

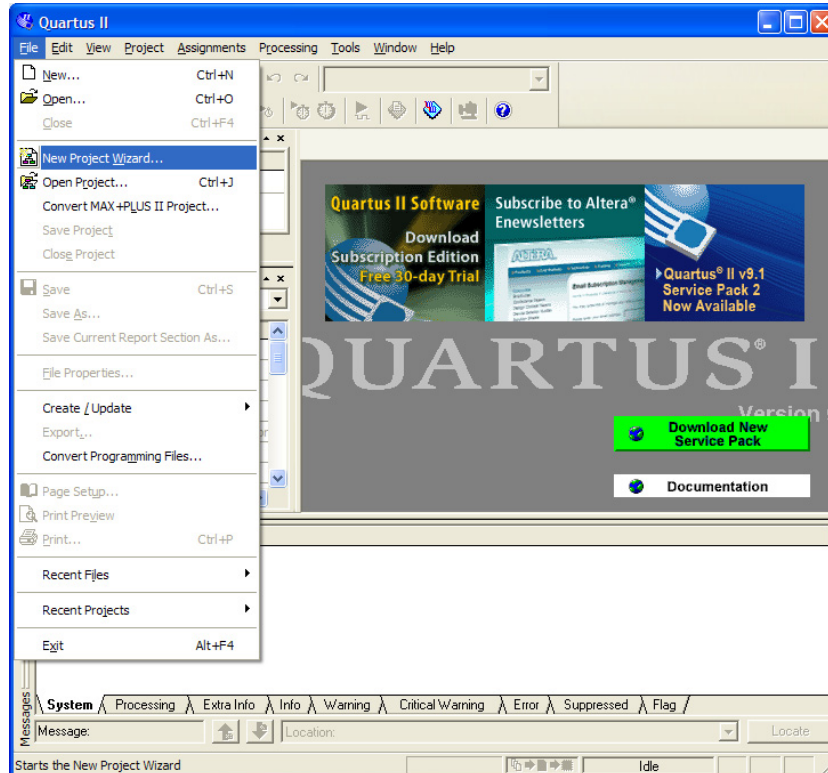
2010 R&E: Computer System Education & Research

Lab 4. Hardware Synthesis & Simulation

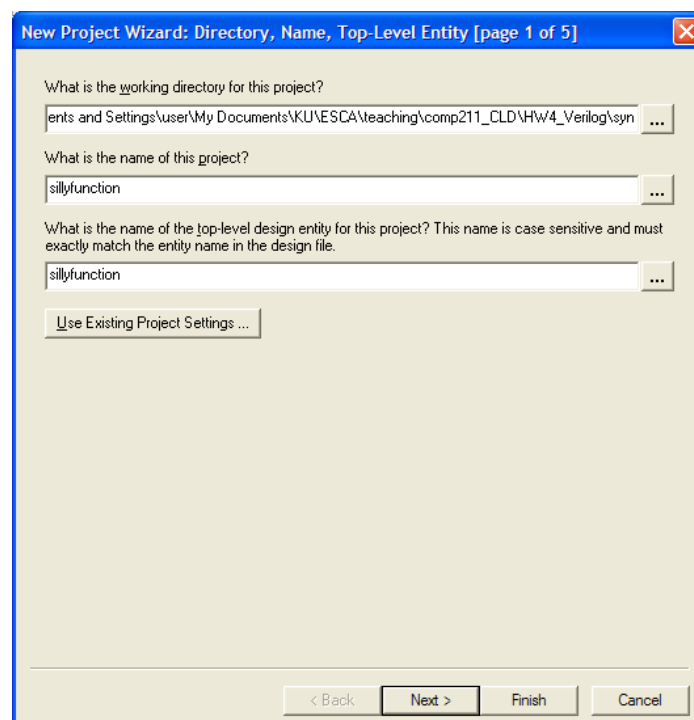
- Download an example Verilog code (sillyfuction.v) with testbench (testbench1.v) from
http://comedu.korea.ac.kr/~suhtw/teaching/comp211_CLD/HW4_Verilog.zip
- Synthesis
 - Install **Altera Quartus-II Web Edition** from
<http://www.altera.com/products/software/quartus-ii/web-edition/qts-we-index.html>
 - Follow the synthesis steps from page 2
 - Screen-capture the synthesized logic
- Simulation
 - Install **Altera ModelSim Starter Edition** on your PC from
<http://www.altera.com/products/software/quartus-ii/modelsim/qts-modelsim-index.html>
 - Follow the simulation steps from page 6
 - Screen-capture the waveform, showing the meaningful section.

