

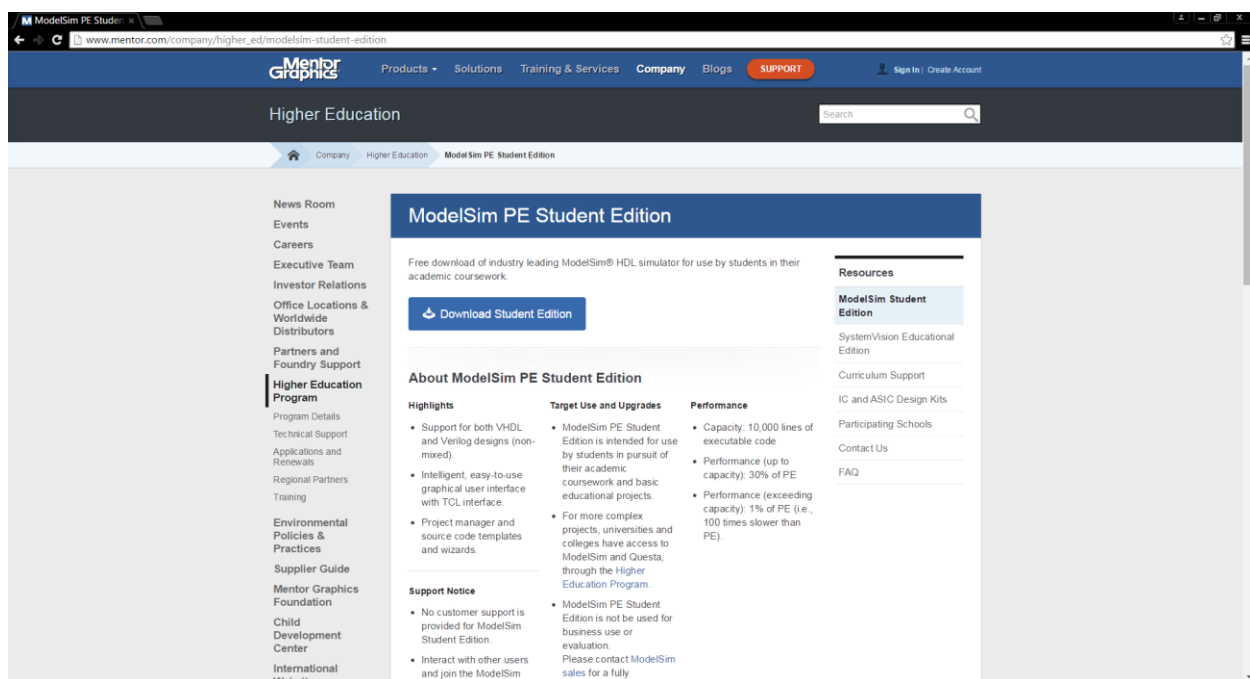
ModelSim Installation and Simulation Tutorial

ModelSim is a verification and simulation tool for VHDL, Verilog, System Verilog and mixed-language designs. In this tutorial we show how to install the ModelSim Student Edition. Please note that only Windows operating system is supported. Mac users can use VirtualBox¹ to install Windows and install ModelSim in Windows OS. Linux users can refer to the appendix for instructions on how to install ModelSim on Linux.

ModelSim Installation

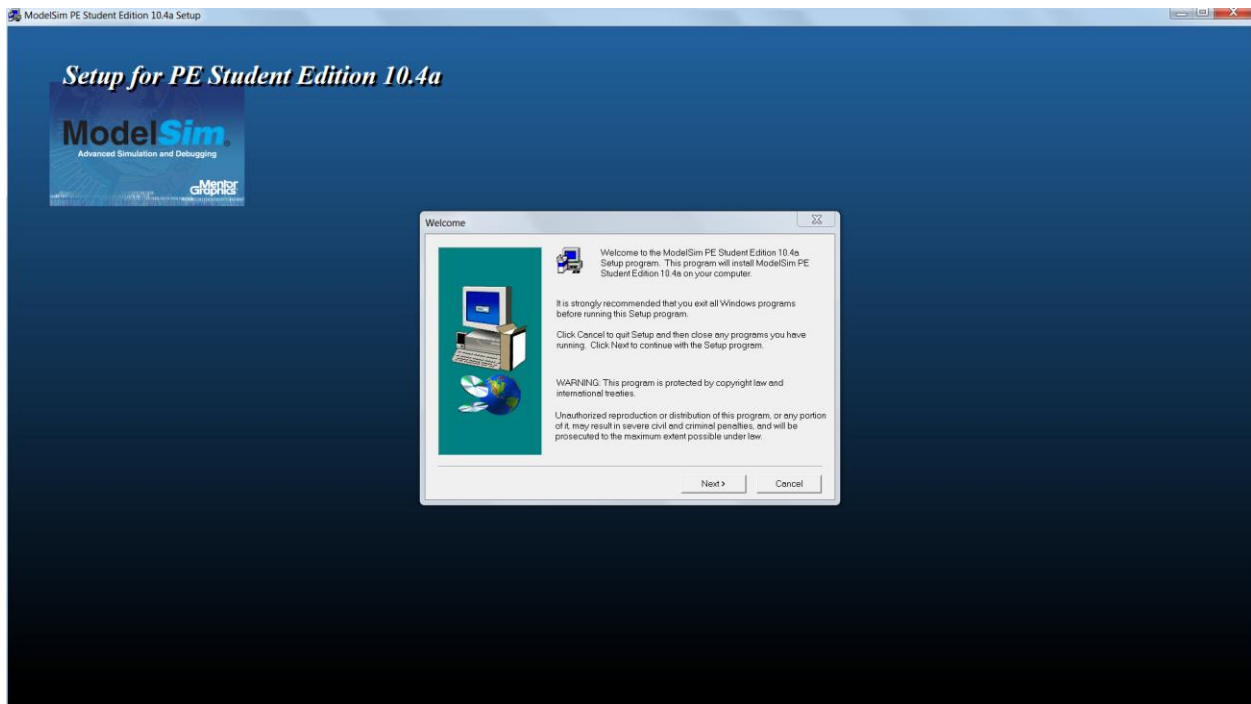
1. Download the ModelSim Student Edition from

