Computer Organization Project

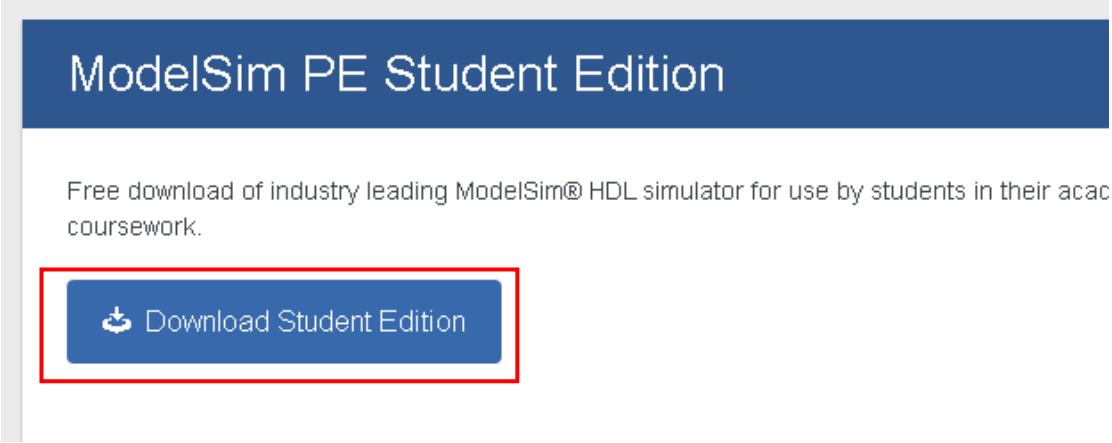
Lab0 – ModelSim Tutorial

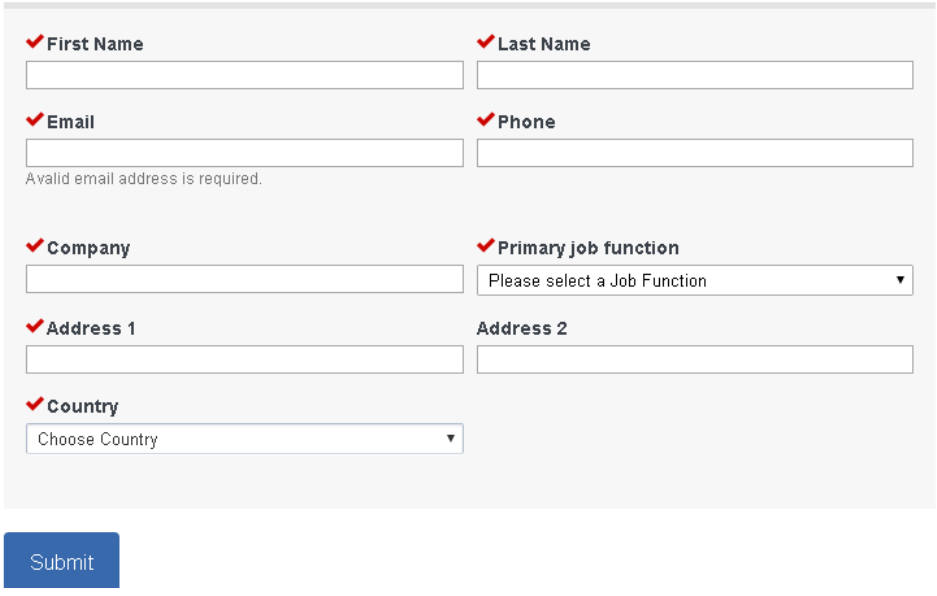
1. Goal

The goal of this Lab Unit is to encourage students to familiar with the environment of ModelSim IDE (Integrated Development Environment). Please download the attached file (Test.v) and go through all steps to finish this Lab.

