



ModelSim® Tutorial

Software Version 6.6

**© 1991-2010 Mentor Graphics Corporation
All rights reserved.**

This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND 03/97

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Contractor/manufacturer is:

Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/terms_conditions/trademarks.cfm.

Table of Contents

Chapter 1	
Introduction	9
Assumptions	9
Before you Begin	9
Example Designs	9
Chapter 2	
Conceptual Overview	11
Basic Simulation Flow	11
Project Flow	12
Multiple Library Flow	13
Debugging Tools	14
Chapter 3	
Basic Simulation	15
Create the Working Design Library	15
Compile the Design Units	17
Load the Design	18
Run the Simulation	20
Set Breakpoints and Step through the Source	22
Chapter 4	
Projects	27
Create a New Project	27
Add Objects to the Project	28
Changing Compile Order (VHDL)	30
Compile the Design	31
Load the Design	32
Organizing Projects with Folders	33
Add Folders	33
Moving Files to Folders	35
Simulation Configurations	35
Chapter 5	
Working With Multiple Libraries	39
Creating the Resource Library	39
Creating the Project	41
Linking to the Resource Library	42
Verilog	42
VHDL	43
Linking to a Resource Library	44

Permanently Mapping VHDL Resource Libraries	45
Chapter 6	
Analyzing Waveforms	47
Loading a Design	48
Add Objects to the Wave Window	48
Zooming the Waveform Display	49
Using Cursors in the Wave Window	50
Working with a Single Cursor	50
Working with Multiple Cursors	52
Chapter 7	
Viewing And Initializing Memories	55
View a Memory and its Contents	56
Navigate Within the Memory	60
Export Memory Data to a File	62
Initialize a Memory	63
Interactive Debugging Commands	66
Chapter 8	
Automating Simulation	71
Creating a Simple DO File	71
Running in Command-Line Mode	72
Using Tcl with the Simulator	75
 Index	
 End-User License Agreement	

List of Examples

List of Figures

Figure 2-1. Basic Simulation Flow - Overview Lab	11
Figure 2-2. Project Flow	13
Figure 2-3. Multiple Library Flow.....	14
Figure 3-1. The Create a New Library Dialog.....	16
Figure 3-2. work Library Added to the Library Window	17
Figure 3-3. Compile Source Files Dialog	18
Figure 3-4. Verilog Modules Compiled into work Library.....	18
Figure 3-5. Loading Design with Start Simulation Dialog	19
Figure 3-6. The Design Hierarchy	20
Figure 3-7. The Object Window and Processes Window	20
Figure 3-8. Using the Popup Menu to Add Signals to Wave Window	21
Figure 3-9. Waves Drawn in Wave Window.....	22
Figure 3-10. Setting Breakpoint in Source Window	23
Figure 3-11. Setting Restart Functions	23
Figure 3-12. Blue Arrow Indicates Where Simulation Stopped.....	24
Figure 3-13. Values Shown in Objects Window	24
Figure 3-14. Parameter Name and Value in Source Examine Window	25
Figure 4-1. Create Project Dialog - Project Lab	28
Figure 4-2. Adding New Items to a Project.....	29
Figure 4-3. Add file to Project Dialog	29
Figure 4-4. Newly Added Project Files Display a '?' for Status.....	30
Figure 4-5. Compile Order Dialog.....	31
Figure 4-6. Library Window with Expanded Library	32
Figure 4-7. Structure(sim) window for a Loaded Design	32
Figure 4-8. Adding New Folder to Project	33
Figure 4-9. A Folder Within a Project.....	34
Figure 4-10. Creating Subfolder	34
Figure 4-11. A folder with a Sub-folder	34
Figure 4-12. Changing File Location via the Project Compiler Settings Dialog.....	35
Figure 4-13. Simulation Configuration Dialog	36
Figure 4-14. A Simulation Configuration in the Project window	37
Figure 4-15. Transcript Shows Options for Simulation Configurations	37
Figure 5-1. Creating New Resource Library	40
Figure 5-2. Compiling into the Resource Library	41
Figure 5-3. Verilog Simulation Error Reported in Transcript.....	43
Figure 5-4. VHDL Simulation Warning Reported in Main Window	44
Figure 5-5. Specifying a Search Library in the Simulate Dialog.....	45
Figure 6-1. Panes of the Wave Window	47
Figure 6-2. Zooming in with the Mouse Pointer	50
Figure 6-3. Working with a Single Cursor in the Wave Window	51

List of Figures

Figure 6-4. Renaming a Cursor	52
Figure 6-5. Interval Measurement Between Two Cursors.....	53
Figure 6-6. A Locked Cursor in the Wave Window	53
Figure 7-1. The Memory List in the Memory window	56
Figure 7-2. Verilog Memory Data Window	57
Figure 7-3. VHDL Memory Data Window	57
Figure 7-4. Verilog Data After Running Simulation.....	58
Figure 7-5. VHDL Data After Running Simulation	58
Figure 7-6. Changing the Address Radix.....	59
Figure 7-7. New Address Radix and Line Length (Verilog.....	59
Figure 7-8. New Address Radix and Line Length (VHDL)	60
Figure 7-9. Goto Dialog.....	60
Figure 7-10. Editing the Address Directly.....	61
Figure 7-11. Searching for a Specific Data Value.....	61
Figure 7-12. Export Memory Dialog.....	62
Figure 7-13. Import Memory Dialog.....	64
Figure 7-14. Initialized Memory from File and Fill Pattern	65
Figure 7-15. Data Increments Starting at Address 251	66
Figure 7-16. Original Memory Content.....	67
Figure 7-17. Changing Memory Content for a Range of Addresses**OK	67
Figure 7-18. Random Content Generated for a Range of Addresses.....	68
Figure 7-19. Changing Memory Contents by Highlighting.....	68
Figure 7-20. Entering Data to Change**OK	69
Figure 7-21. Changed Memory Contents for the Specified Addresses	69
Figure 8-1. Wave Window After Running the DO File.....	72
Figure 8-2. The counter.wlf Dataset in the Main Window Workspace.....	74

List of Tables

Assumptions

Using this tutorial for ModelSim™ is based on the following assumptions:

- You are familiar with how to use your operating system, along with its window management system and graphical interface: OpenWindows, OSF/Motif, CDE, KDE, GNOME, or Microsoft Windows XP.
- You have a working knowledge of the language in which your design and/or test bench is written (such as VHDL, Verilog). Although ModelSim is an excellent application to use while learning HDL concepts and practices, this tutorial is not intended to support that goal.

Before you Begin

Preparation for some of the lessons leaves certain details up to you. You will decide the best way to create directories, copy files, and execute programs within your operating system. (When you are operating the simulator within ModelSim's GUI, the interface is consistent for all platforms.)

Examples show Windows path separators - use separators appropriate for your operating system when trying the examples.

Example Designs

ModelSim comes with Verilog and VHDL versions of the designs used in these lessons. This allows you to do the tutorial regardless of which license type you have. Though we have tried to minimize the differences between the Verilog and VHDL versions, we could not do so in all cases. In cases where the designs differ (e.g., line numbers or syntax), you will find language-specific instructions. Follow the instructions that are appropriate for the language you use.

Chapter 2

Conceptual Overview

Introduction

ModelSim is a verification and simulation tool for VHDL, Verilog, SystemVerilog, and mixed-language designs.

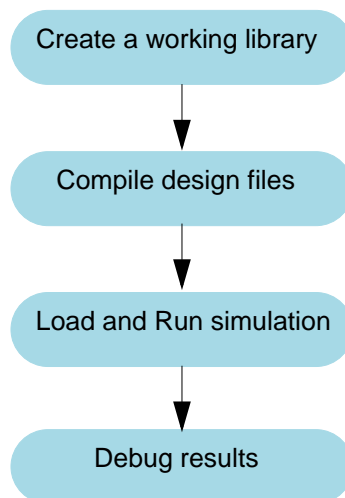
This lesson provides a brief conceptual overview of the ModelSim simulation environment. It is divided into four topics, which you will learn more about in subsequent lessons.

- Basic simulation flow — Refer to [Chapter 3 Basic Simulation](#).
- Project flow — Refer to [Chapter 4 Projects](#).
- Multiple library flow — Refer to [Chapter 5 Working With Multiple Libraries](#).
- Debugging tools — Refer to remaining lessons.

Basic Simulation Flow

The following diagram shows the basic steps for simulating a design in ModelSim.

Figure 2-1. Basic Simulation Flow - Overview Lab



- Creating the Working Library

In ModelSim, all designs are compiled into a library. You typically start a new simulation in ModelSim by creating a working library called "work," which is the default library name used by the compiler as the default destination for compiled design units.

- **Compiling Your Design**

After creating the working library, you compile your design units into it. The ModelSim library format is compatible across all supported platforms. You can simulate your design on any platform without having to recompile your design.

- **Loading the Simulator with Your Design and Running the Simulation**

With the design compiled, you load the simulator with your design by invoking the simulator on a top-level module (Verilog) or a configuration or entity/architecture pair (VHDL).

Assuming the design loads successfully, the simulation time is set to zero, and you enter a run command to begin simulation.

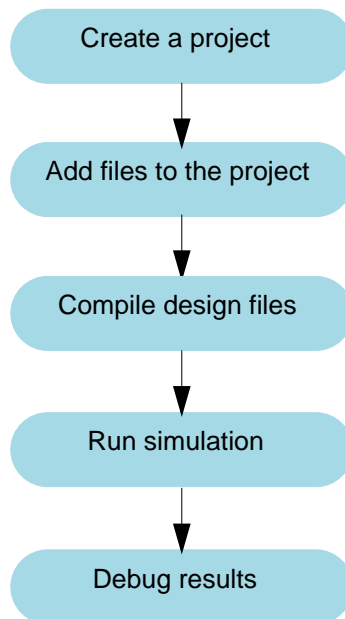
- **Debugging Your Results**

If you don't get the results you expect, you can use ModelSim's robust debugging environment to track down the cause of the problem.

Project Flow

A project is a collection mechanism for an HDL design under specification or test. Even though you don't have to use projects in ModelSim, they may ease interaction with the tool and are useful for organizing files and specifying simulation settings.

The following diagram shows the basic steps for simulating a design within a ModelSim project.

Figure 2-2. Project Flow

As you can see, the flow is similar to the basic simulation flow. However, there are two important differences:

- You do not have to create a working library in the project flow; it is done for you automatically.
- Projects are persistent. In other words, they will open every time you invoke ModelSim unless you specifically close them.

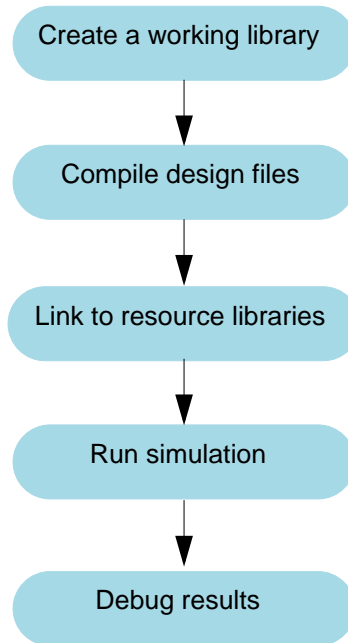
Multiple Library Flow

ModelSim uses libraries in two ways: 1) as a local working library that contains the compiled version of your design; 2) as a resource library. The contents of your working library will change as you update your design and recompile. A resource library is typically static and serves as a parts source for your design. You can create your own resource libraries, or they may be supplied by another design team or a third party (e.g., a silicon vendor).

You specify which resource libraries will be used when the design is compiled, and there are rules to specify in which order they are searched. A common example of using both a working library and a resource library is one where your gate-level design and test bench are compiled into the working library, and the design references gate-level models in a separate resource library.

The diagram below shows the basic steps for simulating with multiple libraries.

Figure 2-3. Multiple Library Flow



You can also link to resource libraries from within a project. If you are using a project, you would replace the first step above with these two steps: create the project and add the test bench to the project.

Debugging Tools

ModelSim offers numerous tools for debugging and analyzing your design. Several of these tools are covered in subsequent lessons, including:

- Using projects
- Working with multiple libraries
- Setting breakpoints and stepping through the source code
- Viewing waveforms and measuring time
- Viewing and initializing memories
- Creating stimulus with the Waveform Editor
- Automating simulation

Chapter 3

Basic Simulation

Introduction

In this lesson you will go step-by-step through the basic simulation flow:

1. [Create the Working Design Library](#)
2. [Compile the Design Units](#)
3. [Load the Design](#)
4. [Run the Simulation](#)

Design Files for this Lesson

The sample design for this lesson is a simple 8-bit, binary up-counter with an associated test bench. The pathnames are as follows:

Verilog – `<install_dir>/examples/tutorials/verilog/basicSimulation/counter.v` and `tcounter.v`

VHDL – `<install_dir>/examples/tutorials/vhdl/basicSimulation/counter.vhd` and `tcounter.vhd`

This lesson uses the Verilog files `counter.v` and `tcounter.v`. If you have a VHDL license, use `counter.vhd` and `tcounter.vhd` instead. Or, if you have a mixed license, feel free to use the Verilog test bench with the VHDL counter or vice versa.

Related Reading

User's Manual Chapters: [Design Libraries](#), [Verilog and SystemVerilog Simulation](#), and [VHDL Simulation](#).

Reference Manual commands: [vlib](#), [vmap](#), [vlog](#), [vcom](#), [view](#), and [run](#).

Create the Working Design Library

Before you can simulate a design, you must first create a library and compile the source code into that library.

1. Create a new directory and copy the design files for this lesson into it.

Start by creating a new directory for this exercise (in case other users will be working with these lessons).

Verilog: Copy *counter.v* and *tcounter.v* files from */<install_dir>/examples/tutorials/verilog/basicSimulation* to the new directory.

VHDL: Copy *counter.vhd* and *tcounter.vhd* files from */<install_dir>/examples/tutorials/vhdl/basicSimulation* to the new directory.

2. Start ModelSim *if necessary*.

a. Type **vsim** at a UNIX shell prompt or use the ModelSim icon in Windows.

Upon opening ModelSim for the first time, you will see the Welcome to ModelSim dialog. Click **Close**.

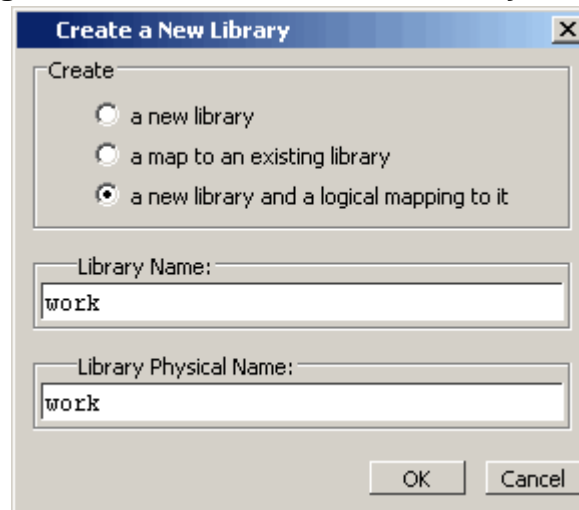
b. Select **File > Change Directory** and change to the directory you created in step 1.

3. Create the working library.

a. Select **File > New > Library**.

This opens a dialog where you specify physical and logical names for the library (Figure 3-1). You can create a new library or map to an existing library. We'll be doing the former.

Figure 3-1. The Create a New Library Dialog



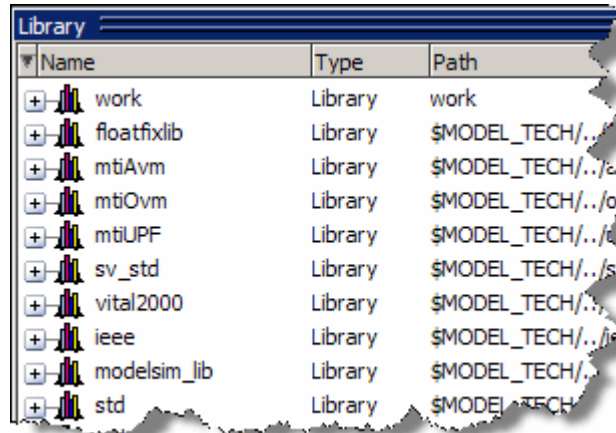
b. Type **work** in the Library Name field (if it isn't already entered automatically).

c. Click **OK**.

ModelSim creates a directory called *work* and writes a specially-formatted file named *_info* into that directory. The *_info* file must remain in the directory to distinguish it as a ModelSim library. Do not edit the folder contents from your operating system; all changes should be made from within ModelSim.

ModelSim also adds the library to the Library window (Figure 3-2) and records the library mapping for future reference in the ModelSim initialization file (*modelsim.ini*).

Figure 3-2. work Library Added to the Library Window



When you pressed OK in step 3c above, the following was printed to the Transcript window:

```
vlib work
vmap work work
```

These two lines are the command-line equivalents of the menu selections you made. Many command-line equivalents will echo their menu-driven functions in this fashion.

Compile the Design Units

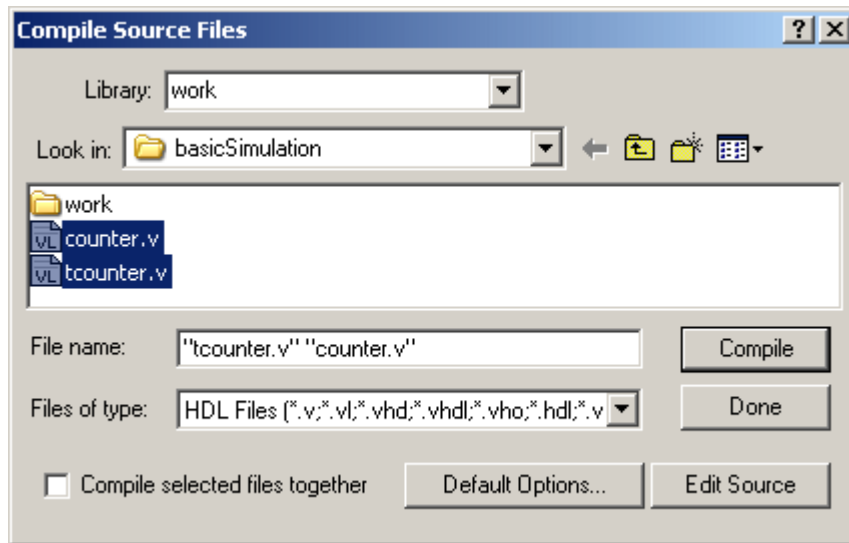
With the working library created, you are ready to compile your source files.

You can compile by using the menus and dialogs of the graphic interface, as in the Verilog example below, or by entering a command at the ModelSim> prompt.

1. Compile *counter.v* and *tcounter.v*.
 - a. Select **Compile > Compile**. This opens the Compile Source Files dialog (Figure 3-3).

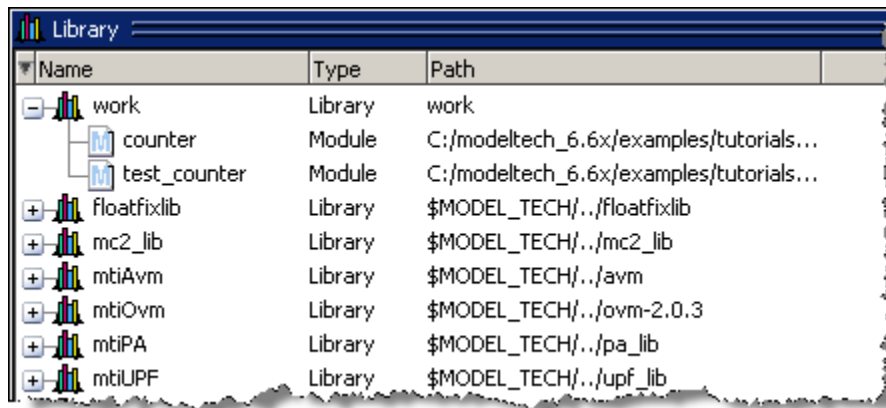
If the Compile menu option is not available, you probably have a project open. If so, close the project by making the Library window active and selecting File > Close from the menus.
 - b. Select both *counter.v* and *tcounter.v* modules from the Compile Source Files dialog and click **Compile**. The files are compiled into the *work* library.
 - c. When compile is finished, click **Done**.

Figure 3-3. Compile Source Files Dialog



2. View the compiled design units.
 - a. In the Library window, click the '+' icon next to the *work* library and you will see two design units (Figure 3-4). You can also see their types (Modules, Entities, etc.) and the path to the underlying source files.