http://www.mentor.com/company/higher_ed/modelsim-student-edition

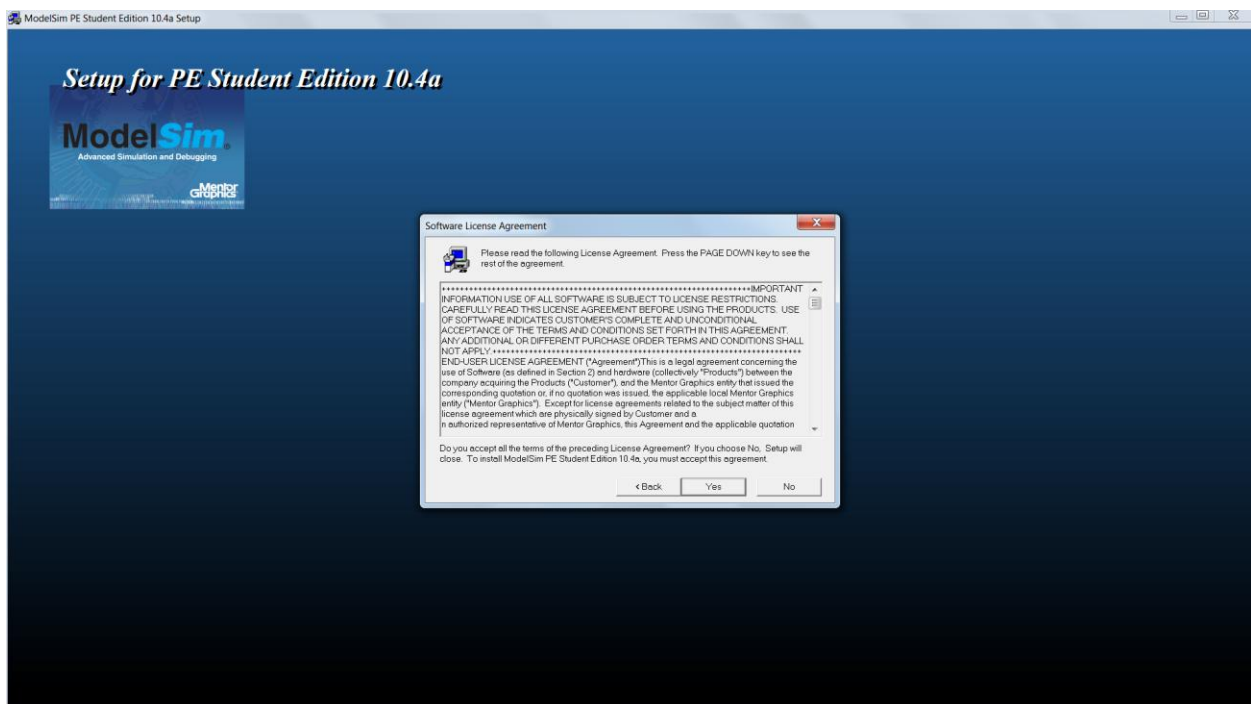


¹ <https://www.virtualbox.org/wiki/Downloads>

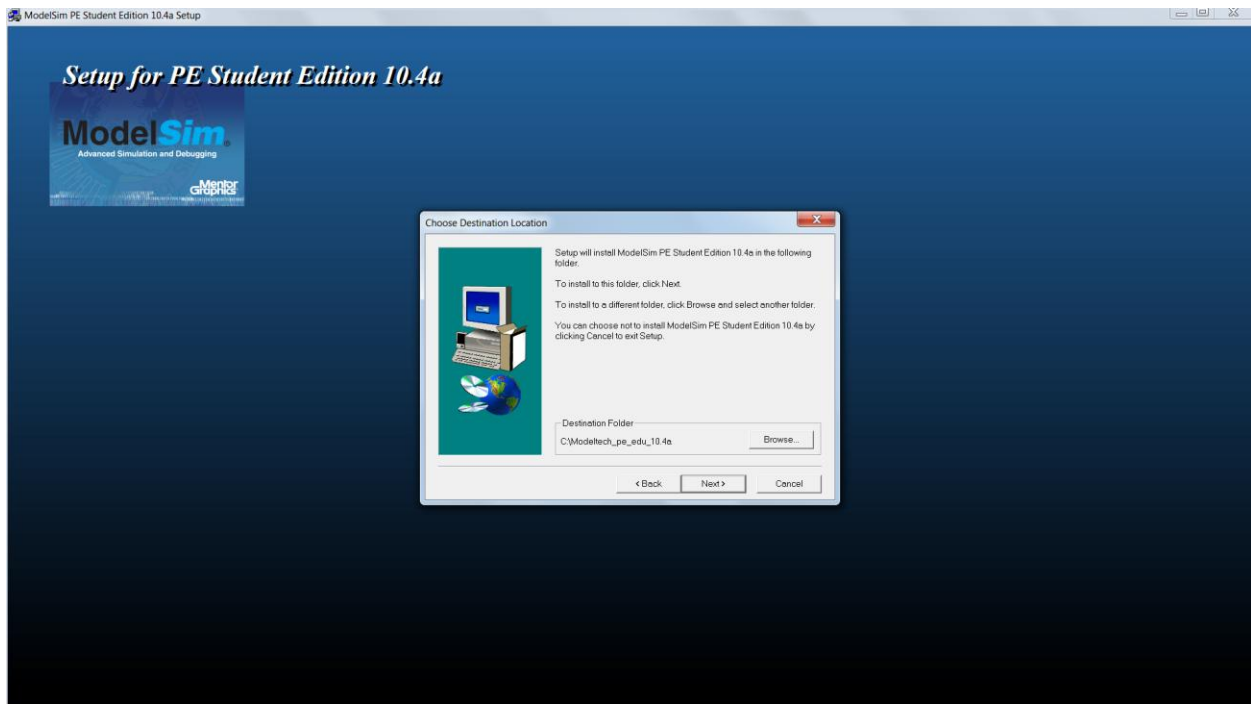
2. After downloading the installer, run it. Click Next.



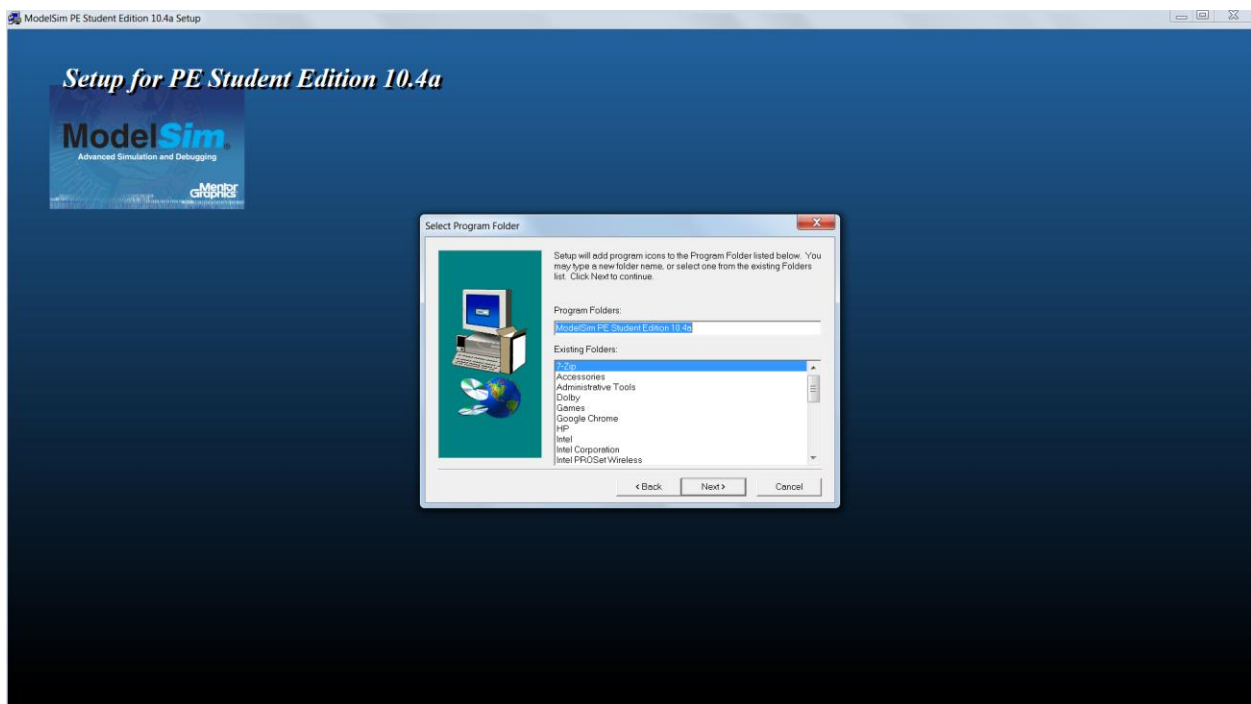
3. Agree with the License Agreement.



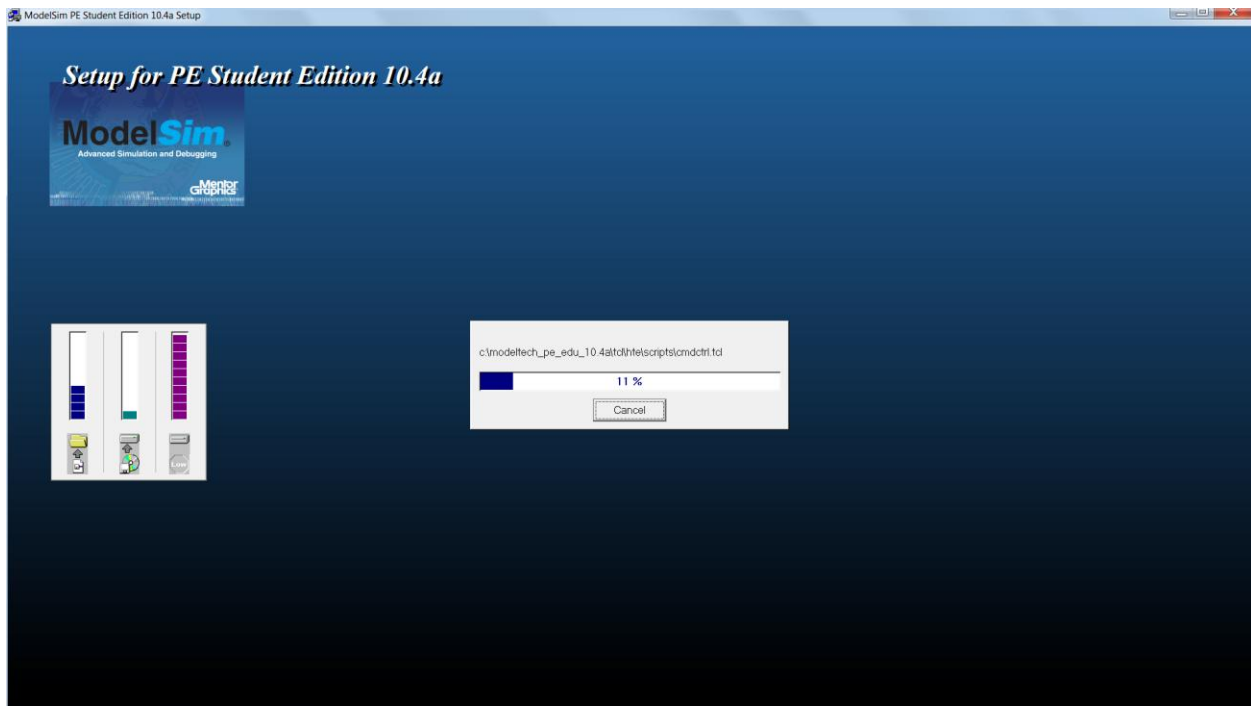
4. Choose the destination folder. You can use the default location and click Next.