- **Synthesis Steps**

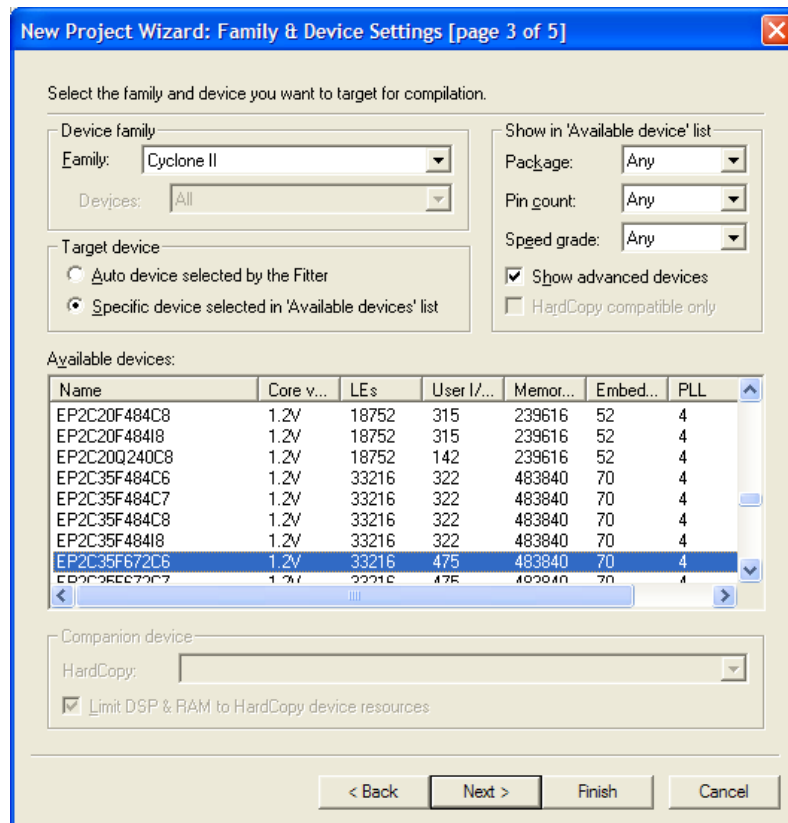
1. Invoke the Quartus-II
2. Create a new project



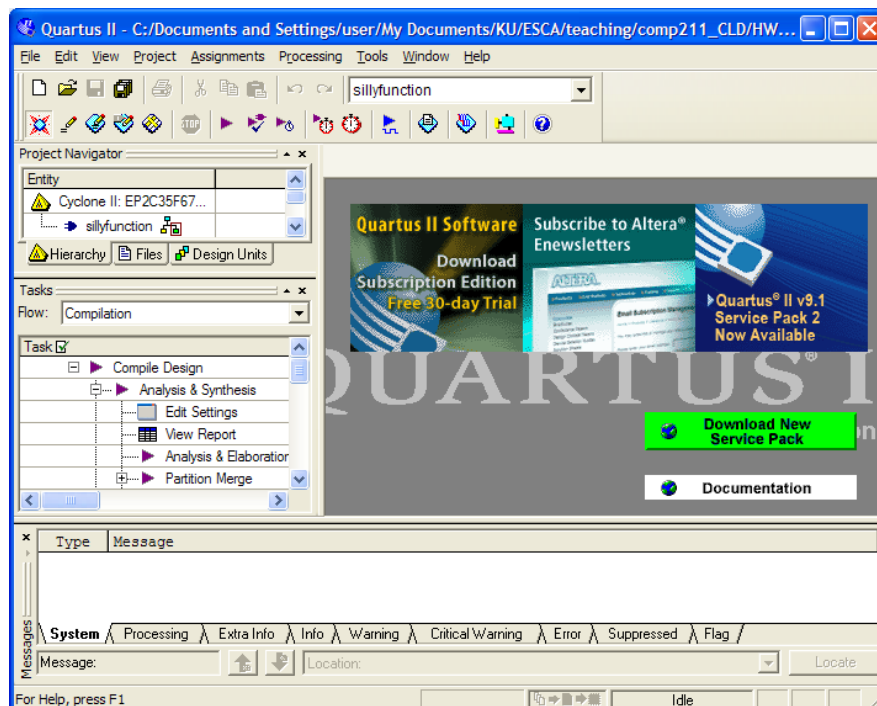
3. Go to any directory where you want to create your project and type a project name of your choice



