EECE 285 – VLSI Design



Cadence Tutorial

EECE 285 VLSI

By: Kevin Dick Co-author: Jeff Kauppila Co-author: Dr. Arthur Witulski





Purpose of Cadence		
1) The Purpose of Cadence		pg. 4
Linux		
1) The Purpose of Linux		
2) Basic Linux commands for r	unning cadence	
3) Logging onto cadence		
4) Lock files		
Navigating in Cadence		
1) Icfb log		pg. 3
2) Library Manager		
3) Schematic, symbol and layou	ut editing	pg. 3
Building a schematic (using in	nverter example)	
1) Creating a new library	1	
2) Creating a new cell view		
3) Making the schematic		
4) Printing the schematic		
Building a symbol		
1) Building the symbol for the	schematic	
Testing the circuit		
1) Setting up the test circuit		
 2) Simulating the test circuit 		
Designing the layout		
1) Opening layout file		
2) Setting up the parameters		
3) Setting up the connections		
4) Running DRC (Design Rule	Checker)	
5) Extracting the circuit	<i>*</i>	
6) Running LVS (Layout versu	s Schematic)	

Hooking up the padframe 1) Fdsa

- 2) Fdasfds
- 3) Fdsafd



Submitting the project to MOSIS

- 1) Extracting the CIF file
- 2) Running checksum and count
- 3) Setting up an account
- 4) Filling out the paper work

Design Rules

1) MOSIS design rules for 0.5µm technology

pg. 6-9



Purpose of Cadence

 Cadence is an Electronic Design Automation (EDA) environment in which different applications and tools can are integrated together. This allows all the stages of IC design and verification to be done in a single environment. The different tools are supported by different fabrication technologies allowing for customization of the Cadence environment to fit the particular technology.

Linux

- Linux is an operating system that is a multi-user, multi-tasking system that can be used on servers, desktops and laptops. Linux was originally developed at Bell Labs in 1969. Linux is used to access cadence and manage the files in its libraries.
- 2) Basic Linux commands used for running cadence
 - A) Command: ls

Function: lists the files in the current directory Example: ls

B) Command: pwd

Function: tells you what directory you are currently in Example: pwd

C) Command: mkdir Function: makes a new directory

Example: mkdir project (makes a new directory called project)

- D) Command: cd
 Function: takes into the specified directory
 Example: cd NCSU AMI06 (takes you into the directory called NCSU AMI06)
- E) Command: cd ..Function: takes you back one directory Example: cd ..
- F) Command: mv

Function: moves a directory to the specified location

Example: mv adder adder2 (moves the directory adder to the current directory and changes the name to adder2)

- G) Command: cp
 - Function: copies a file

Example: cp ../multi . (copies the file named multi from the previous directory to the current directory)

H) Command: gzip

Function: compresses files, so they take up less space Example: gzip inv (compresses the file called inv)

I) Command: gunzip
 Function: uncompresses files that have been compressed by the command gzip



Example: gunzip inv.gz (uncompresses the file named inv.gz)

- J) Command: chmod
 Function: changes the read, write and execute permissions on the files
 Example: chmod nand (lets you change the read, write and execute permission for the file named nand)
- K) Command: rm

Function: removes a file

Example: rm nor.zip (removes the file named nor.zip)

L) Command: diff

Function: compares two files and shows their differences

Example: diff or and (compares the files called or and and showing their differences)

M) Command: clear Function: clears the screen

Example: clear

N) Command: tar -cvf

Function: combines many files or directories into one file Example: tar –cvf homework3.tar layout schematic symbol (combines the directories names layout, schematics and symbol into one and names it homework3.tar)

O) Command: rmdir

Function: removes a directory that is empty

Example: rmdir pad (removes the directory called pad)

P) Command: zip

Function: zips up a fileExample: zip add lock (zips up the file named add and saves it as lock.zip)

Q) Command: --help

Function: Tells you the function of a command Example: chmod –help (tells you the function of the command chmod)

R) Command: zip –r

Function: zips up a directory

Example: zip –r Bickham_HW4 Bickham_HW4 (zips up the directory named Bickham_HW4 and saves it as Bickham_HW4.zip)

- S) Command: unzip
 Function: unzips a file that has been zipped up by the zip command (ends in .zip)
 Example: unzip Homework4.zip (unzips the file named Homework4.zip)
- T) Command: tar –xvf

Function: separates files or directories that have been combined by the tar –cvf command

Example: tar –xvf Homework1.tar (separates the directories and files that have are combined in the Homework1.tar file)

U) Command: rm –r Function: removes a directory and its contents



Example: rm -r nor (removes the directory called nor and all of its contents

V) Command: cp-r

Function: copies a directory and all of its contents

Example: cp –r HW5 (copies the directory called HW5 and all of its contents)

W) Command: history

Function: shows the list of the last 100 commands Example: history

X) Command: find

Function: searches the current directory and all subdirectories for the specified file(s) Example: find project3 (searches the current directory and all subdirectories for files called project3)

- Y) Efads
- Z) Fewfds

AA)

3) Logging onto cadence

3.1) To log into cadence log onto polarbear and then a campus machine by using the ssh – Y command.

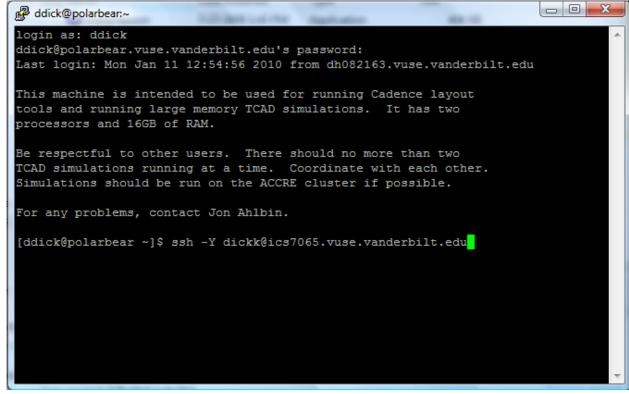


Figure 1: logging onto a campus computer

Here is a list of the campus machines:

ics7055.vuse.vanderbilt.edu ics7056.vuse.vanderbilt.edu



ics7057.vuse.vanderbilt.edu ics7058.vuse.vanderbilt.edu ics7059.vuse.vanderbilt.edu ics7060.vuse.vanderbilt.edu ics7061.vuse.vanderbilt.edu ics7062.vuse.vanderbilt.edu ics7063.vuse.vanderbilt.edu ics7064.vuse.vanderbilt.edu ics7066.vuse.vanderbilt.edu ics7067.vuse.vanderbilt.edu ics7069.vuse.vanderbilt.edu ics7070.vuse.vanderbilt.edu ics7070.vuse.vanderbilt.edu ics7071.vuse.vanderbilt.edu

Next, you need to go into the NCSU_AMI06/ directory.

