# ModelSim v5.7 Quick Reference Guide

Author: David M. Sendek

**Background:** This guide provides you with step-by-step procedures in using ModelSim to compile, link and simulate your VHDL or Verilog source code design. *These procedures provide you with the necessary steps to accomplish labs when using VHDL.* 

### Assumptions:

- 1. Students have a basic understanding of UNIX
- 2. Students know how to sign-into a UNIX environment from Windows NT

References: (a) "Force File and Force Commands", David Sendek, 9 July 2003

- (b) ModelSim SE Tutorial Version 5.5e, 22 Aug 01, Mentor Graphics
- (c) Lesson 2 Basic VHDL Simulation, "Preparing the Simulation", Mentor Graphics, 22 Aug 01, pages T-18 to T-23

### **SETTING UNIX ENVIRONMENT** (steps at the UNIX prompt)

- 1. (Suggestion) Create a directory ee451
- 2. (Suggestion) Create a sub-directory for each lab, ex: lab1, lab2, ...
- 3. %setenv MGC WD \$HOME/path
  - ex: %setenv MGC\_WD \$HOME/ee451/lab1
- 4. Change to your working lab sub-directory
- 5. %source /mentor/csu\_setup/ams\_setup.csu

Notes:

- This permits you to access ModelSim, used for compiling, linking and simulating VHDL source code.
- If you do not modify your .cshrc file, you must repeat steps 3 through 5 each time you re-open your UNIX window.
- There is a **space** between %source and /mentor....

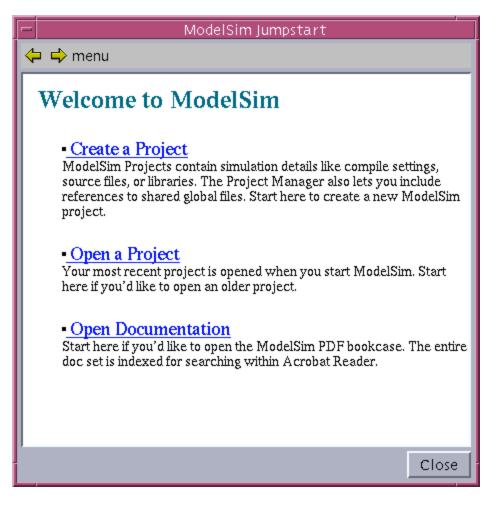
#### **Procedures:**

- 1. Using your favorite text editor, create the Very High Speed Integrated Circuit (VHSIC) Hardware Description Language (VHDL) source code for your design and a separate file with "force" commands to be used in simulating your design (if required). The force file is a set of test vectors to test your design. Reference (a) provides some force commands. Name the file such as "lab1.do"
- 2. From your /ee451/lab1 directory and at the Unix Prompt, start Mentor Graphics ModelSim by entering the following at the Unix prompt: qhsim

3. At the "Welcome to ModelSim 5.7" window, select "Jumpstart".



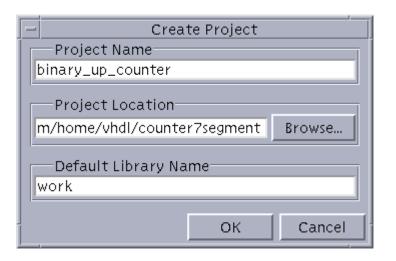
The following screen will be displayed. Select "Create a Project" or File > New > Project. Note: After you have initially created a project, future access to that project can be achieved by selecting "Open a Project".



The following will be displayed <u>if you have already created a project</u> and want to open it at some later time. This will add the files that contain your design. You can then directly proceed to compiling your design.

F	- Open Project
	Open Project
	m/home/vhdl/counter7segment/binary_up_counter
	Browse
	OK Cancel

4. Before you compile any Hardware Design Language (HDL) (such as VHDL or Verilog), you'll need a design library to hold the compilation results. To create a new design library, type in the Project Name and select a directory where the project file will be stored. Leave the default library name set to "work". Finally, select OK.



After pressing OK, the following will be displayed:

-	Add items to the	Project
Г	—Click on the icon to add it	ems of that type:
	۲ Create New File Ac	dd Existing File
	Create Simulation Cre	ate New Folder
		Close

And your ModelSim main window will look like the following:

		•
E	<u> E</u> ile <u>E</u> dit <u>V</u> iew <u>C</u> ompile <u>S</u> imulate <u>T</u> ools <u>W</u> indow	<u>H</u> elp
	🔓 🖻 🖀 🖉 🥩 🎬 🚑 🐹	
ļ	Workspace	
ŀ	Name Status Type Ord # // ModelSim SE Viewer 5.7c Mar 1 unOS 5.9 # //	2003 \$
	# // Copyright Model Technology, a Graphics Corporation company, 2003	Mentor
	# // All Rig served.	hts Re
	# // UNPUBLISHED, ED SOFTWARE.	
	# // CONFIDENTIAL AND PROPRI INFORMATION WHICH IS THE	
	# // PROPERTY OF MENTOR GRAPH RPORATION OR ITS LICENSORS.	ICS CO
ŀ	# // # Loading project binary_up_counter	
ľ	Project Library	
P	Project : binary_up_counter <a>No</a> No Context>	//,

5. The next step is to add the files that contain your design. By now you would have written your VHDL source code file(s). Click "Add Existing File" in the "Add items to the Project" window. For this example, three VHDL files are added. Click the "Browse" button in the "Add file to Project". It should be noted that counter7segment.vhd, counter.vhd & leddcd.vhd are selected. Select "Reference from current location" and then click OK. Close the "Add items to the Project" dialog (This will close the "Add items to the Project" and "ModelSim Jumpstart" windows).

