

Department of Electrical & Computer Engineering

ModelSim Tutorial

Created by:

Dr. Subbarao Wunnava Ph.D, P.E Vivek Jayaram

Florida International University

ModelSim_® Advanced Verification and Debugging

ModelSimSE 6.0c Copyright Mentor Graphics Corporation 2005

Graphics

Software versions

The following documentation supports ModelSim SE 6.0c for Microsoft Windows NT/2000/XP.

Graphical User Interface

ModelSim's graphical user interface controls all internal-window features including menus, buttons, and scroll bars. All of these operations can also be performed using command line programming.

Standards supported

ModelSim VHDL supports IEEE 1076-1987 and 1076-1993 VHDL, the 1164-1993 *Standard Multivalue Logic System for VHDL Interoperability*, and the 1076.2-1996 *Standard VHDL Mathematical Packages* standards. Any design developed with ModelSim will be compatible with any other VHDL system that is compliant with either IEEE Standard 1076-1987 or 1076-1993.

Software Download:

1. ModelSim evaluation media can be requested from the following web link: http://192.94.38.221/model/media_request/default.asp

2. ModelSim software can also be downloaded from the following web link: http://www.model.com/products/60/downloads.asp

1. To open ModelSim SE in VLSI Lab:

Start ModelSim SE by clicking on the shortcut on the Desktop as shown in the figure:



Upon opening Model*Sim* for the first time, welcome to Model*Sim* dialog box will be seen. (If this screen is not available, it can be enabled by selecting **Help > Welcome Menu** from the Main window. It will then display the next time you start Model*Sim*.)

M IMPORTANT Information	
Model Sim.	Welcome to ModelSim 6.0c
IMPORTANT Info	rmation!
• Kev Information	n
ModelSim platfor	n changes
• Product Change	S
Changes in Model	Sim functionality
• New Features	
The latest ModelS	<u>Sim features</u>
ModelSim 6.0 Ap	plication Notes
Taking Advanta	age of SystemVerilog
The ModelSim	Debug GUI
SystemC Verifi Comparing PSI	cation L and OVL
C Don't show this dialog again	Select Jumpstart to Jumpstart M Close

• From the above dialog box, click on *Jumpstart* to go the following window or click *Close* to continue to the ModelSim Main window.

- Create a project in ModelSim.
- Open an existing or example project in ModelSim.
- Open ModelSim documentation.

Select one of the above operations and then click in *close* to close this window.

Note: Without closing this window, further operations cannot be continued!

🕅 Welcome to ModelSim

Welcome to ModelSim

Create a Project

ModelSim Projects contain simulation details like compile settings, source files, or libraries. The Project Manager also lets you include references to shared global files. Start here to create a new ModelSim project.

Open a Project

Your most recent project is opened when you start ModelSim. Start here if you'd like to open an older project.

Open Documentation

Start here if you'd like to open the ModelSim PDF bookcase. The entire doc set is indexed for searching within Acrobat Reader.

Close

2. Create a Project Work Place:

a. Click on "File", then "New", Select "Project".



- b. In the dialog box thus obtained, for "Project Name", we will use "gates" as the example.
- c. In the "Project location" box, click on "Browse" and using down arrow, choose your directory where this project is to be saved.

M Create Project		×
Project Name		
gates		
Project Location C:/Vivek/Circuits	Br	owse
Default Library Name		
	ок	Cancel

Upon selecting OK, if the Project directory does not exist, dialog box appears asking whether to create a directory. Click OK to proceed to go to the following window:

Crea	e Project 🛛 🔀
2	The project directory does not exist. OK to create the directory?
	OK Cancel

In Add items to the Project

window, you can:

- Create New File
- Add Existing File
- Create Simulation
- Create New Folder

Select **Create New File**, to create a new project file. The **Create Project File** dialog box appears as follows, where the **File name** and **File type** has to be specified.



We will use **and_2_gate** as the file name and **VHDL** as the file type as an example.

Click on **OK**, and don't forget to Close the Add items to the project dialog box.

File Name	
and_2_gate	Browse
Add file as type	Folder
VHDL	Top Level 🗾

Then, the main window with **Project** and **Library** tabs will be seen as below. Also notice that the Project name: *gates* is shown in the status bar below the Workspace.