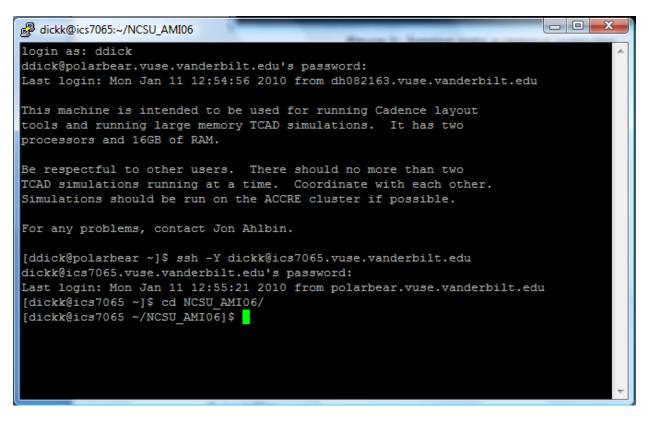


Figure 2: The NCSU_AMI06/ directory

Then, use the command **exec /bin/bash**. In bash, use the command **source soureme.sh** and then **icfb**. This will log you into cadence.



VANDERBILT UNIVERSITY

EECE 285 - VLSI Design

- O X dickk@ics7065:~/NCSU_AMI06 login as: ddick ddick@polarbear.vuse.vanderbilt.edu's password: Last login: Mon Jan 11 12:54:56 2010 from dh082163.vuse.vanderbilt.edu This machine is intended to be used for running Cadence layout tools and running large memory TCAD simulations. It has two processors and 16GB of RAM. Be respectful to other users. There should no more than two TCAD simulations running at a time. Coordinate with each other. Simulations should be run on the ACCRE cluster if possible. For any problems, contact Jon Ahlbin. [ddick@polarbear ~]\$ ssh -Y dickk@ics7065.vuse.vanderbilt.edu dickk@ics7065.vuse.vanderbilt.edu's password: Last login: Mon Jan 11 12:55:21 2010 from polarbear.vuse.vanderbilt.edu [dickk@ics7065 ~]\$ cd NCSU AMI06/ [dickk@ics7065 ~/NCSU AMI06]\$ exec /bin/bash bash-4.0\$ source sourceme.sh bash-4.0\$ icfb&

Figure 3: Logging into cadence

4) Lock files

4.1) If your file says "edit mode only" or otherwise does not respond, there is a good chance you have a .cdslck lock on your schematic or layout file. So, before you start and before you source the sourceme.sh file, type: find \$HOME -name *.cdslck

The find command should show you where the cdsclk files are in your directory. Delete (rm) these files, then you shouldn't have any trouble till the next time it locks the files.

Navigating in Cadence

 The icfg log is the window that gives you gives you a list of the actions that have taken place in cadence, Figure 1. This most recent action is at the bottom of the list. This is where you go to see the result of DRC and LVS checks. The file tab allows you to create a new library or cell, import files, export files and open files in the library manger. The tools tab gives you a list of the different tools that are available in cadence. The options tab allows you to save the session and change the preferences.

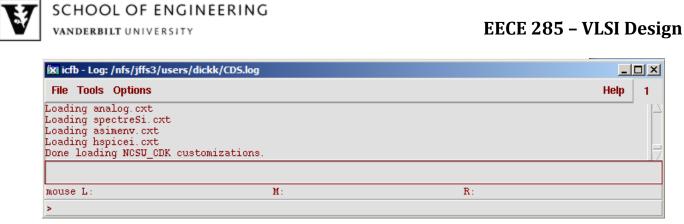


Figure 1: Icfb log

2) The library manager is where the libraries, cells and views are stored, Figure 2. If you select a cell of one of the libraries you can view the different layouts (schematic, extracted, layout, symbol, etc.) of that cell, by double clicking on the type of view. The file tab allows you to create a new library, cell view and category, open a cell view, load or save defaults or exit. The edit tab allows you to copy, rename, delete, change properties, access permission and library path.



VANDERBILT UNIVERSITY

🗙 Library Manager: Directorys/jffs3/users/dickk/NCSU_AMI06								
<u>File Edit View D</u> esign Mar	nager	<u>H</u> elp						
🔲 Show Categories 📃 Show	v Files							
Library	Cell	- View						
0SU_stdcells_ami05_h	JA0122X1	T						
MOSIS_Layout_Test NCSU_Analog_Parts NCSU_Digital_Parts NCSU_TechLib_ami06 NCSU_TechLib_tami06 NCSU_TechLib_tamc02 NCSU_TechLib_tsmc02d NCSU_TechLib_tsmc03 NCSU_TechLib_tsmc03d NCSU_TechLib_tsmc04_4M2H OSU_stdcells_ami05_h US_8ths ahdlLib analogLib basic	AA_README AND2X1 AND2X1 AND2X2 AOI21X1 AOI22X1 BUFX2 BUFX4 CLKBUF1 CLKBUF2 CLKBUF3 DFFNEGX1 DFFNEGX1 DFFSR FAX1 FILL HAX1 INVX1 INVX2	abstract extracted functional hdl layout schematic symbol						
Messages								
	fs/jffs3/users/dickk/NCSU_AMI libFile //nfs/jffs3/users/dic							
5								

Figure 2: Library Manager

3) The virtuoso schematic editing window is where the schematic is built. The buttons on the left side are common used commands. To select an object, make sure that you do not have other commands running by hitting the Esc button on the keyboard, and run the cursor over the object and click on it, Figure 1.

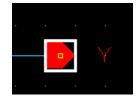
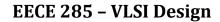


Figure 1: Selecting an object

There is a list of short hand keyboard commands that allows you to quickly call the command. For example "m" is short for move, which will allow you to move an object. These short keyboard commands can be found by going to the tabs at the top of the window and looking at the right of the command, Figure 2.





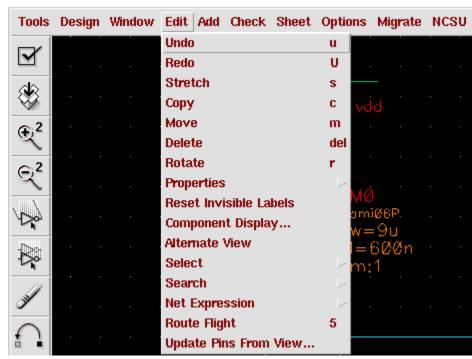


Figure 2: Finding the short cut keyboard commands

To move an object around go to the edit tab and select move, or use the short command of "m", and select the object you wish to move. To delete an object you can click on the

button to the left of the window and click on the object, select the object and hit the delete button and the keyboard, or go to the edit tab and select the delete command and click on the object you wish to delete. If you drag the pointer across the buttons on the left



The

side of the screen they will give a description for what they do

virtuoso symbol editing window is very similar to the virtuoso schematics editing window. The symbol editing window is laid out the same way but has less functions. The virtuoso layout editing window has is similar to the virtuoso schematic editing window but it also has a LSW window that appears with it. This window is to select the different layers to the parts of the transistor and connections between them. To select a layer, click on it in the LSW window and then go to the virtuoso layout editing window and draw the rectangle for that layer.