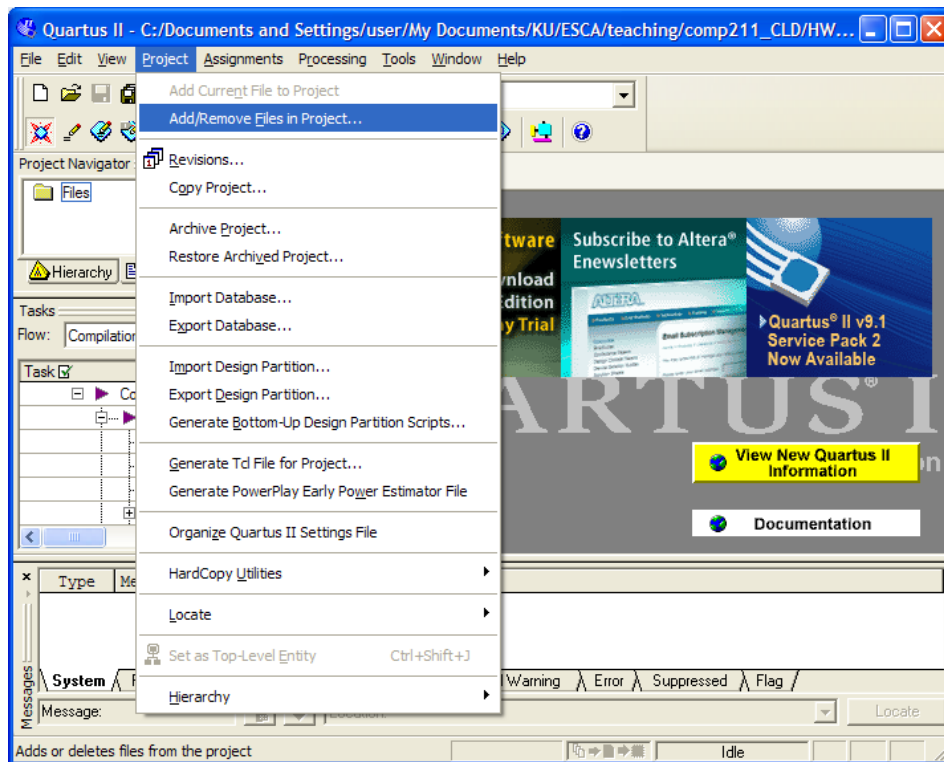
- Choose '**Cyclone-II**' in Device Family and '**EP2C35F672C6**' among available devices (The device is on DE 2 Board shown in the class)



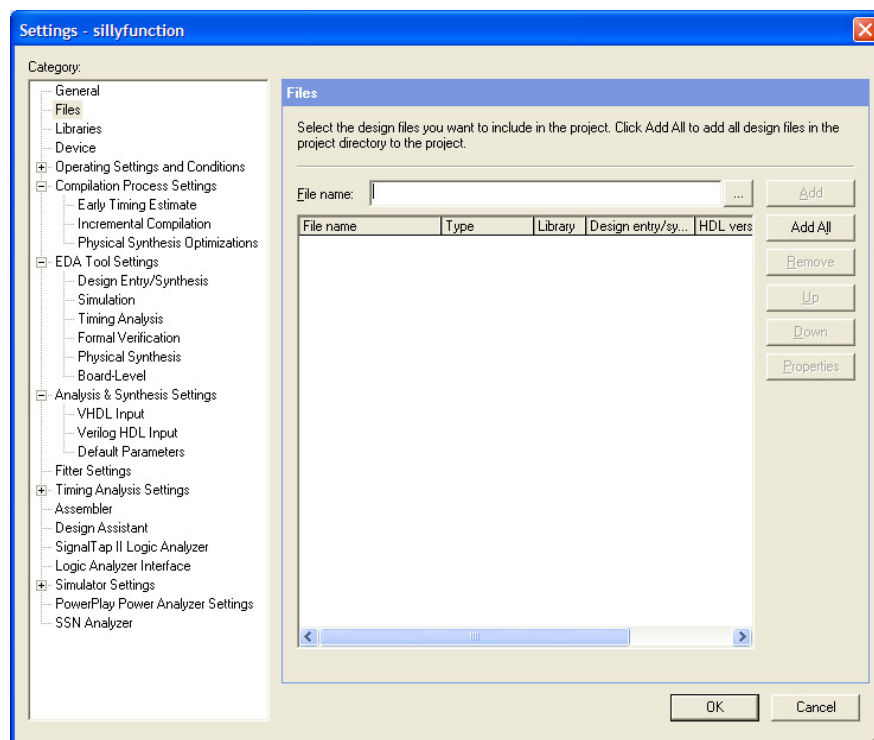
- Then 'Next → Next → Finish'

