Virtuoso Schematic Composer

Virtuoso Schematic Composer is a schematic design tool from Cadence. In this tutorial you will learn how to put electrical components, make wire connections, insert pins and check for connection error.

Start Cadence Custom IC Design

Before start, you should have necessary files and setup done to be able to run Cadence software. If you have not done this, see tutorial on "How to setup Cadence tools?" (Available on class website)

To start Cadence Custom IC Design software, go to the directory you set up for running Cadence (~/cadence), type in icfb command as shown in Figure 1.



Figure 1: Command for running Cadence tool

The Command Interface Window (CIW) and Library Manager window should appear as shown in Figure 2.

And the second sec			
🔄 Show Categories 🛛 🛄 Show F	files		
Library	Cell	View	
	T T	Ť	
		J.	
MOSIS_Layout_Test			
NCSU_Analog_Parts			1
NCSU_Digital_Parts		Librowy Monogon	1
NCSU TechLib ami16		Library Manager	
NCSU TechLib hp06			
NCSU_TechLib_tsmc02		L	
NCSU_TechLib_tsmc02d			
NCSU_TechLib_tsmc03			
NCSU_TechLib_tsmcU3d			
NCSU_TechLib_tsmc04_4M2P			
US 8ths			
ahdlLib			
ambit			
analogLib			
Dasic			
cdsDefTechLib			
cds_assertions			
correctlib			
functional			
ieee			
ncinternal			
ncmodels	V		
	- Prove 10	10	
Messages			
Messages Delete of library "cmpe315" s	ucceeded.		
Messages Delete of library "cmpe315" s Deletion of library done.	ucceeded.		
Messages Delete of library "cmpe315" s Deletion of library done.	ucceeded.		
Messages Delete of library "cmpe315" s Deletion of library done.	ucceeded.		
Messages Delete of library "cmpe315" s Deletion of library done. I icfb - Log:/afs/umbc.edu/users	ucceeded. ;/e/k/ekarat2/home/CDS.log		
Messages Delete of library "cmpe315" s Deletion of library done.	ucceeded. /e/k/ekarat2/home/CDS.log	ц	
Messages Delete of library "cmpe315" s Deletion of library done.	ucceeded. ;/e/k/ekarat2/home/CDS.log th the Library Manager.	Command Interface	
Messages Delete of library "ompe315" s Deletion of library done.	ucceeded. :/e/k/ekarat2/home/CDS.log th the Library Manager.	Command Interface	
Messages Delete of library "cmpe315" s Deletion of library done. icfb - Log: /afs/umbc.edu/users File Tools Options Lefb) Syncing library list with	ucceeded. :/e/k/ekarat2/home/CDS.log th the Library Manager. th the Library Manager.	Command Interface Window (CIW)	=
Messages Delete of library "cmpe315" s Deletion of library done. icfb - Log: /afs/umbc.edu/users File Tools Options .cfb) Syncing library list with .cfb) Syncing library list with	ucceeded. ;/e/k/ekarat2/home/CDS.log th the Library Manager. th the Library Manager.	Command Interface Window (CIW)	- 0 lp 1
Messages Delete of library "cmpe315" s Deletion of library done. icfb - Log: /afs/umbc.edu/users ile Tools Options .cfb) Syncing library list with .cfb) Syncing library list with	ucceeded. /e/k/ekarat2/home/CDS.log th the Library Manager. th the Library Manager.	Command Interface Window (CIW)	

Figure 2: CIW and Library Manager

Create New Library

The design library is a group of cells, you can have multiple views (schematic, layout, symbol, vhdl, etc.) for each cell in the library. The technology (process) defined for the library will be used for all cells in that library.

To create new library, go to **File** -> **New** -> **Library** from Library Manager menu. The New Library dialog as shown in Figure 3 will appear. Type in the name for this library and then click OK. Another dialog will pop up asking for the technology of this library, choose "**Attach to an existing techfile**" as shown in Figure 4. Next you need to choose the technology, select "**NCSU_TechLib_ami06**" as shown in Figure 5.

