-202			
	Annotation	SMA			
	Impedance	50			1
	Datasheet	https://belfuse			
	Purchase	https://www.di			
	TotalSize	[0.007112 0.00			
	GroundPadSize	[0.0023622 0.0			
Sig	gnalPadDiameter	0.0017272			
10	PinHoleDiameter	0.001			
	IsolationRing	0.0040894			
Vert	icalGroundStrips				

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

Name	Туре	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.

They galled She have	Step 2: Select the layers corresponding to the gerber files IN-and-Generfles.glv Bottom adder math v IN-and-Generfles.glv Bottom adder math v Bottom copper v IN-and-Generfles.glv Bottom adder v
----------------------	--

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