Figure 3-4. Verilog Modules Compiled into work Library

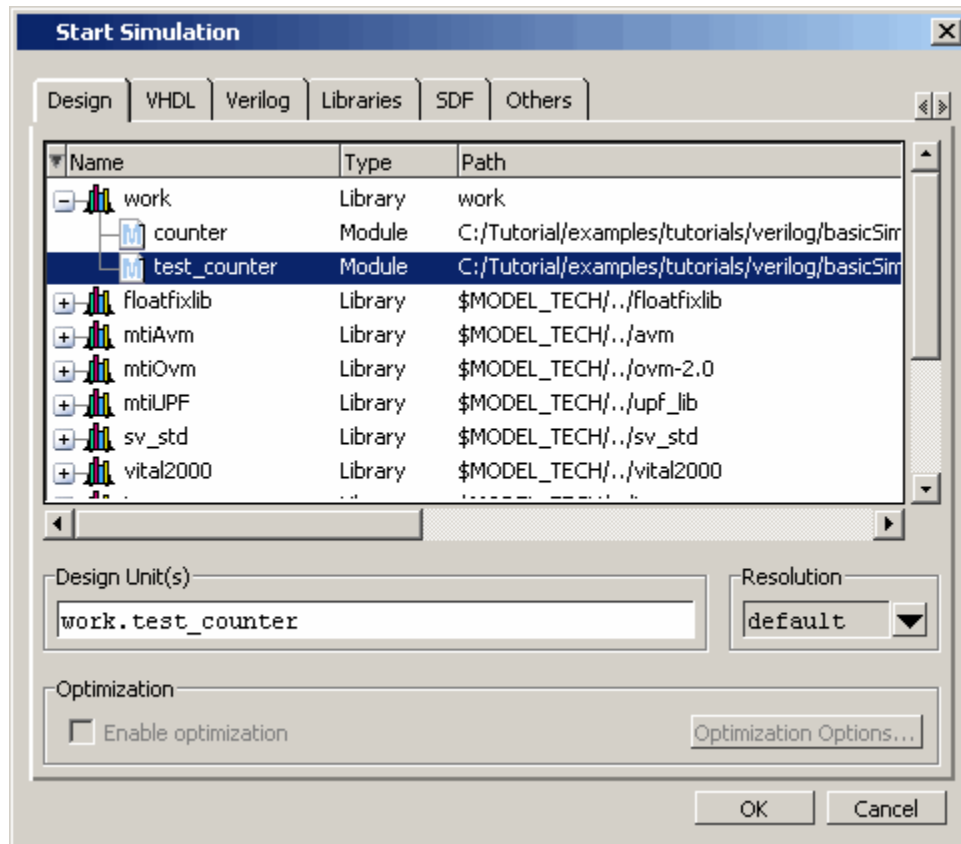


Load the Design

1. Load the *test_counter* module into the simulator.
 - a. In the Library window, click the '+' sign next to the **work** library to show the files contained there.
 - b. Double-click *test_counter* to load the design.

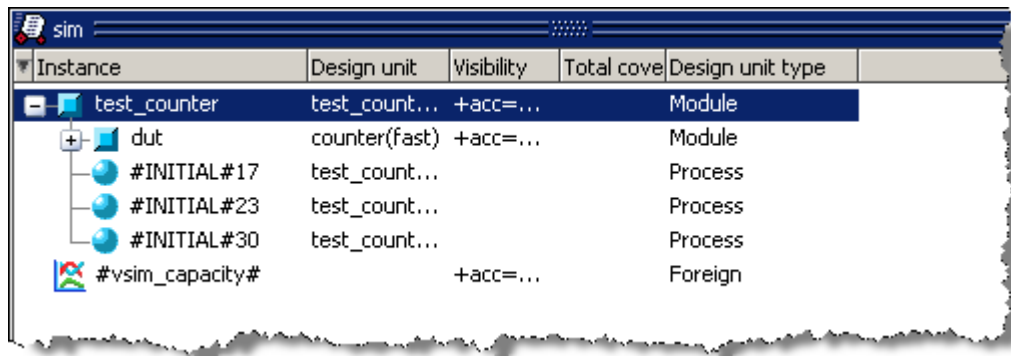
You can also load the design by selecting **Simulate > Start Simulation** in the menu bar. This opens the Start Simulation dialog. With the Design tab selected, click the '+' sign next to the work library to see the *counter* and *test_counter* modules. Select the *test_counter* module and click OK (Figure 3-5).

Figure 3-5. Loading Design with Start Simulation Dialog



When the design is loaded, a Structure window opens (labeled **sim**). This window displays the hierarchical structure of the design as shown in Figure 3-6. You can navigate within the design hierarchy in the Structure (**sim**) window by clicking on any line with a '+' (expand) or '-' (contract) icon.

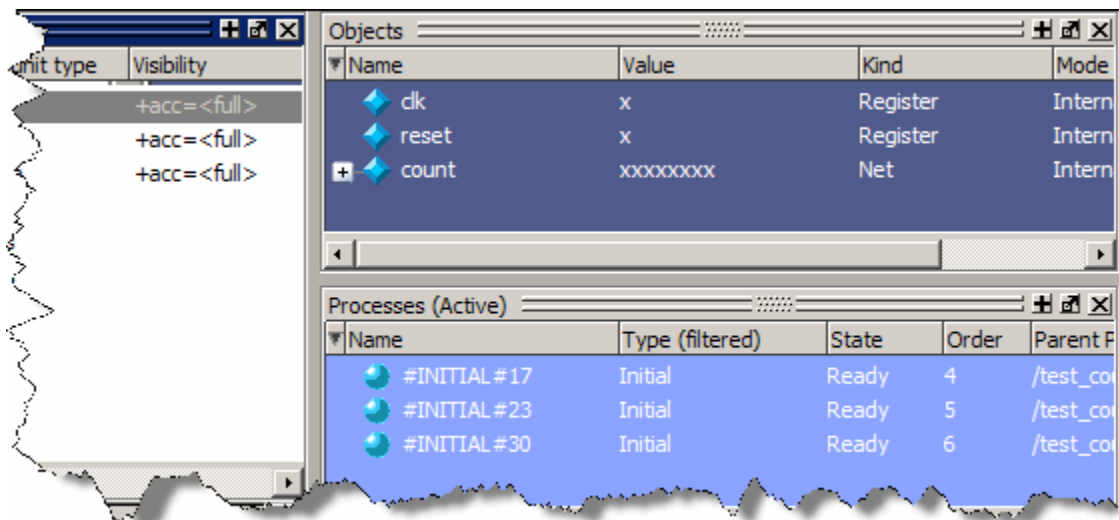
Figure 3-6. The Design Hierarchy



In addition, an Objects window and a Processes window opens (Figure 3-7). The Objects window shows the names and current values of data objects in the current region selected in the Structure (sim) window. Data objects include signals, nets, registers, constants and variables not declared in a process, generics, parameters.

The Processes window displays a list of HDL processes in one of four viewing modes: Active, In Region, Design, and Hierarchical. The Design view mode is intended for primary navigation of ESL (Electronic System Level) designs where processes are a foremost consideration. By default, this window displays the active processes in your simulation (Active view mode).

Figure 3-7. The Object Window and Processes Window



Run the Simulation

We're ready to run the simulation. But before we do, we'll open the Wave window and add signals to it.

1. Open the Wave window.

- a. Enter **view wave** at the command line.

The Wave window opens in the right side of the Main window. Resize it so it is visible.

You can also use the **View > Wave** menu selection to open a Wave window. The Wave window is just one of several debugging windows available on the **View** menu.

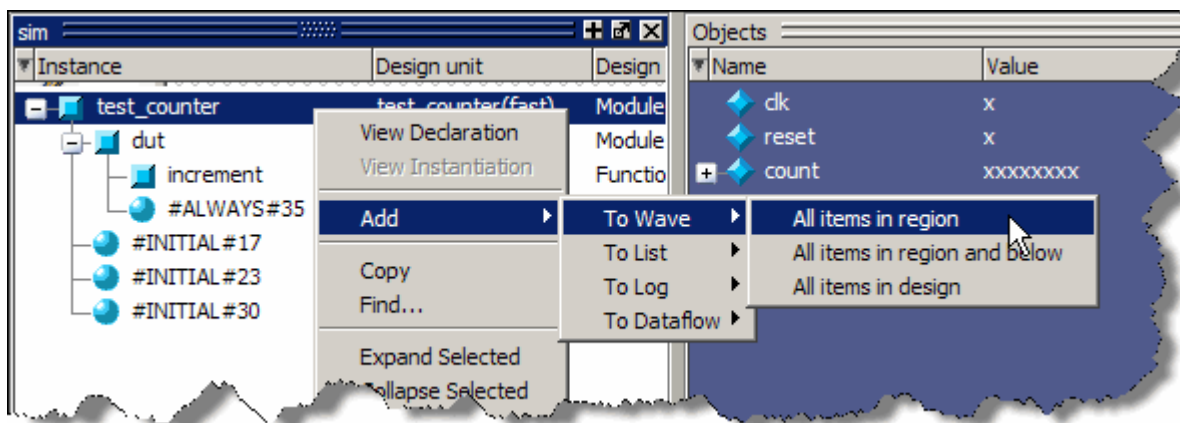
2. Add signals to the Wave window.

- a. In the Structure (sim) window, right-click *test_counter* to open a popup context menu.

- b. Select **Add > To Wave > All items in region** (Figure 3-8).

All signals in the design are added to the Wave window.

Figure 3-8. Using the Popup Menu to Add Signals to Wave Window



3. Run the simulation.

- a. Click the Run icon.

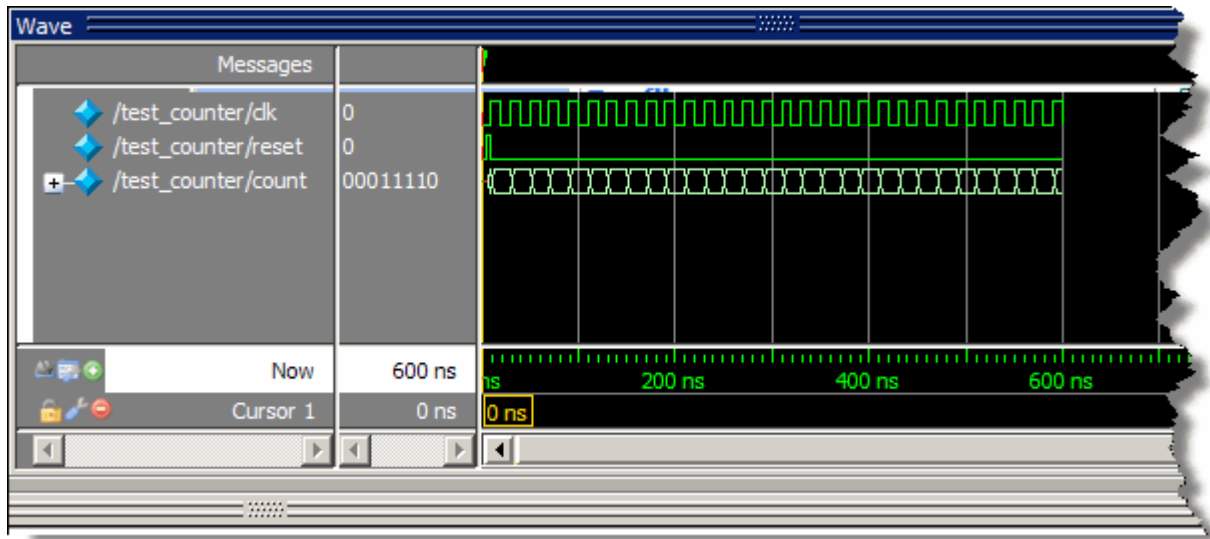
The simulation runs for 100 ns (the default simulation length) and waves are drawn in the Wave window.



- b. Enter **run 500** at the VSIM> prompt in the Transcript window.

The simulation advances another 500 ns for a total of 600 ns (Figure 3-9).

Figure 3-9. Waves Drawn in Wave Window



- c. Click the **Run -All** icon on the Main or Wave window toolbar.

The simulation continues running until you execute a break command or it hits a statement in your code (e.g., a Verilog \$stop statement) that halts the simulation.



- d. Click the Break icon  to stop the simulation.

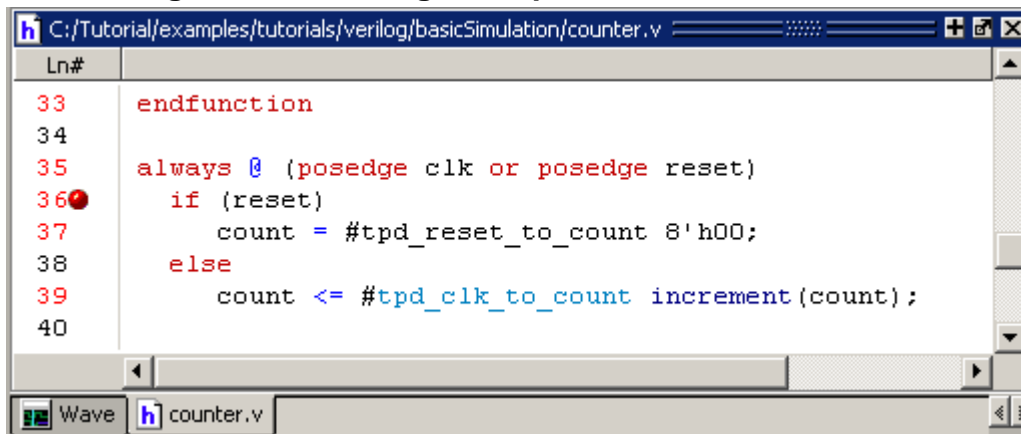
Set Breakpoints and Step through the Source

Next you will take a brief look at one interactive debugging feature of the ModelSim environment. You will set a breakpoint in the Source window, run the simulation, and then step through the design under test. Breakpoints can be set only on executable lines, which are indicated with red line numbers.

1. Open *counter.v* in the Source window.
 - a. Select **View > Files** to open the Files window.
 - b. Click the + sign next to the *sim* filename to see the contents of *vsim.wlf* dataset.
 - c. Double-click *counter.v* (or *counter.vhd* if you are simulating the VHDL files) to open the file in the Source window.
2. Set a breakpoint on line 36 of *counter.v* (or, line 39 of *counter.vhd* for VHDL).
 - a. Scroll to line 36 and click in the Ln# (line number) column next to the line number.

A red ball appears in the line number column at line number 36 (Figure 3-10), indicating that a breakpoint has been set.

Figure 3-10. Setting Breakpoint in Source Window

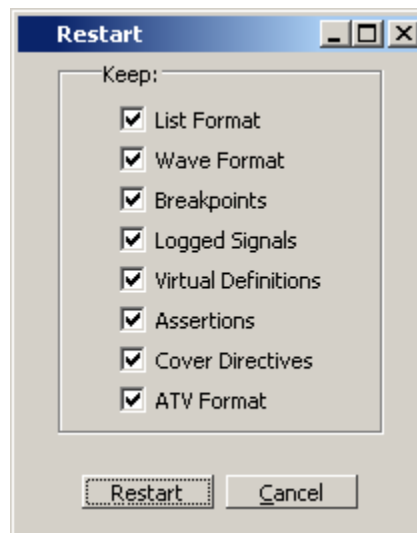


3. Disable, enable, and delete the breakpoint.
 - a. Click the red ball to disable the breakpoint. It will become a black ball.
 - b. Click the black ball again to re-enable the breakpoint. It will become a red ball.
 - c. Click the red ball with your right mouse button and select **Remove Breakpoint 36**.
 - d. Click in the line number column next to line number 36 again to re-create the breakpoint.
4. Restart the simulation.
 - a. Click the Restart icon to reload the design elements and reset the simulation time to zero.



The Restart dialog that appears gives you options on what to retain during the restart (Figure 3-11).

Figure 3-11. Setting Restart Functions

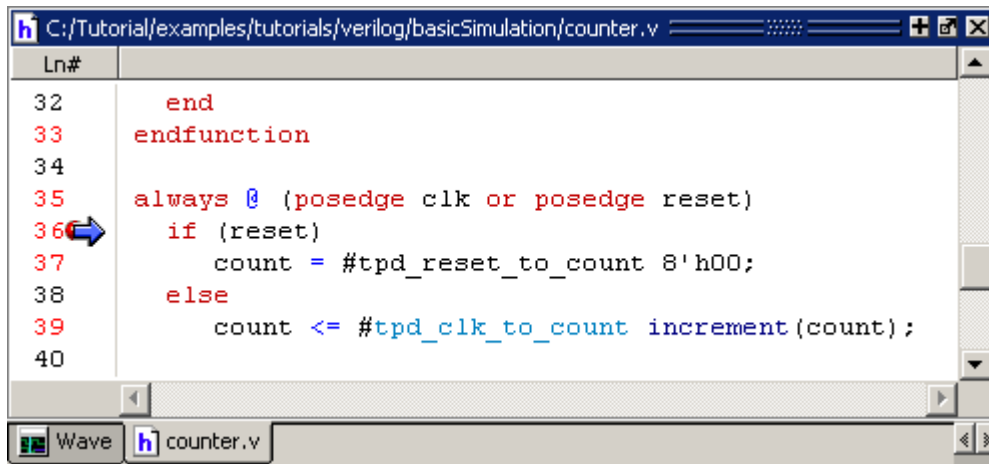


- b. Click the **Restart** button in the Restart dialog.
- c. Click the Run -All icon.



The simulation runs until the breakpoint is hit. When the simulation hits the breakpoint, it stops running, highlights the line with a blue arrow in the Source view (Figure 3-12), and issues a Break message in the Transcript window.

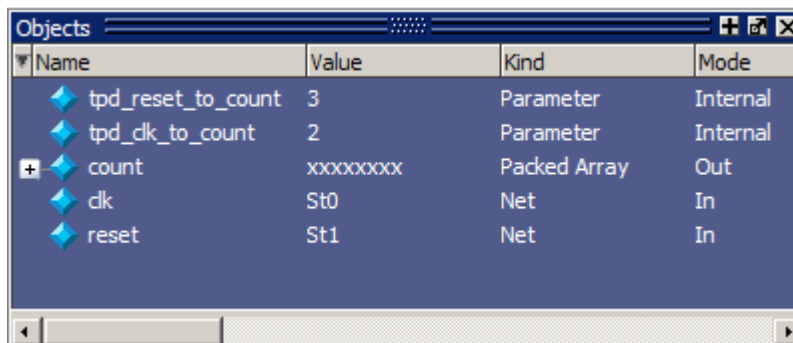
Figure 3-12. Blue Arrow Indicates Where Simulation Stopped.



When a breakpoint is reached, typically you want to know one or more signal values. You have several options for checking values:

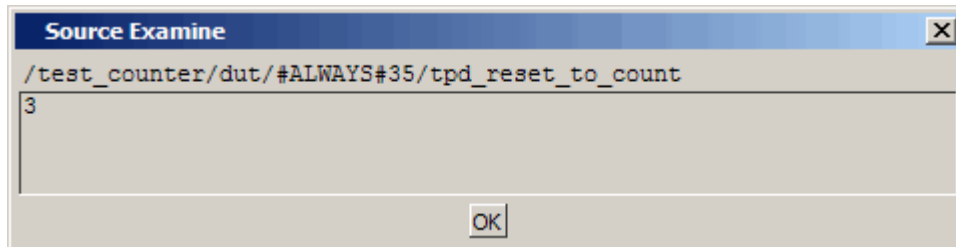
- look at the values shown in the Objects window (Figure 3-13)

Figure 3-13. Values Shown in Objects Window



- set your mouse pointer over a variable in the Source window and a yellow box will appear with the variable name and the value of that variable at the time of the selected cursor in the Wave window
- highlight a signal, parameter, or variable in the Source window, right-click it, and select **Examine** from the pop-up menu to display the variable and its current value in a Source Examine window (Figure 3-14)

Figure 3-14. Parameter Name and Value in Source Examine Window



- use the **examine** command at the VSIM> prompt to output a variable value to the Transcript window (i.e., `examine count`)
5. Try out the step commands.
 - a. Click the Step icon on the Main window toolbar.



This single-steps the debugger.

Experiment on your own. Set and clear breakpoints and use the Step, Step Over, and Continue Run commands until you feel comfortable with their operation.

Lesson Wrap-Up

This concludes this lesson. Before continuing we need to end the current simulation.

