PeakView EM Design & Synthesis Training Lab December 2006



Table of Contents

Install Lab	1
Lesson 1: PeakView Environmental Variables	
Lesson 2: Get Started	
Lesson 3: Get familiar with Peakview	7
Lesson 4: Simulate inductor	14
Lesson 5: Explore circuit parameters	18
Lesson 6: Synthesis a symmetric cell	26
Lesson 7: More on Peakview	28
Lesson 8: MLS Inductor	34
Lesson 9: Use Peakview inductor in Virtuoso.	36
Lesson 10: LEM training	38

Install Lab

Welcome to PeakView's EM Design and Synthesis personal training lab. Prepared here are ten different lessons to help the student quickly learn the in's and out's of using PeakView in conjunction with the Cadence Analog Design Environment. Upon completion of the lab set, the student will be able to use PeakView for synthesizing a broad range of passive components and/or modeling custom designed components.

First, the student should install the lab by following these easy to follow steps:

1. Untar PeakviewTraining.tar.gz

> gunzip < PeakviewTraining.tar.gz | tar xvf -</p>

This will generate a Training directory.

2. Set up the environment variable PEAKVIEW_TRAINING_PATH in the shell initialization file, (**.bashrc** or **.cshrc**, etc...) to point to the lab's installed path. In the bash shell, it is

export PEAKVIEW_TRAINING_PATH=Lab_Install_Path

For example

export PEAKVIEW_TRAINING_PATH=~/Training

Next, source the shell initialization file.

3. Run "setupLab" utility to make sure the lab setting is correct.

> cd Training

> bin/setupLab

Lesson 1: PeakView Environmental Variables

Objective: In Lesson 1, the student will learn how to check PeakView's (referred to as PEAKHOME) environment variables and set up Cadence's "**.cdsinit**" to enable the PeakView integration with the Cadence design environment.

Please follow these steps:

- 1. Open a UNIX terminal, and check PEAKHOME environment.
- 2.

echo \$PEAKHOME

Check the path of PEAKHOME; this is where the PeakView software is installed.

Note: The PeakView software uses the PEAKHOME environment to point to its installation directory. The student should make sure it is properly set.

3. Go to the Lab directory to check the ".cdsinit" setup.



Check the following statement in ".cdsinit" file.

loadContext(strcat(getShellEnvVar("PEAKH
OME")

Note: The student has to add these two statements in their ".cdsinit" file to instruct Cadence load PeakView SKILL wrapper. Depending on the student's environment, these statements can be added to either the "**\$HOME**/.cdsinit" file or to the ".cdsinit" file in the working directory. Ofter times, the CAD group may also provide a system-wised setup for all users. For these personal labs, please add these statements to the ".cdsinit" in the working directory.

4. Check the ".peakviewinit" file:

> cd tutorial/.peakview
> vi .peakviewinit

".**peakviewinit**" is the PeakView initialization file. The student should copy this file to their "**\$HOME**/.**peakviewinit**", or add it to the working directory. The CAD group can also create a system-wised ".**peakviewinit**" which can be shared by all users. For more details, please reference the user's guide.

Lesson 2: Get Started

Objective: In Lesson 2, the student will invoke PeakView from inside the Cadence environment, and run an EM simulation for a simple symmetric inductor.

- 1. Go to Lab directory and start Cadence:
 - > cd \$PEAKVIEW_TRAINING_PATH/Lab
 > icfb &
- 2. In CIW window, click File→Open... to bring up the "Open File" form. Make the following changes and click OK of the form and the lab's HP_filter schematic window will pops up.

	L C	ibrary Name Cell Name	tutorial HP_filter	
V Open	File			/////// ×
ок	Cancel De	faults		Help
Library N	ame tuto	orial 🖃	Cell Names	
Cell Name	e HP_f:	ilter	HP_filter M1 NACTIVE	- F
View Nam	ne scher	natic 💷	M1_NWELL M1_PACTIVE	
	Brov	/se	M1_POLY2 M1_PSUB	
Mode	🔶 edi	t 🔷 read	M2_M1 M3_M2 M4_M3	
Library pa	ath file		M5_M4 MET1_Rect_T	
/home/s	taff/ltao/Tr	aining/Lab/cds.	Libi MET2_Rect_T MET3_Rect_T	7

 In the HP_filter schematic window, select Tools→PkView on the schematic window. A new menu item PkView will appear in the schematic window banner.