SCHOOL OF ENGINEERING VANDERBILT UNIVERSITY



Sort Edit	Help
💹 pwell	drw
NCSU_TechLib	_ami06
Show Object	:ts
📕 Inst 📕 Pin	
AV NV AS	NS
💹 pwell	drw
mvell	drw
active	drw
mactive	drw
pactive	drw
nselect	drw
pselect	drw
poly	drw
🗱 elec	drw
metal1	drw
🚾 metal2	drw
💋 metal3	drw
cc	drw
via	drw
💓 🕶 via2	drw
🚻 glass	drw
∿ highres	drw
Nodrc	drw
nolpe	drw

If you want to view only one layer, select a layer from the LSW window and click on NV, which will select only that layer.





VANDERBILT UNIVERSITY

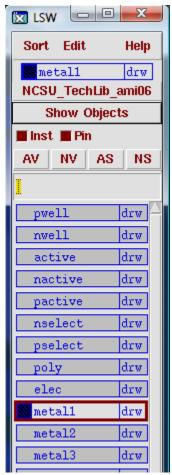


Figure 6: Selecting only one layer

Then go to the virtuoso layout editing window and click only that layer that was selected in the LSW window.



. This will let you view



EECE 285 – VLSI Design

Tools	Design	Window	Create	Edit	Verify	Connectivity	Options	Routing	Migrate	Pcell	Help
۲	· · ·									• •	
Q					*						· · ·
¢											
ତ୍											
~											
ШQ											
											· · ·
Ū,											
34							· · ·				
\cap											
											· · ·
E. C.C.							· · ·				
٦,											
S.											· · ·
[abcd]											
[verse]										<u> </u>	
							.	•	• 1		

Figure 7: Viewing a single layer

Building a schematic

Creating a new library is the first part in building a new schematic. You can do this in one of two ways, first you can create a new library from the Library Manager, or the CIW. In either case select File->New->Library. This will bring up the create library window, Figure 1. Enter the name of the new library, for example digital_lib, and select Attach Existing Library. When there is no path entered, it will place the library in the directory from which you started Cadence. If you started Cadence correctly, this will be your Working Directory.



🔀 New Libra	ary	×
- Library		_
Name	digital_lib	
Directory	y .	
 LVS spectre	. runl	
}/jffs3,	/users/dickk/NCSU_AMIO	Ğ
— Design	Manager	
O Use	NONE	
() Use (ło DM	
ОК	Apply Cancel Hel	p

Figure 1: Create New Library Window

Next, select **Attach to existing tech library** and select one of the available libraries. Remember which one was selected because it will be required for simulation later, Figure 2.

🐹 Tech	nology File for New Library	×							
ок	Cancel	Help							
Techno	Technology File for library "digital_lib5"								
If you	will be creating mask layout	or							
other	physical data in this library, y	ou							
will ne	eed a technology file. If you pl	an							
to use	e only schematic or HDL data,	a							
techn	ology file is not required.								
You c	You can: 🔷 Compile a new techfile								
	Attach to an existing techfile								

Figure 2: Attaching a techfile



2) The next step to creating a schematic is creating a new Cell View. This is done in the Library Manger, but it can be done it two ways. First, select the Library in which you will be adding this Cell. Second, either select File->New->Cell View from the Library Manger or you can type the name of the new cell in the blank under Cell in the Library Manger and hit the enter key. This will bring up a window in which you can specify the type of view you are creating, Figure 3. Since we are creating an inverter, we will name the Cell inv. It is good to give the cell a name that will help you remember what is in it.

a	Ci	Create New File								
ОК	Cancel	Defaults	Help							
Library N	tame _	digital_lib	-							
Cell Nam	e i	inv								
View Na	me 「	schenatid								
Tool	C	Composer-Schematic 🚙								
Library p	ath file									

Figure 3: Create New Cell Window

In the **Create New File window**, select as the tool **Composer-Schematic**. This will cause the View Name to be automatically filled with the name **schematic**. When you are finish click the **OK** tab. This will close the current window and bring up the window in which we will design the schematic of the Inverter.

3) Now we are ready to layout the schematic design of the Inverter. After creating the new cell, a window opens in which we will create our schematic, Figure 4. This window is the Virtuoso Schematic Editing window.



VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

<u></u>						Virtuos	io® Schei	natic E	diting:			
Cmd	1 0	Sel:	0									2
Tools	Design	Window	Edit	Add	Check	Sheet	Options	NCSU				Help
V												
٢												
€ ²												
Q2												
De 1												
2												
¥												
\cap												
Q												
ebc.												
-												
•••												
				Эť.					n el elle	E 919-18	化生活剂 化间子化 医子宫下的 化化	
G		showCli	ckInf	0()			Mi a	chHiMo	usePopUp ()		R:hiDisplayForm(NCSU_createLib	Porn)

Figure 4: Virtuoso Schematic Editing Window

The schematic level design is building the inverter from the transistor level. To add a component click **Add->Instance**. This brings up the Component Browser and a window in which you can specify the component to be added, Figure 5.



VANDERBILT UNIVERSITY

		T al			Add Ins	Add Instance					
		Hide	Cancel	Default	s		H				
		Library	Library NCSU_Analog_Par				Browse				
		Cell	pmos4								
		View	symbol				ĺ				
		Names	μ				ĺ				
		Array		Rows	1	Columns	1				
		Rota	te	S	lideways	1	Upside Down				
Filter		Fingers Width (g	rid units)		1 6 450.	On M	_				
Uncategorized			ស់លោកសា)		450	500.05 8 /	_				
CONTENTS	Ĩ	10000	grid units)		4						
Current_Sources Diodes		Length			300n	nį					
H_Spice_Only Microwave_Parts	6	Longth ((maanaan)		300n	n					
Misc_Parts N Transistors	Drain dif	fusion are	a	3.37	3.375e-13						
P_Transistors		Source o	liffusion a	rea	3.37	5e-13					
Parasitic_Devic R_L_C	83	Drain dif	Drain diffusion perimeter			2.4u M					
	Source	liffusion p	erimeter	2.4u	2.4u M						
		Ource (or mile vor	dimension of						

Figure 5: Component Browser and Add Instance Windows

The first to add is the nmost transistor. To select this transistor, select NCSU_Analog_Parts then select the category N_Transistors and select the nmos4 transistor from the list. In the Add Instance window enter the width as 4u and the length as 2u. Cadence will automatically change this to 4u M and 2u M respectively. Move the cursor into the editing window. Notice that there is an nmos transistor there instead of the normal cursor. To place the component, you need to left click. To rotate the component you need to right click. To exit from adding the current component or any other action press the Esc key. Notice that there are letters next to many of the choices in the menus, these are hot keys and can make you work progress a lot faster if they are learned. Place the nmos transistor in the bottom half of the screen on the right side of the center line.

Next, we will add a pmos transistor. This is located under the library NCSU_Analog_Parts->P_Transistors and select the pmos4. Give it the same width and length and place it on the top half of the screen above the nmos transistor.

The next step is to add the pins. Click **Add->Pin** and a window appears for adding pins to the schematic. We will add pins *vdd*, *vss*, *in*, and *out*. Ensure that the direction is set to inputOutput, Figure 6.



