# Lab 1 - Counter

A project is a collection mechanism for an HDL design under specification or test. Projects in ModelSim ease interaction 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.

Create a project

↓

Add files to the project

↓

Compile design files

↓

Run simulation

↓

Debug results
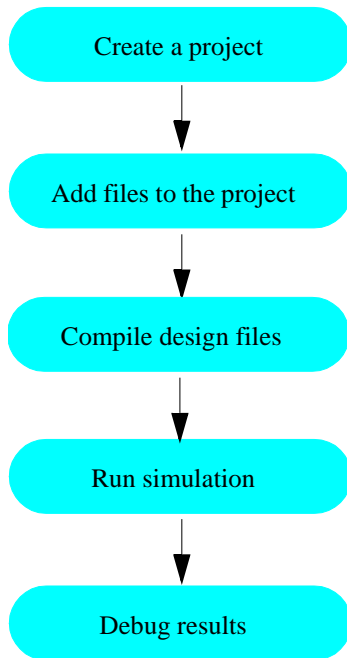
There are two points to remember about projects in ModelSim:

• You have to create a working library in the project flow.

• Projects are persistent. In other words, they will open every time you invoke ModelSim unless you specifically close them.

Introduction

In this lab you will practice creating a project in ModelSim and learn basic VHDL.

At a minimum, projects have a work library and a session state that is stored in a
.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 design for this lab is a simple counter with an
associated testbench. The files can be found on the class website:

www-cs.ccny.cuny.edu/~gertner

Click on **CSC 343** then on **Lab Assignments and Tutorial**.

Creating a new project

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

1 Create a new folder in your home directory. Rename it *lab_1*.

   Go to the class website and download *counter.vhd* and *tcounter.vhd*.

   Create a subdirectory for your project.
Rename the subfolder **counter**.

2 Start ModelSim.

  a Right click on the desktop at a Linux machine. Click on **Open Terminal**. Type **vsim** in the UNIX shell prompt.

    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 subfolder you created in step 1.

3 Create a new project.

  a Select **File > New > Project** from the menu bar in the Main window.

    This opens a dialog where you enter a Project Name, Project Locarion (i.e., directory), and Default Library Name (Figure 1). The default library is where compiled design units will reside.

  b Type **counter** in the Project Name field.

Figure 1: The Create Project dialog
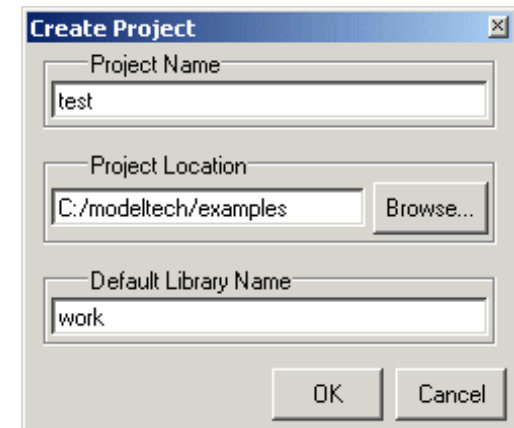
Adding objects to the project

Once you click OK to accept the new project settings, you will see a blank Project tab in the Workspace area of the Main window and the Add items to the Project dialog will appear (Figure 2). From this 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).

1   Add two existing files.

a   Click **Add Existing File**.

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

b   Click **Browse**.

c   Open the *lab_1/counter* directory in your ModelSim installation tree.

d   Select *counter.vhd*, hold the <Ctrl> key down, and then select *tcounter.vhd*.

e   Click **Open** and then **OK**.

f   Click **Close** to dismiss the Add items to the Project dialog.