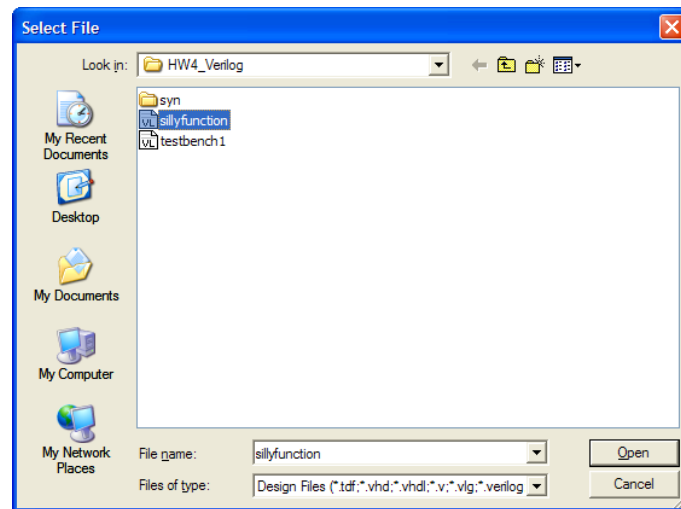


6. Then 'Project → Add/Remove Files' in Project

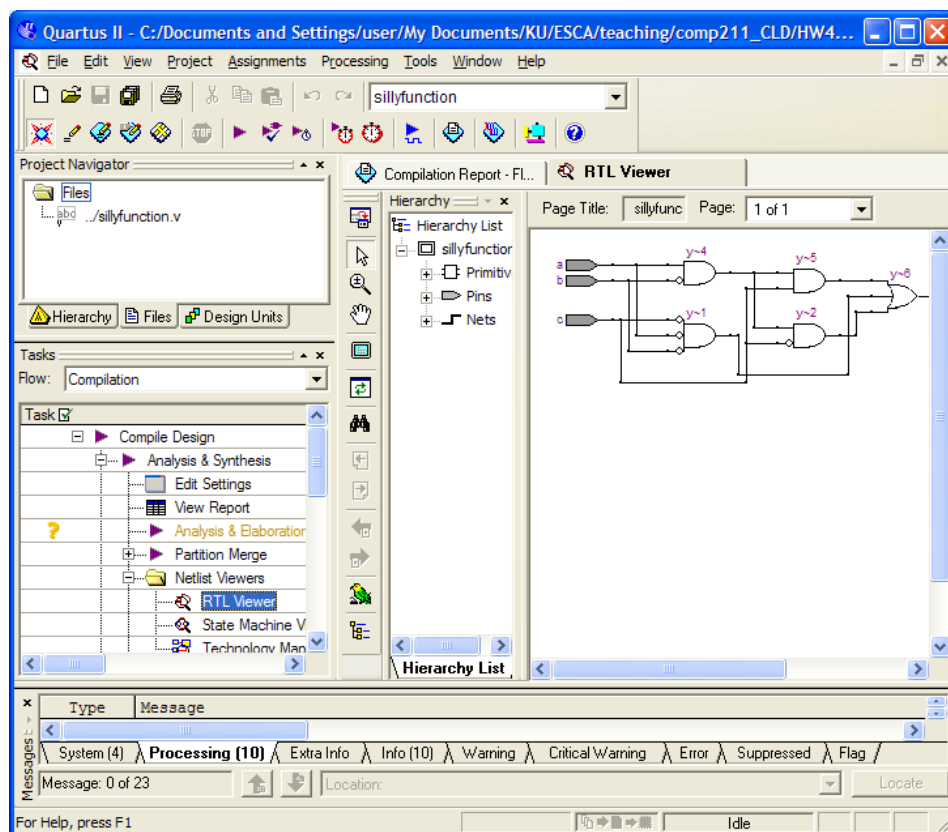


7. Click on '...' besides File name, add 'sillyfunction.v' from the download and then click 'Add' button.



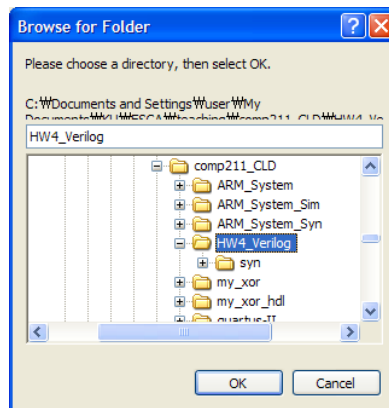
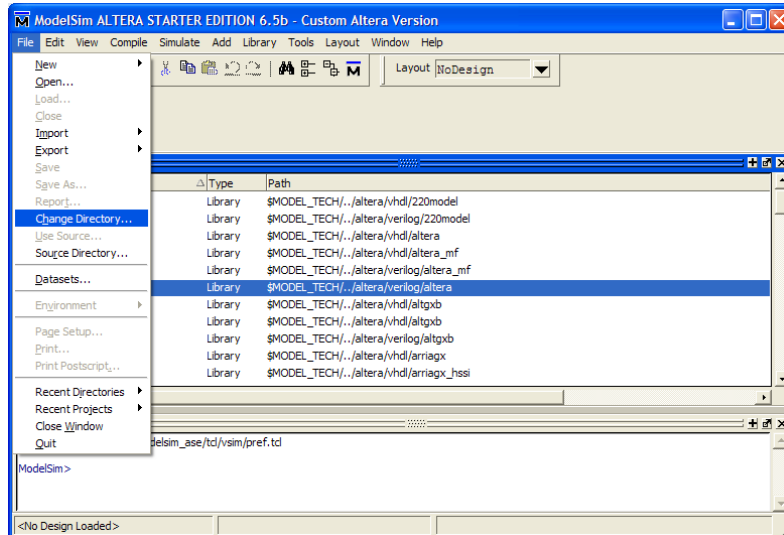


