Cadence Tutorial Schematic Entry & Simulation (Using Virtuoso Schematic and Spectre)

Department of Electronics & Communication Engineering Indraprastha Institute of Information Technology, Delhi, India. The following Cadence CAD tools will be used in this tutorial:

- Virtuoso Schematic for schematic capture.
- **Spectre** for simulation.

We will practice using CADENCE with a CMOS Inverter: creating (1) Schematic (2) Simulation

Computer Account Setup

Please see the Unix/Linux command_before doing this new tutorial.

YOU SHOULD HAVE YOUR ENVIRONMENT SET UP FOR CADENCE AND ADDITIONAL TOOLS

Running the Cadence tools

Log in to your UNIX/LINUX account. Open the terminal window.

Now you should be able to run the Cadence tools. Never run Cadence from your root directory, it creates many extra files that will clutter your root. Instead please create a directory (e.g. cadence).

>> mkdir cadence
>>cd cadence
Now start Cadence by typing
>>csh
>> source cshrc
>>cd cadence_ms_labs_613
>>virtuoso

Please see the Fig. 1 for above command.

🙀 Red Hat Enterprise Linux 5 - VMware Player (Non-commercial use only)	
Player 👻 📔 💌 🔚 🔚	*
Applications Places System 🥪	8:04 PM 🜒
root@localhost:~/cadence	
<u>File E</u> dit <u>V</u> iew <u>T</u> erminal Ta <u>b</u> s <u>H</u> elp	
<pre>[root@localhost ~]# cd cadence/ [root@localhost cadence]# csh [root@localhost ~/cadence]# source cshrc Welcome to Cadence tools [root@localhost ~/cadence]# cd cadence_ms_labs 613 [root@localhost cadence_ms_labs_613]# virtuoso</pre>	
	≡

Fig. 1 Terminal window

The command will start Cadence and after a while you should get a window with the "Virtuoso@ 6.1.5", also called **Command Interpreter Window (CIW)** as below: Fig 2

Ĉ	Virtuoso® 6.1.5 - Log: /root/CDS.log	
<u>File T</u> ools <u>O</u> ptions <u>H</u> elp		cādence
Loading schView.cxt Loading selectSv.cxt Loading LVS.cxt Loading layerProc.cxt Loading xlUI.cxt Loading auCore.cxt Loading dcm.cxt Loading vhdl.cxt Loading vhdl.cxt Loading seismic.cxt Loading ci.cxt Loading ams.cxt Virtuoso Framework License (111 *WARNING* LIB modk180 from File) was checked out successfully. Total checkout time was 1.52s. /root/cadence/cadence ms labs 613/cds lib Line 9 redefines	
4		
mouse L: 1 >	M:	R:



For more information on the various Cadence tools I encourage you to read the corresponding user manuals. You can get to the manuals by pressing **Help -> Virtuoso Documentation** on any Cadence window (e.g. CIW)

Now we need to create a new library (to contain your circuits) so from the **Virtuoso (Fig 2) Command Interpreter Window (CIW)** go to **File -> New -> Library** from the File menu. You will see a "New Library" window (Fig 3). Fill in the name of the new library (e.g. CMOSInverter) in the dialog window (this will create the library in the directory where you started "Virtuoso", you could also choose to set a path if you wanted another directory). Click on "Attach to existing tech library" and click OK.

New Library 🗙						
Library	Technology File					
Name CMOSInverter	Compile an ASCII technology file					
Directory (non-library directories)	 Reference existing technology libraries 					
	 Attach to an existing technology library 					
DRCrun	Do not need process information					
LVS						
bhanu1						
bpLogs daiict	Design Manager: No DM					
pot/cadence/cadence_ms_labs_613						
	J					
	OK Cancel Defaults Apply He	lp)				

Fig. 3 New Library Window

The above steps can be also performed using the Library Manager. After you start Cadence and get the "Virtuoso CIW" window, go to **Tools->Library Manager** or press **F6** on keyboard. It will open the Library Manager window (Fig 4) as shown below. You can create the new library (CMOSInverter) from the Library Manager following the same steps as explained above. Now the "CMOSInverter" library should appear in the Library Manager window. It is easier to work with Library Manager. However, for this document we will work through Virtuoso- CIW window.

