PeakView
EM Design & Synthesis
Training Lab
December 2006

LORENTZ
SOLUTION

Table of Contents

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 -

   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.

   > cd Lab
   > vi .cdsinit

   Check the following statement in "**.cdsinit**" file.

```
loadContext(strcat(getShellEnvVar("PEAKH
OME")
            "/cds/context/pkv.cxt"))
load(strcat(getShellEnvVar("PEAKHOME")
           "/cds/context/pkview.ini"))
```

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

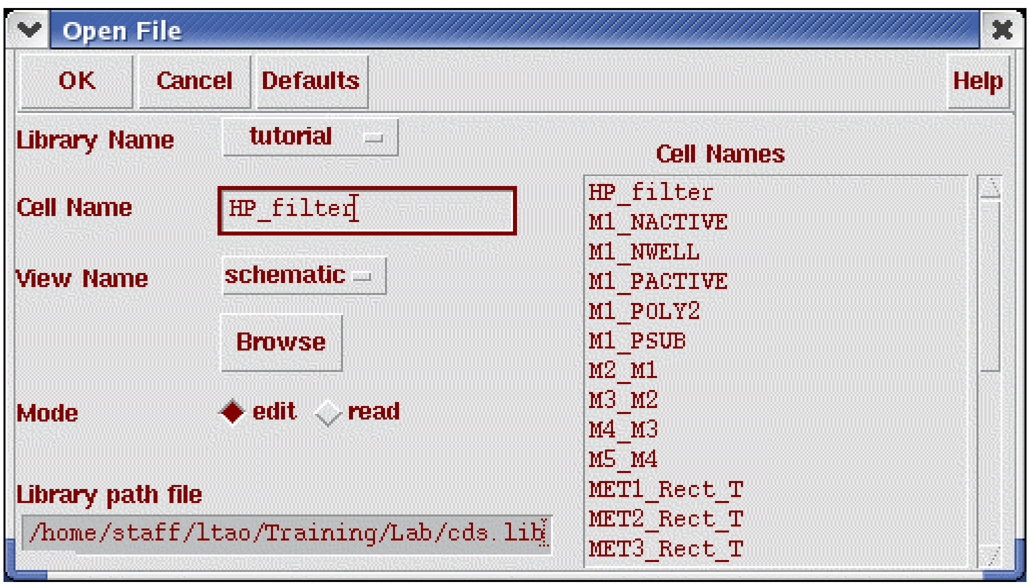
Objective: In Lesson 2, the student will invoke PeakView from inside the Cadence environment, and run an EM simulation for a simple symmetric inductor.

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.

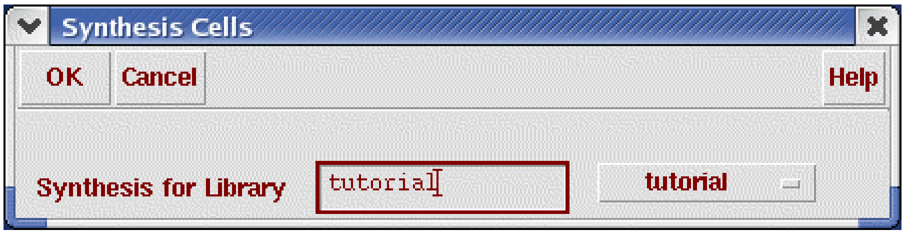
   Library Name tutorial
   Cell Name HP_filter



3. 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.

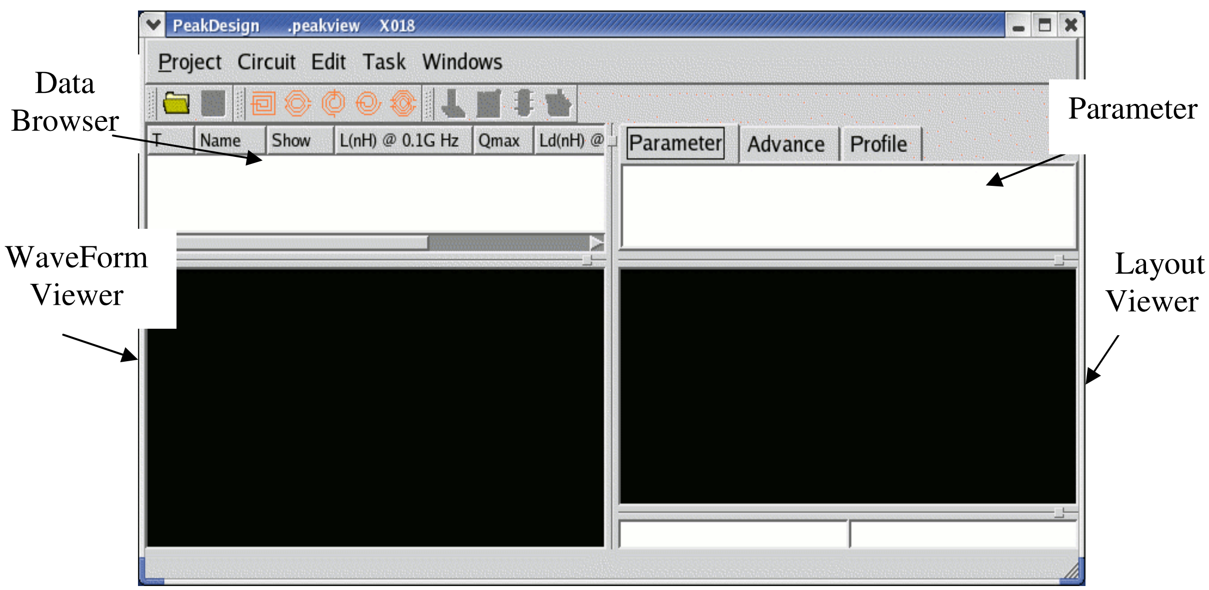
4. Select the PkView menu item in the schematic window. There are four command choices “**Synthesis Cells**”, “**Insert Device**”, “**EM integrity**”,and “**Check in License**”. Opening a layout window, the student will also find “**LEM**” command too. In Lesson 2, the student will use “**Synthesis Cells**”,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



6. Check the PeakView window.

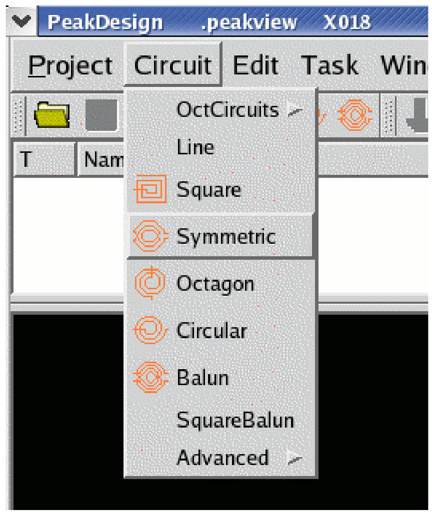
There are four sub windows in the PeakView window, **Data Browser** window, **Waveform Viewer** window, the **Layout Window** and the **Parameter Window**.



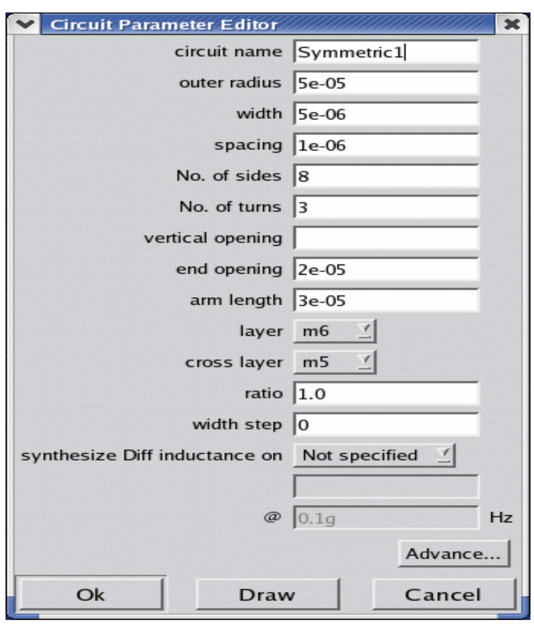
Lesson 3: Getting Familiar with PeakView

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

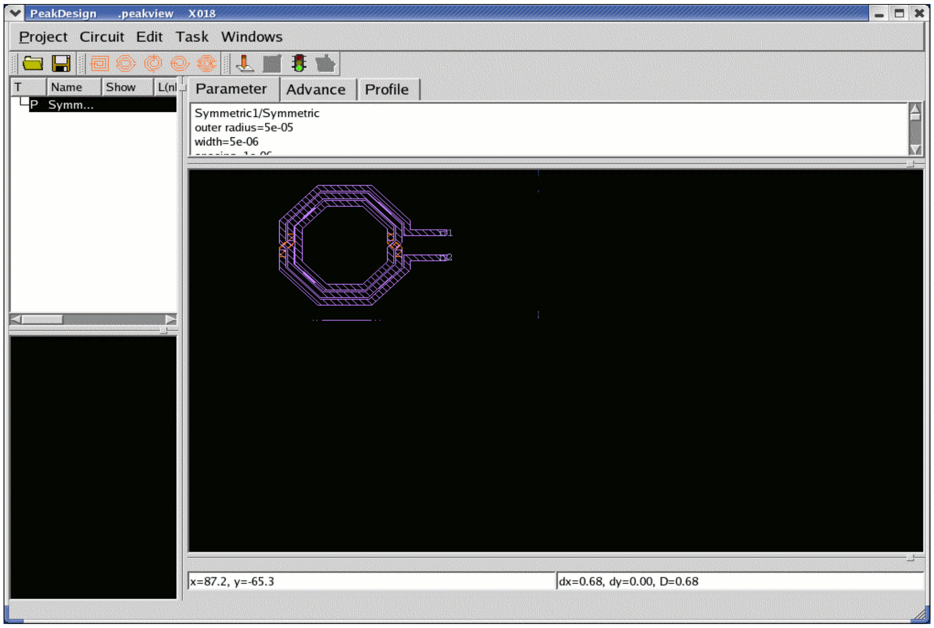
1. In PeakView window, click the "**Circuit**" menu, and then click "**Symmetric**" icon.