1. Select **Simulate > End Simulation**.
2. Click **Yes** when prompted to confirm that you wish to quit simulating.

Introduction

In this lesson you will practice creating a project.

At a minimum, projects contain a work library and a session state that is stored in an *.mpf* file. A project may also consist of:

- HDL source files or references to source files
- other files such as READMEs or other project documentation
- local libraries
- references to global libraries

Design Files for this Lesson

The sample design for this lesson is a simple 8-bit, binary up-counter with an associated test bench. The pathnames are as follows:

Verilog – *<install_dir>/examples/tutorials/verilog/projects/counter.v* and *tcounter.v*

VHDL – *<install_dir>/examples/tutorials/vhdl/projects/counter.vhd* and *tcounter.vhd*

This lesson uses the Verilog files *tcounter.v* and *counter.v*. If you have a VHDL license, use *tcounter.vhd* and *counter.vhd* instead.

Related Reading

User's Manual Chapter: [Projects](#).

Create a New Project

1. Create a new directory and copy the design files for this lesson into it.

Start by creating a new directory for this exercise (in case other users will be working with these lessons).

Verilog: Copy *counter.v* and *tcounter.v* files from */<install_dir>/examples/tutorials/verilog/projects* to the new directory.

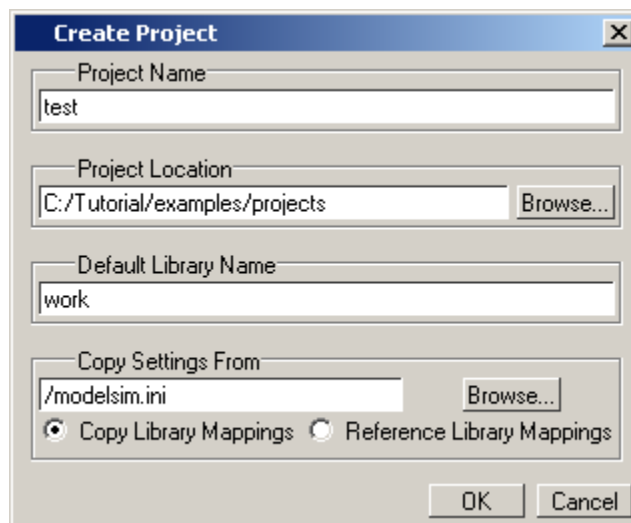
VHDL: Copy *counter.vhd* and *tcounter.vhd* files from */<install_dir>/examples/tutorials/vhdl/projects* to the new directory.

2. If you just finished the previous lesson, ModelSim should already be running. If not, start ModelSim.
 - a. Type **vsim** at a UNIX shell prompt or use the ModelSim icon in Windows.
 - b. Select **File > Change Directory** and change to the directory you created in step 1.
3. Create a new project.
 - a. Select **File > New > Project** (Main window) from the menu bar.

This opens the Create Project dialog where you can enter a Project Name, Project Location (i.e., directory), and Default Library Name (Figure 4-1). You can also reference library settings from a selected .ini file or copy them directly into the project. The default library is where compiled design units will reside.

- b. Type **test** in the Project Name field.
- c. Click the **Browse** button for the Project Location field to select a directory where the project file will be stored.
- d. Leave the Default Library Name set to *work*.
- e. Click **OK**.

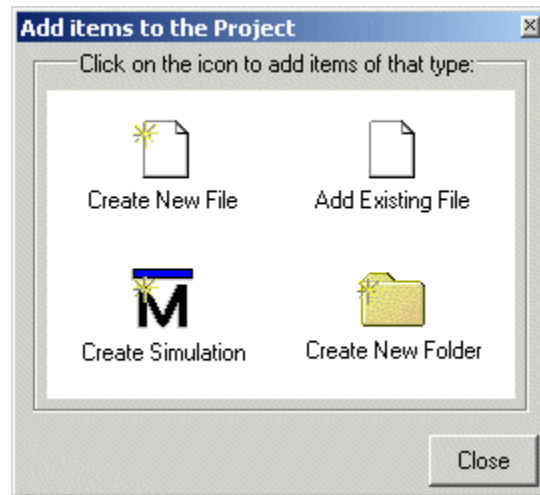
Figure 4-1. Create Project Dialog - Project Lab



Add Objects to the Project

Once you click OK to accept the new project settings, a blank Project window and the “Add items to the Project” dialog will appear (Figure 4-2). From the dialog you can create a new design file, add an existing file, add a folder for organization purposes, or create a simulation configuration (discussed below).

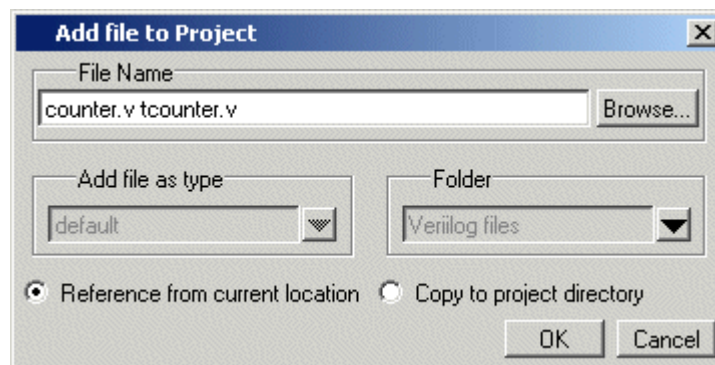
Figure 4-2. Adding New Items to a Project



1. Add two existing files.
 - a. Click **Add Existing File**.

This opens the Add file to Project dialog (Figure 4-3). This dialog lets you browse to find files, specify the file type, specify a folder to which the file will be added, and identify whether to leave the file in its current location or to copy it to the project directory.

Figure 4-3. Add file to Project Dialog



- b. Click the **Browse** button for the File Name field. This opens the “Select files to add to project” dialog and displays the contents of the current directory.
 - c. **Verilog:** Select *counter.v* and *tcounter.v* and click **Open**.
VHDL: Select *counter.vhd* and *tcounter.vhd* and click **Open**.

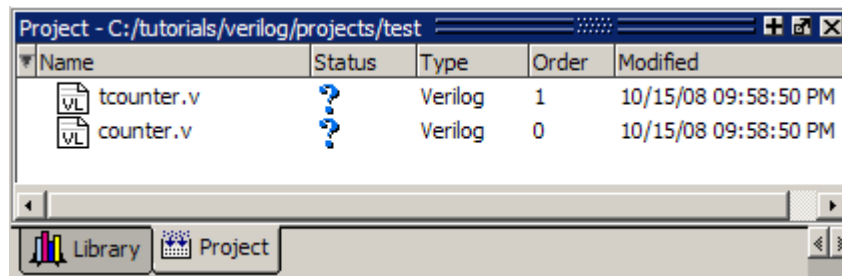
This closes the “Select files to add to project” dialog and displays the selected files in the “Add file to Project” dialog (Figure 4-3).

- d. Click **OK** to add the files to the project.

- e. Click **Close** to dismiss the Add items to the Project dialog.

You should now see two files listed in the Project window (Figure 4-4). Question-mark icons in the Status column indicate that the file has not been compiled or that the source file has changed since the last successful compile. The other columns identify file type (e.g., Verilog or VHDL), compilation order, and modified date.

Figure 4-4. Newly Added Project Files Display a '?' for Status



Changing Compile Order (VHDL)

By default ModelSim performs default binding of VHDL designs when you load the design with `vsim`. However, you can elect to perform default binding at compile time. (For details, refer to the section [Default Binding](#) in the User's Manual.) If you elect to do default binding at compile, then the compile order is important. Follow these steps to change compilation order within a project.

1. Change the compile order.
 - a. Select **Compile > Compile Order**.

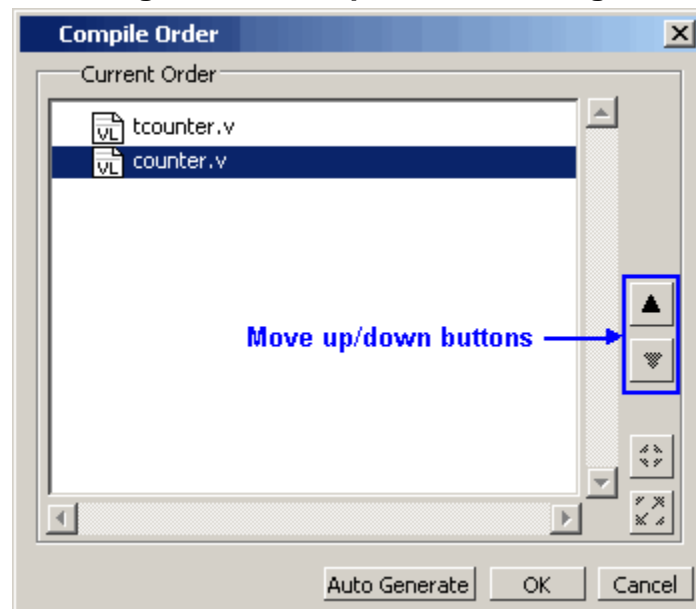
This opens the Compile Order dialog box.

- b. Click the **Auto Generate** button.

ModelSim determines the compile order by making multiple passes over the files. It starts compiling from the top; if a file fails to compile due to dependencies, it moves that file to the bottom and then recompiles it after compiling the rest of the files. It continues in this manner until all files compile successfully or until a file(s) can't be compiled for reasons other than dependency.

Alternatively, you can select a file and use the Move Up and Move Down buttons to put the files in the correct order (Figure 4-5).

Figure 4-5. Compile Order Dialog



- c. Click **OK** to close the Compile Order dialog.

Compile the Design

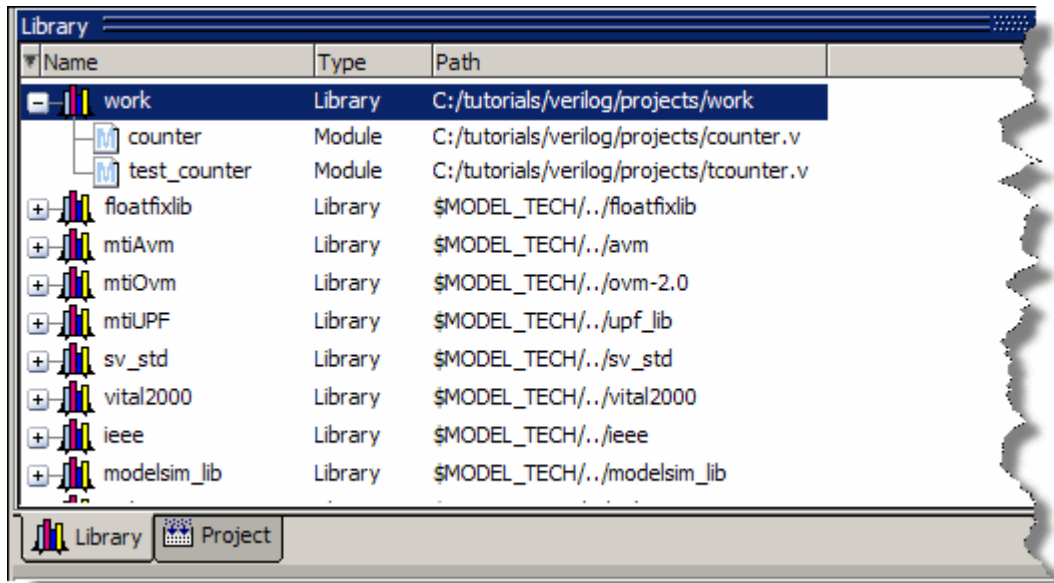
1. Compile the files.
 - a. Right-click either *counter.v* or *tcounter.v* in the Project window and select **Compile > Compile All** from the pop-up menu.

ModelSim compiles both files and changes the symbol in the Status column to a green check mark. A check mark means the compile succeeded. If compile fails, the symbol will be a red 'X', and you will see an error message in the Transcript window.

2. View the design units.
 - a. Click the **Library** tab (Figure 4-6).
 - b. Click the '+' icon next to the *work* library.

You should see two compiled design units, their types (modules in this case), and the path to the underlying source files.

Figure 4-6. Library Window with Expanded Library

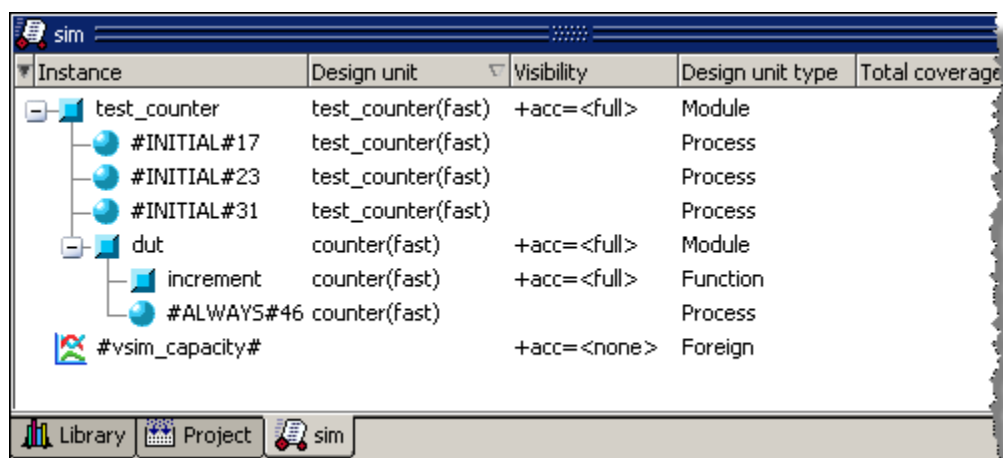


Load the Design

1. Load the *test_counter* design unit.
 - a. Double-click the *test_counter* design unit.

The Structure (sim) window appears as part of the tab group with the Library and Project windows (Figure 4-7).

Figure 4-7. Structure(sim) window for a Loaded Design



At this point you would typically run the simulation and analyze or debug your design like you did in the previous lesson. For now, you'll continue working with the project. However, first you need to end the simulation that started when you loaded *test_counter*.

2. End the simulation.
 - a. Select **Simulate > End Simulation**.
 - b. Click **Yes**.

Organizing Projects with Folders

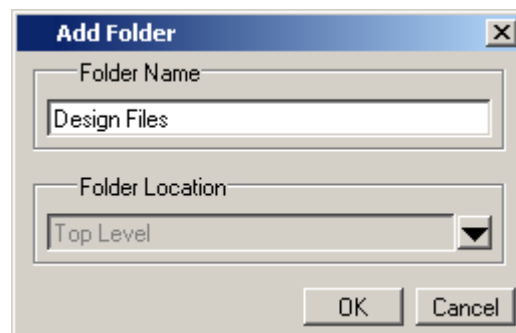
If you have a lot of files to add to a project, you may want to organize them in folders. You can create folders either before or after adding your files. If you create a folder before adding files, you can specify in which folder you want a file placed at the time you add the file (see Folder field in [Figure 4-3](#)). If you create a folder after adding files, you edit the file properties to move it to that folder.

Add Folders

As shown previously in [Figure 4-2](#), the Add items to the Project dialog has an option for adding folders. If you have already closed that dialog, you can use a menu command to add a folder.

1. Add a new folder.
 - a. Right-click in the Projects window and select **Add to Project > Folder**.
 - b. Type **Design Files** in the **Folder Name** field ([Figure 4-8](#)).

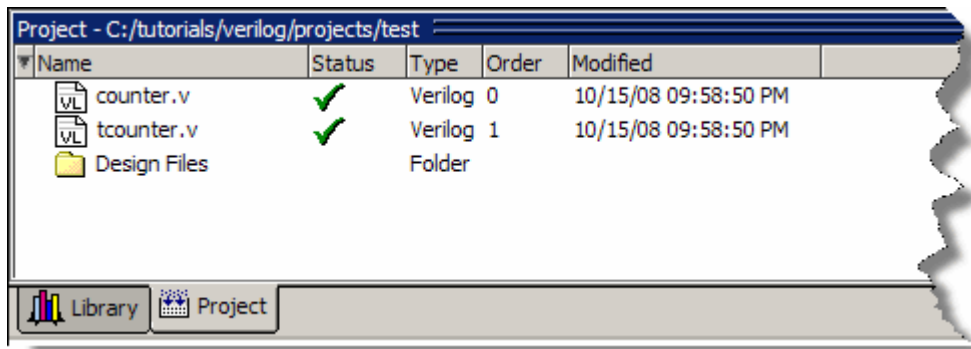
Figure 4-8. Adding New Folder to Project



- c. Click **OK**.

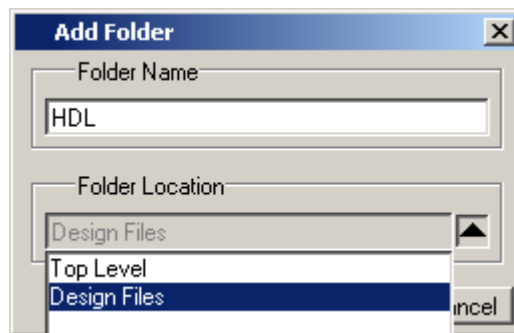
The new Design Files folder is displayed in the Project window ([Figure 4-9](#)).

Figure 4-9. A Folder Within a Project



2. Add a sub-folder.
 - a. Right-click anywhere in the Project window and select **Add to Project > Folder**.
 - b. Type **HDL** in the **Folder Name** field (Figure 4-10).

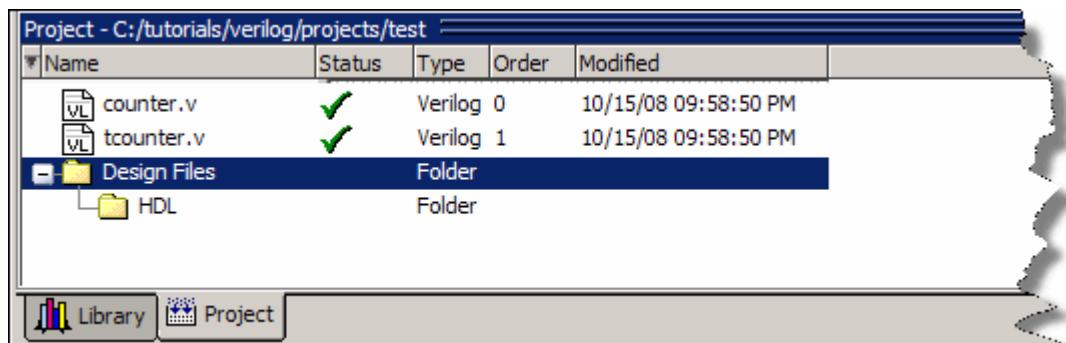
Figure 4-10. Creating Subfolder



- c. Click the **Folder Location** drop-down arrow and select *Design Files*.
 - d. Click **OK**.

A '+' icon appears next to the *Design Files* folder in the Project window (Figure 4-11).

Figure 4-11. A folder with a Sub-folder



- e. Click the '+' icon to see the *HDL* sub-folder.

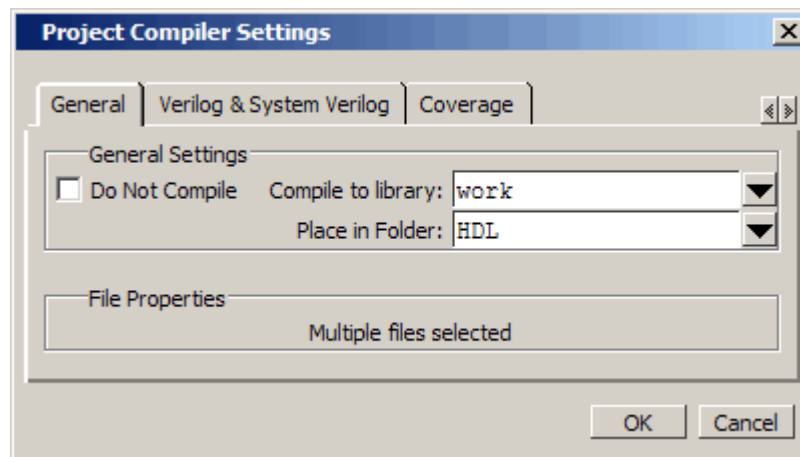
Moving Files to Folders

If you don't place files into a folder when you first add the files to the project, you can move them into a folder using the properties dialog.

1. Move *tcounter.v* and *counter.v* to the *HDL* folder.
 - a. Select both *counter.v* and *tcounter.v* in the Project window.
 - b. Right-click either file and select **Properties**.