Name	cmpe315
Director	Y
 LVS cache_ test	vhdl_code

Figure 3: New Library Dialog

✓ Tec	hnology File for New	r Library 📈 🗙
ок	Cancel	Help
Techno	logy File for library "	cmpe315"
If you other	I will be creating mas physical data in this	sk layout or library, you
to us	eed a technology file e only schematic or l	. IT you pian IDL data, a
techr	ology file is not requi	ired.
You	an: 🔷 <u>Compile a nev</u>	v techfile
	Attach to an e	existing techfile
	Don't poor a	TOC DI IIO

Figure 4: Technology File for New Library

ок	Cancel	Defaults	Apply	Help
New Des	sign Librar	y	cmpe315	
Technolo	and Liberry		NCSU TechLib ami06	

Figure 5: Attach Design Library



Create New Schematic

From Library Manager, click to highlight the library (cmpe315 in this tutorial) you want the new cell to be created in. From the menu, go to **File -> New -> Cell View**. The new cell view dialog as shown in Figure 6 should appear. Type in the cell name (we will make an inverter in this tutorial) and make sure you have "**Composer-Schematic**" selected as a tool. Click OK and Virtuoso Schematic window will appear as shown in Figure 7.

ок	Cance	Defaults	Hel	p
Library Na	me _	cm	pe315	
Cell Name		ind	12	
View Name	,	schematic		
Tool	<	Composer-Scl	hematic _	
Library pat	h file			
rs/e/k/ek	arat!	/home/caden	ce/cds.lib	

Figure 6: New Cell View

Y Vir	tuo so® S	Schemat	ic Edi	iting:	cmpe3	15 inv so	chematic				//////	7777777	//////	777777	//////	77777		/////	//////		////	
Cm	ł:	Se	1: 0																			2
Tools	Design	Window	Edit	Add	Check	Sheet	Options	Migrate	NCSU													Help
		<u></u>									<u>10 3</u>	2 12 A	100	34	<u>99 - 166 - 1</u>	31 - 32 1	- 424 - 3	<u>1</u> 1 - 24	2222	5 3		
\square	12 43 33																					8 - 8 ¹¹
ഷം																						
\otimes	e: .ee.																					
⊕ ²	8 8 9																					
Ľ	8 9 S																					a - 12
(E) ²	10 dt 30																					5 B
\sim	$\kappa \ll \varkappa$																					• •
	18 N N																					a 8
v	12 dt 22																					5 B
\mathbb{R}																						
1	toolt	bar 🔤																				
4								Sche	matic	Dray	wind	I Are	a									
y	10 10 10 10 10 10																					
\cap	* * *																					
α =	8 8 3																					a 8
制制	8 8 8																					a (1
dabab	* * 3																					• •
ų	* * *																					s 0
*	3 8 3																					5 (s
J,	* * 3																					
2000 1000	•>> === ===																					
٠.۲																						
	10. <i>1</i> 8 4																					
-	-10 H (A																					
•D	16 16 38																					a - 6
6773	52 <i>3</i> 2 33																					5 B.
	* * *																					· · · · ·
0	<u>si</u>																					
C1	mouse L	: showC	lickI	nfo()			М	: schHiM	ousePopl	յն ()			I	R: hi	Repea	at()						
	>																					

Figure 7: Virtuoso Schematic

Working with Virtuoso Schematic Composer

In this tutorial, we will make a schematic of the inverter. First, we need NMOS on the schematic. To add components to the schematic, from menu go to Add -> Instance, the Component Browser and Add Instance dialogs will show up. At the Component Browser, shown in Figure 8(a), change library to NCSU_Analog_Parts. You should see N_Transistor in the component list, click on N_Transistors and the component list will change as shown in Figure 8(b). Select nmos4 (4-terminal NMOS), now you can change parameters for the NMOS (e.g., width, length, no. of finger, etc.) at the Add Instance dialog, shown in Figure 9. For this tutorial, we will use transistors with the width of 1.5µm and length of 0.6µm. After changing parameters, now move the mouse over schematic drawing area, left click on the schematic where you want the new component to be placed.

