

Getting Started with Modelsim

Modelsim Installation on Windows OS

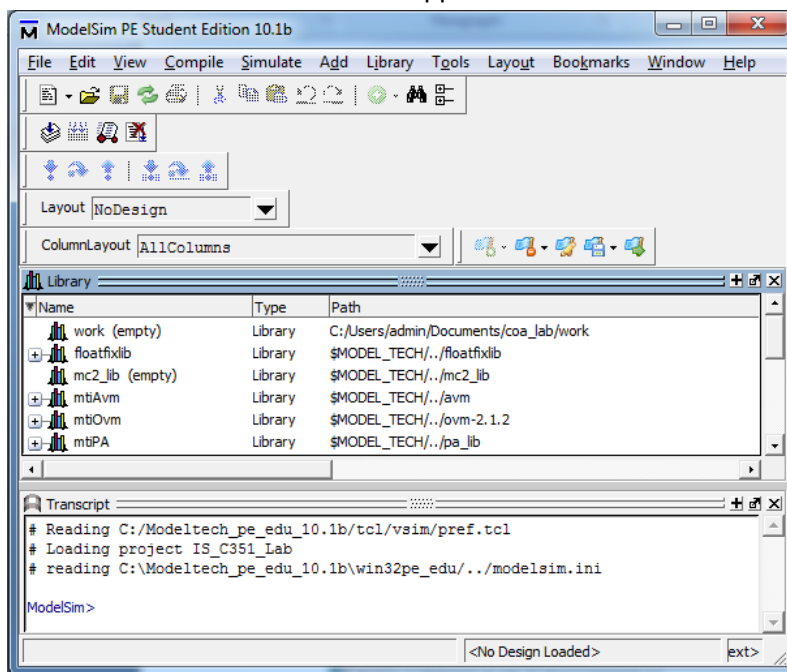
- 1) Login as administrator into the computer
- 2) Download Modelsim student edition from the website: <http://model.com/content/modelsim-pe-student-edition-hdl-simulation>
(Be sure to provide the correct e-mail ID as the license file will be sent to this e-mail)
- 3) Install the software in the system as administrator.
- 4) Check the e-mail for the mail from Mentor Graphics Inc. Make sure you check the spam folder before registering again.
Download the attachment “student_license.dat” file and save it in the Modelsim installation directory (Default – C:\Modeltech_pe_edu_10.1b)
- 5) Run the software. **The software runs successfully only the admin mode.**

Initial Setup

- 1) Go to **Documents** folder and create “**coa_lab**” folder; The “**coa_lab**” folder will contain all the lab files.
- 2) Create a subfolder called “**Lab-1**”; All the files created for “**Lab-1**” will be placed in this folder.
- 3) In future, create one folder for each lab. Ex: **Lab-2**, **Lab-3** etc.

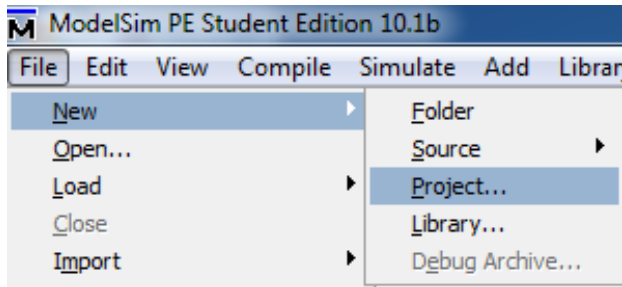
Create Project

- 1) Start Modelsim. The initial screen appears as below.



} **Console / Transcript Window**

- 2) Create a new project by selecting File → New → Project



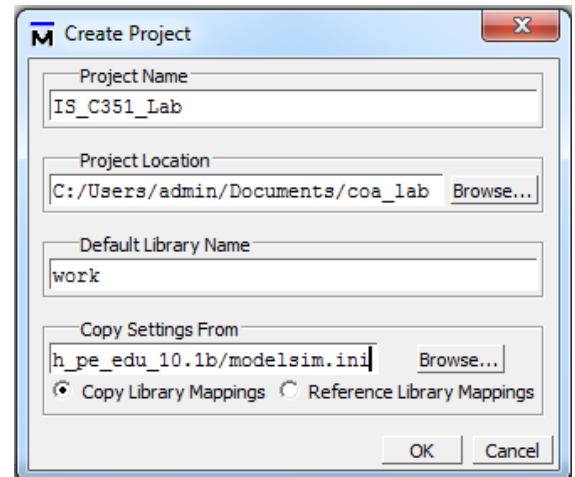
In the new project window that pops up, use the following settings to create a new project.