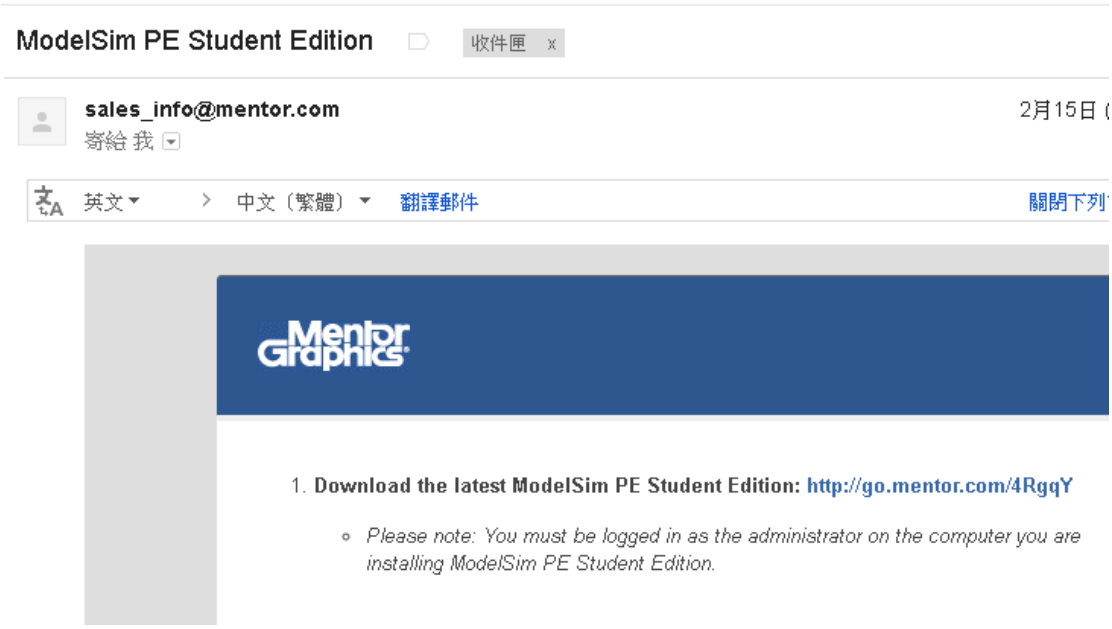
2. Setting Up of ModelSim

a. Please link to https://www.mentor.com/company/higher_ed/modelsim-student-edition to download ModelSim and finish the form.

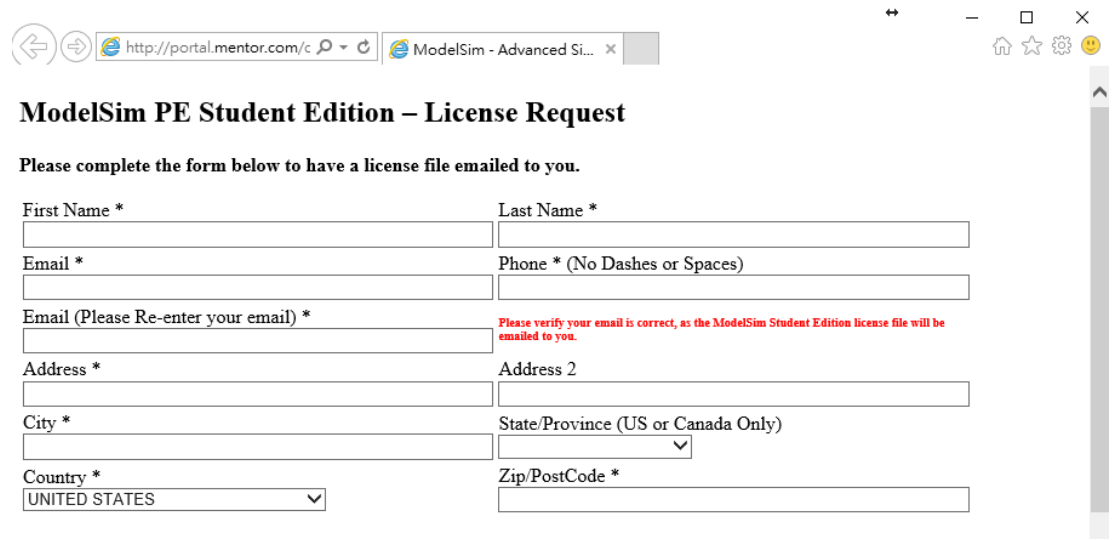




b. Check your E-mail box, and you will accept a mail from ModelSim. Then, you can download it. (The mail may be sent to trash mail box.)



c. At the end of the installation process, select Finish and a browser window will open with the License Request form.



