## Lab 1: Schematic and Layout of a NAND gate

In lab 1, our objective is to:

- Get familiar with Cadence environment.
- Draw a schematic of a simple NAND gate and simulate it.
- Draw layout of a NAND gate using cell library, design rule check (DRC), extract, layout versus schematic (LVS) and simulate using extracted version.
- Compare these two simulations.

In this lab procedure, detailed procedures and snapshots are given for the shake of understanding. You are supposed to understand the procedures, so that you can design your own circuit later. Following lab procedures will not be elaborated so much except where necessary. Please refer to this lab procedure in future labs as reference.

## Login procedure:

After logging in to the computer in the lab using your existing DoE accounts, double click on "**Sun Systems**" icon to start the remote login server on unix machine. Choose one of the computers from the list and click "**Connect**". If you are unable to connect, click "Setting" and make sure all the checkmarks are ON. Ask TAs for help if you still have trouble. When you are prompted to enter your username and password, ask TAs for username and initial password. Make sure you **change your password** as soon as you log in for the first time and **remember** it for the rest of the labs. *TAs might not be able to help you if you forget your password*.

## **PART A: Procedure**

1) Open a console and create a folder in your home directory for Lab 1 "ELEC4708/lab1"

>mkdir ELEC4708

>cd ELEC4708

>mkdir lab1

>cd lab1

2) **Start Cadence** environment with 0.18 technology file.

>startCds -t cmosp18 -b icfb

3) Close the "What's New" window. You will have "**icfb**" window open.

	icfb – Log: /home/bmorshed/C	051095/005109.04		133
File Tools Options CM	AC Gateway CMOSP18-Documentation		Help	
dsNewsClose()				
	Ť.			
<u>í</u>	Ť.			
ouse L	т. М:	R:		

4) The next thing we will do is make a library to hold all your work for lab1. Click on Tools -> Library Manager (or simply press F6). Library manager window should pop up.

Conversion Tool Box		
Library Manager 16		
Library Path Editor		
Verilog Integration		
VHDL Tool Box		
Synopsys Integration		
Router		
Constraint Manager		
Mixed Signal Environment		
Analog Environment		
VSdE		
Technology File Manager		
Display Resource Manager		
CDF		
AMS		
Camera		
SKILL Development	Log: /home/bmorshed/CDSlogs/CDS.log.64	
File Tools Options CMC Gateway	CMOSP18-Documentation Help	1
t ddsOperLibWanager() t		
빈		
I		
mouse L:		
A second seco		

5) Click File -> New -> Library. Type "lab1" in Name field. Click OK.

💻 🛛 Library Manager	: Directoryhome/bmorshed/TA/ELEC4708/lab1	
<u>File E</u> dit <u>Vi</u> ew <u>D</u> es	sign Manager	Help
Show Categories	Show Files	
Library —	Cell View	
Í		
CMCLayoutReference		
CMCpcells	— New Library	
CMCshare ahdlLib	Library	
analogLib	Name lab1	
artisan_io_30 artisan sc 30		
basic	Directory	
cdsDefTechLib cmosp18		
cmosp18_defin_techli		
lab1 package		
passiveLib		
rfExamples tpz973q		
virage_sram	10me/bmorshed/TA/ELEC4708/labl	
vst_n18_sc_tsm_c4		
	– Design Manager – – – – – – – – – – – – – – – – – – –	
	O Use NONE	
	O Use No DM	
	OK Apply Cancel Help	
– Messages		
	morshed/TA/ELEC4708/lab1/libManager.log.182".	- IA
<u>,</u>		

6) Select "Attach to an existing techfile". Click OK.



7) Select "**cmosp18**" from the drop down menu of the Technology Library. Click OK.

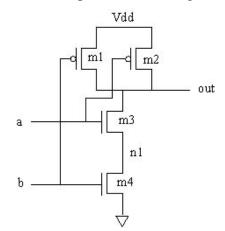
ок	Cancel	Defaults	Ap CMCpcells	Help
	iyu Librar gy Library		analogLib basic cdsDefTechLib cmosp18	
			cmosp18_defin_techlib package passiveLib rfExamples tpz973g virage_sram	

 Highlight the new library you have just created, i.e. "lab1", and click File -> New -> Cell View. Type "NAND" in Cell Name. Keep the default values in other fields as shown. Click OK.

