UNIVERSITY OF WATERLOO

UW ASIC GROUP: Cadence Tutorial

Description:

Part I: logging into UNIX and opening Cadence. Part II: simulation of a CMOS inverter.

PART I: LOGGING INTO UNIX AND OPENING CADENCE

LOGGING INTO UNIX:

UNIX workstations are located in E2 3364, 3366, and 3368.
 NB: Email the leader of the Analog Group to obtain the 5-digit code used to access these rooms.

2.	Use your SunEE	account to log into a UNIX workstation:
	user name:	your engmail user name
	password:	your UW student ID number.

OPENING CADENCE:

- Cadence must be opened from the folder in which you store your Cadence files. Never run Cadence from the root directory. First, open a terminal window:
 - (a) Right click on the desktop,
 - (b) Select Tools,
 - (c) Select Terminal.



- SSH into the uwasic account: ssh -l uwasic bladel.vlsi password: (the password will be provide to you in the tutorial).
- 3. Enter the *cadence* folder: cd cadence

Create a *tutorials* \underline{X} folder (<u>The value of X will be assigned to you in the tutorial</u>): mkdir tutorials \underline{X} cd tutorials \underline{X}

4. You are now in the folder where your Cadence files will be store, and may open Cadence: startCds -t cmosp18

This opens **icfb Command Interpreter Window (CIW)**. A window titled "What's New in 4.4.6" may also open – you may close this.

	icfb – Log: /u4/uwasic/CDSlogs/CDS.log.24072					
File Tools Options	CMC Gateway CMOSP18-Documentation	Help	1			
Completed /CMC/tools completed /CMC/tools >	s/cadence.2003a/IC.5032/tools.sun4v/dfII/local/skill/CMCinit.il s/cadence.2003a/IC.5032/tools.sun4v/dfII/local/.cdsinit					
<[\geq			
Ι						
mouse L:	M: R:					
>						

PART II: SIMULATION OF A CMOS INVERTER

Virtuoso® and Analog Design Environment are the two Cadence CAD tools that we will use. Virtuoso® is used for schematic capture. Analog Design Environment uses the Spectre simulation engine to simulate the schematic.

1. In CIW (icfb, shown above) open the Library Manager: Tools > Library Manager.

– Library Manage	r: Directorywasic/analog/cade	ence/tutorials	•			
<u>File E</u> dit <u>V</u> iew <u>D</u> esign Manager			Help			
Show Categories Show Files						
Library	Cell	View				
¥	¥					
CMCLayoutReference CMCpcells CMCshare ahdLib basic cdsDefTechLib cmosp18 cmosp18_defin_techlib package passiveLib rfExamples tp2973g virage_sram vst_n18_sc_tsm_c4						
- Messages						
View filter set to ""	ence/tutorials/libManager.log.24104".					

2. In Library Manager create a new library: File >New > Library.

-	New Library
Library -	
Name	tutorials
Directory	
rasic/ana	alog/cadence/tutorials
Design №	lanager
🗂 Use R	ONE
🗍 Üse No	I DM
ОК	Apply Cancel Help

(a) Enter the Library Name: *tutorials*.(b) Click OK.

— Technology File for New Librar						
OK Cancel Help						
Technology File for library "tutorials"						
If you will be creating mask layout or other physical data in this library, you						
will need a technology file. If you plan						
to use only schematic or HDL data, a						
technology file is not required.						
You can: OCompile a new techfile						
Attach to an existing techfile						
Don't need a techfile						

(c) Select Attach to existing techfile.(d) Click OK.

-	- Attach Design Library to Technology File								
OK Cancel Defaults Apply Hel									
Ne	New Design Library tutorials								
Technology Library					cmosp18				

(e) Choose **cmosp18** from the Technology Library dropdown list. (f) Click OK.

The tutorials library now appears in the Library Manager window.

SCHEMATIC CAPTURE

1. In Library Manager select the *tutorials* library, then: **File** > **New** > **Cell View**.

F	- Create New File							
OK Cancel Defaults He						Help		
Libi	Library Name tutorials							
Cel	l Name	9	ir	nverter				
Vie	View Name			schematič				
Тос	bl		Co	mposer-S	Schematic	:		
Lib	Library path file							
ic,	ic/analog/cadence/tutorials/cds.lib							

(a) Enter the Cell Name: *inverter*.(b) Click OK.