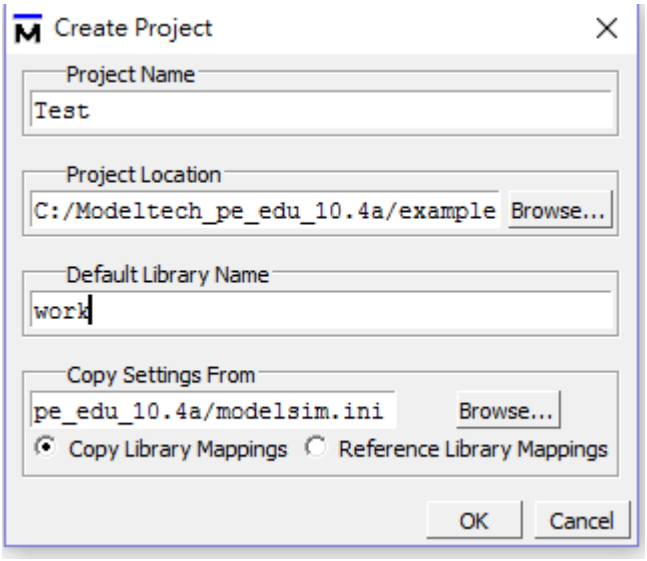
Please fill the form out, and then you will receive an email with license file (student_license.dat). Download the license and copy it to your folder of ModelSim (the default path is C:\Modeltech_pe_edu_10.4a)

d. Double click “ModelSimPE Student Edition 10.4a” and run the ModelSim process.

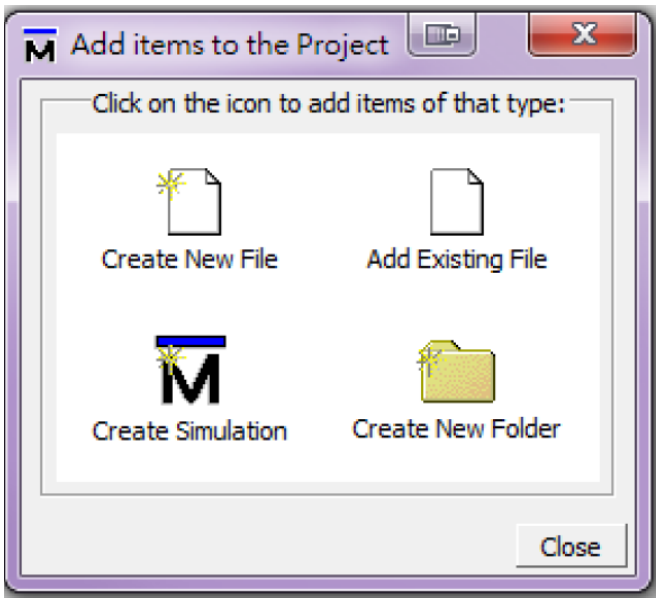
3. Starting to use ModelSim

A. Create a new project and compile your program

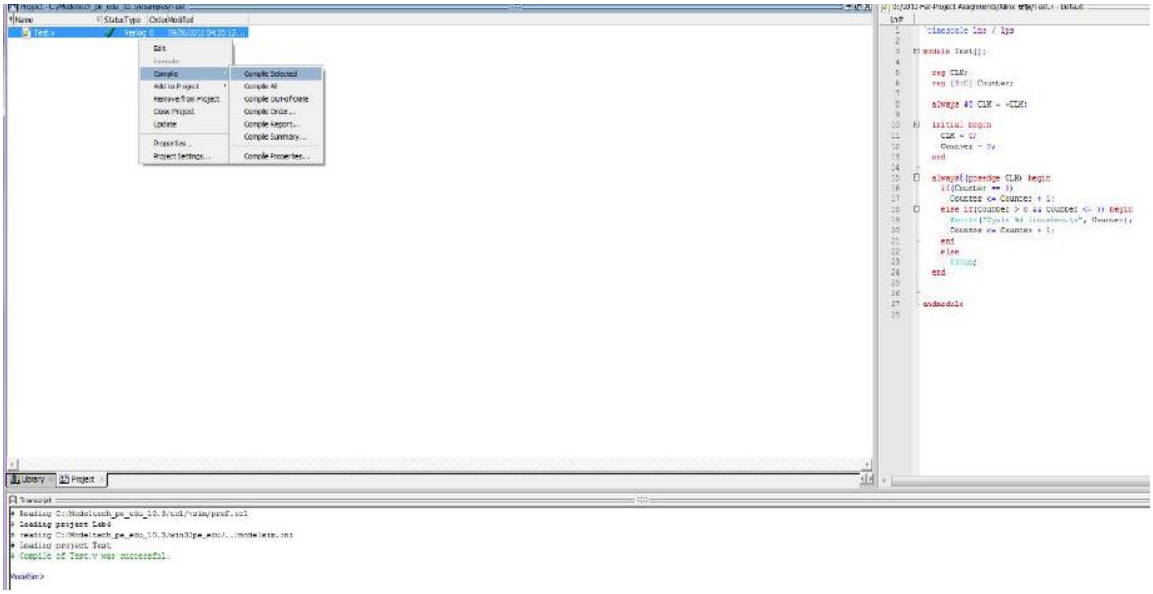
a. Click the toolbar on ModelSim, “File->new->project”, and a window will pop out as below. Fill in the Project Name and then click OK.