8. Double-click on 'RTL Viewer' to generate schematic

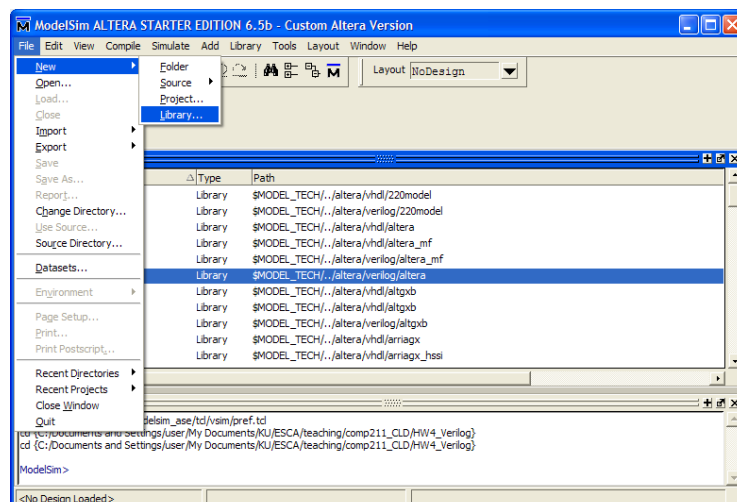


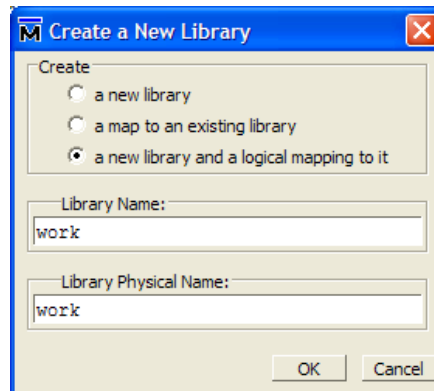
• Simulation Steps

1. Invoke the ModelSim Altera Edition, 'File → Change Directory' and choose a directory where simulation files are going to be saved