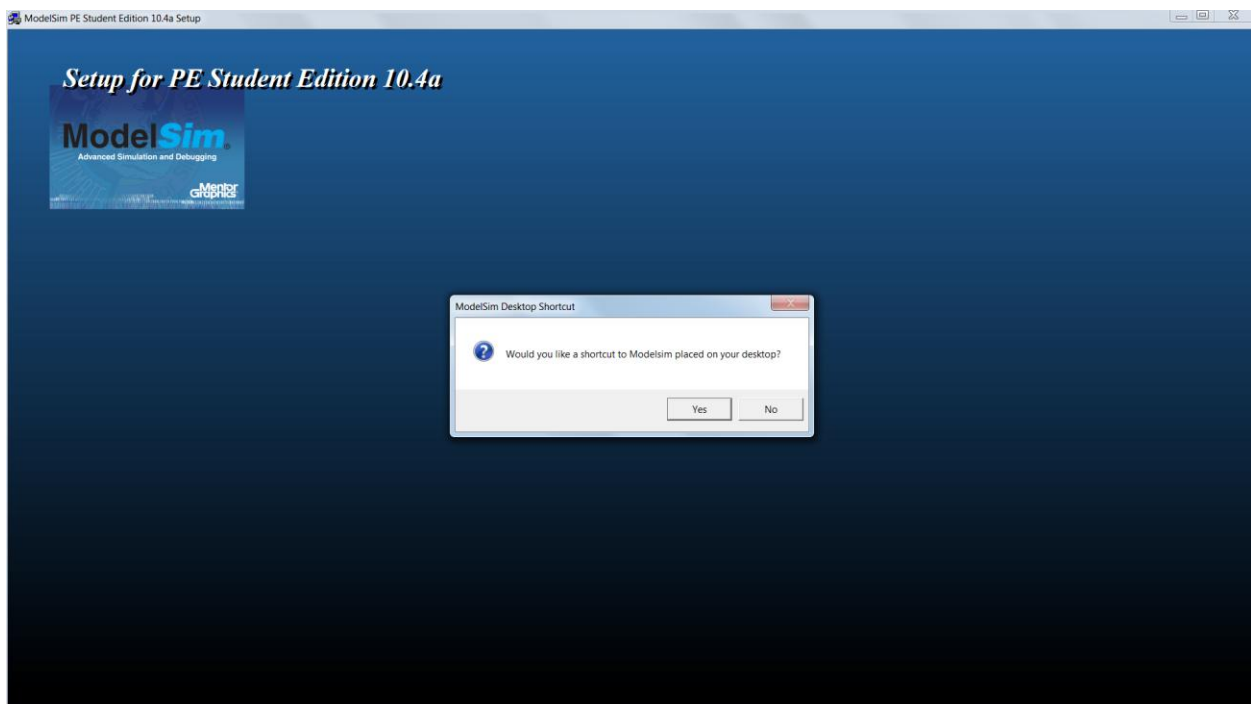
5. Click Next to start installation.



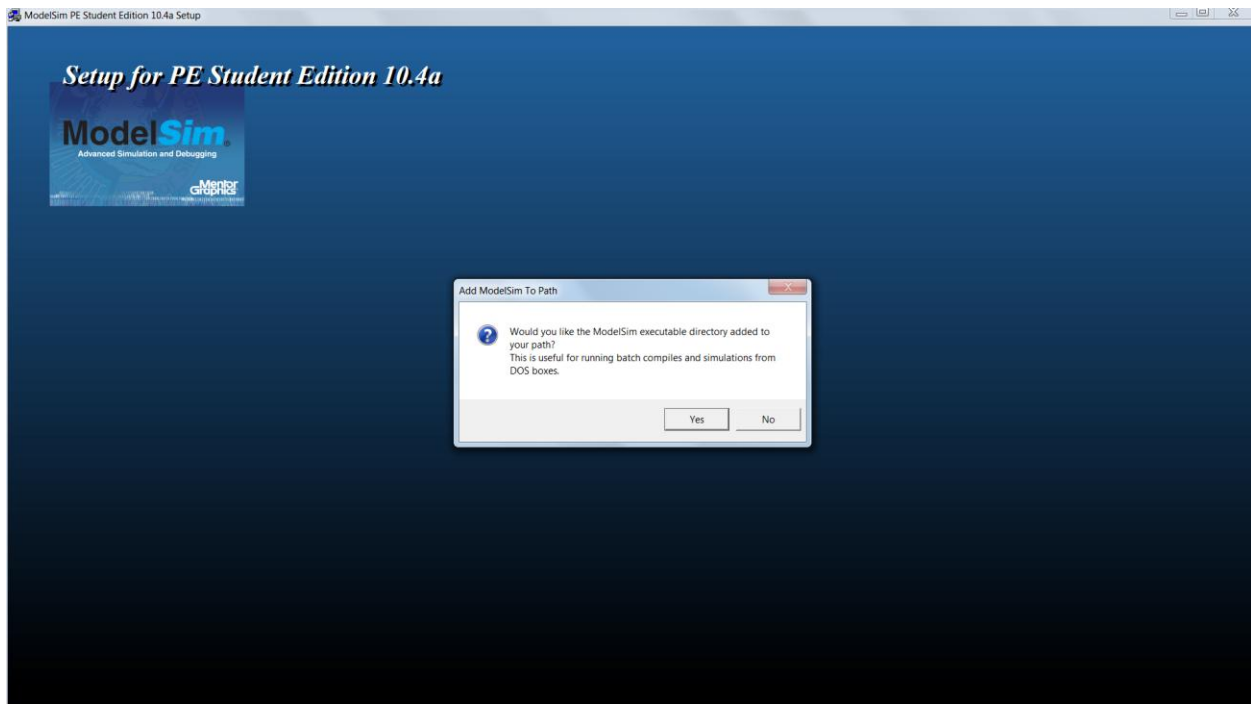
6. ModelSim Student Edition software will be installed in the selected destination folder.



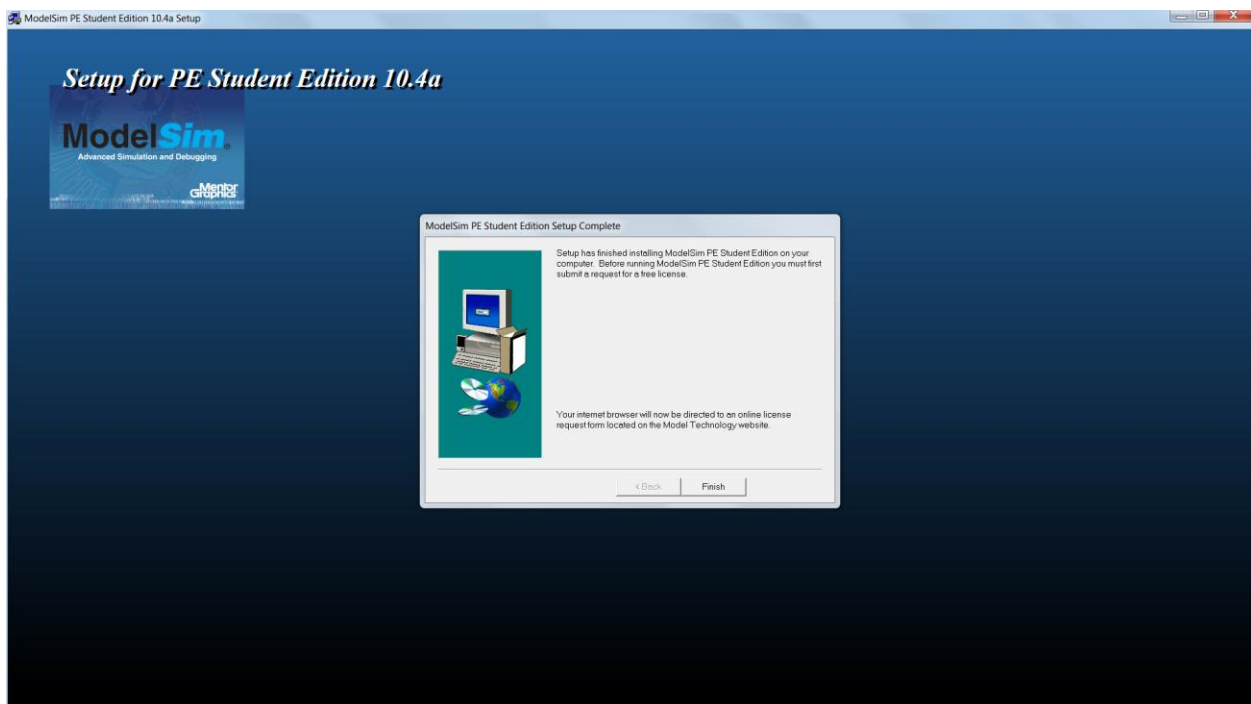
7. You can choose to put a shortcut to ModelSim on your desktop.



8. After Installation is complete, add Modelsim to "path". Click Yes.



9. Click Finish to submit a request for a free license.



10. After you click Finish in the last step, the browser will go to a page to request for free license. Fill the form, providing your valid email ID and click Request License.

The screenshot shows a web browser window with the URL portal.mentor.com/d/license_request.asp. The page title is "ModelSim PE Student Edition – License Request". Below the title, it says "Please complete the form below to have a license file emailed to you." The form is divided into two main sections. The first section contains personal information fields: First Name *, Last Name *, Email *, Phone * (No Dashes or Spaces), Email (Please Re-enter your email) *, Address *, Address 2, City *, State/Province (US or Canada Only), Country * (with a dropdown menu showing "UNITED STATES"), and Zip/PostCode *. A red error message is visible next to the second email field: "Please verify your email is correct, as the ModelSim Student Edition license file will be emailed to you." The second section is titled "Please tell us about yourself" and includes a field for "Please specify your University, College, School, or Institute: *". Below this is a question "Are you a Student or Instructor? *" with radio buttons for Student, Professor / Instructor, and Other, followed by a text input field. If the user is a student, there is a sub-section "Please indicate your grade or position: *" with radio buttons for Freshman, Sophomore, Junior, Senior, Graduate Student, and Other, followed by a text input field. Below this are fields for "Expected graduation year: *" and "Expected graduation term: *" with radio buttons for Spring, Summer, Fall, and Winter. At the bottom, there is a checkbox for "Have you lined up a job where you'll be using a simulator upon graduation?".

11. You will receive an email with ModelSim student edition license key and steps for the final installation. Follow the steps to include valid license to your ModelSim.