🛏 Create New File					
ОК	Cancel	Defaults	Help		
Library I	lame	lab1			
Cell Name					
View Na	me	schematič			
Tool	C	Composer-Schen	natic		
Library p	ath file				
.e/bmors	hed/TA/I	ELEC4708/lab1/	cds lib		

9) A blank **Virtuoso Schemetic Editing** window will open. Move your cursor through the icons on the left side and pop-up descriptions for each will show up. The next thing we

will do is draw the NAND gate using pfet and nfet. We will also add 2 input pins, 1 output pin, 1 VDD pin and 1 GND pin. A circuit diagram of NAND gate is given here.



- 10) To **add an instance** in your schematic, you can click on Instance icon, or click on Add -> Instance, or simply type "**i**" from the keyboard. Add Instance dialog box will show up.
- 11) Click Browse beside Library. Library Manager will pop up.
- 12) In Library, select "**cmosp18**", in cell select "**pfet**", in view select "**symbol**". Then click Close.

-		Add Instance			
Hide	Cancel D	efaults	Help		
Library	cmosp18		Browse		
Cell	pfeť			ary Browser – Add Insta	nce .
View	symbol		Show Categories	any provision - Add Insta	
Names	I.		Library —		- View
Array	Roy	ws 1 Colum	ns 1 cmosp18	pfet	symbol
Rotat	e	Sideways	CMCLayoutReference CMCpcells CMCshare ahdLLib analogLib	netal2_T netal3_T netal4_T netal5_T netal6_T	auLvs hspiceS spectre spectreS symbol
lx Use Cell	I	CMCpcells spcp:	nos artisan_io_30 artisan_sc_30	mimcap nfet	
Multiplier			basic cdsDefTechLib	nfet3 nfet3 na	
Width		500. On M	cmosp18	nfet_na	
Length		180.00n M	cmosp18_defin_techlib lab1	nfet_rf nlpqlobals	
Drain difi	fusion area	0.48u*iPar("w"	passiveLib	nmoscap pcapacitor	
		0 40	rfExamples tpz973q	pfet pfet3	
			viragejsram vst_n18_sc_tsm_c4	pfet_rf pnp poly1_T resistor sample_rwell	
			Close	Filters	Help

13) Click Hide and place the instance in your design. You can place multiple instances of same item. When you are done with placing all the instances of that item, press Esc to get rid of it.

14) You can access the **object property** window by selecting a pfet and pressing "**q**". Edit Object Properties window will pop up and will show properties of that instance. You can change length or width if required. For now, we will use defaults widths for pfets and nfets. However, you can change widths later to enhance performance. Note that the default length value is set to the minimum allowed by the technology.

-			Edi	t Obji	ect Properti	es	
ок	Cancel	Apply	Default	s Prev	ious Next		Help
Apply Show	То		urrent stem 🗖		ance CDF		
	8	rowse	Re	set Inst	ance Labels Di	splay	
Property Value				Display			
	Library Name cmosp18				off		
	Cell Na	ne:	pfet	L.,			off
	View N	ame	synd	olį			off
Instance Name MI					off		
				Add	Delete	Modify	
	CDF Pa	ramete	•		Value		Display
IxUse	Cell			CMCpce	lls spcpmoš		off
Multip	lier			1			off 🛁
Width				500n M			off
Lengti	h (			180.00n M		off	
Drain	diffusion a	urea		0.48u*iPar("w") <u> </u>		off	
Sourc	e diffusior	ı area		0.48u*iPar("w")		off	
Drain	diffusion	peripher	у	0.96u+)	2*iPar("w") M	E.	off
Sourc	e diffusior	n periph	ery	0.96u+	2*iPar("w") M		off 🛁
Drain	diffusion r	es squa	ares	0.27u/:	iPar ("w") <u> </u>		off
Sourc	e diffusior	ı res sq	uares	0.27u/	iPar("w")ľ		off
Drain	diffusion	ength	ĺ				off
	e diffusior						off

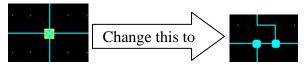
- 15) Repeat steps (10) to (14) for "**nfet**"s from the same library and place them in your design.
- 16) To **add pins**, click on Pin from tools button on the left side, or click Add -> Pin, or simply type "**p**" from the keyboard. Add Pin dialog box will open.
- 17) Type the names of the pins you want to add in sequence, leaving a blank space in between. DO NOT click Hide button. You will need to change Direction property for some of the pins.