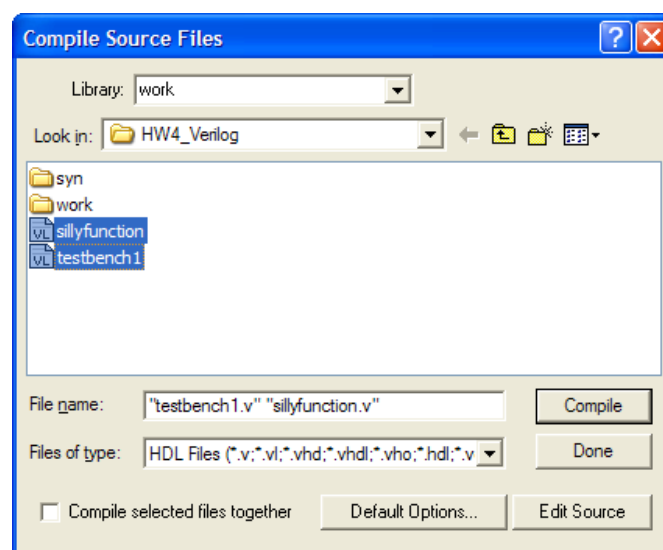
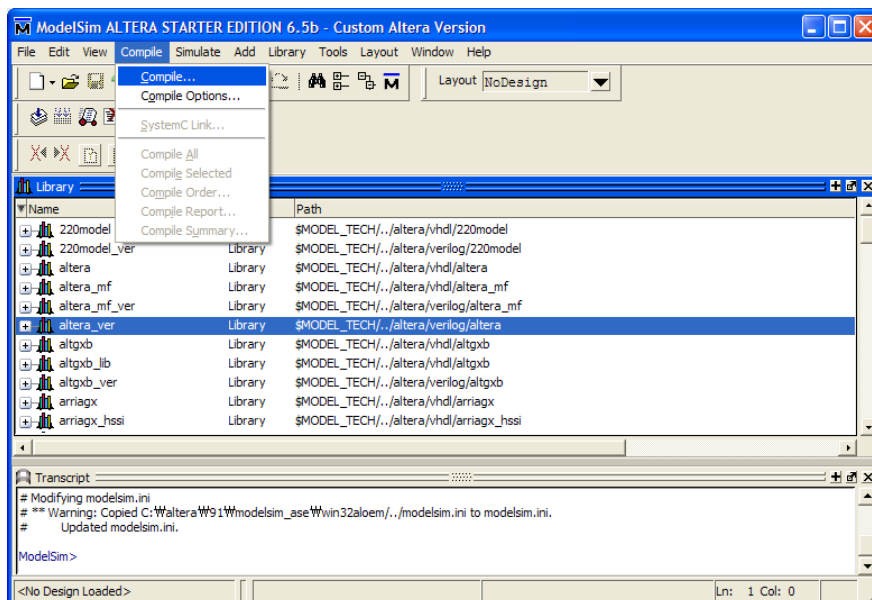


2. 'File → New → Library' and you can type a library name of your choice, but let's leave the name (work) as it is

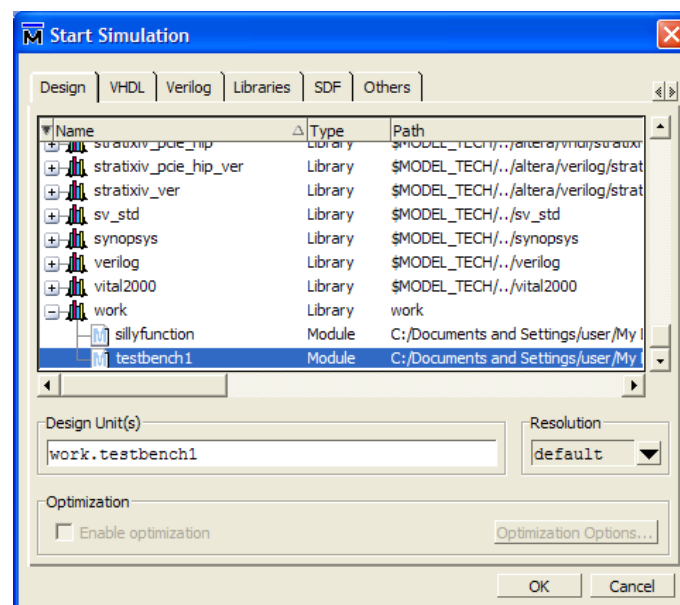
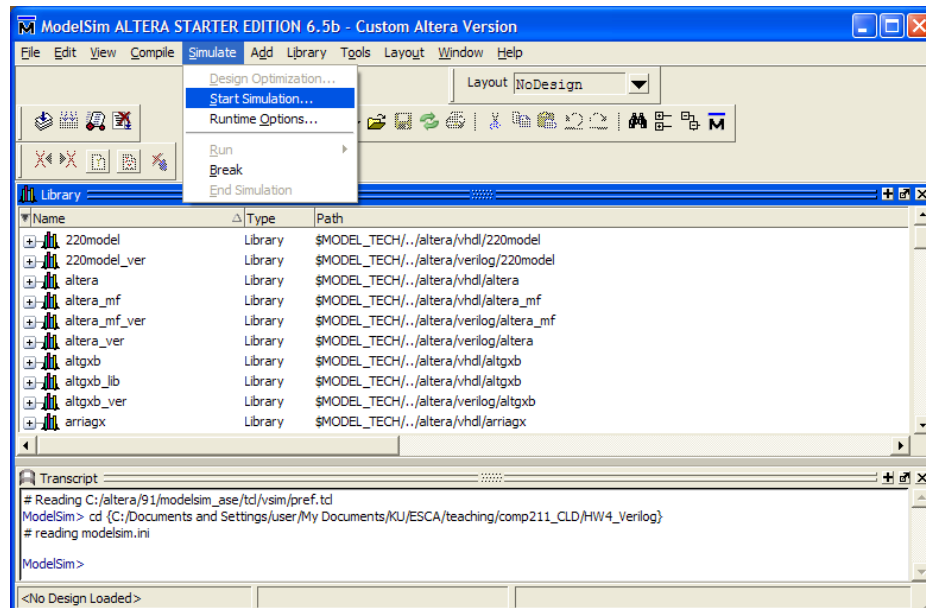