VANDERBILT UNIVERSITY

A			Add Pin	
Hide	Cancel	Defaults		Help
Pin Name	s	ydd vas in	out	
Direction		inputOutput 🚄	Bus Expansio	an 🔹 off 🥪 on
Usage		schematic 💷	Placement	🔹 single 🧹 multiple
Rotat	e		Sideways	Upside Down

Figure 6: Add Pin Window

The order in which the pins are added does not matter. You can also add only one pin at a time. In the current method, the pins will be placed in the order they appear in the list. First, place the vdd pin above the pmos transistor. Note the small diamond that is the actual pin so rotate the object until the diamond faces down toward the pmos transistor. Add the vss below the nmos, the in to the left of the screen, and the out to the right of the screen. In all cases make sure the small diamond faces the transistors.

Now we will add wires to connect the entire Inverter so it will work. Click **Add->Wire** to add wire or you can use the hot key 'w'. Refer to Figure 7 below to see how the connections are made. Notice that as you get closer to a device or node when placing wire, a small diamond appears. This is where you need to click to place a wire. If you make a mistake click on the error and select **Edit->Delete**.

Once you have completed adding all components and wire, Figure 7, click the "check mark" icon on the left of the window. This will check your work for errors and save your work to the library. The same can be done with **Design->Check and Save**. Any errors will be reported in the CIW.

V

SCHOOL OF ENGINEERING

VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

<u>v</u>					Virtu	oso® Sche	ematic	Editing: digi	tal_lil	b inv s	schema	tic					
Cmd	test in the second s	Sel:	0														3
Tools	Design	Window	Edit A	dd Check	Sheet	Options	NCSU										Help
V										a a	5.8	2.2					1
۲								· · · ·									
€ ²								 .									
R																	
×								• •	-								5 505
200								vdd									86 80 M
2								in Vddame	250								5 5 C
1000								out I=3	50.0n 30n								
1								met									
					in	-					- NOU						
ani (1.0									30 60 M
0005								in NES tarrie	TEAL								
ų								W=4	50.Øn								
-									32 n								
۱.۱									_								
r'L																	
_																	
80C																	17 25 4
								e e 💛 e									10.00
•D																	
المتنقا	21																
G	mouse L	showCli	ckInfo());;;		M: a	chHiMo	usePopUp()			R	hiDis	playFor	rm (NC	SU_cr	eateLi	bForm)
	×/																

Figure 7: Finished Schematic of the Inverter

4) When the schematic is finished you can print it out. To do this click **Design->Plot->Submit**. The window that appears allows you to select the options for printing, Figure 8. You can print directly from here, or if desired, the schematic can be saved as a file, such as a PostScript. To save to a file click **Plot Options** on the bottom of the window. In the new window, select **Send Plot Only to File** and enter the desired file name, such as plot.ps.



VANDERBILT UNIVERSITY

A	Submit Plot	
OK Cancel D	efaults Apply	Help
Plotlibrary	cellview viewing area	
Library Name	digital_lib	Browse
Cell Name	ing	
View Name	schematid	
Area to Plot	((-0.9375 -1.8125) (1.3125 1.5	(Full Size)
	Full Size Select	
Plot Scope Hierarchy View Name I Skip Uhrarie Skip Cells Be	s basic sample	down 1
Rotes Template File U Plotter Name def Paper Size A	<pre>1.2/local/cdssetup/schPlotTempl ault Total Pages 1</pre>	ata Load Save
Plot To File Not	Selected Plot Options	
OK Cancel I	Defaults Apply	Help
Plotter Name d Paper Size A	sb lefault portrait ~ landscape + automatic	Image Position
Scale 3.1698	Center Plot 🗍 Fit to Page	
	321 × 10.5000 inches	
	000 × 0.0000	
Total Plot Size 7.1	Server ASTA and and a server of	Total Pages 1
Number Of Copies J Queue Plot Data Send Plot Only T Mail Log To	Local Tmp Directory /tag At 10 - : 00 - PM -	Monday _

Figure 8: Submit Plot and Plot Options Windows



Building a symbol

 This part of the tutorial deals with the layout of the symbolic representation of a circuit, in this case an inverter. As with the schematic, we will create a new cell view for the symbol. You can make the new cell view though any of the methods we have previously discussed. We will also name this view **inv**, but it will be a symbol rather than a schematic, Figure 9.

	eate New File	·
Cancel	Defaults	Help
ame _	digital_lib	
, li	nvį	
ne 🛛	yndolį́	
	Composer-Syn	nbol 🔐
ath file		
	amei ne	ame digital_lib ing ne symbol Composer-Sym

Figure 9: Create New Symbol Window

Make sure symbol is the View Name and Composer-Symbol is the Tool. Press OK to continue and a familiar window will appear in which you will draw the symbol, Figure 10.



VANDERBILT UNIVERSITY

Tools Design Window Edit Add Check Options Help ٢ €² ି୍ଦ Q Ì £ Q [@abc] C mouse L: mouseSingleSelectPt M: schHiMousePopUp() R: schHiCheckAndSave() 🔁 🛱 🗳 🗇 🚍 ŏ

Figure 10: Virtuoso Symbol Editing Window

First, we need to draw a triangle on its side to represent the body of the inverter. Draw a vertical line on the left of the center line and connect the ends at a point on the right of center. To access the tools needed to draw the shapes click **Add->Shape->Polygon**. Figure 11 shows this window and other required settings.

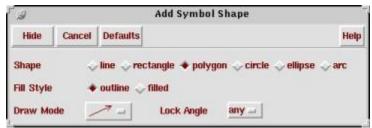


Figure 11: Add Shape Window



After adding the triangle, we need to add a circle at the right point to indicate negation. Click **Add->Shape->Circle** and then on the diagram click where you want the center of the circle, move the mouse until the circle is the size you want, and click again to stop sizing the circle.

Next we need to add the pins to our inverter. As before, click **Add->Pin** to bring up the Add Pin Window. We will again create pins **vdd**, **vss**, **in**, and **out**. As in the schematic make sure the pins are specified as **inputOutput**. The Add Pin Window can be seen in figure 12 below.

A			Add Pin	
Hide	Cancel	Defaults		Help
Pin Name	rs	Judd vas in o	ut	
Direction	0	inputOutput 🚄	Bus Expansion 4	• off 😞 on
Туре		actHi 🚄	Placement 4	🕨 single 😞 multiple
Label Of	fset		Label Location 🔍	none 🔶 left 😞 right
Rotat	e		Sideways	Upside Down

Figure 12: Add Pin Window

Now we need to place the pins. This is done in the same manner as in the schematic, but these pins are represented differently. On these pins, one end of the line has a small box. This box is the actual pin. Place this box away from the symbol. The shortcut of right click to rotate will save a great bit of time here. Because the pin names may be hard to read now, you can move them by clicking **Edit->Move** or using the shortcut '**m**'.

Now we want to label the symbol. We will add two labels here. Begin by clicking **Add-**>**Label**, which will invoke the Add Label box, Figure 13.