Note:

- License files are valid only for the current installation of the software on the computer on which the software is installed.
- If for any reason you need a new license file, you must go through the entire process of download, installation and license request.

12. Save the license file attached to the email to the top level installation directory for ModelSim (e.g. C:\Modeltech_pe_edu_10.4a).

13. Tada! You have ModelSim installed on your computer!

First Simulation

Let's simulate a simple full adder using ModelSim. You can use your favorite text editor to make the file containing the verilog HDL code which describes the design, full_adder.v.

Create a Project

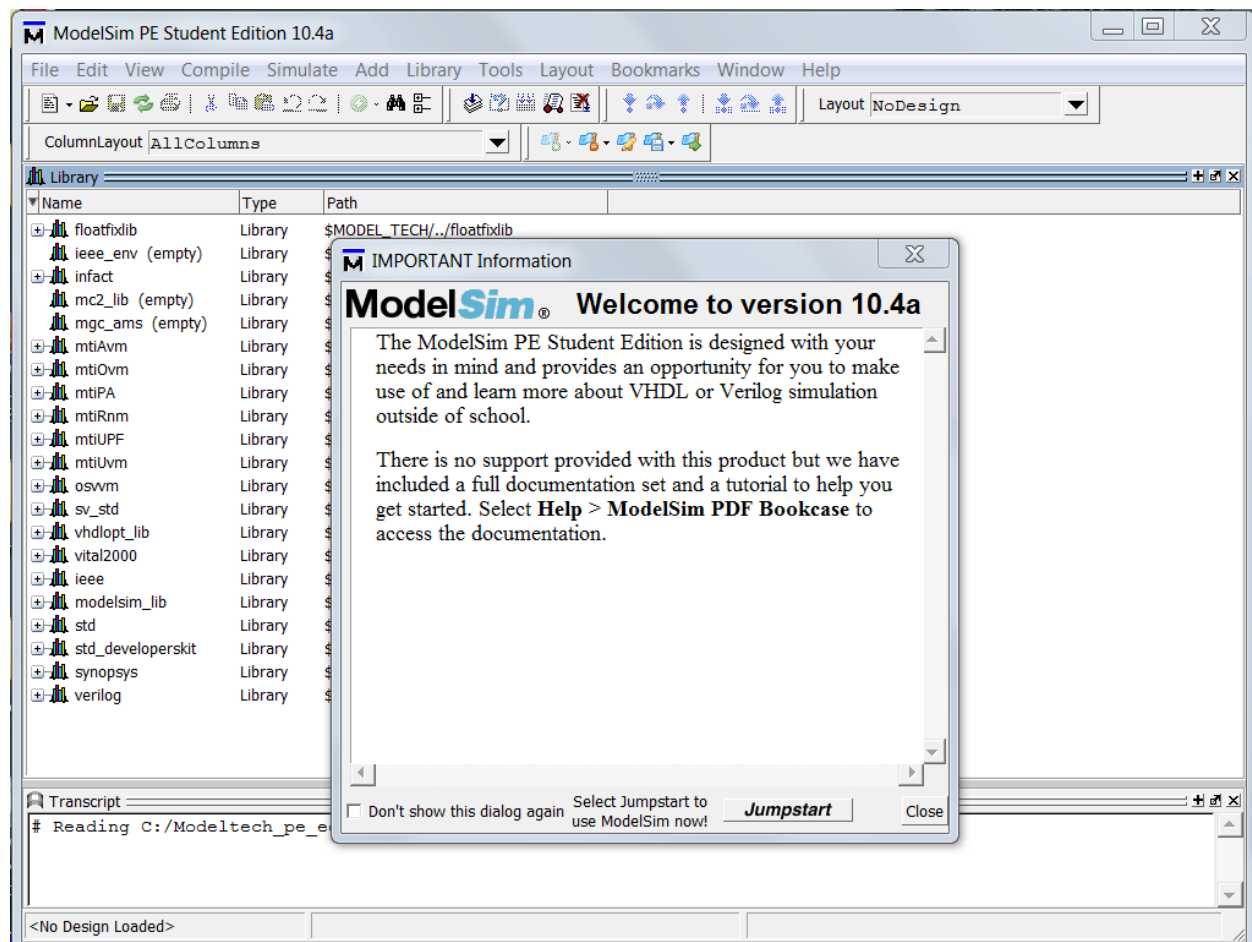
A project is a collection entity for an HDL design under specification or test. Projects ease interaction with the tool and are useful for organizing files and simulation settings.

1. Start ModelSim

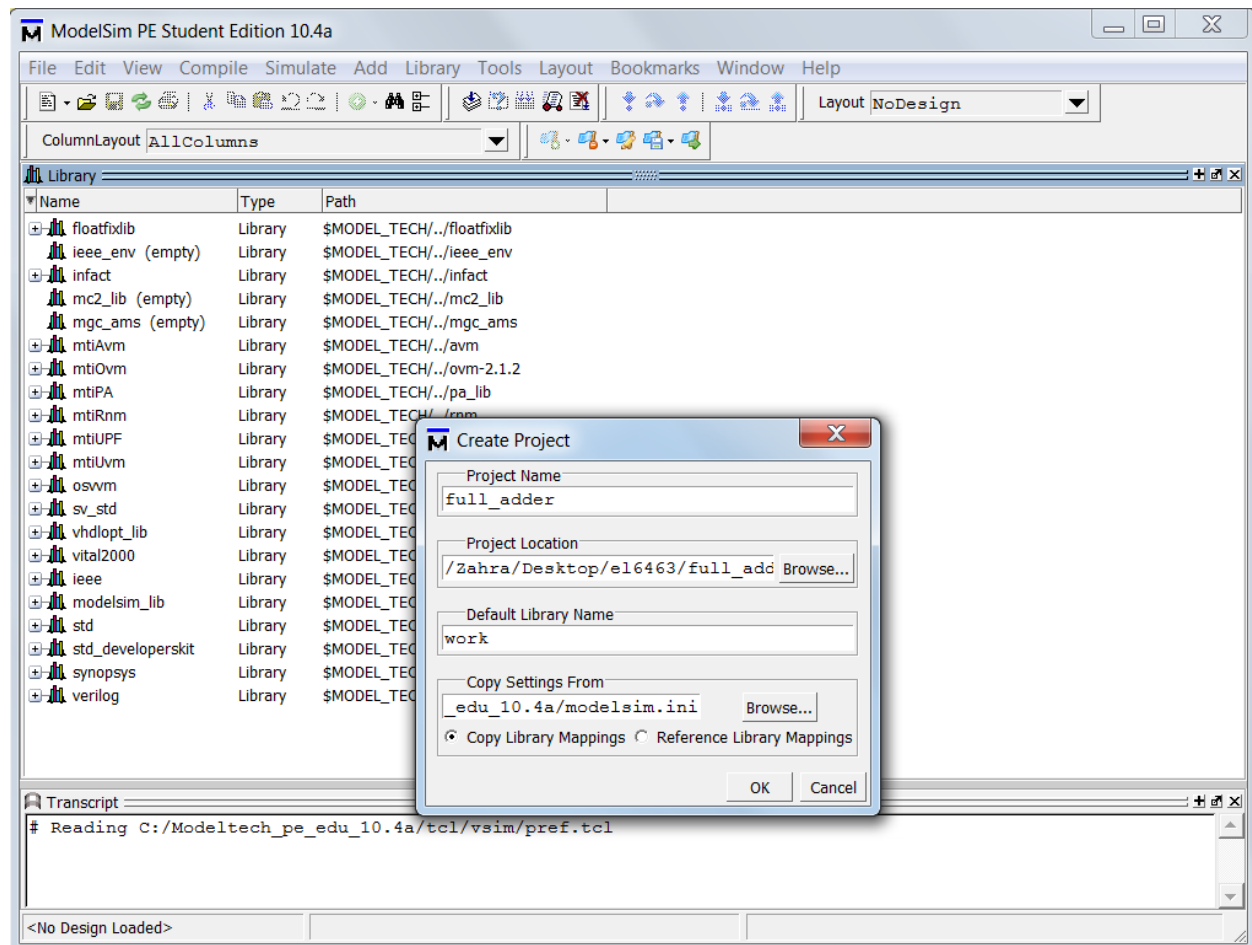
Run the program from a shortcut icon, or the start menu.