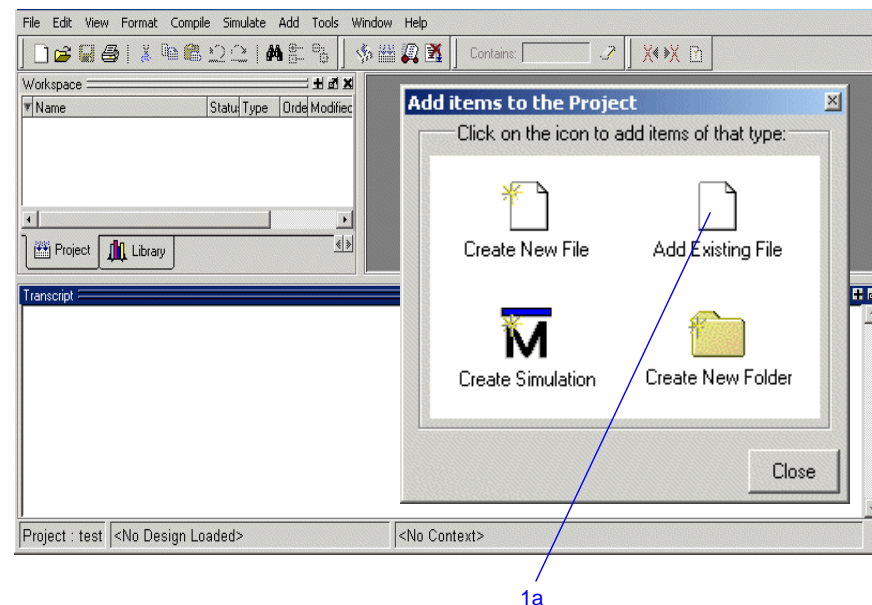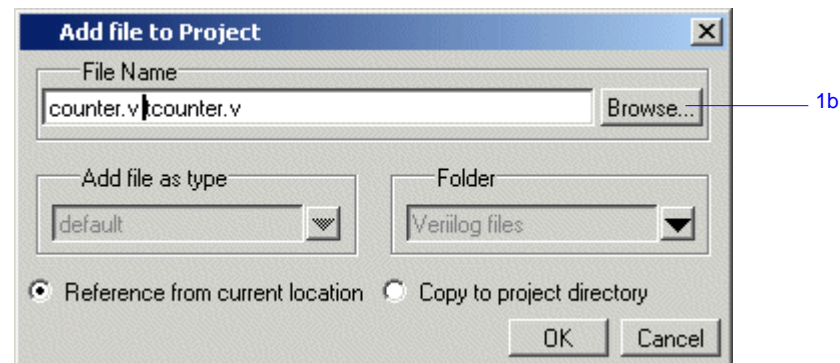**Figure 2: Adding new items to a project**

Figure 3: The Add file to Project dialog

You should now see two files listed in the Project tab of the Workspace pane (Figure 4).

Question mark icons (?) in the Status column mean the file hasn't been compiled or 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.

Changing compile order (VHDL)

By default ModelSim performs default binding of VHDL designs when you load the design with **vsim** (CR-224). However, you can elect to perform default binding at compile time. (For details, see "Default binding" (UM-68) 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 (Figure 5).

   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.

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

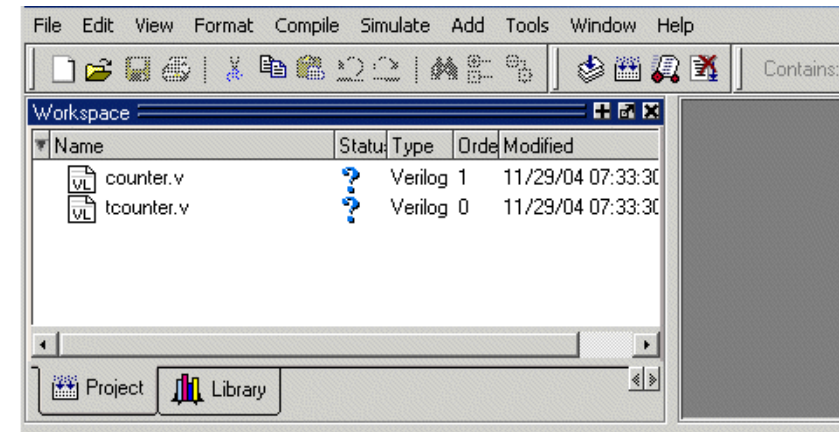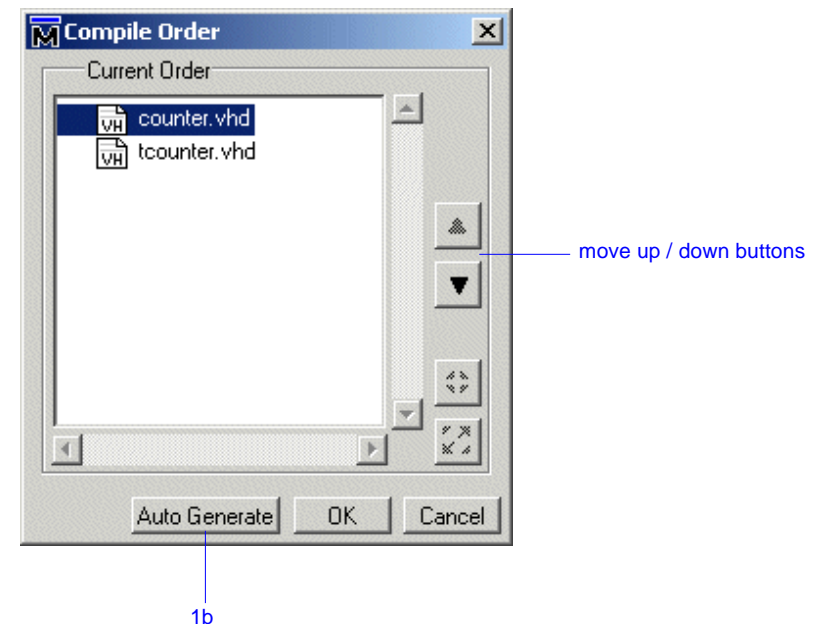**Figure 4: Newly added project files display a '?' for status**