Project Name: IS_C351_Lab

Project Location: C:/Users/admin/Documents/coa_lab

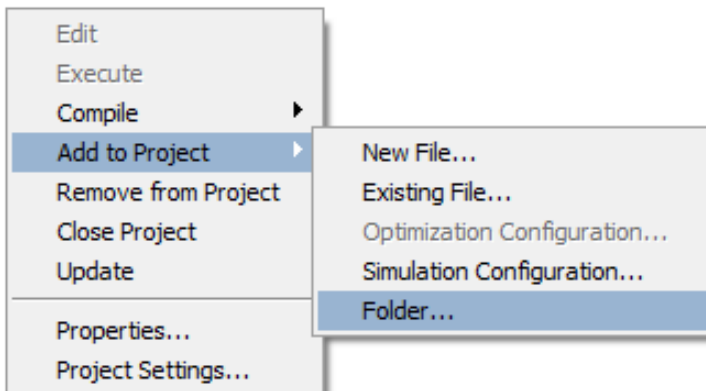
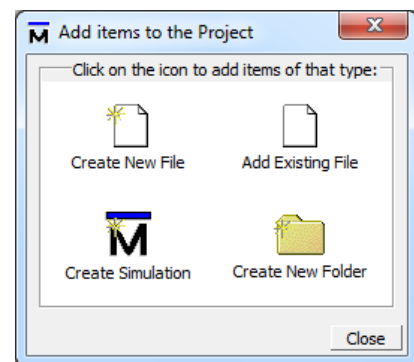
Leave the other settings as they are!

Click OK to create a new project



- 3) Create organization for the labs

A new window (Add items to the Project) shows up. You can close this window immediately.



Right click in the project pane to get options window.

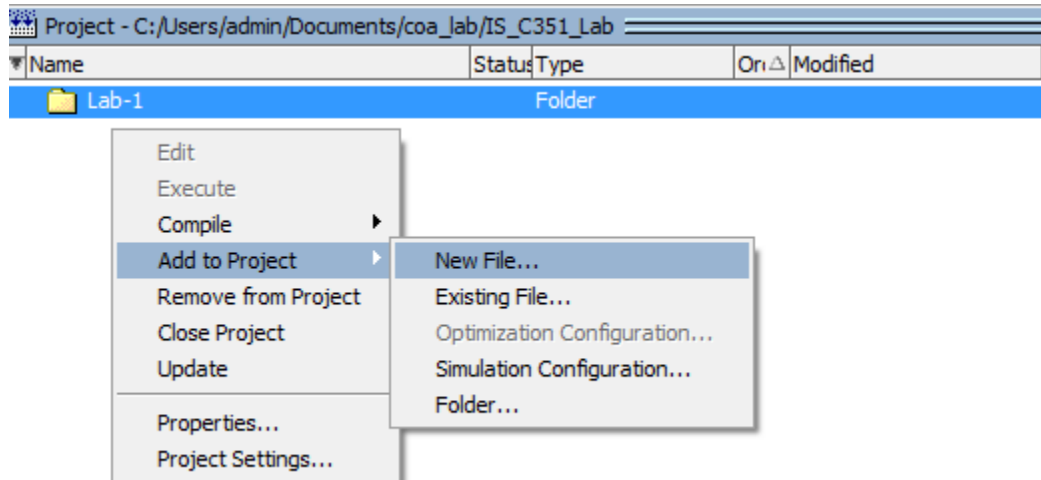
From the available options, select Folder to add virtual folder to the project. In the new window, give "Lab-1" as the name for the virtual folder. The folder is being called

virtual folder because this folder is only visible inside Modelsim project. It is not visible in the Windows OS.

In the future, you can create one virtual folder for each lab to organize your files.

4) Create new verilog file

Right click in the project pane.

