

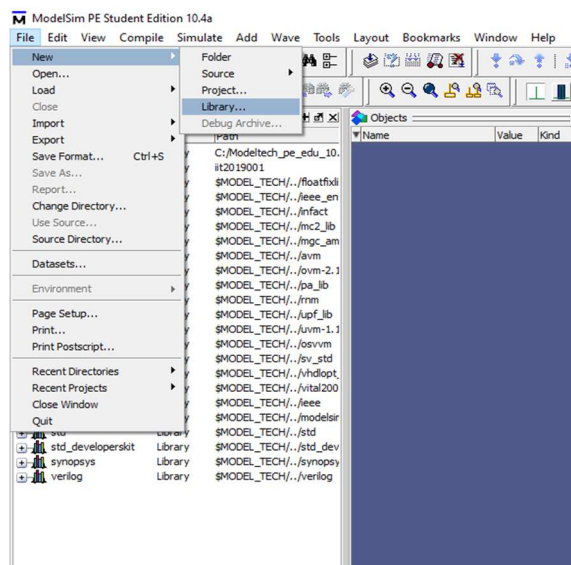
## Getting Started with ModelSim

**Verilog Simulation in ModelSim:** In this tutorial we will simulate a logical AND Gate in ModelSim. ModelSim is a package in Mentor Graphics and is used for logic simulation of HDLs. Steps to be followed:

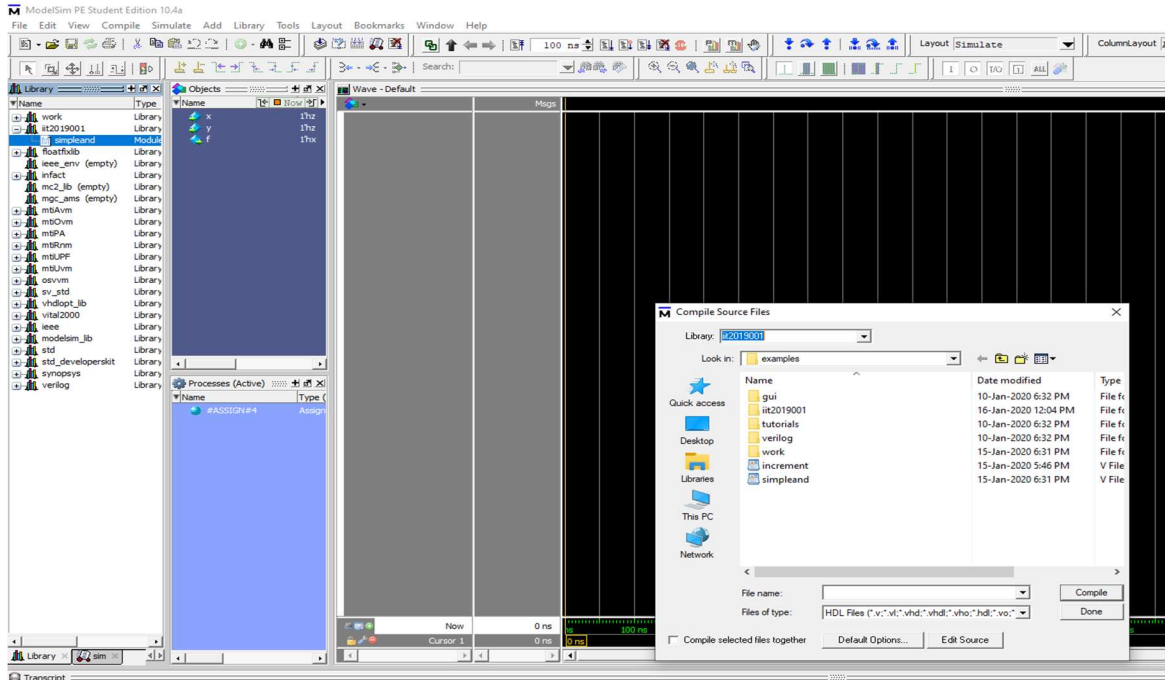
1. The Verilog code used for this tutorial can be downloaded here, <ftp://172.20.54.132> > Desktop > [simpleand.v](#), or you can use your favorite text editor (such as gedit , vim, etc.) to make it.

```
module simpleand(x, y, f);
input x,y;
output f;
assign f = x & y ;
endmodule
```