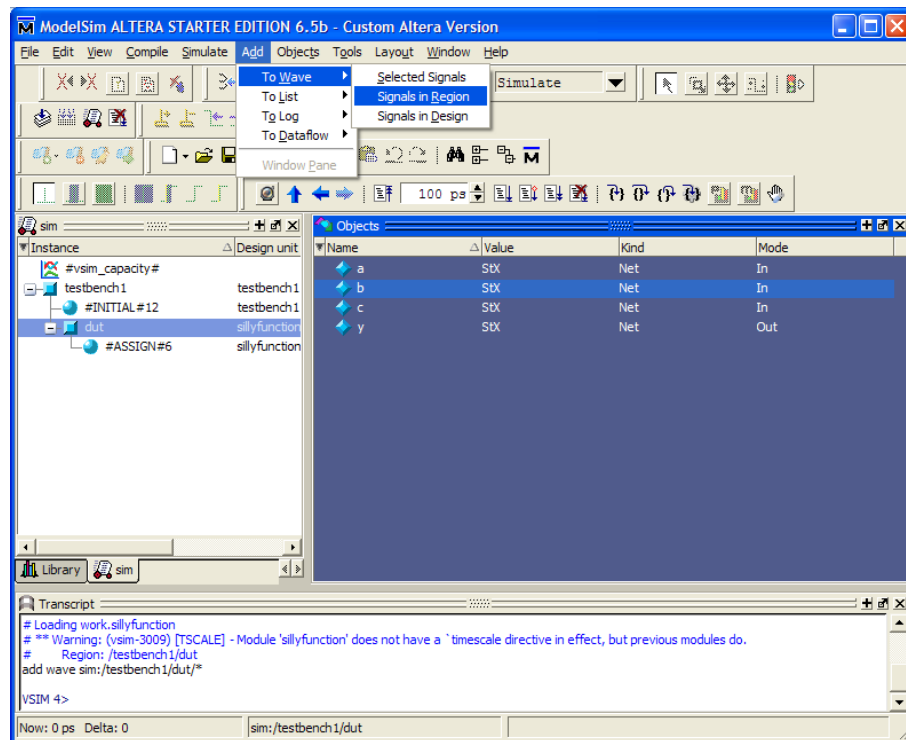
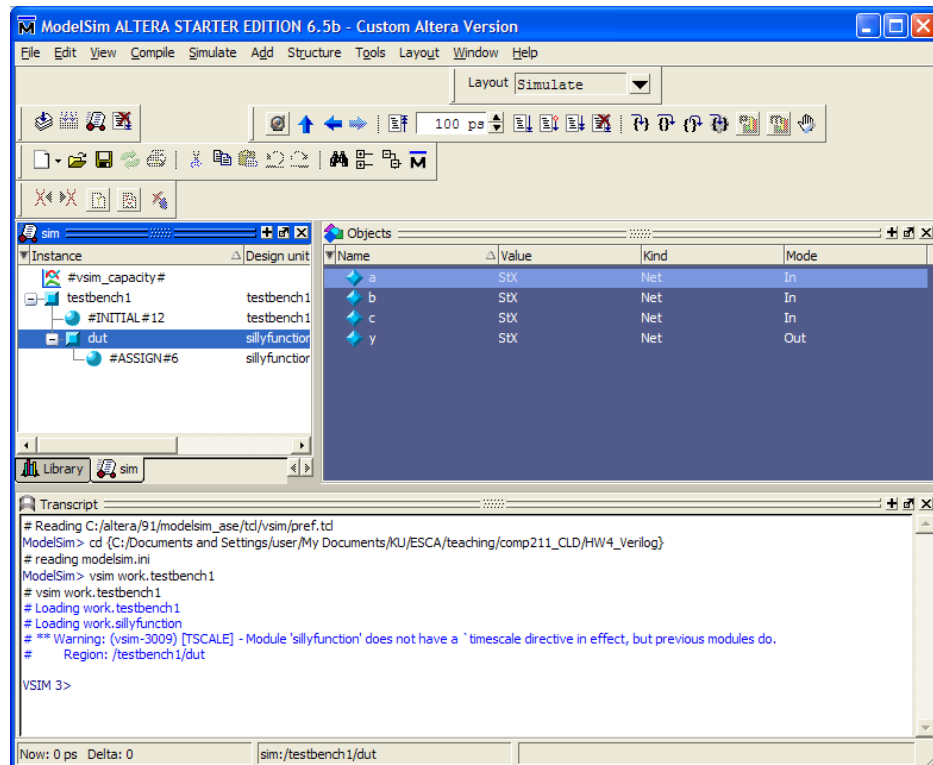
3. Compile the Verilog code you have downloaded by 'Compile → Compile' and choose both sillyfunction.v and testbench1.v