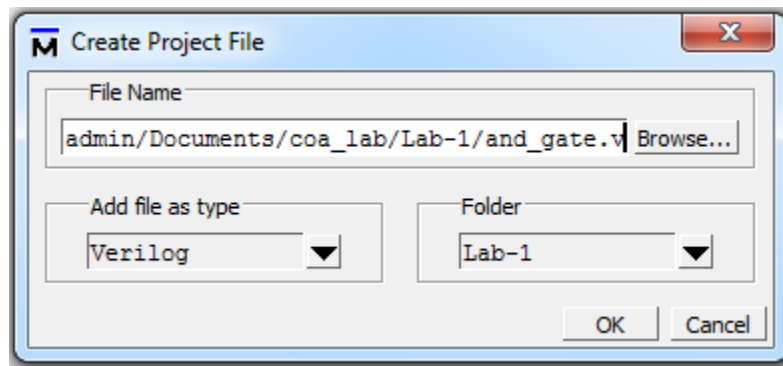


Select "New File" option from the menu. In the new menu that pops up, enter the following information.

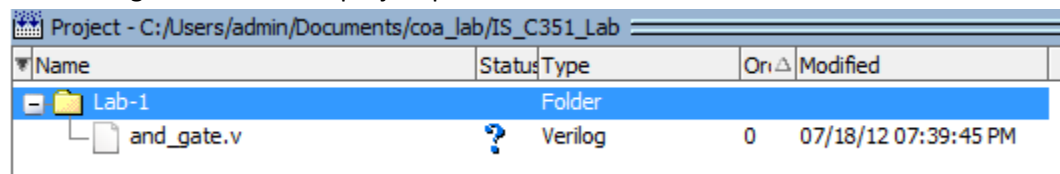
File Name: C:/Users/admin/Documents/coa_lab/Lab-1/and_gate.v

Add File as Type: Verilog

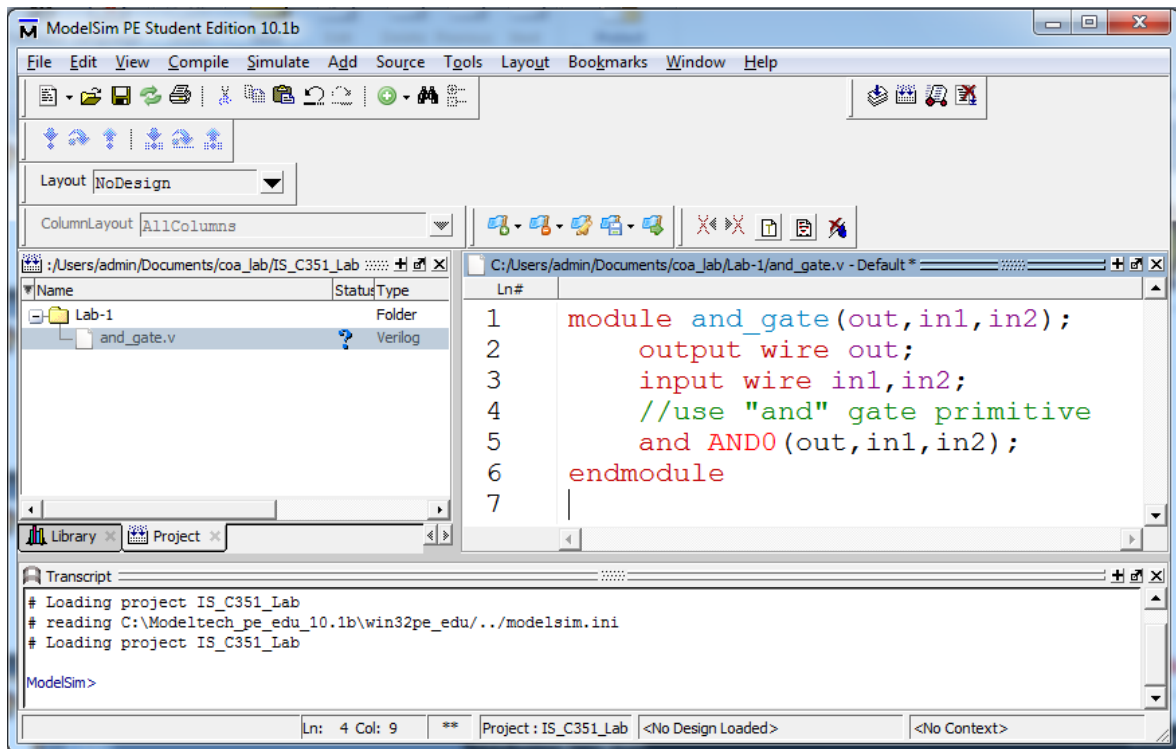
Folder: Lab-1



A new file gets linked in the project pane.