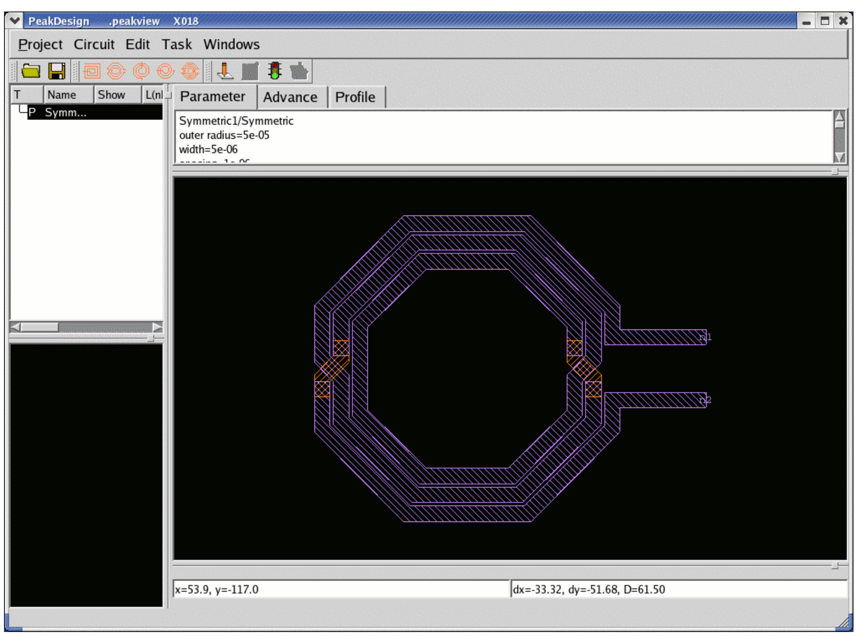
Examine the "**Circuit Parameter Editor**" form. This form has many parameters which defines the physical layout reflected in the layout view window. Please refer the PeakView User Guide for more details regarding the "Circuit Parameter Editor" dialog. For now, don't make any changes and leave the default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

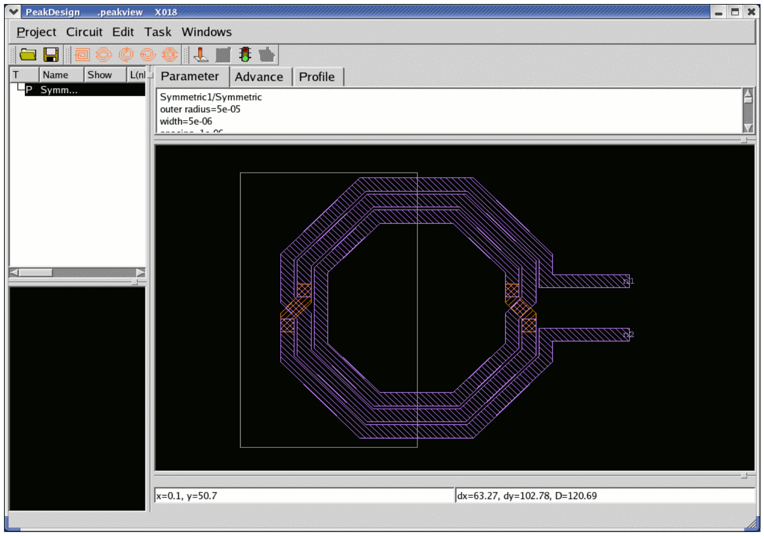


2. Check the Layout Viewer window, you will find the layout of the symmetric inductor. Adjust the sub window size to make layout window larger.



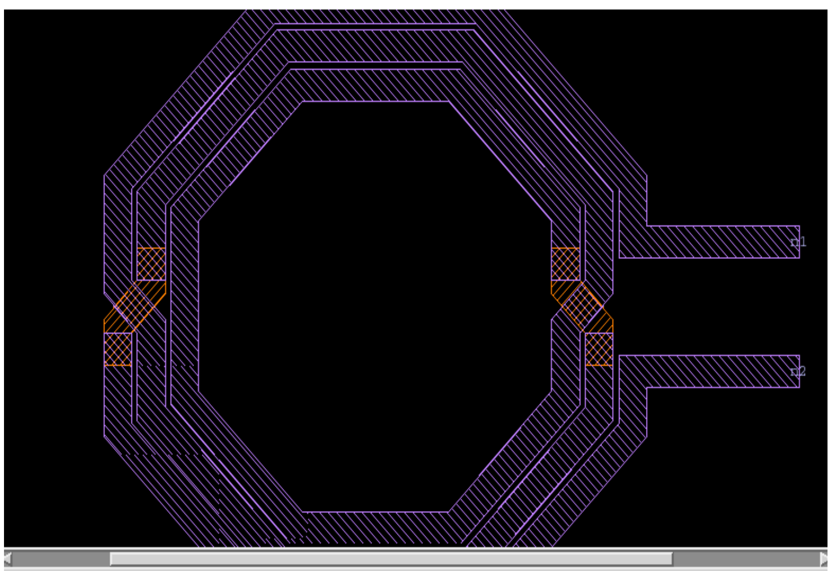
3. Move cursor to layout window, and press “f” key. This will fit the layout in Layout Window. Zoom-in on the layout by moving cursor to the lower left of layout window, press and hold the left mouse button and drag the cursor diagonally to the upper right and release when the inductor is vertically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



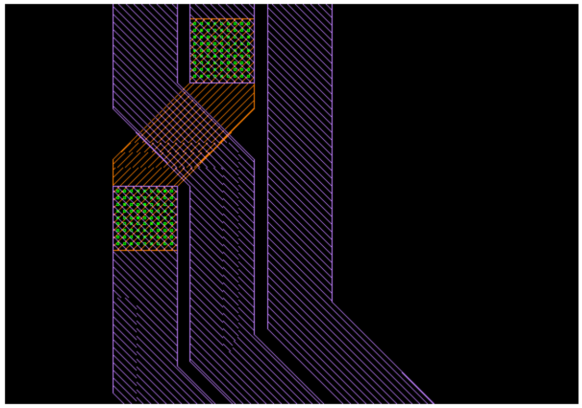


4.

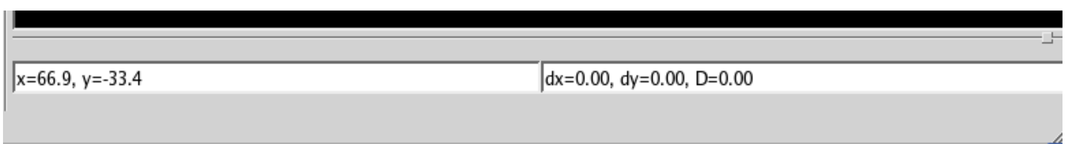
The layout window should looks like this.



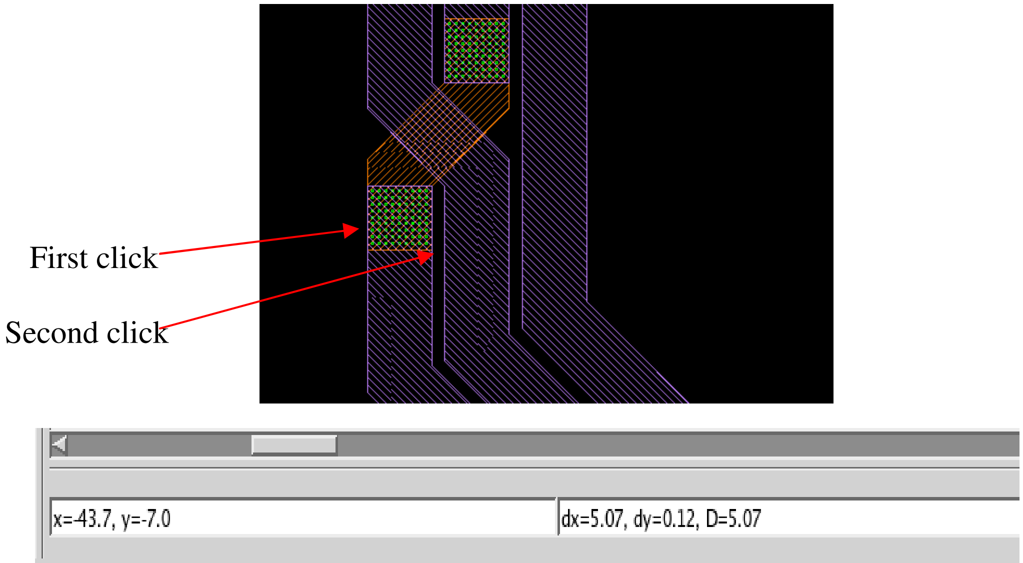
Zoom-in on the layout window near the underpass wire path. It should look similar to this picture.



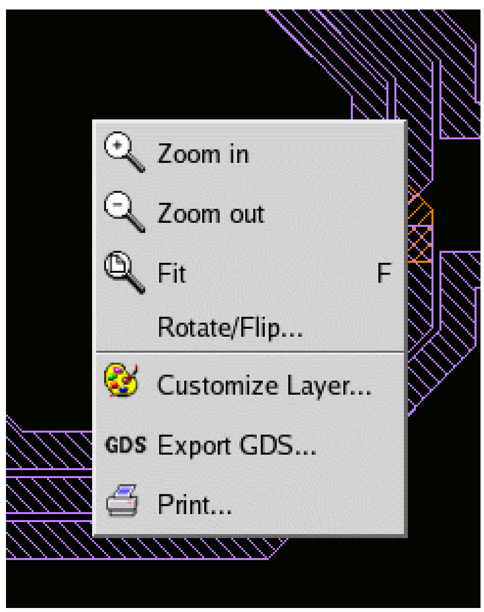
5. The student needs to add center-tap to this device, but first measure to check the wire width needed. Find the coordinate and distance sub-windows (two little boxes under layout sub window). The left one sub-window shows the coordinates of the cursor, the right sub-window shows the distance between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



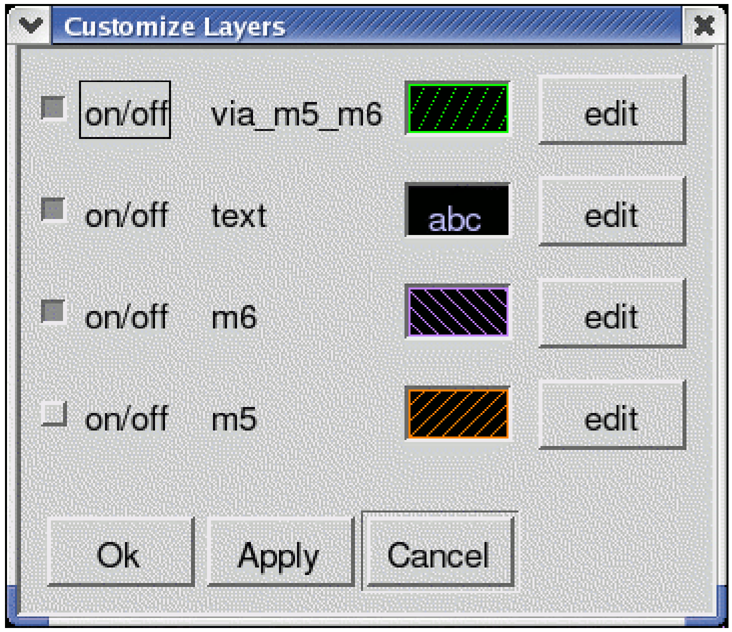
6. Move the cursor to edge of a path shown below, click left mouse button (first click) , and notice the coordinate has changed. Next move the cursor to another parallel edge, click left mouse button again (second click). Now the coordinate sub window has change again and so has the values for dx, dy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



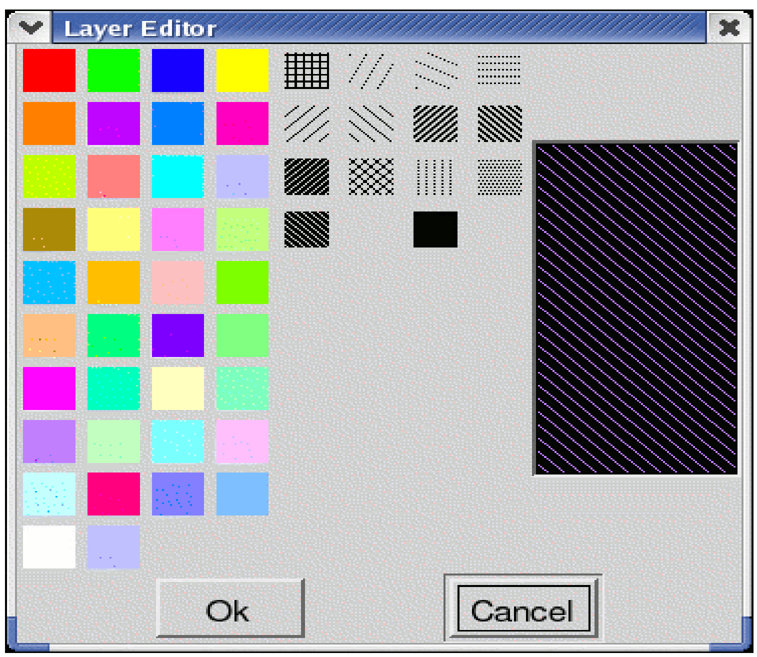
7. Press ‘f’ key to ft the layout. Move cursor to the layout window, and click right mouse button. A layout editing pop-up window will appear.