Let's start our first schematic now!

SCHEMATIC CAPTURE

In the **Virtuoso CIW** window go to **File -> New -> Cell View**. You will get a "Create New file" window (Fig 4). Fill in the information in the dialogue window as below and then press OK.

Library Name : CMOSInverter

Cell Name : myinverter (you can choose other name if you want) View Name : Schematic

Tool : Composer-schematic

	New File	×
_ File		
Library	CMOSInverter	
Cell	myinverter	
View	schematic	
Туре	schematic 🔽	
Application		
Open with	Schematics L	
🔲 Always use t	his application for this type of file	
Library path file		
/root/cadence	e/cadence_ms_labs_613/cds.l	ib
	OK Cancel He	elp

Fig 4 Create New File window

Wait for a while. "The schematic window will appear. You should get the "Virtuoso Schematic Editing" window as shown below (Fig 5). Spend some time analyzing the window. On the left side you have various shortcuts to common used commands such as: placing component instances (looks like an IC), drawing wires, placing ports, stretching, copying, zooming in and out, saving, etc. If you pass the mouse pointer on top of the buttons you get short pop-up help messages. You also have access to these commands (and others) from the menu. It is not possible here to describe all the functionality of Virtuoso Schematic so you are strongly encouraged to read the on-line user manuals.

You should notice that the top bar of the window will display the name of the library (CMOSInverter), cellview (myinverter) and schematic at the end.

Virtuoso ® S	chematic Editor L Editing: CMOSInverter myinverter schematic	
<u>L</u> aunch <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>C</u> r	reate Chec <u>k</u> O <u>p</u> tions <u>M</u> igrate <u>W</u> indow <u>H</u> elp	cādence
		N
0 - 0 - 0 0 W	orkspace:Basic 🔽 🗟 🗔 🔩 🔩 🗛 🗛 🔹	🗣 Searc 🔽 👻
Navigator ? 🗗 🗙		
🔽 Default		
O Saamsh		
Name		· · · · · · · · · · · ·
		· · · · · · · · · · · ·
1		
Property Editor ? 🗗 🗙		
	• • • • • • • • • • • • • • • • • • • •	
		· · · · · · · · · · · ·

Fig 5 Virtuoso Schematic Editing window (Composer)

Let's start our first schematic to create the CMOS Inverter. Expand the Virtuoso Schematic Editing window if necessary. We will pace the NMOS and PMOS transistors on the schematic.

Placing Instances

Click on the "**Instance**" button (icon) on the left side (which looks somewhat like an IC, or go to Add -> Instance), this will pop-up an "Add Instance" window (Fig 6).

Virtuoso® Scl	hematic Editor L	Editing: CMOSInverter	myinverter sche	matic _ 🗆 🗙
<u>L</u> aunch <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>C</u> rea	ate Chec <u>k</u> O <u>p</u> tions	<u>M</u> igrate <u>W</u> indow <u>H</u> elp		cādence
🗅 🗁 🛃 🗔 🚸 (0 🛛 🗙 🕦	T⁄ 🥱 ৫ 🖄 ▾	T T I Q Q	a 🔣 📑 1 🔹
◯ • ◯ • ◯ ◯ Worl	kspace: Basic	- 5 5 5	, 15 16 48 T 3	🔹 🔤 🗸
Navigator ? 🗗 🗙		Add Instance	×	
Y Default				, ^ ; ,
🔍 Search 🔤 🔻	Library		Browse	ter for the following total :
Name 🗠	Cell			
E myinverter	View symb	ool		ter diet feit teder bet teffet i Na die jes alders die die s
	Names			191 193 193 19492 193 19392 1 194 193 193 19493 194 195 1952 1
	🗹 Add Wire Stu	ubs at:		
	0	all terminals 🧕 registered term	iinals only 📃 🔜	
Property Editor 2 R X	Array	Rows I Con	umns I	
	AL Ro	tate 🔵 🕢 Sideways 🕽 🧯	🗧 Upside Down	
		Lide Concel	Defeulte	
		Hide Cancer		
		· · · · · · · · · · · · · · · · · · ·		
IIImouse L.		M·		R.

