

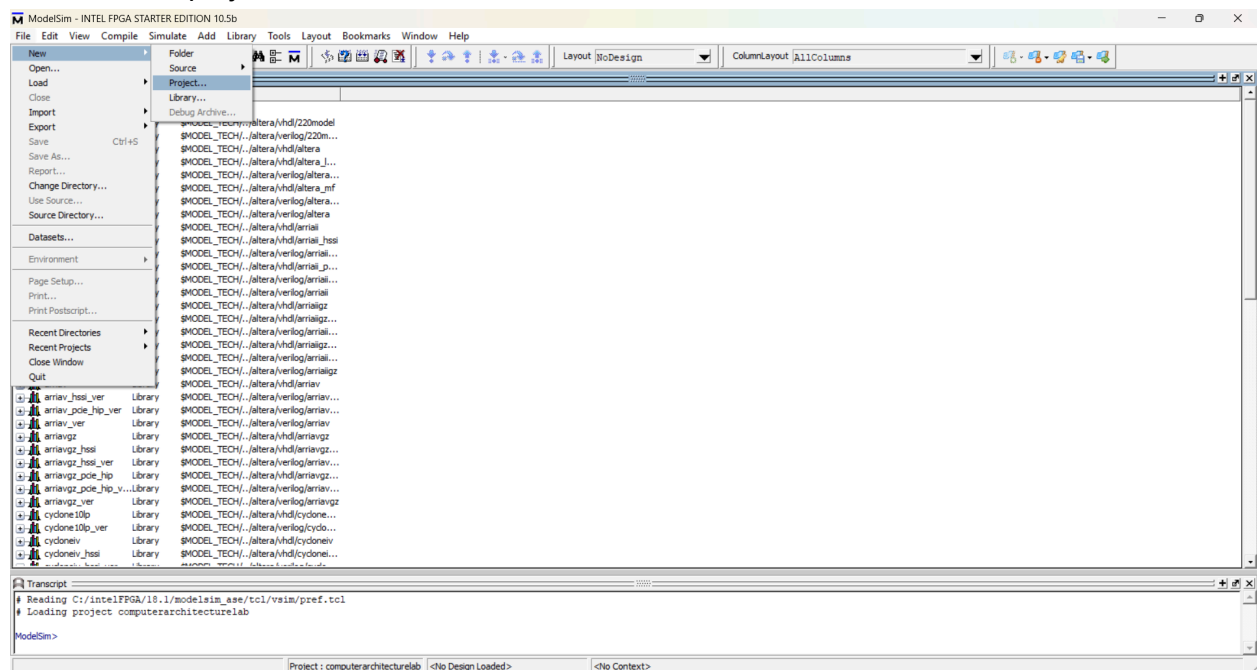
Download ModelSim from the link below

<https://www.intel.com/content/www/us/en/software-kit/750368/modelsim-intel-fpgas-standard-edition-software-version-18-1.html>