F	Add file to Project
Г	File Name
	/ECE/sendekdm/home/vhdl/counter7segment/leddcd.vhd Browse
	Add file as type Folder Top Level
•	Reference from current location 🛛 💠 Copy to project directory
	OK Cancel

The main ModelSim window will show the following:

-	Мо	delSim SE	VHDL 5.7c	•
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>C</u> ompile	e <u>S</u> imulate	e <u>T</u> ools	Window <u>F</u>	<u>t</u> elp
🖻 🖻 🛍 🕹 🕮 🚑 🕷				
Workspace		×		
Name	Status Ty	pe Oro	ED SOFTWARE. # // CONFIDENTIAL AND PROPRIETARY	$ \Delta $
💾 counter7segment.vhd	? VH		INFORMATION WHICH IS THE	
H counter.vhd	? VH		# // PROPERTY OF MENTOR GRAPHICS CO RPORATION OR ITS LICENSORS.	
H leddcd.vhd	? VH	DL 2	# //	
	(	N	# Loading project binary_up_counter # Loading project binary_up_counter	
Project Library			ModelSim>	$\mathbf{N}$
Project : binary_up_counter	gn Loaded>		No Context>	

6. Now you are ready to compile the VHDL or Verilog file(s). To compile the file(s), select Compile > Compile All or highlight the "high-level" design (counter7segment.vhd in this example), right mouse click and select Compile > Compile All. By selecting the "Library" tab, you can observe the compiled design units (the Entity & Architecture blocks).

			M	odelSim SE	VHDL 5.7c	•
<u>F</u> ile <u>E</u> dit <u>V</u> iew	<u>C</u> ompile <u>S</u> i	mulate	<u>T</u> ools	<u>W</u> indow		<u>H</u> elp
🛛 😅 🖻 🛍 🛛 🕸 🗄	1 🚑 🕺					
Workspace ====				<u>×</u>	( _ <del>= </del>	
ime	Status	Туре	Order	Modifie	# // PROPERTY OF MENTOR GRAPHICS CORPORATION OR ITS LICENSORS.	$\Box \Delta$
📊 counter7segmen	it.vhd 🧹	VHDL	0	07/22/03	# //	
🖬 counter.vhd		VHDL	1	07/22/03	# Loading project binary_up_counter	
📊 leddcd.vhd	×	VHDL	2	07/22/03	<pre># Loading project binary_up_counter # Compile of counter7seqment.vhd was successful.</pre>	- 110
					# Compile of counter.vhd was successful.	
				$\vdash$	# Compile of leddcd.vhd was successful. # 3 compiles. 0 failed with no errors.	
Project Library	<u>ا</u>				ModelSime	$\nabla$
Project : binary_up_counte	er 🛛 < No Design Loa	ded>		<no conte<="" th=""><th>xt&gt;</th><th></th></no>	xt>	

Mod	delSim SE VHDL 5.7c 🛛 🔤
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>C</u> ompile <u>S</u> imulate <u>T</u> ools <u>V</u>	<u>√</u> indow <u>H</u> elp
🔁 🖻 🖺 🔌 🕮 🚑 🐹	
Workspace	
Name     Type     Path       Image: Straight of the s	nts/( # // UNPUBLISHED, LICENSED SOFTWAR E. # // CONFIDENTIAL AND PROPRIETARY INFORMATIO
Eleddcd Entity /top/stude     Architecture     Project] Library	<pre>nts/(</pre>
Project : binary_up_counter No Design Loaded>	<no context=""></no>

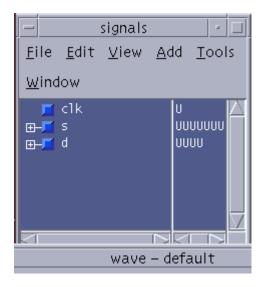
- 7. You may find compile errors in the previous step. These need to be corrected before proceeding any further.
- 8. Now the design must be loaded, once compiled. Double-click the "high-level" design (counter7segment) on the Library page. You'll see a new page appear in the workspace that displays the structure of the counter7segment design unit.