Repeat the steps for placing PMOS, vdd and gnd on the schematic. Using NCSU_Analog_Parts library, you should find PMOS from P_Transistor -> pmos4, vdd from Supply_Nets -> vdd and gnd from Supply_Nets -> gnd. Press **Esc** to end add instance command. (**Esc** can be used to end any commands in Virtuoso Schematic)

Commands	Help 3	Commands	
Library NCSU	_Analog_Parts =	Library	NCSU_Analog_Pa
Flatten 🔄		Flatten	
Filter *		Filter [*]	м.
Uncategorized CONTENTS Current_Sources Diodes H_Spice_Only Microwave_Parts Misc Parts N_Transistors P_Transistors Parasitic_Device R L C	s	(Go u nbsim nbsim4 njfet nmes nmes4 nmos4 nmos4 nmos4_hv nmos_hv	mp 1 level)
(a)			(b)

Figure 8: Component Browser

Hide	Cancel	Default	S		Н							
Library	NCSU_An	alog_Pa	rtă		Browse							
Cell	nmos4											
View	symbol											
Names	lames I vray Rows 1. Columns											
Array												
	Upside	Down										
Model na	ame		ami06	N								
Model na Model T	ame ype		ami06	N item 🔷 use	ir							
Model na Model T Multiplie	ame ype r		amiO6 ♦ sys 1	N Stem 🔶 use	r							
Model na Model T Multiplie Fingers	ame ype r		ami06	N item vuse	r							
Model na Model T Multiplie Fingers Width (g	ame ype r rid units)		ami06	N tem 🔷 use	r							
Model na Model T Multiplie Fingers Width (g Width	ame ype r rid units)		ami06	N tem \diamond use	r 							
Model n Model T Multiplie Fingers Width (g Width (n	ame ype r rid units) ninimum)		ami06	N tem \diamond use	r							
Model n Model T Multiplie Fingers Width (g Width (n Length (ame ype rid units) ninimum) grid units)		ami06	N tem \diamond use	r							

Figure 9: Add Instance

Vir	tuo se	® S	chen	natio	: Edi	ting:	cmpe3	15 inv s	chematic					,,,,,,,,		//////		/////		/////	/////	/////	-	
Cm	d: Ins	tanc	e		Sel:	0																		2
Tools	Des	ign	Wind	ow	Edit	Add	Check	Sheet	Options	Migrate	NCSU												J	Help
	. 22																							
-	£3																							
(f) ²	1											×												
~	13											<u>1</u>												
Q2	83																							8.0
N/2	13											PØ												
	83 -										i-d	amlø	δP											
120	52										. M		.5и Ил											
3	•0											m :1												
0	15																							5
10 •	•55																							
	12										18 15 142	NØ												
Q	•88										<u> </u>		SN'											
	- 12											-W=1	.Su Man											
. ۲	485											m:1	20011											
۲ ۲	53																							
	28																							
	8											i a a												
•	- 53 -																							
	•5											\bigtriangledown												90
	12	1			5	15	80	1	1 2	ST	19 St.	10 (t)	ð.:	<i>.</i>		\$	1		5,2	.*:		3	12	7
G	mous	e L	mou	seA	ddPt	0			M	: schHiM	ousePop	Մթ ()			R	Rota	ate 9	0						12
	Point	at I	ocati	on f	or the	insta	ance.																	

Figure 10: Schematic after placing components

Figure 10 shows the schematic after placing components. You can move components around by go to Edit -> Move, left click at the component you want to move, move the mouse to new location and left click to place(do not drag the component, click once on the component and click again at the new location).