 Select the PkView menu item in the schematic window. There are four command choices "Synthesis Cells", "Insert Device", "EM integrity", and "Check in License". Opening a layout window, the student will also find "LEM" command too. In Lesson 2, the student will use "Synthesis Cells", "LEM", and the "Insert Device" command.



5. Click the "Synthesis Cells" command on the PkView menu. The "Synthesis Cells" dialog will pop up. Make sure the "tutorial" library is selected and Click OK. The PeakView main window will pop up.

Y Syr	nthesis Cells			////// ×
ок	Cancel			Help
Synth	esis for Library	tutorial	tutorial	

6. Check the PeakView window.

There are four sub windows in the PeakView window, **Data Browser** window, **Waveform Viewer** window, the **Layout Window** and the **Parameter Window**.



Lesson 3: Getting Familiar with PeakView

In this Lesson 3, the student will learn how design and simulate a symmetric inductor, and get introduced to PeakView's basic features.

1. In PeakView window, click the "Circuit" menu, and then click "Symmetric" icon.



Examine the "**Circuit Parameter Editor**" form. This form has many parameters which defines the physical layout reflected in the layout view window. Please refer the PeakView User Guide for more details regarding the "Circuit Parameter Editor" dialog. For now, don't make any changes and leave the default settings. Click "**OK**".

Circuit Paramet	ter Editor			×
cir	cuit name	Symmetric	:1	
ou	uter radius	5e-05		
	width	5e-06		
	spacing	1e-06		
No	o. of sides	8		
N	o. of turns	3		
vertic	al opening			
er	nd opening	2e-05		
ä	arm length	3e-05		
	layer	m6 🗹		
c	ross layer	m5 🗹		
	ratio	1.0		
	width step	0		
synthesize Diff indu	ictance on	Not specif	ied 🗹	
	@	0.1g		Hz
			Advance]
Ok	Draw	v 🗌	Cancel	

2. Check the Layout Viewer window, you will find the layout of the symmetric inductor. Adjust the sub window size to make layout window larger.



3. Move cursor to layout window, and press "f" key. This will fit the layout in Layout Window. Zoom-in on the layout by moving cursor to the lower left of layout window, press and hold the left mouse button and drag the cursor diagonally to the upper right and release when the inductor is vertically bisected. Make sure to include the desired wire paths as part of the selection





4.

The layout window should looks like this.



Zoom-in on the layout window near the underpass wire path. It should look similar to this picture.



5. The student needs to add center-tap to this device, but first measure to check the wire width needed. Find the coordinate and distance sub-windows (two little boxes under layout sub window). The left one sub-window shows the coordinates of the cursor, the right sub-window shows the distance between two clicks. If dx, dy and D in the right one is not 0, double click left mouse button to make them 0.

<u></u>	3
x=66.9, y=-33.4	dx=0.00, dy=0.00, D=0.00

6. Move the cursor to edge of a path shown below, click left mouse button (first click), and notice the coordinate has changed. Next move the cursor to another parallel edge, click left mouse button again (second click). Now the coordinate sub window has change again and so has the values for dx, dy and D. Here the measurement shows a dx value neare 5. This is the width of the inductor we specified in the parameter form.



7. Press 'f' key to ft the layout. Move cursor to the layout window, and click right mouse button. A layout editing pop-up window will appear.



8. Click the "**Customize Layer...**" menu item. The "**Customize Layers**" dialog window pops up. Click the "**m5**" button and turn off "**m5**".

Customize	e Layers		×
on/off	via_m5_m6		edit
on/off	text	abc	edit
on/off	m6		edit
니 on/off	m5		edit
Ok	Apply	Cancel	

9. Click the **Edit** button of m6. The "**Layer Editor**" dialog window pops up. You can select color and pattern for m6. Pick one you like.



10. Click **OK** on the "**Layer Editor**" dialog; click **OK** on the "**Customize Layers**" dialog. Review the changes made to the color in the layout window. If necessary, press "f" key to fit the layout to window size. The layout might look something like this one below.



11. Repeat step 7, open "**Customize Layers**" window, turn on layer "**m5**", and click **OK** of that form. Review the changes on the layout and fit to the window size if necessary.

Lesson 4: EM Simulation

In this Lesson 4, the student will learn how to run EM simulation on the symmetric inductor designed in Lesson 3.

1. Adjust the PeakView sub windows making "Waveform Viewer" window larger.



2. Click the "**Run**" (EM simulation) icon (traffic light) on the PeakView toolbar. This will begin to simulate the symmetric inductor (or the currently selected device) previously designed.