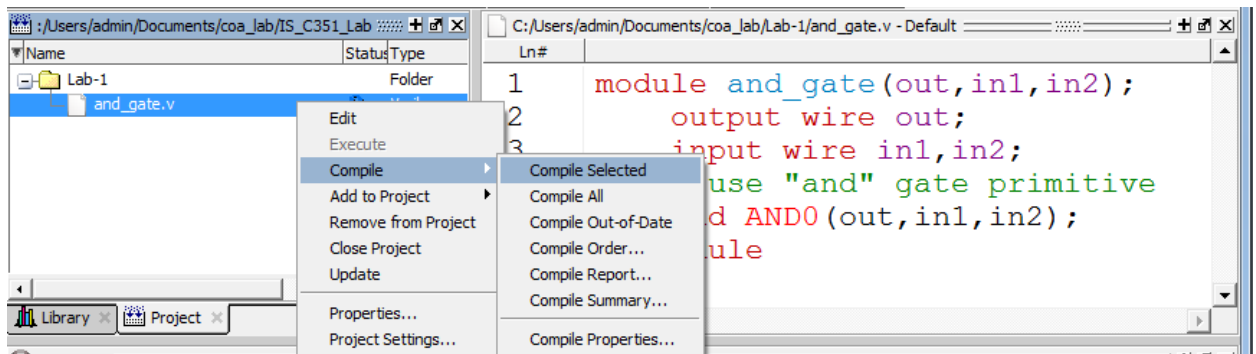


- 5) Edit the file “and_gate.v” by double-clicking on the file name. Enter the verilog code for the **and gate** module.



Save the file using Ctrl+S key combination.