Now, we need to make connections between components. From the menu, go to Add -> Wire (narrow), the add wire dialog as shown in Figure 11 will show up, you can change wire color and style from this dialog. Click on the pin of the component you want to make a connection (when you move mouse over the pin, yellow circle around pin will appear), move mouse to another pin you want to make connection, click again to end the wire. Note that you can make a 90 degree turn on the wire by left click at the turn. You can also end the wire without connecting to the pin by double click.

Figure 12 shows the schematic after add wires. You can check the schematic connection and save the schematic by go to **Design -> Check and Save**. Since we did not make the complete connections, the error/warning dialog will appear as shown in Figure

13. You can see the explanation of errors/warnings on ICW window as highlighted in red in the figure.

Hide	Cancel	Defaults		Hel
Draw M Width	ode	route -	Route Method	full =
Color		cadetBlue		
Line Stv	/le	solid		

Figure 12: Add Wire

Virt	uo so	® S	chem	atic	: Edit	ing: o	mpe	315 inv s	chemat	c//////	/////			77777	/////	/////	/////		/////	/////	/////	/////	-	
Cmd	I: Wire	5		Sel:	0																			2
Tools	Desi	gn	Windo	w	Edit	Add	Chec	k Shee	t Option	is Migra	te N	csi												Help
\mathbf{P}																								
*													vdd											14
€ ²																								4 6
Q2																								×
N.													PØ											4
× 1												-0	am106 w=1.	iP .5u										
-													=60 m:1	Øn										6.6
$\overline{\Omega}$																								
													amiØ6	N.										64
													w=1.	.5u Øn										ः
													rn:1											
- abc																								4
Ř.													and											
-													\checkmark											
										3. 5										2				7
G.	mouse	L	mous	seAd	ddPt()				M: schH	iMou	sePo	թՄբ ()			R:	Togg	le D	raw	Mode				orana a

Figure 13: Schematic after add wires

CMP UMI	E 31 BC	15/C	CMP	PE64	0			Vi	rtuo	DSO	Sch Tı	ema utor	tic ial	Co	mp	ose	r]	Eka	rat]	Laoł	ava	lees	0 n
		28		<i>.</i>									F	ø	20									10	
т) Т											ICL.	J.	⊽ ka ⊠	m1 Ø6	P										à
53.												ML	W	=1.	5.u										5
5												net5	r.	= DØ 1:1	Un										
•30																									S.
																									-+
1 00												net5													S.
						Ĩ	× 9	cher	natic	Che	ck			/////		////	////	×							
•							e I	The	re w	ere I	0 err	ors a	and 4	l war	ning	gs f	ound								R
								10000	616 M	A AVAILA						KO LENED	10. second								
10												Close	,												98
53																_	_								<u>i</u>
433 1																									34
52																									С.
<i>.</i>																									
* 3		ţ.	ц÷.	ŧ.	ŝŧ	đ	ŝ	4	24	*	ţ.		•		÷	j.t.	÷	ŝ	43	ŝ	*	24	*	t 23	54
: mou:	seAd	dPt()						M: s	schHi	Mous	ePopU	p ()					R:	Togg	le I)raw	Mode			
startin	ng po	int fo	or the	e rou	ter o	r sn	ap to	dian	nond	using	the '	"s" ke	y.												
∀ ic	fb - I	Log:	/afs/	umb	c.edu	ı/us	sers/	e/k/el	karatz	2/hon	ne/CI	os.log]										-		×
File	100	is O	ptior	IS													1					8.4	Help	1	
Warni Warni Warni Warni There	.ng: .ng: .ng: .ng: .ng: wei	Pin Pin Pin Pin ce O	"B" "G" "B" "G" erro	on i on i on i on i ors a	insta insta insta insta and 4	ance ance ance ance	e "P(e "P(e "N(e "N(arnir)": f)": f)": f)": f ugs f	loat loat loat loat ound	ing : ing : ing : ing : ing :	input input input input "cmpe	:/outp :/outp :315 i	out. out. inv :	schem	atio	s".									
																							J	100	

Figure 14: Schematic errors & warnings

Now we need to add input and output pins to the schematic. To add pin, go to Add -> Pin, the dialog as shown in Figure 15 should show up. Type in pin name (you can type multiple pin names using space to separate each pin name), and change the direction to appropriate direction for the pin. If you do not have appropriate directions set, you will find problems when trying to simulate the schematic. Figure 16 shows the schematic after 2 pins are placed (INPUT and OUTPUT).

