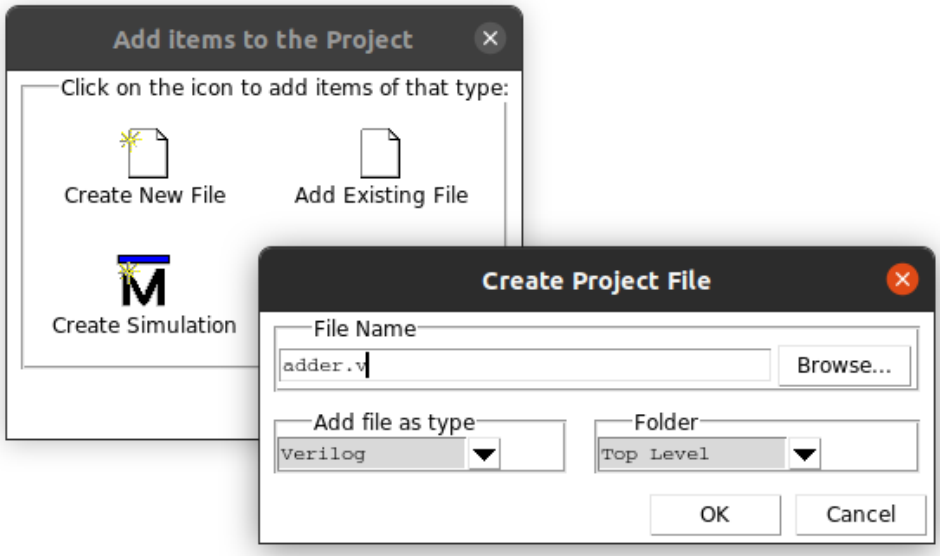
# ModelSim

Very good link for installation:
https://profile.iiita.ac.in/bibhas.ghoshal/COA_2020/Lab/ModelSim%20Linux%20installation.html

Steps:

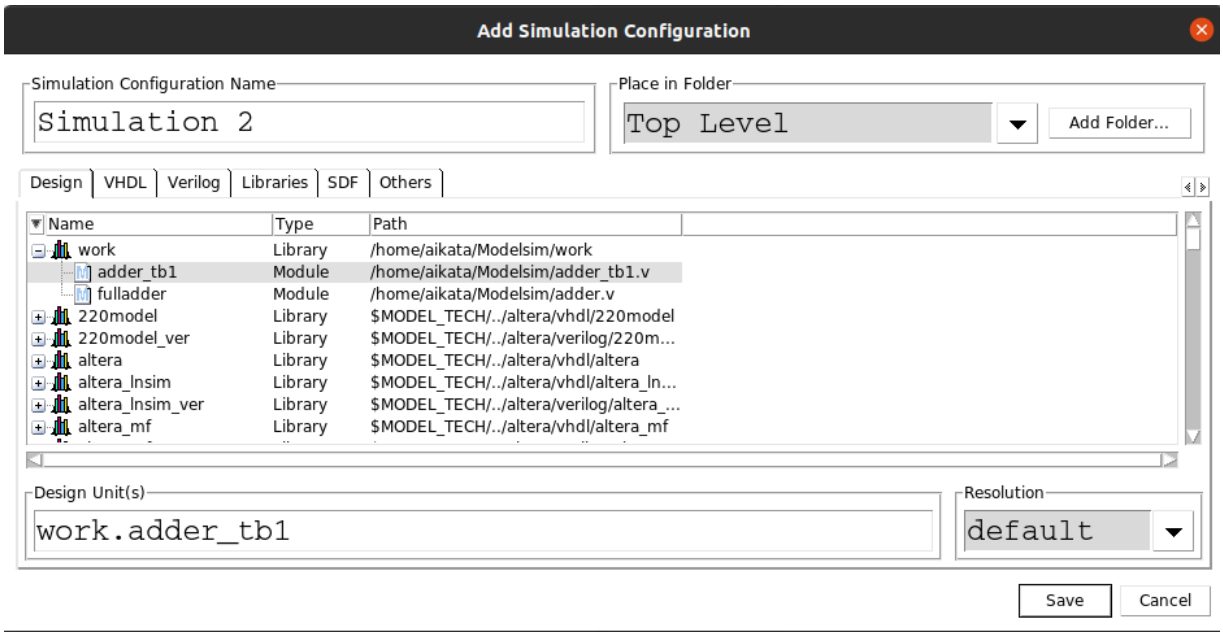
1. Create a project
2. Create a new verilog file (sample file attached as adder.v)