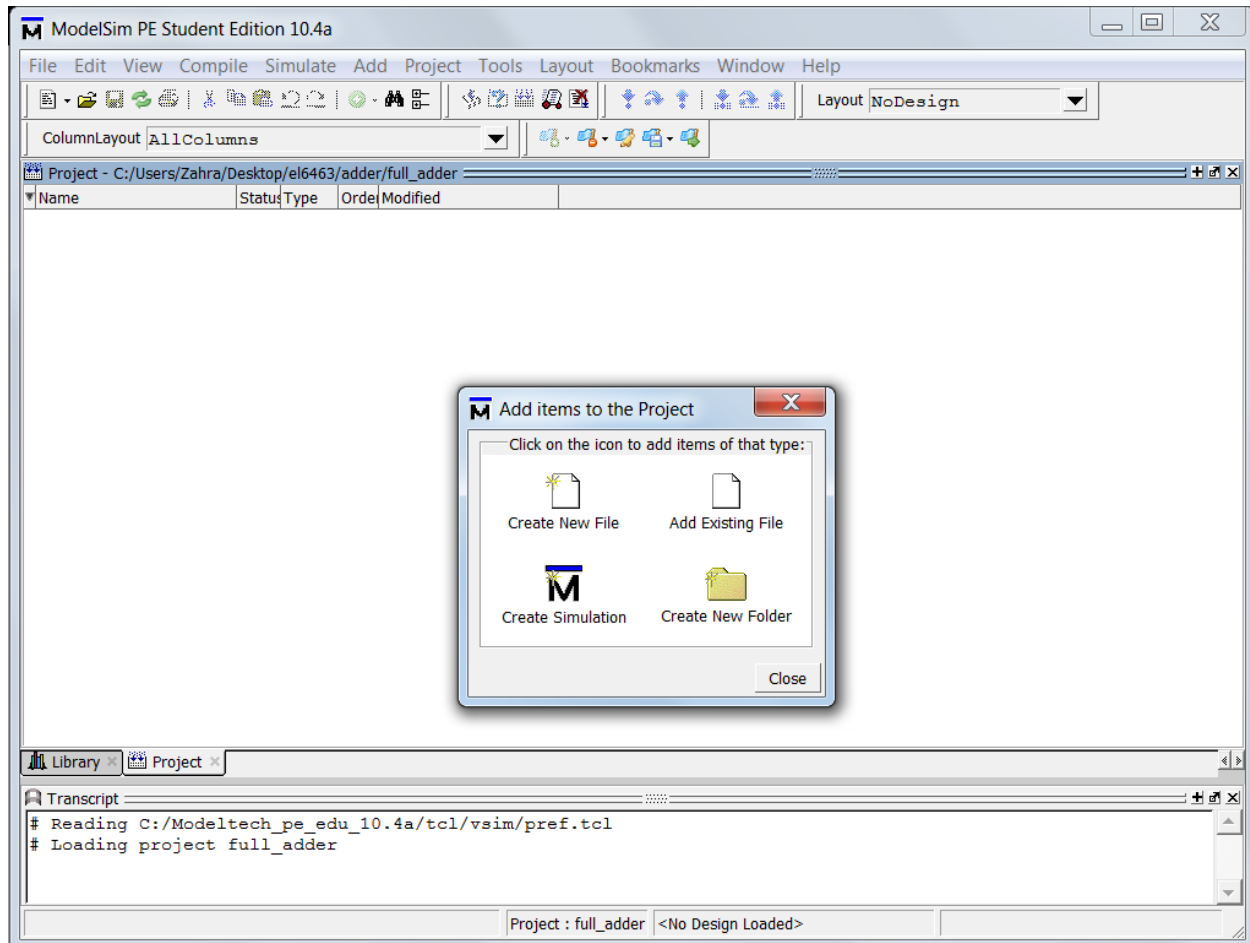
Upon opening ModelSim for the first time, you will see the Welcome to ModelSim dialog.



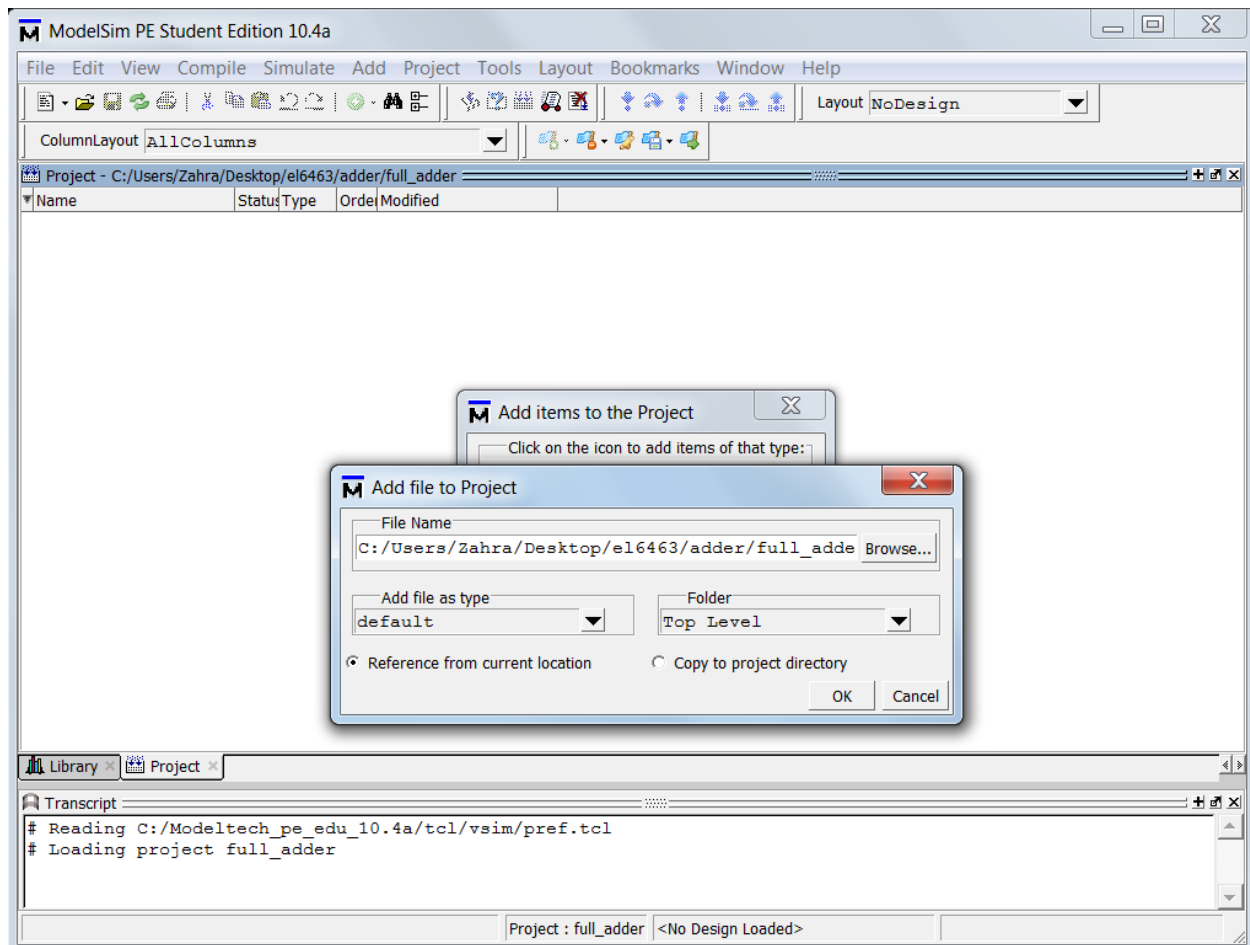
2. Select **Create a Project** from the welcome dialog (Jumpstart) or **File > New > Project** in the main window. Enter a name as the Project Name and select a directory where the project file will be stored. Leave the Default Library Name set to "work".



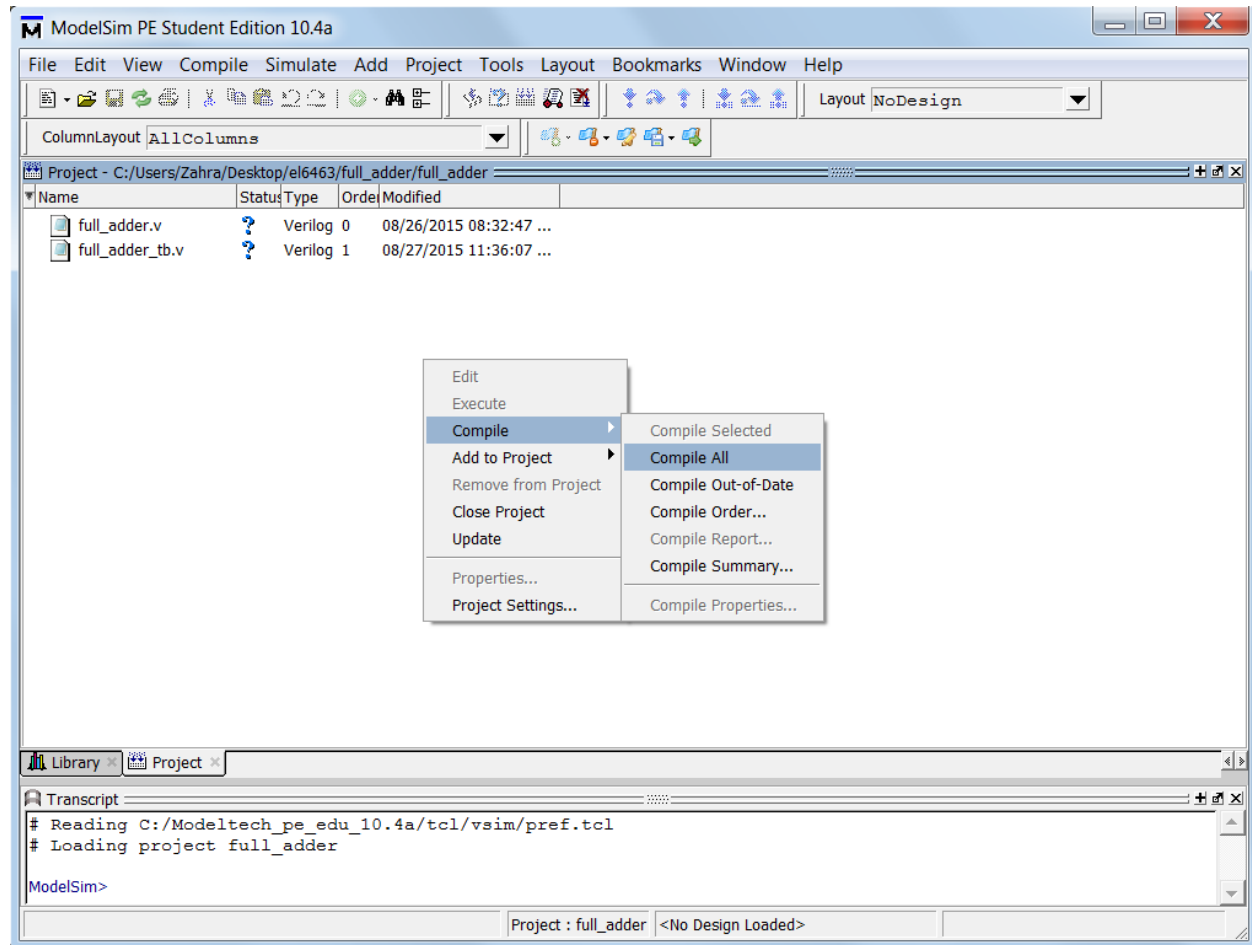
3. Upon selecting OK, you will see a blank Project tab in the workspace area and the **Add Items to the Project** dialog.



