# CPE/EE 427, CPE 527, VLSI Design I: Tutorial #2, Schematic Capture, DC Analysis, Transient Analysis (Inverter, NAND2)

Joel Wilder, Aleksandar Milenkovic, ECE Dept., The University of Alabama in Huntsville

Adapted from San Jose State University, Dept. of Electrical Engineering Author: David W. Parent

### 1. INTRODUCTION

In this tutorial exercise you will learn how to use Cadence's schematic capture tool to create an inverter and then simulate it. You will build the inverter in the schematic tool at the circuit level and will construct a symbol in which to package your inverter design. Next, you will instantiate your inverter symbol into a "test workbench" file, where you will do a spice simulation using Spectre to analyze the transient and DC characteristics of your design. Subsequently, you will do a post-layout simulation that will use the inverter layout from tutorial 1, where the parasitic capacitances are extracted and incorporated into the simulation. Finally, using this workflow as a guide, you will create a two-input static NAND gate on your own and simulate it using Spectre.

You will base your MOSFETs on the TSMC 0.25um 5-metal 2.5V process that was used for Tutorial 1 (MOSIS SCN5M\_DEEP).

### 2. CADENCE STARTUP

From your home directory, change directories into your cadence working directory:

\$ cd cadence

\$ cd lab1\_2

Start the cadence tool:

\$ icfb&

### 3. CREATE DESIGN WORK AREA

In the Library Manager window, select TestLib, the library you created in tutorial 1.

Select the MyINV cell and add a new cell view (as was done in tutorial 1). Ensure the name of the new cell view is MyINV and make sure you select "**Composer-Schematic**" in the Tool pulldown window. Click OK, and the Schematic tool will open, as shown in Figure 1.

-							1	Virl	uo	so	• •	Sche	ema	atic	: Ec	liti	ng	: Ia	b2	IN	V s	ch	em	atio								an a	
Cm	d:		Sel	I: O																													2
Tools	Design	Win	low	Edi	it A	dd	Che	ck	She	et	Op	tions	N	su																		He	Iр
EX.											-		1				1	i.						35 ES 17 ES	- i					8			
	35 9.5 - 3																																
~	38 (A - )																																
€ <sup>2</sup>	343 G¥ - 3																																
~ 2																																	
R																																	
NUN Y	a: 21 - 1																																
12°	$\approx -\infty = 0$																																
	40.52 1																																
120																																	
4																																	
$\cap$	(E) (A = )																																
	46 52 - 5																																
期時	****																																
47k																																	
1	÷ 4 )																																
_	40.52																																
12	38) (r. 1																																
abc																																	
-	- 36 - 36 - 36																																
•D	40.52																																
	30 A.																																
	180 (184 - 19 187 (19)																																31
0	Noune I			iola	Tof	-								abV	Mer	10.07	onI	ο ()						R	ach	7.00	m F d	+71	0.0	.03		1	
\_≦	> wouse L	. sn	ower	LUK.	Inte	50						I.	. 3	stud:	unot	ser	opu	PU						n:	ser	1200	utr 1	(1	. 0 0	.9)			
	82																																

Figure 1. Schematic Capture Tool.

# 4. CREATE INVERTER CIRCUIT

The schematic capture tool will be used to draw the schematic representation of your inverter.

To draw the inverter you need to add a pmos4 transistor, nmos4 transistor, ground connection, power supply connection, input pin, output pin, and wire it together.

To add items to your schematic, go to **Add->Instance** in the composer tool, or press the letter 'i' (or select the icon pictured on the left vertical toolbar). The add instance pop-up appears like in Figure 2. Click on browse to graphically get components, where you will see the window as shown in Figure 3.

-		Add Inst	ance	
Hide	Cancel	Defaults		Help
Library Cell	L			Browse
View Names	symbol <u>i</u>			
Annay		Rows 1	Columns	1
Rotat	te	Sideways	1 1	lpside Down

Figure 2. Adding an instance.

— Com	ponent Brov	wser 🕛 🔲
Commands		Help 5
Library	NCSU_Analog	Parts
Flatten		
Filter	*	
Uncate CONTEN Curren Diodes H_Spic Microw Miscp N_Tran P_Tran P_Tran R_L_C	gorized TS t_Sources e_Only ave_Parts arts sistors sistors tic_Devices	

Figure 3. Browsing for components.

To get the parts, make sure the library is set to NCSU\_Analog\_Parts.