ModelSim SE V	/HDL 5.7c 🛛 🕴
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>C</u> ompile <u>S</u> imulate <u>T</u> ools <u>V</u>	<u>M</u> indow <u>H</u> elp
🔁 🛍 🕄 🤣 🔛 🚑 🐹 🛛 EF 100 fs 🕏 EL EL	±↓ 1 0 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
Workspace 🔤 🖬	
Instance Design Unit Design Unit Ty; 	vsim work.counter7segment # vsim work.counter7segment # Loading /mentor/AMS/modelsim/v5.7c/bin/. ./sunos5//std.standard
☐ ☐ leddcd1 leddcd(behArchitecture ☐ std_logistd_logicPackage	<pre># Loading /mentor/AMS/modelsim/v5.7c/bin/. ./sunos5//ieee.std_logic_1164(body) # Loading /mentor/AMS/modelsim/v5.7c/bin/.</pre>
ばいたい 「「「「」」」」」」」」」」」」」」」」」」」」」」」」」」」」」」」」	<pre>./sunos5//ieee.std_logic_arith(body) # Loading /mentor/AMS/modelsim/v5.7c/bin//sunos5//ieee.std_logic_unsigned(body) # Loading work.counter7segment(structural)</pre>
Project Library sim Files	<pre># Loading work.counter(behavioral) # Loading work.leddcd(behavioral) VSIM 5&gt;  </pre>
Project : binary_up_counter Now: 0 fs Delta: 0	sim/:counter7segment

9. To simulate the design, select View > All Windows from the main ModelSim window. This will open all the ModelSim simulation windows, such as the wave window. This window will be what you will predominately use.

ModelSim SE VHDL 5.7c	source – counter7segment.vhd 🔹 🗆
<u>Eile E</u> dit <u>V</u> iew <u>C</u> ompile <u>S</u> imulate <u>T</u> ools <u>W</u> indow	Help File Edit View Tools Window
Image: Standard Standard Package         Image	Image: Second
# .source .process .signales .dataflow .list .wave         Project Library sim Files         VSIM 6>	als .variab
Project : binary_up_counter Now: 0 fs Delta: 0 sim/:counter7segment	
dataflow	sim/counter7segment
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>N</u> avigate T <u>r</u> ace <u>T</u> ools <u>W</u> indow	<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>Insert Format T</u> ools <u>W</u> indow
Ø   N □ ⊕   X ▷ □ 2 ○ M   J + 4 + > > + + +   Ø Ø Ø 0   0 0 0   10 0	☞ ■ ●   ½ ʰ ʰ ħ   \ X ½ ± 1   \ <u>□</u>   Q Q Q <b>K</b>   ¥   ¥   ↓ ¥ ¥ X   3+
Welcome to the Enhanced Dataflow Window! WARNING: No extended dataflow License exists	Now         0 fs         500         1           Cursor 1         0 fs         0 fs         0

10. From the Signals window menu, select Add > Wave > Signals in Region. This command displays the top-level signals to the Wave window.



This will result in the following:

-	wave – default	
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>I</u> nse	ert F <u>o</u> rmat <u>T</u> ools <u>W</u> indow	
📂 🖬 🎒   🐰 🖻 🛍 11 14 24   3+	) MA   _k 🕺 TE I   N 💽   @, Q, Q, 🗮   EF   E.	Į
☐ :counter7seg ⊞—☐ :counter7seg ⊞—☐ :counter7seg	000000	Z
Now	0 fs ) 500 1	
Cursor 1	0 fs 0 fs	
0 fs to 1 ps		11.

- 11. Of particular interest, if you are using a hierarchical design, the previous step only show the top-level signals. But if you require more detailed resolution, you can select from the signals window Add > Wave > Signals in Design.
- 12. Now you are ready to simulate your design. To simulate your design, you will use your test vectors that you placed in your force file. To execute your force file, to simulate your design, select Tools > Execute Macro from the main window menu. Finally, select the force file you created earlier. Note that a "force file" is also referred to as a "do" file.

-		Execute Do File	
D	irectory: 📝	top/students/GRAD/ECE/sendekdm/home/vhdl/counter7segment	- È
	] work ] test.do		
E	1		
	File <u>n</u> a	ame: test.do	<u>O</u> pen
	Files of <u>t</u>	type: Macro Files (*.do,*.tcl)	<u>C</u> ancel

A sample simulation may look like:

-	wave – default	
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>I</u> nsert	F <u>o</u> rmat <u>T</u> ools <u>W</u> indow	
S ■ ④   X □ □ ■ ▲   S □ ↓ ■ ▲   X   3+	📐 💥 🗠 🛨   🔪 🖳   🍳 🤍 🌉   👫   🗄	<u>.</u>
<pre>:counter7segment:clk</pre>	0       1110111       1110111       1110111         1       1110111       1110111       1       1         1       1       1       1       1       1       1       1         1	X
Now	00000 fs	
Cursor 1		
199518 fs to 199582 fs		

- 13. To exit ModelSim, select File > Quit in the main ModelSim window.
- 14. After simulation, you can use the XESS Field Programmable Gate Array (FPGA) protoboard to verify your design on physical hardware.

## **Useful Feature(s):**

1. If you desire to clear your "wave" window during simulation, simply press the restart icon illustrated below.

