 I. Setting up the user environment to run the Questasim simulation tools.

Throughout this tutorial the Linux prompt is indicated by:
[hvgray][home/x/xy_z]

Your prompt may appear different depending upon the configuration of your account.

Step 1:

We will first create a directory called `Questa.` The directory will be used to contain the Systemverilog code to be simulated, and a directory called `work` (which will be used to hold intermediate files created by the simulation tools). The `work`directory will be created using a special Questasim command (the **vlib**command).

Issue the following commands from the UNIX prompt:
[hvgray][home/x/xy_z]>*cd*
[hvgray][home/x/xy_z] >*mkdir Questa*

Next time to go to that directory use the following command:
[hvgray][home/x/xy_z]>*cd Questa*

Prior to running the **Questasim** tools, it is necessary to set up your Linux computer account. Perform the following from your Linux prompt:

Step 2:

[hvgray][home/x/xy_z] >*source /CMC/ENVIRONMENT/questasim.env*

Alternatively, one may copy the file /CMC/ENVIRONMENT/questasim.env to one's home directory and source it from there (make sure you have the most recent version of the file):
[hvgray][home/x/xy_z] > cd
[hvgray][home/x/xy_z] >cp /CMC/ENVIRONMENT/questasim.env
[hvgray][home/x/xy_z] > source questasim.env

It is necessary to source the questasim.env file every time you login in, or whenever you open a new terminal window.

II. Performing Systemverilog simulation using Questasim

This section will illustrate the use of the Questasim tools used to perform systemverilog simulation. The example will illustrate various features of the tools.
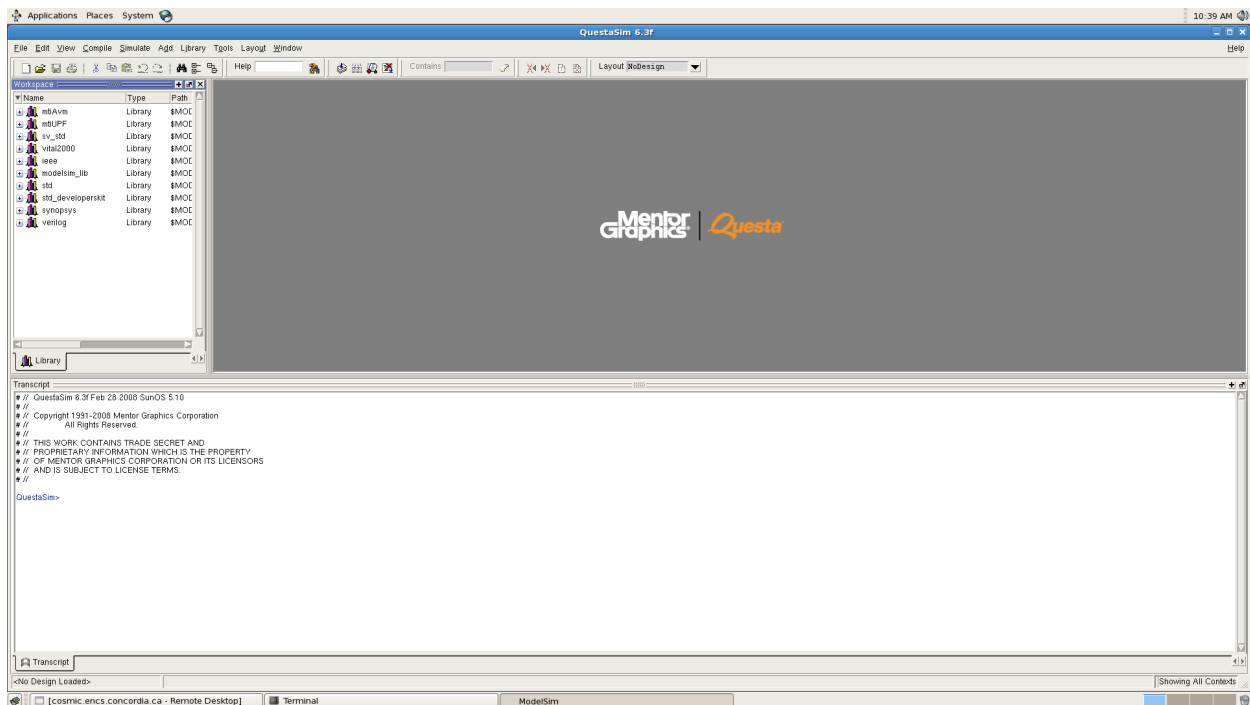
Step 3:
Copy the Systemverilog files to the Questa directory. (Systemverilog files have the extension sv). For example copy the three files at the end of this document to the Questa directory.