In the component browser window, click on supply nets and select vdd. (The items in supply nets are actually global signals, which are automatically given pins. This makes symbols cleaner because only logic ports are designated.)

Stamp down the vdd instance in your schematic like in Figure 4.

-									١	/ir	tu	05(	)®	S	:h	em	ati	iε	Ed	itiı	ng	1:1	ab	21	N١	/ s	ich	ıer	na	tic								•	
	Cmd	:				S	el:	0																															2
To	ols	De	sigr	1 1	Min	dov	,	Edit	P	dd	a	hec	k	She	et	OF	otio	ns	N	csu	ı																	Help	i
F	Y	121		1				20 					ł.			15								12								38) 79					ii.		
	1																																						
8	3																																						
N																																							
œ	2																																						
	~																																						
e	2																																						l
	~																																						
$\mathbf{D}$	0																																						
J.	44.5																																						
Ð	*																																						
	11																																						
Ì																																							ĺ
6	1																																						
a																																							
書																																							
15	17																																						
T	1																																						
	N.																																						
ſ	*																																						
		4		2				1.7									a	1.		-										-				-	1.53			12	
J		mou	se	L:	sh	.owC	11	ckI	nte	0						M	: 3	chł	iıM	ous	eP	opU	р()						K:	sci	oH1	Cre	ate	) In:	st(	)			
ſ		mou >	se	L:	sh	.owC	li	ckI	nf	0						M	: 3	chł	liM	ous	eP	opU	р()						R:	scl	oHi	Cre	ate	In	st(	)			

Figure 4. Stamping down vdd.

Get the gnd symbol and stamp it down in the same manner as you did for vdd.

To add the transistors, click on N\_Transistors for the nmos4 and P\_Transistors for the pmos4. Add an nmos4 and a pmos4 transistor in a similar manner. The pop-up for the nmos4 should look like Figure 5, and the pmos4 pop-up should look like Figure 6.

Make the W/L ratio for your nmos4  $4\lambda/2\lambda$ , and make the W/L ratio for your pmos4  $8\lambda/2\lambda$ . (NOTE: If your transistor does not have the widths and lengths like in Figure 5, you did not attach a tech library to the library you created for tutorial 2 and you will need to start over.)

Stamp them down like Figure 7. It does not have to be exact but neatness will make the design easier to work with. (If you make a mistake and need to get out of add instance mode, press the esc key. Click on the object you want to delete and press the del key.)

-		Ad	d Instance	
Hide	Cancel	Defaults		Help
Library	NCSU_Ana	log_Part	8	Browse
Cell	nnos4			
View	symbol			
Names	Ľ		Ī	
Anray	F	lows 1	Columns	1
Rotat	0	Side	awaye 1	Incida Down
notat	6		eways (	pside Down
			town0Edy	
Model na	11)G		CSMC250N	
Model Ty	pe		System Use	<u>بر</u>
Multiplier			<u>4</u>	
Fingers			1	
Width (gr	id units)		8	
Width			480.0n M	
Willih (m	inimum)		360.0n M	
Length ((	grid units)		4	
Length			240.0n M	
Length (r	ninimen)		240.0n M	
Drain diff	fusion area		2.88e-13	
Source d	iffusion are	ea	2.88e-13	
Drain diff	fusion perir	neter	2.16u M	
Source d	iffusion pe	rimeter	2.16u M	
Drain diff	usion res :	squares	I	
Source d	iffusion res	s squares	Ĩ	
				1.1

Figure 5. Getting an nmos4 device.

-			Add	Instance	2	
Hide	Cancel	Defaults				ł
Library	NCSU_Ana	log_Parts	λ.		Browse	
Cell	pmos4					
View	symbol					
Names	Ľ					
Annay	R	ows 1		Columns	1 <u>ľ</u>	
Rotat	te	Side	ways	î	lpside Down	
Multipliei Fingers Width (gi Width	rid units)		4 1 16 960.0	n Mj		
Willh (n	wimon)		360.0	n M		
Length (	grid units)		4			
Length			240.0	n M		
Longth (	minimum)		240.0	n M		
Drain dif	fusion area		5.76e	-13		
Source d	liffusion are	a	5.76e	-13		
Drain dif	fusion perin	neter	3.120	щ		

Figure 6. Getting a pmos4 device.



Figure 7. Major symbols of the CMOS inverter.

All that remains to be completed is to wire the connections together and add input and output pins. To add a wire, press w. The wire will snap to place at the proper ports of each device by pressing the 's' key near that node.

