Logic Simulation using Verilog XL:

This tutorial includes one way of simulating digital circuits using Verilog XL. Here we have taken an example of two cascaded inverters. The name of this schematic is inverter.

Important Point:

The names of the schematic and the pins in the schematic should not start with a number and should not be keywords in verilog (e.g. and, nand). The length of the names should not be more than 15 characters. You should not use '+' and '-'signs for the names of the pins and the schematics.

1. Open the Schematic cell view:

From the Library manager read the inverter schematic cell view. Save the schematic. The inverter schematic is shown in Fig 1.



Fig 1: Inverter

2. Initializing Verilog Integration:

a. In Fig 1 select *Tools -> Simulation -> Verilog - XL*. The "Setup Environment" window opens as shown in Fig 2.

b. In Fig 2, enter inverter.run1 in the **Run Directory** text box. All other default values are correct. Click *OK*.

[<u>ˈ</u>	Setu	p Environment	
OK Cancel			Help
Run Directory	inverter.run1		
Design To Simu	late		
Library	amitã	Cell	inverter
View	schematic	[
Library Manag	jer Browser Hierarchy	Editor Browser	

Fig 2: Setup Environment

c. The "Virtuoso Schematic Composer Analysis Environment for Verilog-XL Integration" window opens as shown in Fig 3 and the inverter.run1 directory is created.

	Virt	uoso® Sch	ematic Com	poser	Analysis Environi	ment for Verilog-XL Int	egration
				amit3	-inverter-schematic	inverter.run1	Inactive 4
File	Setup	Stimulus	Simulation	Info	Debug		Help
	*						
**	*						
**	Õ						
	-44						
1	ġ,						
Carrent	S						
	<u>&</u>						
and the second s	- Carlor	Verilog_XL	>				
	Sk	Ι					

Fig 3: Environment for Verilog-XL Integration

3. Setting the Netlist Options:

- a. In Fig 3 set the netlisting options by selecting *Setup* -> *Netlist*. The "Verilog Netlisting Options" form opens as shown in Fig 4.
- b. The default settings are as shown in Fig 4. They are all correct.

[['] @	Verilog Netlisting Options					
OK Cancel Defaults Apply						
Netlisting Mode	🔷 Entire Design 🔶 Incremental 🔷 Off					
Netlist These Views	behavioral functional schematic symbol $\!$					
Netlist For LAI/LMSI Models 🔟		More >>				

Fig 4: Verilog Netlisting Options

c. In Fig 4 click *More>>* Additional netlisting options are added to the form as shown in Fig 5.

Simulation using	
Verilog XL	

- d. In Fig 5 set **Global Power Nets** to *vdd* and set **Global Ground Nets** to *gnd*. (The names for the **Global Power Nets** and **Global Ground Nets** should be set as per the names of the power and ground pins in your schematic).
- e. Select Generate Pin Map.
- f. Select Declare Global Locally.
- g. The **Drop Port Range** and **Preserve Buses** options are selected by default. It doesn't matter if we select or deselect them.
- h. Click OK. Thus setting the Verilog Netlisting Options is done.

Verilog Netlisting Options								
OK Cancel De	faults	Apply				Help		
Netlisting Mode 🔷 Entire Design 🔶 Incremental 🔷 Off								
Netlist These Views	beha	wioral fu	nctional sche	matic sy	mbol			
Netlist For LAI/LMSI	Models							
🔳 Generate Test Fi	📕 Generate Test Fixture Template 🛛 📕							
Netlist Uppercase		Generate	Pin Map	I	Preserve Buses			
Netlist SwitchRC		Skip Null	Port		Netlist Uselib			
Drop Port Range		Incremen	tal Config List		Symbol Implicit			
Assign For Alias		Skip Timi	ng Information		Declare Global Locally			
Netlist Explicitly		Support E	Escape Names		Support VhdIImport			
Stop Netlisting at Vie	ews b	ehavioral	functional s	ymbolž				
Global Power Nets	v	dđ				_		
Global Ground Nets gnd								
Global TimeScale Overwrite Schematic TimeScale								
Global Sim Time	1	[Unit	ns 🖃			
Global Sim Precision	1	[Unit	ns 💷			

Fig 5: Additional Verilog Netlisting Options

4. Creating the Stimulus File:

- a. In the "Virtuoso Schematic Composer Analysis Environment for Verilog-XL Integration" window select *Stimulus –> Verilog*.
- b. A dialog box appears as shown in Fig 6. Click No in the dialog box.