Fig 6 Add Instance Window

Now click on the Browse. Another window called "Library Browser – Add Instance" (Fig 6) will pop up. We will select PMOS transistor and will place it on the Virtuoso Schematic window. Follow the steps now. Select as follows in the Library Browser window (Fig 7).

Library =>gpdk180 Cell => pmos View => symbol



Fig. 7 Library Browser window

Change following properties of *pmos* in "Add instance" as given here.

Names => M1 Width =>800 nm Length => 180 nm

Similarly, now we will place the NMOS transistor. Go back to the "Library Browser-Add Instance" by left clicking on this window. Select as follows:

Library =>gpdk180 Cell => nmos View => symbol

Notice carefully. You need to select "nmos" under the cell for NMOS.

Now click on the "Add Instance" window. Change following properties of the *nmos* in the "Add instance" window.

Names => M2, Width => 360 nm, Length => 180 nm

Ok, you have NMOS and PMOS on your schematic. So far so good!!!!

To change the parameters of the instance, select the instance (by clicking on it with the mouse) and then use "properties" icon or press "q".

Now we also need to add wires, I/O pins and power supply.

First let's add wires (narrow) to connect transistor's terminals and form a schematic of the CMOS Inverter.

M	Virtu	ioso®	Sche	ema	tic	Edi	tor L	. Ed	iting	g: CM	os	inverte	r my	inver	ter :	sche	mat	ic				
<u>L</u> aunch <u>F</u> ile	e <u>E</u> dit	<u>V</u> iew	<u>C</u> reate	e Cl	hec <u>k</u>	s o	<u>p</u> tions	Mi	grate	<u>W</u> ind	low	<u>H</u> elp								сā	d e r	nce
	2 6] ની	÷ ()	KI		×	1	T⁄		5	¢	42 -	T	T		Q		F		망	1	»
	• 🛈	6	Works	pace	Ba	asic				-	1	3	3 II	3 243	-13	Ţ		₿	Q	Sear	rc 🔽	•
Navigator	_	? 8	×																			
The fault		-)																			5
Q Search			•							pmo	os) ne 1''									<i>.</i> :
Na Na	ume Pr	1								<u> </u>	r<	w:2u										a
	nmos)																					ŝ
	omos)											m:1	Π									цэ.
										nmo	bs	NM2) ວຣ1″	•								15
										<u> </u>		w:2u										
Property Edi	itor	? 🗗	×								1											
											_] I:18Ø	n,									3
												m:1										
<u><</u> [-22		a a							82			
i≡mouse L:										M:												R:

Fig 8 NMOS and PMOS on schematic

Connecting Wires

To connect the wires, click on the icon "Wire (narrow)" on the left side. You will see an "add Wire" window. You can choose the color whatever you want to. Now activate the Virtuoso Schematic Editing window by clicking on its title bar.

Move the mouse over or click on the **s** key on your keyboard. This snaps the wires to connect between the little diamond-shapes displaying by the nodes. You can click on the node (diamond-shape) with left mouse button, move the mouse over (you will see wire attached) and then double click at other end to connect wire between those points. Connect all the wires likewise. When you are done hit **"Esc"** on the keyboard. You can delete unwanted wires if connected accidentally. To delete the wire, select that wire by left mouse click and then hit Delete on the keyboard.