b. For a project, you may create a new folder or add an existing file. In this Lab Unit, please select “Add Existing File” to add “Test.v” into your project.



c. Then, the file “Test.v” will show up in the project window as shown below:

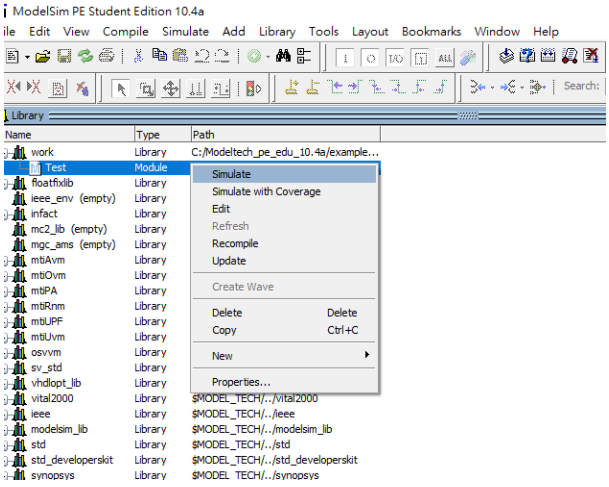


For editing the program, you may double click Test.v and then an editor window will be shown on the right-hand side.

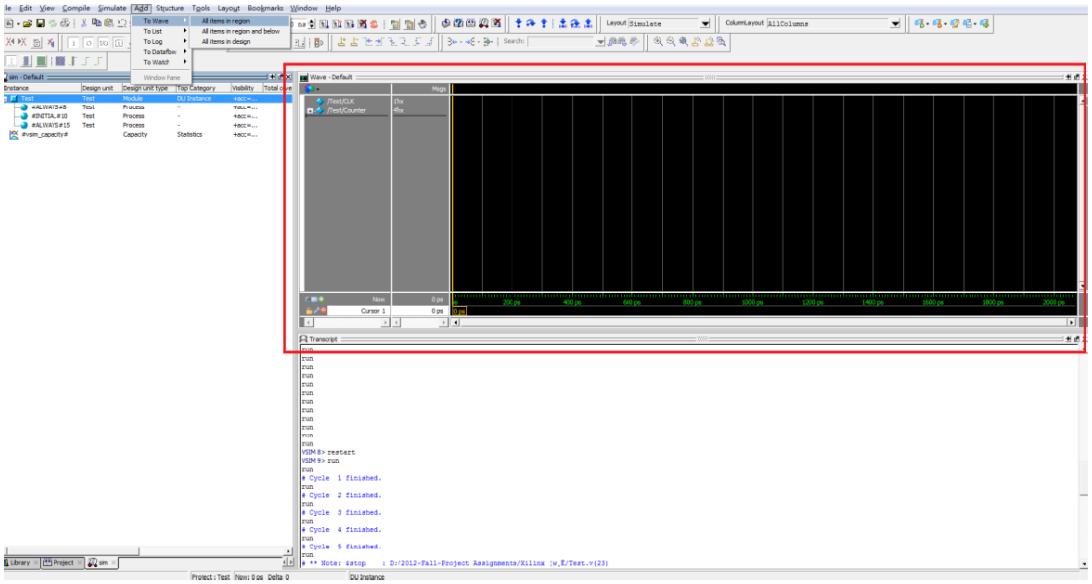
For compiling the program, right click “Test.v” and then select “Compile->Compile Selected”. The Verilog program will be compiled and the compiled module(Test) will be generated in the work library.