This opens the Project Compiler Settings dialog (Figure 4-12), which allows you to set a variety of options on your design files.

Figure 4-12. Changing File Location via the Project Compiler Settings Dialog



- c. Click the **Place In Folder** drop-down arrow and select *HDL*.
- d. Click **OK**.

The selected files are moved into the HDL folder. Click the '+' icon next to the HDL folder to see the files.

The files are now marked with a '?' in the Status column because you moved the files. The project no longer knows if the previous compilation is still valid.

Simulation Configurations

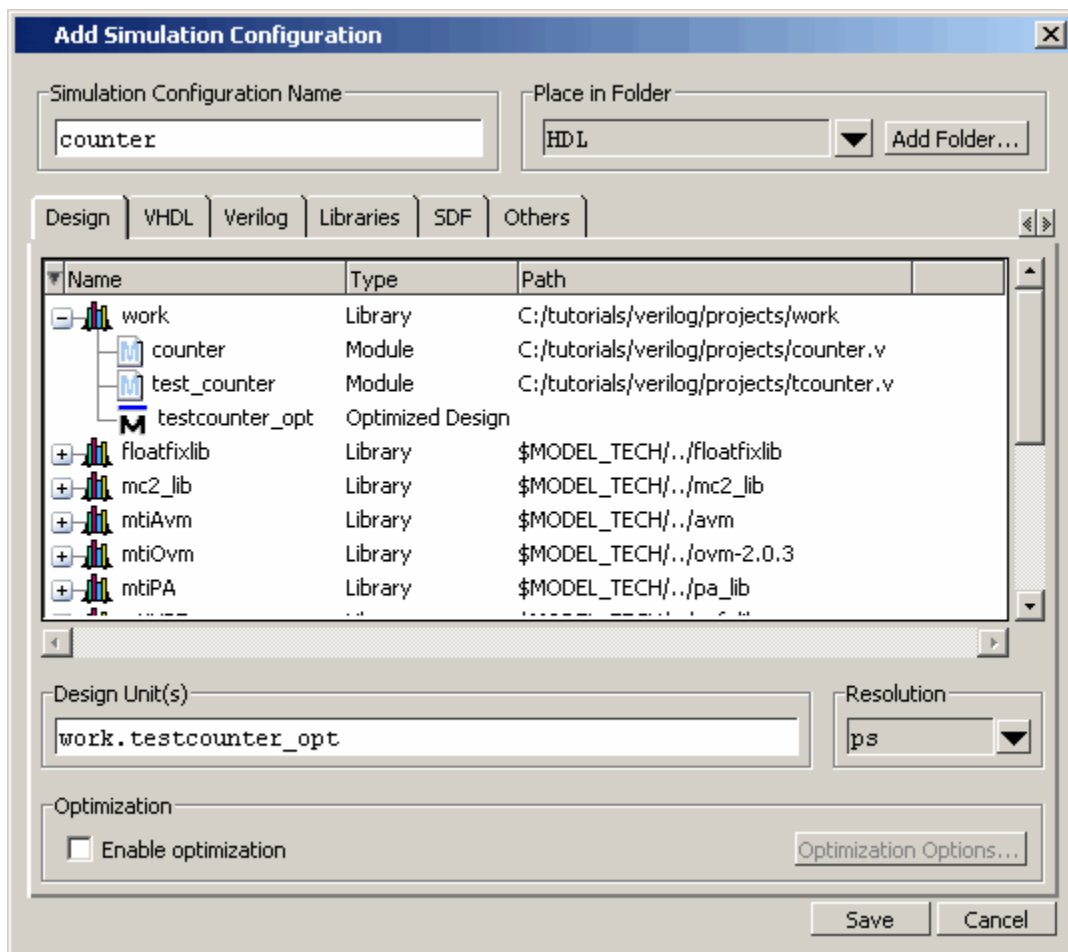
A Simulation Configuration associates a design unit(s) and its simulation options. For example, let's say that every time you load *tcounter.v* you want to set the simulator resolution to picoseconds (ps) and enable event order hazard checking. Ordinarily, you would have to specify those options each time you load the design. With a Simulation Configuration, you specify options for a design and then save a "configuration" that associates the design and its options.

The configuration is then listed in the Project window and you can double-click it to load *tcounter.v* along with its options.

1. Create a new Simulation Configuration.
 - a. Right-click in the Project window and select **Add to Project > Simulation Configuration** from the popup menu.

This opens the Add Simulation Configuration dialog (Figure 4-13). The tabs in this dialog present several simulation options. You may want to explore the tabs to see what is available. You can consult the ModelSim User's Manual to get a description of each option.

Figure 4-13. Simulation Configuration Dialog



- b. Type **counter** in the **Simulation Configuration Name** field.
- c. Select *HDL* from the **Place in Folder** drop-down.
- d. Click the '+' icon next to the *work* library and select *test_counter*.

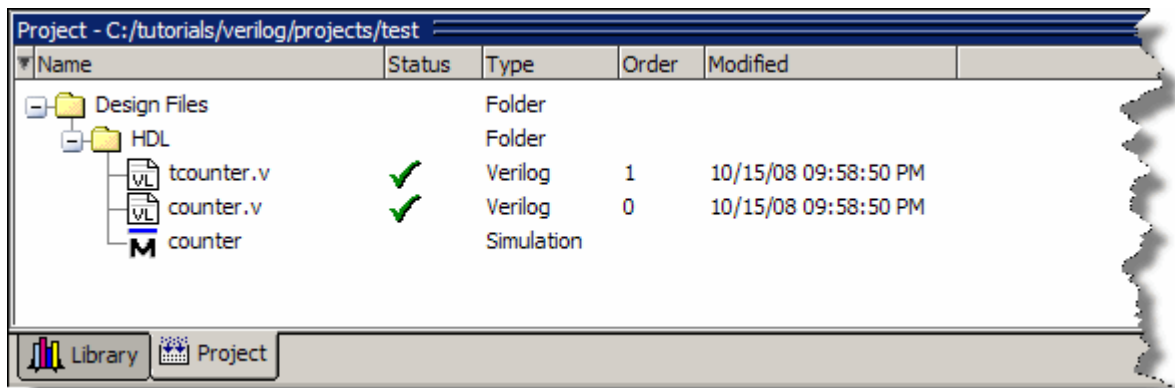
- e. Click the **Resolution** drop-down and select *ps*.
- f. For Verilog, click the Verilog tab and check **Enable hazard checking (-hazards)**.
- g. Click **Save**.

The files *tcounter.v* and *counter.v* show question mark icons in the status column because they have changed location since they were last compiled and need to be recompiled.

- h. Select one of the files, *tcounter.v* or *counter.v*.
- i. Select **Compile > Compile All**.

The Project window now shows a Simulation Configuration named *counter* in the HDL folder (Figure 4-14).

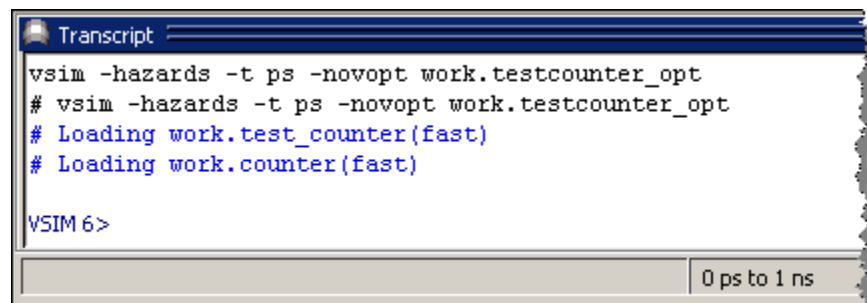
Figure 4-14. A Simulation Configuration in the Project window



2. Load the Simulation Configuration.
 - a. Double-click the *counter* Simulation Configuration in the Project window.

In the Transcript window of the Main window, the **vsim** (the ModelSim simulator) invocation shows the **-hazards** and **-t ps** switches (Figure 4-15). These are the command-line equivalents of the options you specified in the Simulate dialog.

Figure 4-15. Transcript Shows Options for Simulation Configurations



Lesson Wrap-Up

This concludes this lesson. Before continuing you need to end the current simulation and close the current project.

1. Select **Simulate > End Simulation**. Click Yes.
2. In the Project window, right-click and select **Close Project**.

If you do not close the project, it will open automatically the next time you start ModelSim.

Chapter 5

Working With Multiple Libraries

Introduction

In this lesson you will practice working with multiple libraries. You might have multiple libraries to organize your design, to access IP from a third-party source, or to share common parts between simulations.

You will start the lesson by creating a resource library that contains the *counter* design unit. Next, you will create a project and compile the test bench into it. Finally, you will link to the library containing the counter and then run the simulation.

Design Files for this Lesson

The sample design for this lesson is a simple 8-bit, binary up-counter with an associated test bench. The pathnames are as follows:

Verilog – `<install_dir>/examples/tutorials/verilog/libraries/counter.v` and `tcounter.v`

VHDL – `<install_dir>/examples/tutorials/vhdl/libraries/counter.vhd` and `tcounter.vhd`

This lesson uses the Verilog files `tcounter.v` and `counter.v` in the examples. If you have a VHDL license, use `tcounter.vhd` and `counter.vhd` instead.

Related Reading

User's Manual Chapter: [Design Libraries](#).

Creating the Resource Library

Before creating the resource library, make sure the `modelsim.ini` in your install directory is “Read Only.” This will prevent permanent mapping of resource libraries to the master `modelsim.ini` file. See [Permanently Mapping VHDL Resource Libraries](#).

1. Create a directory for the resource library.

Create a new directory called `resource_library`. Copy `counter.v` from `<install_dir>/examples/tutorials/verilog/libraries` to the new directory.

2. Create a directory for the test bench.

Create a new directory called *testbench* that will hold the test bench and project files. Copy *tcounter.v* from `<install_dir>/examples/tutorials/verilog/libraries` to the new directory.

You are creating two directories in this lesson to mimic the situation where you receive a resource library from a third-party. As noted earlier, we will link to the resource library in the first directory later in the lesson.

3. Start ModelSim and change to the *resource_library* directory.

If you just finished the previous lesson, ModelSim should already be running. If not, start ModelSim.

- a. Type **vsim** at a UNIX shell prompt or use the ModelSim icon in Windows.

If the Welcome to ModelSim dialog appears, click **Close**.

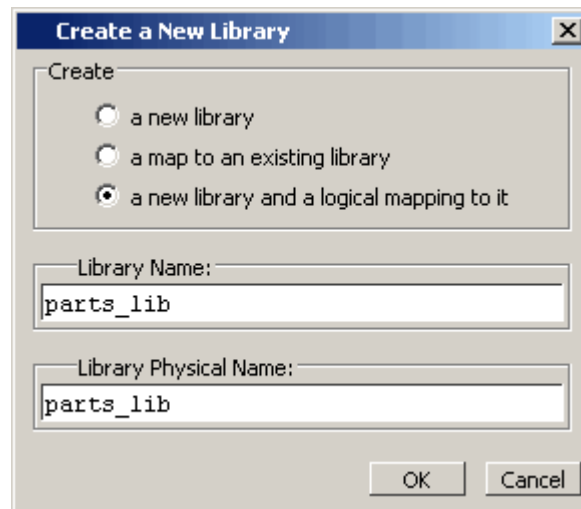
- b. Select **File > Change Directory** and change to the *resource_library* directory you created in step 1.

4. Create the resource library.

- a. Select **File > New > Library**.

- b. Type **parts_lib** in the Library Name field (Figure 5-1).

Figure 5-1. Creating New Resource Library



The Library Physical Name field is filled out automatically.

Once you click OK, ModelSim creates a directory for the library, lists it in the Library window, and modifies the *modelsim.ini* file to record this new library for the future.

5. Compile the counter into the resource library.


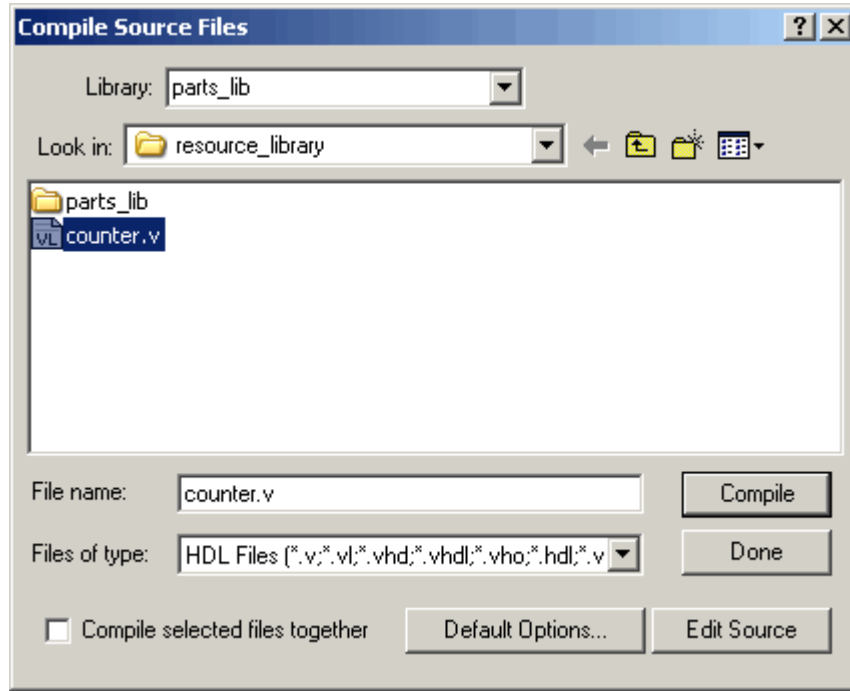
- a. Click the Compile icon on the Main window toolbar. 
- b. Select the *parts_lib* library from the Library list (Figure 5-2).

Figure 5-2. Compiling into the Resource Library



- c. Double-click *counter.v* to compile it.
- d. Click **Done**.

You now have a resource library containing a compiled version of the *counter* design unit.

6. Change to the *testbench* directory.
 - a. Select **File > Change Directory** and change to the *testbench* directory you created in step 2.

Creating the Project

Now you will create a project that contains *tcounter.v*, the counter's test bench.

1. Create the project.
 - a. Select **File > New > Project**.
 - b. Type **counter** in the Project Name field.
 - c. Do not change the Project Location field or the Default Library Name field. (The default library name is *work*.)

- d. Make sure “Copy Library Mappings” is selected. The default *modelsim.ini* file will be used.
 - e. Click **OK**.
2. Add the test bench to the project.
 - a. Click **Add Existing File** in the Add items to the Project dialog.
 - b. Click the **Browse** button and select *tcounter.v* in the “Select files to add to project” dialog.
 - c. Click **Open**.
 - d. Click **OK**.
 - e. Click **Close** to dismiss the “Add items to the Project” dialog.

The *tcounter.v* file is listed in the Project window.
 3. Compile the test bench.
 - a. Right-click *tcounter.v* and select **Compile > Compile Selected**.

Linking to the Resource Library

To wrap up this part of the lesson, you will link to the *parts_lib* library you created earlier. But first, try loading the test bench without the link and see what happens.

ModelSim responds differently for Verilog and VHDL in this situation.

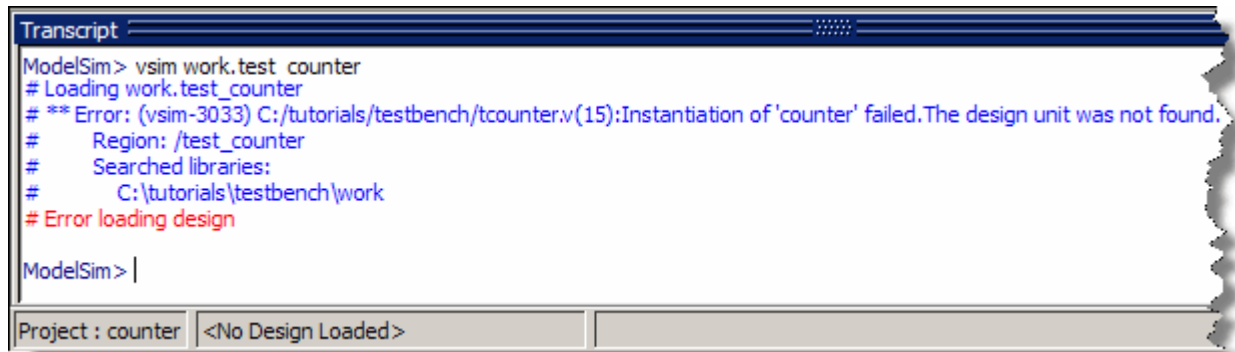
Verilog

Load the Verilog Test Bench

1. Load a Verilog design with a missing resource library.
 - a. In the Library window, click the '+' icon next to the *work* library and double-click *test_counter*.

The Transcript reports an error ([Figure 5-3](#)). When you see a message that contains text like "Error: (vsim-3033)", you can view more detail by using the **verror** command.

Figure 5-3. Verilog Simulation Error Reported in Transcript

The screenshot shows the ModelSim Transcript window. The text in the transcript is as follows:

```
ModelSim> vsim work.test_counter
# Loading work.test_counter
# ** Error: (vsim-3033) C:/tutorials/testbench/tcounter.v(15):Instantiation of 'counter' failed.The design unit was not found.
#   Region: /test_counter
#   Searched libraries:
#     C:/tutorials/testbench/work
# Error loading design

ModelSim> |
```

At the bottom of the window, there is a status bar with the text "Project : counter" and "<No Design Loaded>".

- b. Type **verror 3033** at the ModelSim> prompt.

The expanded error message tells you that a design unit could not be found for instantiation. It also tells you that the original error message should list which libraries ModelSim searched. In this case, the original message says ModelSim searched only *work*.

- c. Type **quit -sim** to quit the simulation.

The process for linking to a resource library differs between Verilog and VHDL. If you are using Verilog, follow the steps in [Linking to a Resource Library](#). If you are using VHDL, follow the steps in [Permanently Mapping VHDL Resource Libraries](#) one page later.

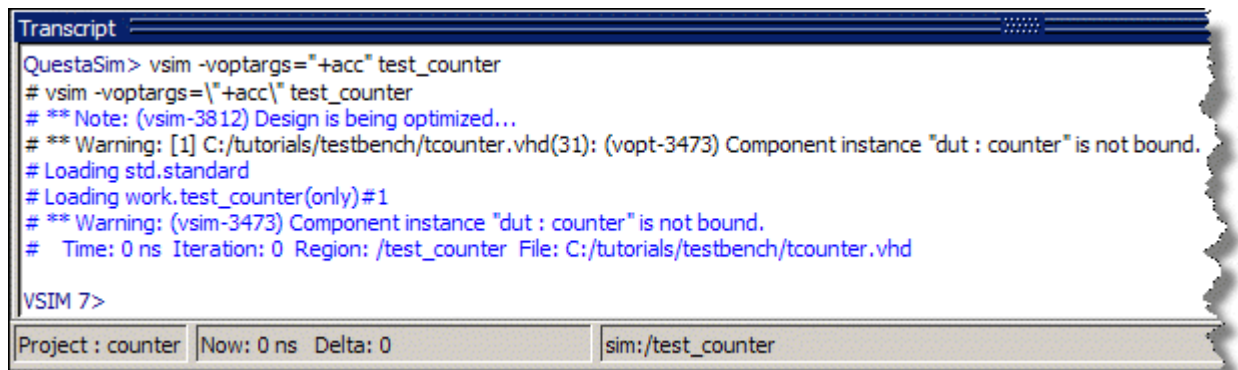
VHDL

Load the VHDL Test Bench

1. Load the VHDL test bench with a missing resource library.
 - a. In the Library window, click the '+' icon next to the *work* library and double-click *test_counter*.

The Main window Transcript reports a warning ([Figure 5-4](#)). When you see a message that contains text like "Warning: (vsim-3473)", you can view more detail by using the **verror** command.

Figure 5-4. VHDL Simulation Warning Reported in Main Window



```
Transcript
QuestaSim> vsim -voptargs="+acc" test_counter
# vsim -voptargs="+acc" test_counter
# ** Note: (vsim-3812) Design is being optimized...
# ** Warning: [1] C:/tutorials/testbench/tcounter.vhd(31): (vopt-3473) Component instance "dut : counter" is not bound.
# Loading std.standard
# Loading work.test_counter(only)#1
# ** Warning: (vsim-3473) Component instance "dut : counter" is not bound.
# Time: 0 ns Iteration: 0 Region: /test_counter File: C:/tutorials/testbench/tcounter.vhd
VSIM 7>
```

Project : counter Now: 0 ns Delta: 0 sim:/test_counter

- b. Type **error 3473** at the VSIM> prompt.