The Virtuoso® Schematic Editing window will open, and we can now create the CMOS inverter schematic.

2. Before we begin creating the CMOS inverter schematic review the following tips.

Hot-Key Action		Hot-Key	Action
i	Add Instance	q	Properties
m	Move	[or]	Zoom in or out
shift + x	Check and Save	р	Add Pin(s)
ctrl + e	Ascend from Symbol	shift + e	Descend into Symbol
W	Add Wire	f	Fit to Screen
с	Сору		

Virtuoso® Tips: Virtuoso® uses Hot-Keys. The following table will be useful.

Library Manager Tips: Only two libraries are of interest:

- 1) cmosp18 (contains MOSFETs)
- 2) analogLib (contains res, cap, vdd, vss, vdc, gnd, vpulse).
- First add an n-channel MOSFET to the schematic in Virtuoso®:
 (a) In Virtuoso® press "i" to open the Add Instance window.

-	Add Instance	
Hide	Cancel Defaults	Help
Library	I	Browse
Cell	Ĭ.	
View	symbol	
Names	¥ 	
Array	Rows 1. Columns	1
Rotat	e Sideways U	lpside Down

(b) Click the **Browse** button to open the Library Browser.

analoqLib	metal6 T	symbol
basic	mimcap	
cdsDefTechLib	nfet	
cmosp18	nfet3	

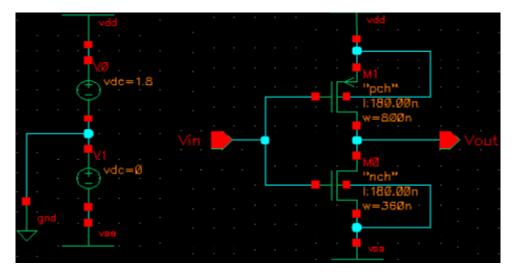
(c) Select the **cmosp18** library. A list of cells appears to the right.

(d) Select the **nfet** cell. A list of views appears to the right.

(e) Choose the **symbol** view.

(f) Go back to the Virtuoso® window, and place the n-channel MOSFET using your mouse.

Follow a similar procedure to create the *inverter* schematic shown below:



- NB: (a) To change the voltage of vdc and the length and width of the MOSFETs, click on the symbol and press "q".
 (b) The red elements that are labelled Vin and Vout are pins. To add a pin press "p", enter the pin name (Vin or Vout), click Hide, then place the pin using your mouse.
- 4. Check and save (shift + x) the *inverter* schematic.
- 5. Create a symbol from the schematic. In Virtuoso®: **Design > Create Cellview > From Cellview**.

ŀ	-		Cellvi	ew From Cellvie	w		
	ок	Cancel	Defaults Apply				Help
L	Library Cell Nan		tutorials]			Browse	
	From Vie	ew Name	schematic	To View Name Tool / Data Type	symbol <u>i</u> syn	nbol	
	Display Edit Opt	Cellview tions	■ ■				

- (a) Enter the Cell Name: *inverter*.
- (b) Click OK.

	Symbol Generation Options	
OK Cancel	Apply	Help
Library Name tutorials	Cell Name View Name invertex symbol	
Pin Specificatio	ns	Attributes
Left Pins	Vin <u>ľ</u>	List
Right Pins	Vout 1	List
Top Pins	Ĩ	List
Bottom Pins	1	List
Load/Save 🔄	Edit Attributes 🔄 Edit Labels 🔄 Edit Pro	perties 🔄

(c) Click OK.

The symbol appears in a new Virtuoso® window as shown below.

 t		[@inst(anc	eΝ	am	e]
Vin	[@partName]					



and close the *inverter* symbol Virtuoso® window.

Sanity Check: If you open Library Manager, and look in the *tutorials* library, you will find that the *inverter* cell contains a both *schematic* and a *symbol*.

So far so good!