	Ad	d Symbol Labe	el .	
Cancel	Defaults			Help
[[FinstanceNam	ue][
ht 0.	0625	Label Choice	instance label	1
e 📘	stick 🔟	Label Type	🔷 normal Label	
ion lo	werCenter _	1	* NLPLabel	
	ht 0	Cancel Defaults Cancel Defaults [@instanceNan ht 0.0625 e stick	Cancel Defaults Cancel Defaults [@instanceName][ht 0.0625 e stick Label Type	[@instanceName] ht 0.0625 Label Choice instance label e stick Stick Label Type will shele

Figure 13: Add Label Window

The first label is going to be the *instance label*. This is the default label as seen above. Place this label near the symbol of the inverter. The second label will be the name label. If the Add Label box disappeared after adding the instance label, bring it back up. Change label type to **normalLabel** and type the name of the device in the Label field, Figure 14. Place this label on or near the symbol. When we place the symbol in schematics, all of the symbols will have the same name, but different instances, e.g. U1, U2, etc.



🕅 Add Sy	mbol Lab	el				×
Hide	Cancel	Defaults				Help
Label	I	ΩV	-			
Font Heig	ht 0	. 0625 <mark>.</mark>	Label	Choice	instance label	-
Font Styl	e 🗌	stick =	Label	Туре	🔶 normalLabel	
Justificat	ion lo	werCenter	-		NLPLabel	
					↓ ILLabel	
Rotate	e					

Figure 14: Add Symbol Label Window

The last task is to add the selection box to the symbol. Click **Add->Selection Box** and click on the choice of **Automatic** on the window that appears.

The symbol is now finished and should look similar to the symbol below in **Figure 15**. If you are satisfied with the symbol, save it by clicking **Design->Save**.

VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

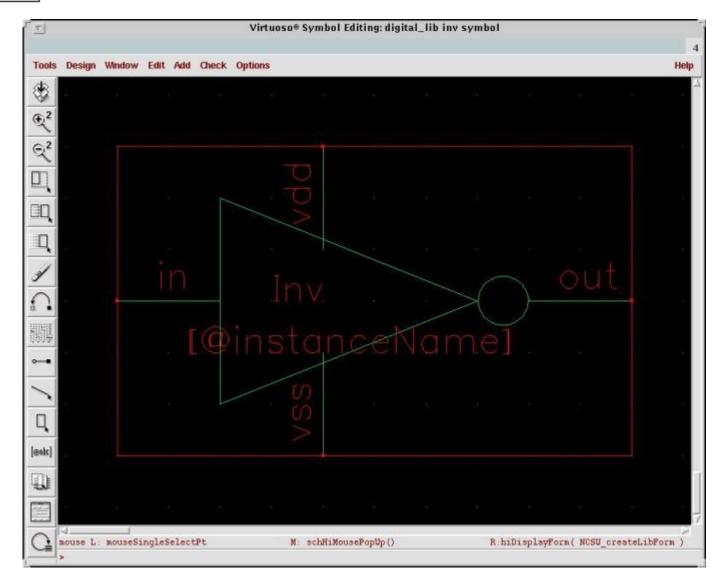


Figure 15: The Finished Inverter Symbol

Simulating the Circuit

This final part of the tutorial deals with simulating a circuit that has been designed, in this case the inverter. To test out the inverter, we will make a new schematic of a network with the inverter in it. To create a new schematic, we need a new cell view. We can create a new library entirely, but for simplicity, we will create a cell view within our current library. This is done in the same manner in which it was done in all of the previous steps. We will call the new cell view **inv_test**. Make sure that the selected tool is **Composer-Schematic**.

In the new schematic, we need to add the inverter, two DC voltage supplies, a ground connection, and a capacitor. First we will add the inverter. Click **Add->Instance**, change the library to your library, and select inv. Place it in the center of the schematic. Next we will add the two voltage supplies. The first we will add is the input supply. Click **Add->Instance**, change to NCSU_Analog_Parts, click Voltage Supplies, and select vdc, Figure 14. Place the first supply to the left of the **in** pin on the inverter. You may want to rotate the supply so that it is horizontal with the positive terminal closest to the **in** pin. Before placing the next supply (the vdd supply) we need to change a parameter for the supply. In the Add Instance window, set the DC Voltage to 5 V. Place the supply at the top of the schematic.

S.			Add Inst	ance	
Hide	Cancel	Defaults			Help
Library 1	ICSU_An:	alog_Par	tš		Browse
Cell	zdoj				
View	symboli				
Names 🏾					
Array	F	Rows	1	Columns	
Rotate		Si	deways		Upside Down
AC magnitu	de		V.		
AC phase			Ľ.		
DC voltage			5 ¥ <u>I</u>		
Noise file n	ame		ļ.		
Number of	noise/fre	eq pairs	<u>ď</u>		
_			Y		

Figure 14: Add instance of 5V DC voltage supply

Next we need to add the ground connection below the **vss** pin of the inverter. This is found under Supply_Nets in NCSU_Analog_Parts. We also need to add a capacitor at the output of the inverter to act as a load. This is found under Parasitic_Devices in NCSU_Analog_Lib. When all of the components are placed, we need to add a pin at the output of the circuit. Add a pin called out. This pin should be an INPUToutput pin as have all of the previous pins. The finished circuit should look similar to the one below:

VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

<u>v</u>]													٧i	rtu	05	0@	S	che	em	at	tic	Ed	liti	ng	i: n	ny.	_li	ib i	inv	'_t	es	t s	ch	e m	nat	ic												
Cmd	:			Se	1: 0																																											
Tools	Desi	ign	Win	dov	,	Edi	it	Ac	ld	C	he	ck	S	hee	et	0	pti	ons		NC	csi	J											_													100	H	elp
7																																																
2																																																
2																																																
×								Γ						ł	-(0	-	i.																														
																5=0																																
×8×																PV.																																
																		1				/dd																										
/																					0	1	1.																									
1000										~														1			8	1 80									-28											
]								1	-	U	Nigs	-								NJA					,/	/	×	7									2	01	UL.									
																						X		/									C															
																						10										1		c:1p														
*hunt																						3																										
,								-				-01							•••			1	qu	d					344																			
-																						7	7.																									
ត																																																
\supset																																																
	8 S	8	8 B	8	22	93 9	÷2	12	: 3	3	2	£3	<u> </u>	<u>i</u>	12	ŝ	ŝŝ		\$3	ŝ	32	1	S - S	8 B	2	2	ŝ	ŝ.	22	ŝ	1	2		\$î -	33	2	Ş	32	82	ŝ	1	22	32	22 - T	2	22	12 - S	
	√ mouse	L :	sho	wC.	Lic	kI	nfo	o ()		_	_	_	_			_	3	M :	30	hF	HiM	lou	seP	op	Up	()				_				R	: e	ch	Hi	Dre	at	eIn	ist	0			_	_	1	×
	>																																															

Figure 15: The test schematic

2) We are now ready to simulate this schematic. Click **Tools->Analog Environment** to open the window that gives access to the simulation tools, Figure 16.

