

Chapter 4

PATHWAVE

TECHNICAL OVERVIEW

Planar Electromagnetic (EM) Simulation in ADS

PathWave Advanced Design System (ADS)

Background

Electromagnetic (EM) simulation is a key consideration for many circuits. EM simulation has always been an important part of the RF design process. However, as next-generation technologies (i.e. PCIe Gen 5) require high throughput, EM simulation is becoming more important in areas where it has traditionally been neglected. Additionally, EM and electrical simulation have typically required different design environments, which has not only increased complexity of the design process but has also introduced significant potential for error.

Keysight ADS provides two key electromagnetic simulators integrated within its environment, making it convenient for the designers to perform EM simulations on their designs. Unlike circuit simulators, EM simulators are used on layout. Additionally, ADS is capable of EM and circuit co-simulation, reducing the need to use multiple design environments.

This chapter illustrates the flow that can be used to perform EM simulations using ADS. Towards the end of the chapter, EM/circuit co-simulation is explored.



Case Study 1: Microstrip Bandpass Filter

Step 1: Creating the Schematic Design for a BPF

Create a new workspace and select units as "mil." Create a new schematic cell and place components for a coupled line bandpass filter topology as shown in Figures 1 and 2. This section will walk through component selection and design considerations.

×

 \sim



Figure 1. Setup of Components



Figure 2. Schematic of Microstrip Bandpass Filter

In order to prepare this 5-section Coupled Line BPF, do the following:

- 1. Place Components
 - a. Place five **MCFILs** (Coupled Filter Section) from the **TLines-Microstrip** library palette onto the schematic.
 - b. Place two **Terms** (Port Impedance Termination for S-Parameters) from the **TLines-Microstrip** library palette onto the schematic.
 - c. Place one **S_Param** block (S-Parameter Simulation) from the **Simulation-S_Parameters** library palette onto the schematic.
 - d. Place one **MSUB** (Microstrip Substrate) from the **TLines-Microstrip** library palette onto the schematic.
 - e. Place one **VAR** (Variable Equation) onto the schematic.
 - f. Place a **ground** at the end of each Term block.
- 2. Arrange the components as shown in Figure 2.
- 3. Edit Component Values
 - a. Define the lengths for the MCFIL components:
 - 1st and 5th sections: 245 mils
 - 2nd and 4th sections: 195 mil
 - 5th sections: 237 mil
 - b. Define the output terminal value of the Term blocks: $Z = 50 \Omega$.
 - c. Define the frequencies on the S-Params block:
 - Start Frequency: 3 GHz
 - Stop Frequency: 7 GHz
 - Step Size: 0.01 GHz
 - d. Define the following values on the MSUB block:

H = 25 mil (Height of the dielectric)
Er = 9.9 (Relative Dielectric Constant)
Cond = 4.1E7 (Metal Conductivity, in this Case it is Set for Gold)
T = 0.7 mil (Metal Thickness)
TanD = 0.0009 (Loss Tangent)

- Define the values in the VAR block. To do this, double click on the block. In the window, enter a name and value for the variable. Press Add to create the variable as seen in Figure 3.
 - w1 = 10.2 w2 = 24 w3 = 28 s1 = 10 s2 = 17.4 s3 = 27.7

Edit Instand	ce Parameters						X
Library name: Cell name: View name: Instance name:	ads_datacmps VAR symbol VAR1		Swap Co	omponent			
Select Paramete	er		riab	le or Equation Entry M	lode		
w1=10.2 w3=28 s1=10 s2=17.4 s3=27.7 w2=24	,	~	Name Variab	ard w1 le Value Equation Education Education Education Tune/Opt/Stat/D	Noi ditor OE Setup	ne	~
Add Variable Value:	Cut Pa	ste		Component Options		Reset	
ОК	A	pply		Cancel		Help	

Figure 3. Directions on how to Change the VAR Value

4. Run the simulation and observe the results by plotting S(1,1) and S(2,1). The bandpass filter should look like what is shown in Figure 4.



Figure 4. Output Graph of Bandpass Filter

Step 2: Creating the Layout from the Schematic

 To generate the layout of your schematic from the schematic window, go to Layout > General/Update Layout, as shown in Figure 5. Click OK on the next screen that appears.