Install the software

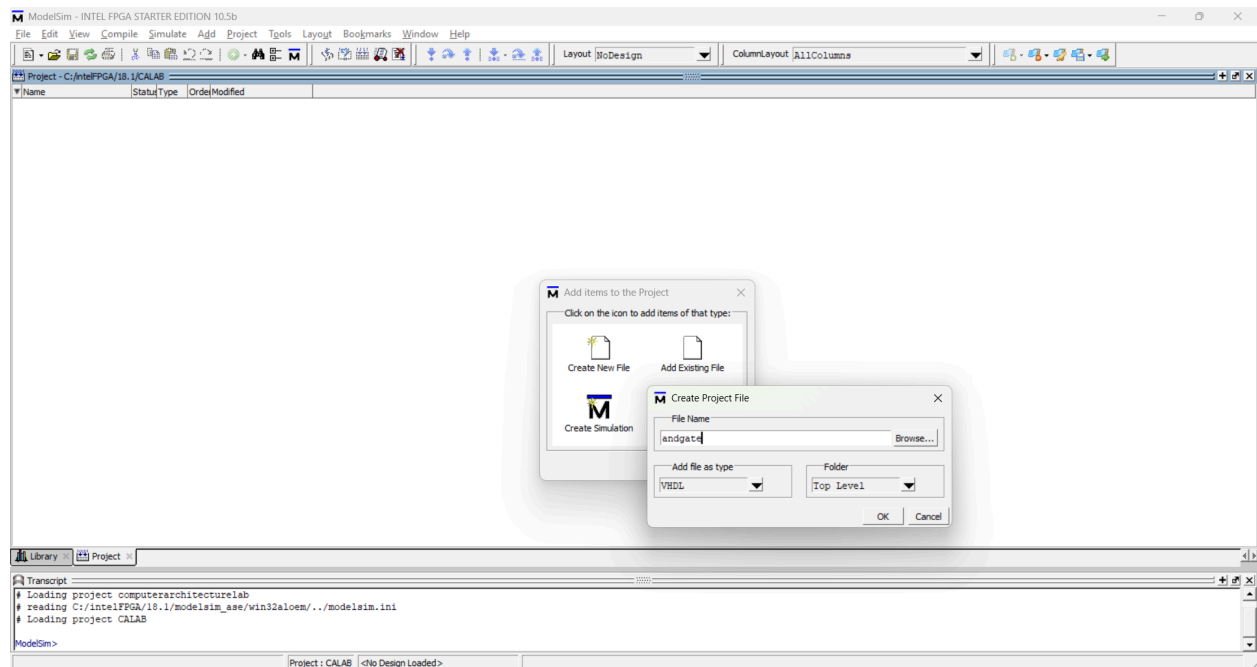
Open the software

Create a new project

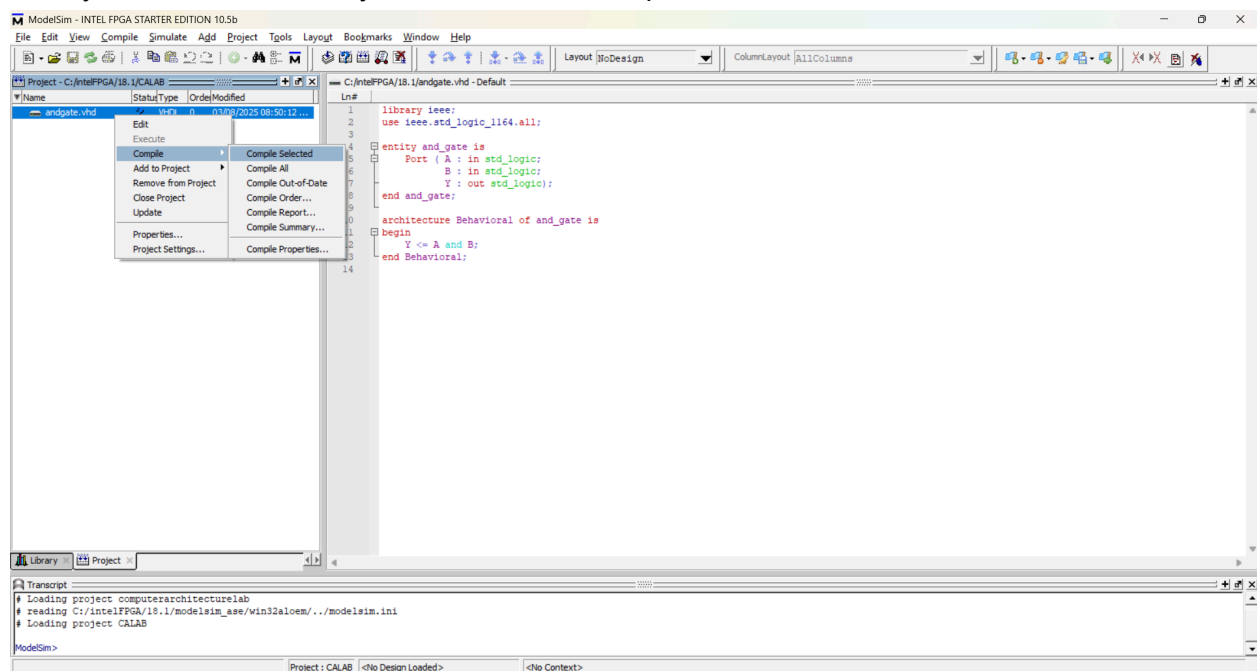




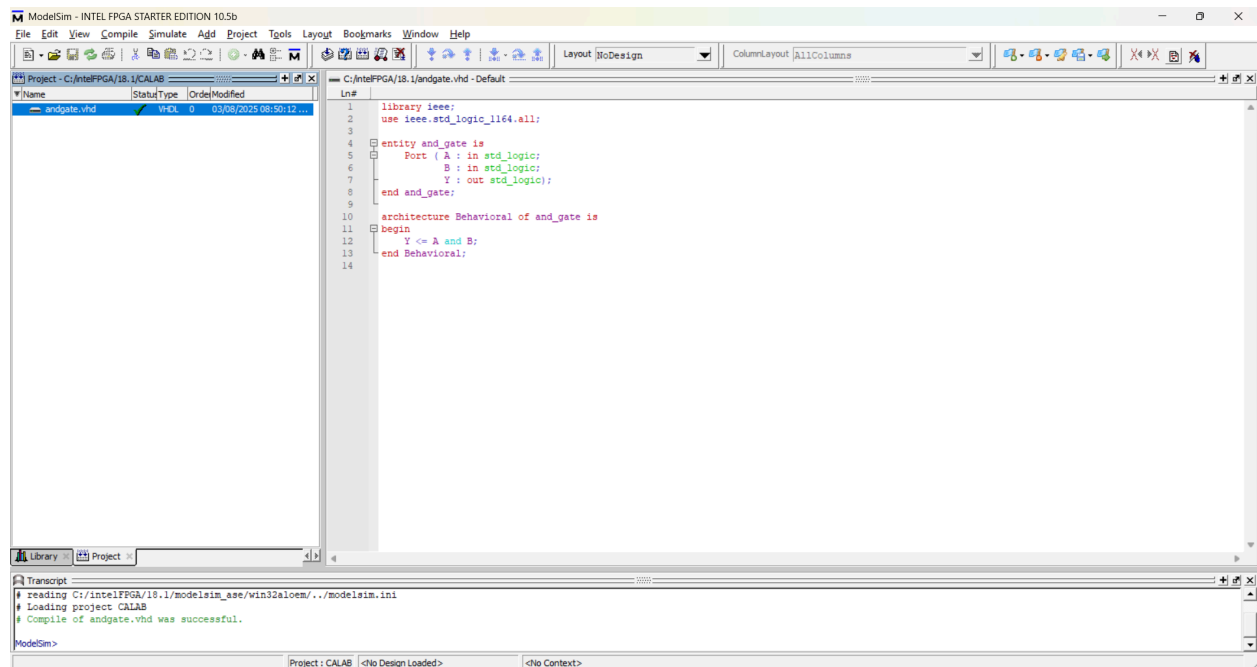