The expanded error message tells you that a component ('dut' in this case) has not been explicitly bound and no default binding can be found.

- c. Type **quit -sim** to quit the simulation.

Linking to a Resource Library

Linking to a resource library requires that you specify a "search library" when you invoke the simulator.

1. Specify a search library during simulation.

- a. Click the Simulate icon on the Main window toolbar.



- b. Click the '+' icon next to the *work* library and select *test_counter*.

- c. Click the Libraries tab.

- d. Click the Add button next to the Search Libraries field and browse to *parts_lib* in the *resource_library* directory you created earlier in the lesson.

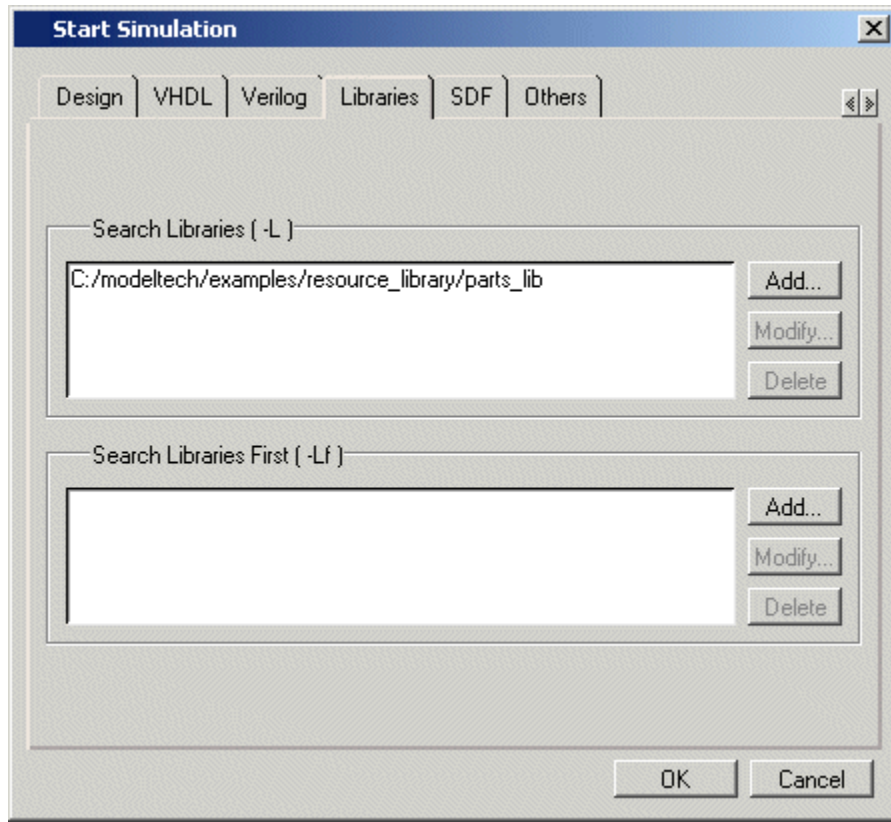
- e. Click OK.

The dialog should have *parts_lib* listed in the Search Libraries field (Figure 5-5).

- f. Click OK.

The design loads without errors.

Figure 5-5. Specifying a Search Library in the Simulate Dialog



Permanently Mapping VHDL Resource Libraries

If you reference particular VHDL resource libraries in every VHDL project or simulation, you may want to permanently map the libraries. Doing this requires that you edit the master *modelsim.ini* file in the installation directory. Though you won't actually practice it in this tutorial, here are the steps for editing the file:

1. Locate the *modelsim.ini* file in the ModelSim installation directory (`<install_dir>/modeltech/modelsim.ini`).
2. IMPORTANT - Make a backup copy of the file.
3. Change the file attributes of *modelsim.ini* so it is no longer "read-only."
4. Open the file and enter your library mappings in the [Library] section. For example:

```
parts_lib = C:/libraries/parts_lib
```
5. Save the file.
6. Change the file attributes so the file is "read-only" again.

Lesson Wrap-Up

This concludes this lesson. Before continuing we need to end the current simulation and close the project.

1. Select **Simulate > End Simulation**. Click **Yes**.
2. Select the Project window to make it active.
3. Select **File > Close**. Click **OK**.

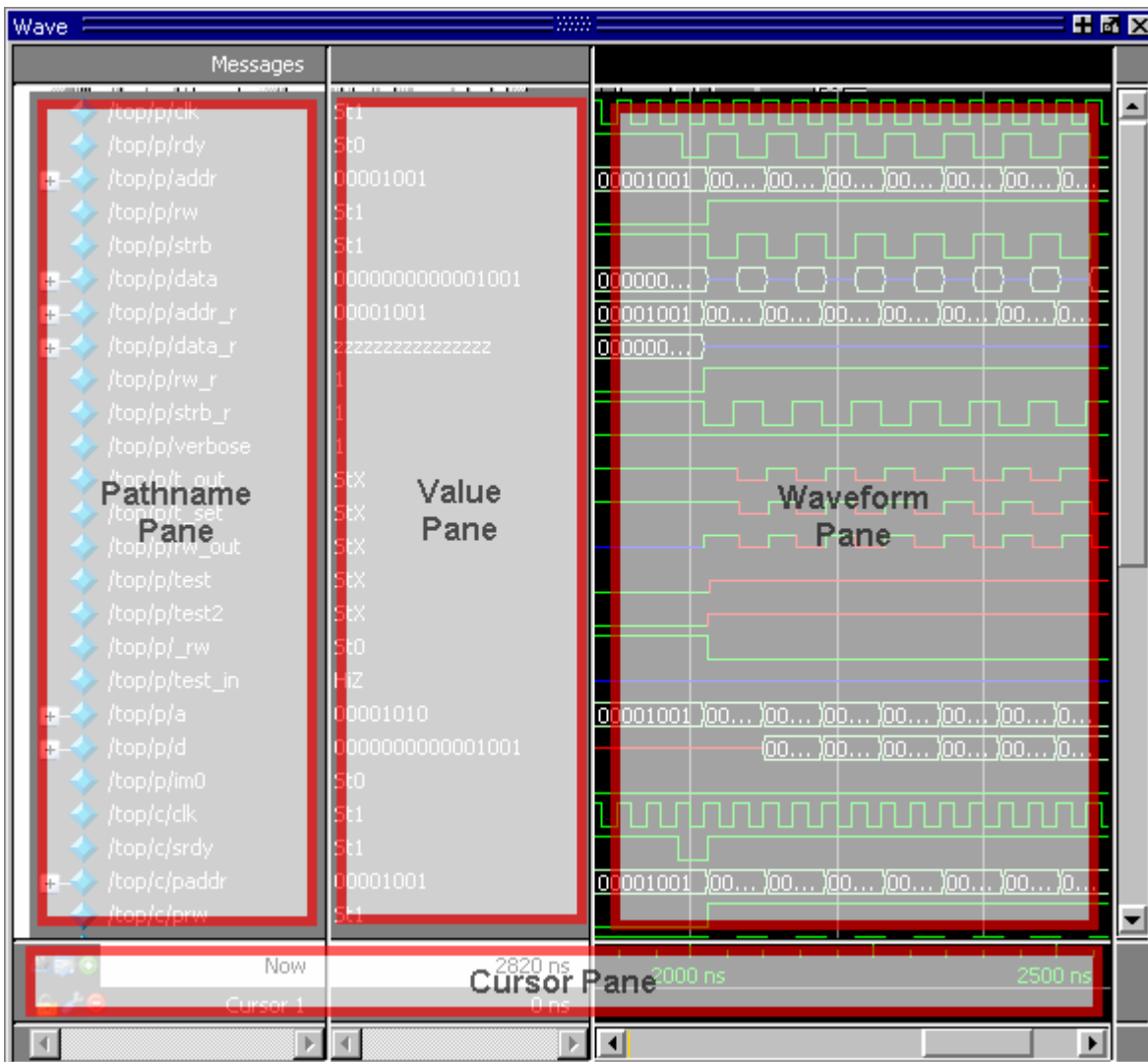
Chapter 6

Analyzing Waveforms

Introduction

The Wave window allows you to view the results of your simulation as HDL waveforms and their values. The Wave window is divided into a number of panes (Figure 6-1). You can resize the pathnames pane, the values pane, and the waveform pane by clicking and dragging the bar between any two panes.

Figure 6-1. Panes of the Wave Window



Related Reading

User's Manual sections: [Wave Window](#) and [Recording Simulation Results With Datasets](#)

Loading a Design

For the examples in this lesson, we will use the design simulated in [Basic Simulation](#).

1. If you just finished the previous lesson, ModelSim should already be running. If not, start ModelSim.
 - a. Type **vsim** at a UNIX shell prompt or use the ModelSim icon in Windows.
If the Welcome to ModelSim dialog appears, click **Close**.
2. Load the design.
 - a. Select **File > Change Directory** and open the directory you created in the “Basic Simulation” lesson.
The *work* library should already exist.
 - b. Click the '+' icon next to the *work* library and double-click *test_counter*.
ModelSim loads the design and opens a Structure (sim) window.

Add Objects to the Wave Window

ModelSim offers several methods for adding objects to the Wave window. In this exercise, you will try different methods.

1. Add objects from the Objects window.
 - a. Open an Objects window by selecting **View > Objects**.
 - b. Select an item in the Objects window, right-click, and then select **Add > To Wave > Signals in Region**.
ModelSim opens a Wave window and displays signals in the region.
2. Undock the Wave window.

By default ModelSim opens the Wave window in the right side of the Main window. You can change the default via the Preferences dialog (**Tools > Edit Preferences**). Refer to the [Simulator GUI Preferences](#) section in the User's Manual for more information.

- a. Click the undock icon on the Wave window.



The Wave window becomes a standalone, un-docked window. Resize the window as needed.

3. Add objects using drag-and-drop.

You can drag an object to the Wave window from many other windows (e.g., Structure, Objects, and Locals).

- a. In the Wave window, select **Edit > Select All** and then **Edit > Delete**.
- b. Drag an instance from the Structure (sim) window to the Wave window.
ModelSim adds the objects for that instance to the Wave window.
- c. Drag a signal from the Objects window to the Wave window.
- d. In the Wave window, select **Edit > Select All** and then **Edit > Delete**.

4. Add objects using the [add wave](#) command.

- a. Type the following at the VSIM> prompt.

```
add wave *
```

ModelSim adds all objects from the current region.

- b. Run the simulation for 500 ns so you can see waveforms.

Zooming the Waveform Display

There are numerous methods for zooming the Waveform display.

1. Zoom the display using various techniques.

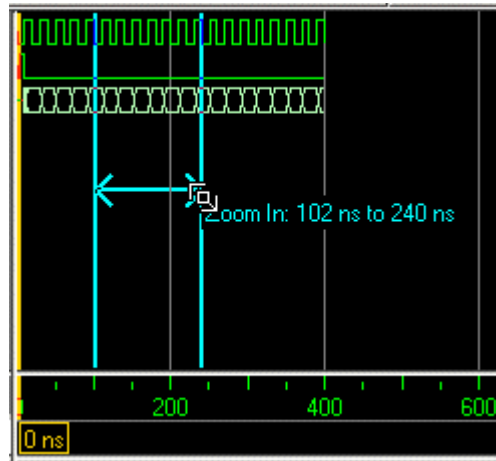
- a. Click the Zoom Mode icon on the Wave window toolbar.



- b. In the waveform display, click and drag down and to the right.

You should see blue vertical lines and numbers defining an area to zoom in ([Figure 6-2](#)).

Figure 6-2. Zooming in with the Mouse Pointer



- c. Select **View > Zoom > Zoom Last**.

The waveform display restores the previous display range.

- d. Click the Zoom In icon a few times.



- e. In the waveform display, click and drag up and to the right.

You should see a blue line and numbers defining an area to zoom out.

- f. Select **View > Zoom > Zoom Full**.

Using Cursors in the Wave Window

Cursors mark simulation time in the Wave window. When ModelSim first draws the Wave window, it places one cursor at time zero. Clicking anywhere in the waveform display brings that cursor to the mouse location.

You can also:

- add additional cursors;
- name, lock, and delete cursors;
- use cursors to measure time intervals; and
- use cursors to find transitions.

First, dock the Wave window in the Main window by clicking the dock icon.



Working with a Single Cursor

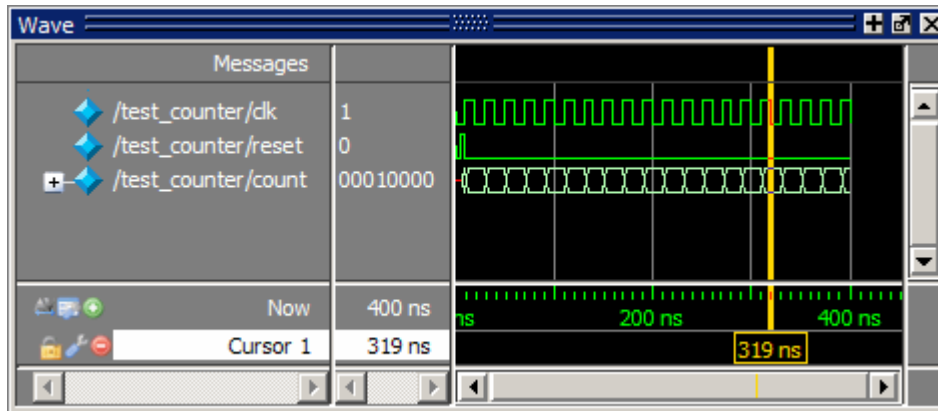
1. Position the cursor by clicking and dragging.

- a. Click the Select Mode icon on the Wave window toolbar.
- b. Click anywhere in the waveform pane.



A cursor is inserted at the time where you clicked (Figure 6-3).

Figure 6-3. Working with a Single Cursor in the Wave Window



- c. Drag the cursor and observe the value pane.

The signal values change as you move the cursor. This is perhaps the easiest way to examine the value of a signal at a particular time.

- d. In the waveform pane, drag the cursor to the right of a transition with the mouse positioned over a waveform.

The cursor "snaps" to the nearest transition to the left. Cursors "snap" to a waveform edge if you click or drag a cursor to within ten pixels of a waveform edge. You can set the snap distance in the Window Preferences dialog (select **Tools > Window Preferences**).

- e. In the cursor pane, drag the cursor to the right of a transition (Figure 6-3).

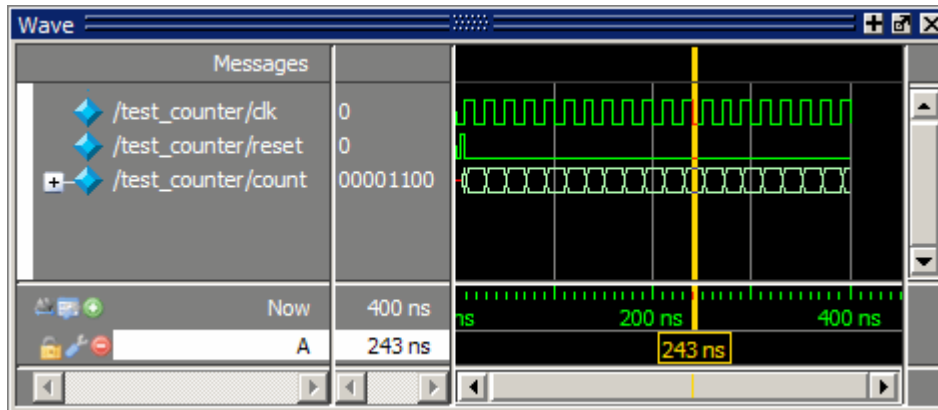
The cursor doesn't snap to a transition if you drag in the cursor pane.

2. Rename the cursor.

- a. Right-click "Cursor 1" in the cursor pane, and select and delete the text.
- b. Type **A** and press Enter.

The cursor name changes to "A" (Figure 6-4).

Figure 6-4. Renaming a Cursor



3. Jump the cursor to the next or previous transition.

a. Click signal *count* in the pathname pane.

b. Click the Find Next Transition icon on the Wave window toolbar.



The cursor jumps to the next transition on the selected signal.

c. Click the Find Previous Transition icon on the Wave window toolbar.



The cursor jumps to the previous transition on the selected signal.

Working with Multiple Cursors

1. Add a second cursor.

a. Click the Insert Cursor icon on the Wave window toolbar.

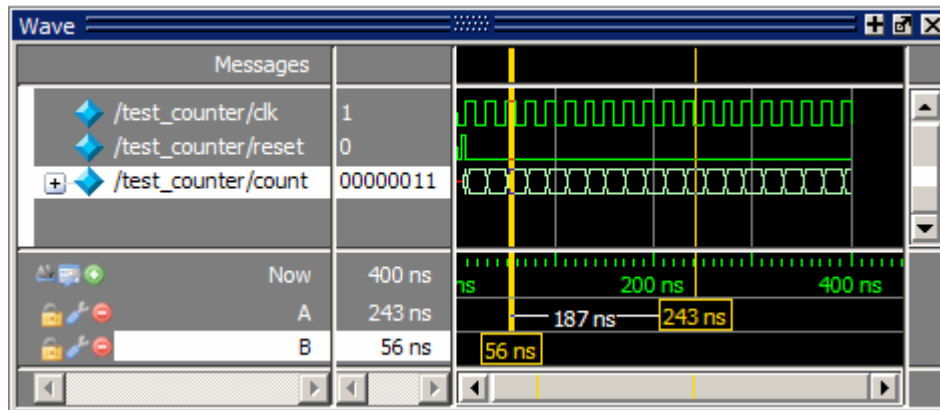


b. Right-click the name of the new cursor and delete the text.

c. Type **B** and press Enter.

d. Drag cursor *B* and watch the interval measurement change dynamically (Figure 6-5).

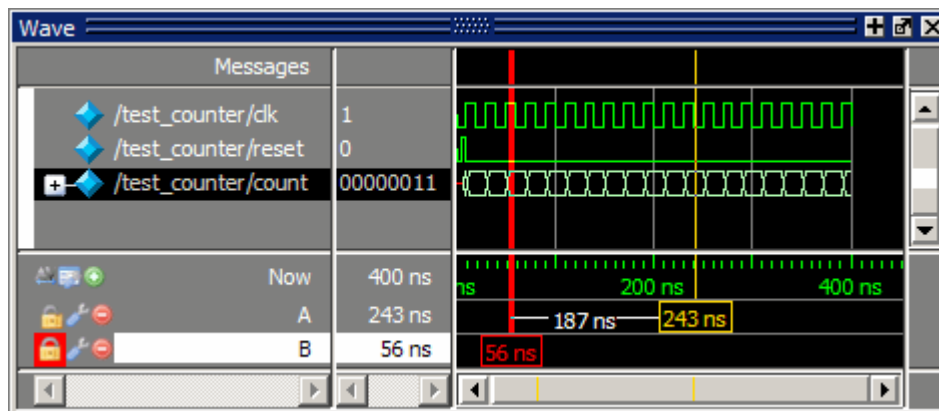
Figure 6-5. Interval Measurement Between Two Cursors



2. Lock cursor *B*.
 - a. Right-click the yellow box associated with cursor *B* (at 56 ns).
 - b. Select **Lock B** from the popup menu.

The cursor color changes to red and you can no longer drag the cursor (Figure 6-6).

Figure 6-6. A Locked Cursor in the Wave Window



3. Delete cursor *B*.
 - a. Right-click cursor *B* (the red box at 56 ns) and select **Delete B**.

Lesson Wrap-Up

This concludes this lesson. Before continuing we need to end the current simulation.

1. Select **Simulate > End Simulation**. Click Yes.

Chapter 7

Viewing And Initializing Memories

Introduction

In this lesson you will learn how to view and initialize memories. ModelSim defines and lists any of the following as memories:

- reg, wire, and std_logic arrays
- Integer arrays
- Single dimensional arrays of VHDL enumerated types other than std_logic

Design Files for this Lesson

The installation comes with Verilog and VHDL versions of the example design located in the following directories:

Verilog – *<install_dir>/examples/tutorials/verilog/memory*

VHDL – *<install_dir>/examples/tutorials/vhdl/memory*

This lesson uses the Verilog version for the exercises. If you have a VHDL license, use the VHDL version instead.

Related Reading

User's Manual Section: [Memory and Memory Data Windows](#).

Reference Manual commands: [mem display](#), [mem load](#), [mem save](#), and [radix](#).