Wire it up like Figure 8.

									λ	/ir	tu	osc	)®	Sch	ie	ma	tic	Ec	liti	ng	j: I	ab	2	IN'	V s	sch	ıer	na	tic								• [	
	Cmd	Ŀ				S	el:	0																														2
То	ols	De	esig	n	Win	idow	1	Edit	t A	<b>\</b> dd	C	heck	¢	Shee	t	Opti	ons	N	csi	J.																	Hel	p
	r			ł.			i.	3 					i.					į.			l		33 	÷.							35 							
	1																																					
R.	3																																					
	-																																					
Œ	4																				1																	
	2																					101		÷.														
Q	5																				Ĩ																	
N.L.	1																																					
P P	2															ie g	1.04	÷.	- (4) <mark>-</mark>	الم	4	terrie	254	2	14													
	41															82				Ч	4	W=1	96Ø	. бп 8-														
173	ð															88					1	m:1																
1	1																																					
Ĭ																														÷,	•							i I
6	1															5) 16 - 5			-			tenic W=4	2581 48Ø	۹ .Øn														
â	•																				7	1=2	40.	Øn														
北部																						ma																
131	1																				1																	ł.
H																																						
	*																																					
J	x																																					-
		4_		-			12	-1.7		- 0											00	<b>T</b> 1	0				_	-							0		1	l.
Ĵ		mou	ise	<u>Б</u> :	sh	10WC	11	CKI	.nr	0()						M :	sch	util	10U:	seP	opt	ub ('	)					K:	sci	nHi	sat	rette	sTT.	vie	w()			
		<u>ੈ</u>																																				

Figure 8. Wiring a CMOS inverter.

To add the input and output pins, go to **Add->pins** and a pop up like Figure 9 should appear. Fill it out exactly like Figure 9. The pin names must match the pin names of the symbol view you are going to create later. If they do not match (direction or name), the design, check & save routine will fail.

-			Add Pin	
Hide	Cancel	Defaults	E.	leip
Pin Nam	es	ip op		
Direction		input	Bus Expansion 🖲 off 🔵 on	-
Heado		schematic	Placement  single multiple	

Figure 9. Add pin pop-up.

Stamp down the input pin like in Figure 10. Change the direction of the Y pin in the add pin pop-up to output. Stamp it down like in Figure 10.

### VLSI Design I, Tutorial 2

					Virt	tuo	so® S	Sche	ema	tic E	diti	ng:	TestLit	o My	INV :	sche	ema	tic						j,	• 🗖
Cm	d:		Sel: O																						22
Tools	Design	Windo	w Edi	t Ado	l Che	ck	Sheet	Opti	ions	NCS	U													3	Help
$\blacksquare$	548 -																								
۲													vdc												
€ <sup>2</sup>	÷																								60 60
ଙ୍	(9)												•												-
V Sol	<u>:</u> *:											er Victor													1
₩.	145 2年										_iţ	신	r Po vdql <sub>smo</sub>	25dF											
4	44)										T		w=' 1=2	960. 40.(	.Øn Øn										-
$\cap$	98				b l		ж Э				_	्य २३	m:1						5						•
聯	(*) (*)																								
Q	1										ļi	o p c	p <mark>_NØ</mark> nd! <sub>tsmo</sub>	:25dN											-
1	νte Γ										2. 2.			480	.Øn										
ŗ	N26										. 3	gnd	! = 1=2 m:1	40.1	2n										20
abc																									
~													T dB2												
*																									
	38) 	-87		5-0 -	1.4		10	10) 		<u>*</u> .		6.5		620	17		2,5		1.1	12	12	(†) 	2.5		1
G	mouse L	: shou	Click	Info(	)				M	: sch	HiMou	isePc	pUp ()				R: :	sevRu	nEng	ine(′	sevS	essio	n1)		12
	>																						1990. <b>f</b>		

Figure 10. Completed schematic.

To connect the substrate of the pmos transistor to vdd, press the esc key to get out of the add pin mode and then press w to enter the wire mode. While in wire mode connect the nmos substrate to ground as well.

To save and check your schematic for errors, go to **Design->Check & Save.** Any errors will be highlighted in the schematic window. (The CIW window will let you know the results of the schematic check.) You can exit the schematic capture tool at this time (go to **Window->Close**).