VANDERBILT UNIVERSITY

r Affir	ma Analog Circuit Design Environment (1)	
Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ł
Library my_lib	# Type Arguments Enable	JAC FTRAN JDC
Cell inv_test View schematic		III III XYZ
Design Variables	Outputs	Œ,
# Name Value	# Name/Signal/Expr Value Plot Save March	4
		8
>	1	\sim

Figure 16: The Analog Environment Window

Make sure the simulator is set to **spectreS**. Go to the tab, Setup, and select Simulator/Directory/Host... In this window make sure that simulator is set to specterS.

🗽 Choosi	ng Simulat	or/Director	y/Host Virtuoso® Analog Design Environment (1)	x
ОК	Cancel	Defaults		Help
Simulator		spectr	eS =	
Project D	irectory	/nfs/jf	fs3/users/dickk/cadence/simulation	
Host Mod	le	🔶 local	\Diamond remote $ \diamondsuit $ distributed	
Host				
Remote D)irectory			

Figure 17: Setting up the simulator

Now we need to specify what we want to do with our simulation. We first need to specify our analysis, click **Analyses->Choose**. In the window that appears, select **dc** and click **Component Parameter**. Next double click **Select Component**, so that you can select the input DC voltage source on the schematic, Figure 18. This will bring up another window in which you need to select the parameter of the source that will be used in the analysis. Select **dc**.



VANDERBILT UNIVERSITY



Figure 18: Select Component Parameter

Next enter the starting and stopping voltages for the supply (Start: 0V, Stop: 5V). Select linear for **Sweep Type** and make the **Step Size** 0.1. The Analysis window should appear similar to the one below:

SCHOOL OF ENGINEERING IVERSITY

EECE 285 – VLSI Design

	ZAN			ъъ	 т.		
· ·	10.0	4 1.4	18	K 8	 	JIN	

ок	Cancel	Defaults	Apply			Help
Analysi			 ◆ dc ◆ sens ◆ pac ◆ pdisto 	 ↓ ac ↓ sp ↓ pnoise ↓ qpnoise 		
0 0	c 0	rating Poin	DC Analys	is		
💷 De	mperat sign Va mpone	ture		oonent Name Select Co neter Name	/VQ̃ omponent dcš	
∳ Ce Sweep Line	art-Sti nter-S Type ear	op Sta Ipan	art (<u>č</u>	Sto Size er of Steps	p <u></u> [5]0.1]	
Add Sp	ecific I	Points 🔟				
Enabled					Option	s

Figure 19: Analysis Setup Window

Click Ok to close the window. The next step is to select the outputs. Click Outputs->To Be Plotted->Select On Schematic. This will bring the schematic window to the front. Select the nodes at the input and output of the inverter, Figure 20.



VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

<u> </u>									irma Analog Circuit Des		
Cmd	:	Sel:	0	Sta	atus: Si	electing	outputs to	be plotted		T=27 C Simulator: spect	re
Fools	Design	Window	Edit	Add	Check	Sheet	Options	NCSU			Hel
7											
2											
2											
2						ipuð	net4				
\mathbf{N}							()- 				
24							ັດ 				
>>>							vdc				
×8×											
11								N 000 00			
/				ij	net6						
2				Ð	C 2		in		out 💦 👝 a	atil	
÷.								NV / V			
58E								00 100 00	a cout ⁴ CØ		
聯									, , , , , , , , , , , , , , , , , , ,		
									gndl 🙀		
T											
•											
								ын <mark>1984</mark> на нь			
··.*											
-											
\supset											
	4										1
		showCli	ckInf	o()			M: 8	schHiMousePopUp()	R:sevChange	eOutsOnSchematic('sevSe	ssio

Figure 20: Test Schematic with input and output nodes selected

This is done by clicking them once. Minimize the schematic and return to the Analog Environment window. This window should now look like the one below, Figure 21.



Status: Selecting outputs to be plotted T=27 C Simulator: spectre											
Variables Outputs Simulation	Results Tools Help										
Analyse	s 🕹										
# Type Arguments	Enable										
1 dc 0 5	100m Line. yes										
Output:	· []										
# Name/Signal/Expr Va	lue Plot Save March										
1 out	yes allv no 🍼										
2 net6	yes allv no										
	10										
	18										
	10										
	# Type Arguments 1 dc 0 5 1 dc 0 5 Outputs # Name/Signal/Expr Val 1 out 1 out										

Figure 21: Analog Environment Window after simulation setup

Finally, we are ready to run our simulation. Click Simulation->Netlist and Run. This will run the simulation and should produce an output file and a plot of the inverter's input and output, Figure 22.

VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

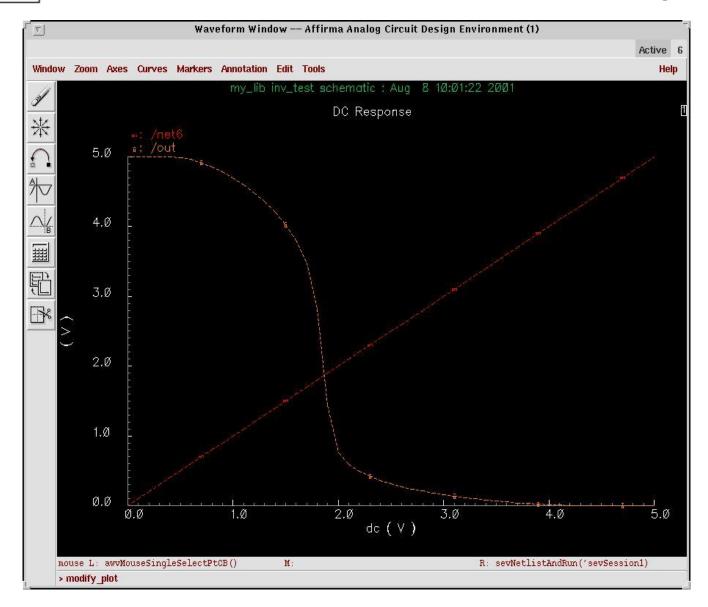


Figure 22: Input/Output plot for the inverter



VANDERBILT UNIVERSITY

File	Help
equations 5 bsim3v3 2 capacitor 1 vsource 2	
Entering remote command mode using MPSC service (spectr spectre0_23730,).	e, ipi, vO.O,

DC Analysis `dc': VO:dc = (O V -> 5 V)	
<pre>Important parameter values: reltol = 1e-03 abstol(I) = 1 pA abstol(V) = 1 uV temp = 27 C tnom = 27 C tempeffects = all qmin = 1 pS</pre>	
$\begin{array}{llllllllllllllllllllllllllllllllllll$	$(2 \ \%)$ $(2 \ \%)$
modelParameter: writing model parameter values to rawfi element: writing instance parameter values to rawfile. outputParameter: writing output parameter values to raw	

Figure 23: Spectre output file

Designing the Layout

1) Opening Layout File

1.1) Log onto cadence.