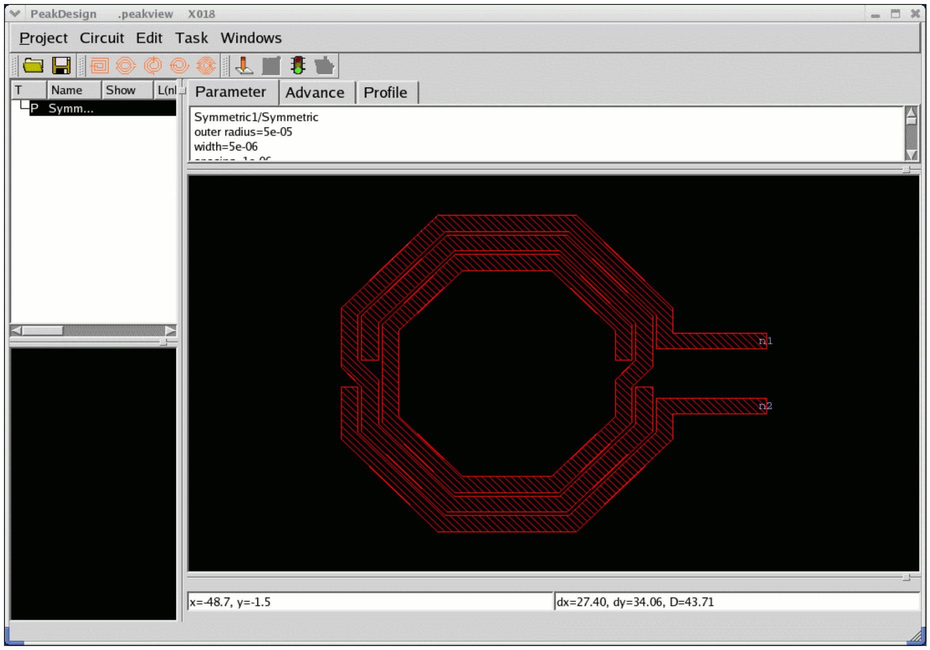
8. Click the “**Customize Layer…**” menu item. The “**Customize Layers**” dialog window pops up. Click the “**m5**” button and turn off “**m5**”.



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.

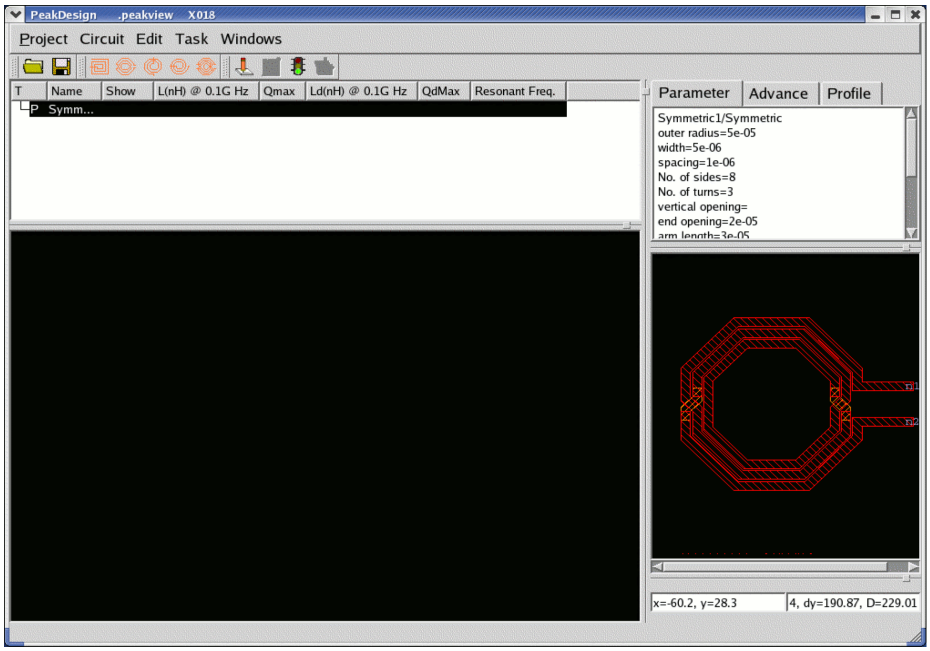


11. Repeat step 7, open "**Customize Layers**" window, turn on layer "**m5**", and click **OK** of that form. Review the changes on the layout and fit to the window size if necessary.

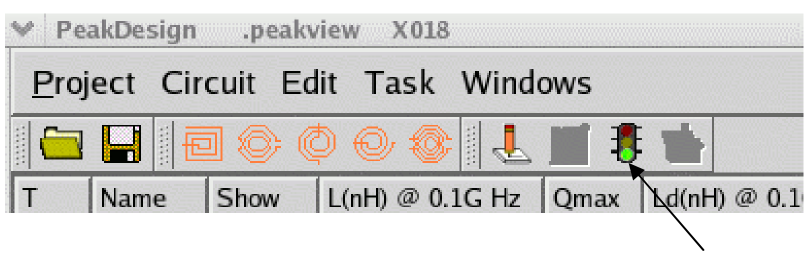
Lesson 4: EM Simulation

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.

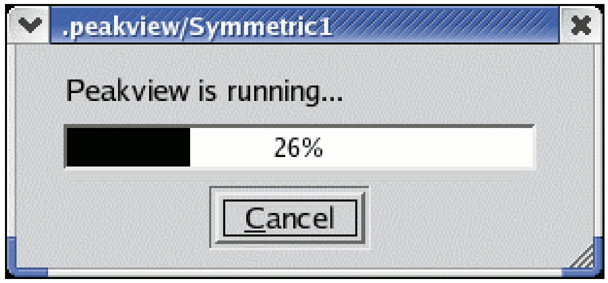


2. Click the “**Run**” (EM simulation) icon (traffic light) on the PeakView toolbar. This will begin to simulate the symmetric inductor (or the currently selected device) previously designed.

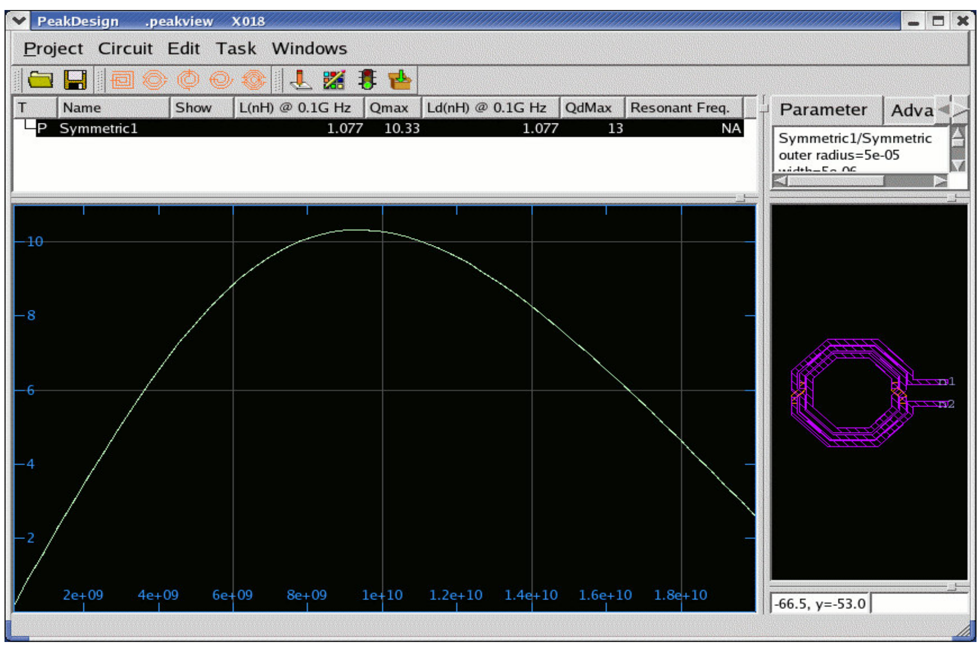


Run EM
Simulation Button

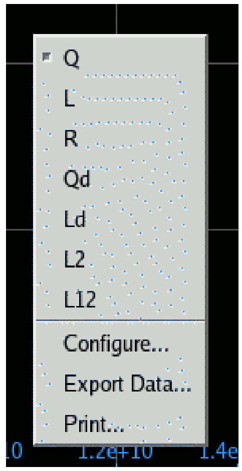
3. A progress bar will pop up displaying the simulation progress. For this device, it should take only a couple of minutes to complete.



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

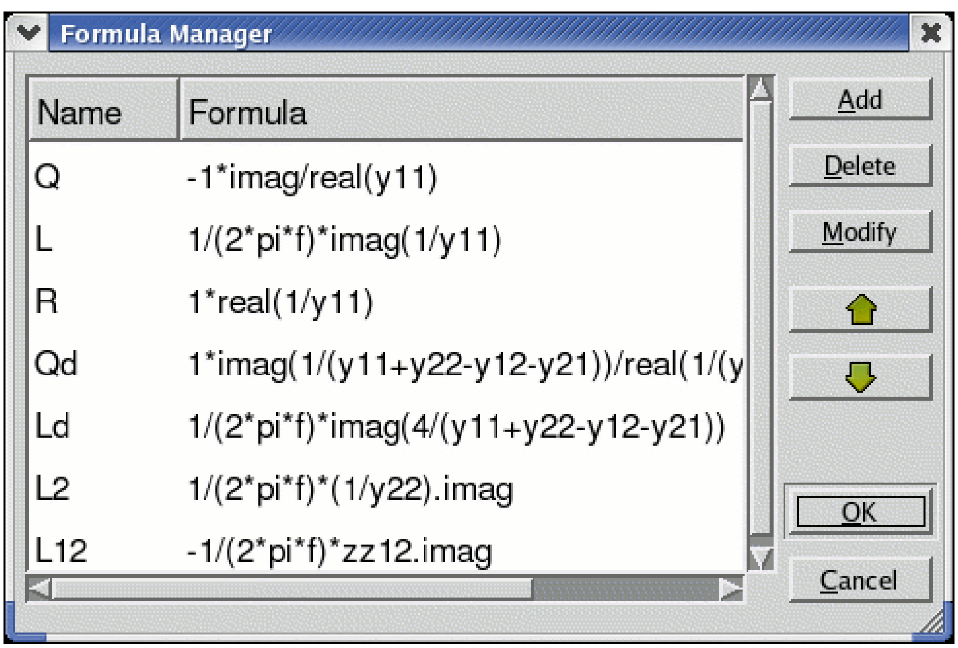


5. Move the cursor to the "**Waveform Viewer**" window, click the right mouse button. The "**Formula List"** will pop up. Notice **Q** is selected (check box) on this dialog, this instructs PeakView to display the plot for the value for Q as a function of frequency.