Create a new file(make sure the type is VHDL)



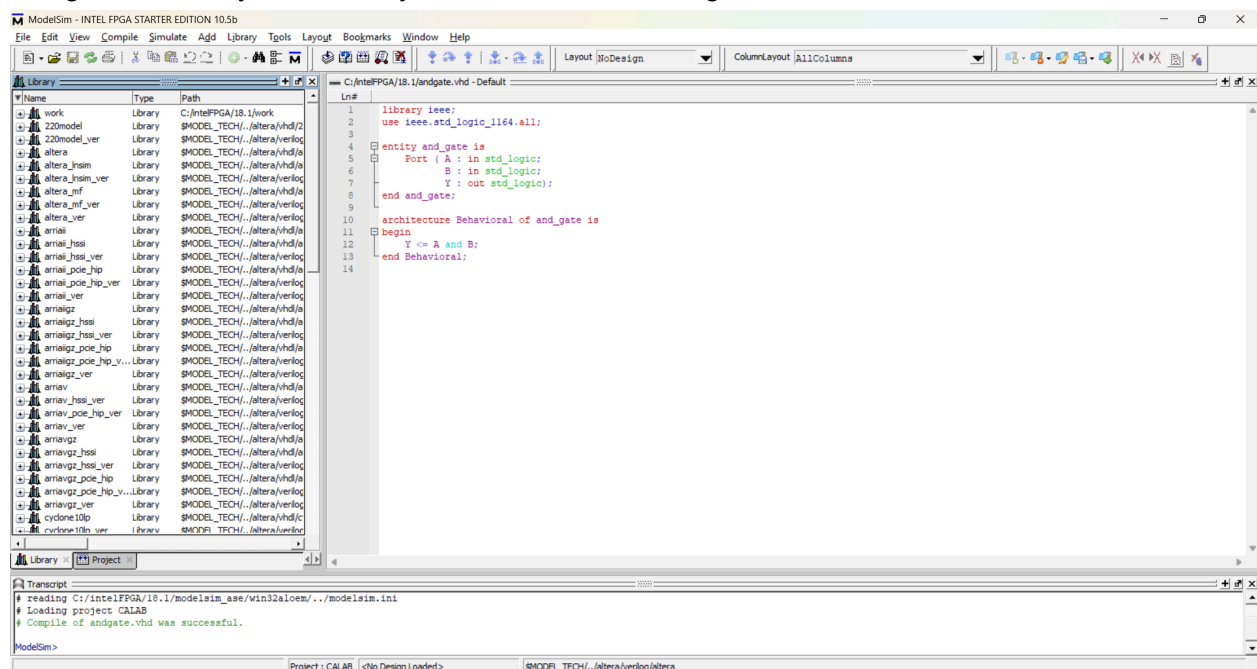
Write your code in the newly created file and compile the file