### 5. CREATE INVERTER SYMBOL

To use a test bench (test bench means a separate schematic where you will connect voltage sources and whatever else for simulating your design) you must have a symbol view of the inverter. Do not use this inverter's schematic view as a test bench; otherwise, it will not pass LVS (LVS means layout versus schematic).

To create a symbol view:

1. Go to the Library Manager window and create a new cell view in your current working library for lab2. When the window appears for creating the new cell, select the "Composer-Symbol" tool from the pulldown window and make sure the cell name is the same as the previously

created inverter schematic cell. Click OK and the symbol editor window will appear, as shown in Figure 11.

			Vi	rtuc	oso® §	symb	ol Ec	litin	g: la	ıb2	INV	syml	ool				
Cmd:		Se	1:0														6
Tools	Design	Window	Edit	Add	Check	Optio	ns										Help
۲																	
€ <sup>2</sup>																	653 619 619
Q <sup>2</sup>																	
																	- 619 - 819
Q																	8.68 5.55 1.551
· //																	
$\cap$																	635 619
~ <b>.</b>																	
>																	673 679
Q																	- 6369   - 6359   - 6659
[@abc]																	
Q																	
				83454 83254	19 19 19 19 19 19		en enne en enne		(* *) (* *)		60.04 60204	• )• (• • (• (•		0 80,98 ( 80 <u>9</u> 9	• • •	かれた 第1日日	
	iouse L	: mouse!	Single	Selec	tPt	1	1: scł	uHiMo	usePo	օրՄը (	0		R:hi0	loseWi	ndow (g	jetCuri	cent₩in

Figure 11. Symbol editor.

The symbol editor allows you to draw any kind of shape to represent your logic gate. The important part is to have the input and output pins match whatever it is you are representing. In this case, you need to match the input pin A and output pin Y of the inverter. Rather than drawing an inverter symbol from scratch, it is much easier to copy one that has already been drawn.

To import a symbol:

 Go to Add->import symbol and a pop-up window will appear. Click on browse and select the inverter in the digital library of the NCSU kit (NCSU\_Digital\_Parts->inv->symbol).
 Stamp the symbol down into the symbol editor, as shown in Figure 12. (note that the names of the pins will need to be changed so that they match the names that were assigned to the inverter in the schematic cell)

- 3. Press the esc key to get out of add symbol mode.
- 4. Press the f key to fit the symbol in your window.
- 5. For each of the pin names, do the following:
  - 1. Select the actual pin (the red square), press 'q', and change the 'A' ('Y') to 'ip' ('op')
  - 2. Select the text label, press 'q', and change the 'A' ('Y') to 'ip' ('op')

6. Go to **Design->check & save**. You should have no errors (check the CIW window). If there are errors, the pins must be wrong in your schematic. Go back to your schematic and fix the pin assignments. Also, check for direction and name. If there is one pin misnamed, you will see the following two errors:

1. A pin in the schematic view was not found in the symbol view.

2. A pin in the symbolic view was not found in the schematic view.



Figure 12. The inverter's symbol.

Close the symbol editor by going to **Window->Close**.

# 6. CREATE INVERTER TEST BENCH

Now that you have created a schematic and symbolic view of your inverter, it is time to create a test bench. This will be a virtual test bench, but it will have to have a power supply, input vectors, and a ground (plus the inverter) to test its transient response and DC response.

To create the inverter test bench:

In the Library Manager, go to **File->New->Cell view**. Create a new cell in your lab2 library and name it INV\_TB, and assign this cell to the Composer-Schematic tool.

In the schematic editor press 'i' or go to **Add->Instance**. (This is the same way you added the pmos4 and nmos4 previously in your inverter schematic.) Click on browse, and this time select your lab2 library. Click on flatten and select your inverter (this is the symbol that you just created). Stamp it down in your schematic.

While still in add instance mode, add the **gnd** and **vdd** global symbols just like you did when creating the schematic for your inverter. Add the power supply (vdc from voltage sources in NCSU\_Analog\_Parts) and set the DC voltage to 2.5 Volts, as shown in Figure 13.

		A	dd Instance	È.
Hide	Cancel	Defaults		Hel
Library	NCSU_An	alog_Par	tă	Browse
Cell	vdď			
View	symboli			
Names	Ĭ.			
Annay		Rows	1 Col	lumns 1 <u>1</u>
Rotat	te	Si	deways	Upside Down
AC phase DC volta Noise file Number	e ge e name of noise/fr ture coeff	eq pairs icient 1	2.5 V I. Q	
Tempera			17.	
Tempera Tempera	ture coeff	icient 2	I.	