Figure 5. How to Open the Layout Tab

 ADS will generate a layout based on schematic. The layout should look like what is shown in Figure 6. The layout view will be added to the view list under the cell name. This can be verified in the ADS main window.



Figure 6. Filter Layout

Step 3: Setup and Run EM Simulation

Several things need to be setup before EM simulation can be run. As a brief outline, the following things need to be done:

1. Connect the pins in the layout, which will be defined as Ports in the EM Setup window.

NOTE: We do not need to convert Pins to Ports where we don't need to see the results.

- Select the right simulator Momentum (Method of Moments) or FEM (Finite Element Method).
- 3. Define the proper substrate.
- 4. Define the simulation frequency plan.
- 5. Define the conductor meshing properties.
- 6. Create EM module and symbol This is an optional step. This is only needed if you are going to perform EM/Circuit co-simulation. This will be explained in a later section.

Now, let's begin the EM simulation setup for the bandpass filter.

1. Connect the pins at the input and output of the filter structure. This is outlined in Figures 7 and 8.

Lab4_Mstrip_Filter [Lab4_Mstrip_Filter1_lib:Lab4_Mstrip_Filter:layout] * (Layout):26



Figure 7. How to Add a Pin



Figure 8. Where to Place Pin

2. Click the EM setup form the EM simulation toolbar on the layout page as shown in Figure 9.

 $\boxed{ Lab4_Mstrip_Filter [Lab4_Mstrip_Filter1_lib:Lab4_Mstrip_Filter:layout] * (Layout):26 }$ $\boxed{ File Edit Select View Insert Options Tools Schematic EM Window DesignGuide Help }$ $\boxed{ C } \boxed{ C$

Figure 9. How to Open EM Simulation

- 3. In the box that appears, name the New EM Setup View **emSetup** and click the button that says **Create EM Setup View**. This is shown in Figure 10.
 - a. The EM setup window is as shown in Figure 11. You can note the '!' indicating that there is something missing from the setup. Hovering the mouse over the warning will show that there is missing information in the substrate because we have not defined the substrate yet.
- 4. In the emSetup box that appears, select the Substrate tab, and click New. After naming the substrate, click Create Substrate to accept the 25 mil Alumina template. We can modify the properties of the same to suit our applications. This window is shown in Figure 12.

EM New EM Setup View	\times
EM New EM Setup View Create a new EM setup view to contain your EM simulation settings.	
Library: Lab4_Mstrip_Filter1_lib	
Cell: Lab4_Mstrip_Filter	
View: emSetup1	
Use template	
<none></none>	\sim
Create EM Setup View Cancel Help	
Related topics What is a cell?	

Figure 10. How to Save EM Setup View



Figure 11. Substrate Properties Window

EM New Su	ubstrate	Х							
Create a new substrate file in the specified library.									
Library:	Lab4_Mstrip_Filter1_lib								
File name:	substrate1								
Template:	25milAlumina	\sim							
Create S	Substrate Cancel Help								

Figure 12. Create New Substrate Window

5. The substrate editor below will be opened. Click the **Alumina** dielectric layer and then click the "…" button, as shown in Figure 13.

substrate1 [Lab4_Mstrip_Filter1_	lib] (Substrate):28				- 🗆 X
le Technology Edit View Op	ptions Tools Window H	lp			
े 📁 🔚 🦻 🥙 🤄	🖗 🔜 TE 👯 📃				
Substrate Name: substrate1	cond			AIR Alumina (9.9) 25 mil	Use right mouse context menus to add or delete substrate items. Select items on the substrate and view their properties below. Shortcuts in the Edit menu can be used to quickly the next substrate item. Material Alumina (9.9) Thickness 25 Bounding area layer: milto:linkert Substrate Layer
Substrate Layer Stackup		🛟 🗙 Substrate Vias		¢ ×	
Type Name Dielectric	Material Thickness AIR	Туре	Name Top	Bottom Materia	3
1 Conductor	Gold 0.7 mil				
Dielectric	Alumina 25 mil				
Cover	PERFECT_C 0 um				
		K		,	

Figure 13. Substrate Editor window

a. In the Dielectrics tab, modify Alumina to have **Er = 9.9** and **TanD = 0.0009**, as shown in Figure 14. These are the same parameters that were used in the schematic design.

