

## **Introduction:**

Modelsim is a software application that is used for simulating digital logic models. This document will describe the steps required to perform a behavioral simulation on a project or module.

For this tutorial, the author will be using a 2-to-4 Decoder to simulate. The module has three enable signals (2 active high, and 1 active low).

### **1. Create Test Bench Waveform (.tbw) file**

The test bench file is a VHDL simulation description. Modelsim reads and executes the code in the test bench file. The test bench file contains an instance of the module being simulated. The file being simulated is referred to as the UUT (Unit Under Iest).

- a. In Project Navigator, open the project that contains the module for simulation,
- b. Right-click on any source file in the Sources Pane,
- c. Select New Source from pop-up menu,
  - i. In the New Source wizard, select **Test Bench Waveform**,
  - ii. Type in the desired filename and select the current project directory, click **Next**,
  - iii. Select the Source module to associate the waveform file with, click **Next**,
  - iv. Select **Finish**.
- d. Timing generation wizard appears
  - i. For combinatorial circuits, chose **Combinatorial** under Clock Information section,
  - ii. For sequential circuits, setup the clocks according to the clocks used to implement the design, (for the S3SK board use 50 MHz),
  - iii. Length of simulation is defined in the 'Initial Length of Test Bench' field,
    1. To change the length of simulation after the timing generation wizard is complete, right-click on an open area of the waveform area in the test bench waveform,
    2. Select **Set End of Test Bench**,
    3. Change the value, click **OK**.

# Modelsim Tutorial

## ECGR2181

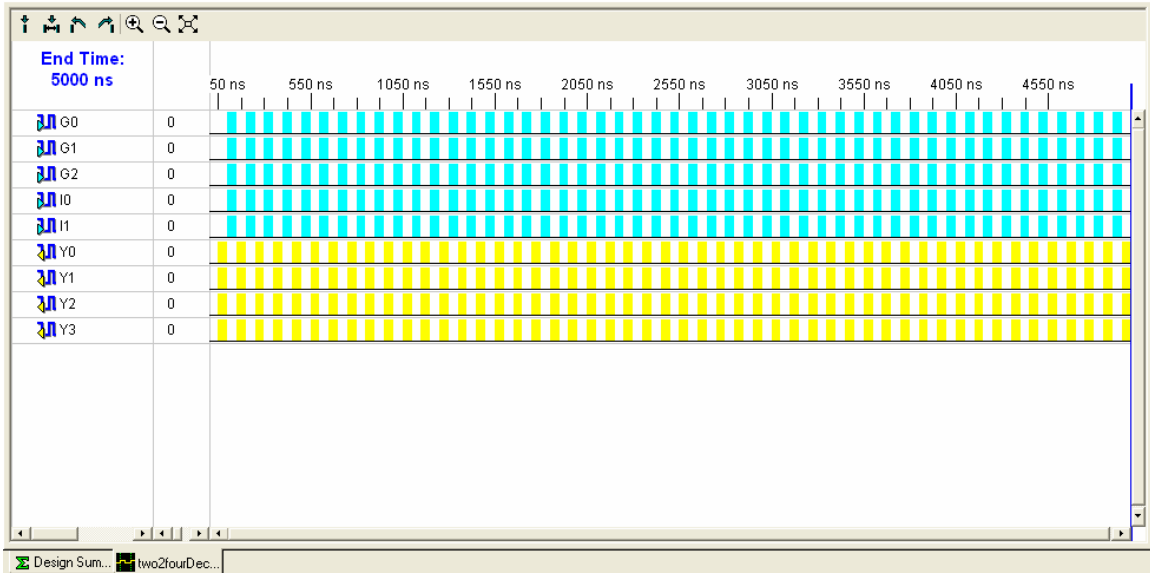


Figure 1, Test Bench Waveform file just after creating test bench file.

- e. Change input waveforms (inputs have blue areas; outputs have yellow areas) by clicking on the waveform the value for the remainder of the waveform will change, change the input waveforms until all combinations of inputs are checked, (Note: output signal changes, in the test bench editor, are overridden in Modelsim)

Another way is to change the waveform,

1. Right click on the waveform area,
2. Select **Set Value** from the pop-up menu,
3. Either set value (0 or 1) or,
4. Click Pattern Wizard ... I'll leave the rest to you,
5. Click **OK**

- f. Click **Save**.