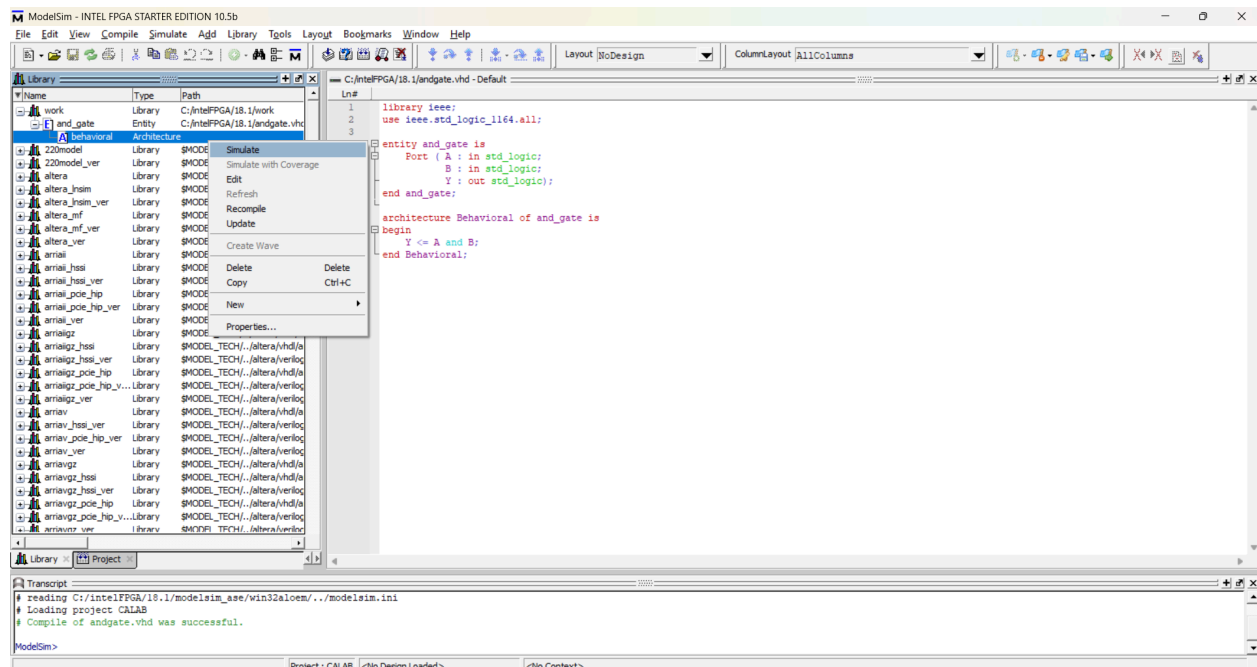
If the compilation is successful you will see the following message in your log