- 6) Compile the verilog file

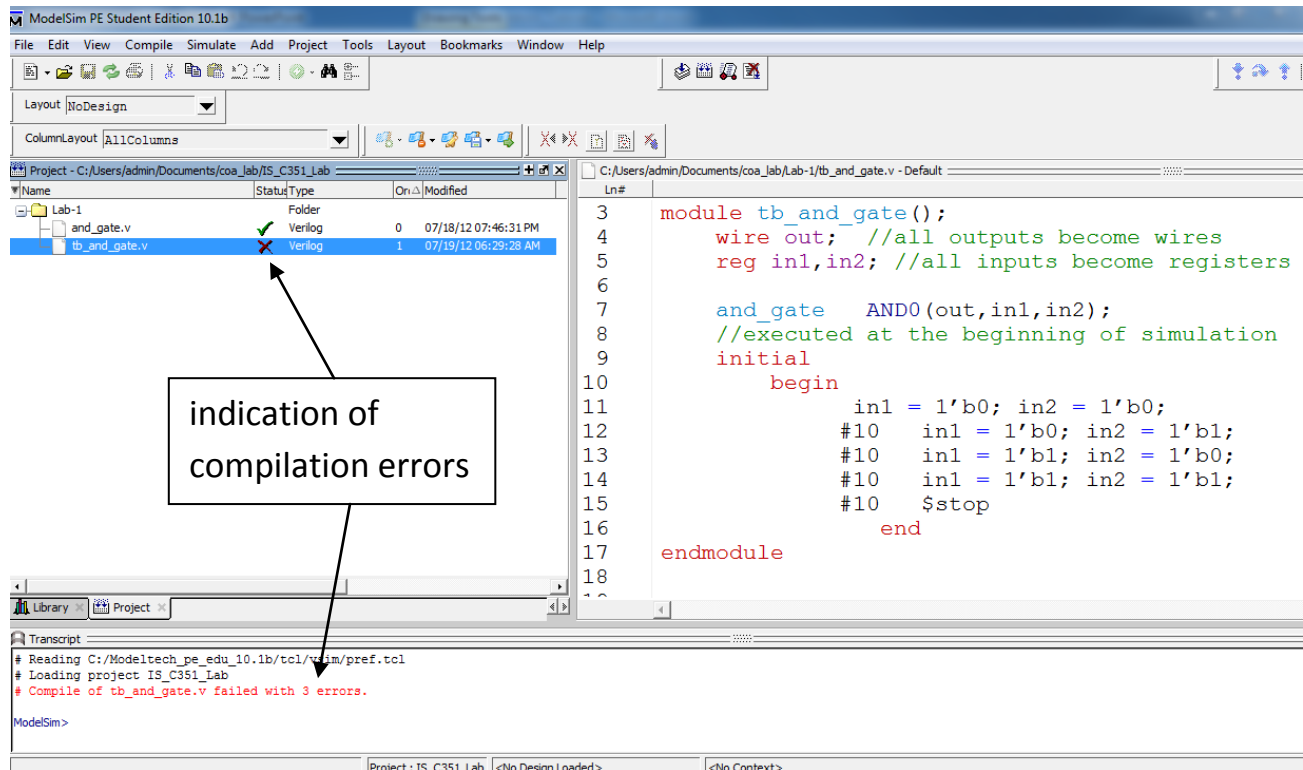


After successful compilation, the status of the file changes from ? to ✓. The transcript window shows the message –

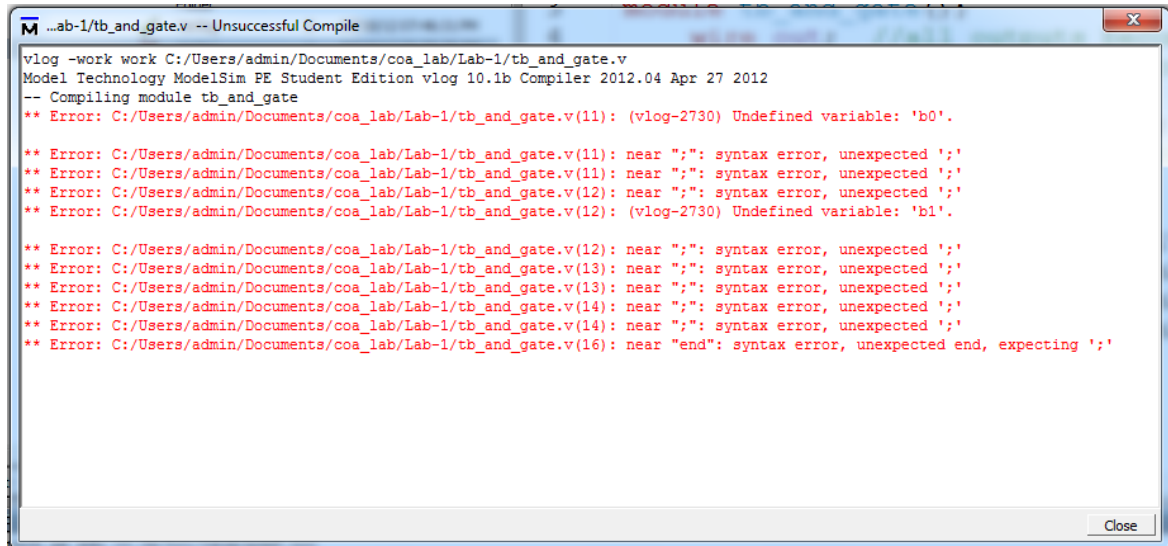
Compile of and_gate.v was successful.

If the verilog file contains any errors, the errors are listed in the transcript window.

7) Create the testbench by repeating the steps #4-6 for “tb_and_gate.v” file.



Click on the error message in the transcript / console window to get a detailed list of errors.