♥ PeakDesign .peakview X018	
Project Circuit Edit Task W	/indows
	🕹 🔳 🦉 💼 👘
T Name Show L(nH) @ 0.1G	Hz Qmax Ld(nH) @ 0.1
	\sim
	Run EM
	Simulation But

3. A progress bar will pop up displaying the simulation progress. For this device, it should take only a couple of minutes to complete.

and a second	.peakview/Symmetric1
	Peakview is running
-	26%
	<u>C</u> ancel
	26%

4. Upon simulation completion, a waveform will appear in the "**Waveform Viewer**. It might look something like this:



5. Move the cursor to the "**Waveform Viewer**" window, click the right mouse button. The "**Formula List**" will pop up. Notice **Q** is selected (check box) on this dialog, this instructs PeakView to display the plot for the value for Q as a function of frequency.

	⊭ Q	
	E	
	R	
	Qd	
	Ld	
	L2	
	L12	
	Configure	
	Export Data	
	Print	
.0	1.2e+10	4e

- 6. In formula list window, click **L**, you will see the L plot displayed in the waveform window. Repeat this procedure, picking the other values (R, Qd, Ld, L2,...) and displaying their respective waveforms.
- 7. Repeat step 5, pop up the formula list window again, and select "Configure...". The "Formula Manager" dialog will pop up. Here, the student can check the definition for Q, L, R, etc... All of these definitions user-configurable. Later in this Lesson, the student will have the opportunity to configure their own formulas. Cancel this dialog at this time.

Y Formula	a Manager		×
Name	Formula	Add	
Q	-1*imag/real(y11)	<u>D</u> elete	
L	1/(2*pi*f)*imag(1/y11)	<u>M</u> odify	
R	1*real(1/y11)		
Qd	1*imag(1/(y11+y22-y12-y21))/real(1/(y	Ţ	
Ld	1/(2*pi*f)*imag(4/(y11+y22-y12-y21))		-
L2	1/(2*pi*f)*(1/y22).imag	ОК	키
L12	-1/(2*pi*f)*zz12.imag	Cancel	
<u> </u>			

8. Repeat step 5, popup the "Formula List" again. This time, select the Q value. Move cursor to "Waveform Viewer" window. Position the cursor somewhere near the apex in the waveform. Click the left mouse button, and then dragging the cursor to "box in" a portion of the curve and releasing will zoom in the display on the waveform. Fit the curve to the window and try zooming in on a different part of the curve.



Lesson 5: Explore Circuit Parameters

In this Lesson 5, the student will copy the previously designed Symmetric1 inductor and modify some of its parameters to investigate their impact on the devices electrical characteristics.

1. Click the PeakView Edit menu item, and click "Copy".

pe	akview XU18		11
t	Edit Task Windows		
9	🖶 Edit		
ov	Cut	Ctrl+X	G
	Сору	Ctrl+C	
	Paste	Ctrl+V	
12000	Delete	Del	
10000	Hide Chart		
	🔏 Chart Color		
	Hide All		
100000000000000000000000000000000000000	Customize Brows	er	
	Options		
1			

Click the PeakView Edit menu item, and click "Paste". A new device is created and added to the "Data Browser" window with the name "copy_of_Symmetric1". The student may need to adjust the width of the display columns to better see the device name and the other column parameters.

🖻 🔲 🗐 🗇 🖗	© 🕸	🕹 🕻 🕻 📥				
Name	Show	L(nH) @ 0.1G Hz	Qmax	Ld(nH) @ 0.1G Hz	QdMax	Resonant Freq.
P Symmetric1		NA	10.34	NA	13	N
P copy_of_Symmetric1		NA	10.34	NA	13	N

3. Double click **copy_of_Symmetric1** cell and the "**Circuit Parameter Editor**" dialog will pop up. Change the following parameters:

Circuit Name	barber
Outer radius	7e-05
Width	8u

Click "**Draw**" button on the "**Circuit Parameter Editor**" dialog, examine layout window to make sure the layout is reasonable and changed parameter values were realistic. Click **OK** on the dialog.