Compile and Load the Design

1. Create a new directory and copy the tutorial files into it.

Start by creating a new directory for this exercise (in case other users will be working with these lessons). Create the directory and copy all files from *<install_dir>/examples/tutorials/verilog/memory* to the new directory.

If you have a VHDL license, copy the files in *<install_dir>/examples/tutorials/vhdl/memory* instead.

2. Start ModelSim and change to the exercise directory.

If you just finished the previous lesson, ModelSim should already be running. If not, start ModelSim.

- a. Type **vsim** at a UNIX shell prompt or use the ModelSim icon in Windows.

If the Welcome to ModelSim dialog appears, click **Close**.

- b. Select **File > Change Directory** and change to the directory you created in step 1.

3. Create the working library and compile the design.

- a. Type **vlib work** at the ModelSim> prompt.

- b. **Verilog:**

Type **vlog *.v** at the ModelSim> prompt to compile all verilog files in the design.

VHDL:

Type **vcom -93 sp_syn_ram.vhd dp_syn_ram.vhd ram_tb.vhd** at the ModelSim> prompt.

4. Load the design.

- a. On the Library tab of the Main window Workspace, click the "+" icon next to the *work* library.

- b. Double-click the *ram_tb* design unit to load the design.

View a Memory and its Contents

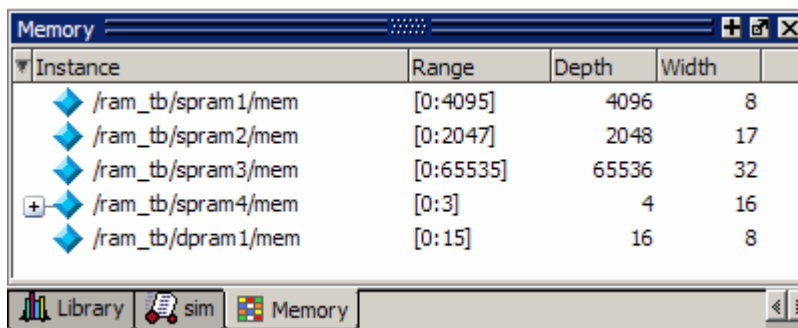
The Memory window lists all memory instances in the design, showing for each instance the range, depth, and width. Double-clicking an instance opens a window displaying the memory data.

1. Open the Memory window and view the data of a memory instance

- a. If the Memory window is not already open, select **View > Memory List**.

A Memory window opens as shown in [Figure 7-1](#).

Figure 7-1. The Memory List in the Memory window



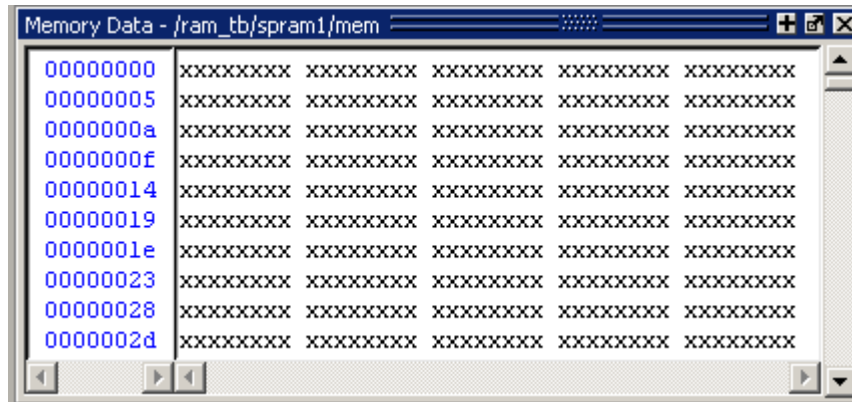
Instance	Range	Depth	Width
◆ /ram_tb/spram1/mem	[0:4095]	4096	8
◆ /ram_tb/spram2/mem	[0:2047]	2048	17
◆ /ram_tb/spram3/mem	[0:65535]	65536	32
+ ◆ /ram_tb/spram4/mem	[0:3]	4	16
◆ /ram_tb/dpram1/mem	[0:15]	16	8

- b. Double-click the `/ram_tb/spram1/mem` instance in the memory list to view its contents.

A Memory Data window opens displaying the contents of `spram1`. The first column (blue hex characters) lists the addresses, and the remaining columns show the data values.

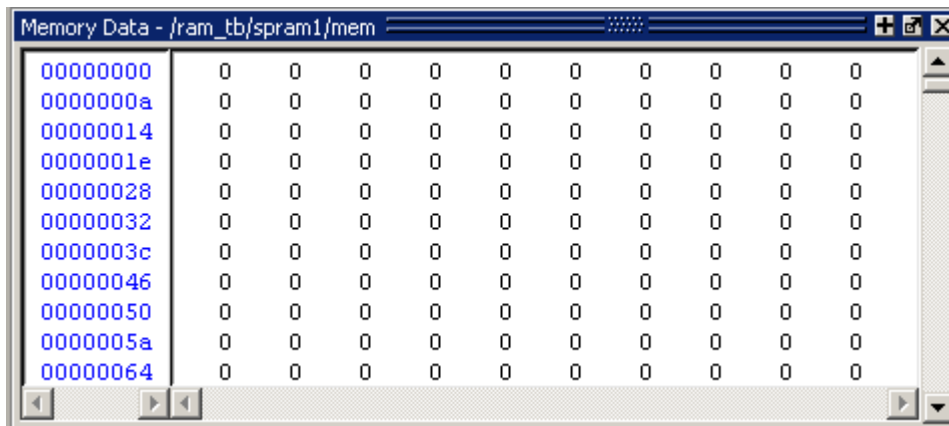
If you are using the Verilog example design, the data is all **X** (Figure 7-2) because you have not yet simulated the design.


Figure 7-2. Verilog Memory Data Window



If you are using the VHDL example design, the data is all zeros (Figure 7-3).

Figure 7-3. VHDL Memory Data Window



- c. Double-click the instance `/ram_tb/spram2/mem` in the Memory window. This opens a second Memory Data window that contains the addresses and data for the `spram2` instance. For each memory instance that you click in the Memory window, a new Memory Data window opens.
2. Simulate the design.
 - a. Click the **run -all** icon in the Main window. 

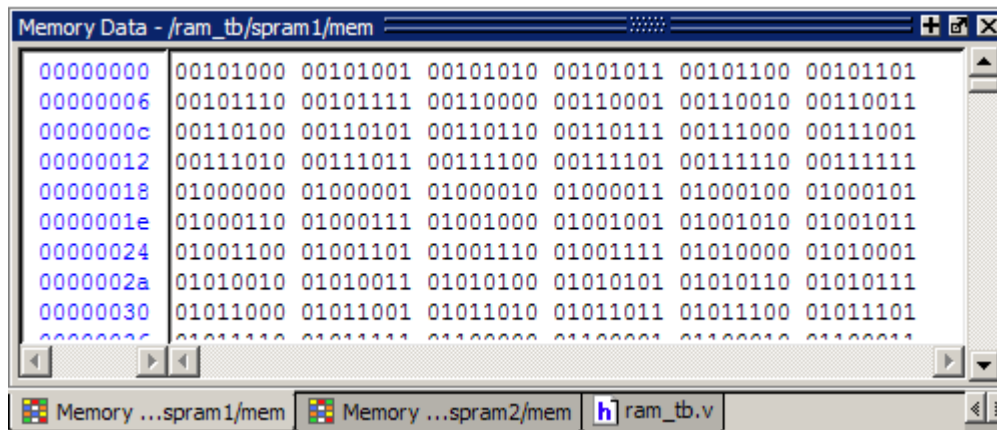
A Source window opens showing the source code for the *ram_tb* file at the point where the simulation stopped.

VHDL:

In the Transcript window, you will see NUMERIC_STD warnings that can be ignored and an assertion failure that is functioning to stop the simulation. The simulation itself has not failed.

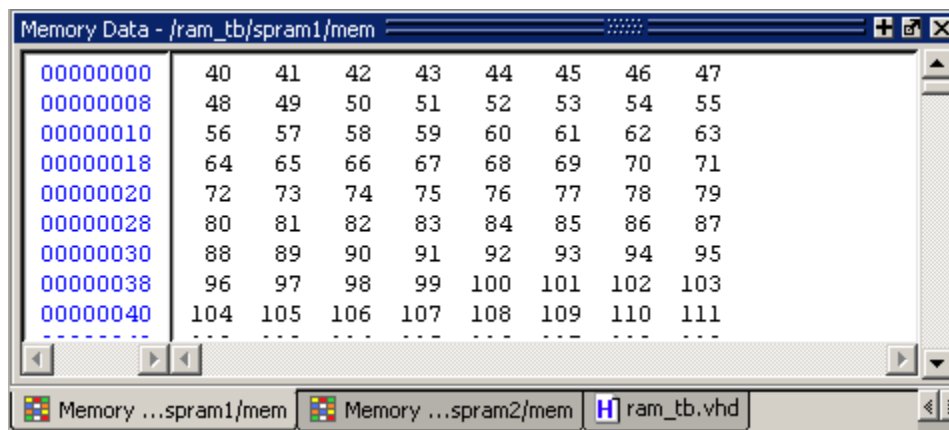
- b. Click the **Memory ...spram1/mem** tab to bring that Memory data window to the foreground. The Verilog data fields are shown in [Figure 7-4](#).

Figure 7-4. Verilog Data After Running Simulation



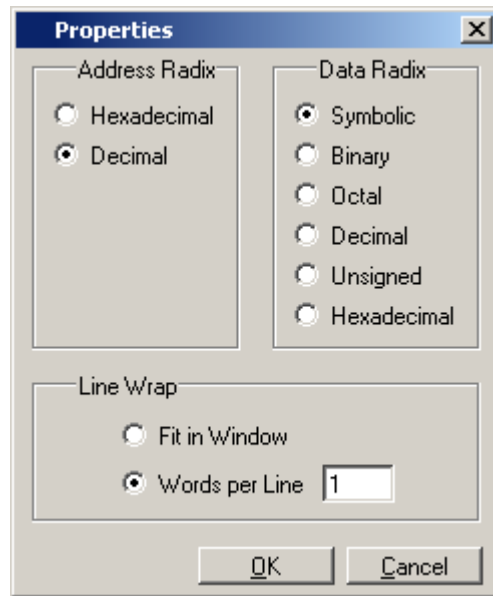
The VHDL data fields are show in [Figure 7-5](#).

Figure 7-5. VHDL Data After Running Simulation



3. Change the address radix and the number of words per line for instance */ram_tb/spram1/mem*.
 - a. Right-click anywhere in the spram1 Memory Data window and select **Properties**.
 - b. The Properties dialog box opens ([Figure 7-6](#)).

Figure 7-6. Changing the Address Radix



- c. For the **Address Radix**, select **Decimal**. This changes the radix for the addresses only.
- d. Select **Words per line** and type **1** in the field.
- e. Click OK.

You can see the Verilog results of the settings in [Figure 7-7](#) and the VHDL results in [Figure 7-8](#). If the figure doesn't match what you have in your ModelSim session, check to make sure you set the Address Radix rather than the Data Radix. Data Radix should still be set to Symbolic, the default.

Figure 7-7. New Address Radix and Line Length (Verilog)

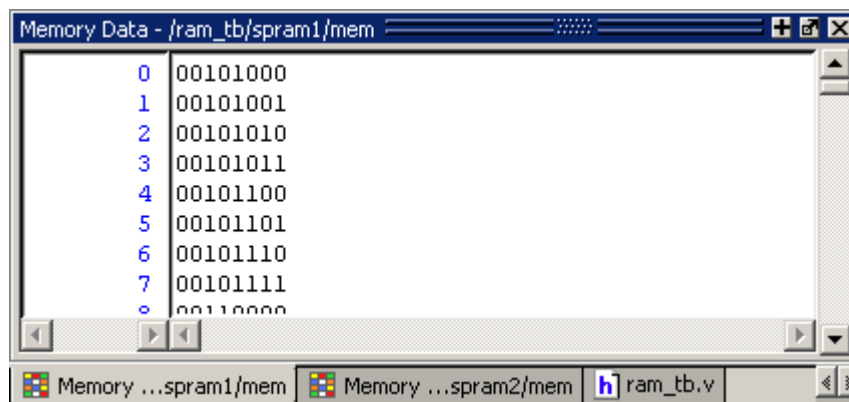
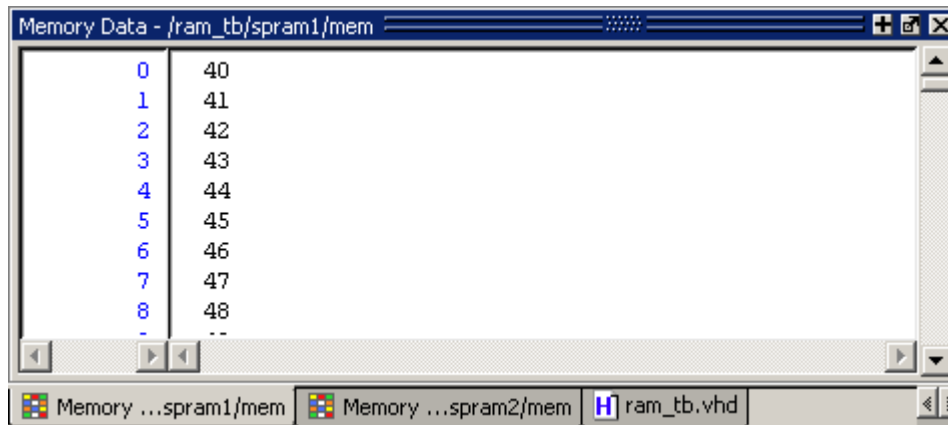


Figure 7-8. New Address Radix and Line Length (VHDL)



Navigate Within the Memory

You can navigate to specific memory address locations, or to locations containing particular data patterns. First, you will go to a specific address.

1. Use Goto to find a specific address.
 - a. Right-click anywhere in address column and select **Goto** (Figure 7-9).

The Goto dialog box opens in the data pane.

Figure 7-9. Goto Dialog

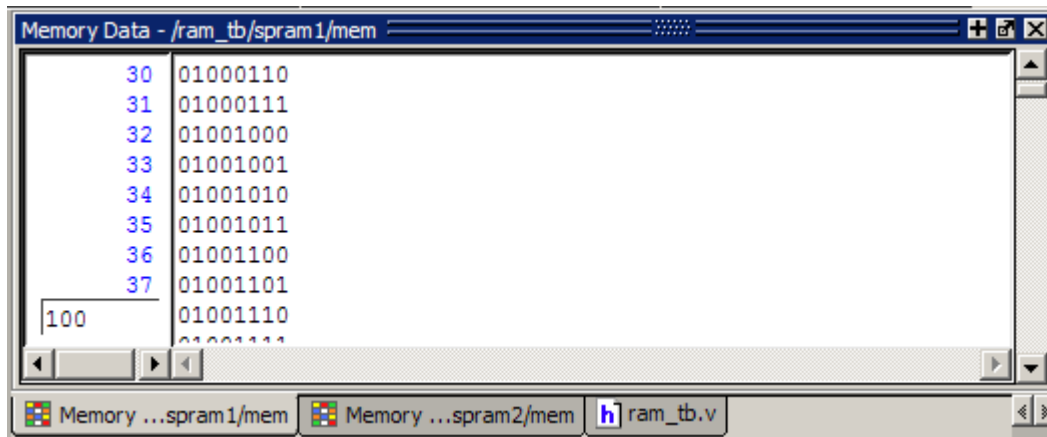


- b. Type **30** in the Goto Address field.
 - c. Click **OK**.

The requested address appears in the top line of the window.

2. Edit the address location directly.
 - a. To quickly move to a particular address, do the following:
 - i. Double click address 38 in the address column.
 - ii. Enter address 100 (Figure 7-10).

Figure 7-10. Editing the Address Directly



iii. Press the Enter or Return key on your keyboard.

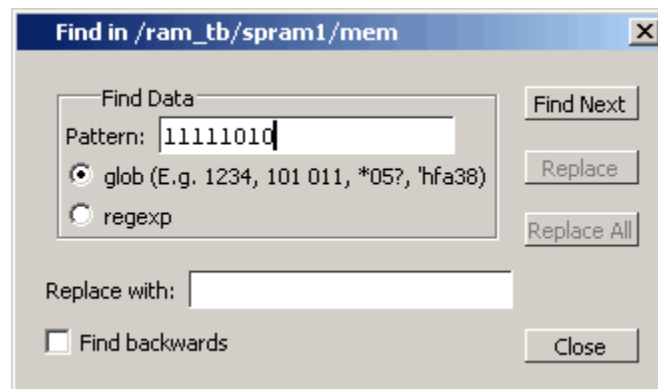
The pane jumps to address 100.

3. Now, let's find a particular data entry.

a. Right-click anywhere in the data column and select **Find**.

The Find in dialog box opens (Figure 7-11).

Figure 7-11. Searching for a Specific Data Value



b. **Verilog:** Type **11111010** in the **Find data:** field and click **Find Next**.

VHDL: Type **250** in the **Find data:** field and click **Find Next**.

The data scrolls to the first occurrence of that address. Click **Find Next** a few more times to search through the list.

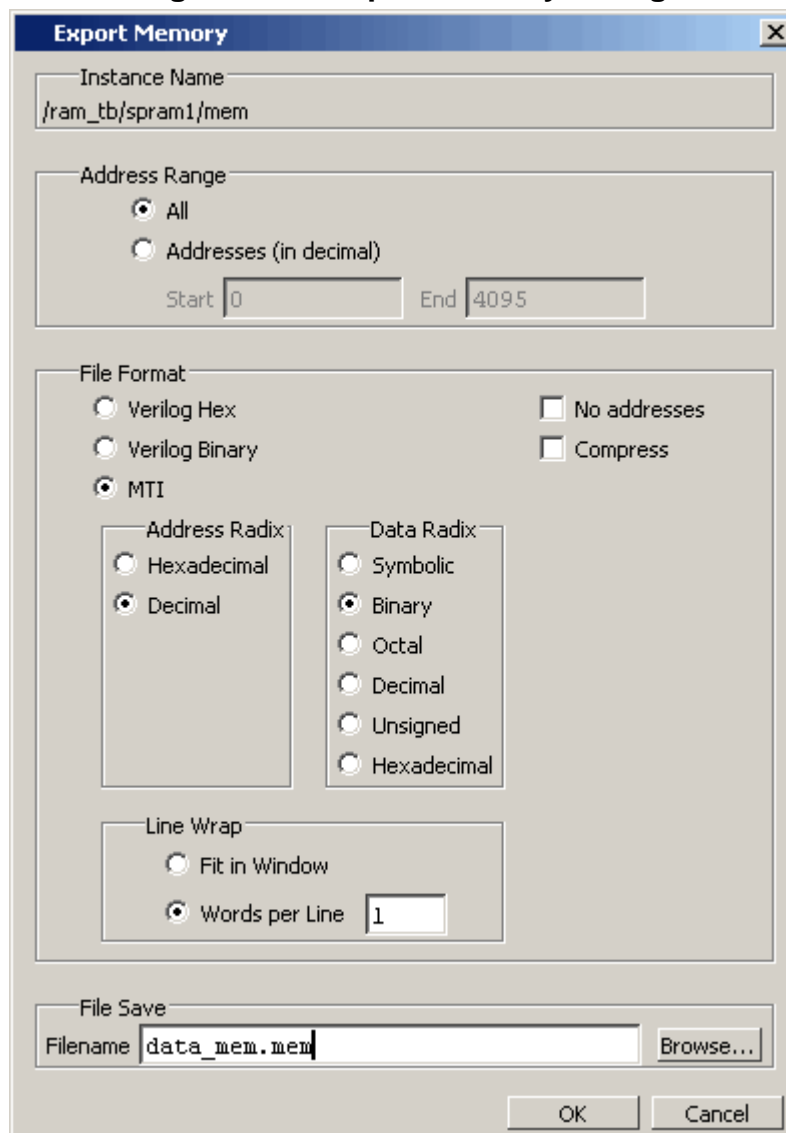
c. Click **Close** to close the dialog box.

Export Memory Data to a File

You can save memory data to a file that can be loaded at some later point in simulation.

1. Export a memory pattern from the `/ram_tb/spram1/mem` instance to a file.
 - a. Make sure `/ram_tb/spram1/mem` is open and selected.
 - b. Select **File > Export > Memory Data** to bring up the Export Memory dialog box (Figure 7-12).