Tutorial Release Date: 02/12/2005



Fig 6: Dialog Box

c. A "Stimulus Options" form opens as shown in Fig 7. The default Mode is Copy.

Stimulus Options								
OK Cancel Apply		Help						
Mode 🐟 Select 🔷 Edit	Mode 🔷 Select 💠 Edit 🔶 Copy 💠 Delete							
Copy From:								
Directory /u/nahamit/	/cadence/inverter.run1							
File Name								
/ control pinMap raw si.env	View File							
Сору То:								
Run Directory	/u/nahamit/cadence/inverter.run1							
File Name	testfixture.new_							
Design Instance Path	test. top							
Current Test Fixture								
🔲 Make Current Test F	ixture 📃 Check Verilog Synt	ax						

Fig 7: Simulation Options

d. In Fig 7 select the verilog file generated using BitGen in the **File Name** text box. The verilog file selected can be seen in Fig 8.

r g		Stimulus Options		
OK Can	el Apply			Help
Mode 🔶 S	elect 🔷 Edit 🔶 (Copy 🔷 Delete		Ì
Copy From:				_
Directory	/u/nahamit/cade	nce		
File Name	textfixture.ver	ilod	Unix file	
	tech1.dpux test testing textfixture.ver tsmc25N.m%	ilog	View File	
Copy To:				_
Run Direc File Name Design Ins	tory stance Path	/u/nahamit/cadence/inverte testfixture.nev	er.runl	
Current Te	est Fixture			
🔄 🔲 Make (Current Test Fixture	e 📃 Chec	k Verilog Syntax	

Fig 8: Simulation Options (Verilog File selected)

- e. You can view the stimulus file by clicking **View File** in Fig 8.
- f. Click **OK** in Fig 8.
- g. Another "Stimulus Options" form opens as shown in Fig 9.

Simulation using Verilog XL

l 😥		Stimulus Options
OK Cano	el Apply	Help
Mode 🔷 S	elect 🔷 Edit 🔶 C	Copy 🔷 Delete
Copy From:		
Directory	/u/nahamit/cade	nce
File Name	textfixture.ver	ilog Unix file
	tech1.dpux test testing	View File
	textfixture.ver tsmc25N.m%	ilog
Copy To:		
Run Direct	tory	/u/nahamit/cadence/inverter.run1
File Name		testfixture.new
Design Instance Path		test.top
The User	Suppiled File is a	Stimulus 🖃
Current Te	est Fixture	
🔲 Make C	urrent Test Fixture	e 📃 Check Verilog Syntax

Fig 9: Simulation Options

- h. Click **OK** in Fig 9.
- i. In the "Virtuoso Schematic Composer Analysis Environment for Verilog-XL Integration" window again select *Stimulus -> Verilog*.
- j. The "Simulation Options" window opens as shown in Fig 10.

f @	Stimulus Options	Ξ.
ок	Cancel Apply	Help
Mode 4	🕈 Select 🔷 Edit 💠 Copy 🔷 Delete	
Directo	/u/nahamit/cadence/inverter.run1	
File Na		
	testfixture.new	
	View File	
Design	Instance Path test. top	
Curren	t Test Fixture	

Fig 10: Simulation Options

Portland State University

- k. In Fig 10 select textfixture.new and click **OK**.
- 1. Thus creating the netlist and the stimulus file is done.

5. Starting the Simulation:

a. In the "Virtuoso Schematic Composer Analysis Environment for Verilog-XL Integration" window click *Setup -> Record Signals* …. The "Record Signals Options" form opens up as shown in Fig 11.

ſø	Record Signals Options					
ОК	Cancel	Defaults	Apply	Help		
🔳 Sav	/e T	'op Level Pr	rimary I/O = in Simulation History File	shm. db		
Wavefo	rm Displa	y Package	Signalscan 🖃			
L				More >>		

Fig 11: Record Signals Options

b. In Fig 11 change the **Top Level Primary I/O** tab to **All Signals** as shown in Fig 12.