✔ Circuit Para	meter Editor		<u>//</u> ×
	circuit name	barber	
	outer radius	7e-05	
	width	8u	-
	spacing	1e-06	-
	No. of sides	8	-
	No. of turns	3	-
V	ertical opening		-
	end opening	2e-05	-
	arm length	3e-05	-
	layer	m6 🝸	
	cross layer	m5 🝸	
	ratio	1.0	-
	width step	0	-
synthesize Diff	inductance on	Not specified	
]
	@	0.1g	Hz
		Advanc	e
Ok	Draw	v Cance	el

4. Start a new EM simulation by clicking the "**Run**" button. PeakView will simulate the new inductor. Before checking the simulation result, predict how the electrical characteristics will change when compared to the original Symmetric1 inductor. If the width and outer radius are made larger , how this will affect L, R, and Q? Go to next page to see our prediction.

Our prediction: Increasing the outer radius increases the total length of the winding. Doing so, we expect the inductance to increase. The overall area of the device is also increased; as a result, the capacitance should also increase.

Recall.... $f_{res} \propto \left. \frac{1}{\sqrt{LC}} \right.$, as L & C increase, f_{res} will decrease.

Therefore, we expect to see a steeper L curve at lower frequencies for "barber" than for "symmetric".

It is hard to predict R at this time, increasing width will decrease R, but increasing outer radius will increase R, therefore, the combined effect cannot be predicted with any certainty.

Since Q depends on R,L, and C, an accurate prediction on Q behavior is also not possible at this time.

After the EM simulation completes, examine the "Waveform Viewer". Two curves will appear for each electrical plot. They have exactly the same color. But, the curve that is highlighted is the curve of barber.



5. Move cursor to the **barber** cell in the **Data Browser** sub window, click right mouse button, the "**Edit Circuit**" dialog will pop-up. Click the "**Chart Color...**" item. The "**Select Color**" window will pop up.



6. In the "Select Color" window, pick a new color for all barber waveforms. OK this form. Notice the color of the barber curve has changed.

Select color	×
Basic colors	
<u>C</u> ustom colors	Hue: 0 Red: 255
	Set: DEE Green: 0
Define Custom Colors >>	<u>V</u> al: 255 Bl <u>u</u> e: 0
	Add to Custom Colors

7. Move cursor to "**Waveform Viewer**" window, pressing the "f" key will fit the waveform to the window. Compare the plots for Q, L and R. Do they agree with the prediction? The L plot might look something like the picture below. What other information can you deduce from these plots?



- 8. Based on the previous **barber** discussion, one should be able to only predict as the outer radius increases, L will increase, the resonant frequency will decrease. Now, let's confirm this idea by expanding the investigation and performing a parameter sweep on the outer radius.
- 9. Click the PeakView **Circuit** menu, select the **Symmetric** device. Change its name to **rabbit**, and insert a variable "**x**" for the outer radius value.

Circuit Paramet	er Editor			×
cir	cuit name	rabbit		
O	uter radius	x		
	width	5e-06		
	spacing	1e-06		
N	o. of sides	8		
N	o. of turns	3		
vertic	al opening			
er	id opening	2e-05		
a	arm length	3e-05		
	layer	m6 ⊻		
c	ross layer	m5 ⊻		
	ratio	1.0		
	width step	0		
synthesize Diff indu	ctance on	Not specifi	ed 🗹	
	@	0.1g		Hz
			Advance	
Ok	Draw	v	Cancel	

10. Clicking OK on the "Circuit Parameter Editor" dialog will pop up of the "Sweep Parameter Setting" dialog. Previously, the outer radius value was set as a variable x. Here, the sweep values for x must be specified. Make the following changes to the dialog values:

	Start Stop Points	5e-05 8e-05 4			
♥ Sweet	ep Param	eter Setti	ng		×
x start	5e-05	end 8e-0	5 points	4	
	k		Cancel		

11. Click **OK** on "**Sweep Parameter Setting**" dialog. Notice in the Data Browser sub window, four new "**children**" cells have been added under the "**parent**" cell rabbit with names rabbit_1, rabbit_2, rabbit_3, rabbit_4.

q.
NA
+10
4

12. Adjust the PeakView "Layout Viewer" window to making it larger. Selecting rabbit_1 in the "Data Browser" window will display its layout in the "Layout Viewer" and the exact parameters are recorded in the "Parameter Window". The student can cycle through the other children cells by selecting rabbit_2, rabbit_3 and rabbit_3 in the "Data Browser" window.



- 13. Measure (learned in Lesson 4) the outer radius of each "child" cell to make sure the parameter sweep was correctly set. Double check the measurement by comparing to the outer radius parameter value in the "Parameter Window".
- 14. Readjust the PeakView "Waveform Viewer" making it larger. Select the **parent** cell **rabbit** in the "**Data Browser**" window, and click "**Run**" button to simulate all four "**children**" cells. This will take several minutes to complete.