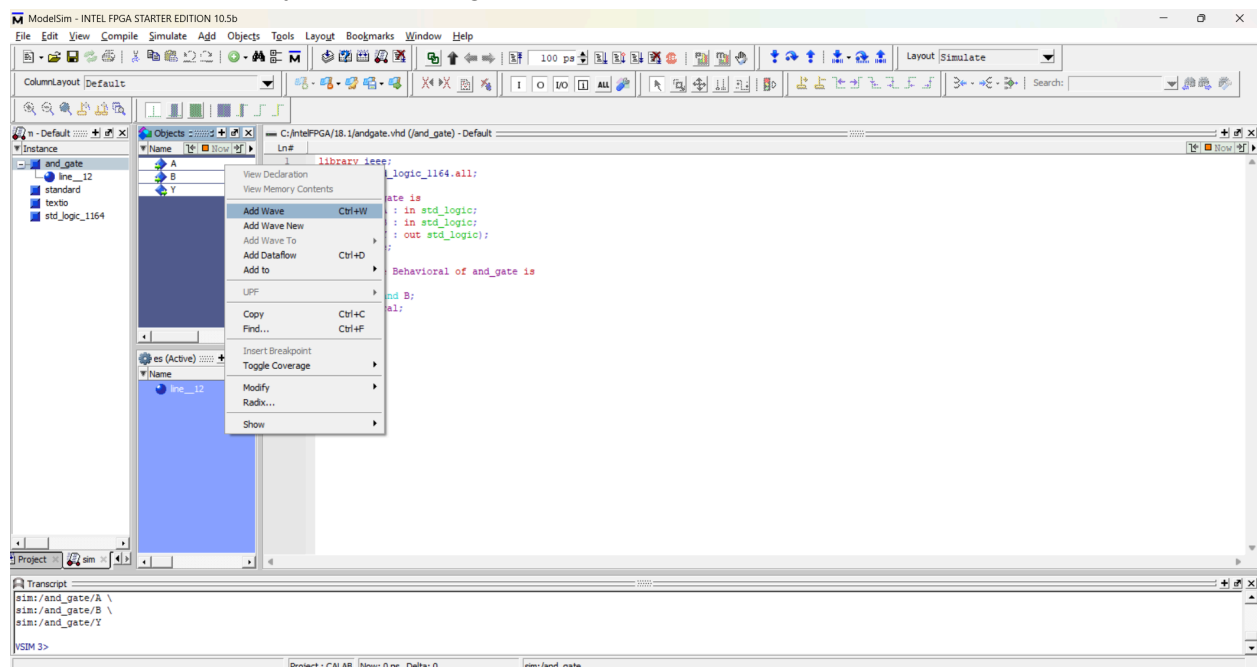
Navigate to Library menu and you will see the following screen



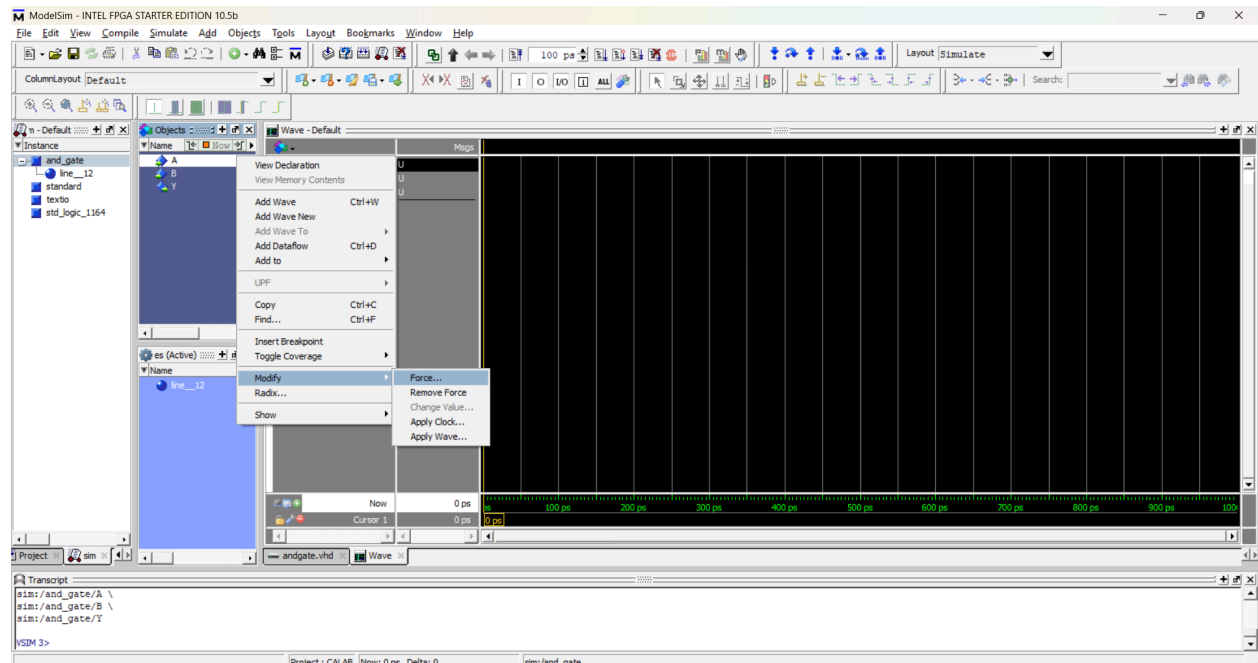
Open work tab and you will see the compiled block. Right click on the block and click simulate