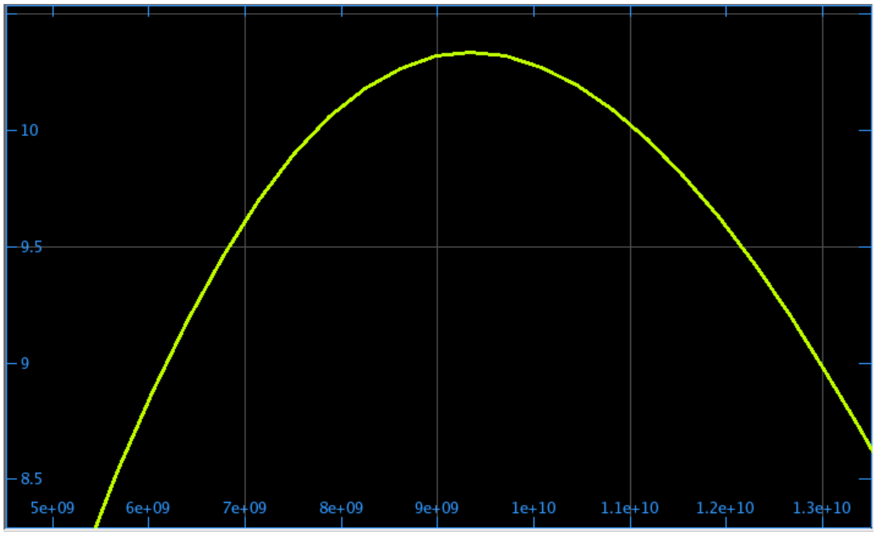


6. In formula list window, click **L**, you will see the L plot displayed in the waveform window. Repeat this procedure, picking the other values (R, Qd, Ld, L2,….) and displaying their respective waveforms.

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.



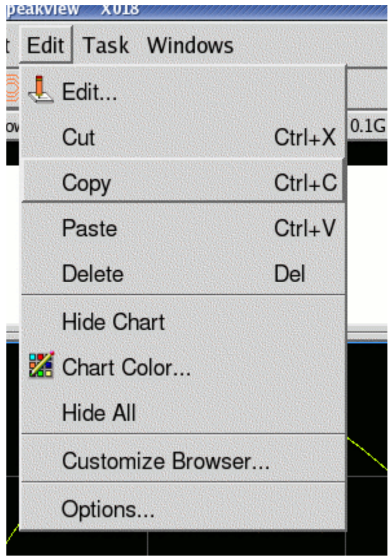
8. Repeat step 5, popup the "**Formula List**" again. This time, select the **Q** value. Move cursor to "**Waveform Viewer**" window. Position the cursor somewhere near the apex in the waveform. Click the left mouse button, and then dragging the cursor to "box in" a portion of the curve and releasing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



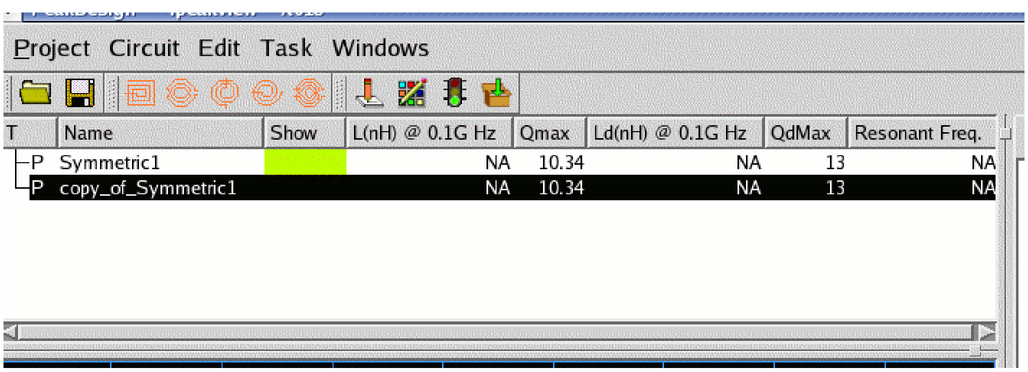
Lesson 5: Explore Circuit Parameters

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**”.



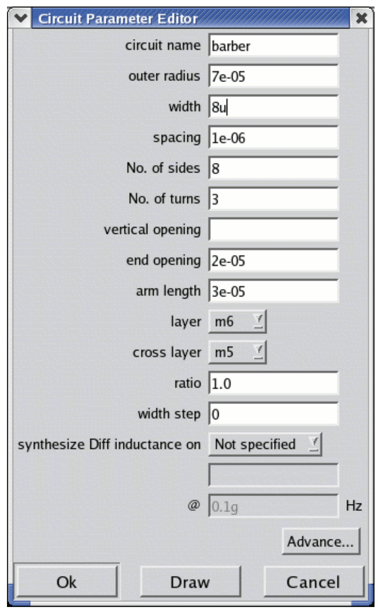
2. Click the PeakView Edit menu item, and click “**Paste**”. A new device is created and added to the “**Data Browser**” window with the name “**copy_of_Symmetric1**”. The student may need to adjust the width of the display columns to better see the device name and the other column parameters.



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.



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.

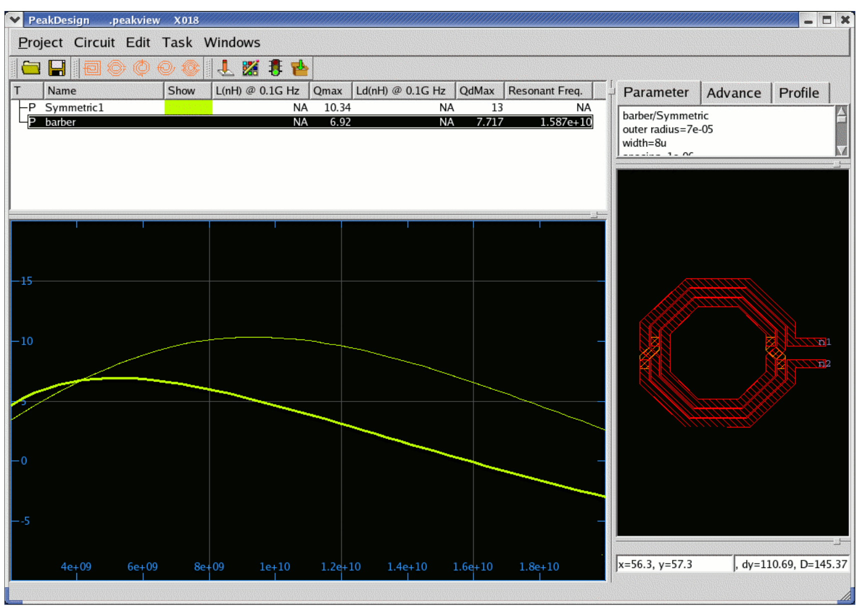
Recall…. $f_{res} \propto 1/\sqrt{LC}$***, as L & C increase,*** f_{res} ***will decrease.***

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

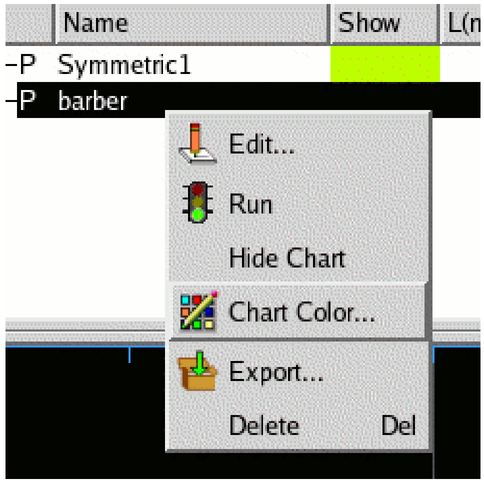
It is hard to predict R at this time, increasing width will decrease R, but increasing outer radius will increase R, therefore, the combined effect cannot be predicted with any certainty.

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

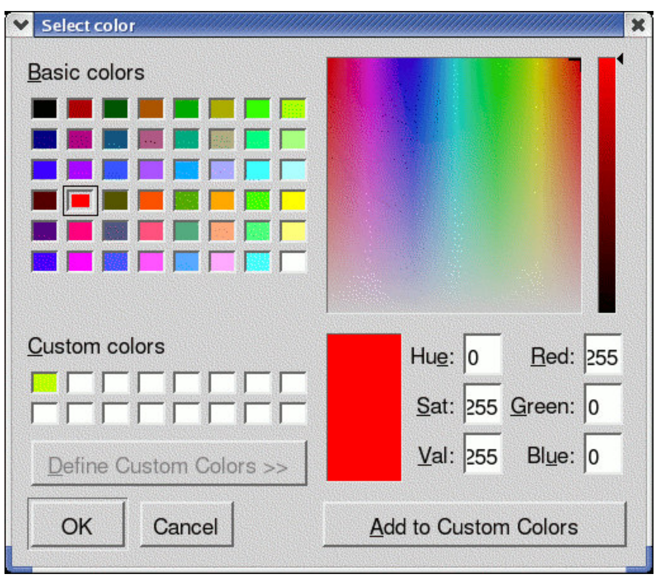
After the EM simulation completes, examine the "Waveform Viewer". Two curves will appear for each electrical plot. They have exactly the same color. But, the curve that is highlighted is the curve of barber.



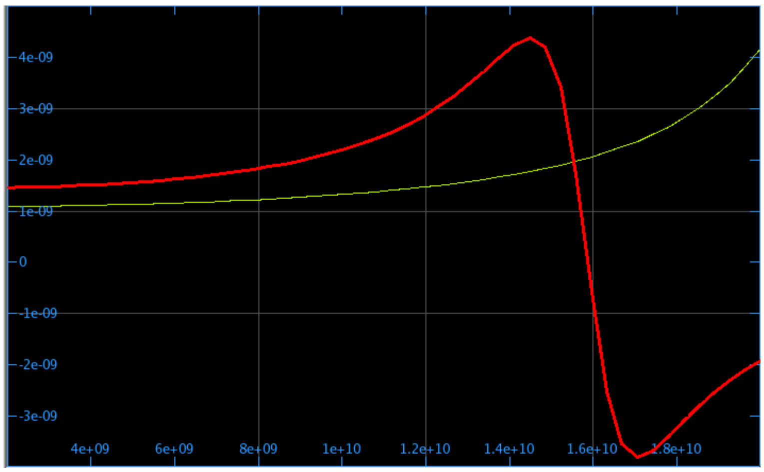
5. Move cursor to the **barber** cell in the **Data Browser** sub window, click right mouse button, the "**Edit Circuit**" dialog will pop-up. Click the "**Chart Color…**" item. The "**Select Color**" window will pop up.



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.

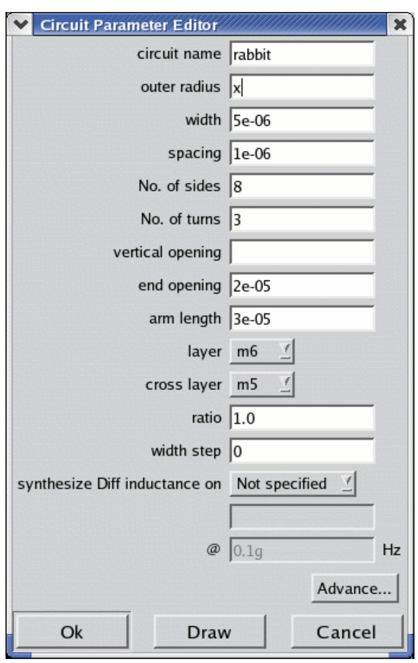


7. Move cursor to "**Waveform Viewer**" window, pressing the "f" key will fit the waveform to the window. Compare the plots for Q, L and R. Do they agree with the prediction? The L plot might look something like the picture below. What other information can you deduce from these plots?



