

Don Bouldin <dbouldin@gmail.com>

# ElectricalandComputerEngr65148040FA2011: Calibre PEX

1 message

### Zachary Michael Crane <zcrane@tennessee.edu> Fri, Sep 2, 2011 at 10:31 AM

I figured I'd save everybody the horror and suffering that it took for me to figure this out.

The first time you run calibre to generate your extracted parasitic schematic, you're going to have to define pins for your transistors and also your parasitics. I didn't see this mentioned anywhere in the tutorial, so if I missed it, shame on me and you can disregard this. Otherwise, check out page 4 of this PDF, and do note that they are using a different technology.

The transistors can be found in the default library that calibre first pulls up, so you can just press the "auto map pins" button and you're done with the nmos and pmos.

This isn't the case for the parasitic devices. To find them:

In the library box, you'll want to type analogLib (don't miss the capital L) In the cell box, you'll want to type either res or cap, depending on which parasitic device you're mapping.

The view should default to symbol

Over to the right, in the box entitled "pin map," if you've found the library path correctly, all you should have to do is click, and the window will update with pins that are available to map, and then you can click auto map pins again and you should be good.

link: <u>http://www.uio.no/studier/emner/matnat/ifi/INF4420/v10/</u> PostLayoutTutorialTSMC90nm.pdf

# Cadence Tips (not complete)

#### How to run Layout XL

In the schematic view go to: Tools – Design Synthesis – Layout XL (click "ok" on the two pop-up windows)

Now you should have the Virtuoso layout window popping up.

Go to: Design – Gen from source... (click ok)

## Post layout simulation (tsmc90nm)

In order to be able to run post layout simulations, the following needs to be fulfilled.

- 1) ALL pin names have to be capital letters (schematic and layout)
- 2) GND and VDD need to be pins specified as "inputOutput" (schematic and layout)

In Virtuoso, when assigning pins to nets (wires) "Create Symbolic Pin – Display Pin Name Option..." (Shift-P), make sure that the pin layer (pn) is specified for the metal layer ("M1" in this case) your pin is supposed to be in. This is indicated with the red arrow in Figure 1. LVS requires this as well.

			1 11 1	8	
Create Symbolic Pin		- O X	Y Pin Na		y <b>_ D X</b>
Hide Cancel		Help	ок	Cancel	Help
Terminal Names			Height		0. Ž
Keep First Name X Pitch	0 Y Pitch 0		Font		stick 💷
Mode 🔶 sym pin <	🦯 auto pin 🔶 shape pin		Text O	ptions	Drafting
🔳 Display Pin Name	Display Pin Name Option				Overbar
I/O Type 🔷 input Switch	♦ output 🔷 inputOu ⊖ jumper	ıtput	Layer	•	↓ Pin Layer
Pin Type met1_T =					× ,
Pin Width	Pin Length <sup>0</sup>		Justific	ation	centerCenter
Access Direction Top Bo	ttom 🔳 Left 📕 Right ne		∑ Ro	tate	/ Sideways 🗧 Upside Dow

Figure 1: Create pin

After DRC and LVS are complete without errors, you can then do parasitic extraction.

In Virtuoso layout , go to Calibre – PEX.

In the Rules tab, specify the directory for the PEX rules file "calibre.rcx". See Figure 2.

	Calibre Interactiv	/e - PEX : pexSetup [/ifi/midgard/h01/amirh/tsmc90nmlp2]	
	<u>F</u> ile <u>T</u> ranscript	Setup	<u>H</u> elp
	Rules	Calibre-PEX Rules File	
- <b>- - - - - - - - - -</b>	 Inputs	/ifi/midgard/h01/amirh/tsmc90nmlp2/Calibre/rcx/calibre.rcx View Loa	ad
	Outputs		
	PEX Options	Calibre-PEX Run Directory	
	Run <u>C</u> ontrol	/ifi/midgard/h01/amirh/tsmc90nmlp2	
	Tr <u>a</u> nscript		
	Run <u>P</u> EX		
	Start R <u>V</u> E		

Figure 2: PEX setup - Rules

In the Inputs - Netlist tab, select "Export from schematic viewer". See Figure 3.