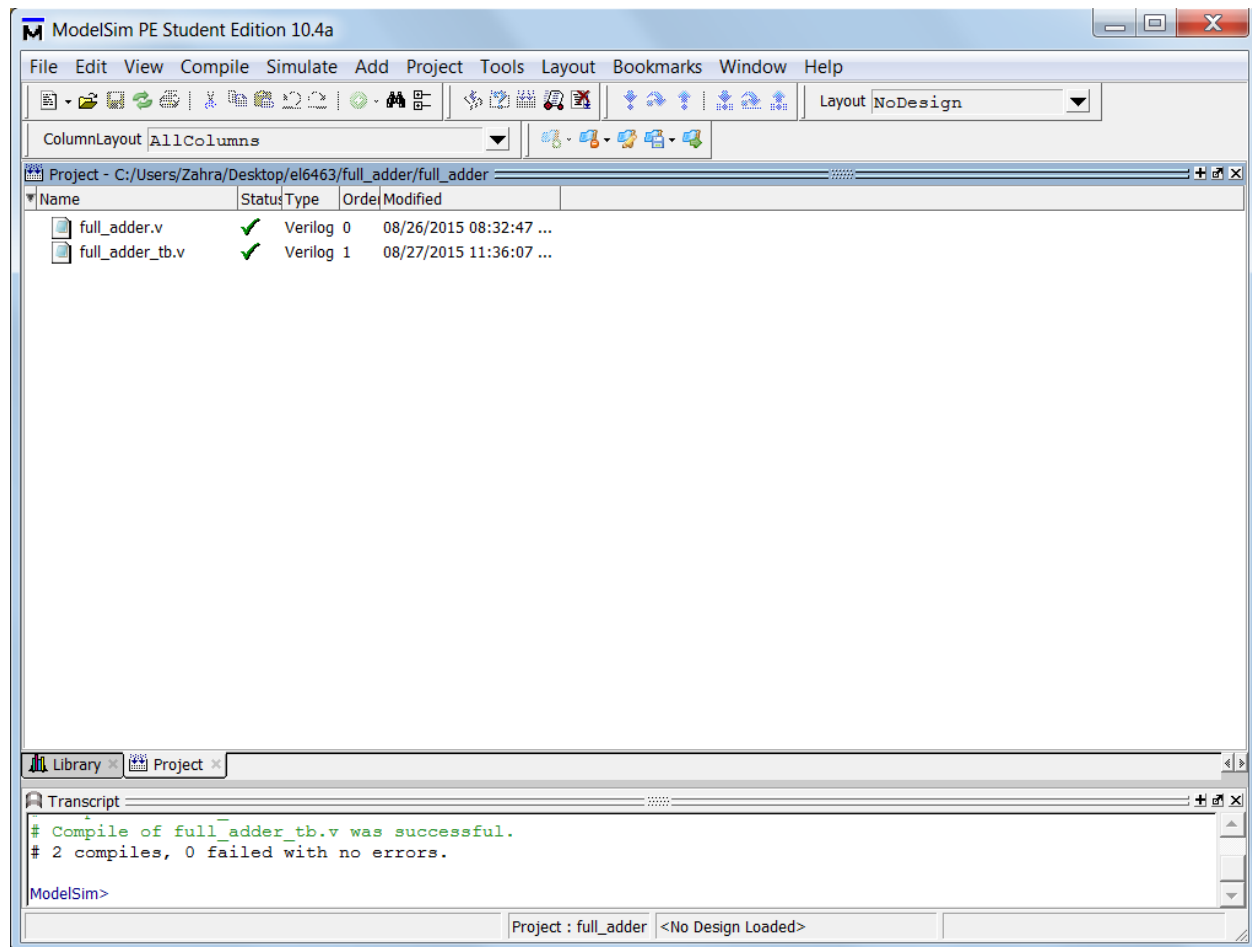
4. Click **Add Existing File** in the **Add Items to Project** dialog. Click the Browse button and select your full_adder.v and full_adder_tb.v files to add it to the project. Click OK. When you are done adding files, close the **Add Items to Project** dialog.



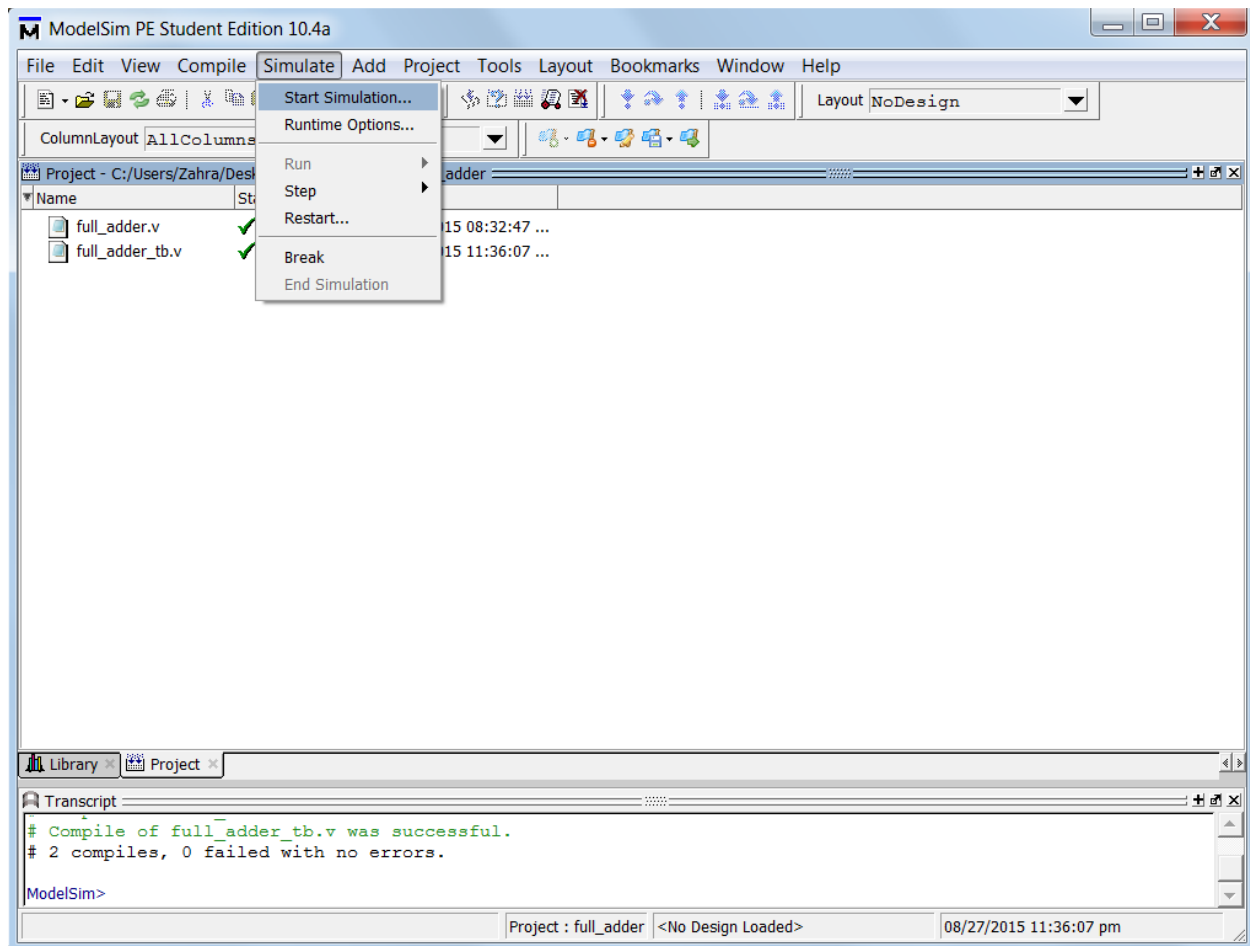
5. To compile the file(s) added to the project, right click in the main window and choose **Compile > compile All**.