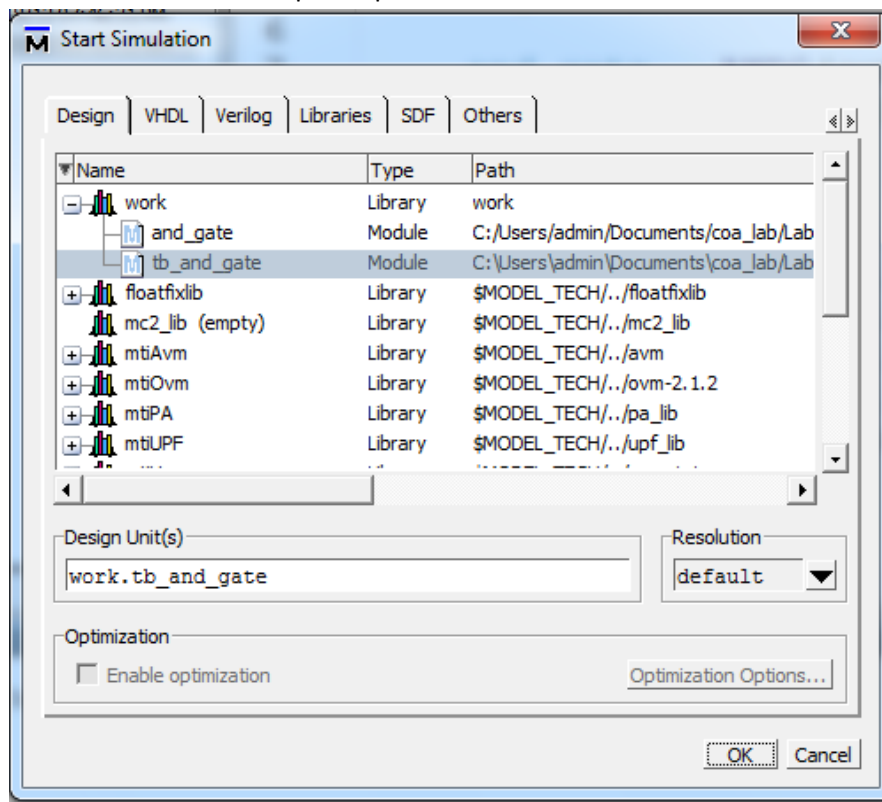


Correct the errors in the file and compile the verilog file again.

You can also compile the file by typing the command
Modelsim>vlog Lab-1/tb_and_gate.v
in the console window

```
Transcript
ModelSim> vlog Lab-1/tb_and_gate.v
# Model Technology ModelSim PE Student Edition vlog 10.1b Compiler 2012.04 Apr 27 2012
# -- Compiling module tb_and_gate
#
# Top level modules:
#     tb_and_gate
ModelSim>
```