Step 4:

To invoke the graphical user interface use the command:
[hvgray][home/x/xy_z/Questa] >*vsim*

The graphical user interface of Questasim consists of a main Questasimwindow that hosts several of subwindows, called panes. A pane can be manipulated by clicking on the controls at its right.

Questasim's mechanism to keep all source files of a design together is called a *project*.

Create a project with an appropriate name using the following step:
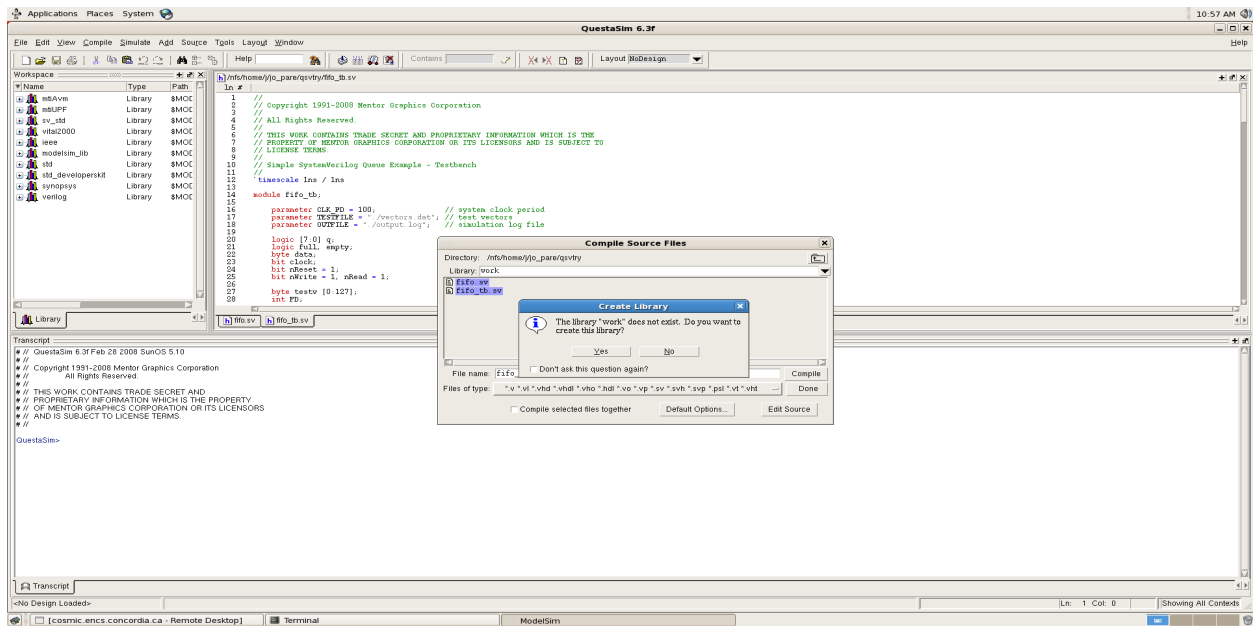
File -> New -> Project
In future sessions, if your project is not yet open at start up, open it using
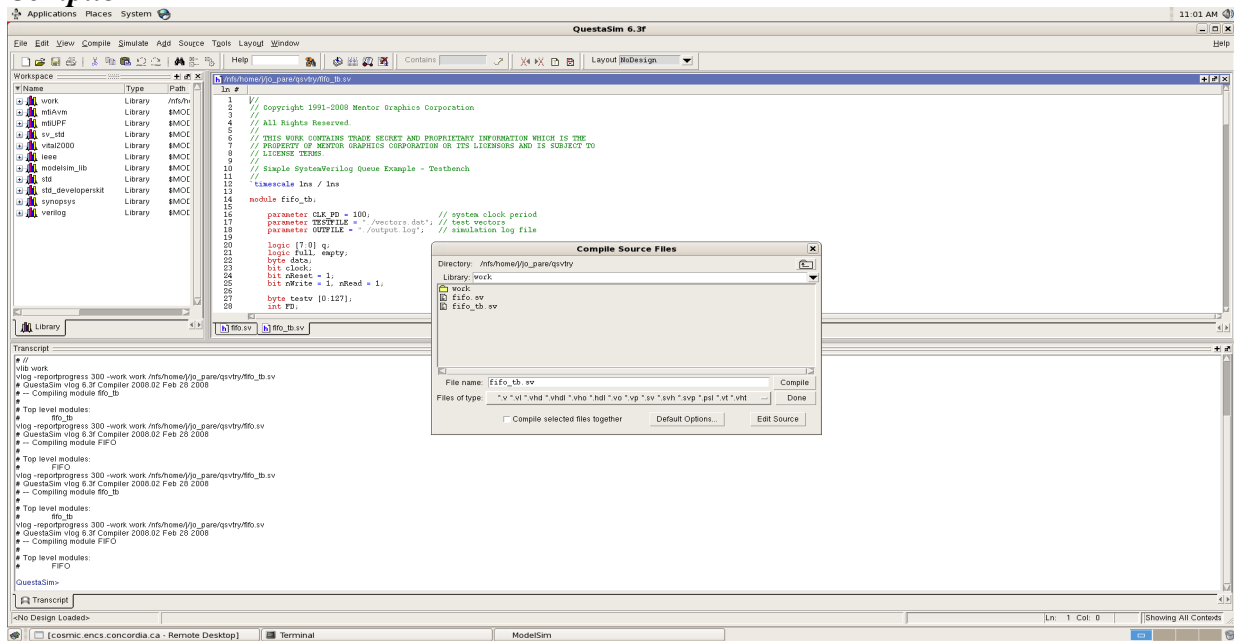### File -> Open . . .
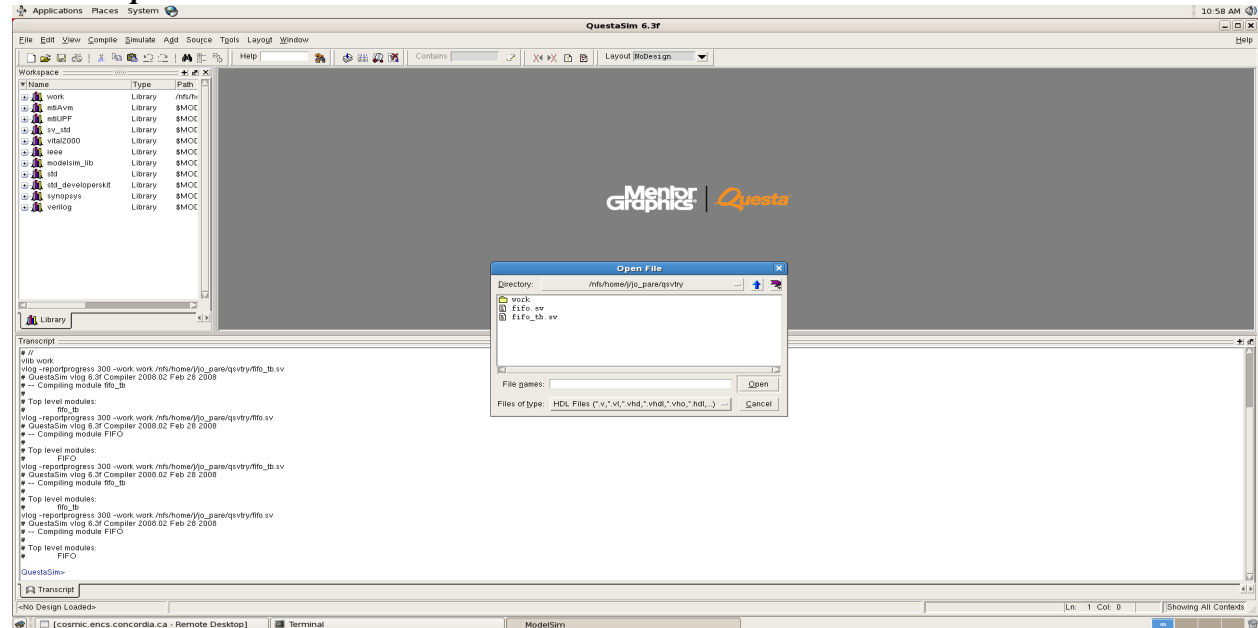
Select the required files (Press control and click on the required files. In this case fifo.sv and fifo_tb.sv) and click *open*