		Add Pin		
Hide Cance	el Defaults			Help
Pin Names	A B C VDD GND	[		
Direction	input 📟	Bus Expansio	n 🔵 off	) on
Usage	schematic	Placement	🕒 single	e multiple
Attach Net Expr	ession: 🔵 No	).Yes		
Property Name	L			
Default Net Ham	e I			
Fout Height	0 0685	Fout Style	stick	
Justification	lowerCenter 🗆	Entry Style	fixed off:	set
Rotate	ş	lideways		Upside Down

- 18) Click on the schematic where you want to place pin A. For pin A and B, direction should be **input**, for pin C, direction should be **output**, for pin VDD and GND, direction should be **inputOutput**. Now click "Cancel" of the Add Pin dialog box.
- 19) To **add wires**, click on Wire (narrow) from tools button on the left side, or click Add -> Wire (narrow), or simply type "**w**" from the keyboard. Add Wire dialog box will open.

-			Add Wir	e	
Hide	Cancel	Defaults			Help
Draw M Width	lode [	route -		Route Method	full
Color		cyan			
Line Sty	/le [	solid			

- 20) Click hide, and connect the pins with FETs properly to form a NAND gate. Notice that you have to connect the substrate of pfets and nfets properly.
- 21) Click on "**Check and Save**", the first icon on the left hand side toolbar, or click Design > Check and Save, or simply type "X" from the keyboard to save the file.
- 22) You might get errors and warnings at this point. Note that Virtuoso will produce warning if a junction has more than 3 wires connected. To avoid this, you can reroute one wire to form 2 junctions instead of 1.



23) Get rid of all errors and warnings. Ask TAs if you need help. Next thing we will do is create a symbol for our NAND gate.

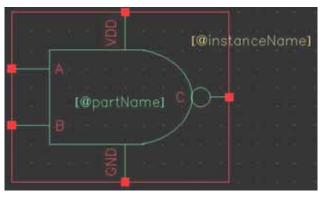
- 24) Click **Design -> Create Cellview -> From Cellview**. "Cellview From Cellview" dialog box will open.
- 25) Keep the default values as shown and click OK.

OK Cancel De	efaults Apply			He
jbrary Name	lab1]			Browse
Cell Name	NANDĚ			
From View Name	schematic	To View Name	symbolį	
		Tool / Data Type	symt	ool —
Display Cellview				
Edit Options				

26) Symbol Generation Options dialog box will open. Cut and paste GND from Top Pins field to Bottom Pins field. The dialog box should look like this. Click OK.

Cell Name	View Name
NANDĚ	symboli
	Attributes
	List
	List
	List
	List

27) Virtuoso Symbol Editing window will open. You can keep the default rectangular shape or change it to your own as you like. To change shape, select and delete rectangular box, click on Line icon in the left side toolbox, select shape and line width as necessary and draw a new shape. You can also move the pins around if you like. This step is optional.



- 28) Save and close the symbol window and schematic window. You will also need to create an **inverter schematic and symbol** using the same procedure (step 8 to 27). Name your inverter as **inv0**. Next we will make a testbench to simulate our NAND gate.
- 29) With "icfb" window highlighted, press F6 to open the library manager. Select lab1 from the library field. Click File -> New -> Cell view. In the Create New File dialog box, enter **testNAND** in the Cell Name field. Keep other default values. Click OK.



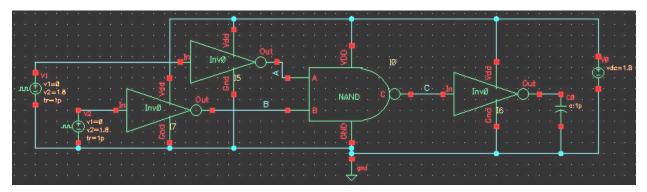
30) A blank Virtuoso Schematic Editing window will open. Press "i" to Add Instance. Click Browse. From Library Browser, select lab1 from Library, NAND from Cell, symbol from view. (From now on, this selection sequence will be given as: lab1 -> NAND -> symbol) Click Close. Place the NAND gate in your blank Virtuoso window. Press Esc.