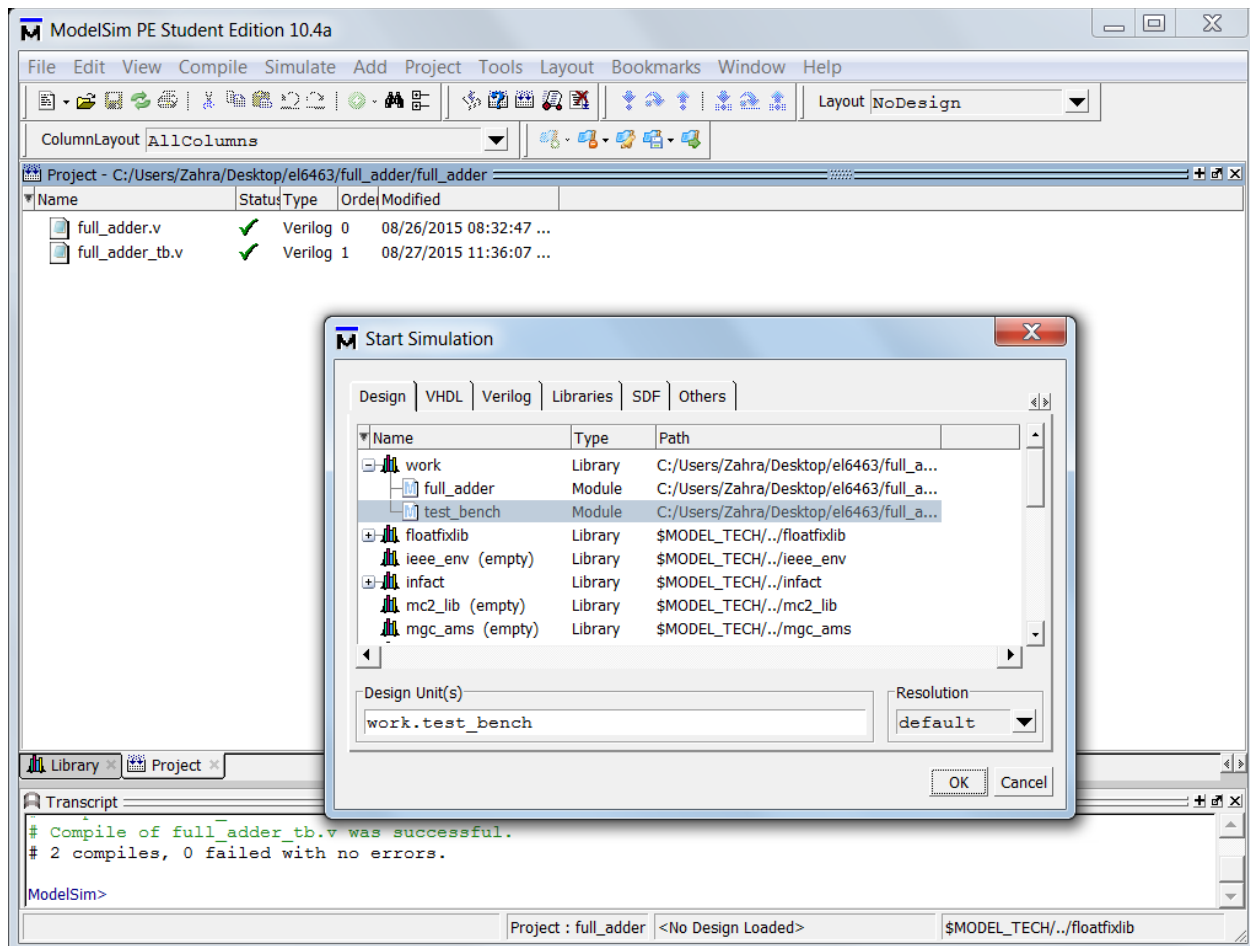
If the compilation is successful, the "?" in the Status would change to "✓", otherwise a red cross "×" will show in the status and there will be an error message in the **Transcript** command line window. In this case, you can double click the file to debug the Verilog code and fix the errors.



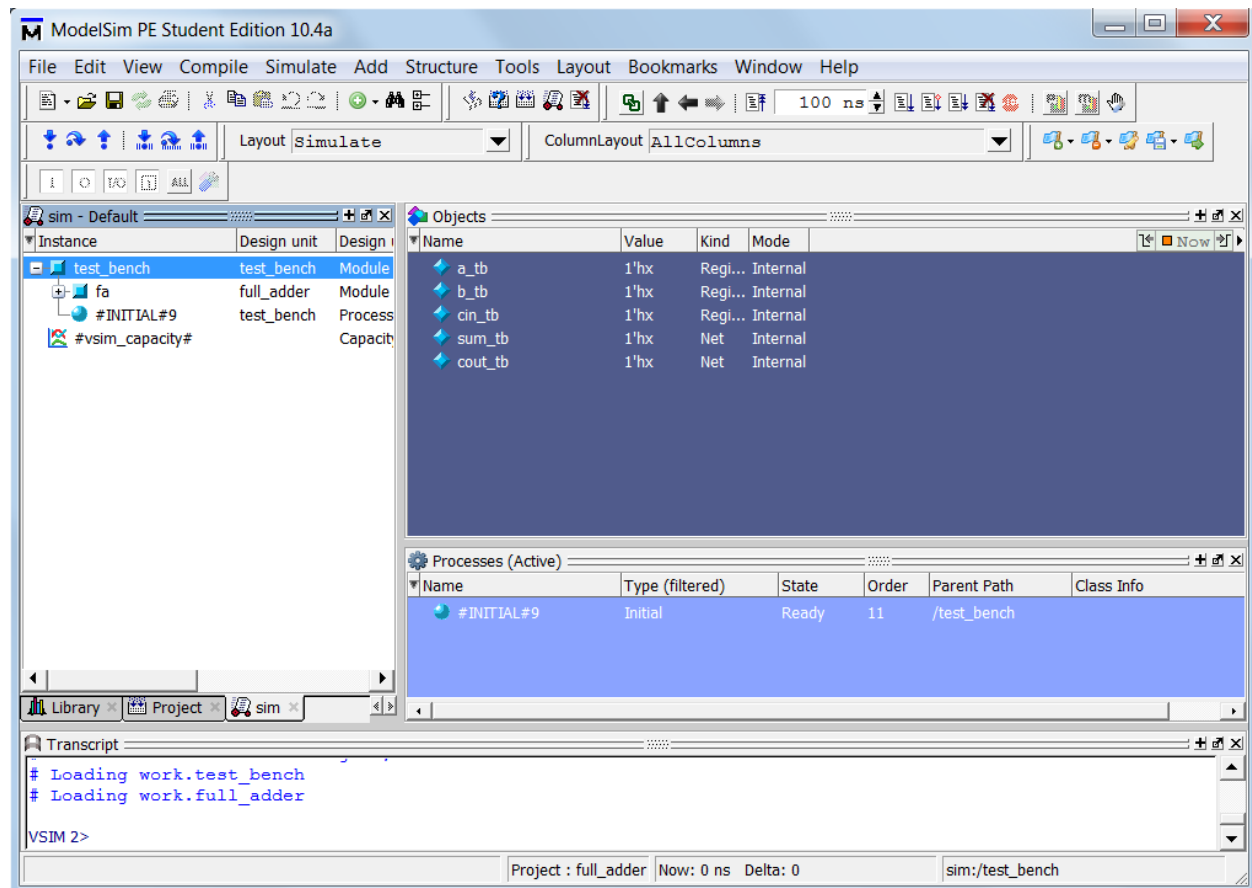
6. Now we can go ahead and simulate our design. Select **Simulate > Start Simulation...** .



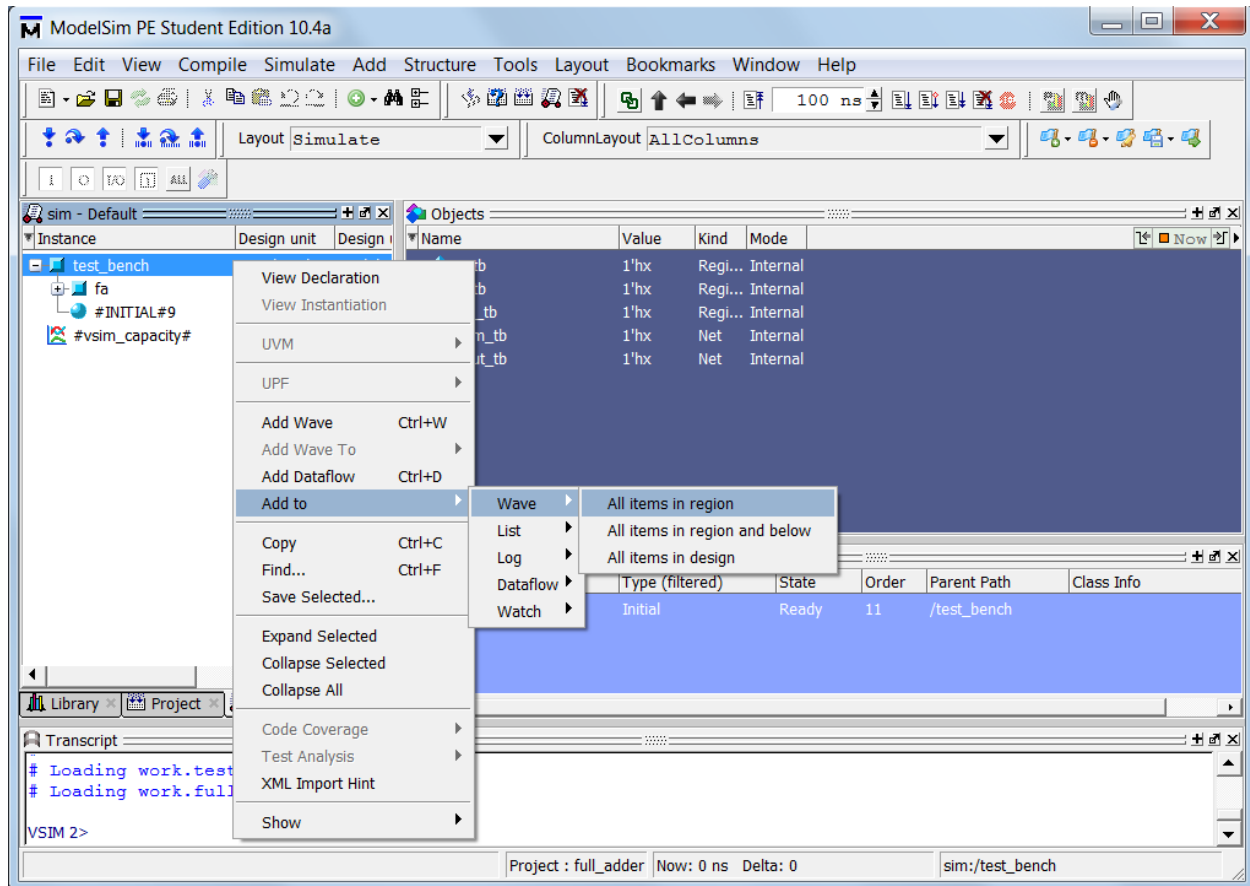
7. Choose your design (test_bench) under work in the **Start Simulation** dialog. Click OK.