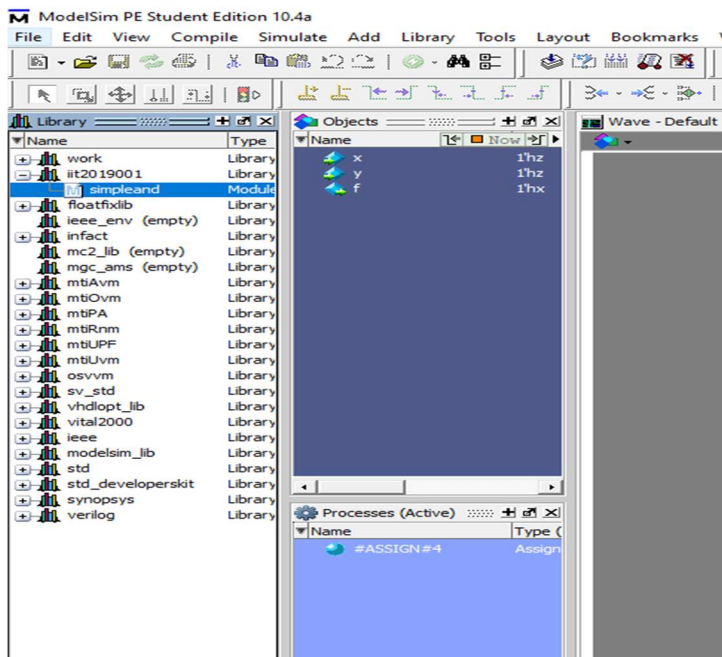
2. Open ModelSim by typing in the terminal `./modelsim.sh` in your home directory. (You should have the `modelsim.sh` script file in your home directory that initializes the environment variables and runs ModelSim. You can get it <ftp://172.20.54.132> > Desktop > [modelsim.sh](#) if you don't already have it.)
3. Click on Compile > New > Library to create a new library, and name it as your roll no. '**iit2019XXX**'. You will see the library appear in the workspace. You are supposed to use this library in every lab for keeping your compiled code.



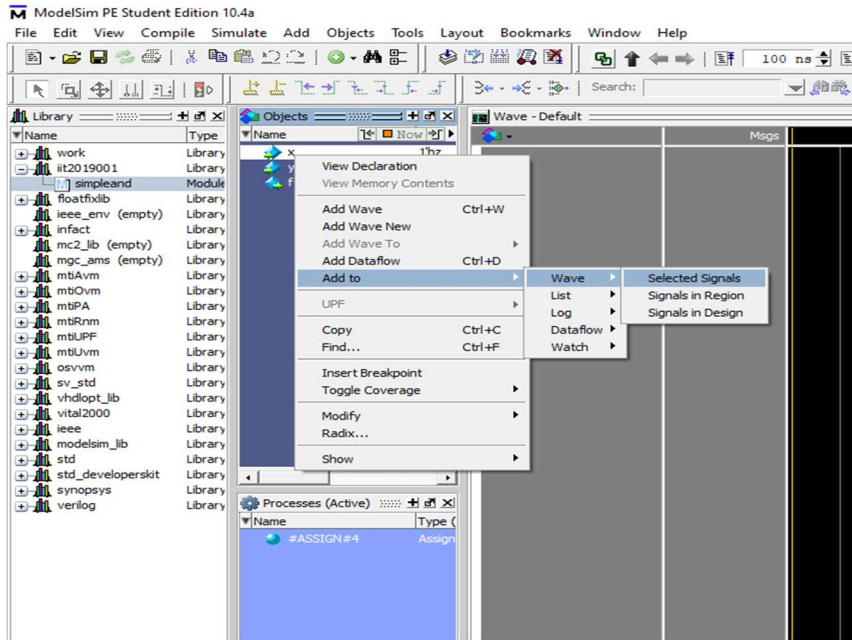
4. Click on the Compile button on the Compile toolbar around the top of the window. Choose '**iit2019XXX**' as the library from drop-down menu, and then select the file **simpleand.v** by browsing to its location and compile it.