If it is the first time and the work directory is not yet made it will ask to permission to make one. Click *yes*.
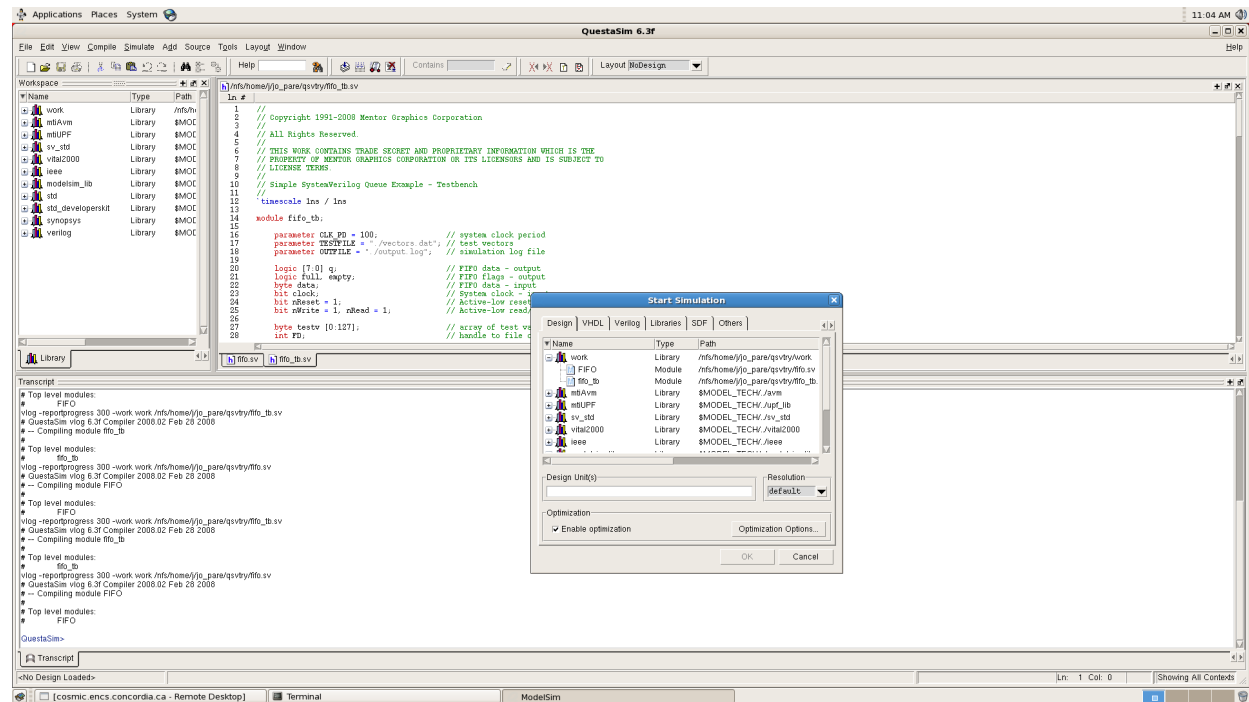
To compile all files, use:

Compile->

Select the required files (case fifo.sv and fifo_tb.sv) and click *compile*. After compilation without any error click *done*.

To execute:
Simulate-> Start Simulation.

In the pane click on the **work** (+ on the left of work) and select the required modules and click **ok**.

Questasim makes use of many panes/windows. The windows that are necessary for theexercisesare *Objects* (for seeing the signals and variables belonging to the selectedcomponent) and *Wave* (forvisualizing signals changing over time). Open these using:

View -> Wave

From the workspace pane select the required module. The signals of the selected module can be seen in the *Objects* pane. Select the required signals and move it to the wave pane.

Now run the simulation.

Simulate->Run->Run-all

The main goal of a simulation is to verify a design. The simulation results in signal transitionsthroughtime. For the purposes of the exercises, it is often sufficient to perform the verification by displayingthe transitions along a time scale and zooming in at specific parts of the *wave forms* displayed. Waveforms are displayed in the *Wave* window. To get signals displayed; you should first choose the design unit in which the signal occurs in the *workspace pane* of the main window, which behaves like a browser in which you can navigate through the hierarchyof your design. A design unit is simply selected by clicking on it. Parts of the hierarchy can be

hiddenor displayed by clicking on the little squares containing a `-' or a `+' respectively. All signals of theselected design unit are listed in the *Objects* window.

Selected signals are transferred to the *Wave* window with the command:

Signals: ***Add -> Wave -> Selected signals***

If you are sure that you want to trace all signals of the currently selected design unit, use:

Signals: ***Add -> Wave -> Signals in region***

An alternative way of transferring selected signals to the *Wave* window is to use *drag & drop*. Select first the signals as described above and put the mouse cursor in the *Objects* window. Then click onthe left mouse button and drag the selected objects to the *Wave* window without releasing the button.Release the button when the cursor has arrived at the desired location. This *drag & drop* mechanismalso works within *Wave* itself e.g. to reorder the signals.

Once all signals to be traced have been notified to the *Wave* window, the actual simulation can start. Youmay, however, want to modify the display properties of the signals. If you want e.g. to display a binaryvector as a decimal number, you should first select the signal in the *Wave* window and then execute:

Wave -> Format -> Radix -> Decimal

In most cases, you will not need to modify the default radix of a signal.Sometimes, it is interesting to display a sequence of digital values as a continuous wave form. Theeasiest way to achieve this is by issuing the command:

Wave -> Format -> Format -> Analog (automatic)

A useful feature is the availability of multiple time cursors. To create a new cursor, execute:

Add -> Wave -> Cursor

in the docked *Wave* pane or

Add -> Cursor

in the undocked *Wave* pane.