<u>File Edit View Format Compile Simulate</u>	<u>A</u> dd <u>T</u> ools <u>W</u> indow <u>H</u> elp	
🛛 🍪 🖽 🎜 🕅 🎢 Under % 🦳 1 🗄		tains:
☐ ≥ ☐ 4 ¥ % 6 2 2	M 8: 93	
Workspace 🔤 🖬 🗗 🗶	HIC: //ivek/Circuits/and 2 gate vhd	
Name Statu: Type Orde	1n #	
wh and 2_gate whd WHDL 0		
	I Program Ed	ltor
	2 Write Prog	rams nere
	H and_2_gate.vhd	
个	Assertions 🚽 🖻 🕺	Functional Coverage
	Vame Failure	▼ Name
.1		
		4
Project Library	a	Total Coverage: 09
		li -
ranscript		
ModelSimal		
	↑	
Project : gate <no design="" loaded=""></no>	<no context=""></no>	
		11040

design units. It can be hidden or displayed and allows to enter command-line by selecting View > Workspace

3. Write Program

Now, double click on the **and_2_gate.vhd** to see a source file at the we have to open the source window, where we write our program. In order to open the source file, Click on "File", then "New", Select "Source" and in that Select either "VHDL" or "Verilog" depending on the language you are writing program. Let us consider "VHDL" for our example.



4. Write source code

Upon clicking on "VHDL", we get to see that a new window by name "source_edit" is opened. It also shows thefile name as "new.vhd"



We use this window as the program editor and write codes.

5. Save the source file

Upon writing the code, we should save the source file by clicking "File" and selecting "Save as" we get a window in which type source file name in the project directory. In our example, the project directory is "circuits" and source file is saved as "and2.vhd". Remember to type the extension name as ".vhd" for VHDL files, and ".v" for Verilog files.

🛐 source_edit - and2.vhd	
File Edit Object Option	ns Window
	ΤΗ Ø
<u>N</u> ew •	
<u>O</u> pen	
<u>U</u> se Source	
Source <u>D</u> irectory	
<u>P</u> roperties	ight (c) 2004 - VLSI Laboratory, ECE Departm
Save	IGHTS RESERVED.
Save <u>A</u> s	ntire noticed above must be reproduced on al 토
Compile	▶ <i>//.</i>
<u>C</u> lose	

6. Compilation

Click on "File" and select "Compile" and in the dialog box thus obtained select the "and2.vhd" which is to be compiled and press "Compile". After completion of compilation, press " Done".

🖪 source_edit - and2.vhd	
File Edit Object Option	ns Window
	ři Ö
<u>N</u> ew	
<u>O</u> pen	
<u>U</u> se Source	
Source <u>D</u> irectory	
<u>P</u> roperties	ight (c) 2004 - VLSI Laboratory, ECE Departm
Save	IGHTS RESERVED.
Save <u>A</u> s	ntire noticed above must be reproduced on al 🗾
Comoile	▶ <i>Ii</i> .
<u>C</u> lose	

7. Error correction

If the program has any errors, you will see "red comments" showing "Errors" in the command window.



Click on the Error message and it will direct you to the error line it the source file.



Correct the error's and Compile again following the same steps as before.

8. Load design for simulation

After the source file is compiled successfully without any compilation errors, Click on the Library tab and you'll see the compiled design units listed. If you don't see the design units, make sure the Library: field shows "work."

🕅 ModelSim PE 5.5					_ 🗆	×
<u>F</u> ile <u>E</u> dit <u>D</u> esign <u>V</u>	jew <u>P</u> roject	<u>R</u> un	<u>C</u> ompare	<u>M</u> acro	<u>O</u> ptions	
<u>W</u> indow <u>H</u> elp						
🕸 🚘 🖻 🛍 💵	0 +	li i:	14 🕺	<u> i</u> , i Q,		
	💐 03 Mar 9 20	101				
Library: work	# Loading	packag(e standard s stal logio	1164		
⊞ <mark>E</mark>) andgate						
# Compiling architecture behavioral of andgate						
# ~ Compiling architecture behavioral of andgate test, benc						
Project Library	# Loading	entity ar	idgate			-
Project : gates	<no design="" l<="" th=""><th>oaded></th><th>></th><th></th><td><no context<="" td=""><td>1.</td></no></td></no>	oaded>	>		<no context<="" td=""><td>1.</td></no>	1.