Figure 13. Adding the power supply (VDD from specification).

Add the load capacitor according to Figure 14. Make sure it is 10f F not 10 F! This simulation will not work if the capacitance is too big!

Add the test vector (vpulse) according to Figure 15.

		1	Add Instan	се	
Hide	Cancel	Defaul	lts		Hei
Library	NCSU_An	alog_P	arts		Browse
Cell	capi				
View	symbol				
Names	ļ.				
Anray		Rows	1	Columns	1 <u>ľ</u>
2.24					
Rotat	e		Sideways	1	Jpside Down
Rotat Capacita Initial co Model na	nce ndition me		Sideways		Jpside Down
Rotat Capacita Initial co Model na Width	nce ndition ume		Sideways		Jpside Down
Rotat Capacita Initial co Model na Width Length	nce ndition une		Sideways           10f řť           í           í           í           í           í		
Rotat Capacita Initial co Model na Width Length Multipliei	nce ndition ume		Sideways		
Rotat Capacita Initial co Model na Width Length Multipliei Temp ris	nce ndition me r e from am	ıbient	Sideways		
Rotat Capacita Initial co Model na Width Length Length Multiplien Temp ris Tempera	ie nce ndition me e from am ture coeff	ıbient icient 1	Sideways		

Figure 14. Adding a load capacitor (10f F from specification).

-		Ad	dd Instanc	e			
Hide	Cancel	Defaults		He			
Library	NCSU_Ar	alog_Par	tš	Browse			
Cell	vpulse			Ī			
View	symbol						
Names	Į.						
Array		Rows	1 <u> </u>	olumns 1			
Rota	te	Si	deways	Upside Down			
AC phase Voltage	e 1		I. OV				
Voltage (	2		2.5 V				
Delay tir	ne		Q s				
Rise tim	e		100p s				
Fall time			100 <u>p</u> s				
Pulse width			500pi s				
Period			ln s				
DC volta	ge		-				
Noise file	e name						

Figure 15. Adding a test vector (From specification).

						- 27	/irt	uo	so	° So	chε	ema	tic	Edi	itin	g:	Te	stLi	b	My	INV	(_T	Bs	sch	em	atio	a)							• 🗖
Cm	d:		5	Sel: (	0																													23
Tools	Desig	n s	Mindo	w I	Edit	Ada	d C	hec	k S	heet	t O	ptior	ns M	ICS	U																			Help
R	a la li																																	a 12
	êc (13																																	
	8 S -																																	
æ.2																																		
R																																		
$\Theta^2$							1		vd																									
~	n is N Sa																												-					
100	6 G I							120	1072																									
-Disco							vđ	<u>"</u>	- Y Ø	dc=	2.5																							
De	18 Sa							5	2																									
4	6 G (						gлo	<u>11</u>	Ľ																									
7	5 5																																	8 B
$\cap$	M Na - I																																	
1111111	ê (3																			1			IØ											
調購	9. 24 17 52 - 1											ìr	- <del></del>					-	ĺβ			0		8-	0	P	c	D	or OF					
1 E												nn (	4	v1:8	5						în	V						-	- 0	=10	Øf			
~												30.0	Ý	v2=	2.5													-						
1												gnđ		4	100												gnr	11	l.					
	2 64 - 1																																	
٦.	6 G																																	
abc	9 St -																		7	7														3 B
7	6 B																																	
•D	êr (9																																	
[manana]	1. S 																																	
0				o1.4	1.7	6 - 1	,		_				v																- 1				1.5	12
L	mouse	Ц:	show	UL1(	ckin	ro(	)						M :	sch	HIMO	nuse	Pob	up ()	)					H	(: S	evKu	mer	ıgır	te (7	sev	5688	nor	1)	

Figure 16. The completed test bench.

Press the esc key to get out of add instance mode. Move the symbols around as needed by pressing the 'm' key for move.

Press the esc key to get out of move mode and wire up the schematic according to Figure 16. Go to **Design->Check & Save** to check for errors. If you receive warnings and errors about pins being in one view and not the other, you have either name differences in the schematic vs. symbol or pin direction differences in schematic vs. symbol. The symbol should be perfect so check the schematic you did to make sure the pins are labeled correctly and the directions are correct.

You are now ready to simulate your inverter!