Figure 5: The Compile Order dialog box

move up / down buttons

1b

ModelSim Tutorial

Compiling and loading a design

1   Compile the files.

   a   Right-click anywhere in the Project tab 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 pane.

2   View the design units.

   a   Click the **Library** tab in the workspace.

   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 6).
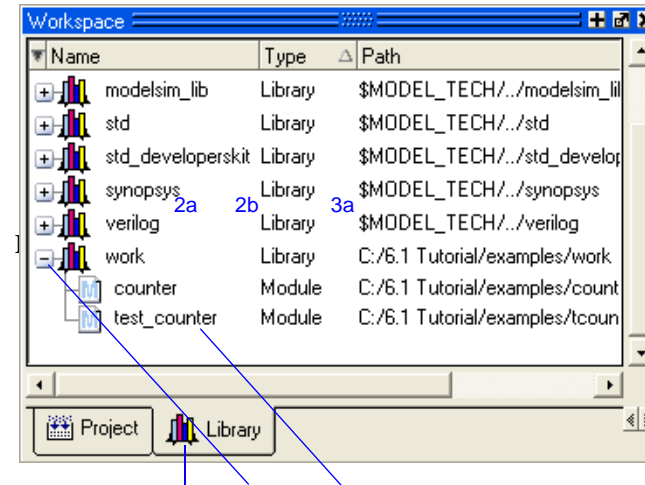
3   Load the *tcounter* design unit.

   a   Double-click the *tcounter* design unit.

       You should see 3 new tabs in the Main window Workspace. The *sim* tab
       displays the structure of the *tcounter* design unit (Figure 7). The
       Files tab contains information about the underlying source files. The
       Memories tab lists all memories in the design.
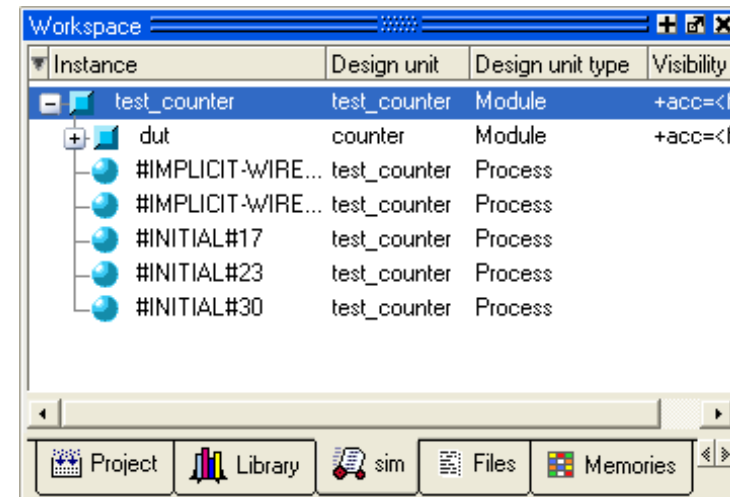
Running the simulation

Now you will run the simulation.

1   Set the graphic user interface to view the Wave debugging pane in the Main window.

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

     This opens one of several panes available for debugging. To see a list of the other panes, select **View > Debug Windows** from the menu bar. Most debugging windows will open as panes within the Main window. The Dataflow window will open as a separate window. You may need to move or resize the windows to your liking. Panes within the Main window can zoomed to occupy the entire Main window or undocked to stand alone.

2   Add signals to the Wave window.

   a   In the Workspace pane, select the **sim** tab.

   b   Right-click *tcounter* to open a popup context menu.

   c   Select **Add > Add to Wave** (Figure 8).

     Four signals are added to the Wave window.

3   Run the simulation.

   a   Click the Run icon in the Main or Wave window toolbar.

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

   b   Type **run 500 ns** at the VSIM> prompt in the Main window.