15. When the EM simulation completes, select the Symmetric1 cell in "Data Browser" window, click the right mouse button. The "Edit Circuit" dialog pops up. Select "Hide Chart" to the hide waveforms for the Symmetric1 inductor. Repeat this step to hide curves of barber cell too.



16. Now, there are four curves of rabbits remaining in the wave form window. Select rabit_1, the curve of rabbit_1 will be highlighted. Select rabbit_2, rabbit_3 and rabbit_4 to check their curves. Are these curves agreed with what we predicted? You might find more regular patterns from these four curves.

Lesson 6: Synthesis a symmetric cell

In this Lesson 6, the student will be introduced to PeakView's EM synthesis capabilities by synthesizing a 1nH symmetric inductor.

1. Click PeakView's **Circuit** menu, select **Symmetric** item. After the "**Circuit Parameter Editor**" dialog pops up. Change the following parameter values to following:

Name	fox
Synthesis Diff inductance on	outer radius
(set the inductance synthesis target)	1n
(set the target frequency) @	5g

Circuit Parame	ter Editor		×
ci	rcuit name	fox	
o	uter radius	4.7747e-05	
	width	5e-06	
	spacing	1e-06	
N	o. of sides	8	
N	o. of turns	3	
vertic	al opening		
er	nd opening	2e-05	
	arm length	3e-05	
	layer	m6 🝸	
c	ross layer	m5 🝸	
	ratio	1.0	
	width step	0	
synthesize Diff indu	ictance on	outer radiu	s 🗹
		1n	
	@	5g	Hz
			Advance
Ok	Draw	/	Cancel

- 2. Click **OK** on the "**Circuit Parameter Editor**" dialog. Then click the **Run** button to synthesize a symmetric inductor to the specified target inductance value of **1nH** at **5Ghz**.
- 3. After synthesis completes, select the **rabbit** cell in "**Data Browser**" and hide its waveforms.

4. Now, only **fox's** waveforms will appear in the "**Waveform Viewer**". If there are other curves still remaining, hide them. Turn on **fox's L** plot. Zoom in to around 5G Hz to make sure the inductance value is 1nH as targeted.



Lesson 7: Simulation Options and Formluas

In this Lesson 7, the student will learn more details about PeakView's simulation options and how to enter their own waveform formula.

1. Click PeakView's **Circuit** menu item and select **OctCircuit** to see how various types of octagonal structures available.



2. Click PeakView's **Circuit** menu item and select **Advanced** to see the various advanced structures available. Click "**Transformer_p2s1**.



3. Change circuit name on the "Circuit Parameter Editor" dialog to giraffe, and then click the "Advance..." button. The "Advanced Option Editor" dialog will pop up. Change nu to 1, nv to 1, and nz to 0. Here, we will take advantage of PeakView's ability to trade-off EM simulation runtime vs. accuracy. This is often useful when exploring specific device architectures when many iteration are required. Later, after converging on a useful structure, the student can reset the simulation options for higher accuracy. Here, the setting will instruct PeakView to run the EM simulation faster but with less accuracy.

Advanced Option Editor	×
Simulation Option nu	
Slotting options	default 💆
☐ guard ring ☐ rectangl	e shield circular shield
Ok	Cancel

- 4. Click **OK** on the "**Advanced Option Editor**" form; click **OK** of "**Circuit Parameter Editor**" dialog. And then click "**Run**" button
- 5. After the simulation finishes, examine the transformer's waveform plots. If there are any other electrical curves, hide them
- 6. Move the cursor to the "Waveform Viewer", clicking the right mouse button will bring up the "Formula List" window. Click "Configure..." and the "Formula Manager" dialog will pop up.

7. In the "Formula Manager" window, click "Add" to open the "Formula Editor" dialog as shown in the following figure.



8. In the "Formula Editor", replace "new" with "K", check the lower radio button ,and input following equation into the open box:

imag(zz21)/sqrt(abs(imag(zz11)*imag(zz22)))

This equation defines the coupling coefficient for this transformer. Click "**Ok**" to save this formula. The new formula for "**K**" is now shown in "**Formula Manager**" window.