Mat	Material Permittivity (Er)		Er)	Permeability (MUr)		Djordjevic					
Aaterial Name	Library	Real	Imaginary	TanD	Real	Imaginary	Туре	TanD Freq	Low Freq	High Freq	Con
Alumina	Lab4_Mstri	9.9		0.0009	1		Svensson/Djordjevic	1 GHz	1 KHz	1 THz	25.2
FR_4_Core	Lab4_Mstri	4.6		0.01	1		Svensson/Djordjevic	1 GHz	1 KHz	1 THz	
FR_4_Prepreg	Lab4_Mstri	4.6		0.01	1		Svensson/Djordjevic	1 GHz	1 KHz	1 THz	
SolderMask	Lab4_Mstri	3.3			1		Svensson/Djordjevic	1 GHz	1 KHz	1 THz	
<i>,</i>	_										

Figure 14. Edit Dielectric Properties

b. In the Conductors tab, go to Add from Database and select Gold from the list. The database is shown in Figure 15. Click OK. After defining the dielectric and conductor properties, click OK.

Add Materi	ials From Data	base		
elect materials	to add to the teo	hnology:		
Conductors	Dielectrics	Semiconductors		
Name	Туре	Real	Imag	
70_30_brass	Conductivity	1.57e7 Siemens/m Siemens/m		
Ag	Conductivity	6.17e7 Siemens/m Siemens/m		
AI	Conductivity	3.72e7 Siemens/m Siemens/m		
Aluminium	Conductivity	3.72e7 Siemens/m Siemens/m		
Au	Conductivity	4.1e7 Siemens/m Siemens/m		
Copper	Conductivity	5.8e7 Siemens/m Siemens/m		
Cu	Conductivity	5.8e7 Siemens/m Siemens/m		
Gold	Conductivity	4.1e7 Siemens/m Siemens/m		

Figure 15. Material database, showing conductors

c. From the main substrate definition window, select the **conductor** layer from graphic window. Select **Gold** from the material dropdown and define the thickness as **0.7 mil**.

ڬ substrate1 [Lab	o4_Mstrip_Filter1	_lib] (Substrate):22									-		\times
File Technology	Edit View C	Options Tools	Window Help											
0 📁 🔒 🦻) (? 🕂 🤣 (😴 🖓												
Substrate Name: s	ubstrate1									Use right mouse	context menus to add or	delete sub	strate item	15.
									^	Select items on	the substrate and view th	eir propert	les below.	
										Shortcuts in the	Edit menu can be used to	quickly ed	it the next	
										substrate item.				
										Collectu	ayer			
								AIR		Lay	cond (1)		· ·	
										Proci	Conductor		1	~
		cond						Alumina (9.	9)	Material	Cold	lapes from	ldyer	
25								25 mil	-/	Material	(Gold		· .	
								1		Operation	 Sheet Intrude into substr 	ate		
0 mil										1	Expand the substra	ate		
											Above interface			
										Position	 Below interface 			
									~	Thickness	0.7	m	il i	~
Substrate L	ckup			¢ x	Substrate Vias				¢ ×	Angle	90		degrees	
	Name	Material	Thickness		Туре	Name	Тор	Bottom	Materia	Surface	Top <none></none>		×	
Diele		AIR								roughness model	Bottom <none></none>		×	
1 Conductor	cond (1)	Gold	0.7 mil									21		
Dielectric		Alumina	25 mil							Precedence	1 📮	10		
Cover		PERFECT_C	0 um							To move the up or down.	e layer up or down on the	substrate,	just drag it	2
					<				>					

Figure 16. Modify substrate

- d. Click Save and close the substrate editor window.
- Click **Ports** in the EM setup window to inspect that there are two ports defined (one for each Pin placed in layout) as shown in Figure 17. You might need to drag down the bottom border of the **S-Parameter Ports (view only)** box to see Port 2.