# 7. SIMULATION IN SPECTRE SPICE USING THE ANALOG DESIGN ENVIRONMENT

Now that you have created a schematic and symbolic view of your inverter and created a test bench in which to test your inverter, you are ready to run the Analog Design Environment tool. This is a tool to simulate your circuit in a physics-based environment that solves for voltages and currents over time. This is different from a functional simulation, which is just doing a high, low, Z or X analysis.

To simulate your circuit:

From the schematic editor window of your test bench circuit, go to **Tools->Analog environment**. A pop-up window like Figure 17 should appear. Since this is the first time you are running the Analog design tool, you will need to customize the environment.

— Virtuos	o® Analog Design Environment (1)	
Status: Ready Session Setup Analyses	T=27 C Simulator: spectre Variables Outputs Simulation Results Tools	S & Help
Design	Analyses	K
Library lab2	# Type         Arguments         Enable           1         tran         0         2n         ves	J AC F TRAN J DC
View schematic	2 dc 0 5 yes	
Design Variables	Outputs	D#X
# Name Value	# Name/Signal/Expr Value Plot Save March	4
		1
>	Plotting mode: Replace	2

Figure 17. Analog Design Environment.

You will be using the Spectre Spice simulator, which is a spice-like simulator, using BSIM3 model decks, but the underlying algorithms are different. These algorithms are transparent to the user.

Go to **Setup->Simulator/Directory/Host**. A pop-up like Figure 18 should appear. Make sure the simulator is set to spectreS and click OK when done.

ок	Cancel	Defaults	Help
Simulato	r	spectre S 📃	
Project I	Directory	<pre>~/cadence/simulation</pre>	
Host Mo	de	● local	
Høst			
Qomoto	Directory	1	

Figure 18. Setting up the simulator to use.

Now you need to make sure that Analog design tool can find the model decks for your process. Go to **Setup->Model path** and a pop-up like Figure 19 should appear. Add the model path as indicated in Figure 19.

ок	Cancel	Defaults	Apply	Apply & Ru	n Simulat	ion	Help
Directi	ories	/app	s/cade	nce2005/10	cal/mode	ls/spectre/	'nom
New D	irectory	I Add 4	Above	Add Below	Change	Delete	

Figure 19. Setting up the model path.

#### Transient Analysis:

You will use the transient analysis to measure propagation delay times. To set up the transient analysis, go to **Analyses->Choose** and select the 'tran' radio button. Enter **2n** for the stop time of the transient analysis (equates to 2 nano-seconds).

#### DC Analysis:

In order to measure VINV (where VOUT=VIN), you need to do a DC analysis as well. VINV is a measure of the noise performance, and the closer VINV is to VDD/2, the better the noise performance of the inverter. Go to **Analyses->Choose** and fill out the pop-up according to Figure 20.

Click on Select Component and then in the schematic click on the vpulse.

Select VDC when the pop-up like Figure 21 appears.

Select the start and stop voltages to be 0 and 2.5. Your pop-up should look like Figure 22.

<ul> <li>Choosing Analyses</li> </ul>	Virtuoso®	Analog Desig
OK Cancel Defaults App	ly	Help
Analysis ◯tran ◯ac ●dc ◯xf	)sp )pd )pss )nd	listo 🔵 spss iise
DC Save DC Operating Point	Analysis	
Sweep Variable	Component Name	[
Component Parameter	Select Co	mponent
_ Model Parameter	Parameter Name	dď
Sweep Range		
Start-Stop Start	0 Stor	n 2.5
Center-Span		
Sweep Type		
Automatic		
Add Specific Points 📃		
Enabled 🔳		Options

Figure 20.	Setting	up a	DC	analysis.
------------	---------	------	----	-----------

- Select Component Parameter									
OK Ca	ncel	Help							
de	vdc	"DC voltage"							
mag	acm	"AC magnitude"							
phase	acp	"AC phase"							
val0	v1	"Voltage 1"							
val1	<b>v</b> 2	"Voltage 2"							
period	per	"Period"							
delay	td	"Delay time"							
rise	tr	"Rise time"							
fall	tf	"Fall time"							
width	pw	"Pulse width"							
tc1	tc1	"Temperature coefficient							
tc2	tc2	"Temperature coefficient							
tnom	tnom	"Nominal temperature"							
4									

Figure 21. Choosing to sweep DC voltage.