	Add Instance			
Hide	Cancel Defaults	Help		
Library	Inbl	Browse		
Cell	KANTÎ	Hibran	Browser - Add Ins	tance
View	syaboli	Show Categories	The sectors and a sector store	
Names		Library	Cell	Wew.
Amay	Rows 4 Columns	lah1 CHCLayoutReference	NAND	[sysbol
Rotal	e Sidoways U	CHCpcells CHCshare whdLibb analogLib artisan_so_30 artisan_so_30 basic odsDefTechLib tmospl0_defin_techLib HDD package passiveLib cfDramples tpr973p virage_sism vet_n18_sc_tsm_c4	testhand	
		Oose	Filterst.	Help

31) Press "i" again to Add Instance. Click Browse. From Library Browser, select **analogLib** - > vdc -> symbol. Click Close. Place the "vdc" in your design. Press Esc.

		Ad	d Instance			
Hide	Cancel	Defaults		Help		
Library	analogL	iĥ		Browse		
Cell	vde					ng antonin sing sing sing sing sing sing sing s
View	symboli			Library	Browser – Ad	d Instance
Names				🔲 Show Categories	Cell	View
Array		Rows 1	Columns	janalogLib	vdc	jsymbol
Rotat AC magn AC phase DC volta Noise file Number (	itude 9 ge		eways Ups	CMCLayoutReference CMCpcells CMCshare ahdlLib antisan_io_30 artisan_sc_30 basic cdsDefTechLib cmosp18 cmosp18_defin_techlib lab1 package passiveLib rfExamples tp2973g virage_sram vst_n18_sc_tsm_c4	tline ulwire u2wire u5wire u5wire usernpn vam vcc vcca vcca vcca vccs vccs vccs vccs	A ams auCdl auLvs cdsSpice hspiceD hspiceS spectre spectreS symbol
				Close	Filters	Help

- 32) Press "i" again to Add Instance. Click Browse. From Library Browser, select **analogLib** > **gnd** -> **symbol**. Click Close. Place the "gnd" in your design. Press Esc.
- 33) Press "i" again to Add Instance. Click Browse. From Library Browser, select analogLib > vpulse -> symbol. They will serve as input signals for input A and B (through inverters). Click Close. Place 2 (two) "vpulse" in your design. Press Esc. Note that to complete this lab, you might need to use vpwl as input signals. Ask TAs if you need help.
- 34) Press "i" again to Add Instance. Click Browse. From Library Browser, select analogLib > cap -> symbol. This will serve as output load for output inverter. Click Close. Place the "cap" in your design. Press Esc.
- 35) Press "i" again to Add Instance. Click Browse. From Library Browser, select **lab1** -> **Inv0** -> **symbol**. Place three inverters in you design. Two of the inverters between vpulses (or vpwl) and input A and B. Third inverter between C and output capacitor.
- 36) Select wire (narrow) and connect all the symbols properly. If you need to **pan around** use the **cursor arrow keys**. Use "[" or "]" to **zoom in or out** by factor of 2. Use "**z**" to **zoom to window**. Your design should be similar to this one.



37) Now we will setup the properties of some instances. Select V0 (vdc) and press "q" to access the Edit Object Properties dialog box. In the DC Voltage field, enter **1.8 V** and click OK. Note that standard supply voltage for CMOS 0.18u technology is 1.8V.

ок	Cancel Apply	Defaults Prev	vious Next		H
Apply Show	10	current ins vstem 🗖 user			
	Browse	Reset Ins	tance Labels	Display	
	Property		Value		Display
	Library Name	analogLib			off
	Cell Name	vdoj			off
	View Name	symboli			off
	Instance Nam	e <sup>VQ</sup>			off
		Add	Delete	Modify	
	User Property	Master \	√alue	Local Value	Display
	lvsignore	TRUE			off
	CDF Paramete	r	Value		Display
AC ma	gnitude	1			off
AC pha	ase	<u> </u>			off
DC vo	Itage	1, 8			off 🚐
Noise	file name				off
Numbe	er of noise/freq p	oairs 🧕			off.
XF ma	gnitude	Ĭ.			off
PAC m	agnitude	I.			off
PAC pl	nase	Ĩ			off 🛁
Tempe	erature coefficie	nt 1			off
		nt 2 🗓			