Figure 7-12. Export Memory Dialog



- c. For the Address Radix, select **Decimal**.
- d. For the Data Radix, select **Binary**.

- e. For the Words per Line, set to 1.
- f. Type **data_mem.mem** into the Filename field.
- g. Click OK.

You can view the exported file in any editor.

Memory pattern files can be exported as relocatable files, simply by leaving out the address information. Relocatable memory files can be loaded anywhere in a memory because no addresses are specified.

2. Export a relocatable memory pattern file from the */ram_tb/spram2/mem* instance.
 - a. Select the Memory Data window for the */ram_tb/spram2/mem* instance.
 - b. Right-click on the memory contents to open a popup menu and select **Properties**.
 - c. In the Properties dialog, set the Address Radix to **Decimal**; the Data Radix to **Binary**; and the Line Wrap to 1 **Words per Line**. Click OK to accept the changes and close the dialog.
 - d. Select **File > Export > Memory Data** to bring up the Export Memory dialog box.
 - e. For the Address Range, specify a Start address of **0** and End address of **250**.
 - f. For the File Format, select **MTI** and **No addresses** to create a memory pattern that you can use to relocate somewhere else in the memory, or in another memory.
 - g. For Address Radix select **Decimal**, and for Data Radix select **Binary**.
 - h. For the Words per Line, set to 1.
 - i. Enter the file name as **reloc.mem**, then click OK to save the memory contents and close the dialog. You will use this file for initialization in the next section.

Initialize a Memory

In ModelSim, it is possible to initialize a memory using one of three methods: from an exported memory file, from a fill pattern, or from both.

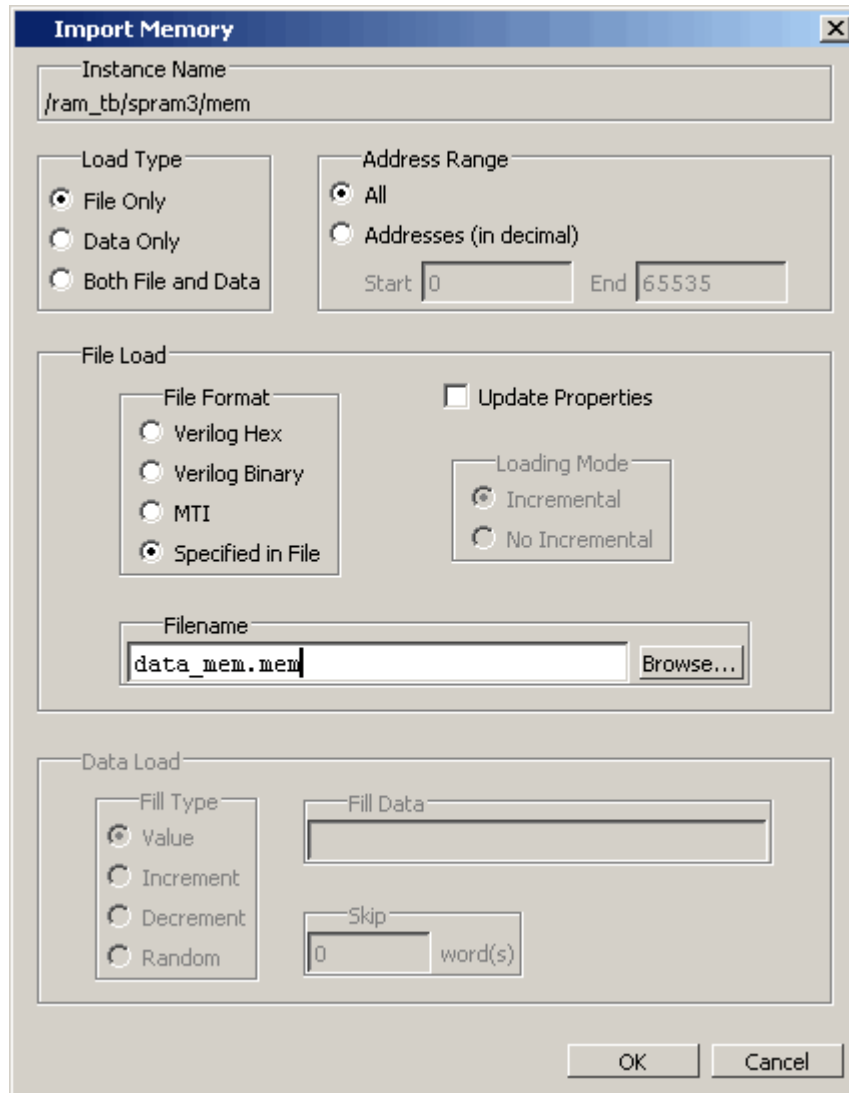
First, let's initialize a memory from a file only. You will use the one you exported previously, *data_mem.mem*.

1. View instance */ram_tb/spram3/mem*.
 - a. Double-click the */ram_tb/spram3/mem* instance in the Memories tab.

This will open a new Memory Data window to display the contents of */ram_tb/spram3/mem*. Familiarize yourself with the contents so you can identify changes once the initialization is complete.

- b. Right-click and select **Properties** to bring up the Properties dialog.
 - c. Change the Address Radix to **Decimal**, Data Radix to **Binary**, Words per Line to 1, and click OK.
2. Initialize *spram3* from a file.
 - a. Right-click anywhere in the data column and select **Import Data Patterns** to bring up the Import Memory dialog box (Figure 7-13).

Figure 7-13. Import Memory Dialog

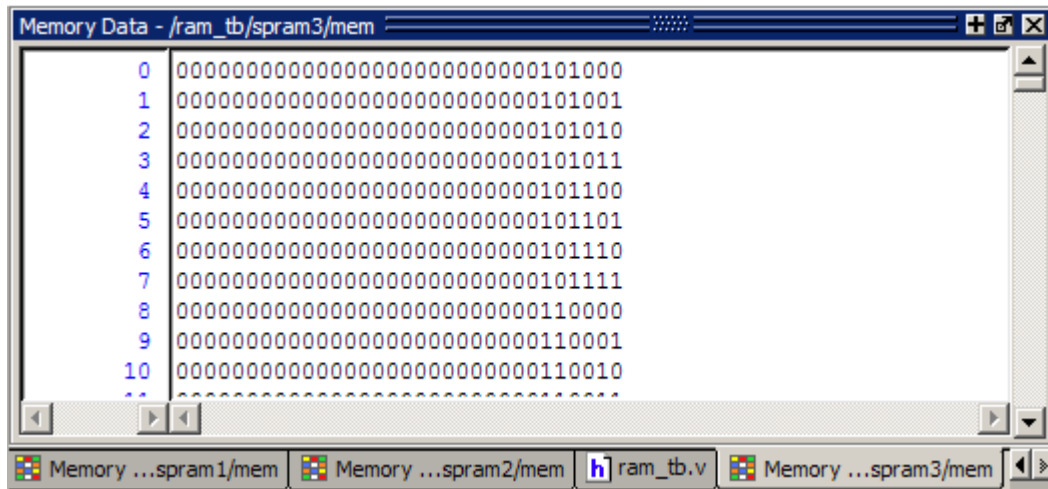


The default Load Type is File Only.

- b. Type *data_mem.mem* in the Filename field.
- c. Click **OK**.

The addresses in instance `/ram_tb/spram3/mem` are updated with the data from `data_mem.mem` (Figure 7-14).

Figure 7-14. Initialized Memory from File and Fill Pattern



In this next step, you will experiment with importing from both a file and a fill pattern. You will initialize `spram3` with the 250 addresses of data you exported previously into the relocatable file `reloc.mem`. You will also initialize 50 additional address entries with a fill pattern.

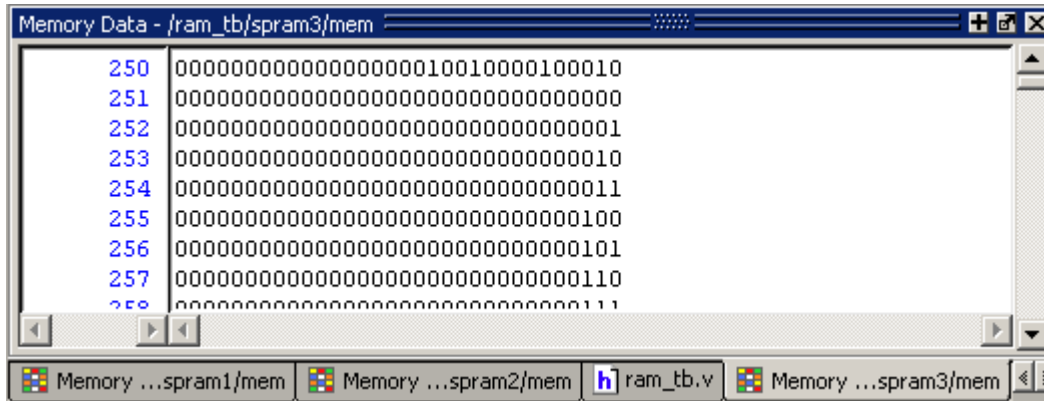
3. Import the `/ram_tb/spram3/mem` instance with a relocatable memory pattern (`reloc.mem`) and a fill pattern.
 - a. Right-click in the data column of `spram3` and select **Import Data Patterns** to bring up the Import Memory dialog box.
 - b. For Load Type, select **Both File and Data**.
 - c. For Address Range, select **Addresses** and enter **0** as the Start address and **300** as the End address.

This means that you will be loading the file from 0 to 300. However, the `reloc.mem` file contains only 251 addresses of data. Addresses 251 to 300 will be loaded with the fill data you specify next.

- d. For File Load, select the MTI File Format and enter **reloc.mem** in the Filename field.
- e. For Data Load, select a Fill Type of **Increment**.
- f. In the Fill Data field, set the seed value of **0** for the incrementing data.
- g. Click **OK**.
- h. View the data near address 250 by double-clicking on any address in the Address column and entering **250**.

You can see the specified range of addresses overwritten with the new data. Also, you can see the incrementing data beginning at address 251 (Figure 7-15).

Figure 7-15. Data Increments Starting at Address 251



Now, before you leave this section, go ahead and clear the memory instances already being viewed.

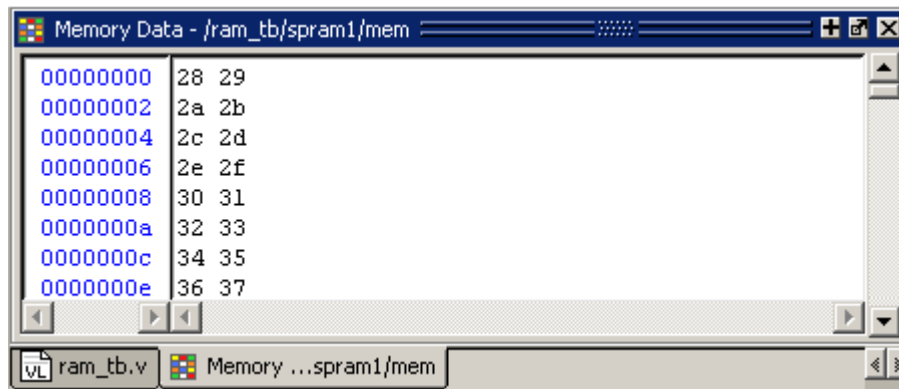
4. Right-click in one of the Memory Data windows and select **Close All**.

Interactive Debugging Commands

The Memory Data windows can also be used interactively for a variety of debugging purposes. The features described in this section are useful for this purpose.

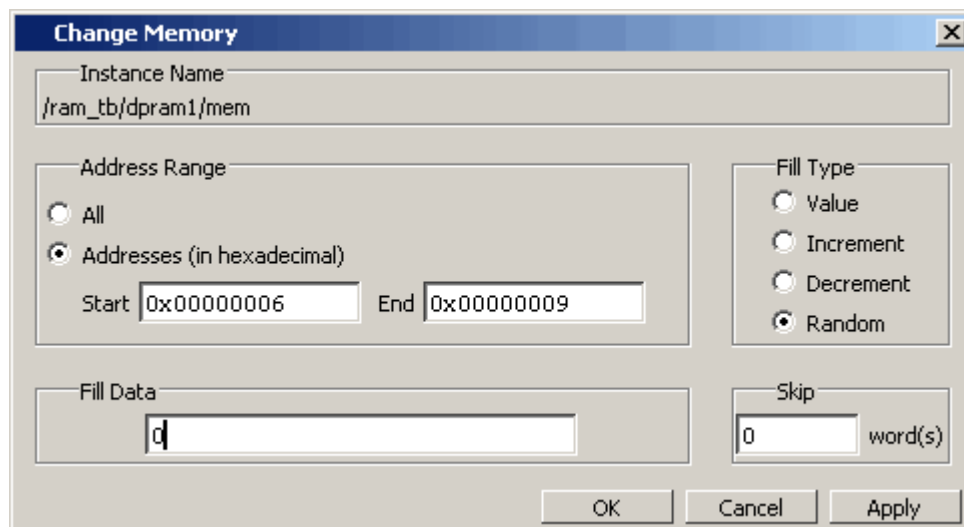
1. Open a memory instance and change its display characteristics.
 - a. Double-click instance */ram_tb/dpram1/mem* in the Memories window.
 - b. Right-click in the *dpram1* Memory Data window and select **Properties**.
 - c. Change the Address and Data Radix to **Hexadecimal**.
 - d. Select **Words per line** and enter **2**.
 - e. Click **OK**. The result should be as in Figure 7-16.

Figure 7-16. Original Memory Content



2. Initialize a range of memory addresses from a fill pattern.
 - a. Right-click in the data column of `/ram_tb/dpram1/mem` and select **Change** to open the Change Memory dialog (Figure 7-17).

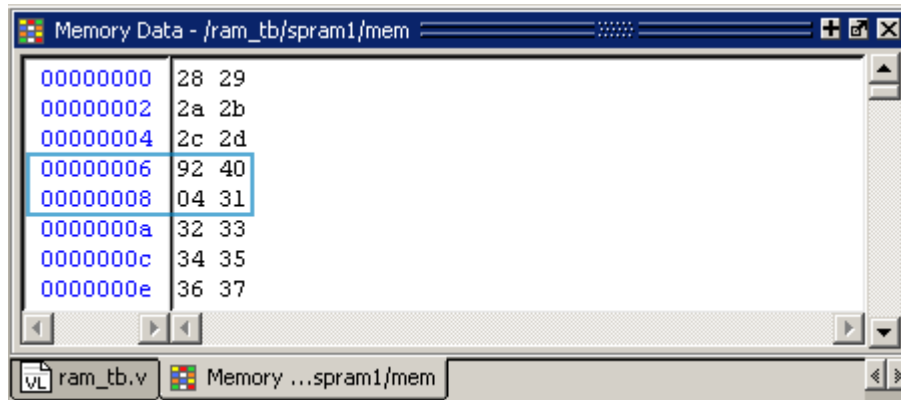
Figure 7-17. Changing Memory Content for a Range of AddressesOK**



- b. Select **Addresses** and enter the start address as **0x00000006** and the end address as **0x00000009**. The "0x" hex notation is optional.
- c. Select **Random** as the **Fill Type**.
- d. Enter **0** as the **Fill Data**, setting the seed for the Random pattern.
- e. Click **OK**.

The data in the specified range are replaced with a generated random fill pattern (Figure 7-18).

Figure 7-18. Random Content Generated for a Range of Addresses

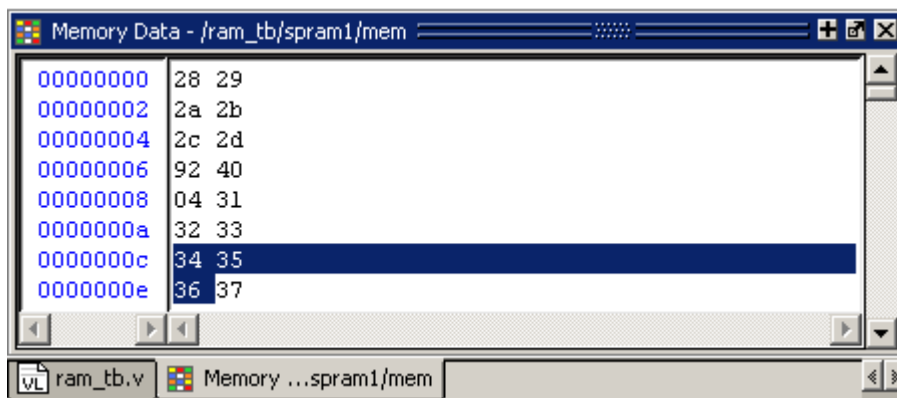


3. Change contents by highlighting.

You can also change data by highlighting them in the Address Data pane.

- a. Highlight the data for the addresses **0x0000000c:0x0000000e**, as shown in [Figure 7-19](#).

Figure 7-19. Changing Memory Contents by Highlighting

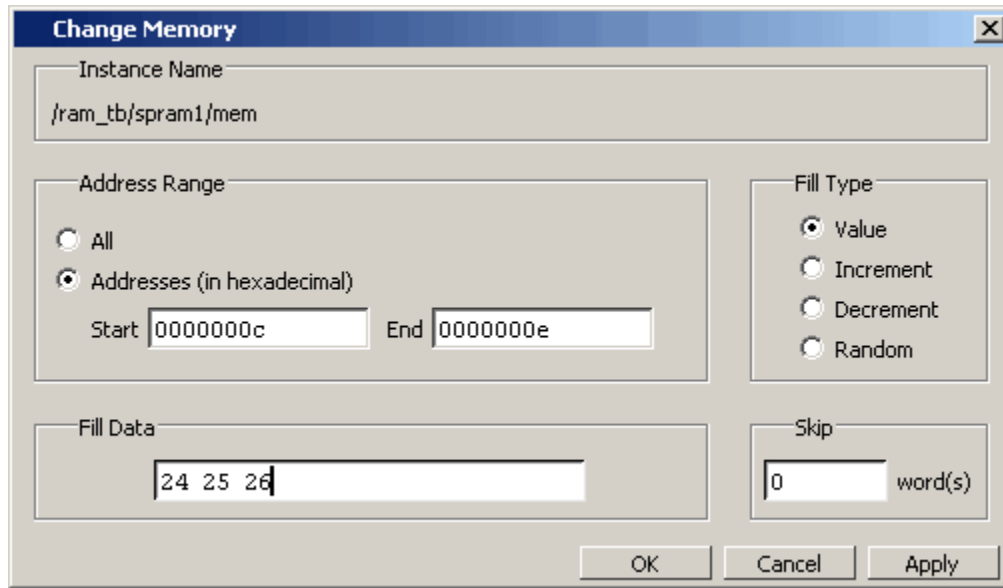


- b. Right-click the highlighted data and select **Change**.

This brings up the Change memory dialog box. Note that the Addresses field is already populated with the range you highlighted.

- c. Select **Value** as the Fill Type. (Refer to [Figure 7-20](#))
- d. Enter the data values into the Fill Data field as follows: **24 25 26**.

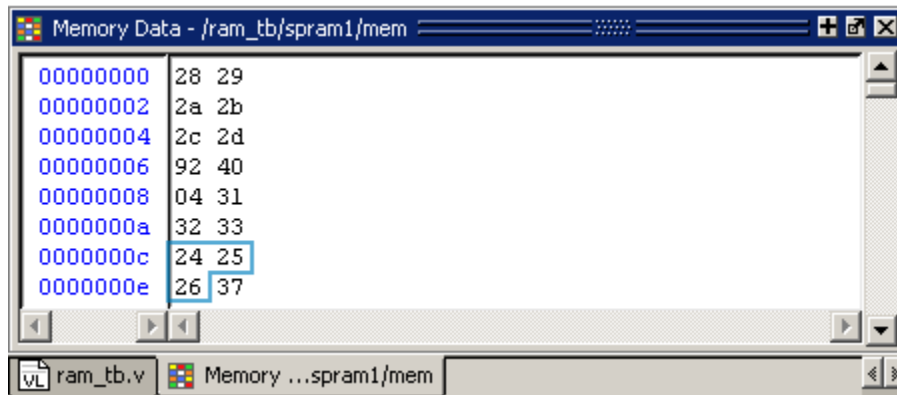
Figure 7-20. Entering Data to ChangeOK**



e. Click **OK**.

The data in the address locations change to the values you entered ([Figure 7-21](#)).

Figure 7-21. Changed Memory Contents for the Specified Addresses



4. Edit data in place.

To edit only one value at a time, do the following:

- a. Double click any value in the Data column.
- b. Enter the desired value and press the Enter or Return key on your keyboard.