- Choosing Analyses	Virtuoso® Analog Desi Iv Heli
Analysis tran ac ac xf	sp pdisto spss pss noise
DC Save DC Operating Point	Analysis
Sweep Variable Temperature Component Parameter Model Parameter	Component Name /v1 Select Component Parameter Name de
Sweep Range Start-Stop Center-Span Sweep Type Automatic	g Stop 2.5
Enabled	Options

Figure 22. Completed DC Analysis Setup.

### Adding Wire Names:

In order to easily see which signal is the input and which is the output, it is advisable to add wire names to the wires connecting A (input) and Y (output) of the inverter. Go to Add->Wire names and add names 'A' and 'Y' to the resulting window, as shown in Figure 23. Select the inverter input net and then the inverter output net in your test bench schematic in order to 'name' the nets.

		Ad	ld Wire Name
Hide	Can	cel Defaults	Help
Wire N	lame	Net Expression	
Names		ip op	
Font He	ight	0.0625	Bus Expansion 🖲 off 🔵 on
Font St	yle	stick 📖	Placement 💿 single 🔵 multiple
Justific	ation	lowerCenter	- Purpose 💿 label 🔵 alias
Entry S	ityle	fixed offset —	Show Offset Defaults
Rota	ate	1	

Figure 23. Adding Wire Names.

#### Choosing which nets to plot:

Now you need to choose which vectors you want to plot. Go to **Outputs->To be plotted->Select->on schematic**. Go back to your inverter test bench schematic and click on the net labeled 'ip' and the net labeled 'op'. Observe that the nets change color as you select them. (If you click directly on a pin it will probe the current, and the terminal of the device you clicked will have circles drawn around them. Clicking on a net, as you're doing here, probes the voltage.)

Press the esc key to get out of selection mode. Your Analog design window should now look like Figure 24.

– Virtuos	o® Analog Design Environment (1)	* [
Status: Ready	T=27 C Simulator: spectreS	3
Session Setup Analyses	Variables Outputs Simulation Results Tools	Help
Design	Analyses	Ł
<b>ibrary</b> TestLib	# Type Arguments Enable	J AC
<b>Cell MyINV_TB</b> <b>Mew</b> schematic	1 tran 0 2n yes 2 dc 0 2.5 yes	
Design Variables	Outputs	[]ŧ
≭ Name Value	# Name/Signal/Expr Value Plot Save March	4
	1 op yes allv no 2 ip yes allv no	100
	Plotting mode: Replace 💷	to

Figure 24. Selecting which vectors to plot.

In the Analog design tool, go to **Simulation->Run**. The CIW and the Analog design windows will show you the process it goes through to simulate your circuit. If all went well, a waveform window containing your transient and DC analyses should appear. Observe these waveforms to ensure that they match your expectations for your inverter circuit design.

If the simulations worked, go back to the CIW and go to **Options->Save defaults** and click OK on the pop-up window. This will save your simulator environment, but not your plot settings.

Note: The transient simulation should look similar to Figure 25. Go ahead and add markers (go to the Marker menu) on the input and output waveforms at the voltages shown in Figure 25 (you will use this for comparison purposes with the post-layout simulation).



Figure 25. Transient Analysis.

# **Possible Simulation Problems:**

Doh! The simulation did not work!

If you get error messages like "model XXX not found", you did not set the path correctly to point to the directory containing the models for your PMOS and NMOS FETs.

If you get a spice error message, you probably did not set your vpulse, or vdd correctly.

If it takes a long time to run, and/or the simulator times out, the capacitor could be too big, or the width or length of a MOSFET could have the wrong units (for example, instead of 4 microns it is 4 meters).

If the simulation runs but the output does not look correct, the capacitor could be too big or the DC voltage may not be set correctly.

To fix or change properties of the inverter circuit, go to **Design->Hierarchy->descend edit** and click on your inverter. Change the pop-up to schematic and click OK. To edit a MOSFET or anything else on your inverter circuit, press 'q' and click on what you want to edit. To return to the top level (the level of your test bench), go to **Design->Hierarchy->Return to the top**.

# Saving your State:

In order to save certain setup specifications in your simulation, save the state. To do this, go to **Session->Save the state** in your Analog design window.

#### 8. POST LAYOUT SIMULATION

Now, you will extract the parasitic capacitances from your inverter layout (done in Tutorial 1) in order to include them with your transient simulation.

First, open up your inverter layout from the Library Manager window. Go to **Verify->Extract**. Click on **Set Switches** and select **Extract\_parasitic\_caps** in the resulting window and click OK. Your Extractor window should now look similar to Figure 26.