EM Lab4_Mstrip_Filter1_lib:	Lab4_Mstri	p_Filter	:emSetu	o (EM Setup fo	or simulatio	on)		-		×		
File Tools View Help												
🔚 🖱 🦻 🦉 🖁	M M	3		🗊 📶 -	& 🚵							
Mom uW Cayout Partitioning Substrate Ports Frequency plan	Ports Defined S-param	by input	layout ts (view or	Dpen Layout Por	t Editor							
Output plan	Numbe	er	Gnd La	ayer Name	Feed Type	Ref In	npedanc	e [Ohm]	Ohm] Ref Offset [
E Options Resources Model Notes	~ 1	✓ 1										
	~ <u>1</u>	2	<impli< td=""><td>cit> P2</td><td>Auto</td><td>50 + 0</td><td>)i</td><td></td><td>N/A</td><td>></td></impli<>	cit> P2	Auto	50 + 0)i		N/A	>		
	Layout Pins (view only)											
	Name	Layer	Net	Connected to	Purpose	X [mil]	Y [mil]	Number	Layer N	Num		
	O• P1	cond	P1	1(+)	drawing	0	0	1	1			
	○• P2	cond	P2	2(+)	drawing	1117	-178.9	2	1			
	<									>		
	Hide	e connec	ted layout	pins								
	Auto	o-select	Auto-	center 🗌 Auto	-zoom							

Figure 17. Edit substrate ports

7. In the Frequency Plan section, define the **Sweep Type** to be **Adaptive**.

Fstart = 3 GHz Fstop = 7 GHz Npts = 101

EM Lab4_Mstrip_Filter1_lib File Tools View Help I Tools 10 10 10 10 10 10 10 10 10 10 10 10 10	EM	Mstrip_Filte	er.emSet	up (EM ⊦ 🗍	Setup for s	imulatio	on)	-		×
V Mom uW Layout	Fr	equency Plan Add	Rer	nove						
Ports Ports Frequency plan Output plan	1	Type Adaptive	Fstart 3 GHz	Fstop 7 GHz	Npts 101 (max)	Step -	Enabled	- N	lore	
E Options Resources										
						Gener	ate: S-Param	eters	~ S	imulate

Figure 18. Adjust Frequency

8. In the Options section, go to the **Mesh** tab. Enable **Edge Mesh**, and select **Auto-determine edge width**. Leave the other fields as default. This is shown in Figure 19.

M Lab4_Mstrip_Filter1_lib le Tools View Help	xLab4_Mstrip_Filter.emSetup (EM Setup for simulation) — 🗆	~
🖥 🖻 🥙	🏗 🕅 🔁 늘 🗁 🗐 🍈 🐁 🌉	
Mom uW	Simulation Options	
Partitioning	Preset: <none></none>	
Substrate	Description Physical Model Preprocessor Mesh Solver Expert	
Frequency plan	Global Layer Specific Net Specific Shape Specific	
Options	Mesh frequency	
Resources	Highest simulation frequency	
Model		
E Hotes	O mean nequency 0	
	Mesh density	
	O Maximum cellsize 0 mil ~	
	Cells/Wavelength 20	
	☑ Edge mesh	
	Auto-determine edge width	
	O Use edge width 0 mil ∨	
	·	
	Generate: S-Parameters Simulat	e

Figure 19. Adjust Simulation Options

- 9. Click the Save button and click the Simulate button at the bottom right-hand side of the EM setup window.
- 10. Once the simulation has started, the following two windows will appear: Job Manager and Momentum Simulation Status.
 - a. Job Manager window (Figure 20): Displays simulation job status. Its status turns to "Done" once the simulation is finished. A completed simulation is shown in Figure 22.

4	EEsof Job Manag	er				_		×	
File	e View Job Qi	ueue Settings	Help						
3	ob Summary								
	Tag	Status	Host	Queue	Workspace::Lib:Cell:View				
	M210122153746	Done	localhost		k::Lab4_Mstrip_Filter1_lib:Lab4_Mstrip_Filter:layou				

Figure 20. View Jobs in Job Manager

b. Momentum Simulation Status window (Figure 21): Displays simulation status. It shows the time when simulation finishes and displays the frequency points used to achieve the converged results. This is because the sweep type was set to adaptive. Adaptive sweeps automatically stop once the results converge.