- g. Figure 2 illustrates the final test bench waveform file ready for simulation.

# Modelsim Tutorial

## ECGR2181

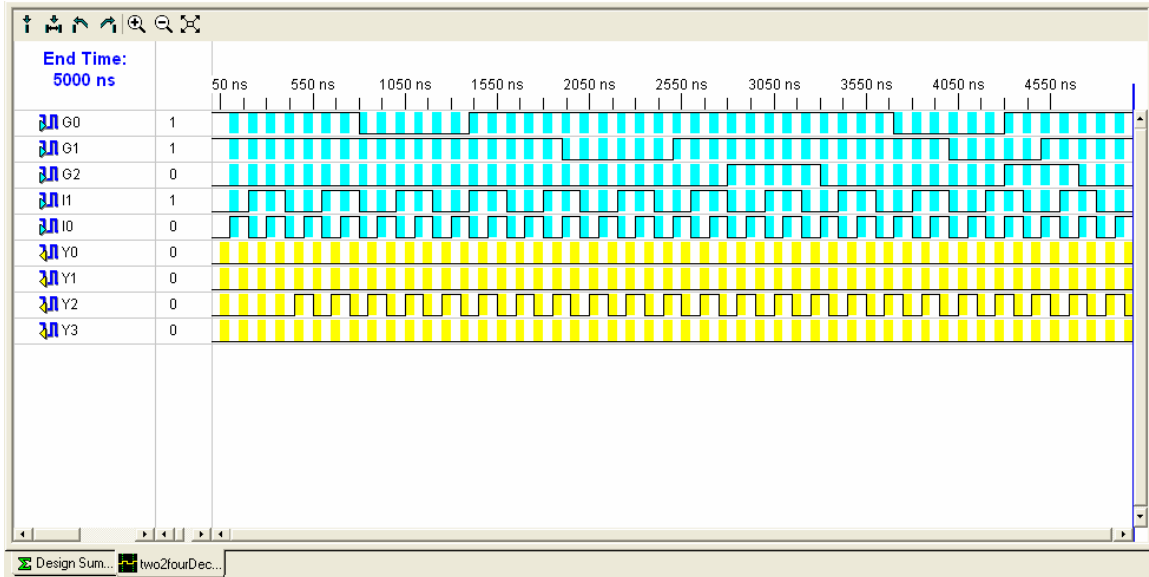




Figure 2, Test Bench Waveform file for demo.

## 2. Initiate Modelsim:

- In the Sources Pane, the Test Bench Waveform file will show up under the module being simulated,
- In the Process View Pane, double-click on **Simulate Behavioral Model** process,
- Modelsim application should now be starting,
- After the application is finished performing it's initial tasks, click on the **Wave** tab, in the Waveform pane, to view the waveforms,
- Zoom out by repeatedly clicking on the minus-magnifying glass button, on the toolbar , until waveform features are distinguishable,
- The user can reorder the waveforms to make viewing the signals easier,
  - Drag and drop the waveforms into the appropriate order,
  - The user can add dividers to make grouping the signals easier.
    - Right-click on the waveform names,
    - Select **Insert Divider** from the pop-up menu,
    - Type the divider name and click **OK**.
- To add internal signals from the UUT, into the waveform list
  - In the Workspace pane, select UUT
  - In the Object pane, drag the signals to observe into the Wave pane, and
  - Rearrange signals to suit.
- Click on the **Restart** button on the toolbar ,

# Modelsim Tutorial

## ECGR2181





- i. Click on **Run All** button on the toolbar . This action causes the simulation to run again.

Figure 3 shows the Modelsim application after initial setup. Values of all waveforms at a particular time can be read in the panel next to the waveform list. The time of the values is given by the Yellow Line marker. The current marker can be moved by clicking at the desired time in the waveform area. A **new marker** can be **added** by clicking on the  button on the toolbar. The **current marker** can be **removed** by clicking on the  button. And finally, the current marker can be moved to the next rising/falling edge by using the  buttons.