۲ <u>م</u>	Record Signals Options						
ОК	Cancel	Defaults	Apply	Help			
■ Sav Wavefo	ve rm Displa	All Sig y Package	jnals 🖃 in Simulation History File	shm. db			
1				More >>			

Fig 12: Record Signals Options

- c. In Fig 12 click **OK**.
- d. In the "Virtuoso Schematic Composer Analysis Environment for Verilog-XL Integration" window click *Simulation -> Start Interactive*. At this stage the stimulus and the netlists are compiled. If there are any errors in the netlist or the stimulus file then the errors show up. If everything is correct, then the stimulus file and the netlists are compiled and the window shows up as shown in Fig 13.

Tutorial Release Date: 02/12/2005

		Virtuoso® Schematic Composer Analysis Environment for Verilog-XL Integration	-
		amit3-inverter-schematic inverter.run1 test 0.00 ns Stopped 1	8
File	Setup	Stimulus Simulation Info Debug)
**	" I	For technical assistance please contact the Cadence Response Center at 1-877-CDS-4911 or send email to support@cadence.com	i.
×	►	For more information on Cadence's Verilog-XL product line send email to	
ÞI	Õ	talkv@cadence.com	
1 1	Ś.	Compiling included source file "testfixture.verilog" Continuing compilation of source file "/u/nahamit/cadence/inverter.runl/testfixture.template"	
	-11	Compiling source file "/pkgs/cadence-2000/ic4.45-new/local/lib/NCSU_Analog_Parts/nmos/functional/verilog.v" Compiling source file "/pkgs/cadence-2000/ic4.45-new/local/lib/NCSU_Analog_Parts/pmos/functional/verilog.v" Compiling source file "ibnl/cds0/netlist"	
ĸ	Å	Compiling source file "ihnl/cds1/netlist" SST2 Database Write API DWAPI Version 04.10-s014 06/16/2003	
Ì	\bigotimes	Copyright 1997-2002 Cadence Design Systems, Inc. Highest level modules:	
惫	۶	test	
۲	۲	Type ? for help C1 >	1
(Fe)	Ŧ	5	f
P	Str		1
			1

Fig 13: Compiling the netlists and the stimulus file

- e. In Fig 13 click *Simulation -> Continue*.
- f. Fig 13 now shows up as shown in Fig 14. Thus the simulation is complete.

Tutorial Release Date: 02/12/2005

	Virtuoso® Schematic Composer Analysis Environment for Verilog-XL Integration																	
									amit3-	-inverte	er-sche	matic	inverte	r.run1	test	0.00 ns	Inactive	e 18
File	Setup	Stimulus	Simulation	Info	Debug													Help
₽	*	For more	informatio	n on (adorna (a	Perileg	VI pro	duct	line	and an	sil to							
×	*	talkv@cad	lence.com	n on i	Jauence 5	veriiog	-ve bro	Juucc	. line s	senu em	Iaii Cu	,						
**	\bigcirc	Compiling source file "/u/nahamit/cadence/inverter.run1/testfixture.template" Compiling included source file "testfixture.verilog" Continuing commilation of source file "(u/nahamit/cadence/inverter run1/testfixture template"																
100 100 100 100		Compiling Compiling	Compiling source file "/pkgs/cadence-2000/ic4.45-new/local/lib/NCSU_Analog_Parts/nmos/functional/verilog.v" Compiling source file "/pkgs/cadence-2000/ic4.45-new/local/lib/NCSU_Analog_Parts/pmos/functional/verilog.v"															
	-4-5	Compiling SST2 Data	Compling source file "ihnl/cdsU/netlist" Compiling source file "ihnl/cds1/netlist" SST2 Database Write API DWAPI Version 04.10-s014 06/16/2003 Copyright 1997-2002 Cadence Design Systems, Inc.															
	×.	Copyright																
Carrier	S	test	highest level modules: test															
A	<u>&</u>	Type ? fo C1 >	or help															
.		O simulat CPU time:	O simulation events (use +profile or +listcounts option to count) + 12 accelerated events CPUL time: 0.3 secs to commile + 0.1 secs to link + 0.0 secs in simulation															
<u>I</u>	C.	End of To	ool:VERILOG	-xl04.	10.008-s	Dec 26	6, 2004	4 01:	:02:17									Y
M	Gr	Verilog-XL																
		-		_				_				_		_	_		_	

Fig 14: Simulation is complete

6. Viewing the Waveforms:

- a. In Fig 14 click on the **View Waveforms** button. (This is the button which is enabled in Fig 14 in lower left corner).
- b. On clicking **View Waveforms** a Logic Verification tool **SimVision** opens as shown in Fig 15.