Calibre Interacti	ive - PEX : pexSetu	i/midgard/h01/amirh/tsmc90nmlp2]	- D ×
<u>F</u> ile <u>T</u> ranscript	<u>S</u> etup	7	<u>H</u> elp
<u>R</u> ules Inputs	Layout Ne	etlist H-Cells Blocks Probes	
<u>O</u> utputs	Files:	inverter.src.net	▼ View
P <u>E</u> X Options	Format:	SPICE -	Export from schematic viewer
Run <u>C</u> ontrol Tr <u>a</u> nscript	Top Cell:	inverter 🗸	
Run <u>P</u> EX	ĺ		
Start R <u>V</u> E	]		

Figure 3: PEX setup - Inputs

In the Outputs – Netlist tab, make sure that the format is "CALIBREVIEW" and that "SCHEMATIC" names are used. See Figure 4.

Calibre Interactiv	ve - PEX : pexSetup [/ifi/midgard/h01/amirh/tsmc90nmlp2]	O X
<u>F</u> ile <u>T</u> ranscript	Setup	<u>H</u> elp
Rules	Transistor Level R + C + CC No Inductance	
<u>Inputs</u>	Netlist Nets SVDB	
Outputs PEX Options	Netlist Nets SVDB	$\rightarrow$
Run Control	Format: CALIBREVIEW Use Names From: SCHEMATIC	
Tr <u>a</u> nscript		
	File: inverter.pex.netlist	
Run <u>P</u> EX	View netlist after PEX finishes	
Start R <u>V</u> E		

Figure 4: PEX setup - Outputs

#### Run PEX

If everything goes well, you should get the Calibre View Setup window popping up.

Calibre View Setup		. 🗆 🗙
OK Cancel		Help
Output Library:	INF4420Prosjekt]	
Cellmap File:	./calview.cellman	
	View Edit	
Calibre View Name:	calibre	
Calibre View Type:	⇔ maskLayout 🔶 schematic	
Create Terminals:	$igoplus$ if matching terminal exists on symbol $ \diamondsuit $ Create all terminals	
Reset Properties:	$m = 1_{\underline{i}}^{\mathbf{r}}$	
Magnify Devices By:	Τ <sup>μ</sup> <sub>k</sub> ,	
Magnify Parasitics By:	T.	
Device Placement:	◆ Layout Location 🧠 Arrayed	
Parasitic Placement:	🔷 Layant Lacadon 🔸 Arrayed	
Open Calibre CellView:	🔷 Read-mode 💊 Edit-mode 🔶 Don't Open	1
Always Show Dialog	• 1	

*Figure 5: Calibre View Setup* 

You should click on the Open Calibre CellView "Read-mode" to verify that there are parasitic components in your extracted netlist.

#### Click OK

**When running PEX for the first time (i.e. ./calview.cellmap file is empty):** After clicking OK in Calibre View Setup, another window will pop up, Map Calibre Device, see Figure 6. This window wants you to map device pins from already existing cells in the library.

For example, if the specified device is "nch\_mac": Click "Browse", find the "nch\_mac" *symbol* in the tsmcN90rf library, and click on "Auto Map Pins".

Click OK.

An identical window pops up and asks you to perform the same procedure again. Repeat the process for all the other devices you have used in your design.

Map Cali	bre Device		- C ×
ок с	ancel Auto Map Pins		Help
Device:	nch_mac	Library:	tsmcN90rf
Pins:	bdgs	Cell:	nch_mac
Pin Map:	b=B d=D	View:	symboli
	g=G s=S		Browse
		Terminals:	BDGS
· · · · · · · · · · · · · · · · · · ·			

Figure 6: Device Mapping

If the extraction completed without errors/warnings, you should see your new netlist appear like the one shown in Figure 7.

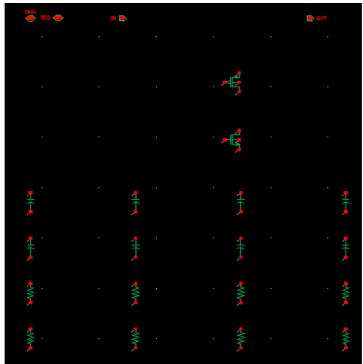


Figure 7: Extracted netlist

Figure 7 shows ports/pins, transistors, and parasitic components (R and C). Sometimes it is

necessary to open the new netlist in "Edit mode", because the extraction fails and some ports are missing. Then you can add these port manually like you do in the schematic. Just make sure the names match. However, everything you modify in this schematic must also be modified in the netlist file (<cellname>.pex.netlist).