SEsof Job M210122153746 (Lab4_Mstrip_Filter:layout)	_		\times
File View Job Help			
Info for Job: M210122153746		Statu	: Done
ADS Workspace::Design: C:\Users\calatham\ADSexample_Ch4_vrk::Lab4_Mstrip_Filt Simulator: Momentum Input/output directory: C:\Users\calatham\ADSexample_Ch4_wrk\simulation\Lab4_ Working directory: C:\Users\calatham\ADSexample_Ch4_wrk\simulation\Lab4_Mstrip_	terl_lib:Lab4 _Mstrip_Filte ip_Filterl_li	_Mstrip_F rl_lib\La b\Lab4_Ms	'ilter b4_Ms trip_
Tofe All series Research			/
Momentum MomEngine_64 2021.10.414 (*) built: Oct 15 2020 Copyright 1592 - 2020 Keysight Technologies Simulation Mode: uW Substrate valid Automatically determined a snap distance of 0.0021 mil for layout healing. Generating mesh at 7 GHZ S-parameter simulation Automatic selection: direct dense matrix solver Using multi-threading (8 threads) Matrix size: 330 (reduced: 360) Simulation temperature: 298.15 K The simulation results have been saved in dataset "Lab4_Mstrip_Filter_MomUW"			
Status M. ges			
calculating S-parameters Adaptive: 100.0 % of frequency range covered S-parameter simulation finished Simulation finished on: Fri Jan 22 15:37:56 2021 Job finished on: Fri Jan 22 15:37:59 2021 Exit status: Done			~
Current Item Progress			0%

Figure 21. Momentum Simulation Status Window

11. The Momentum simulation data display will open automatically. Delete all the graphs and insert a new rectangular graph. Select S(1,1) and S(2,1) to be plotted in dB. The simulation results are shown in Figure 22.



Figure 22. Graph of Momentum Simulation

Note:

When we perform Adaptive frequency sweep, ADS generates 2 dataset files; one for the points which are simulated (in our case: 15 freq points) and the other one with "_a" suffix indicating the adaptive rational polynomial fitted curve as illustrated. The dataset with "_a" suffix is recommended to be used for display purposes.

- Lab4_Mstrip_Filter_MomUW.ds
- Lab4_Mstrip_Filter_MomUW_a.ds

Comparison of Schematic and EM Simulation Results

The last step is to compare results. In order to see both the results on the same graph, double-click the graph from the Momentum simulation (from step 3). Click on the drop-down list to locate the dataset for the schematic-based simulation. It should have the same cell name but will not contain '_MomUW' (for example, 'Lab4_Mstrip_Filter'). Plot both S(1,1) and S(2,1) in dB. Observe the response for both circuit simulation as well as Momentum simulation.

In Figure 23, the results from the momentum simulation are shown in red and blue; the results from the schematic simulation is shown in green and black. Notice that the results are generally in agreement. The bandwidth of the filters is nearly identical. There is slight difference in S(1,1). This has several possible causes, such as coupling between the microstrips and edge effects, which are be taken into consideration in the Momentum simulation but not the schematic simulation.



Figure 23. EM and Schematic Graph

Case Study 2: Design and Simulation of a Patch Antenna

Theory

A microstrip antenna in its simplest configuration consists of a radiating patch on one side of a dielectric substrate, which has a ground plane on the other side. The patch conductors are usually made of copper or gold and can be any shape. However, conventional shapes are normally used to simplify analysis and performance prediction. The radiating elements and feed lines are usually photo etched on the dielectric substrate. The basic configuration of a microstrip patch antenna is shown in Figure 24.



Figure 24. Microstrip Patch Antenna

The radiating patch may be square, rectangular, circular elliptical, or any other configuration. Square, rectangular and circular shapes are the most common because of the ease of analysis and fabrication. Some of the advantages of microstrip antennas compared to conventional microwave antennas are:

- Low weight, low volume
- Low fabrication cost
- Easy mass production
- Linear and circular polarization are possible with simple feeds
- Easily integrated with MIC
- Feed lines and matching networks can be fabricated simultaneously with antenna structures

Patch antennas find various applications from military to commercial because of their many benefits. Patch arrays are extensively used in phased array radar applications and in applications requiring high directivity and narrow beam width.

Objective

Design a patch antenna at 2.4 GHz and simulate the performance using ADS.

