This brief document provides guidance for in-lab activity related to the VHDL portion through screen captures and related descriptions below.

The in-lab activity involves the four files in the system.zip package, and two additional files that are provided in the Lab 4 portion of the course Webpage

All of these files should be placed in a SYSTEM folder, as shown below.
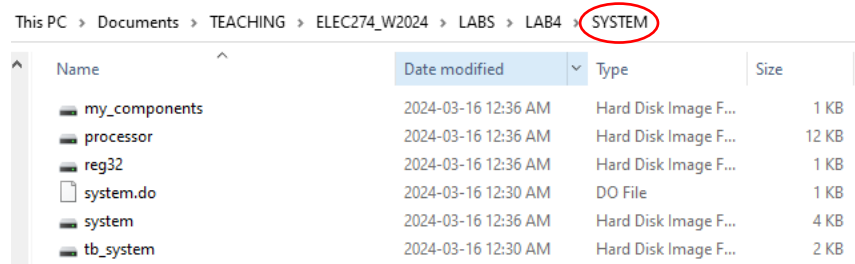
The ModelSim tool should be executed by using the Search feature of the Microsoft Windows Start menu. Because it appears that the license has not been renewed, we will be using the unlicensed version ('starter edition') of the ModelSim tool. Choose the correct version.
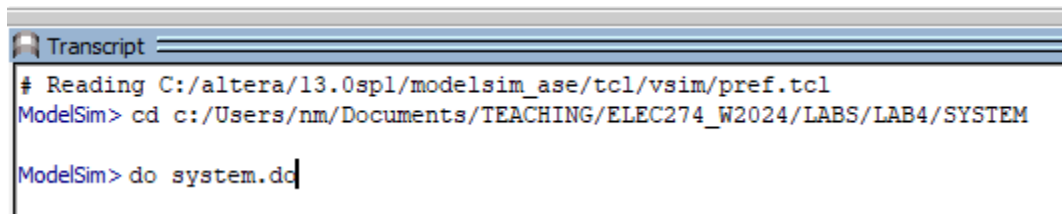

When the tool starts up, a "welcome" dialog box appears. Click on Close.
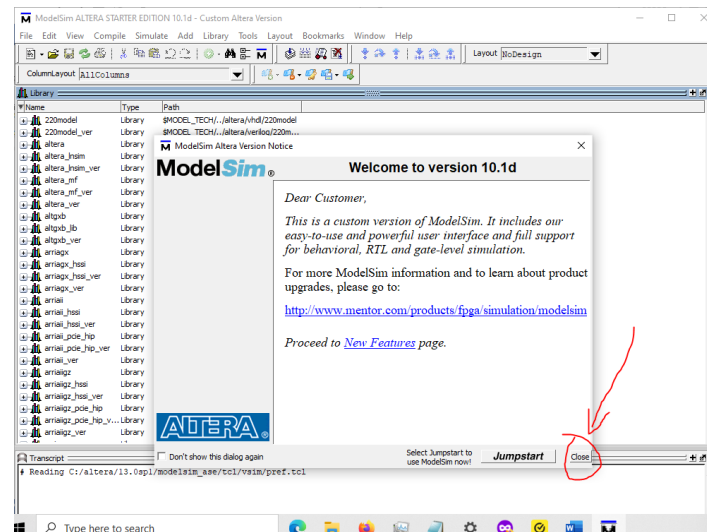
In the Transcript window at the bottom, type a 'cd' command to change the working directory to the SYSTEM folder. The screen capture below illustrates the intent, but the path to your working directory should reflect where you have created your SYSTEM folder. For example, it could be
c:/Users/*your_netID*/Desktop/SYSTEM   or   c:/Users/*your_netID*/Documents/SYSTEM

ModelSim supports foldername/filename completion. As you type letters of each part of the path to your SYSTEM folder, ModelSim shows you a list of matching names above your typing area. If you press the Tab key, the first item from that list is automatically completed, and you can then type "/" followed by the first few characters of the next part of the path to your working folder. Press the Enter key when the full path has been typed to change the working directory to that folder.

If the 'cd' command is successful, then execute the simulation-setup script in the system.do file with the 'do' command shown in the screen capture below.