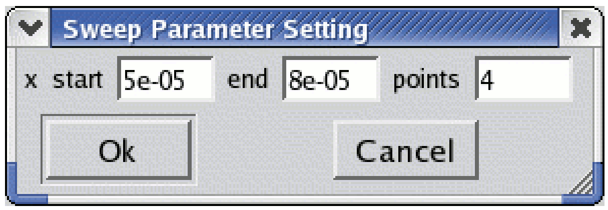
8. Based on the previous **barber** discussion, one should be able to only predict as the outer radius increases, L will increase, the resonant frequency will decrease. Now, let's confirm this idea by expanding the investigation and performing a parameter sweep on the outer radius.

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.

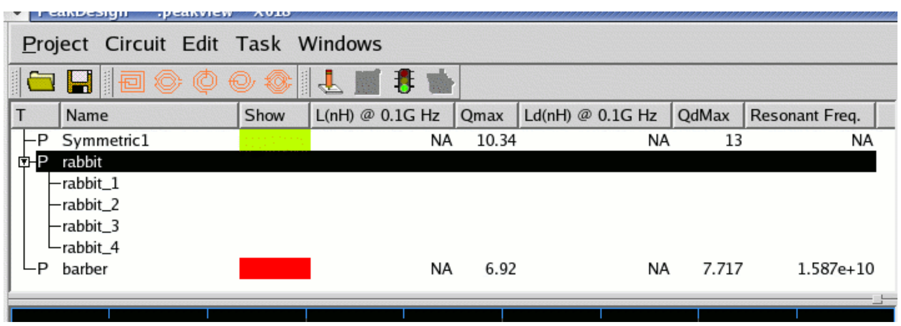


10. Clicking **OK** on the "**Circuit Parameter Editor**" dialog will pop up of the "**Sweep Parameter Setting**" dialog. Previously, the outer radius value was set as a variable **x**. Here, the sweep values for **x** must be specified. Make the following changes to the dialog values:

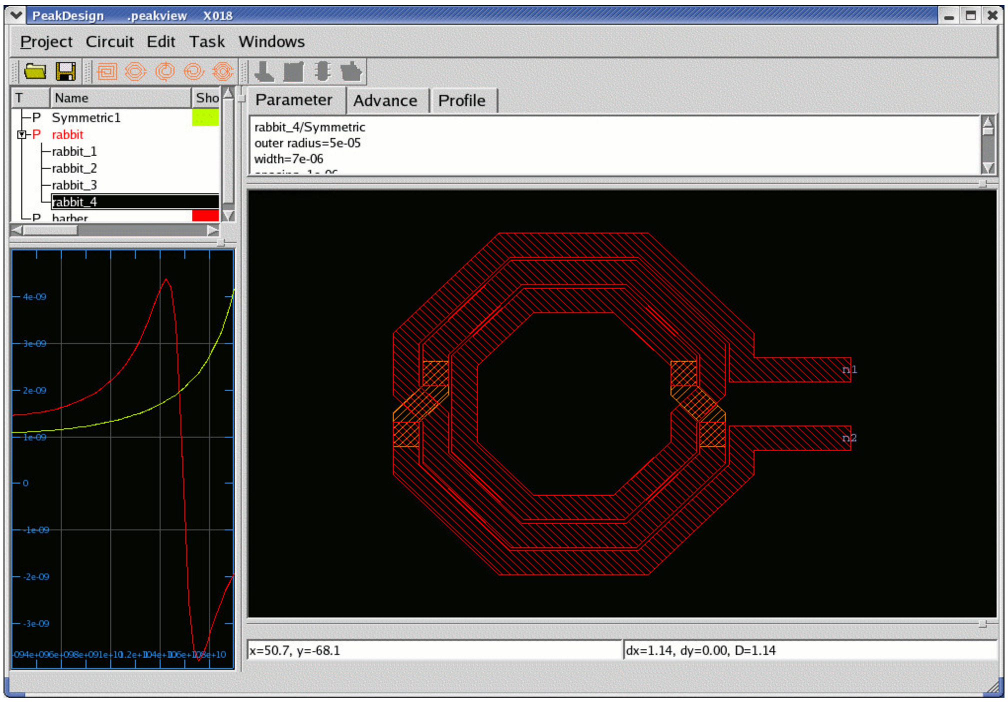
Start 5e-05
Stop 8e-05
Points 4



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.



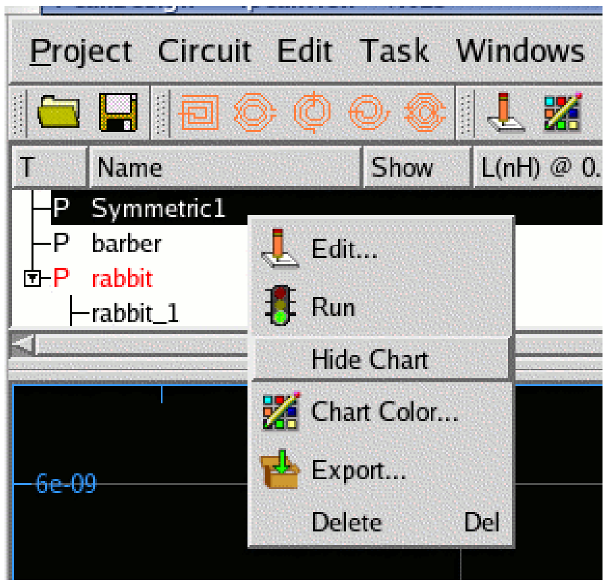
12. Adjust the PeakView "**Layout Viewer**" window to making it larger. Selecting **rabbit_1** in the "**Data Browser**" window will display its layout in the "**Layout Viewer**" and the exact parameters are recorded in the "**Parameter Window**". The student can cycle through the other children cells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



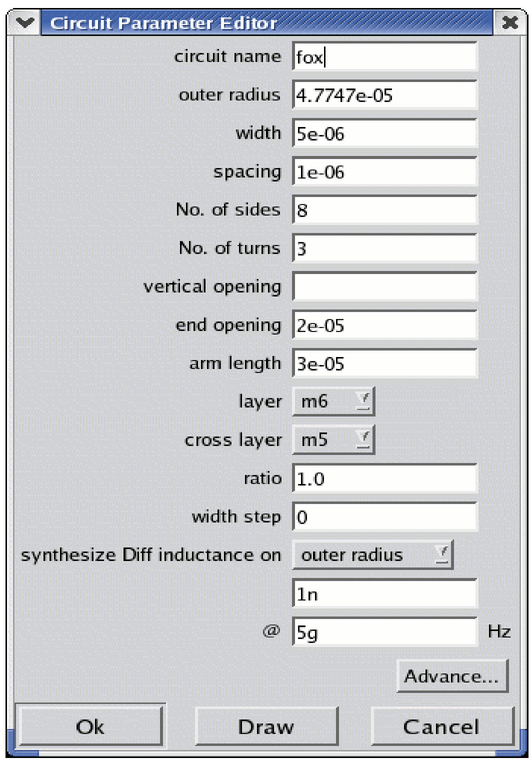
16. Now, there are four curves of rabbits remaining in the wave form window. Select rabit_1, the curve of rabbit_1 will be highlighted. Select rabbit_2, rabbit_3 and rabbit_4 to check their curves. Are these curves agreed with what we predicted? You might find more regular patterns from these four curves.

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:

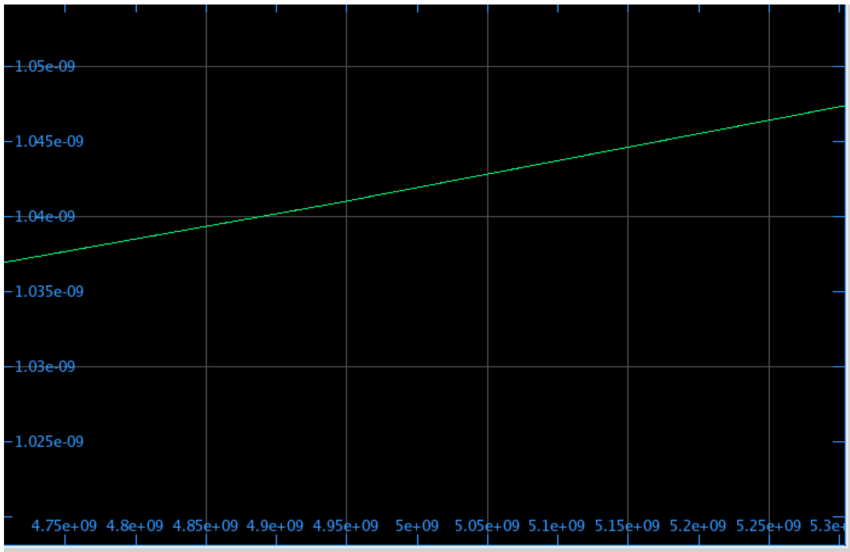
Name	fox
Synthesis Diff inductance on	outer radius
(set the inductance synthesis target)	1n
(set the target frequency) @	5g



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.

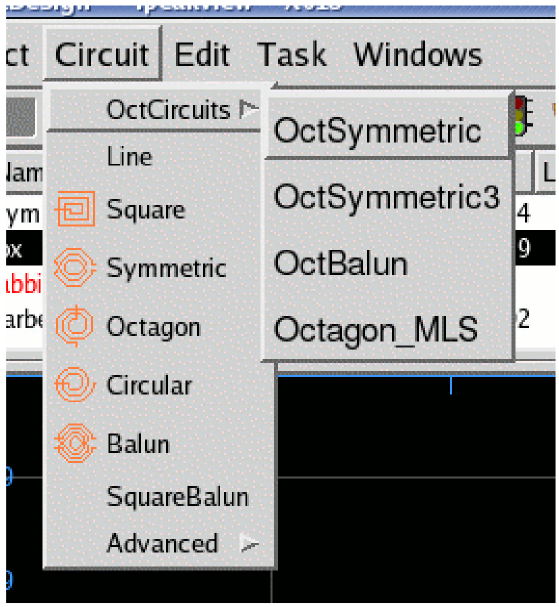
4. Now, only **fox's** waveforms will appear in the "**Waveform Viewer**". If there are other curves still remaining, hide them. Turn on **fox's L** plot. Zoom in to around 5G Hz to make sure the inductance value is 1nH as targeted.



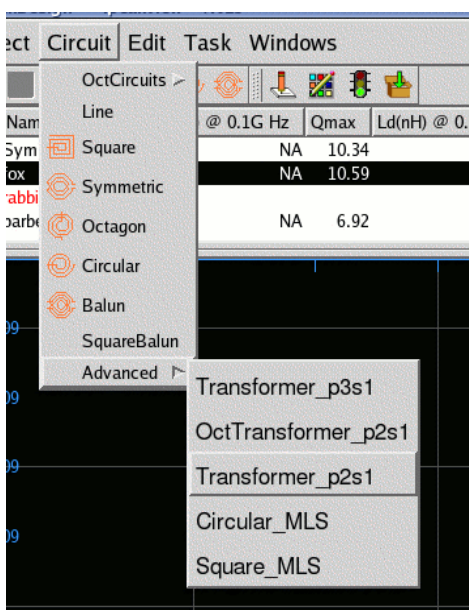
Lesson 7: Simulation Options and Formluas

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

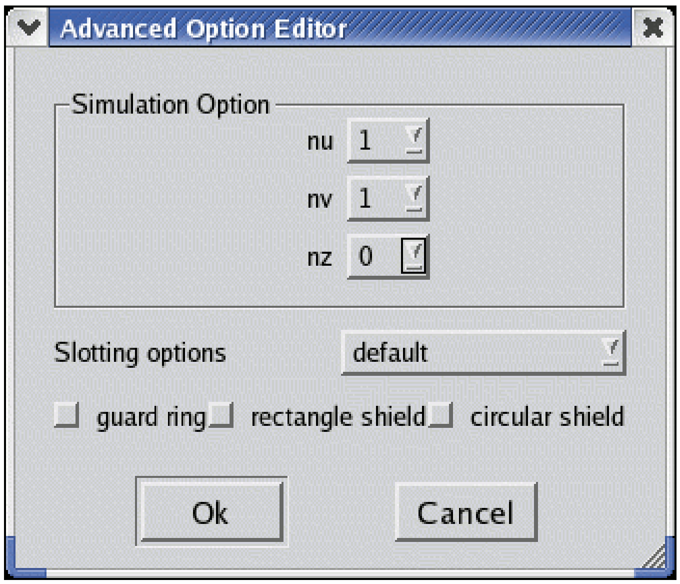
1. Click PeakView's **Circuit** menu item and select **OctCircuit** to see how various types of octagonal structures available.