See Fig 9

📡 Virtuoso ®	Schematic	Editor L Editin	g: CMOSInverter my	inverter schematic	
<u>L</u> aunch <u>F</u> ile <u>E</u> dit <u>V</u> iew	<u>C</u> reate Chec <u>k</u>	O <u>p</u> tions <u>M</u> igrate	e <u>W</u> indow <u>H</u> elp		cādence
	• 🗘 🖾 🕽	K 🕕 T⁄ 🖪)	T Q Q Q [3 1 75 1. »
	Workspace: Bas	sic	🗖 🗣 🚭 🛛 🗖	5 12 - 13 I B	🗣 Searci 🔽 👻
Navigator ? 🗗	×				
🍸 Default 🔤 🛛			nmon PMØ		· · · · ·
🔍 Search 🔤 🤊	• • • • • •			,	• • · ·
Name 🗠	ī				
Myinverter					
PMO (pmos)			l:18Øn		
			Tm:1		
			nmos NMØ	· · · ·	
	1.1.1.1.1.1		"nmos1"	e e e e e e	
 		• · ·	<u> </u>		
Property Editor ? 🗗	×				
			l:18Øn		
			- m:1		
					1
⊪mouse L:			M:		R:

Fig 9 CMOS with no pins

Let's connect the I/O pins now.

Adding Pins

To add the *input* and *output* pins, click on the "Pin" icon at the lower left corner. The "Add Pin" form appears.

Under the Pin Name type *Vin*. Note that Direction in the form reads input, as shown below (Fig 10).



Fig 11 CMOS with pin, gnd and supply (Vdd)

The final schematic should look somewhat like this (Fig 17). Success?

It's a good idea to save your design from time to time in case the system crashes.

Everything worked fine so far!!!! Congratulation!!!

Check and Save your design

Now you need to Check and Save your design (either click the top left button or go to Design -> Check and Save). Make sure you look at the **"Virtuoso"** CIW window and there are no errors or warnings, if there are any you have to go back and fix them!

The Virtuoso window will give the message as shown below.

"Schematic" check completed with no errors.

"CMOS Inverter myinverter schematic" saved.

Let's now perform the simulation on the inverter circuit to see the final results!!!

In the Virtuoso Schematic window go to Launch \rightarrow ADE L Then you will get the simulation window or ADE pop-up window

Please see Fig. 12.

Tools -> Analog Environment. You will get "Virtuoso Analog Design Environment (1)" window (Fig 12).

In the Virtuoso Analog Design Environment, go to "Setup -> Model Libraries......".

Virtuoso® Analog Design E	nvironment L Editing: CMOSInverter myinverter	schematic	_ 🗆 🗙
Launch File Edit View Create Chec	<u>k Options Migrate W</u> indow <u>H</u> elp		cādence
🕌 Virtuoso® Analog Design Envir	ronment (1) - CMOSInverter myinverter schema	tic _ 🗆 🗙	
Launch S <u>e</u> ssion Set <u>up A</u> nalyses <u>V</u> ar	iables <u>O</u> utputs <u>S</u> imulation <u>R</u> esults <u>T</u> ools <u>H</u> elp C	ādence	S S
📇 🧽 🧊 27 💩 🎾 🖞) 🗹 🖻		Searc 🔽 👻
Design Variables	Analyses	? 🗗 🗙 🚃	
Name Value	Type Enable Arguments	ODC	
Traine Yalke		85	a a a a
			8 8 8 8
	N	P-m-+	
		×	a a a a
			5 6 32 st
	Outputs	? # X 👩	1 1 1 1
	Name/Signal/Expr Value Plot Save Save Opt	tions	
		VV	a a sa sa
			5 5 St of
			1 1 11 11
1	Dist after simulation. Auto		8 8 8 8 8 8 8 8 8 8 8 8 8 8 8 8 8 8 8
>	Plotting mode: replace		
mouse L:	M:	R:	MousePopUp()

Fig. 12 ADE Window

Now you need to choose the type of simulation. From "Virtouso Analog Artist" (Fig 13) go to Analyses -> Choose... (Fig 13). In this case we will choose a transient analysis. Enter the stop time for transient analysis. Let's type 200p for stop time. Click OK.