-			Ext	ractor		
ок	Cancel	Defaults	Apply			Help
Extract N	<b>Aethod</b>	🛈 flat	) macro c	ell () full	hier () inc	remental hier
Join Nets	: With San	ne Name			Echo Com	mands 📃
Switch N	lames		Extract_pa	rasitic_(	caps	Set Switches
Run - Spe	cific Com	mand File				
Inclusion	Limit		1000			
View Nar	nes	Extracted	extracted		Excell	excell
Rules File	P		divaEXT.r	ulį		
Rules Lib	rary		U_Tech	Lib_ami06	*	
Machine			🖲 local 🔵	remote	Machine	Į

Figure 26. Extractor Window.

Click on OK to perform this operation. Now close the layout view of your design. Bring up the extracted view of your layout by double-clicking it from the library manager. Next, you will perform a layout-versus-schematic comparison as a design check. Go to **Verify->LVS** and you will see the window as shown in Figure 27. Fill it out according to this window.

-		LVS			
Commands				Help 19	
Run Directory	L∀ŠĮ			Browse	
Create Netlist	schematic		extracted		
Library	TestLib		TestLib		
Cell	MyINV		MyINV		
View	schematič		extracted		
	Browse	Sel by Cursor	Browse	Sel by Cursor	
Rules File	divaLVS.	divaLVS.rul		Browse	
Rules Library	MCSU_	_TechLib_tsmc03	١ď		
LVS Options	📕 Rewirin	ıg	_ Device	Fixing	
Correspondence Switch Names	File	Cross Reference	Termina	file Create	
Priority 🕅	Run lo	cal 💷 👔			
Run	Output	Error Display	Monito	or Info	
Backannotate	Parasiti	ic Probe Build	Analog	Build Mixed	
	[ ]		a la constanti a		

Figure 27. LVS Window.

Click **Run** and wait *patiently* for a popup window to tell you that the LVS process has completed successfully. Note that this it not telling you that the layout and schematic netlists are identical. To find out the results from the LVS, click on **Output**. Determine if the netlists match from the resulting window.

After ensuring that the netlists match, in the LVS window, select "Build Analog". This creates a file called analog\_extracted (in the same cell view that contains your layout and schematic). You will use this file when doing the transient analysis in the Analog Design Environment.

Go back to the Analog Design Environment window, and go to **Setup->Environment**, and insert "analog\_extracted" in front of "schematic" in the Switch View List, as shown in Figure 28.

### VLSI Design I, Tutorial 2

Environment Options				
OK Cancel Defaults App	Hel			
Init File	I			
Undate File	T			
Demonster Denne Checking File	17			
Parameter Range Checking File	ja.			
Recover from Checkpoint File				
Netlist Type	🔵 flat 🖲 hierarchical 🔵 incremental			
Switch View List	; spice cmos_sch cmos.sch analog_extracted schematic $v$			
Stop View List	spectreS spice			
Instance-Based View Switching				
Instance View List Table	Į.			
Instance Stop List Table				
Print Comments				
Include/Stimulus File Syntax	CdsSpice  spectre			
Include File	Ĩ.			
Stimulus File	L.			

Figure 28. Environment Options.

Run the transient analysis, and your waveform should look similar to Figure 29. (Place markers at the voltages shown, and verify that the timing is different from your previous results.)



Figure 29. Transient Analysis with Parasitic Capacitances.

# 9. TWO-INPUT NAND GATE ANALYSIS

Using the procedure developed and demonstrated in the inverter circuit design/test, design a two-input NAND gate circuit (shown in Figure 26 for your reference) with accompanying symbol, and then create a test bench schematic for simulation purposes (you will not do a layout here!).



Figure 26. Two-input NAND Gate Circuit.

### **10. ASSIGNMENT**

- Turn in the transient analysis waveform printouts from your inverter:
  - Without parasitic capacitances (20%)
  - With parasitic capacitances (20%)
- To demonstrate your design and test procedure, hand in the following printouts:
  - your two-input NAND gate circuit (10%)
  - your two-input NAND gate test bench circuit (10%)
- To prove the operation of your NAND gate circuit, hand in printouts of the transient analysis waveforms of the inputs and output to prove the entire truth table shown in Table 1 (40%)

Input 1	Input 2	Output	Grade %
0	0	1	10
0	1	1	10
1	0	1	10
1	1	0	10

Table 1.	Two-Input NAND Gate Truth Table.
----------	----------------------------------