Simulation using Verilog XL

					De	sign Bı	rowse	r 1 –	SimVi	ision						
<u>F</u> ile <u>I</u>	<u>E</u> dit	<u>V</u> iew	<u>S</u> elect	<u>W</u> ind	ows										ŀ	<u>H</u> elp
~	9	\$°2	c* []		b						Q	•		•	-	-
Scope	Tree:				Ş,	Signal	ls/Varia	ables (ofscop	e:					•	Д,
									_		_		_			
Filter:	×				-		B	3	•		POP	Filte	er: 🖹			-
0														0 object	ts sele	cted

Fig 15: SimVision

c. In Fig 15 click *File -> Open Database*. The "Open Database" window opens as shown in Fig 16.

ſ	Open Database							
<u>D</u> irectory:	/u/nahamit/cadence 🛁							
 . abstra . amit-1 . amit-2 . amit-1 . amit-1 . amit-1 . amit-1 . amit-1 . hld 	act	(((() () () () () () () () () () () (
File <u>r</u>	<u>n</u> ame:	<u>O</u> pen						
Files of	f type: Transition Files (*.trn)	<u>C</u> ance						

Fig 16: Open Database

Simulation using Verilog XL

d. In Fig 16 browse to .../inverter.run1/RunObject.0/shmDir/shm.db/ and select the file *shm.trn* as shown in Fig 17.

	Open Database	
<u>D</u> irectory:	/u/nahamit/cadence/inverter.run1/RunObject.0/shmDir/shm.db	- 1
🗈 shm. trn	l	
N		
File <u>i</u>	name: shm.trn	<u>O</u> pen
Files of	f type: Transition Files (*.trn)	<u>C</u> ancel

Fig 17: Selecting the file

- e. In Fig 17 click **Open**.
- f. Now the "SimVision" window appears as shown in Fig 18.



Fig 18: SimVision after opening database

g. In Fig 18 the circuit hierarchy can be seen as shown in Fig 19.

r ∑ Des	sign Browser 1 – SimVision
<u>File Edit View Select Windows</u>	Help
🛎 🗐 🔍 🗠 🖉 🖬 👘	🔃 - 🧱 - 💼 -
Scope Tree:	Signals/Variables of scope:
□ test □ top □ 10 □ M0 □ M1 □ 11 □ M0 M1	
Filter: *	⊷2 ⊷ ⊷ ⊷ ⊷ ⊷ <th< td=""></th<>
8	0 objects selected

Fig 19: Hierarchy of the circuit

h. In Fig 19 click on test and the signals on the top level of the circuit appear as shown in Fig 20.



Fig 20: Hierarchy of the circuit Portland State University i. In Fig 20 select the signals to be viewed and click on the button circled in Fig 21.

r ∑ Des	sign Browser 1 – SimVision
<u>File Edit View Select Windows</u>	Help
൙ 🗍 🖓 🗠 🗠 🛛 🏥 🖡 🗎	🔯 (🚟) 💼 · 🖬 ·
Scope Tree: 🔍	Signals/Variables of scope: shm::test
shm test Image:	tin tip out
Filter:	
ම	2 objects selected

Fig 21: Simvision

j. Now "Waveform 1 – Simvision "window opens as shown in Fig 22

		Wavef	orm1−SimVisi	on		
<u>F</u> ile <u>E</u> dit <u>V</u> iew Explore	For <u>m</u> at <u>W</u> indows					<u>H</u> elp
🗃 🖌 🗠 🗍 👪	🗈 × 548 🛚	🕦 🤤 🛛 🚺	🗟 🔒 🗖	F I 😽 F0	RE 🕂 Q	• 🚟 • 💼 • 🔲 •
Search Names:	🛚 🍂 🦓 Search T	imes: Marker 🔻		(<mark>]</mark>)		
x1 TimeA 💌 = 0	- ns 🗉 🛝 🛛 📩	76		Time Ranı	ge: 0 : 1124ns	
Baseline = 0 Cursor-Baseline = 0	Baselin TimeA	e = 0 = 0		1		
_	Cursor - 0	100ns 200ns	300ns 400ns	500ns 600ns	700ns 800n	s 900ns 1000ns 🕨
in in	o					
en out	0	_				
						_
8						2 objects selected

Fig 22: Waveform 1- Simvision

k. The waveforms at all the hierarchies of the circuit can be viewed in a similar way.