Step 1: Calculating Patch Antenna Dimensions

Select an appropriate substrate thickness (h) and dielectric constant (ϵ_r) for the design of the patch antenna. In this case, we will use the following Dielectric for design:

```
Height = 1.6 mm
Metal Thickness = 0.7 mil (1/2 oz. Copper)
\epsilon_r = 4.6
TanD = 0.001
Conductivity = 5.8E7 s/m
```

Calculate the physical parameters of the patch antenna as shown in the geometry in Figure 25 using the given formula.



Figure 25. Dimensions of Patch Antenna

The width and length of the radiating surface is given by:

$$W = L = \frac{c}{(2f\sqrt{\epsilon_r})} = 29.2 \text{ mm}$$

where

Velocity of light, $c = 3 \times 10^8$ m/s

Frequency, f = 2.4 GHz

Relative Permittivity, $\varepsilon_r = 4.6$

The depth of the feed into the patch is given by:

H = 0.822L / 2 = 12 mm

The other dimensions are

$$Y = W / 5 = 5.8 mm$$

Step 2: Creating Patch Antenna Geometry

- 1. Open ADS and create a new workspace. Name it Lab5_PatchAntenna_wrk.
- 2. Open the new layout cell and name it Patch_Antenna
- Use Insert > Polygon and Insert > Coordinate Entry commands to enter (x,y) coordinates to enter the required points to construct the Patch Antenna geometry according to the calculations. The patch antenna is shown in Figure 26.



Figure 26. Dimensions of Patch Antenna

Step 3: Antenna Simulation

1. Connect a pin at the feed point of the antenna shown in Figure 27.



Figure 27. Feed point for microstrip antenna

Go to the EM Setup window. In the Substrate section, click New. Create a new substrate with the 25 mil Alumina as the template. Define the substrate with the parameters (substrate height, ε_r, and TanD) from step 1. Define the material as Copper (select it from Add from Database list). The definition is shown in Figures 28, 29, and 30.





Conductors	Dielectrics S	emiconductors	Surface Roughne	SS					
Ma	terial			Loss Parar	neters			Permeat	oility (MUr)
Material Nan Copper	h Library Lab5_Patc	Parameter Tyr Conductivity	Real 5.8e7 Siemen	Imaginary	Tnom	TC1	TC2	Real 1	Imaginary
<									
						Add Conductor	Add From Da	tabaco Ro	move Conduct

Figure 29. Material definitions for conductor layer

Material Permetability (MU) Material Name Library Real Imaginary TanD Real Imaginary Type Alumina Lab5_Patc 9.6 1 Svensson/Djor 1 Svensson/Djor	Djordjevi		
Material Name Library Real Imaginary TanD Real Imaginary Type Alumina Lab5_Patc 9.6 1 Svensson/Djor			
Alumina Lab5_Patc 9.6 1 Svensson/Djor	TanD Freq	Low Freq	High
	rdjevic 1 GHz	1 KHz	1 THz
FR_4_Core Lab5_Patc 4.6 0.01 1 Svensson/Djor	rdjevic 1 GHz	1 KHz	1 THz
<			2

Figure 30. Material definitions for dielectrics

- 3. Set the Simulation Frequency range as **2.1 GHz** to **2.7 GHz** (adaptive sweep). Add a new **Single Point** of **2.4 GHz**. The frequency definition is shown in Figure 31.
- 4. In the Output Plan section, select **User Specified Frequencies** under 'Save Currents For'. This will ensure that the currents are saved at the specified frequencies. These will be needed to see the far field.

EM Lab5_PatchAntenna_lib:	PatchAntenna	emSetup * ((EM Setu	p for simu	lation)					-		×
File Tools View Help	9 💥 📼			m 🔺	1							
	ём ёй 💶			. .								
Layout	Frequency P	20										
Partitioning Substrate	Add	Kem	ove									_
Ports	Type	Fstart	Fstop	Npts	Step	Enabled		Mor	e			
Output plan	1 Adaptiv	2.1 GHz	2.7 GHz	50 (max)			-					_
Resources	2 Single	2.4 GHz					-					_
Model												
								Generate	S-Parameters	v	Simu	late

Figure 31. Frequency setup for simulation

5. Run the simulation. The results are shown in Figure 35.



Discrete Frequencies vs. Fitted (AFS or Linear)



Step 4: Antenna Radiation Pattern

1. For the far field antenna pattern, go to **EM > Post Processing > Far Field**. This is shown in Figure 33. The visualization will open in another window.