SCHOOL OF ENGINEERING VANDERBILT UNIVERSITY

- 1.2) Select the inverter file from your library.
- 1.3) Open the schematic of the inverter.

Library Manager: Directorys/jffs3/users/dickk/NCSU_AMI06			
<u>File Edit View D</u> esign Manager <u>H</u> elp			
🔄 Show Categories 🔄 Show Files			
Library	Cell	View	
[digital_lib	Įinv	Ischematic	
NCSU_TechLib_ami16 NCSU_TechLib_hp06 NCSU_TechLib_tsmc02 NCSU_TechLib_tsmc03 NCSU_TechLib_tsmc03d NCSU_TechLib_tsmc04_4M2F US_8ths ahdlLib analogLib basic cdsDefTechLib digital_lib functional rfExamples rfLib	inv_test	layout schematic symbol	
Messages			
in /usr/local/isde/cadence/IC5141/tools.lnx86/dfII/etc/cdsDotLibs/composer/c to suppress this warning message.			

Figure 1: Opening schematic view

- 1.4) Go to the tab labeled Tool, and then Design Synthesis and select Layout XL.
- 1.5) This will open the Setup Options window. Select create new and click OK.

1.6) In the create new file window, fill in the cell name, make sure Virtuoso is selected for the Tool and then click OK. This will bring up the XL layout editor window and the LSW window.





VANDERBILT UNIVERSITY

🔀 Create	New File	e			×		
ок	Cance	el	Defaults		Help		
Library N	ame		dię	gital_lib			
Cell Name			n⊽				
View Name		13					
Tool			Virtuo	SO –			
Library p	ath file						
ffs3/users/dickk/NCSU_AMI06/cds.lik							
- 		_					

Figure 2: Creating a new file

V

SCHOOL OF ENGINEERING

VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

X: -5.5		Layout Editing: digital_lib inv Y: 10.0 (F) Sel			Select: (D DI	RD: OFF	dX:		dY:	E g
Tools	Design	Window	Create			Connectivit	y Options	Place	Routing	Migrate	
۲											
Q											
Ð											
Q											
<u> </u>											
Ц											
))											
\cap											
법법											
899											
Ľ											· ·
Q,											
[abcd]											

Figure 3: XL Layout Editor

2) Setting up the parameters

2.1) In the XL layout editor, go to the Design tab, and click on Gen From Source. This will bring up the Layout Generation Option window, in which you can select the options for generating the layout. Make sure that they match the selection below.

000	X Layout Generation Options	
OK Cancel	Defaults	Help
Layout Genera	ation	
Generate:	I/O Pins Instances Boundary Transistor Chaining Transistor Folding Preserve Mapp	ings
I/O Pins		
Apply	Pin Type Layer / Master Width Height Num O	reate
Defaults:	Geometric - netali dg - 0.9 1	•
Select:	Number Selected: 0 Add	a Pin
Term Name	Net Name Pin Type Layer / Master Width Height Num O	reate
"out" "vdd"	"out" "Geometric" ("metall" "draving") 0.9 0 "vdd" "Geometric" ("metall" "draving") 0.9 0	9 1 9 1 9 1 9 1
Update	Pin Type Layer / Master Width Height Num O Geometric	reate
Pin Label Sh	nape: 🔶 Label 😞 Text Display 😞 None 🦳 Pin Label Opti	ions
Boundary Layer:		
L	ectangle - Left: 0 Bottom: 0	
Boundary Area	a Estimation	
Utilizatio	n (%) - 25 Aspect Ratio (W/H) - 1	1
Area Calci	ulation: PRBoundary Based	
_ Load	I Template File for Layout Generation	
mels log	Browse Load	

Figure 4: Setting up layout options



2.2) Go to the tab, Options, and select Display. This will allow you to select what is being displayed on the layout screen. To see the different levels inside the transistor change the display level stop to 32.

Display Options	
OK Cancel Defaults Apply	Help
Display Controls	Grid Controls
 Open to Stop Level Axes Access Edges Instance Origins Instance Pins EIP Surround Array Icons Pin Names Label Origins Dot Pins Use True BBox Net Expressions Cross Cursor Dynamic Hilight Stretch Handles Shape Information 	Type none ♦ dots lines Minor Spacing 1 Major Spacing 9 X Snap Spacing 0.9 Y Snap Spacing 0.9
Path Display Borders and Centerlines	Size 🛱 Style empty =
Show Name Of instance master Array Display Display Levels Full Start I Border Source Stop 31	Snap Modes Create orthogonal Edit orthogonal
$igstar{}$ Cellview \diamondsuit Library \diamondsuit Tech Library \diamondsuit File	<pre>~/.cdsenv</pre> Browse
Save To Load From	Delete From

Figure 5: Display options

3) Setting up the connections

3.1) To move your components around select the move tool, on the left side bar. Move the components into the first quadrant and position them so that they can be connected.3.2) When you select a component to move in the XL layout editor window, the component will be highlighted in the schematic window. It will also show you what else it is connected to by showing a line to the connection. This will help you, if you forget what something is connected to.



VANDERBILT UNIVERSITY

EECE 285 – VLSI Design

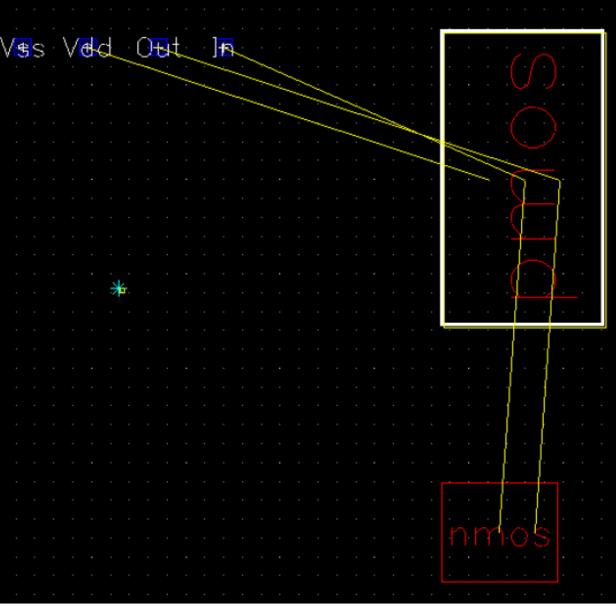
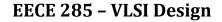


Figure 6: Moving pins and transistors

3.3) To connect the components together go to the LSW window and select the intended layer and select the rectangle tool from the left side bar in the LX layout editor window. Connect the nmos and pmos gates with poly and the pins with metal1. To connect the input pin to the poly you will need the poly to metal 1 via.



SCHOOL OF ENGINEERING VANDERBILT UNIVERSITY



	x								
Sort Edit	Help								
poly	drw								
NCSU_TechLib_ami06									
Show Objects									
📕 Inst 📕 Pin									
AV NV AS	NS								
💹 pwell	drw 🛆								
nwell	drw								
active	drw								
nactive	drw								
pactive	drw								
nselect	drw								
pselect	drw								
poly	drw								
elec	drw								
metal1	drw								
metal2 metal3	drw drw								
CC Metals	drw								
via	drw								
via2	drw								
glass	drw								
A highres	drw								
nodrc	drw								
nolpe	drw								
M pad Figure 7: LSW	drw								

Figure 7: LSW window

3.4) Make sure to keep your cells the same height so that they will be easy to connect to each other.