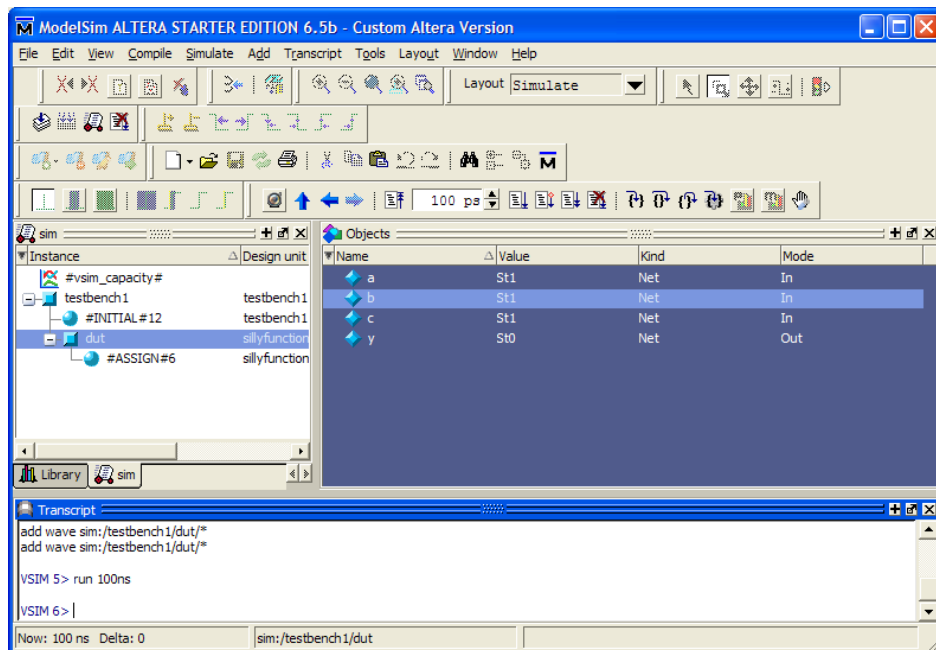
4. Run simulation by 'Simulation → Start Simulation' and choose testbench1 under work library. Then press OK



5. Choose 'dut' and add all signals (a, b, c, and y) to the waveform by 'Add → To Wave → Signals in Region'



6. Run simulation for 100ns by typing "run 100ns" in the Transcript pane.



7. See the full screen of the waveform by 'View → Zoom → Zoom Full'. Is the output (y) shown as you expected?