✓		Formul	a Editor		× (
Definit	ion of				
K	- ÷ <u>1</u>	<u> </u>	imag	<u> </u>	<u> 11 </u>
	* limad	zz21)/sa	rt(abs(ir	nad(zz11	1)*ima <u>@</u>
	Ok			Ca	ncel



9. Place cursor on the "**Waveform Viewer**", click the right mouse button and select "**K**" in the formula list. The waveform for the coupling coefficient will be displayed.



10. Repeat steps 6 and 7 to bring up "**Formula Editor**". This time, use the dropdowns to create a new formula. The dropdown is useful for simple parameters like S, Y, Z parameters for two port networks. Change the formula as in the following picture. Here the S21 parameter is defined a formula.

•		Formula Editor	3
Definit	ion of		
S21	-* 1	<u> </u>	<u> (</u> s ⊻ 21 [])
	~		
	Ok		Cancel
			Cancel

- 11. Click **OK** on the "**Formula Editor**" dialog; click **OK** on the "**Formula Manager**" window. Move the cursor to the "**Waveform Viewer**" window and display the **S21** waveform just defined. Turn off waveforms for other devices if necessary.
- 12. Examine layout of transformer in layout window
- 13. Move cursor to the top of "**Data Browser**" sub window, click the right mouse button, select "**Customize Browser** ...",



The "**Column Formula Editor**" pops up. Turn on the "**Area**" button at the bottom of this form and click **OK**.

~	Column Formula Editor		×
	Name of characteristic	Formula	Add
	☑ L(nH) @ 0.1G Hz	1/(2*pi*f)*(1/y11).imag*1e9	Delete
	🗹 _{Qmax}	-1*y11.imag/y11.real	Delete
	☑ Ld(nH) @ 0.1G Hz	1/(2*pi*f)*imag(4/(y11+y22-y12-y2	Modify
	☑ _{QdMax}	1*imag(1/(y11+y22-y12-y21))/real(
	Resonant Freq.	1/(2*pi*f)*(1/y11).imag*1e9	.
	□ L2(nH) @ 0.1G Hz	1/(2*pi*f)*(1/y22).imag*1e9	
	□ L12(nH) @ 0.1G Hz	-1/(2*pi*f)*zz12.imag*1e9	
	4		<u>C</u> ancel
	Additional data item:	Area	

14. The "**Area**" column will now appear in the "**Data Browser**" window. The area of each cell will be shown in the "**Data Browser**" window. Click the **Area** column heading and the cells will be sorted by area.

1		* * *		U U					
Т	Name	Show	L(nH) @ 0.1G Hz	Qmax	Ld(nH) @ 0.1G Hz	QdMax	Resonant Freq.	Area	A
-P	fox		NA	10.59	NA	13.34	NA	1.1259e-08	
¢-₽	rabbit							1.1989e-08	
	-rabbit_1		NA	9.062	NA	10.89	1.985e+10	1.1989e-08	
	-rabbit_2		NA	10.34	NA	13	NA	1.2162e-08	
	-rabbit_3		NA	11.32	NA	14.71	NA	1.2336e-08	
	-rabbit_4		NA	12.12	NA	16.24	NA	1.2511e-08	
	Symmetric1		NA	10 34	NA	13	NA	1 2162e-08	M
					· · · · · · · · · · · · · · · · · · ·				

15. Click **Name** or other column heading to sort the cells as desired.

Lesson 8: Advanced Passive Design

In Lesson 8, the student will learn how to design multi-layer structures (MLS). MLS can achieve the same electrical performance but with much smaller area (25-40% less) space. Here the student will design an MLS inductor and compare it to a symmetric inductor of same inductance.

1. Synthesize a symmetric 8nH inductor. Click PeakView's **Circuit** menu and select **Symmetric** to generate a symmetric inductor. Change its circuit name to **Symmetric_8n**, the number of turns to be 5, set synthesis option to "synthesize Diff inductance" on "outer radius", and target inductance to 8nH at 0.1GHz as shown in the following figure.