38) Select V1 (vpulse) and press "q" to access the Edit Object Properties dialog box. In the Voltage 2 field, enter 1.8 V; Rise time field, enter 1p; Fall time field, enter 1p; Pulse width field, enter 1n; Period field, enter 2n and click OK. If you are using vpwl, you should enter time-voltage pairs for your desired pulse. *Note that you can also enter a variable or expression in any field and later assign a value to that variable during simulation. For example, you can enter "T" in Period field, "T/2" in Pulse width. Then during simulation, specify T=2n. Variables give you flexibility of sweeping values.* 

	Edit Obje	ect Propert	ies	
OK Cancel Apply [	)efaults Prev	ious Next		Help
арруто	rrent inst tem ∎ user	ance CDF		
Browse	Reset Inst	ance Labels Di	isplay	
Property		Value	11	Display
Library Name	analogLib			off
Cell Name	vpulse			off
View Name	symboli			off
Instance Name	Vĺ			off
	Add	Delete	Modify	
User Property	Master V	'alue L	ocal Value	Display
lvsignore	TRUE	J.		off
CDF Parameter		Value		Display
AC magnitude	1			off
AC phase	1			off
DC voltage	l			off
Voltage 1	0 V			off
Voltage 2	1.8 V <u>í</u>			off
Delay time	Į.			off
Rise time	1p s			off
Fall time	1p s			off
Pulse width	1n s			off
Period	2n s			off
Frequency name for 1/pe	riod İ			off
Noise file name				off
Number of noise/freq pai	rs 🤨			off

- 39) Select V2 (vpulse) and press "q" to access the Edit Object Properties dialog box. In the Voltage 2 field, enter 1.8 V; Rise time field, enter 1p; Fall time field, enter 1p; Pulse width field, enter 2n; Period field, enter 4n and click OK.
- 40) Click the icon name "Wire Name" on the left had side toolbar, or click Add -> Wire Name, or press "l" (small L). The Add Wire Name dialog box will open. Type "A B C" in the Names field. Click Hide. Click the wire connected to input pin A of the NAND gate first, then the wire connected to input pin B, then the wire connected to output pin C.

-		Ad	d Wire Name		
Hide	Cancel	Defaults			Help
Wire N	ame No	et Expression			
Names		ав ф			
Font He	ight	0.0625	Bus Expansion	O off	on
Font St	yle	stick	Placement	🔘 single	multiple
Justifica	ation l	owerCenter	Purpose	🔿 label	alias
Entry S	tyle fi	xed offset	Show O	offset Defa	ults
Rota	ite				

- 41) Look at the picture in step (36). Each component of your design should show similar values.
- 42) To simulate our design, we will use Virtuoso Analog Design Environment. Click Tools
   -> Analog Environment to access the dialog box for Virtuoso Analog Design Environment.

Status: Ready	T=27 C Simulator: spectr	122211
session setup Analy	ses Variables Outputs Simulation Results Tools	He
Design	Analyses	-1
Library lab1 Cell testNAND	# Type Arguments	
View schematic		1
Design Variables	Outputs	D
# Name Value	# Name/Signal/Expr Value Plot Save March	3
		1
	Plotting mode: Replace	1

43) Click Setup -> Model Libraries. In the Model Library Setup window, Enter this line to Model Library File: /CMC/kits/cmosp18.5.2/models/spectre/icfspectre.init. Click OK. Then click Add to add this model in your dialog box. Click OK.