If you needed to cancel the edit function, press the Esc key on your keyboard.

Lesson Wrap-Up

This concludes this lesson. Before continuing we need to end the current simulation.

1. Select **Simulate > End Simulation**. Click Yes.

Chapter 8

Automating Simulation

Introduction

Aside from executing a couple of pre-existing DO files, the previous lessons focused on using ModelSim in interactive mode: executing single commands, one after another, via the GUI menus or Main window command line. In situations where you have repetitive tasks to complete, you can increase your productivity with DO files.

DO files are scripts that allow you to execute many commands at once. The scripts can be as simple as a series of ModelSim commands with associated arguments, or they can be full-blown Tcl programs with variables, conditional execution, and so forth. You can execute DO files from within the GUI or you can run them from the system command prompt without ever invoking the GUI.

Note



This lesson assumes that you have added the `<install_dir>/<platform>` directory to your PATH. If you did not, you will need to specify full paths to the tools (i.e., vlib, vmap, vlog, vcom, and vsim) that are used in the lesson.

Related Reading

User's Manual Chapter: [Tcl and Macros \(DO Files\)](#).

Practical Programming in Tcl and Tk, Brent B. Welch, Copyright 1997

Creating a Simple DO File

Creating a DO file is as simple as typing a set of commands in a text file. In this exercise, you will create a DO file that loads a design, adds signals to the Wave window, provides stimulus to those signals, and then advances the simulation. You can also create a DO file from a saved transcript file. Refer to "[Saving a Transcript File as a Macro \(DO file\)](#)."

1. Change to the directory you created in the "Basic Simulation" lesson.
2. Create a DO file that will add signals to the Wave window, force signals, and run the simulation.
 - a. Select **File > New > Source > Do** to create a new DO file.
 - b. Enter the following commands into the Source window:

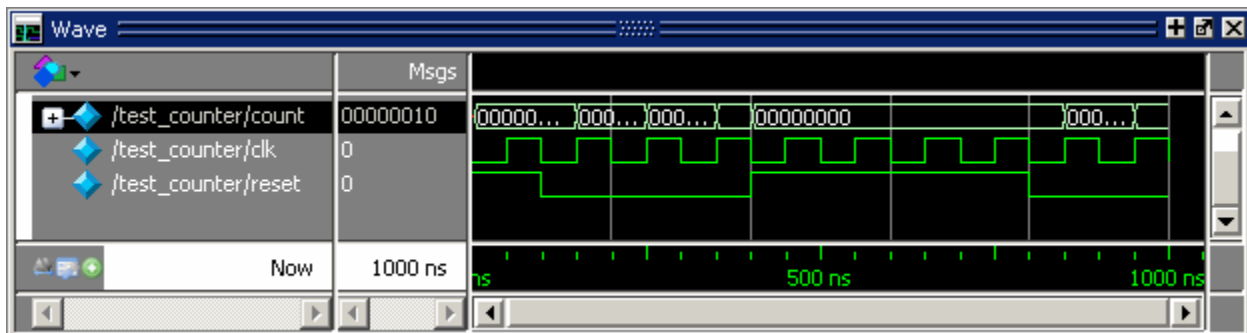
```
vsim test_counter
```

```
add wave count
add wave clk
add wave reset
force -freeze clk 0 0, 1 {50 ns} -r 100
force reset 1
run 100
force reset 0
run 300
force reset 1
run 400
force reset 0
run 200
```

3. Save the file.
 - a. Select **File > Save As**.
 - b. Type **sim.do** in the File name: field and save it to the current directory.
4. Execute the DO file.
 - a. Enter **do sim.do** at the VSIM> prompt.

ModelSim loads the design, executes the saved commands and draws the waves in the Wave window.

Figure 8-1. Wave Window After Running the DO File



5. When you are done with this exercise, select **File > Quit** to quit ModelSim.

Running in Command-Line Mode

We use the term "command-line mode" to refer to simulations that are run from a DOS/ UNIX prompt without invoking the GUI. Several ModelSim commands (e.g., vsim, vlib, vlog, etc.) are actually stand-alone executables that can be invoked at the system command prompt. Additionally, you can create a DO file that contains other ModelSim commands and specify that file when you invoke the simulator.

1. Create a new directory and copy the tutorial files into it.

Start by creating a new directory for this exercise. Create the directory and copy the following files into it:

- `<install_dir>/examples/tutorials/verilog/automation/counter.v`
- `<install_dir>/examples/tutorials/verilog/automation/stim.do`

This lesson uses the Verilog file `counter.v`. If you have a VHDL license, use *the* `counter.vhd` and `stim.do` files in the `<install_dir>/examples/tutorials/vhdl/automation` directory instead.

2. Create a new design library and compile the source file.

Again, enter these commands at a DOS/ UNIX prompt in the new directory you created in step 1.

- a. Type **vlib work** at the DOS/ UNIX prompt.
- b. For Verilog, type **vlog counter.v** at the DOS/ UNIX prompt. For VHDL, type **vcom counter.vhd**.

3. Create a DO file.

- a. Open a text editor.
- b. Type the following lines into a new file:

```
# list all signals in decimal format
add list -decimal *

# read in stimulus
do stim.do

# output results
write list counter.lst

# quit the simulation
quit -f
```

- c. Save the file with the name `sim.do` and place it in the current directory.

4. Run the batch-mode simulation.

- a. Enter the following command at the DOS/UNIX prompt:

```
vsim -c -do sim.do counter -wlf counter.wlf
```

The `-c` argument instructs ModelSim not to invoke the GUI. The `-wlf` argument saves the simulation results in a WLF file. This allows you to view the simulation results in the GUI for debugging purposes.

5. View the list output.

- a. Open `counter.lst` and view the simulation results. Output produced by the Verilog version of the design should look like the following:

```
ns      /counter/count
delta   /counter/clk
        /counter/reset
0 +0    x z *
3 +0    0 z *
50 +0   0 * *
100 +0  0 0 *
100 +1  0 0 0
150 +0  0 * 0
152 +0  1 * 0
200 +0  1 0 0
250 +0  1 * 0
.
.
.
```

The output may appear slightly different if you used the VHDL version.

6. View the results in the GUI.

Since you saved the simulation results in *counter.wlf*, you can view them in the GUI by invoking VSIM with the **-view** argument.

Note

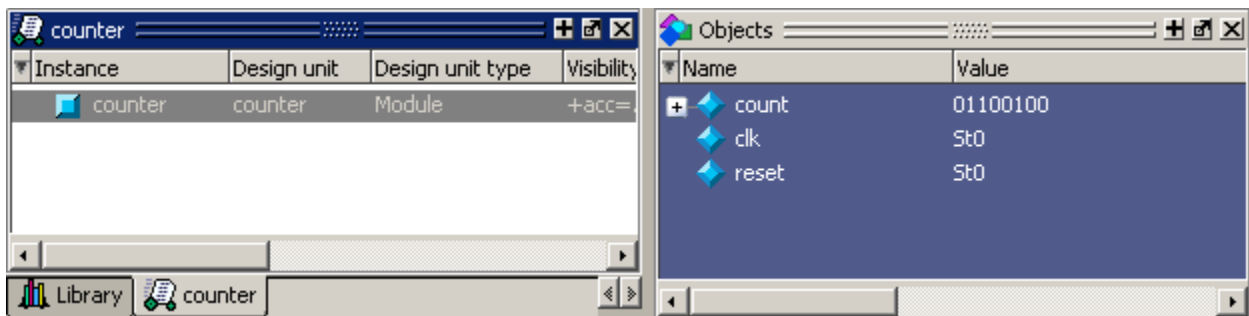


Make sure your PATH environment variable is set with the current version of ModelSim at the front of the string.

a. Type **vsim -view counter.wlf** at the DOS/ UNIX prompt.

The GUI opens and a dataset tab named "counter" is displayed (Figure 8-2).

Figure 8-2. The counter.wlf Dataset in the Main Window Workspace



b. Right-click the *counter* instance and select **Add > To Wave > All items in region**.

The waveforms display in the Wave window.

7. When you finish viewing the results, select **File > Quit** to close ModelSim.

Using Tcl with the Simulator

The DO files used in previous exercises contained only ModelSim commands. However, DO files are really just Tcl scripts. This means you can include a whole variety of Tcl constructs such as procedures, conditional operators, math and trig functions, regular expressions, and so forth.

In this exercise, you create a simple Tcl script that tests for certain values on a signal and then adds bookmarks that zoom the Wave window when that value exists. Bookmarks allow you to save a particular zoom range and scroll position in the Wave window.

1. Create the script.

- a. In a text editor, open a new file and enter the following lines:

```
proc add_wave_zoom {stime num} {  
    echo "Bookmarking wave $num"  
    bookmark add wave "bk$num" "[expr $stime - 50] [expr $stime + 100]" 0  
}
```

These commands do the following:

- Create a new procedure called "add_wave_zoom" that has two arguments, *stime* and *num*.
- Create a bookmark with a zoom range from the current simulation time minus 50 time units to the current simulation time plus 100 time units.

- b. Now add these lines to the bottom of the script:

```
add wave -r /*  
when {clk'event and clk="1"} {  
    echo "Count is [exa count]"  
    if {[examine count]== "00100111"} {  
        add_wave_zoom $now 1  
    } elseif {[examine count]== "01000111"} {  
        add_wave_zoom $now 2  
    }  
}
```

These commands do the following:

- Add all signals to the Wave window.
- Use a **when** statement to identify when *clk* transitions to 1.
- Examine the value of *count* at those transitions and add a bookmark if it is a certain value.

- c. Save the script with the name "*add_bkmrk.do*" into the directory you created in the [Basic Simulation](#) lesson.

2. Load the *test_counter* design unit.

- a. Start ModelSim.

- b. Select **File > Change Directory** and change to the directory you saved the DO file to in step 1c above.
 - c. Enter the following command at the QuestaSim> prompt:

```
vsim test_counter
```
3. Execute the DO file and run the design.
 - a. Type **do add_bkmrk.do** at the VSIM> prompt.
 - b. Type **run 1500 ns** at the VSIM> prompt.

The simulation runs and the DO file creates two bookmarks.
 - c. If the Wave window is docked in the Main window make it the active window (click anywhere in the Wave window), then select **Wave > Bookmarks > bk1**. If the window is undocked, select **View > Bookmarks > bk1** in the Wave window.

Watch the Wave window zoom in and scroll to the time when *count* is 00100111.
Try the **bk2** bookmark as well.

Lesson Wrap-Up

This concludes this lesson.

1. Select **File > Quit** to close ModelSim.

— A —

add wave command, 49
al, 55

— B —

break icon, 22
breakpoints
 setting, 22
 stepping, 25

— C —

command-line mode, 72
Compile, 17
compile order, changing, 30
compiling your design, 12
cursors, Wave window, 50

— D —

design library
 working type, 13

— E —

error messages, more information, 43
external libraries, linking to, 42

— F —

folders, in projects, 33

— L —

libraries
 design library types, 13
 linking to external libraries, 42
 mapping to permanently, 45
 resource libraries, 13
 working libraries, 13
 working, creating, 15
linking to external libraries, 42

— M —

mapping libraries permanently, 45
memories
 changing values, 67

 initializing, 63

memory contents, saving to a file, 62

— O —

options, simulation, 35

— P —

projects
 adding items to, 28
 creating, 27
 flow overview, 12
 organizing with folders, 33
 simulation configurations, 35

— Q —

quit command, 43, 44

— R —

run -all, 22
run command, 21

— S —

saving simulation options, 35
simulation
 basic flow overview, 11
 restarting, 23
 running, 20
simulation configurations, 35
stepping after a breakpoint, 25

— T —

Tcl, using in the simulator, 75
time, measuring in Wave window, 50

— V —

vcom command, 56
verror command, 43
vlib command, 56
vlog command, 56
vsim command, 16

— W —

Wave window

adding items to, [48](#)

cursors, [50](#)

measuring time with cursors, [50](#)

zooming, [49](#)

working library, creating, [11](#), [15](#)

— Z —

zooming, Wave window, [49](#)

End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:
www.mentor.com/eula

IMPORTANT INFORMATION

USE OF THIS SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE SOFTWARE. USE OF SOFTWARE INDICATES YOUR COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.

END-USER LICENSE AGREEMENT (“Agreement”)

This is a legal agreement concerning the use of Software (as defined in Section 2) between the company acquiring the license (“Customer”), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity (“Mentor Graphics”). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, if received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

1. ORDERS, FEES AND PAYMENT.

- 1.1. To the extent Customer (or if and as agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement (“Order(s)”), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not these documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order will not be effective unless agreed in writing by an authorized representative of Customer and Mentor Graphics.
- 1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will invoice separately. Unless provided with a certificate of exemption, Mentor Graphics will invoice Customer for all applicable taxes. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. Notwithstanding anything to the contrary, if Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under such orders in the event of default by the third party.
- 1.3. All products are delivered FCA factory (Incoterms 2000) except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all products delivered under this Agreement, to secure payment of the purchase price of such products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation and design data (“Software”) are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form; (b) for Customer's internal business purposes; (c) for the term; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer requests any change or enhancement to Software, whether in the course of receiving support or consulting services, evaluating Software or

otherwise, any inventions, product improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

3. **ESC SOFTWARE.** If Customer purchases a license to use development or prototyping tools of Mentor Graphics' Embedded Software Channel ("ESC"), Mentor Graphics grants to Customer a nontransferable, nonexclusive license to reproduce and distribute executable files created using ESC compilers, including the ESC run-time libraries distributed with ESC C and C++ compiler Software that are linked into a composite program as an integral part of Customer's compiled computer program, provided that Customer distributes these files only in conjunction with Customer's compiled computer program. Mentor Graphics does NOT grant Customer any right to duplicate, incorporate or embed copies of Mentor Graphics' real-time operating systems or other embedded software products into Customer's products or applications without first signing or otherwise agreeing to a separate agreement with Mentor Graphics for such purpose.
4. **BETA CODE.**
 - 4.1. Portions or all of certain Software may contain code for experimental testing and evaluation ("Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. This grant and Customer's use of the Beta Code shall not be construed as marketing or offering to sell a license to the Beta Code, which Mentor Graphics may choose not to release commercially in any form.
 - 4.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.
 - 4.3. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 4.3 shall survive termination of this Agreement.
5. **RESTRICTIONS ON USE.**
 - 5.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Software available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Software and ensure that any person permitted access does not disclose or use it except as permitted by this Agreement. Log files, data files, rule files and script files generated by or for the Software (collectively "Files") constitute and/or include confidential information of Mentor Graphics. Customer may share Files with third parties excluding Mentor Graphics competitors provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") mean Mentor Graphics' proprietary syntaxes for expressing process rules. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Software or allow its use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Software, or disclose to any third party the results of, or information pertaining to, any benchmark. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, reverse-compile, reverse-engineer or in any way derive from Software any source code.
 - 5.2. Customer may not sublicense, assign or otherwise transfer Software, this Agreement or the rights under it, whether by operation of law or otherwise ("attempted transfer"), without Mentor Graphics' prior written consent and payment of Mentor Graphics' then-current applicable transfer charges. Any attempted transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and licenses granted under this Agreement. The terms of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.
 - 5.3. The provisions of this Section 5 shall survive the termination of this Agreement.
6. **SUPPORT SERVICES.** To the extent Customer purchases support services for Software, Mentor Graphics will provide Customer with available updates and technical support for the Software which are made generally available by Mentor Graphics as part of such services in accordance with Mentor Graphics' then current End-User Software Support Terms located at <http://supportnet.mentor.com/about/legal/>.

7. LIMITED WARRANTY.

7.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Software, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Software will meet Customer's requirements or that operation of Software will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under the applicable Order and does not renew or reset, by way of example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Software has been subject to misuse, unauthorized modification or improper installation. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF SOFTWARE TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF SOFTWARE THAT DOES NOT MEET THIS LIMITED WARRANTY, PROVIDED CUSTOMER HAS OTHERWISE COMPLIED WITH THIS AGREEMENT. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) SOFTWARE WHICH IS LICENSED AT NO COST; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."

7.2. THE WARRANTIES SET FORTH IN THIS SECTION 7 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO SOFTWARE OR OTHER MATERIAL PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.

8. **LIMITATION OF LIABILITY.** EXCEPT WHERE THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT PAID BY CUSTOMER FOR THE SOFTWARE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 8 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

9. **LIFE ENDANGERING APPLICATIONS.** NEITHER MENTOR GRAPHICS NOR ITS LICENSORS SHALL BE LIABLE FOR ANY DAMAGES RESULTING FROM OR IN CONNECTION WITH THE USE OF SOFTWARE IN ANY APPLICATION WHERE THE FAILURE OR INACCURACY OF THE SOFTWARE MIGHT RESULT IN DEATH OR PERSONAL INJURY. THE PROVISIONS OF THIS SECTION 9 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

10. **INDEMNIFICATION.** CUSTOMER AGREES TO INDEMNIFY AND HOLD HARMLESS MENTOR GRAPHICS AND ITS LICENSORS FROM ANY CLAIMS, LOSS, COST, DAMAGE, EXPENSE OR LIABILITY, INCLUDING ATTORNEYS' FEES, ARISING OUT OF OR IN CONNECTION WITH CUSTOMER'S USE OF SOFTWARE AS DESCRIBED IN SECTION 9. THE PROVISIONS OF THIS SECTION 10 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

11. INFRINGEMENT.

11.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Software product infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay any costs and damages finally awarded against Customer that are attributable to the action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.

11.2. If a claim is made under Subsection 11.1 Mentor Graphics may, at its option and expense, (a) replace or modify Software so that it becomes noninfringing, or (b) procure for Customer the right to continue using Software, or (c) require the return of Software and refund to Customer any license fee paid, less a reasonable allowance for use.

11.3. Mentor Graphics has no liability to Customer if the claim is based upon: (a) the combination of Software with any product not furnished by Mentor Graphics; (b) the modification of Software other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of Software as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code; (g) any Software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; or (h) infringement by Customer that is deemed willful. In the case of (h), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.

11.4. THIS SECTION IS SUBJECT TO SECTION 8 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY SOFTWARE LICENSED UNDER THIS AGREEMENT.

12. **TERM.**

- 12.1. This Agreement remains effective until expiration or termination. This Agreement will immediately terminate upon notice if you exceed the scope of license granted or otherwise fail to comply with the provisions of Sections 2, 3, or 5. For any other material breach under this Agreement, Mentor Graphics may terminate this Agreement upon 30 days written notice if you are in material breach and fail to cure such breach within the 30 day notice period. If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term.
 - 12.2. Mentor Graphics may terminate this Agreement immediately upon notice in the event Customer is insolvent or subject to a petition for (a) the appointment of an administrator, receiver or similar appointee; or (b) winding up, dissolution or bankruptcy.
 - 12.3. Upon termination of this Agreement or any Software license under this Agreement, Customer shall ensure that all use of the affected Software ceases, and shall return it to Mentor Graphics or certify its deletion and destruction, including all copies, to Mentor Graphics' reasonable satisfaction.
 - 12.4. Termination of this Agreement or any Software license granted hereunder will not affect Customer's obligation to pay for products shipped or licenses granted prior to the termination, which amounts shall immediately be payable at the date of termination.
13. **EXPORT.** Software is subject to regulation by local laws and United States government agencies, which prohibit export or diversion of certain products, information about the products, and direct products of the products to certain countries and certain persons. Customer agrees that it will not export Software or a direct product of Software in any manner without first obtaining all necessary approval from appropriate local and United States government agencies.
 14. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. All Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to US FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in this Agreement, except for provisions which are contrary to applicable mandatory federal laws.
 15. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.
 16. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FLEXIm or FLEXnet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this section shall survive the termination of this Agreement.
 17. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of the Mentor Graphics intellectual property rights licensed under this Agreement are located in Ireland and the United States. To promote consistency around the world, disputes shall be resolved as follows: This Agreement shall be governed by and construed under the laws of the State of Oregon, USA, if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply. Notwithstanding the foregoing, all disputes in Asia (except for Japan) arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the Chairman of the Singapore International Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. This section shall not restrict Mentor Graphics' right to bring an action against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.
 18. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.
 19. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements, including but not limited to any purchase order terms and conditions. Some Software may contain code distributed under a third party license agreement that may provide additional rights to Customer. Please see the applicable Software documentation for details. This Agreement may only be modified in writing by authorized representatives of the parties. All notices required or authorized under this Agreement must be in writing and shall be sent to the person who signs this Agreement, at the address specified below. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.