In our example, we see an entity "andgate" in the Library field.

Next up in this exercise is to load one of the design units. Double-click on the "andgate" which is in the library field to load into simulation.

You'll see a new page appear in the workspace that displays the structure of the *andgate* design unit.

Clicking on "Design" and selecting "Load Design" can also do this step of loading designing unit



Select the required architecture that has to be loaded for simulation and press "Add" and then click on "Load". In our example click on "behavioral".

🕅 Load Design				_ 🗆 ×
Design				
Library: work				•
□ <mark>E</mark>) andgate A) behavioral				
Simulate work.andgate(behavioral)		Ad	d Simula	tor Resolution
	Load	Exit	Save	Cancel

The Load Design dialog box lets you select the library and top-level design unit to simulate. You can also select the resolution limit for this simulation. By default, the following will appear for this simulation run:

- Simulator Resolution: default (the default is 1 ns)
- Library: work
- Design Unit: andgate(behavioral)

If the Design Unit is an entity (like **andgate** in this design), you can click on the plus-box prefix to view any associated architectures as there can be more than one architecture for a single entity.

9. View Signals Window

Next, Click on "View" and select "Signals" to open the signals window.



10. Assign values to the signals

From the Signals window, click on each signal and assign values for each signal by clicking on "Edit" and select "Force". You can also use "Clock" if you want to give clock input to the selected signal.

			🟪 signals		
🟪 signals	_ 🗆 ×	File	Edit View Window		
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>W</u> indo	w				
	1		<u>С</u> ору		
a	U		<u>S</u> ort	<u> </u>	
Ь Р	U		Select <u>A</u> ll		
C C	U		<u>U</u> nselect All		
			<u>E</u> xpand Selected		
			Co <u>l</u> lapse Selected		
			E <u>x</u> pand All		
			Collapse All		
		sim:	F <u>o</u> rce	//,	
			<u>N</u> oForce		
			Cloc <u>k</u>		
			Justify Values	•	
sim:/andgate	1.		<u>F</u> ind		

Force Selected Signal 🛛 🗙
Signal Name: sim:/andgate/a
Value: 1
Kind
Freeze O Drive O Deposit
Delay For: 0
Cancel After:
<u> </u>

You can force either a 1'' or a 0'' for the selected signals as shown.

11. View signals in the waveform window

To view signals in the waveform window, select signals to be displayed in the waveform window and click on "View" and select "Wave" and click on "Selected Signals". This will display selected signals in the waveform window.



You can perform the same operation by opening waveform window by clicking on "View" and then selecting "Wave" in the main ModelSim command window and then by dragging and dropping the signals onto the waveform window.

12. Running the Simulation

Now you will exercise two different **Run** functions from the toolbar buttons on either the Main or Wave window. (The **Run** functions are identical in the Main and Wave windows.)



Select the **Run** button first. This causes the simulation to run and then stop after 100 ns. You may change the run time in the Main window.

elle Edit Qursor Z	oom Compare Book	whork Figmat Window
	BIANE	
/andgate/a /andgate/b /andgate/c	1 0	
	200 no	570 1
	0 rss	
O ns to 1 us		a de la companya de la

You may change values for the signal inputs and run to see different outputs.

뺖 wave - default				
<u>F</u> ile <u>E</u> dit <u>C</u> ursor <u>Z</u> oom	Co <u>m</u> pare <u>B</u> ookr	nark F <u>o</u> rmat <u>V</u>	⊻indow	
☞◼▰▏▓▆▆▏▙▓▕▙ਤ▏QQQQ ₽▌▋₽₽₩				
			Restart	
📕 /andgate/a	1			
/andgate/b	1			
/andgate/c	1			

Select the Restart button on either the **Main** or **Wave** toolbar to restart the waveform simulation and to give different values to the signals from the beginning.

13. Save Project

To save the project, click on "File" and select "Save" and select the directory in which the project is to be saved.