SCHOOL OF ENGINEERING

EECE 285 – VLSI Design

Tools	Design	Window	Create	Edit	Verify	Connectivity	Options	Routing	Migrate	NCSU		Help
*	 <u>.</u> .	· · ·	· · ·	· ·					· · · ·	· · ·	· · · · · · · · · · · · · · · · · · ·	
							V////X#2	*****				
Q								b Aq	d			
												· · ·
Ð												
~									-			
୍												
~												
•												· · ·
Q												
30												
\sim												· · ·
£]												
:::::::::::::::::::::::::::::::::::::::									Y · ·			
									4			
Ð.								A				
999												
۲ ۲												
-												
S.												
L.												
[abcd]												
[assa]								<mark>a</mark> gn				
П												
				1	Figur	∙e 8∙ Fin	allavo	ut de	sign o	fan	inverter	

Figure 8: Final layout design of an inverter

4) Running DRC (Design Rule Checker)

4.1) Throughout the design process, make sure to run DRC, which is located under the tab, Verify. When the DRC window appears click the OK button for it to run. The DRC check will give you the number of the rule, if there is a violation.



VANDERBILT UNIVERSITY

DRC								
ок	Cancel	Defaults	Apply			Help		
Checking Method 🔹 flat 😞 hierarchical 😞 hier w/o optimization								
Checking	Limit		Ť 📃	ental 🔷 by a	area	_		
		Coordin	ate			Sel by Cursor		
Switch N	ames		I			Set Switches		
Run-Spe	cific Comr	nand File						
Inclusion	Limit		1000 <mark>(</mark>	Limit	Rule Errors	0		
Join Nets	With San	ne Name		Limit	Run Errors	0		
Echo Con	imands							
Rules File	I.		divaDR	C. rul <mark>j</mark>				
Rules Lib	ra ry		U_ T	echLib_ami0	6			
Machine			🔶 local	🔷 remote	Machine			
Use Error	Database	9						

Figure 9: DRC window

4.2) The results of the DRC can be viewed in the icfb log window. It will list the number of errors as well as the rule that was violated.

🔀 icfb - Log: /nfs/jffs3/users/dickk/CDS.log		x
File Tools Options	Help	1
DRC startedMon Jan 11 10:56:47 2010 completedMon Jan 11 10:56:47 2010 CPU TIME = 00:00:00 TOTAL TIME = 00:00:00 ********* Summary of rule violations for cell "INVX1 layout" ******** Total errors found: 0		
>		

Figure 10: Checking the results of the DRC

4.3) One of the DRC errors that you might encounter is an off the grid error. This happens when wire nets or symbols are not on the same grid. There is a default grid setting, but it can be changed. It is under the display options. It is a good idea to keep them on the default settings. This can also occur if you use symbol created on a different grid setting. For example if you create an inverter symbol instance on a grid with minimum of 0.3 you can draw the inverter symbol such that it is 2.4 X 3.6 in size. However, if your grid is set at 0.5, then errors will occur during the DRC check. A 0.5 setting could handle 2.5 X 3.5. In other words, if you



use a W and L that is not a multiple of the minimum grid setting you will get an "off the grid error".

5) Extracting the circuit

5.1) After you have finished the circuit you will need to extract it. This can be done by going to Extract under that tab Verify. This will allow you to check the circuit against the schematic to see if they are equivalent. When Extract is selected a window will appear that gives the extract information. Click on the OK button.

Extractor		×
OK Cancel Defaults	Apply	Help
Extract Method 🔶 fl	at \diamondsuit macro cell \diamondsuit full hier \diamondsuit inc	cremental hier
View Names Extracte	d extracted Excell	excell
Switch Names	I	Set Switches
Run-Specific Command File		
Inclusion Limit	1000 Limit Rule Errors	0
Join Nets With Same Name	Limit Run Errors	0
Echo Commands		
Rules File	divaEXT. rul <mark>i</mark>	
Rules Library	U_TechLib_ami06	
Machine	◆ local ◇ remote Machine	
Use Error Database		

Figure 12: Extract window

6) Running LVS (Layout versus Schematic)

6.1) When you are done laying out the inverter in the layout window you can compare it to the schematic to see if they match. This can be done by using LVS. LVS is located under the tab Verify.

6.2) When LVS is opened an LVS contents window will appear. This is the window that lists the contents of what is going to be compared. If it does not match what you desire to compare then go to the Artist LVS window and fill in the correct information for the schematic and layout. You will need to fill in the Library that the schematic and layout are in, the cell name for the schematic and layout and the view name. Then click on the Run button, located in the bottom left.



VANDERBILT UNIVERSITY

🔀 Artist LVS Fo	orm Contents Different	×
OK Cance	4	Help
The selected l	LVS Run directory does not match the Run Form.	
Use	🔷 Form Contents 🔶 Run Directory Contents	
Differences:	Extracted Library Name: OSU_stdcells_ami05_h Extracted Cell Name: padframe40_7 Extracted View Name: extracted Schematic Library Name: OSU_stdcells_ami05_h Schematic Cell Name: padframe40_7	



🕅 Artist LVS	-	these through t	Manager -			
Commands				Help 4		
Run Directory	LVS			Browse		
Create Netlist	📕 schema	tic	📕 extract	ed		
Library	OSU_stdc	ells_amiO5_h <mark>i</mark>	0SU_stdce	ells_amiO5_h <mark></mark>		
Cell	invx1 <mark>,</mark>		INVX1			
View	schemati	4	extracted			
	Browse	Sel by Cursor	Browse	Sel by Cursor		
Rules File	divaLVS.:	rul		Browse		
Rules Library	■ OSU_s	tdcells_ami05_	h			
LVS Options	📕 Rewirin	g	_ Device	Fixing		
	_ Create	Cross Reference	Termina 📕	ds		
Correspondence	File 🔲 🖸	kk/ncsu_ami06;	/lvs_corr_:	file Create		
Switch Names	I					
Priority 🦉	Run bac	kground 💷 📜				
Run	Output	Error Display	Monito	r Info		
Backannotate	Parasiti	c Probe Build	Analog	Build Mixed		

Figure 12: Artist LVS window

6.3) When the LVS is complete a window will appear that will let you know if the layout and schematic matched. The result of LVS can also be seen in the icfb window.





Figure 13: LVS Result window

🔀 icfb - Log: /nfs/jffs3/users/dickk/CDS.log		x
File Tools Options	Help	1
Total errors found: 0		" <u> </u>
Total errors found: O Loading seCore.cxt LVS job is now started		
The LVS job has completed. The net-lists match. Run Directory: /nfs/jffs3/users/dickk/NCSU_AMI06/LVS		
mouse L: showClickInfo() M: leHiMousePopUp() R: setExtForm()		
>		

Figure 14: icfb window