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.

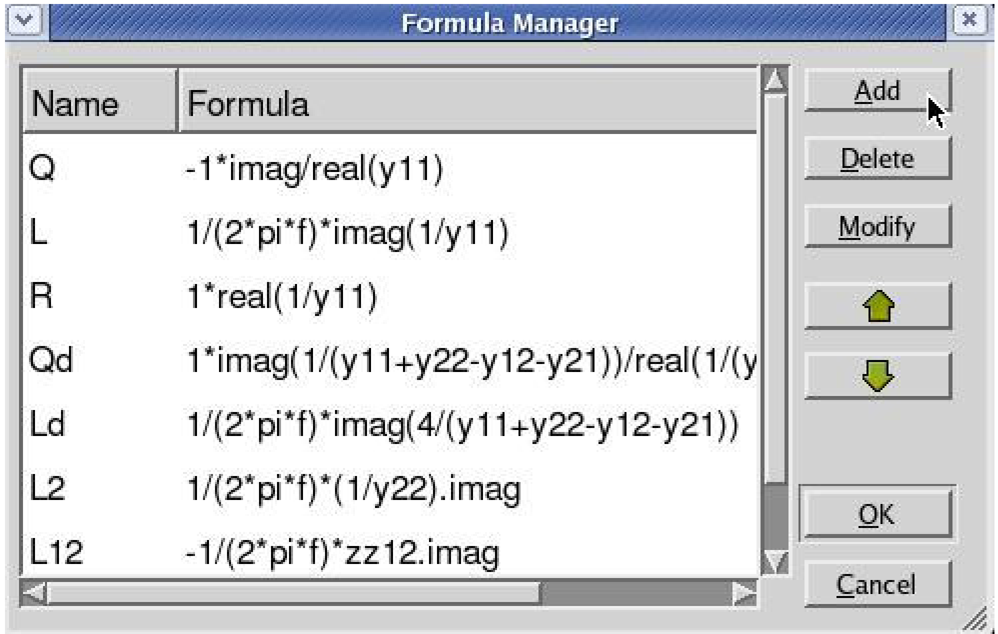


4. Click **OK** on the "**Advanced Option Editor**" form; click **OK** of "**Circuit Parameter Editor**" dialog. And then click "**Run**" button

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.

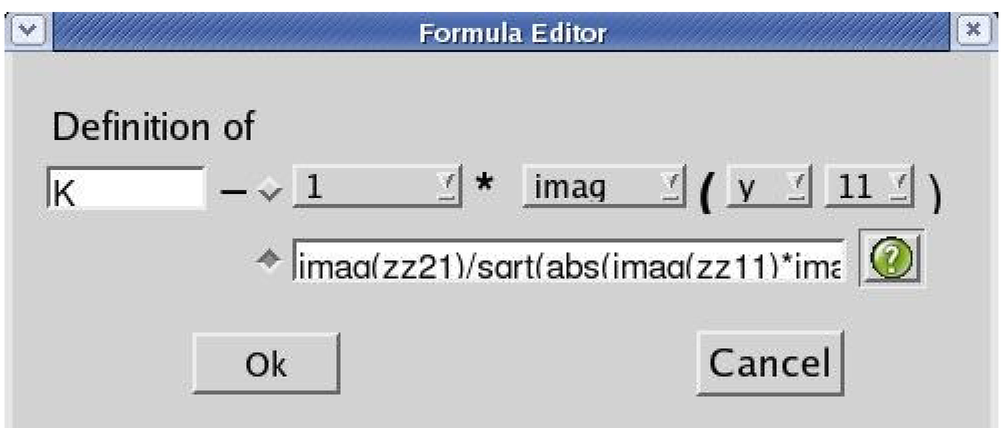
7. In the "**Formula Manager**" window, click "**Add**" to open the "**Formula Editor**" dialog as shown in the following figure.

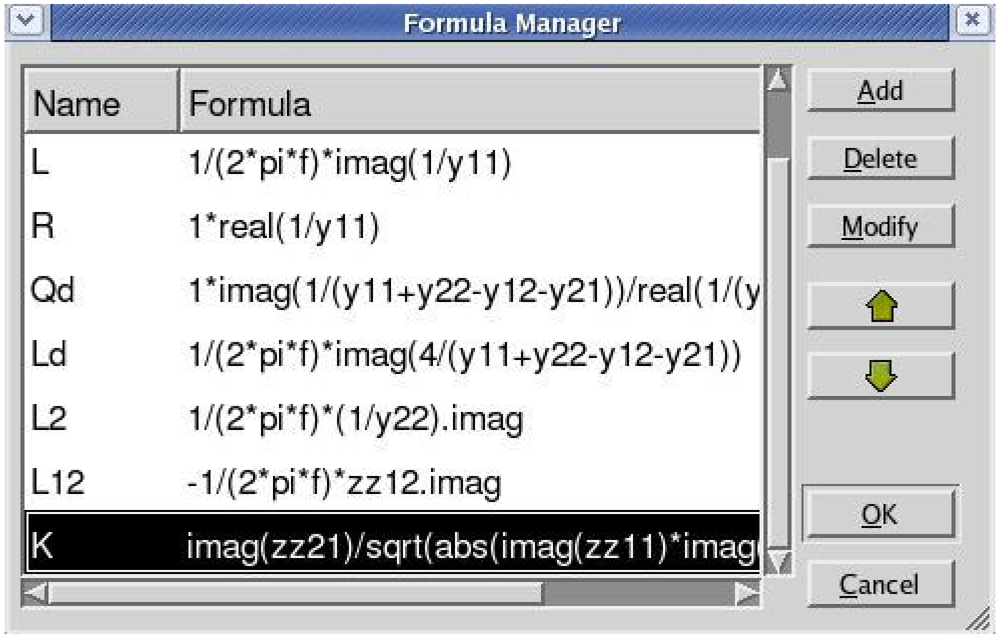


8. In the "**Formula Editor**", replace "**new**" with "**K**", check the lower radio button ,and input following equation into the open box:

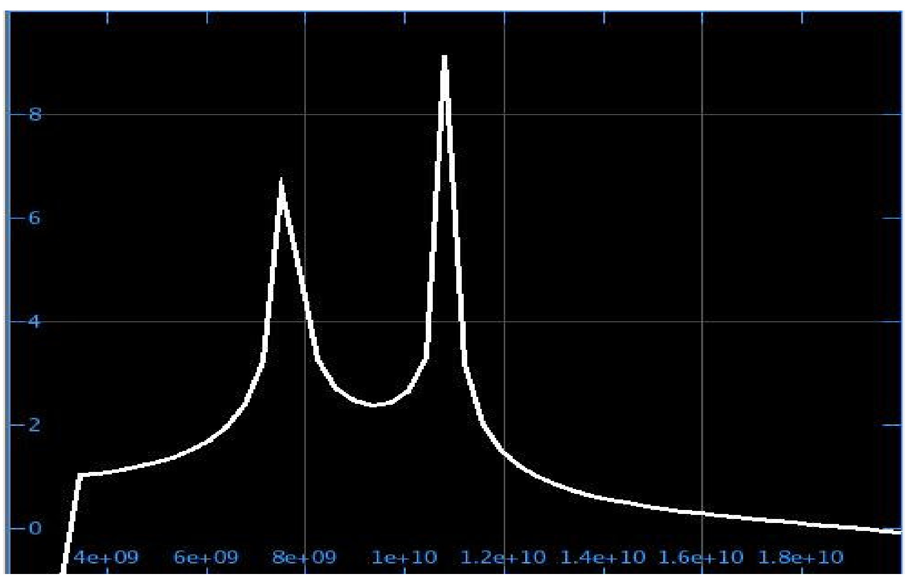
   imag(zz21)/sqrt(abs(imag(zz11)*imag(zz22)))

   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.

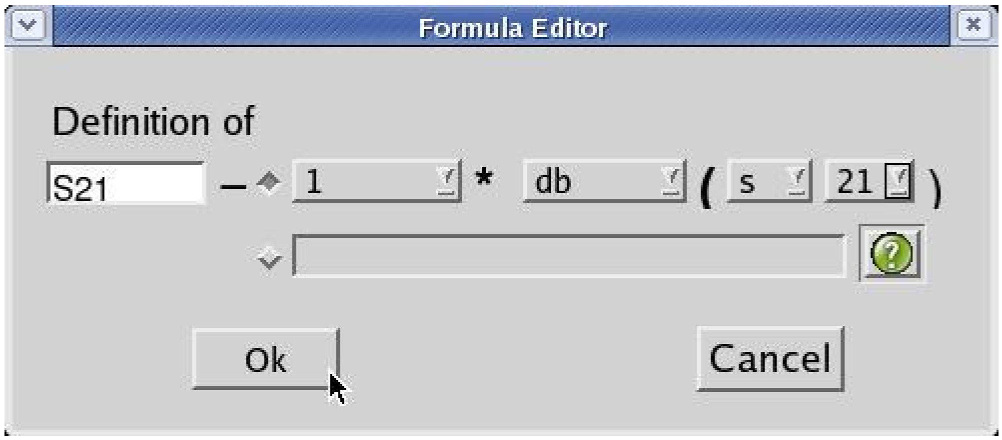




9. Place cursor on the "**Waveform Viewer**", click the right mouse button and select "**K**" in the formula list. The waveform for the coupling coefficient will be displayed.



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.

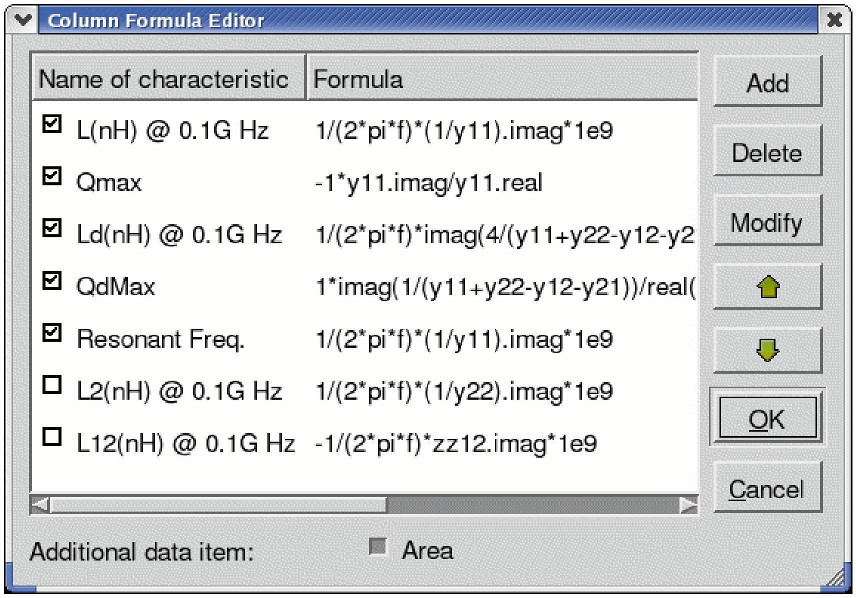


11. Click **OK** on the “**Formula Editor**” dialog; click **OK** on the **“Formula Manager**” window. Move the cursor to the “**Waveform Viewer**” window and display the **S21** waveform just defined. Turn off waveforms for other devices if necessary.