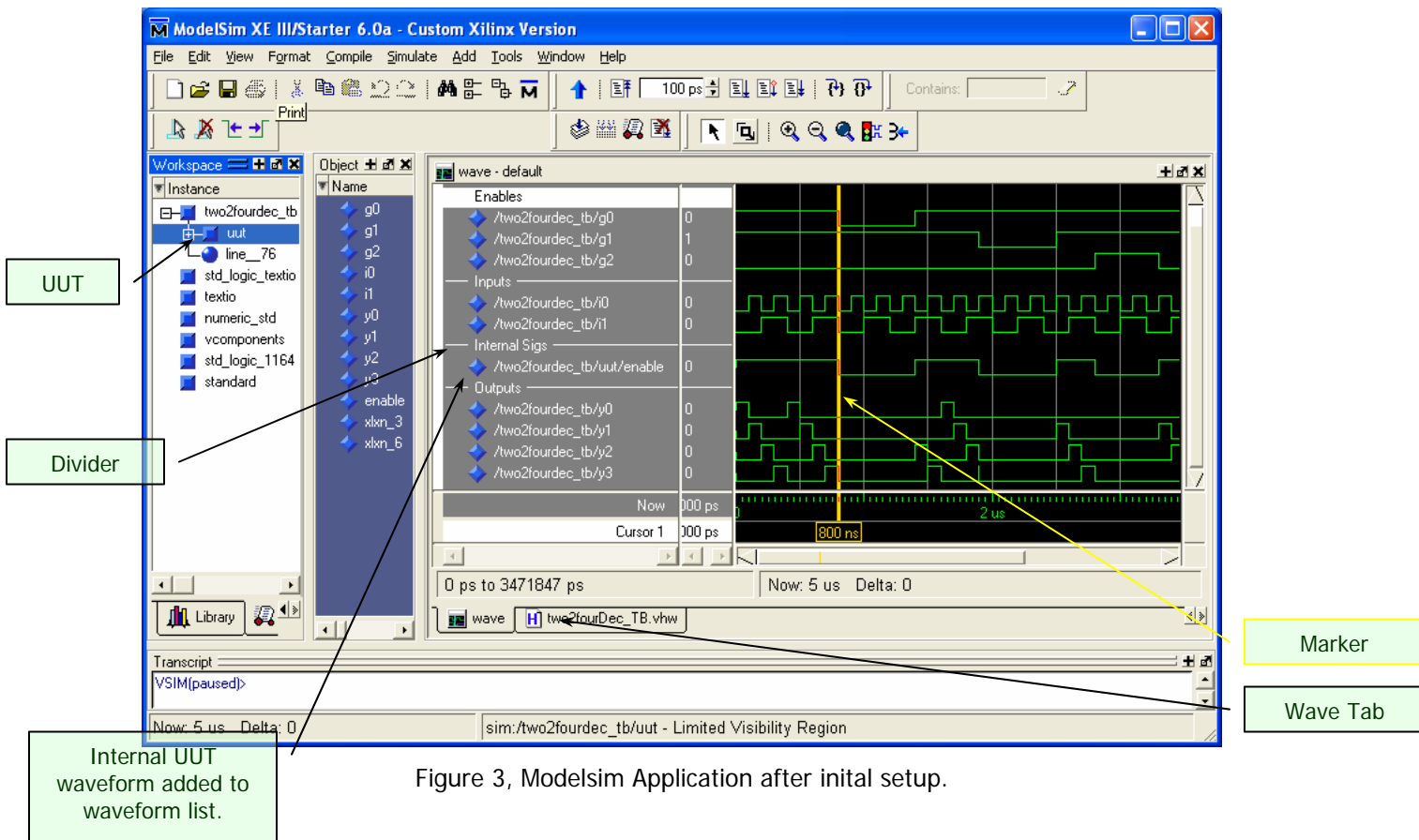





Figure 3, Modelsim Application after initial setup.

### 3. Modelsim Application features

- a. Creating macros,  
Macros can be used to save the setting you have created, so that you will not have to repeat the above steps for multiple design iterations; i.e, many runs of the same simulation file.