Fig 13 Choosing Analysis (Transient Analysis)

For DC analysis, please see the Fig. 14.

📓 Virtuoso 🕻	Choosing Analyses -	- Virtuoso® Ai	nalog Design Env 🗙	nverter schematic	
Launch <u>F</u> ile <u>E</u> dit Virtuoso® An	Analysis 🔾 tran 💿	dc 🔾 ac sens 🔾 dcmatch	 ○ noise ○ stb 	schematic 💶 🗙	cādence
Launch S <u>e</u> ssion	⊖ pz ⊖	sp 🔾 envlp	O pss	cādence	1 1
I 📥 🔊 I 🔭 🖥	O pac O psp O	pstb 🔾 pnoise qpss 🔾 qpac	 ○ pxf ○ qpnoise 		Searc 🔽 🔻
Design Variables	◯ qpxf ◯ ◯ hbnoise ◯	qpsp 🔾 hb hbsp	🔾 hbac		
Name -	D Save DC Operating Point Hysteresis Sweep Sweep Variable Temperature Design Variable Component Parameter Model Parameter	Component Nam	le /V3 Component e dc	Save Options	
> Results in /root/si immouse L: 2(3) Stop	Sweep Range Start-Stop Start Center-Span OK	Cancel Defa	Stop 18	Replace R: Simulator: spectre	MousePopUp(Jlator: spectre

Fig 14 Choosing Analysis (DC Analysis)

Now in the "Virtouso Analog Artist" (Fig 15) go to "Outputs -> to be plotted -> select on schematic". That will bring your inverter cell view window in front. Select node voltages by clicking on the net. We will click on input and output nets (wires) to select input and output voltages. The currents can be selected by clicking on the terminals (red squares).

Virtuoso® Analog Design Environment L Editing: CMOSInverter myinverter schematic	: _ –
Launch Eile Edit View Create Check Options Migrate Window Help	cādence
Virtuoso® Analog Design Environment (1) - CMOSInverter myinverter schematic _ □ × Launch Session Setup Analyses Variables Outputs Simulation Results Tools Help Cadence	₽ <mark>51</mark> »
Image: Setup Delete Design Variables Import Name Value Itrat Export To Be Saved . To Be Plotted . Save All Save All Outputs Image: Save Options	Searc
> Plot after simulation: Auto Plotting mode: Replace	
mouse L: M: R:	MousePopUp()

Fig. 15 ADE for Output plot

👫 Virtuoso® Analog Design Envi	ronment (1) - CMOSInverter myinverter schematic 🗕 🗆 🗙
<u>L</u> aunch S <u>e</u> ssion Set <u>u</u> p <u>A</u> nalyses <u>V</u> ar	iables <u>O</u> utputs <u>S</u> imulation <u>R</u> esults <u>T</u> ools <u>H</u> elp cādence
🛛 🚰 🖉 🔽 🖓 🛵 🎾	A 2
Design Variables	Analyses ? 5 ×
Name Value	Type Enable Arguments Otrans
	l l
	Name/Signal/Expr Value Plot Save Save Options
	1 vout 🗹 🗌 allv
	Dist after simulation, Auto Disting made, Benlace
>	

Fig. 16 ADE after setup

Now we can finally simulate! Click on the "Netlist and Run Simulation" button (looks like a green light) on the right or go to "Simulation -> Netlist and Run".

It will start simulation. You will need to wait for a while. You should check your "Virtuoso" window for messages while it is running the simulation.

In case you have errors you will need to go back and correct them. This can be tricky!

CAUTION Each time you change the schematic you have to do Check and Save!. You should finally get the desired simulation results, input and output





You can zoom in your waveforms.



Fig. 18 Simulation Results (Transient Analysis)

- 1. Symbol generation
- 2. Delay Calculation (From Schematic)
- 3. Power Calculation (From Schematic)
- 4. Area Calculation (From Layout)

Lab Assignment: #2

Design and simulate a 2-input NAND gate. Calculate the area, power, current, and Delay.

Good Luck