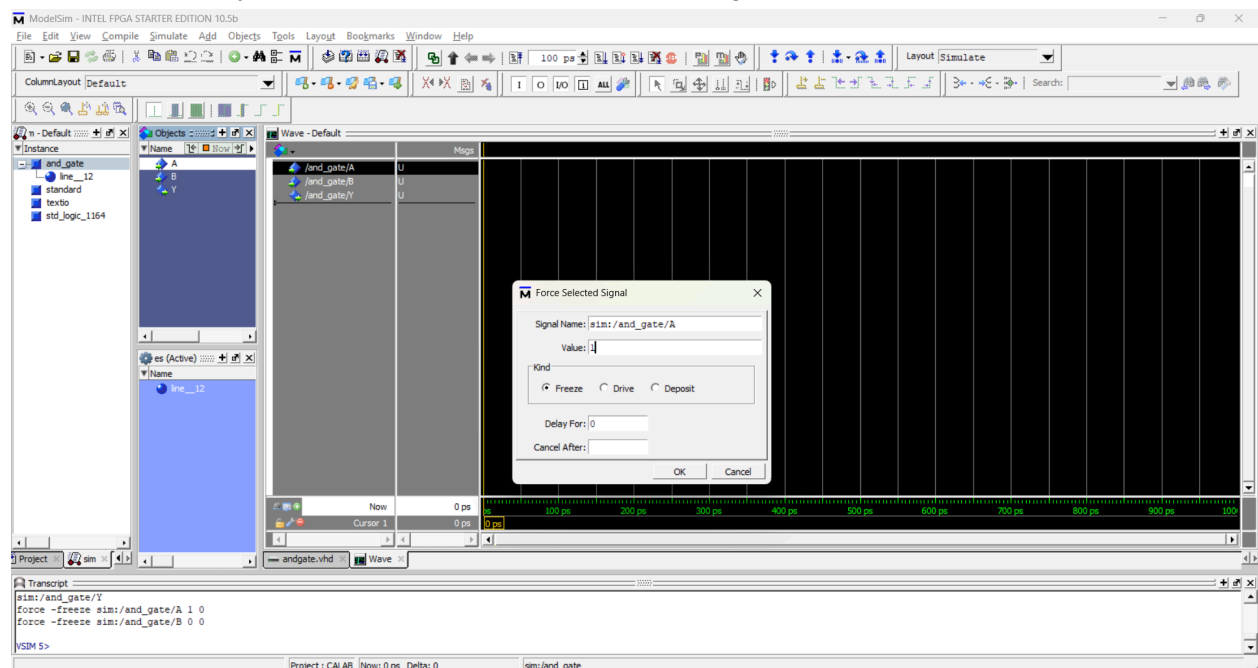
Select all the entities you see and right click and select Add Wave