## Modelsim Tutorial ECGR2181

- i. Ensure that the Waveform Pane is selected,
- ii. Select **File** from the top menu, then click **Save**,  
(or by clicking on the Save button )
- iii. Type in desired name, use \*.do as the file extension,
- iv. Click the **Save** button,
- v. Now open the macro file that was just saved, by clicking  
on the Open button ,
- vi. Figure 4 shows the macro file in place of the waveform area,
- vii. Add a line after line 2 that reads "delete wave \*"  
This will delete the current waveforms and add the ones from the macro  
to create your saved setup. If this step is skipped then the waveforms  
will be repeated each time you run the macro.
- viii. Click the **Save** button , click **Save** again.
- ix. Close the wave.do file, by clicking on the **X** in the upper right of pane.

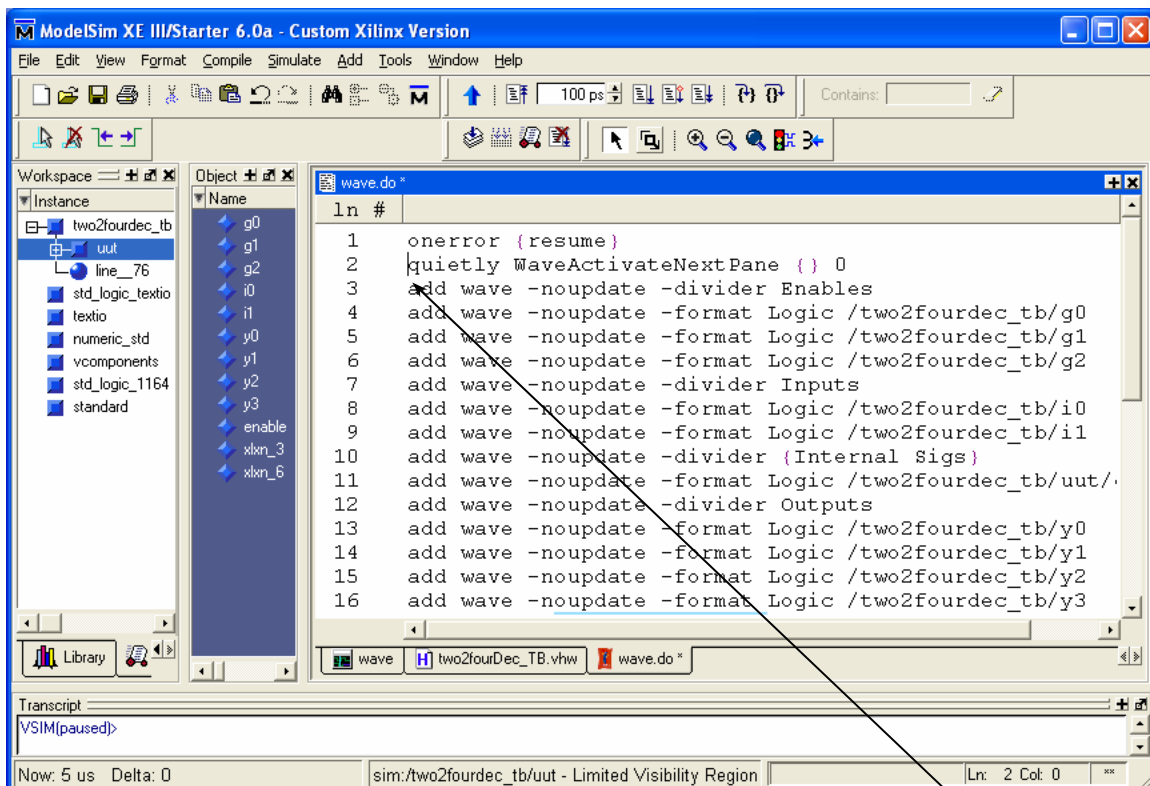


Figure 4, Macro file illustrated.

Add  
"delete wave \*"  
before line 3.

### b. Running Macros:

The macros can be run at anytime. However, each time that you start Modelsim the waveforms will revert back to your initial settings. Make sure that Modelsim is finished performing the initial tasks before executing this step.

- i. Click **Tools** menu, then select **Execute Macro...**,

## Modelsim Tutorial

### ECGR2181

- ii. Re-run the simulation (steps 2.h and 2.i), this step is required as the waveforms will not reflect any design changes that occurred since the user first saved the macro file.

This concludes the tutorial on the basic operation of Modelsim. There are many other features that can be used to enhance your experience with Modelsim. If there are any omissions that could help others please do not hesitate to contact the course instructor with the details.