

SynthWorksSM

VHDL Training Solutions for hardware design and test.

ModelSimTM Quick Reference

1. Steps to Running Modelsim

Start the Simulator
Set the Working Directory
Create a Work Library
Compile a Design into the Library
Simulate the Testbench
Display Waveforms
Run and Stop a Simulation
Debugging

In the following, **Menu** = Modelsim Menu and **Command** = Modelsim command / transcript window.

2. Start the Simulator

Windows: Click on ModelSim Icon
UNIX: vsim -i &

3. Set the Working Directory

Windows: Set properties on Modelsim command
UNIX: Start tool in desired directory
Menu: File > Change Directory

Note: cannot change directory if simulation running, use:
Menu: Simulate > End Simulation

4. Create a Work Library

Menu: File > New > Library
Command: vlib work -- Library name = work
vmap work work

5. Compile a Design into the Library (Analyze)

Menu: Compile > Compile -- Select file in pop-up
Command: vcom Inc.vhd -- file name = Inc.vhd

All VHDL files in a design must be compiled into a library.
Compile in a bottom up order.

6. Simulate the Testbench (Elaborate)

Workspace Library Tab:
Click on "+" in front of work in the workspace library tab and double click on the testbench.

Command: vsim TheTb
Where TheTb is the Testbench Entity or Configuration name.

Time resolution may be specified while starting a simulation.
See 10.2 Simulator Time Resolution.

7. Display Waveforms

Command: add wave -r * -- all signals in design
Can also use instance path names through design.

Workspace Sim Tab: One Step
Right click on a design name in the workspace sim tab and select "Add > Add to Wave".

Sim Tab + Object Window: Three Steps
Main Menu: View > Debug Windows > Objects
Workspace Sim Tab: Select design
Objects Menu: Select signals and
right click: Add to Wave > Selected ...

8. Run and Stop a Simulation

8.1 Run for Specified Time

Command: Run 100 ns
Run 100 -- in time base units

Menu Two Steps
Buttons Specify Run Time in Box
Menu: Simulate > Run > Run XXX ns

8.2 Stopping

Menu: Simulate > Break

8.3 Restarting a Simulation

Menu: Simulate > Run > Restart
Command: restart -f

8.4 Run All

Menu: Simulate > Run > Run -All
Command: run -all

Stop simulation with: report "Done" severity failure ;

9. Debugging

9.1 Finding Signal Changes

Waveform Menu: Edit > Search

9.2 Using the Waveform Viewer

Detach waveform viewer by pressing the box with the arrow going out (top left corner of waveform portion of window).

9.2.1 Zooming

Waveform Menu: View > Zoom > ...
Choice of the following: Full, Last, In 2x, Out 2x, Range
Waveform window, use Right Mouse Button for zoom menu.
Click and drag with Middle Mouse Button to zoom to a specific area.

9.2.2 Adding Cursors

Waveform menu: Add > Cursor
Use find next change buttons to move cursor to change on the selected signal. Also drag cursor and it will try to snap to an edge of a selected signal.

Measure time of an event by using two cursors.

9.2.3 Specifying Signal Radix (Hex, Unsigned, ...)

Select signal(s) and then:
Waveform menu: Format > Radix > Hex
Format > Radix > Unsigned
In the signal portion of the Waveform window, use Right Mouse Button to access the Format menu.

9.3 Setting Breakpoints

Workspace Sim Tab: Select design
Main Menu: View > Source
Source Window:
Set: Click on line of code
Delete: Click on line of code (toggle)

Alternately:
Menu: Tools > Breakpoints
Can set and delete line and signal breakpoints.

9.4 Single Stepping

Main Menu: Simulate > Run > Step
Simulate > Run > Step -Over
Simulate > Run > Continue -- after break

9.5 Forcing Values on Signals

Select design in Workspace Sim Window, right click on signal in objects window, and select Force.

10. Special Topics

10.1 Running Script Files

Menu: Tools > TCL > Execute Macro
Command: do <script_file_name>

10.2 Simulator Time Resolution

Command: vsim -t ns
Menu: Simulate > Start Simulation
Current value of Simulator Resolution = Default. Press on Default to change.

10.3 Turning Coverage on (step 1)

Command: vcom -cover all design.vhd
Cover also has individual switches.

10.4 Simulating with Coverage (step 2)

Command: vsim -coverage Cfg_Uart1_Rx

10.5 Creating Coverage Reports

Command: coverage report -file uart1.txt

10.6 Saving Coverage Results

Command: coverage save uart1.cov

10.7 Merging Coverage Results

Command: vcover merge to.cov f1.cov f2.cov f3.cov

10.8 Loading Coverage Results

Command: vsim -viewcov to.cov

11. Help Menu & Manuals

Menu: Help > Manuals >

© 1999 - 2007 by SynthWorks Design Inc. Reproduction of entire document in whole is permitted. All other rights reserved.

SynthWorks Design Inc.

VHDL Hardware Synthesis and Verification Training
11898 SW 128th Ave. Tigard OR 97223 1-(503)-590-4787
<http://www.SynthWorks.com> jim@synthworks.com

ModelSim is a trademark of Model Technology Inc.
SynthWorks is a servicemark of SynthWorks Design Inc.