Now we have to specify that we want to run simulation based on the extracted netlist. In the Library Manager go to: File - New - Cell View . The Cell Name must be your testbench cell. The view is "config" and the tool used is the Hierarchy-Editor. Illustrated in Figure 8.

🔆 Create New File 📃 🗖 🔀						
ОК	Canc	el	Defaults		Help	
Library N	ame		INF44	20Prosjekt	-	
Cell Name		ir	nverter_t	b	]	
View Name		config				
Tool		Hierarchy	-Editor 😑	]		
Library path file						
dgard/h01/amirh/tsmc90nmlp2/cds.libj						

Figure 8: Create config view

Click Ok.

The Hierarchy editor pops up automatically. In the Top Cell View, type "schematic". See Figure 9. Press "Use Template...", choose the template name "spectre" and click OK. Click OK in the "New Configuration" window as well.

📓 Cadence® hierarchy editor					
File Edit View Plug-Ins He	lp				
1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2					
Top Cell					
Library: Cell: Open					
Global Bindings					
Library List:					
View List:	-				
Stop List:					
Cell Bindings New Configuration	Б				
Library					
Library: INF4420Prosjekt Cell: inverter_tb View: hematic Browse					
⊂ Global Bindings					
Library List:					
View List:					
Stop List:					
Description					
	Ľ.				
Messages					
as authorized in a					
	~				
Use/reproduction set forth at FAR 1 OK Cancel Use Template Help	F				

Figure 9: Hierarchy editor

Now you should get all cells listed in the hierarchy editor. In order to be able to run simulations based on the extracted netlist. You must right-click on "inverter" cell, go to "Set Cell View" and select "calibre". See Figure 10.

🖀 Cadence® hierarchy editor: New Configuration (Save Needed)							
File Edit View Plug-Ins Help							
) D 🚅 🔒		🕅 n n 😣 🚺	E 🖩 🖷				
Top Cell							
Library: INF4	442 <u>0 P</u> i	rosiekt	ell: inverter_tb	View: schematic	Open		
Library. jiw	17201			view. jschemade	open		
-Global Bindin	igs —						
Library List:	myLik	)					
View List:	cnoct	re cmos_sch cmos.sch s	chomotic verilogo obdl				
VIEW LISU	speci	re chios_sch chios.sch si	chemanic venioga anui				
Stop List:	spect	re					
Cell Bindings	;						
Library		Call	View Found	View to Use	Inherited View List		
INF4420Prosje	≘kt ir	nverter	schematic		spectre cmos_sch cmo		
INF4420Prosje	ekt ir	nverter_tb	schematic		spectre cmos_sch cmo		
analogLib	V	dc	spectre		spectre cmos_sch cmo		
analogLib	V	pulse	spectre		spectre cmos_sch cmo		
tsmcN90rf	n	ch_mac	spectre		spectre cmos_sch cmo		
tsmcN90rf	p	ich_mac	spectre		spectre cmos_sch cmo		
Massaas							
Messages							
Bound cell (INF4420Prosjekt Inverter) to view "calibre". Bound cell (INF4420Prosjekt inverter) to view "schematic". Deleted view binding for cell (INF4420Prosjekt inverter). Bound cell (INF4420Prosjekt inverter) to view "calibre". Deleted view binding for cell (INF4420Prosjekt inverter).							
Ready 🔥 Update Needed Filters OFF NameSpace: CDBA							
-Cell Bindings							
Lihran		Cell	View/Found	View to Use	Inherited View List		
INF4420Prosje	≘kt ir	nverter	calibre	calibre	spectre cmos_sch cmo		
INF4420Prosje	ekt ir	nverter_tb	schematic		spectre cmos_sch cmo		
analogi ib	Y	ndr	spectre		spectre cmos, sch cmo		

Figure 10: Change from schematic view to calibre view

Save and exit.

In the Library Manager, open your testbench using the "config" view (not schematic view). Click OK in the window that pops up.

Setup your simulation as your normally do. In the Analog Environment, go to Simulation – Netlist – Create.

Run the simulation.