# ENGI 5131 --- Tutorial 4

## Layout vs. Schematic and Extracted View Simulation

Lakehead University

Wai Wong Winter 2013

# cādence™

## Objective

To simulate a circuit with extracted parasitic from a layout and perform post-layout simulation.

Open the inverter layout that you have created in the last tutorial. If the pin names (vdd!, gnd!, in, out) are not visible in the layout view, click **Option**  $\rightarrow$  **Display Options** and select the **Pin Names** button in the **display controls** box. Save it.

## Extracting circuit from a layout

We are going to extract the parasitic capacitor from our layout for post layout simulation. In the Layout Editor menu, click Verify → Extract. The extractor window will pop up. Click the set switches button and select parasitic\_caps.

Display O	ptions 🔹 🔺 🗙
OK Cancel Defaults Apply	Help
Display Controls	Grid Controls
Open to Stop Level Nets	Type 🔷 none 🔶 dots 🔷 lines
Access Edges	Minor Spacing
EIP Surround Array Icons	Major Spacing
📕 Pin Names 📄 🗌 Label Origins	X Snap Spacing 0.01
Dot Pins Use True BBo	Y Snap Spacing 0.01
📕 Dynamic Hilight 🔄 Stretch Hand	les
Shape Information	Filter
Path Display Borders and Centerline	s – Size 🦉 Style empty –
Show Name Of instance master Array Display Display Levels	r Snap Modes
Full Start Q	Create orthogonal -
Source Stop	Edit orthogonal =
♦ Cellview ↓ Library ↓ Tech Library	File .cdsenv Browse
Save To Load F	From Delete From



### **Layout Versus Schematic**

In the Layout Editor menu, click **Verify**  $\rightarrow$  **LVS**  $\rightarrow$  **OK.** Click the **Run** button and wait for LVS to finish.

2	Artist LVS Form Contents Different	~ + X
ок	Cancel	Help
The sele	cted LVS Run directory does not match the Run Form.	
Use	$\diamond$ Form Contents $\blacklozenge$ Run Directory Contents	
Differen	Ces: Extracted Library Name: ct18 Extracted Cell Name: invt2 Extracted View Name: extracted Schematic Library Name: ct18 Schematic Cell Name: invt2 Schematic View Name: schematic	

	Artist LVS		^ <u>-</u> +						
Commands			Help						
Run Directory	LVS		Browse						
Create Netlist	schematic	🔳 extracted	1						
Library	ct18	ct18							
Cell	invtŽ	invtž							
View	schematid	extracted							
	Browse Sel by Cursor	Browse	Sel by Curso						
Rules File	divaLVS. rulį Browse								
Rules Library	cmosp18								
LVS Options	Rewiring	📕 Device Fix	king						
LVS Options	Rewiring Create Cross Reference	Device Fix Terminals	king						
LVS Options Correspondence	Rewiring     Create Cross Reference     File     Iwong1/tsmc180,	Device Fis Terminals /lvs_corr_fi	king le Create						
LVS Options Correspondence Switch Names	Rewiring     Create Cross Reference     File     Ilworg1/tsmc180,     I	Device Fib Terminals /lvs_corr_fi	king le Create						
LVS Options Correspondence Switch Names Priority	Rewiring Create Cross Reference File Ilvong1/tsmc180,  Run local I	Device Fit     Terminals     /lvs_corr_fi	king le Create						
LVS Options Correspondence Switch Names Priority 0 Run	Rewiring     Create Cross Reference     File     Iwong1/tsmc180,     I     Run     local     I     Output     Error Display	Device Fix     Terminals /lvs_corr_fi      Monitor	king le Create						

_	/ho	me/wlwong]	/tsmcl	80/LVS/si.out			~	-	+
-ile								Help	1
	1	pfet							
Ter	minal correspor	dence points							
N3	N2	in							
N2	N3	out							
avices	in the rules b	ut not in th	e netli:	st:					
	pfet rf nfet r	f 3Vnfet rf	3Vpfet :	of numoscap 34	/nmoscap r	fet na	3Vn:	fet	
	3Vnfet_na 3Vpf	et capacitor	mimcap	resistor	· ·	-			
he net	-lists failed t	to match.							
		layou	t scher	Matic					
		ín	stances						
	un-matched	0	0						
	rewired	0	0						
	size errors	0	0						
	pruned	0	0						
	active	2	2						
	total	2	2						
			nets						
	un-matched	n	n						
	merged	ň	ň						
	nruned	ň	ň						
	active	4	4						
	total	4	4						
			rminale	_					
		00	2	>					
(	un-matched	2							
<	un-matched	2							
<	un-matched matched but different type	2	شمسر 0						

Click the **Output** button to display the report. Scroll down and look for the line "The netlists failed to match" to verify there is no mismatch between the layout and schematic diagram. But if you follow the tutorial closely, you should find two errors <u>in the report</u>.

To locate errors in the schematic diagram, open the schematic of the inverter; return to the Artist LVS window and click "**Error Display**". The mismatched items will be highlighted in the schematic editor.