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

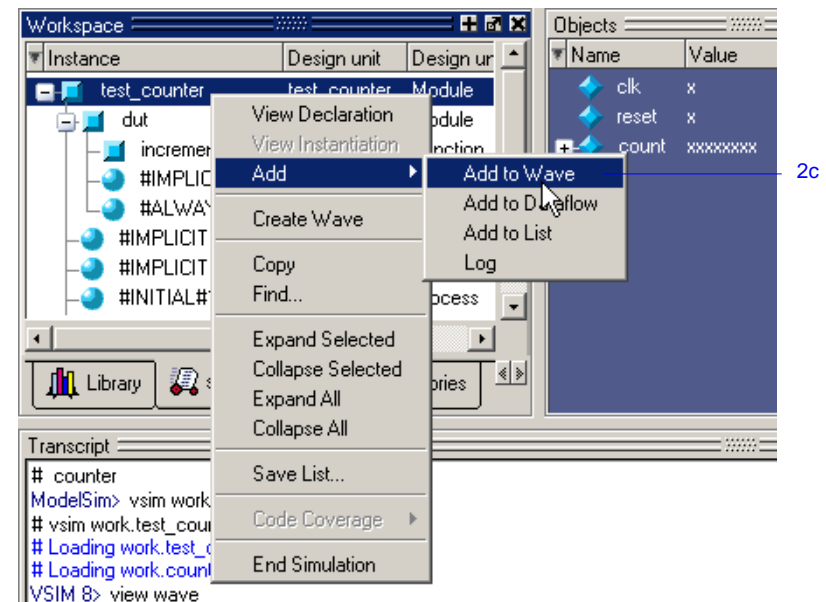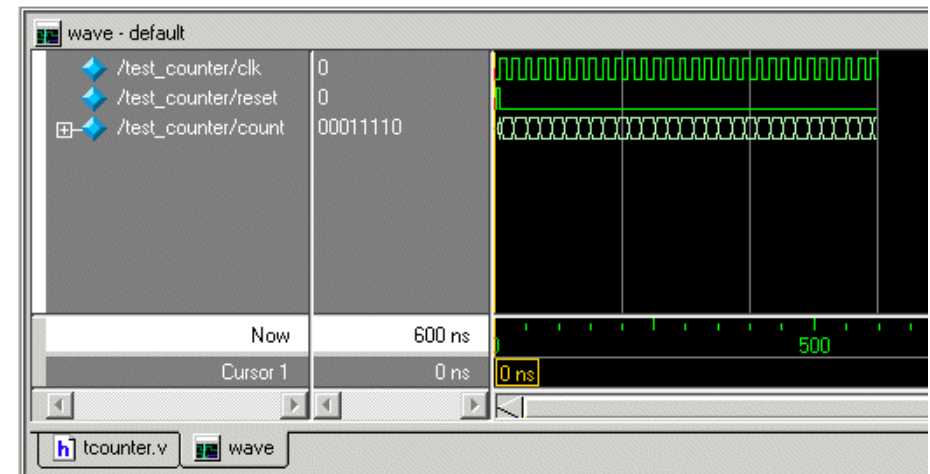**Figure 8: Adding signals to the Wave window**

Figure 9: Waves being drawn in the Wave window

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

The simulation continues running until you execute a break
command or it hits a statement in your code that halts the simulation.

d   Click the Break icon.

The simulation stops running.

Zooming the waveform display

Zooming lets you change the display range in the waveform pane. There are numerous methods for zooming the display.

1 Zoom the display using various techniques.

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

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

    You should see blue vertical lines and numbers defining an area to zoom in (Figure 10).

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

    The waveform pane returns to the previous display range.

d   Click the Zoom In 2x icon a few times.

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

    You should see a blue line and numbers defining an area to zoom out (Figure 11).