After pressing the Enter key for the second command above, the ModelSim window will flicker/flash/update as the simulation is initialized and the waveform subwindow is created inside the main window.
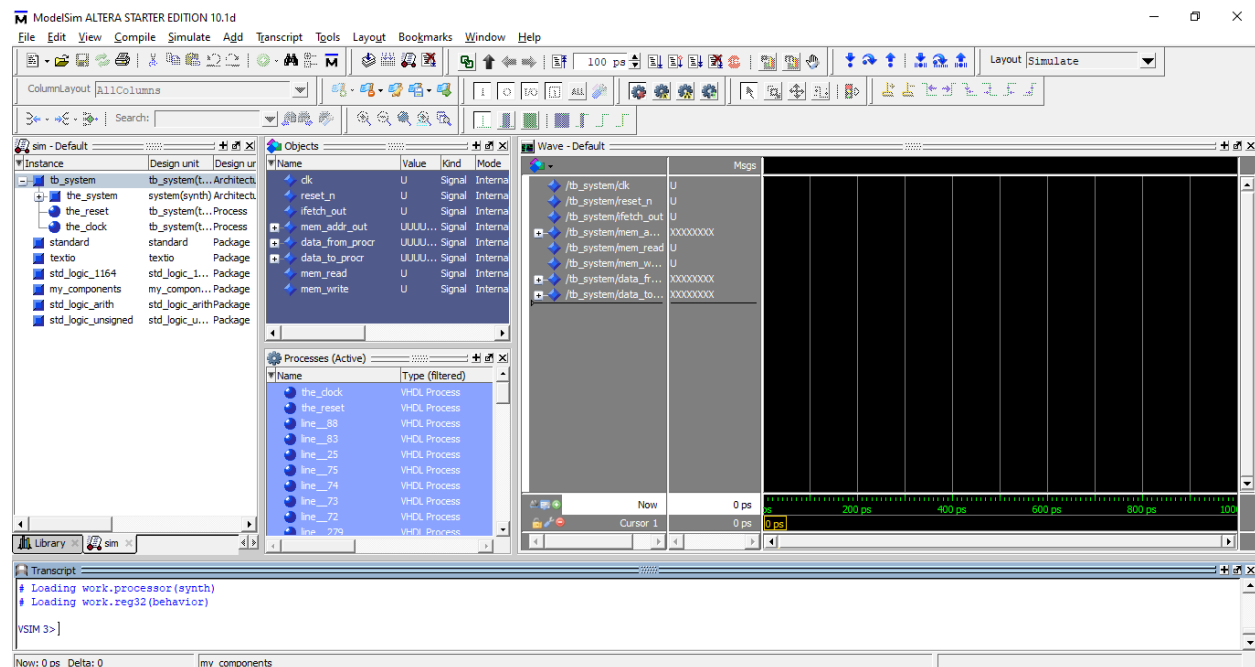
WAIT until the tool is ready with a new "VSIM >" prompt appearing in the Transcript window.

When the waveform window is ready, type a 'run' command in the Transcript window with a desired time interval for simulation. The testbench file tb_system.vhd has a process that models a 50-MHz clock. That means the clock cycle time is 20 ns. Therefore, each five-step instruction requires 100 ns of simulation time. For example, 'run 500 ns' will simulate the execution of five instructions.

After typing a 'run' command, the waveform window will show waveforms, but with a zoomed-in view. Left-click the mouse anywhere in the black waveform area, then use the minus '–' key on the keyboard to zoom out until you can see a number of clock cycles, and finally use the scroll bar (or type the left/right arrow keys on the keyboard) to adjust the view to see the different parts of the simulation timeline. At points of interest, you can adjust the zoom level with the '+' or '–' keys, and you can scroll left/right to observe the simulated system behavior.
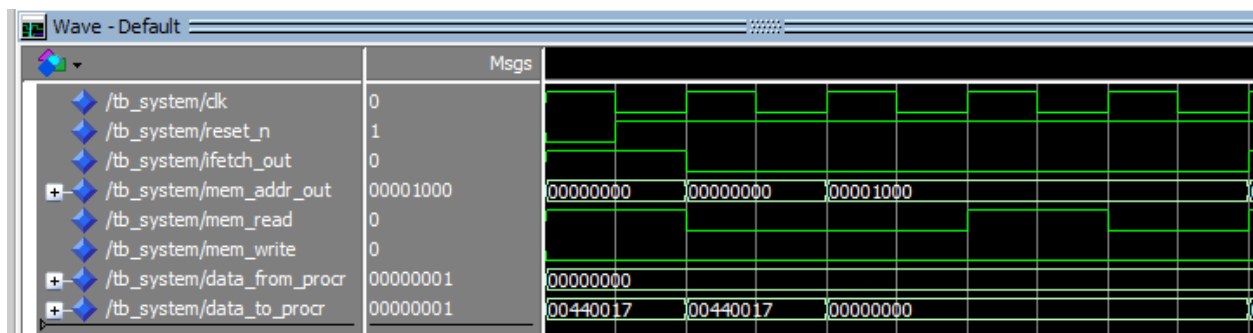
The signal names at the left side may initially not be entirely visible. Click and hold on the white vertical divider to move it to the right and expose more of the signal names.

Once waveforms are visible, the learning activity involving the study of the simulated behavior can commence.

You can continue simulation past the current end point with another 'run' command specifying a duration of $k*100$ ns for k more dynamically executed instructions. Once again, click in the waveform window, adjust the zoom level with the '+' or '–' keys, and scroll left/right as desired to observe the additional simulation results.

To start over from the beginning, type the "restart" command at the prompt in the Transcript window to reinitialize and create an empty waveform display. A window will pop up to prompt you on what elements to keep. Just click on OK to keep everything.

A restart does not run a simulation automatically. It simply resets to time zero. After a restart command, you must type a "run" command to simulate for a specified duration of time.