To complete the connections, we need to tie bulk terminal of PMOS to vdd and bulk terminal of NMOS to gnd. We also need to connect gate of NMOS and PMOS to INPUT pin and make connection from drain of both transistors to OUTPUT pin. Figure 17 shows the complete schematic. Run Check & Save again, you should have no errors/warnings as shown in Figure 18.

Hide	Cancel	Defaults	Help
Pin Namo Direction Usage	es	INPUT] input = Bus Expansio chematic = Placement	m ♦ off ↓ on ♦ single ↓ multiple
		ion: 🔺 No 🔿 Yes	

Figure 15: Add Pin

💙 Vir	tuo so	® S	chema	tic E	diti	ng: d	mpe	315	inv sc	hematic			97777777		. 🗆 🗙
Cm	d: Pin		S	el: 0											2
Tools	Desi	gn	Window	w Ed	lit	Add	Chec	ck S	Sheet	Options	Migrate	SU			Help
	÷.														
	32											a seese seese seese			
02	÷														
X	v'														
⊖²	÷														
~															
1												vdd! PØ			
-D>+	•										· · ·				
2%	÷,											. w= 1.3u			
A.	4.											meto meto meto meto meto meto meto meto			
0	*														
f	à.							NP					J I		6
制制												u sue e national en la companya de la comp			
	÷.										· · · -				
ų												w=1.5u			
												and]=600n			
_															
1,	5														
abcl	÷														
7	£.,											i de 🗯 de les les les les les les les les			a.
•D	e'														
	÷														
0	MO1125	e Le	molie	adda	2± ()	i i				× M	schHiM	ePonTho() R: Rotate 00			
	Use t	he o	ptions	form	i to	supp	ilv tei	rmin	al nam	nes for th	e pins.	n hotale 50			_

Figure 16: Schematic after adding pins

Virtuoso Schematic Composer Tutorial



Figure 17: Complete Schematic



Figure 18: CIW showing no errors/warnings after Check & Save



Using following shortcut keys help working with schematic faster

- i to Add Instance
- **m** to Move
- w to Add Wire (narrow)
- **p** to Add Pin
- **u** to Undo
- c to Copy
- To zoom in on the schematic, hold **right-botton** mouse to create zooming area and release to zoom in.
- To zoom out, press **Shift+z**
- To fit whole schematic on screen, press **f**

Create Symbol View

The symbol views are used in hierarchical design. You can create a symbol view of a cell from its schematic and then use the symbol of that cell in a higher level cell. For example, a buffer cell can be created using 2 inverter cells.

To create symbol view of the schematic, go to Design -> Create Cellview -> From Cellview, the create cellview dialog as shown in Figure 19 should appear. Make sure that you have To View Name set to "symbol" and Tool/Data Type set to "Composer-Symbol". Click OK and new symbol view will pop up as shown in Figure 21. Close the new symbol view.

ок	Cancel	Defaults	Apply			Help
Library	Name	cmpe	315			Browse
Cell Na	me	inv				
From V	/iew Nam	ie <mark>sche</mark> n	natic 💷	To View Name	symbolį	
				Tool / Data Type	Compose	er-Symbol 💷
Display	y Cellviev	w 🔳				