f   Select **View > Zoom > Zoom Full**.
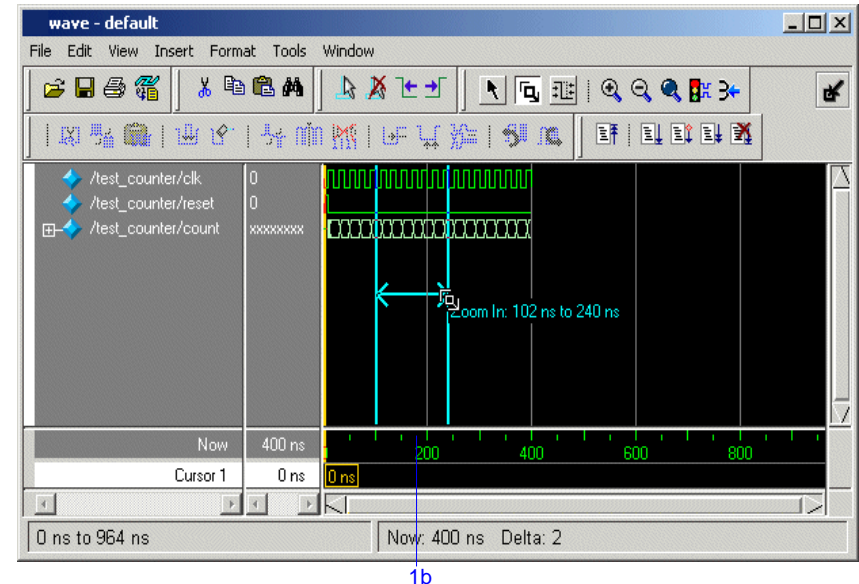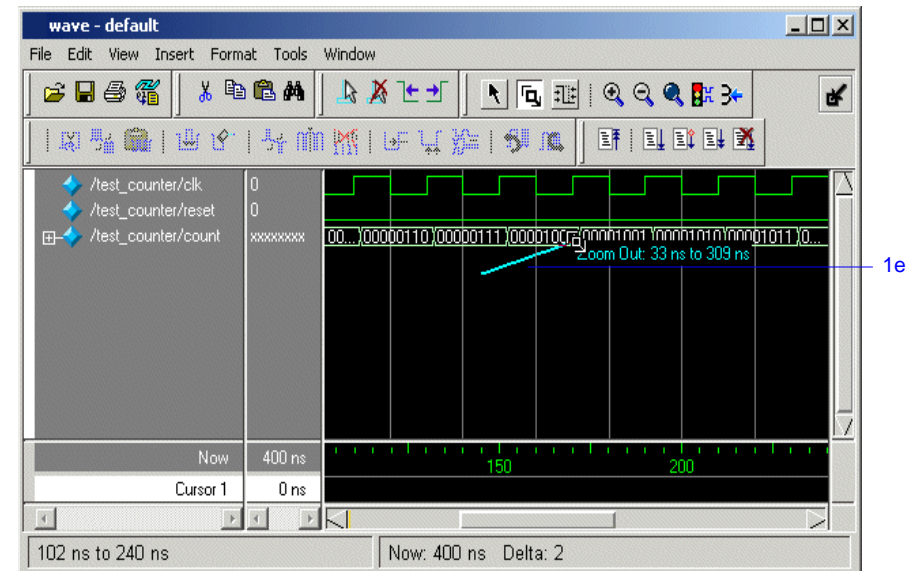
Figure 10: Zooming in with the mouse pointer

Figure 11: Zooming out with the mouse pointer

ModelSim Tutorial

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 pane 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.

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 12).

   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   Confirm that the waveform is correctly counting upward.
   .
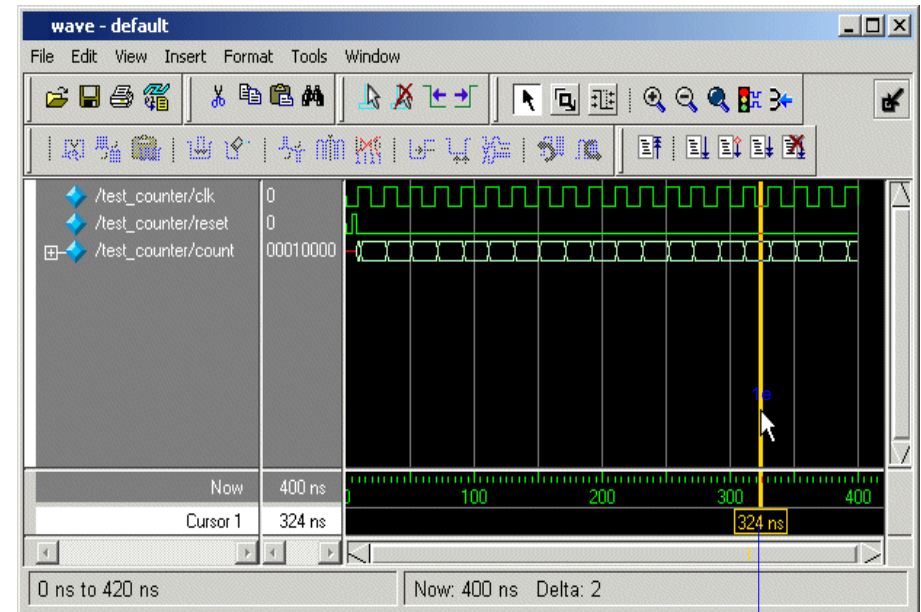
ModelSim Lab wrap-up

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

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

2 Select the Project tab in the Main window Workspace.

3 Right-click in this tab to open a context popup menu and select **Close Project**. Then click OK.

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