- 2. Click "**Ok**" of this form and "**Run**" to synthesize the symmetric inductor.
- Click PeakView's Circuit menu and select OctCircuits->Octagon_MLS to generate an MLS inductor. Change its circuit name to Octagon_MLS_8n, the number of turns to be 4, set the layer_options to be m5/m4, set the synthesis option to "synthesize Inductance" on "outer radius", and the target inductance to be 8nH at 0.1GHz as shown in the following figure

Circuit Para	ımeter Editor 🛛 🗶
circuit name	Octagon_MLS_8n
inlen	0.00016
width	5e-06
spacing	1e-06
nturn	4
opening	2e-05
top layer	m6 🗹
layer_options	m5/m4
arm length 1	3e-05
arm length 2	3e-05
synthesize Inductance on	inlen !
	8n
@	0.1g Hz
	Advance
Ok Dra	aw Cancel

- 4. Click **OK** of this form and "**Run**" button of PeakView window to synthesize the MLS structure.
- After completing synthesis, in the "Data Browser" window, click "Area" to sort the cells by area. Compare the area of Octagon_MLS_8n to Symmetric_8n. Although they have same inductance value at 0.1GHz, the area of Symmetric_8n is 5.29e-8m², and the area of Octagon_MLS_8n is only 1.78e-8m², a 66% area savings!

Т	Name	Show	L(nH) @ 0.1G Hz	Qmax	Ld(nH) @ 0.1G Hz	Area 🗸
-P	Octagon_MLS_8n		8.003	6.256	8.007	1.6788e-08
-P	Symmetric_8n		7.992	3.796	7.998	5.2932e-08

Lesson 9: Using PeakView Devices in Cadence.

In this Lesson9, the student will see how easy it is to move all the devices created in PeakView back to the Cadence environment and use them in real design.

1. Click the Save button to save the project.



- 2. Click PeakView's **Project** menu and select **Return** to exit PeakView and transfer all newly created devices back to the Cadence environment.
- In Cadence CIW , open Tools→Library Manager....All the cells designed, simulated or synthesized in PeakView will be listed here. Select barber. The four views available include: layout, spectre, spectreS and symbol.

Library	Cell	View	
tutorial	barber	Ĭ	
IS_8ths hhdLib malogLib basic dsBefTechLib fErxmples fLib putorial	M1_NACTIVE M1_NVELL M1_PACTIVE M1_POLY2 M1_POLY2 M1_PSUB M2_M1 M3_M2 M4_M3 M5_M4 MTT1_Rect_T MTT2_Rect_T MTT3_Rect_T MTT4_Rect_T MTT4_Rect_T MET5_Rect_T Symmetric1 TM_M5 TOPMET_Rect_T barber fox giraffe m1_T m2_T m3_T m4_T m5_T poly2_T rabbit_1 rabbit_2 rabbit_4 ruleVia tm_T zebra	A layout spectre spectres symbol	

- 4. Open the layout and symbol view of the **barber** cell and examine. The layout is identical to the previous PeakView result. Close the layout and symbol windows.
- 5. Open the **HP_filter** schematic window up. Click "**PkView**→**Insert Device** …" .



6. On the "Create PeakView Device" dialog, click the cell cyclic menu to pick one of the cells previously created in PeakView. Notice the "Sim Model Mode" and "Noise Analysis" fields at the bottom of the form. PeakView provides three models for circuit simulation: a broadband subcircuit model, passive subcircuit model and nport model. Select the "Sim Model Mode" option to set the type of model to be netlisted. PeakView also provide noise analysis model. If "yes" is selected for this cell, the netlist procedure will add the noise model to spectre netlist.

✓ Create	e Peakviev	v Device	×		
Hide	Cancel	Defaults	Help		
Library Cell	tutorial zebra	Peakview Graphical Browse			
View	giraffe fox robbit_1				
Names	rabbit_3				
Rotat	-rabbit_1 Charber	Sideways Upside Down			
	Symmetr	ic1			
SubCircu	it File	zebra. š			
SYZ File		zebra. s2p	zebra. s2př		
Sim Mod	el Mode	$igstar{}$ broad_band subckt $igstar{}$ passive s	subckt 🔷 nport		
Noise An	alysis	⇔yes 🔶 no			

7. Cancel "Create PeakView Device" form.

Lesson 10: LEM training

In this Lesson 10, the student will learn how to use PeakView's unique Layout EM Extraction (LEM) capability. LEM provides a "1-click" EM modeling of any passive structure directly from the Virtuoso layout. Suppose the student needed to change the layout created by PeakView. For example, we need to add a center tap to the **barber** cell. To do this, all that is necessary is to add this center tap manually, add a pin to that tap and run LEM to create all the same Cadence design views, CDF parameters as if it were synthesized by PeakView. Here, there is a copy of barber cell with the center tap already added, but the student needs to add the third pin to it.