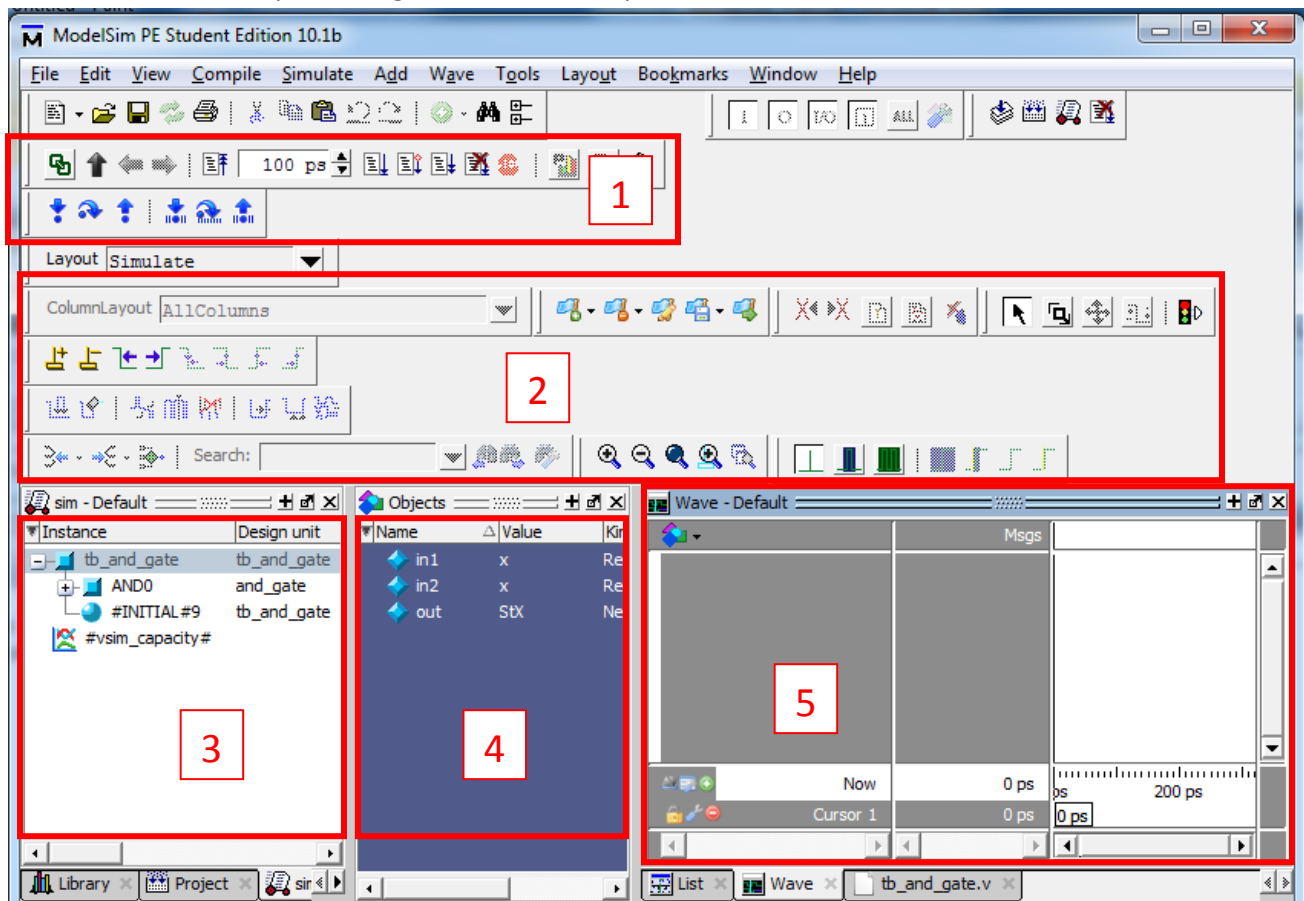
- 8) Simulate the Testbench
Start the simulation by selecting Simulate → Start Simulation
A new “Start Simulation” window opens up.



Expand the **work** library and select **tb_and_gate** module for simulation.

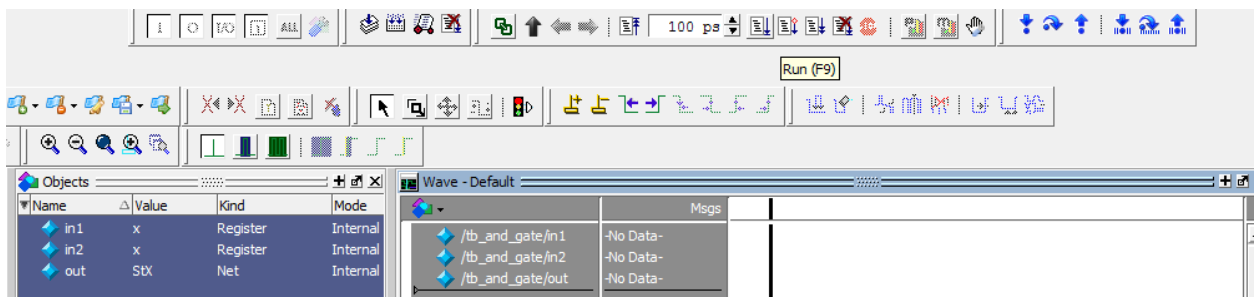
You can also start the simulation by typing the command
Modelsim>vsim -gui tb_and_gate
in the console window

The window layout changes to simulation layout.



- 1 – Simulation options
- 2 – Waveform window options
- 3 – Instances window
- 4 – Variables / Signals window
- 5 – Waveform window

To add signals to the waveform window, select the signals from signals window (window #4), drag and drop them into the waveform window (window #5).



Run the simulation by pressing the **F9** key. Waveforms appear in the waveform window. The



button can be used to zoom-in to the waveforms.