6. Now we will create a new schematic that contains our *inverter* symbol:

In Library Manager: **File > New > Cell View**:

-	Create New File								
	ок	Cance	el	Defaults		Help			
Lib	Library Name tutorials								
Cel	l Name	e	inverter_sim]						
Vie	w Nan	ne	s	chematic					
Тос	Tool Composer-Schematic								
Library path file									
ic/analog/cadence/tutorials/cds.lib									

(a) Enter the Cell Name: *inverter_sim* (<u>double check that the Library Name is *tutorials*).(b) Click OK.</u>

7. Create the *inverter_sim* schematic as shown below:

								-103			
· · · 	Vin		inv	/er	ter		'out	: -			
									·	cø	
.m.(+) v1:20.20 v2=1.8 · · · tr=5m									1	- c=5	Øf :
gnd									•	gnd	
									1	7	

NB: (a) The symbol with the square wave beside it is the *vpulse* symbol (from the analogLib library). Change the properties of the *vpulse* symbol by clicking on its symbol and pressing "q":

0
1.8
5n
30n

(b) Change the value of the capacitor to 50f by clicking on its symbol and pressing "q".

8. Check and save (shift + x) the *inverter_sim* schematic.

We now simulate the *inverter_sim* schematic.

SIMULATION

1. Open Analog Design Environment from Virtuoso®: **Tools > Analog Environment**.

Cadence Status: Selecting outputs	Analog Design Environment (1)	•
	to be plotted T=27 C Simulator: spectre Variables Outputs Simulation Results Tools	S 5 Help
Design	Analyses	۲.
Library tutorials	# Type Arguments Enable	⊐ AC F TRAN
Cell inverter_sim	1 tran 0 20n yes	- DC
View schematic		x Y Z
Design Variables	Outputs	I ŧ∕
# Name Value	# Name/Signal/Expr Value Plot Save March	
	1 net6 yes allv no	
> Select on Schematic Outp	its to Be Plotted	\sim

2. Setup the Analog Design Environment:

Setup > Simulator/Directory/Host.

Choosing Simulator/Directory/Host Cadence® Analog								
ок	Cancel	Defaults	Help					
Simulator		spectr	eS 💷					
Project Directory		./simul	ation					
Host Mode		🖲 local	remote olistributed					
Host								
Remote ()inectory							

(a) Choose spectre (not spectreS, the figure above is incorrect) from the Simulator dropdown list.(b) Click OK.

Setup > Models Libraries...

(a) Set the path to: /CMC/kits/cmosp18/models/spectre/icfspectre.init(b) Click Add, then Click OK.

3. Perform a transient analysis. In Analog Design Environment:

Analysis > Choose.

😑 Choosing Analyses — Ca	idence® Analog Desig						
OK Cancel Defaults Apply	Help						
Analysis 🛈 tran 🔵 ac 🔵 s	ppdistospss						
⊖dc ⊖xf ⊖p	ss 🔵 noise						
Transient Analysis							
Stop Time 20n							
Accuracy Defaults (empreset)							
conservative moderate liberal							
Enabled	Options						

(a) Select **tran** (transient analysis).

(b) Enter the Stop Time: 100n. (not 20ns, the figure above is incorrect) This is the simulation duration.(b) Click OK.

Outputs > To be plotted > Select On Schematic. To plot the *inverter* input and output, in the *inverter_sim* schematic:

inverter_sim schematic:

- (a) first click the blue wire at the *inverter*'s input.
- (b) then click the blue wire at the *inverter*'s output.

(Be careful where you click.)

Session > Save State...

Sav	ving St	tate —— Cadence® Analog Design Env	ironr
ок	Cancel	Apply	Help
Save A	ls	may28v1]
Existin	g States		
What to) Save	🖬 Analyses 📰 Variables	
		■ Outputs ■ Model Path	
		Environment Options 📕 Simulator Option	IS
		📕 Convergence Setup 🛛 📕 Waveform Setup	
		📕 Graphical Stimuli 🔰 Conditions Setur	1
		Results Display Setup	

(a) In the Save As textbox enter the date followed by a version number as shown above.

Simulation > Run.

In a moment the Waveform Window will open as shown below. Note that the *inverter* has inverted the input square pulse.