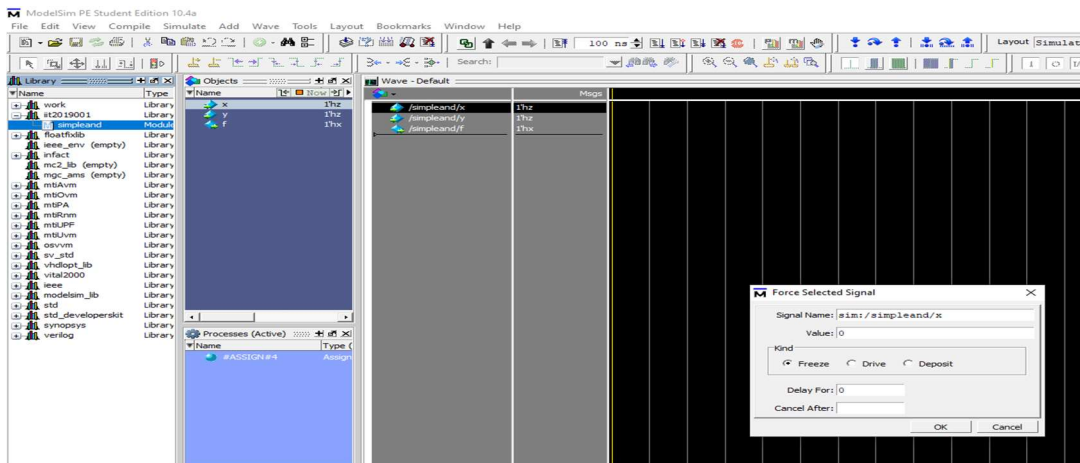
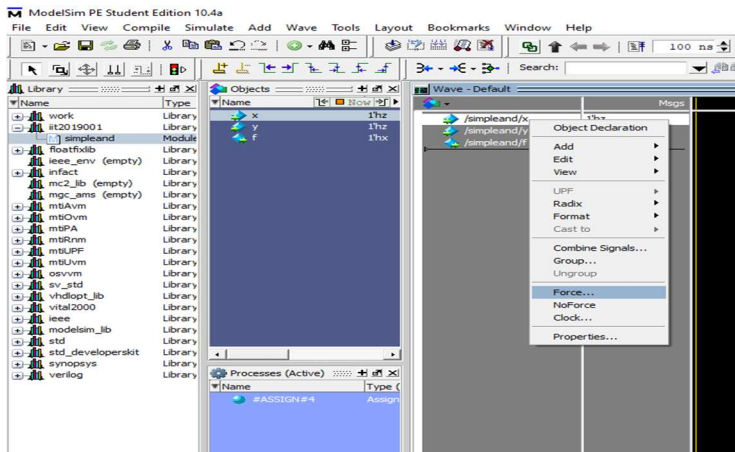
5. You should see the file, **simpleand**, under the **iit2019XXX** library in the Workspace after successful compilation. Double-click this file. This should take you into Simulation mode, and also bring up the Objects window. You can click on View > Objects in the menu bar to bring up the objects window if you accidentally close it later.