	Arti	st LVS Error Dis	play	~ +	×
ок	Cancel Explai	Clear Display	Probe Form		Help
Display	First Ne	kt Prev La	ışt All	ALL of 2	
Error Co	lor =	Cycl	e Colors 🗌	Auto-Zoom	
Ali 📕 None 🗌	Unmatched Pruned Merged	nets 🔳 instanc nets 🔳 instanc nets	es 🔳 paramet es Net Display	ers 🔳 termin Limit 🔄 🛛 101	nals )





#### **Creating symbols**

On the menu bar of the schematic editor, click **Design**  $\rightarrow$  **Create Cellview**  $\rightarrow$  **From Cellview**  $\rightarrow$  **OK.** In the **Symbol Generation Options** window, rearrange "**Pin Specifications**" as shown in the figures below. Click **OK** to continue. Save it and close the Symbol Editor window.

	Symi	bol Generation Opti	ons	~ + X													
OK Cancel	Apply			Help		١	/irtuoso€	® Symbo	l Editin	g: ct18 i	nvt2 sym	bol				^	_ + X
Library Name		Cell Name	View Name		t Add	Check	Options										13 Help
ct18		invtž	symbolį														
Pin Specificati	ons			Attributes													
Left Pins	[in			List													
Right Pins	outľ			List					-								
Top Pins	VDD			List													
Bottom Pins	GND			List								 @	inct	ana	- Nar	~~1	
Exclude Inherit	ed Connection Pi	ins:										L	insc	unc	ENUI	1161	
🔶 None 📣	All 🔷 Only the	se:															
		r															
Load/Save	Edit Att	ributes 📃 🛛 Edit La	bels 🔄 Edit Prop	erties 📃			[@	par	tNai	me]			- T				
			• <b>—</b> •														
			[@abc]														
			C nouse L	: mouseSing	gleSele	ctPt		M: s	schHiMou	sePopUp	0		R: ivh	HiLVS()			
			>														

	Virtuos	so® Sch	emat	ic Ed	iting:	ct18 tu	t02 sche	ematic -	- Virtuoso®	Analog	Desigr	Environ	ment	(1) (on	cmc.la	kehead	J.ca)	^ _	+ X
Cm	d:	Sel	: 0	S	Status:	Ready										T=27 C	Simulate	or: spect	re 3
Tools	Design	Window	Edit	Add	Check	Sheet	Options	Migrate											Help
$\mathbf{\nabla}$																			
8																			
~~																			
€²																			
C <sup>2</sup>																			
Ľ		. 🖕 .																	
	· ·								. •										
14									· <mark>-</mark> · · ·						· · · T				
$\aleph$		. 📘 .															18		
	· ·		-16							· · · · · · · · · · · · · · · · · · ·							· •		
3		· (‡) 🖓	u— 1.0								 ou	t1 <sup>°°°</sup>							
	· ·	.   .							invt2	out			— in		invt	2 1 1	out 🚽	- <b>E</b>	
£1																			
		· _ ·			v1:	ø.ø												· •	
術時				υιι	÷) v2:	=1.6												- +	- c=1f
					tr=	=20p													
*															T			. T	
٦,																			
abc									gnd										
<b>-</b> *																			
•D	· ·																		
	· · ·																		
	· ·																		~
C	mouse L	: schSin	gleSe	lect	Pt()			M: s	schHiMousePo	յթմբ ()			R	:hiZoo	mRelati	.veScale(	getCurre	ntWindo	ow() 1

Next, create a new schematic in the same library that you are using to test the new symbol. The new created symbol can be instantiated as usual like other components, except that it located in is the Cell you are working with.

Library Browser - A	dd Instance (on cmc.lakeh	eadu.ca) 🛛 🔺 🗕 🔶 🗶
Show Categories		
Library	Cell	View
[ct18	invt2	İsymbol
CMCLayoutReference CMCpcells CMCshare ahallib analogLib basic c18 cdSDefTechLib cmosp18_defin_techlib et18 package passiveLib rffxamples sample sbaLib tp2973g vst_nl8_sc_tsm_c4	invt2 tut02 zpcellSCRATCHZaFev680559	symbol
Close	Filters	Help

🔲 Virtuoso® Analog De	esign Environment (1) (on cmc.lakeheadu.ca 🔺 😑	÷ X
Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ļ
Library ct18	# Type Arguments Enable	⊐ AC ■ TRAN ⊐ DC
<b>Cell</b> tutO2 <b>View</b> schematic	i cian o in cons yes	III X Y Z
Design Variables	Outputs	l∎″
# Name Value	<pre># Name/Signal/Expr Value Plot Save March 1 in</pre>	J.
	2 out1 yes all no 3 out2 yes all no 9 yes all no	8
		8
> Results in /home/wlwong1/	Plotting mode: Replace -	$\sim$

Load the Analog Design Environment and set up a transient analysis as shown below.

In the ADE menu bar, click **Setup**  $\rightarrow$  **Environment**. Add the word 'extracted' to the **Switch View** List line. Click **OK** and run the netlist.

			Envi	ronment Options	~	+	×
ок	Cancel	Defaults	Apply			I	Help
Switch V	view List		sj	pectre cmos_sch cmos_sch extracted schemati	c ve	ril	Loga
Stop Vie	w List		S	pectre			
Paramet	er Range	Checking F	file 🚶				
Analysis	Order		Ĭ.				
Print Co	mments			I			
userCma	l Line Optio	n	Ĭ				
Automat	ic output I	log		ſ			
Use SPI	CE Netlist	Reader(sp	op): 🗌	Y 🗌 N			
savesta	te(ss):			Y 🗆 N			
recover	(rec):			Y 🗆 N			
SKI Plug	jin for PLL	. Macro Mo	del 🗌	1			

The simulation results are below.