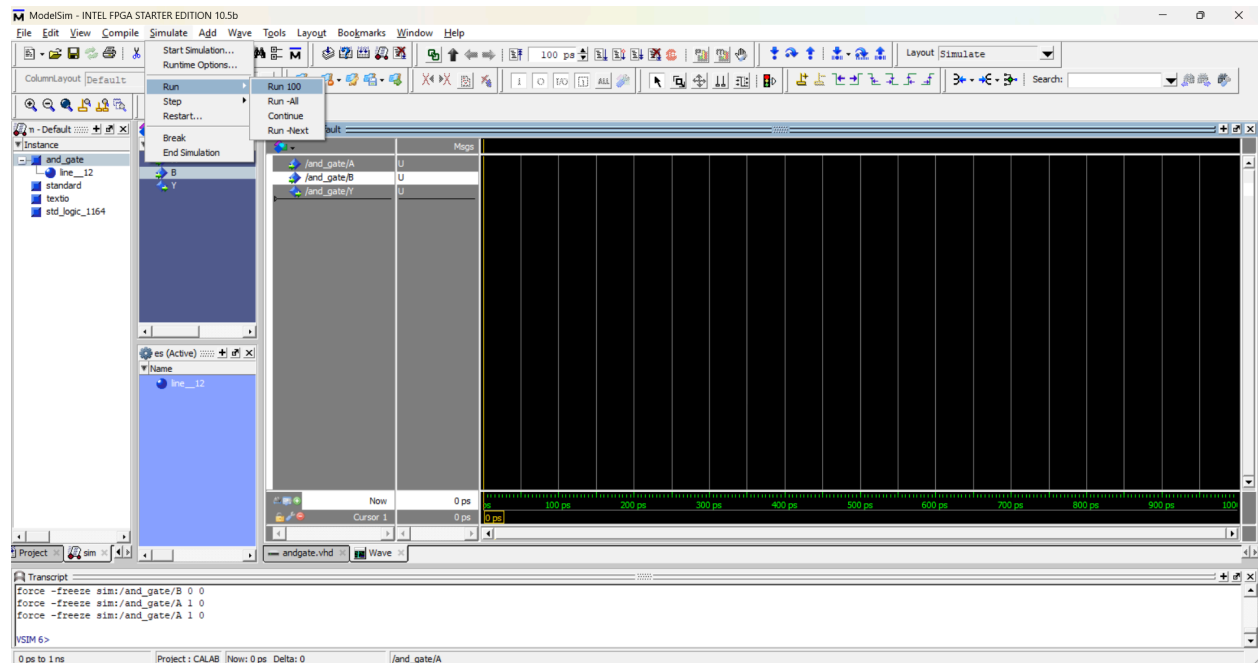
Update the value of each input. Select a particular input eg. A , right click on it and click Modify



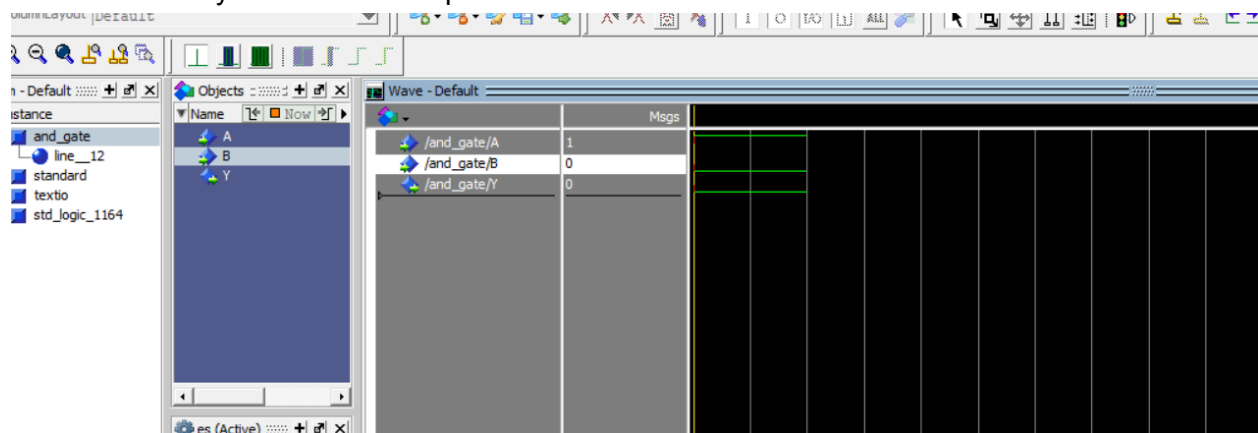
Update the value you want to provide. Let's provide a high value on input A ie.1



Select Simulate on the menu , select RUN and choose Run 100



The waveforms you see is the output



As we can see here, we passed A=1 and B=0 so the and gate output at C is 0