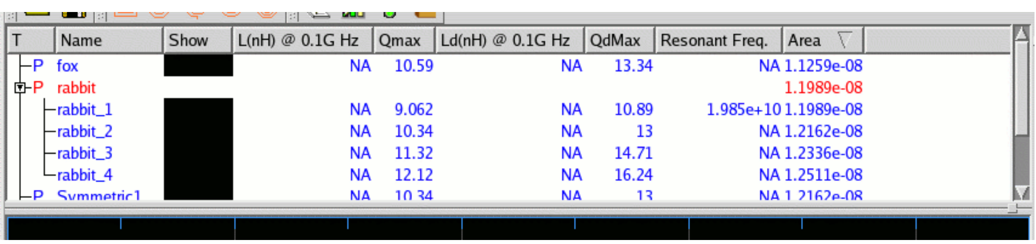
12. Examine layout of transformer in layout window

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**.



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.



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

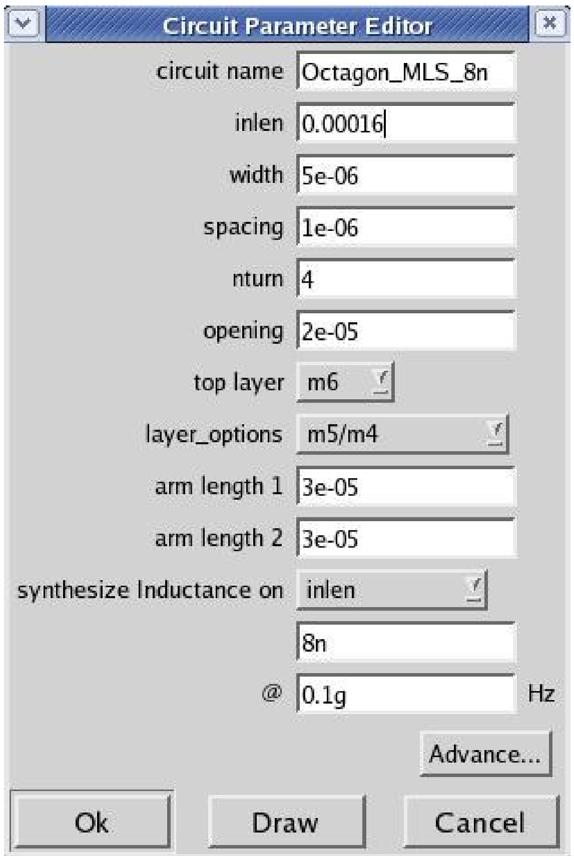
Lesson 8: Advanced Passive Design

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.

3. 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



4. Click **OK** of this form and "**Run**" button of PeakView window to synthesize the MLS structure.

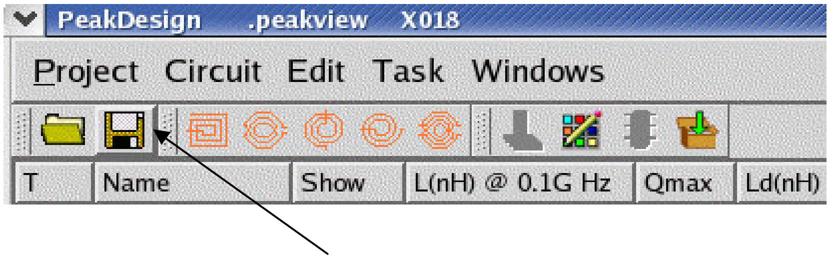
5. After completing synthesis, in the **"Data Browser"** window, click "**Area**" to sort the cells by area. Compare the area of **Octagon_MLS_8n** to **Symmetric_8n**. Although they have same inductance value at 0.1GHz, the area of **Symmetric_8n** is 5.29e-8m^2, and the area of **Octagon_MLS_8n** is only 1.78e-8m^2, a 66% area savings!

T	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.

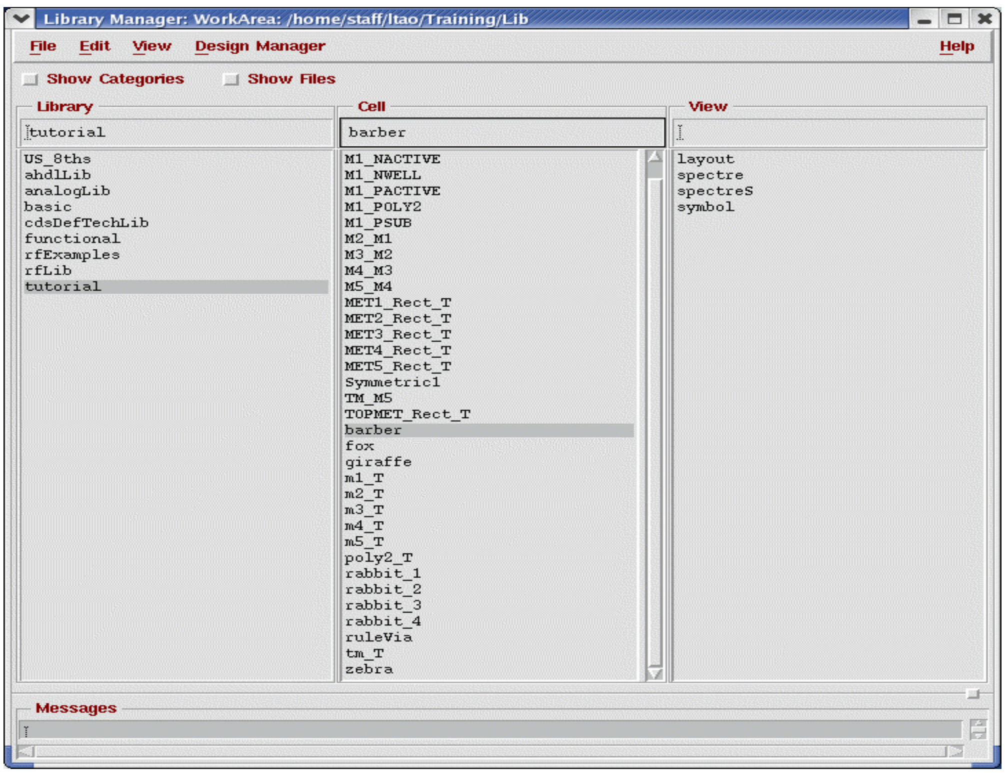
1. Click the **Save** button to save the project.



Save button

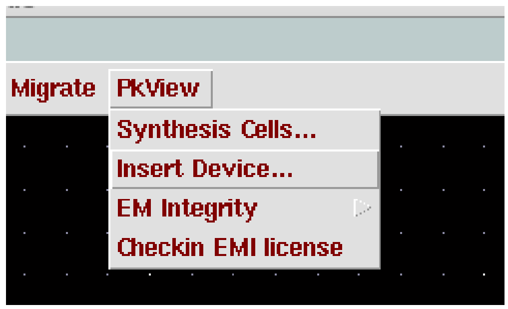
2. Click PeakView's **Project** menu and select **Return** to exit PeakView and transfer all newly created devices back to the Cadence environment.

3. 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.

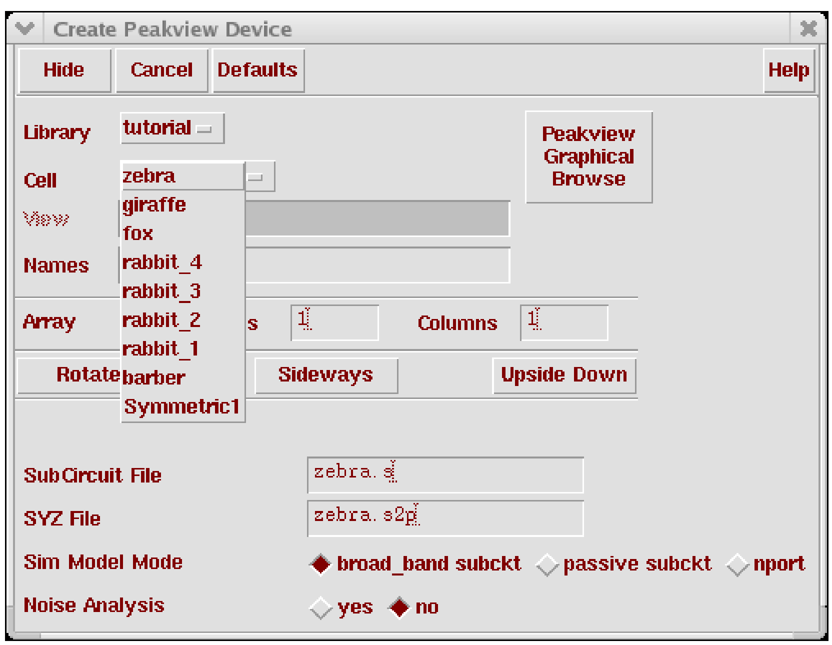


4. Open the layout and symbol view of the **barber** cell and examine. The layout is identical to the previous PeakView result. Close the layout and symbol windows.

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.