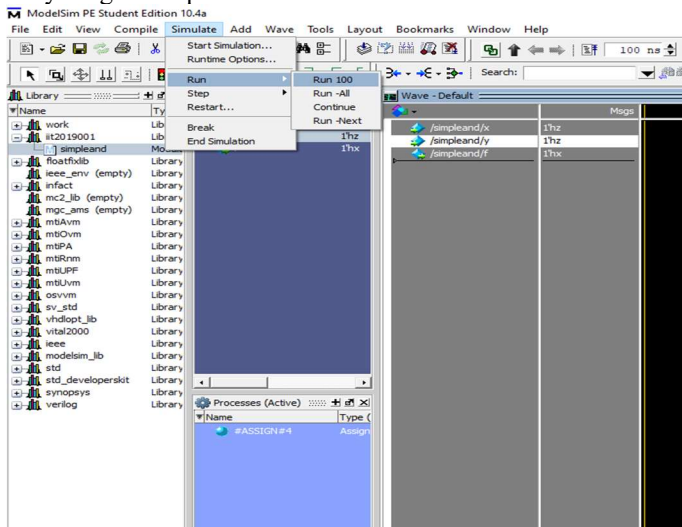
6. Right-click on each object in the Objects window, and add it to the wave by clicking Add to Wave > Selected Signals one by one. You will see all objects will be added to wave window.



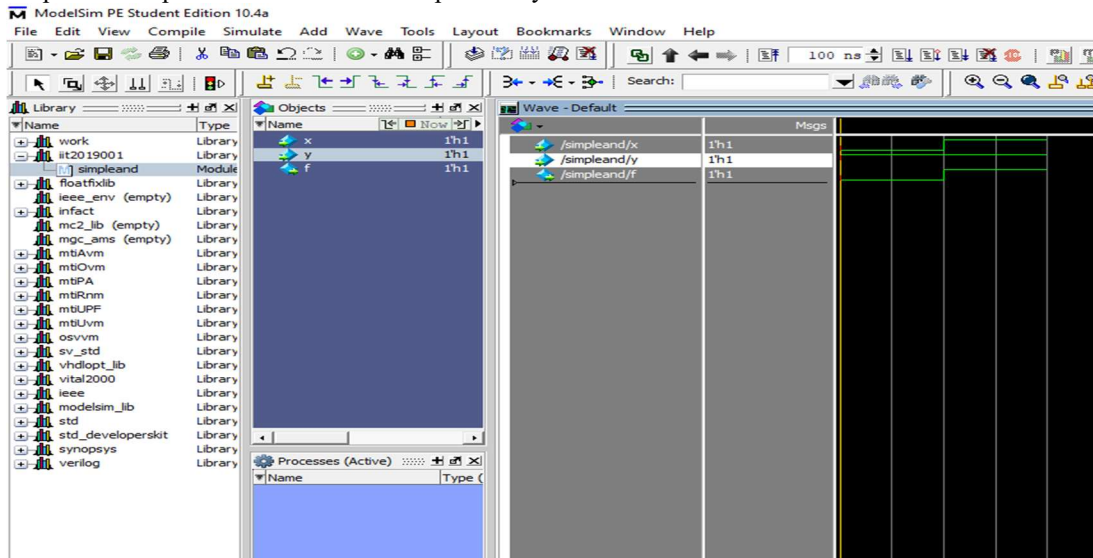
7. Add inputs by right-clicking it and clicking on Force > Value > 0 or 1 > OK for each input you have taken.



8. Create Wave in the wave window by selecting Simulate > Run > Run 100 and you will see the wave form for your given input for 100 ns. You can increase or decrease this time as well.



9. Repeat the steps 7 and 8 for different inputs and you will see the waveform in the wave window.



**Alternate way to initialize input from VSIM terminal in Transcript window:**

- 1- At the command line in the Transcript tab, type the following for one input:  
force x 0; force y 1; force cin 1; run 100 then press Enter to see the waveform.
- 2- At the command line in the Transcript tab, type the following for multiple input :