8. Here is the general view of the simulation window. In the left window all your modules are shown. When you choose a module, all the signals in this module would be shown in the right window.



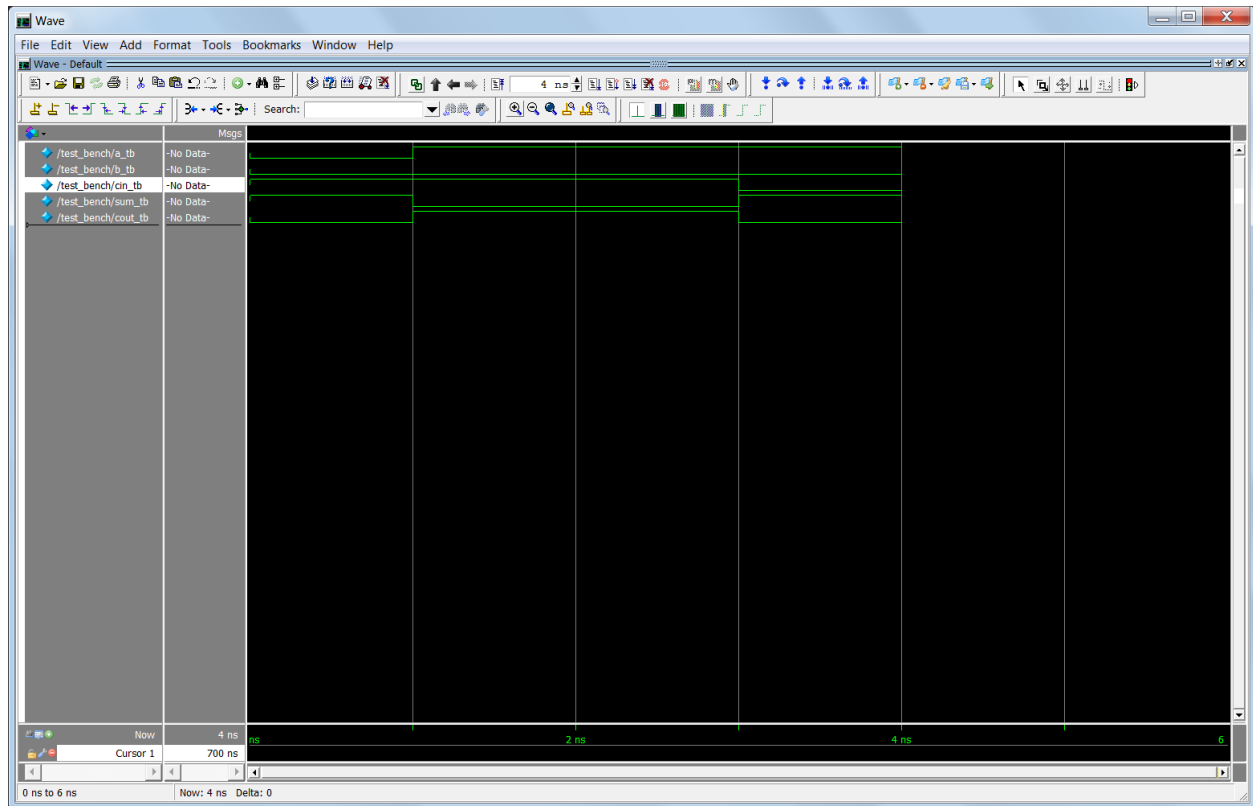
9. To setup the waveform window, right click on your module and choose **Add to > Wave > All items in region**.



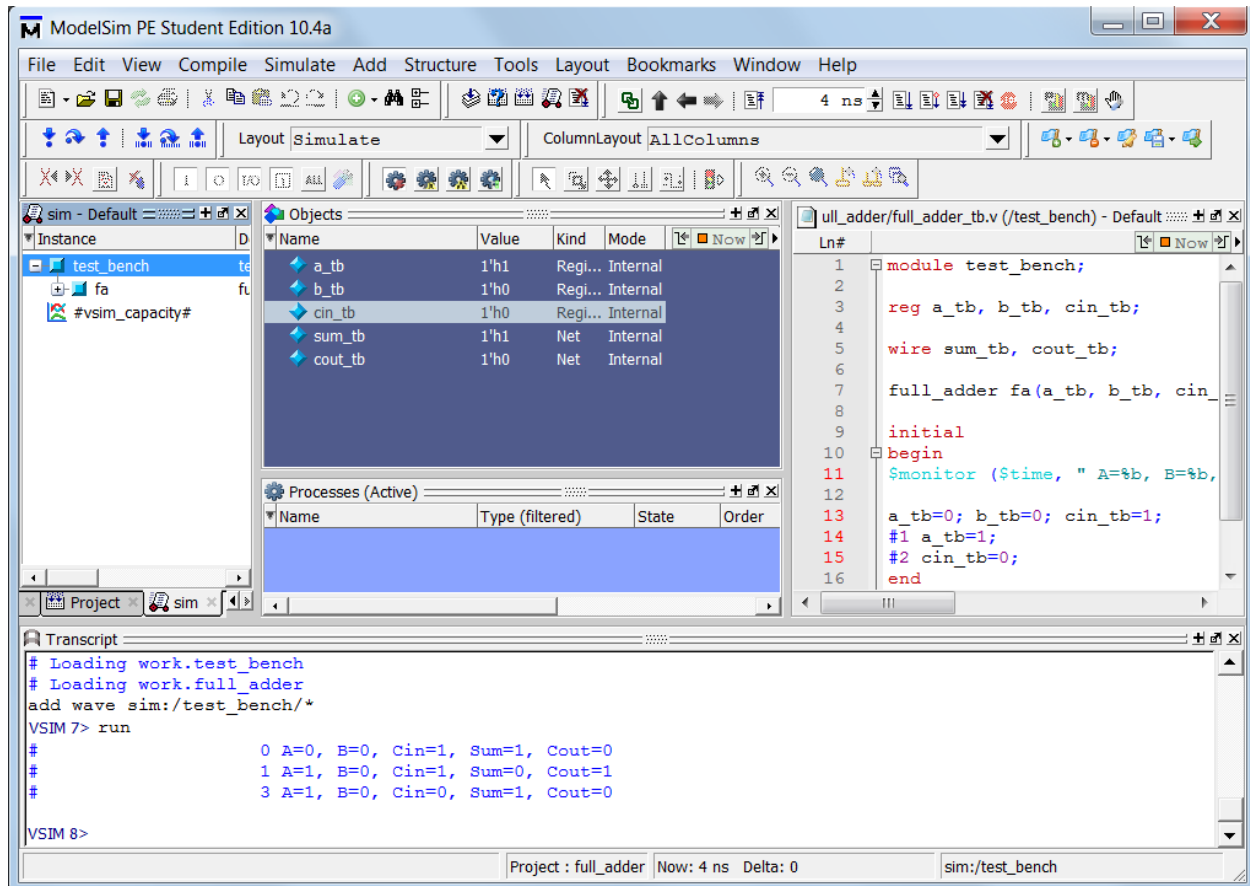
10. Here is the waveform window. On the toolbar, change the simulation time to 4ns, and run the simulation by clicking on the **Run** button.



11. Use the **Zoom In** button to check the details of the waveforms.



12. The outputs of the \$display function in the testbench would be printed in the **Transcript** command line window.



Appendix

Installing ModelSim Student Edition in Linux

ModelSim PE is a Windows-only application. To use ModelSim in Linux you can use Wine² which allows applications designed for Windows to run on Unix-like operating systems.

1. Install Wine using the **Software Center** or the following command:

```
sudo apt-get install wine
```

2. Obtain the .exe file for ModelSim Student Edition from the link given in the first page of this document.

3. Run the above file using Wine with the following command:

```
sudo wine modelsim-pe_student_edition.exe
```

The installation process will begin. Continue with the normal installation process as explained in steps 2 to 11 in previous section. After obtaining the license file, you have to save it to the top level installation directory for ModelSim PE Student Edition. This is the directory that contains the sub-directory 'wine32pe_edu'. It should look like this:

```
/home/username/.wine/drive_c/Modeltech_pe_edu_10.4a
```

4. You should be able to find ModelSim in the list of installed applications and use it!

Follow the instructions in this document to perform your first simulation using ModelSim.

² <https://www.winehq.org/>