1. In Library Manager, choose **apprentice_without_pin cell**, click right mouse button, and click "**Copy** .." item.

♥ Library Manager: WorkArea: /hom	e/staff/ltao/Training/Lib		×
<u>File Edit View Design Manager</u>			<u>H</u> elp
🔄 Show Categories 📃 Show Files	;		
- Library	Cell	View	
Itutorial	[apprentice_without_pin	I	
US_8ths ahdlLib analogLib avTech basic cdsDefTechLib functional rfExamples rfLib tutorial	HP_filter M1_NACTIVE M1_NVELL M1_PACTIVE M1_POLV2 M1_PSUB M2_M1 M3_M2 M4_M3 M5_M4 METI_Rect_T MET2_Rect_T MET4_Rect_T MET4_Rect_T MET4_Rect_T Symmetric1 TM_M5 TOPMET_Rect_T apprentice without pin	A layout spectre spectres symbol	
	barber C	Copy	
	giraffe R	lename	
	m1_T D	elete	
	m3_T P	roperties	
	m4_T C	Jieck bi	
	poly2_T C	Jieck Out	
	rabbit_1 C	ancel Checkout	
	rabbit_3	lpdate	
	rabbit_4 S	how file Status	
	Salar S	idmit	
- Messages			
Ť			
A			

2. In the "**Copy Cell**" form, change Cell name of "To" section as **apprentice**. And click **OK** on this form.

	×	
Library	tutorial	
Cell	japprentice_	without_pin
То		
Library	<u>]</u> tutorial	
Cell	apprentice	
Options		
🔄 Copy I	lierarchical	
R :	Skip Libraries	US_8ths ahdlLib analogLib avTech basic cdsDefTechLib
_ 1	Exact Herarchy	? ?
	Extra Views	¥.
Copy	All Views	
Vii	νων Τα Ορμγ	symbol spectreS spectre layout
🔄 Updat	e Instances:	Of Entire Library
		x x

3. Open **apprentice** layout. The center tap is already added. Now, add a pin to the end of this center tap.



4. Zoom in to the under path to make sure MET4 is connected with TOPMET by MET5 and vias. Then press "f" key to fit the layout.



5. Choose MET4 as current layer in LSW window.

¥ LS	SW	-	
Sort	Edit		Help
M	ett4		drw
	tuta	rial	
:	Show ()bject	s
AV	NV	AS	NS
I			
VI.	A2		drw
i ME	тЗ		drw
ME	тЗ		161
VI	A3		drw
in the second se	т4		drw —
	T4 T4		drw 1b1

 Click Create → Pin command in the layout window. Type n3 for "Terminal Names" field. Change Mode to "shape pin", and select "rectangle" as the Mode.

💙 Create Shape P	in	
Hide Cancel		Heij
Terminal Names	nJ	
🔄 Keep First Nam	e X Altch	1 0 Y Pitch 0
Mode	🔶 rectangle	\sim dot \sim polygon \sim circle \sim auto pin \sim sym pin
🔄 Display Pin Nar	ne	Display Pin Name Option
ИО Туре	⇔input ⇔switch	◇ output ◆ inputOutput ◇ jumper
Snap Mode	orthogonal 🖃	
Access Direction	Top Bo	ottom 🔳 Left 📕 Right
As ROD Object	Any _ No	aut
ROD Name	rectO	

7. Create a little rectangle at the left end of the MET4 center tap as shown in the following picture.



- 8. Cancel the **Create Pin** command by press **Esc** key.
- 9. Click **Tools**→**PkView** of **apprentice** layout window. The PkView menu will appear in the layout window.
- 10. Click **PkView→Layout EM→LEM** to invoke LEM.



11. A LEM dialog pops up asking whether to make this cell a PeakView inductor or not. Click **Yes**.



12. A PeakView window will pop up and the EM simulation will automatically start. Notice in the "**Data Browser**" window, the letter "**L**". The other cells have letter "**P**". The "L" designates this cell as a LEM cell, and a "P" designates this cell a PeakView generated cell.

▼ PeakDesign .peakView XU18								
Project Circuit Edit Task Windows								
			國非學					
Т	Name	Show	L(nH) @ 0.10	G Hz	Qma:			
HL	apprentice							
-P	Symmetric1			NA	10.			
⊢P	fox			NA	10.			
¢-₽	rabbit				1000			
- P	zebra			NA	2.6			
– P	giraffe			NA	2.4			
LP	barber			NA	6.			

13. After the EM simulation is finished, the student must click **Project**-→**Return** and PeakView will generate all Cadence views and information as needed.