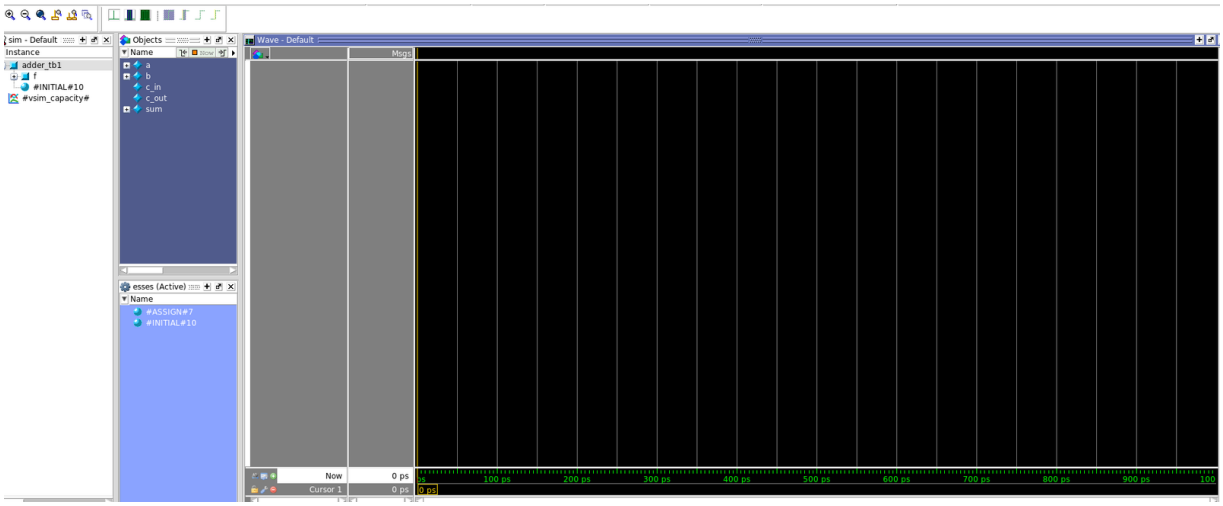

For zooming in I used ctrl+shift+'+'

3. Now you can add a testbench file (sample file adder_tb1.v)
4. Now go to compile and run compile all option. If everything is correct you will see green ticks in front of the files.
5. The go to the Simulate option and run simulate start. If nothing happens, then you need to point it to the simulation file. For this right click in the directory hierarchy display and go to option 'add to the project' . Here select Simulation

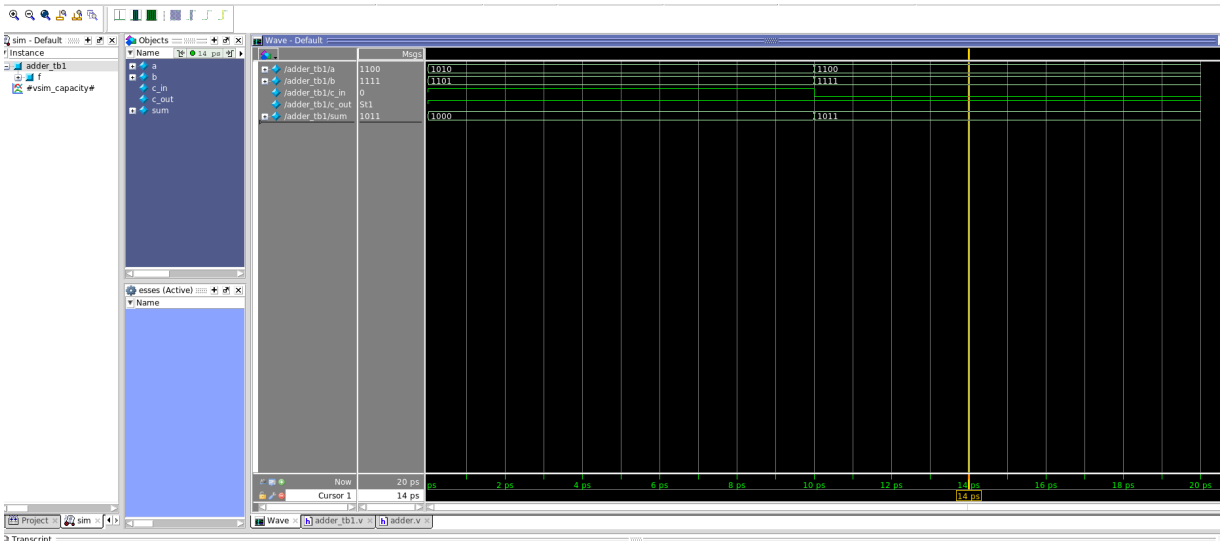
configuration and in this window point to the testbench file. Then save this. Then again start the simulation.




6. Now the simulation will start but an empty wave with objects on the right.



7. To this add the signals and then in the Simulate option go to Run>Run-all. Now you should be able to see the simulated waves.


sim - Default
Instance
adder_tb1
f
#vsim_capacity#
Objects
Name
a
b
c_in
c_out
sum
Wave - Default
Msgs
/adder_tb1/a
/adder_tb1/b
/adder_tb1/c_in
/adder_tb1/c_out
/adder_tb1/sum
1100
1111
0
St1
1011
1010
1101
1000
1100
1111
1011
Now
20 ps
Cursor 1
14 ps
2 ps
4 ps
6 ps
8 ps
10 ps
12 ps
14 ps
16 ps
18 ps
20 ps
Project
sim
Wave
adder_tb1.v
adder.v