/CMC/kits/cmosp18.5.2/models/spectre/icfspectre.init	ок	Cancel	Defaults	Apply		H
Addel Library File	#Disa)	ole Model	Library F	ile	Sectio	on Enable
lodel Library File Section (opt.)	/CM	C/kits/cmo	sp18.5.2/1	models/spectre/icfspect	re init	Disable
Nodel Library File Section (opt.)						
						եր
	lodel	library File				Down

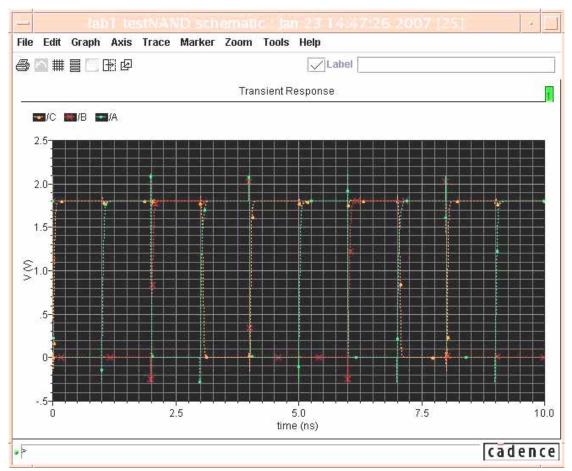
44) Click **Analysis -> Choose**. Choosing Analysis dialog box will show up. Select Analysis type as **tran** and enter **10n** in Stop Time field and select **Enabled**. Click OK.

	and the second second			
Analysis	🔘 tran	dc	lac	noise
	xf	sens	dcmatch	sth
	pz	sp.	envip	pss
	i pac	pnoise	pxf	
	psp	opss	qpac	
	qpnoise	qpxf	qpsp	
	Tran	sient Analys	sis	
Stop Time	104			
	Defaults (empr ervative mo		beral	

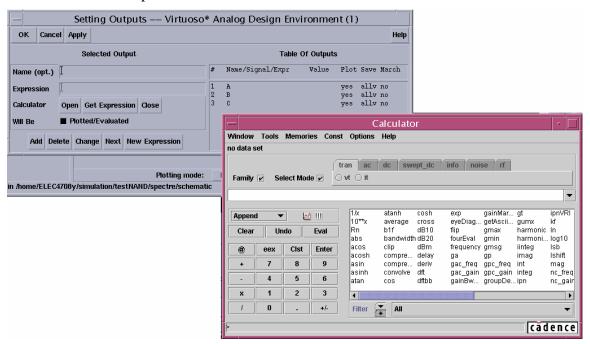
45) Click **Outputs -> To Be Plotted -> Select On Schematic**. Click the input wire connected to **A**, the input wire connected to **B** and the output wire connected to **C**. Now your Virtuoso Analog Design Environment should look like this.

Status: Ready	T=27 C Simulator: spectre	e 14
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	-Ę
Library lab1	# Type Arguments. Enable	a AC # TRA
<b>Cell</b> testNAND <b>View</b> schematic	1 tran 0 10n yes	
Design Variables	Outputs	E
# Name Value	# Name/Signal/Expr Value Plot Save March 1 A yes allv no 2 B yes allv no 3 C yes allv no	/
	Plotting mode: Replace	

46) Click **Simulation -> Netlist and Run**, or the traffic light with green light on the right side. Turn off "Welcome to Spectre". A message box should open and show log file. If the simulation is successful, input/output waveforms will be shown in a new window.



- 47) To see the waveform separated, select axis -> strip. To find delay manually, look at transistors of output and the corresponding input causing the transition. For example, in this case, the first output transition is at around 3 ns. To find exact value at 50% output (0.9 V), you can use Trace marker (under Marker -> Place) and click at 0.9 V crossing of the waveform. It is show the corresponding time. If you placed marker at wrong place, select that marker and click Edit -> delete.
- 48) To find delays automatically, you can use the built in Calculator. In the analog design environment, select Output -> Setup. Click Open beside Calculator. The calculator window should open as shown:

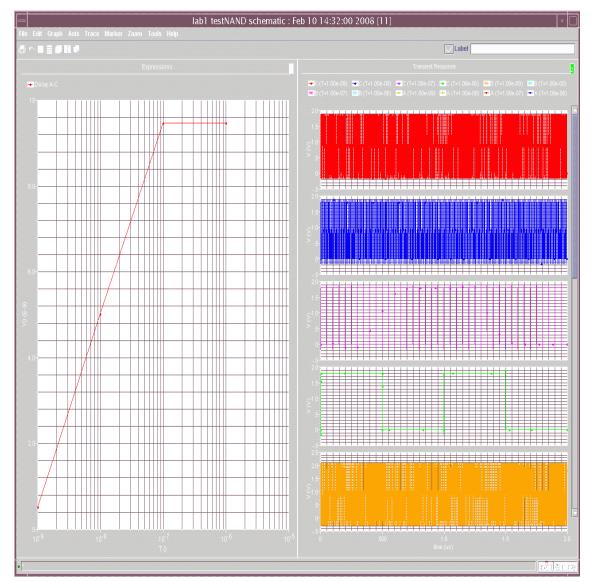


- 49) Click on delay. Place the cursor in Signal 1 box and click Vt. The schematic window should pop up. Select wire A. Place the cursor in Signal 2 box and click Vt again. The schematic window should pop up again (if it does not, select it and then select Vt again). Select wire C. Change Threshold value 1 to 0.9 (do not use the numpad buttons). You can choose edge type (rising, falling, either), so that any combinations of rising and falling can be found. Click the button >>>. Change Threshold value 2 to 0.9. Click Apply. Go to Setting Outputs window and click Get Expression. Give a Name of the expression. Click Add and then OK. Click Simulation -> Netlist and Run. The value will be shown in this window.
- 50) If you need to use a variable (for sweeping etc.), then you can do it using the schematic window. It is a good practice to assign variables to the parameters that you might want to change/sweep. Give meaningful names variables so that you can recognize them later. For example, if you want to sweep the frequency of excitation, select any of the Vpulse, and press "q" to open the properties window. Enter Period = T and Pulse Width = T/2. Do the same for the other Vpulse, too.
- 51) From the Analog Design Environment, click Variables -> Copy from Cellview. T should appear under Design Variable list.

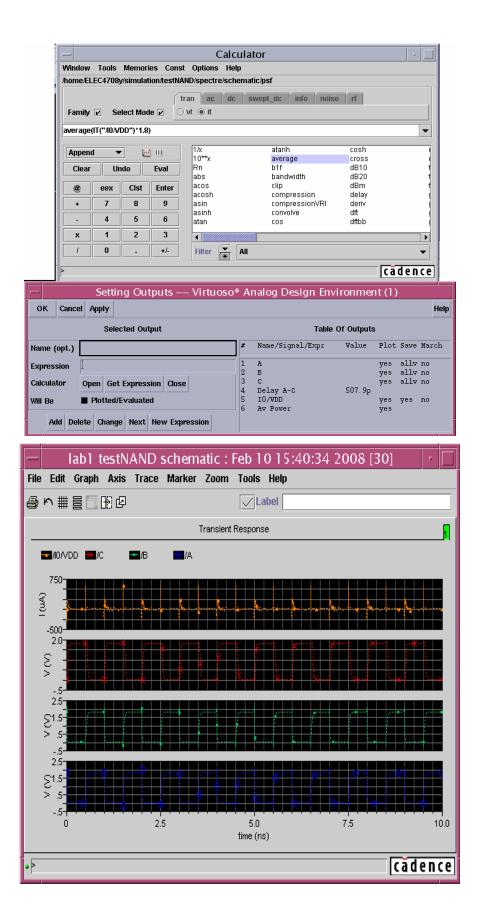
— Virtuoso	Analog Design Environment (1)	•
Status: Ready	T=27 C Simulator: spectr	e 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	÷Ę
Library lab1	# Type Arguments Enable	⊐ AC = TRAN ⊐ DC
Cell testNAND View schematic	1 tran 0 10n yes	JDC III X Y Z
Design Variables	Outputs	l∎″,
# Name Value	# Name/Signal/Expr Value Plot Save March	
	1         A         yes         allv no           2         B         yes         allv no           3         C         yes         allv no           4         Delay A-C         1.008n	101 101
> Results in <i>I</i> home/ELEC4708	Plotting mode: Replace	

52) For Parametric Analysis, click Tools -> Parametric Analysis. To collect the variable name, click Setup -> Pick Name for variable -> Sweep 1. Click on T and select OK. In the range field, choose From = 1n and To = 1u. Select Step Control as Logarithmic and Total Steps = 4. In the Analog Design Environment window, click the Transient response setup and change Stop Time to 2u. In the Parametric Analysis window, click Analysis -> Start. It will take some time to simulate, as the Stop Time of Transient response is quite high (!). However, if you would like to find effect any pulse, you need to have at least one complete cycle. When the simulation finishes, look at the waveform window. Highlight Transient Response part and click the Strip button (Axis -> Strip). Try to interpret the waveform window.