Figure 20: Create Cellview From Cellview

Vir	tuo so®	Syn	1bol E	diting	g: cm	pe315	inv s	ymbo	1////														////	
Cmc	1:		Sel	: 0																				9
Tools	Design	Wi	ndow	Edit	Add	Check	C Op	tions																Help
ako I	83	24	č.	3		13	25	Ċ.			1		3	1	38		3	22	2	35	÷	12	32	
Ŵ																								- 3
€²																								
C2																								
R																								-0
																								22
Щ																								2
																								•
																								•
y																TH	à	et a			aar	al		-
5			T								- L.K. IS							JLU						
å i			-		- 11	APC		×.	@	bar	tN	am	ie]					-						
																								•
••••																								•2
_																								
1																								
Ц																								
																								2
[@abc]																								-8
																								•2
	1994			n 91	14									-						-				
C	mouse :	5: M	iouseS	ingle	Seled	etPt				M: 3	chHiM	louseE	opUp	0			R: s	chHiVi	lewTo'	View	0			

Figure 21: New created symbol view

Create Plot from Schematic

You can create plot of the schematic by go to **Design -> Plot -> Submit**, the Submit Plot dialog as shown in Figure 21 will appear. Click on **Plot Options**, the Plot Option dialog as shown in Figure 22 should show up. Select "**Send Plot Only To File**", type the file name (postscript file), Click OK to close Plot Options dialog and then click OK again on Submit Plot dialog.

Virtuoso Schematic Composer Tutorial

	Cancel	Defaults	Apply		Hel
Plot	🔷 librar	y 🔶 cellvi	iew 🔷 viewir	ng area	
Libi	rary Name	cmpe3	15		Browse
Cel	Name	inv_s	inį		
Vie	w Name	schem	atiď		
Are	a to Plot	((-2.)	31875 -0.562	25) (2.39375 1.25)) (Full Size)
			Full Size	Select	
			1		
	View Nam	e List scl	hematic	arcny vievels du	wn -
	View Nam Skip Libra Skip Cells	e List scl ries ba: Below	hematic sic sample	arcny vieveis uu	
Plot Wit Notes	View Nam Skip Libra Skip Cells h ∎ head	e List sci ries ba: Below ser note	hematic sic sample es _] grid/axe	archy vievers au	
Plot Wit Notes	View Nam Skip Libra Skip Cells h ■ head	e List scl ries baa Below	hematic sic sample es grid/axe	s	
Plot Wit Notes Templa	View Nam Skip Librau Skip Cells h I head	e List sch ries baa Below Jer _ note	hematic sic sample s grid/axe	s ∕schPlotTemplate	Load Save
Plot Wit Notes Templa Plotter	View Nam Skip Libra Skip Cells h ■ head E File	e List scl ries bas Below ler note	hematic sic sample s _ grid/axe 5.1/cdssetup	s /schPlotTemplate	Load Save
Plot Wit Notes Templa Plotter Paper S Plot To	View Nam Skip Libra Skip Cells h ■ heau Le File Name S Name S Name S Name S	e List scl ries baa Below ler _ note	hematic sic sample ss grid/axe 5.1/cdssetup ss Total	s ∕schPlotTemplatej	Load Save

Figure 21: Submit Plot

OK Can	cel Default	ts Apply	He
Display Type Plotter Name Paper Size Orientation	psb default A = \bigcirc portra	= = ait ↓ landscape ♦ aut	Image Position
Scale 0.070	14 <u> </u>	enter Plot 🔄 Fit to Pag	je
Plot Size Offset	10.5000 <u>,</u> 0.0000	x 6.2373 inches x 0.0000	
Plot Size Offset Total Plot Size	10.5000 0.0000 e 10.5000	x 6.2373 inches x 0.0000 x x 6.2373 x	Total Pages 1
Plot Size Offset Total Plot Size Number Of Co Queue Plot	10.5000 0.0000 e 10.5000 ppies 1 t Data At	x 6.2373 inches x 0.0000 x x 6.2373 x Local Tmp Directory 10 PM	Total Pages 1
Plot Size Offset Total Plot Size Number Of Co Queue Plot Send Plot (10.5000 0.00000 e 10.5000 ppies 1 : Data At Only To File	x 6.2373 inches x 0.0000 x x 6.2373 x Local Tmp Directory 10 PM inverter.ps PM	Total Pages 1

Figure 22: Plot Options