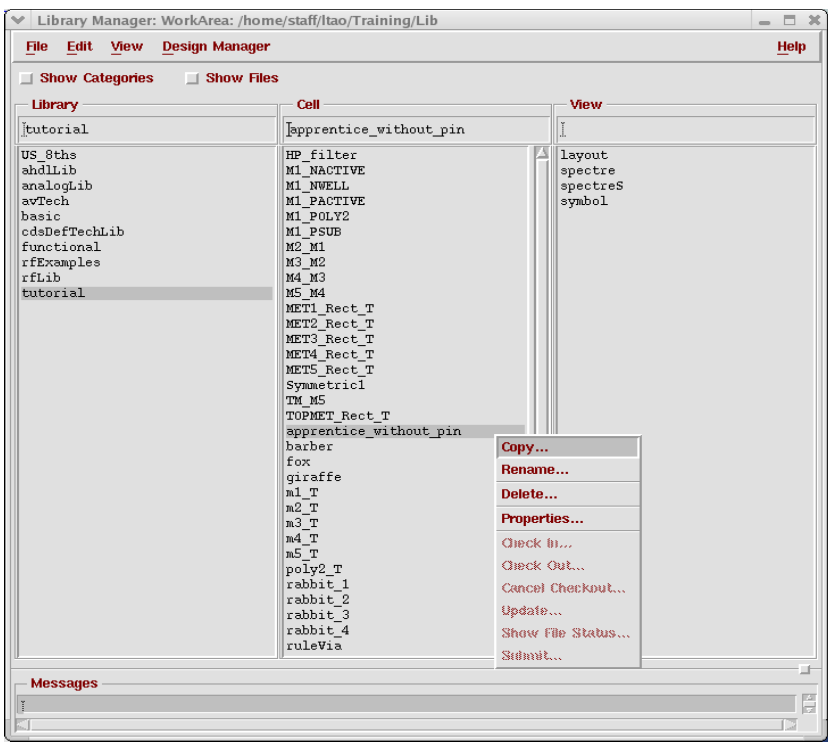


7. Cancel "**Create PeakView Device**" form.

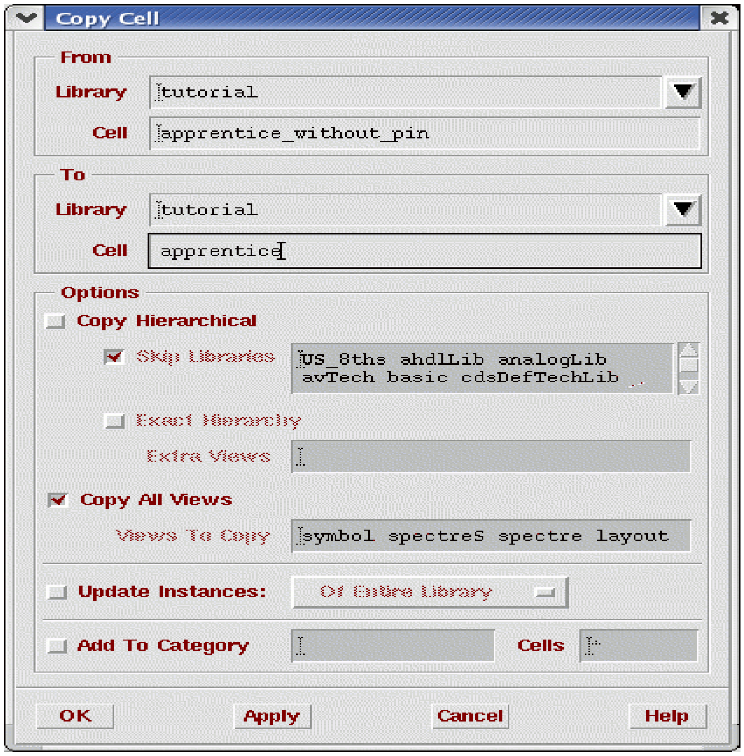
Lesson 10: LEM training

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.

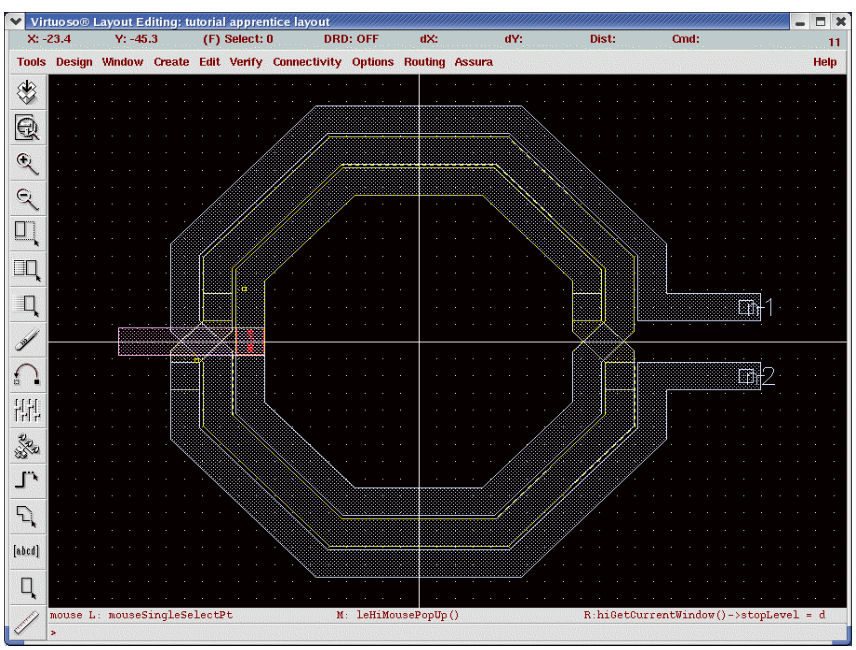
1. In Library Manager, choose **apprentice_without_pin cell**, click right mouse button, and click "**Copy** .." item.



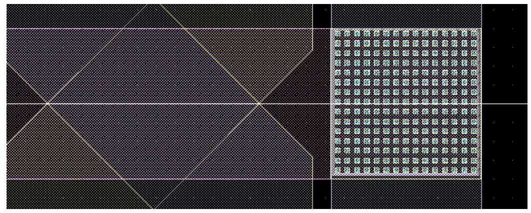
2. In the "**Copy Cell**" form, change Cell name of "To" section as **apprentice**. And click **OK** on this form.



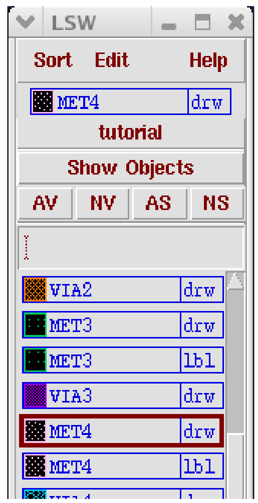
3. Open **apprentice** layout. The center tap is already added. Now, add a pin to the end of this center tap.



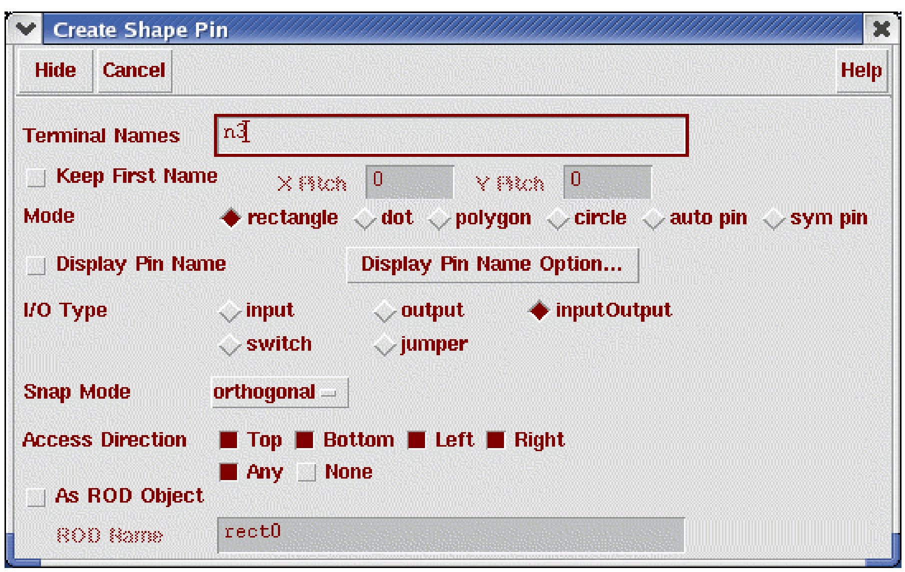
4. Zoom in to the under path to make sure MET4 is connected with TOPMET by MET5 and vias. Then press "f" key to fit the layout.



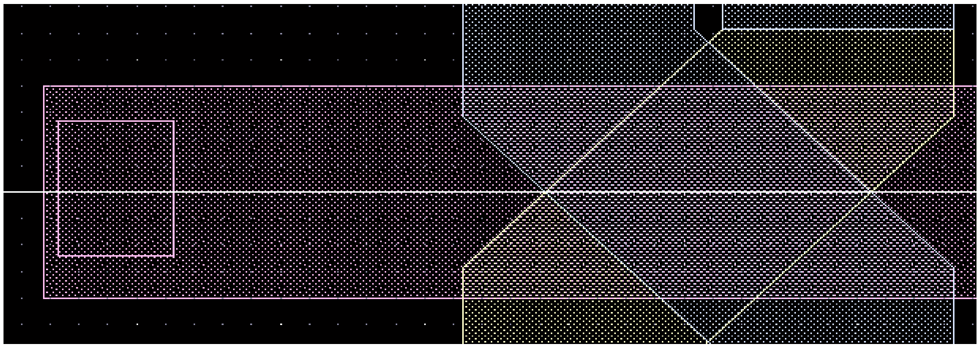
5. Choose MET4 as current layer in LSW window.



6. 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.



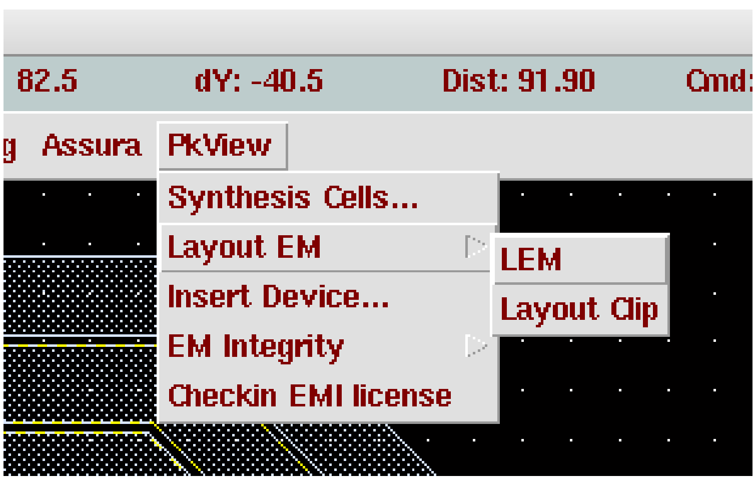
7. Create a little rectangle at the left end of the MET4 center tap as shown in the following picture.



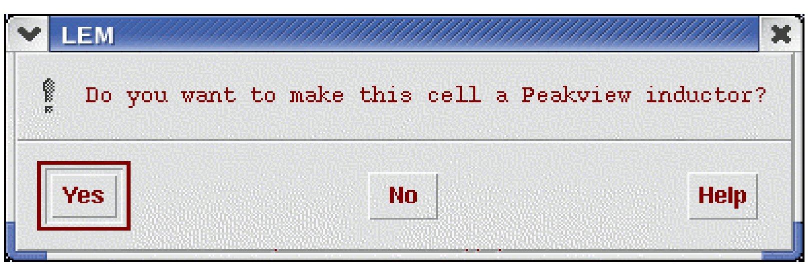
8. Cancel the **Create Pin** command by press **Esc** key.

9. Click **Tools→PkView** of **apprentice** layout window. The PkView menu will appear in the layout window.

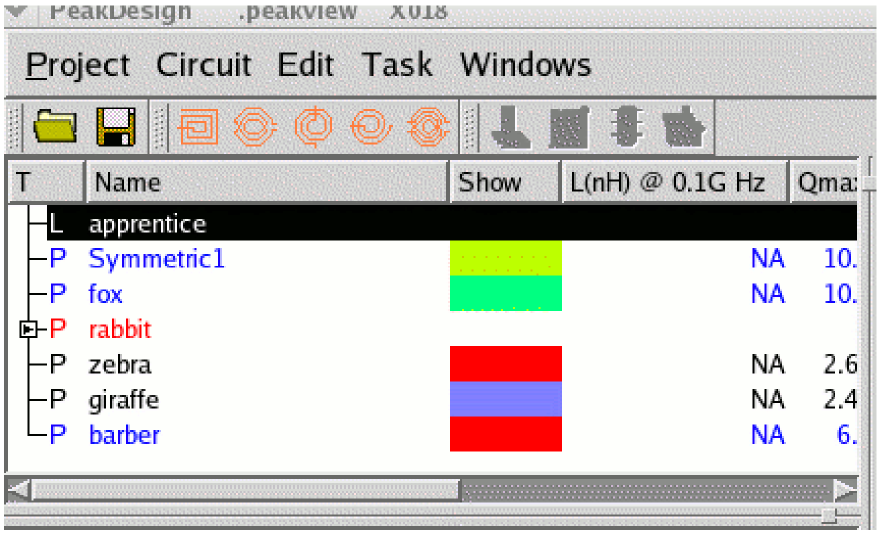
10. Click **PkView→Layout EM→LEM** to invoke LEM.



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.



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