B. Simulate the program

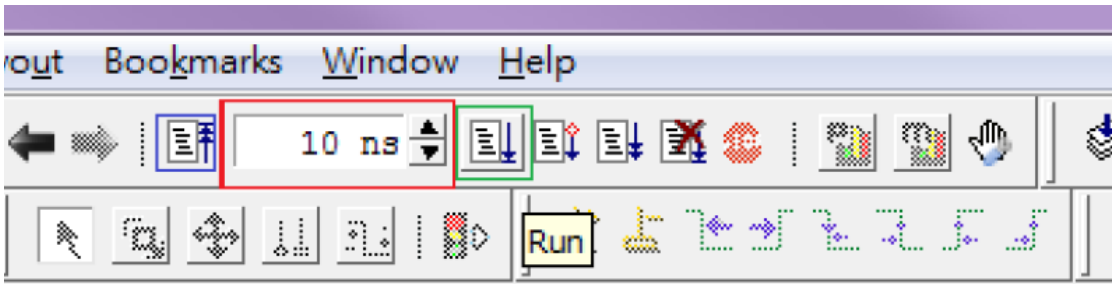
a. Switch your window to “Library”, extend “work”, and then right click module “Test” and select “Simulate”.



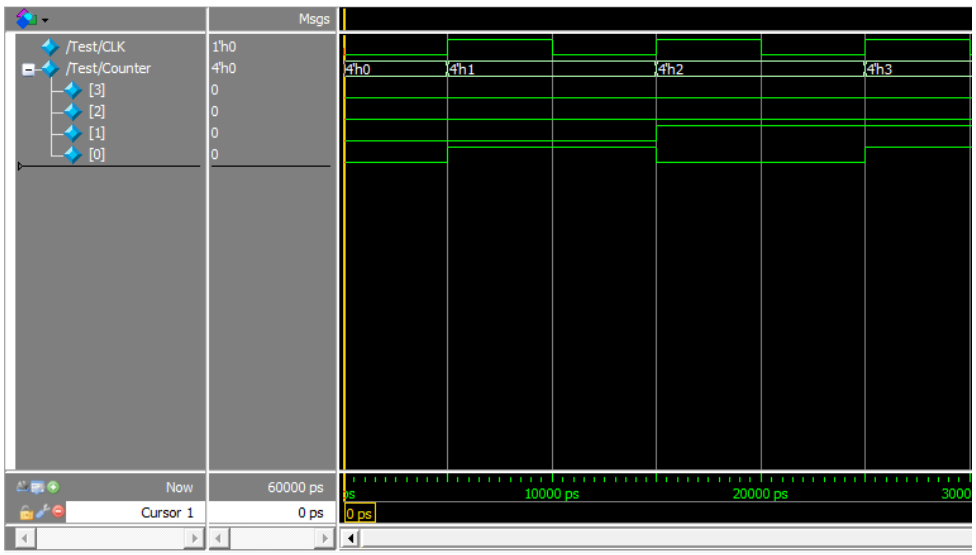
b. After “Simulation”, click the toolbar on ModelSim, “Add→To Wave→All item in region”, and a wave form diagram window will show up as below.



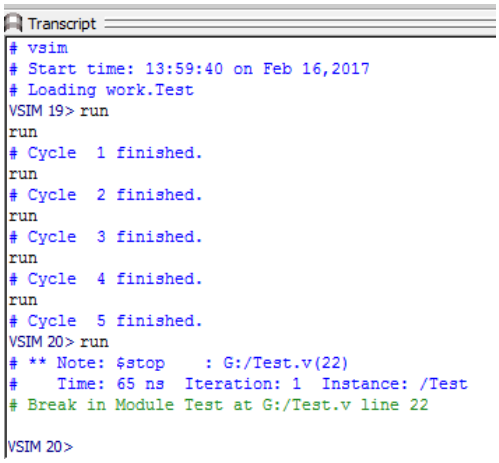
c. Set the time period of each simulation step as 10ns (the red square in the following figure) in this Lab Unit. Each click of “Run” (the green square) will generate the result of the simulation for10ns.Click“Restart” (the blue square) to reset the simulation, if necessary.



d. After complete all the steps described above, you will get the result of simulation. The wave form diagram shown below is the simulation results of “Test” for 30ns.



Besides the wave form diagram, you may also need to get some output from the “Transcript” window which will be shown below the waveform diagram. (If the “Transcript” window is not shown, click the toolbar on ModelSim, “view->Transcript”, and then the window will show up.)



4. Deadline

No need to hand in the report of Lab0. Please be familiar with ModelSim for the implementation of the succeeding Lab Units.