Figure 33. Opening the far field display

Far field results will be displayed in a separate window, shown in Figure 34. If the window layout does not match the figure, you can use Window > Tile. To see the antenna parameters, go to Plot Properties (from the bottom tabs). Go to the Far Field tab (in the middle) and select Antenna Parameters to see all the required data. If the frequency is not set to 2.4 GHz, go to the Solution Setup tab and select 2.4 GHz.



Figure 34. Far field visualization window

3. In the Plot Properties tab, go to the **Far Field Cut** tab (in the middle). Click **Enable** and enter the desired **Phi**. Click **Display Cut in Data Display**.

Plot Properti	es		8
Scaling opt	ons		
	Display Maxim	um Field Location	s
Animation of	ptions		
Display upd	ate (ms)	1	100
Phase incre	ment (deg)	j	10
0	F	hase	360
1		т. т. т.	1 1 1 1
Shaded	Arrow Contou	ır 🛛 Far Fields	Far Field Cut
🔽 Enabl	e		
○ Thet	a 0 🕂		
Phi	0 -		
			1
Datase	at Name ∣em⊢ar		
	Display Cut	in Data Display	
Properties	Solution Setup	Plot Properties	

Figure 35. Edit Far Field Cut window

4. The results, shown in Figure 36, will open in a new window.





Results and Conclusions

From Figure 32, the patch antenna is shown to have a narrow bandwidth centered around 2.4 GHz, which matches expectations. In the far field visualization, only the 2.4 GHz frequency selection shows gain. Neither 2.1 GHz nor 2.7 GHz show gain, which is to be expected based on the S parameters. If you were to look at the radiation pattern closer to 2.4 GHz, you would notice that the gain is negligible until get within the antenna's bandwidth (about 2.37 GHz to 2.42 GHz).

Case Study 3: EM/Circuit Co-Simulation

What is EM/Circuit Co-Simulation?

Often, there is a requirement of having discrete components such as resistors, inductors, capacitors, transistors, etc. in the layout. However, EM solvers cannot simulate these discrete components directly. In this case, we use co-simulation, where we create a layout component and place it into the schematic, where the discrete components are connected.

Typical Process for EM/Circuit Co-Simulation

- 1. Connect pins in the layout where we need to make connections for discrete components.
- 2. Define the substrate and other EM settings (such as mesh, simulation frequency range, etc.).
- 3. Create an EM model and symbol for this layout component.
- 4. Place this layout component in the schematic and connect the required discrete components.
- 5. Set up the appropriate simulation in the schematic.
- 6. If it hasn't been done, Momentum/FEM simulation will be performed. Otherwise, the most recent data will be used.

Step 1: Create Layout

Create the layout manually using MLIN and MTEE components (from the TLines-Microstrip palette). The layout should be similar to what was shown in Figure 37. To match the results shown at the end of the section, the transmission line widths are 5 mm, and the lengths are 20 mm. For the two transmission lines connected to the top of the MTEE components, the lengths are 10 mm. In the rectangle at the bottom (using the polygon tool), make sure the add VIAs. This will be connected to ground.

P4	TL1	P3	P4	TL2 Tee1 TL4 P¶1	TL3	P 5	P6	TL7	TL5	TL6	R 7	P.	8	TL8	P
		ж	ж	P12 X	ж	ж		ا بر	₽10		ж ж				

Figure 37. Layout of circuit with MLIN and MTEE

- 2. Place pins wherever we need to make connections or assemble discrete parts along with layout.
- 3. In this case we have used the "cond" layer for conductors and a "hole" layer for VIA to provide a path to ground. To insert the hole layer, right click on the dielectric and select **Map Conductor Via**.
- Create a substrate using the 25 mil Alumina template, with Er = 9.9 and TanD = 0.0009. Define the conductor layer as Gold with conductivity of 4.1E7 and thickness of 0.7 mil. The "hole" layer will also have Gold conductivity as shown in Figure 38.





5. Setup the frequency plan as needed for schematic simulation, e.g. in this case we will keep it from 0.01 GHz to 1 GHz with 101 points.

Step 2: Create EM/Circuit Co-Simulation Component and Symbol

- To set up EM/circuit co-simulation, a model will need to be created. Go to EM > Component > Create EM Model and Symbol. Select both options and press OK. This is shown in Figure 39.
- 2. Click Simulate to begin the Momentum simulation.