- Virtuos	o® Analog Design Environment (1)	•
Status: Ready	T=27 C Simulator: spectre	. 4
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	-K
Library lab1	# Type Arguments Enable	I AC F TRAN
Cell testNAND View schematic	1 tran 0 2u yes	
Design Variables	Outputs	
<pre># Name Value 1 T 1</pre>	# Name/Signal/Expr     Value     Plot Save March       1     A     yes allv no	<i>#</i>
	2 B yes allv no 3 C yes allv no 4 Delay A-C wave yes	畿
	· · · · · · · · · · · · · · · · · · ·	<b>₩</b>
> Results in /home/ELEC470	Plotting mode: Replace	
Parametric Ar	alysis – spectre(0): lab1 testNAND schematic	] : //
Tool Sweep Setup Analysis		Help 6
Sweep 1	Variable Name T	ation
Range Type From/To	From 11 <u>1</u> To 11 <u>1</u>	
Step Control	Total Steps 4	Select _



53) To find average power, click on Outputs -> To be plotted -> Select on Schematic. Then click on VDD node of the NAND gate (to save the transient current). Now, open the Calculator from Setting Output window. Click *it* and then VDD node of the NAND gate. Multiply the expression with 1.8, which is the supply voltage here (numpad does not work). Then click "Average". Click Get Expression in the Setting Outputs window and give a name (eg. Av Power). Click Add, and then OK. In the Analog Design Environment window, click the variable T and change its value to 1n. Click Transient simulation setup and change Stop time to 10 ns. Click Simulation -> Netlist and Run. Average power consumption will be shown in Analog Design Environment window. You can also sweep the frequency and plot the average power using the Parametric Analysis procedure described before.



Status: Ready		T=27 C Simulator: spec	tre
Session Setup Analyses	s Variables Outputs Simulat	tion Results Tools	Help
Design	A	nalyses	۲,
L <b>ibrary</b> lab1	# Type Argumen	ts Enable	TRP
Cell testNAND	1 tran 0 :	10n yes	1 DC
<b>view</b> schematic			I X Y
Design Variables	C	Dutputs	ŀ
≭ Name Value	# Name/Signal/Expr	Value Plot Save March	
l T 1n	2 B	yes allv no	
	3 C	yes allv no	10
	4 Delay A-C 5 IO/VDD	507.9p	10
	6 Av Power	yes yes no 16.22u	1000
	D	otting mode: Replace 💴	1

## Helpful hints for the lab 1 part A:

- If you want to find out what is inside a symbol, select it and press "E" or "e" to **descend** (first one lets you edit, second one only to view as read only). Press "^e" when you are done to **ascend** to the top level.
- For delay measurement, use 2 vpwl as sources and a capacitor as load (lab 1 part A, Q1 & 2). Set the timing of the vpwl's properly to get the given waveform at A and B. For lab 1 part A Q3 & 4, use one "vpulse" and connect both inputs with it.
- You can save your settings of Analog Design Environment by selecting Session -> Save State. To load a previously saved state, select Session -> Load State.
- Anytime you make any change in schematic window, save it before you simulate it.
- To find delay, you need at least 2 inverter symbols, eg. Inv0 and Inv1, as the inverters in the source and drain are different size. However, a more efficient way to do this is to use the keyword **pPar("varname")**, where varname is the name that will be shown in symbol attributes, accessed in higher level. For example, in your inverter schematic, set value for nMOS and pMOS width as pPar("InvWidth"). Then create the symbol. Now, if you place this inverter symbol in another higher level schematic, when you select the inverter and view it's property (by pressing 'q'), you should notice a new variable at the bottom named InvWidth. You can assign different value for same symbol in different instances.
- Delay measurements should be done in transient analysis mode. Make sure A and B matches given wave shape (not X and Y). Look closely to your graph for rise times and fall times (You should be able to find all cases as required from this plot) and tabulate them. Find propagation delay (maximum) and contamination delay (minimum) from the table.
- Use variables for defining widths of MOSFETs, so that you can change them from during simulations keeping  $\beta$  constant.