EM Lab4b_EMCircuitCosim_	lib:Lab4b_EMCircuitCosim_schematic:emSetup * (EM Setup for simulation)	_		\times
File Tools View Help				
🔚 🖲 🍠 🥙 皆	🕅 🗱 🔁 늘 🗁 🗐 🗊 🗞 🏧			
 Mom uW Layout Partitioning Substrate Ports Frequency plan Output plan Output plan Resources Model Notes 	EM Model View name: emModel Append text to the name Content Reduce to decrease size Only S-Parameter data. No currents nor fields for post-processing. Em EM Model exists when the simulation is launched Update the existing EM Model when the simulation finishes	Au X "emModel" ymbol"	uto-update	Now
	Ge OK	Cancel	Simu	late

Figure 39. Creating the emModel and symbol

Step 3: Simulation and Database Generation Process

- 1. When the simulation is finished, the Momentum data display will be opened, but we do not need to see it right now as the discrete components are not part of the simulation.
- 2. Observe the ADS main window and we will notice 'emModel' and 'symbol' now appearing under the cell on which the simulation is being performed...we will use 'emModel' for EM/Circuit co-simulation. This is shown in Figure 40.



Figure 40. emModel and symbol shown in main window

 Open a new schematic cell, drag and drop the emModel from the Main Window on this schematic cell. From the tool bar, click Choose view for simulation and select emModel from the list as shown in Figure 41.

-																			•		<u> </u>	•
٠	÷				۰.	·		•	۲			• •				۲		•		۲	n an	•
E	•	·	÷									• •			·						<mark>.</mark>	•
·	·	·	·	·	·			·	•			•			·	·	•	·	•		<mark>.</mark>	•
	•	·	·	•		·	•	·				•		•	·	·	•	·	•		Choose View for Simulation	\times
1	·			·	·	·		•	•						•	·		·	•	•		
	•	•			•	•		•					j.	·		•	1	•	1	•		
Ι.	·	•		·	·	·		•	•			• •		•	·	·		•	•	•		
	·	·	·	·	·	·		·	•			• •		·	·	·	÷	·	·	•	Use this view for simulation:	
•	•	·	·	÷	•	•	•	•	•						•	•		•	•		EM emModel	
1	·	•	•									۲	>								S lavout	
1	·	·				•			1			• •		•	1	•		•				
1		•	•				1															
1	·	·			•	·		•	•	•		• •		•	•	·	•	1		•	schematic1	
-											_										⊳ symbol	
•	La	b4t)_E	MC	Irc	uitC	osi	m	lay	ou	t_n	hm .		·	·	·	·	·	·	•		
•	X1	IIVIC	Jael	•				•							•	·	•	•	•			
	~	•																				
•		•																			OK Cancel Help	
·		·																				

Figure 41. emModel shown in the new schematic

- 4. To change the size of the symbol, open the symbol file from the main window. When in the edit window, select all components. Under Edit > Scale/Oversize, select Scale. Change the values to the scale (for example, 0.5) and click OK. When returning to the schematic view, you may need to update the symbol. Go to Edit > Component > Update Component Definitions. Click OK.
- 5. Place the desired discrete components on the schematic and connect them to the layout component as shown in Figure 42.



Figure 42. Discrete components connected to the emModel

6. Run the simulation.



7. In the data display, insert a rectangular plot to observe S(1,1) and S(2,1) as shown in Figure 43.

Figure 43. S(1,1) and S(2,1) plot for emModel with discrete components

Results and Conclusions

This exercise was not focused on creating a specific circuit (i.e. a bandpass filter). Instead, the focus of this lab was to become familiar with the process of EM/circuit co-simulation. The layout was first constructed using MLIN and and MTEE components, with pins added where the discrete components would be placed. The layout model was then placed into a schematic view where discrete components were connected. This simulation method allows for discrete components to be added into an EM simulation.

Conclusion

Congratulations! You have completed Planar Electromagnetic (EM) Simulation in ADS! Check out more examples: www.keysight.com/find/eesof-ads-rfmw-examples.

Learn more at: www.keysight.com

For more information on Keysight Technologies' products, applications or services, please contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus

