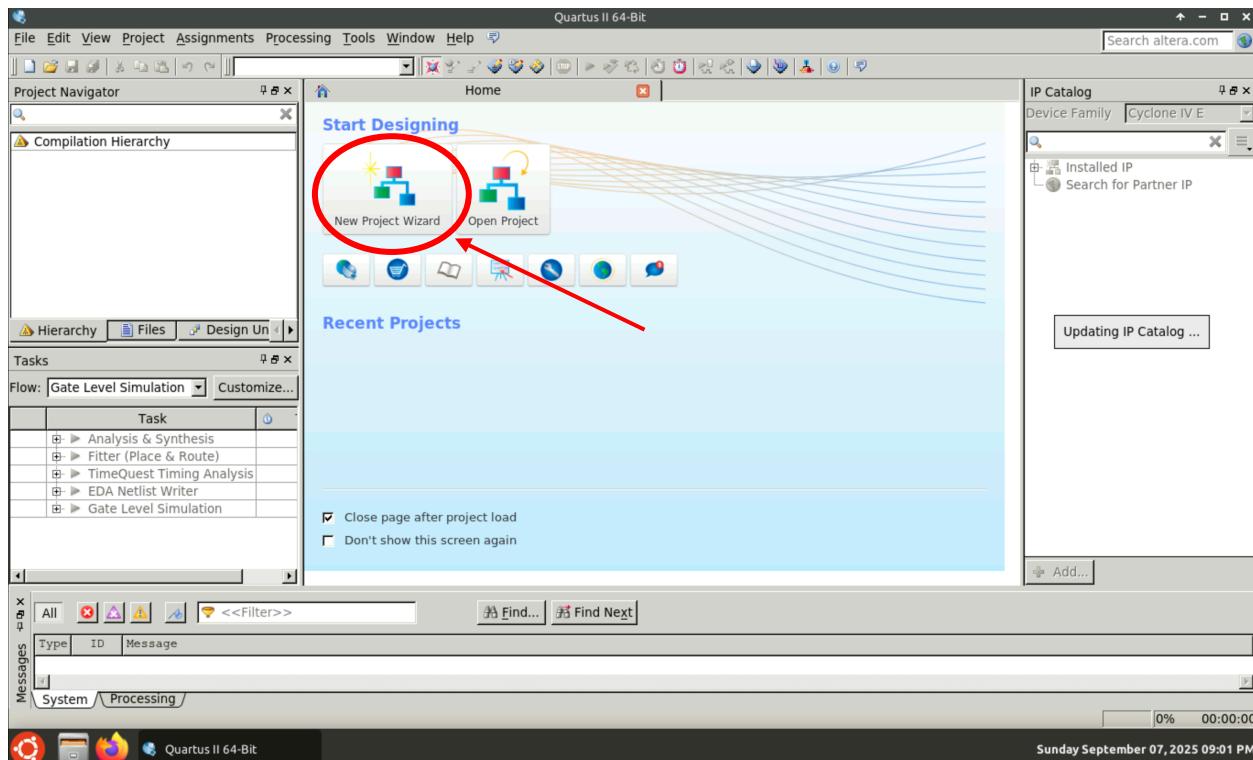


2. Launching Quartus

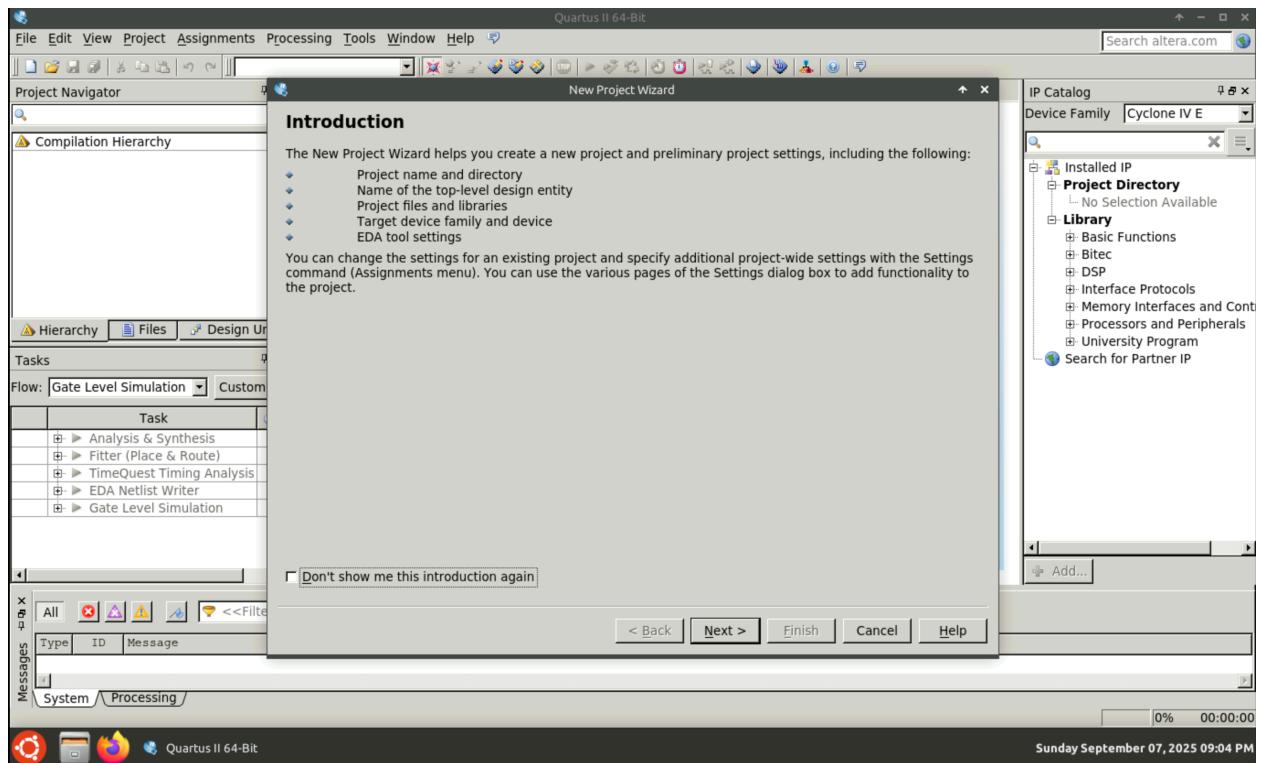
1. Pick one of the many PCs in the EE103 Lab and login with your SoE account that you created a password for in the first instruction set.
2. Click on the Quartus icon on the desktop, as shown below.



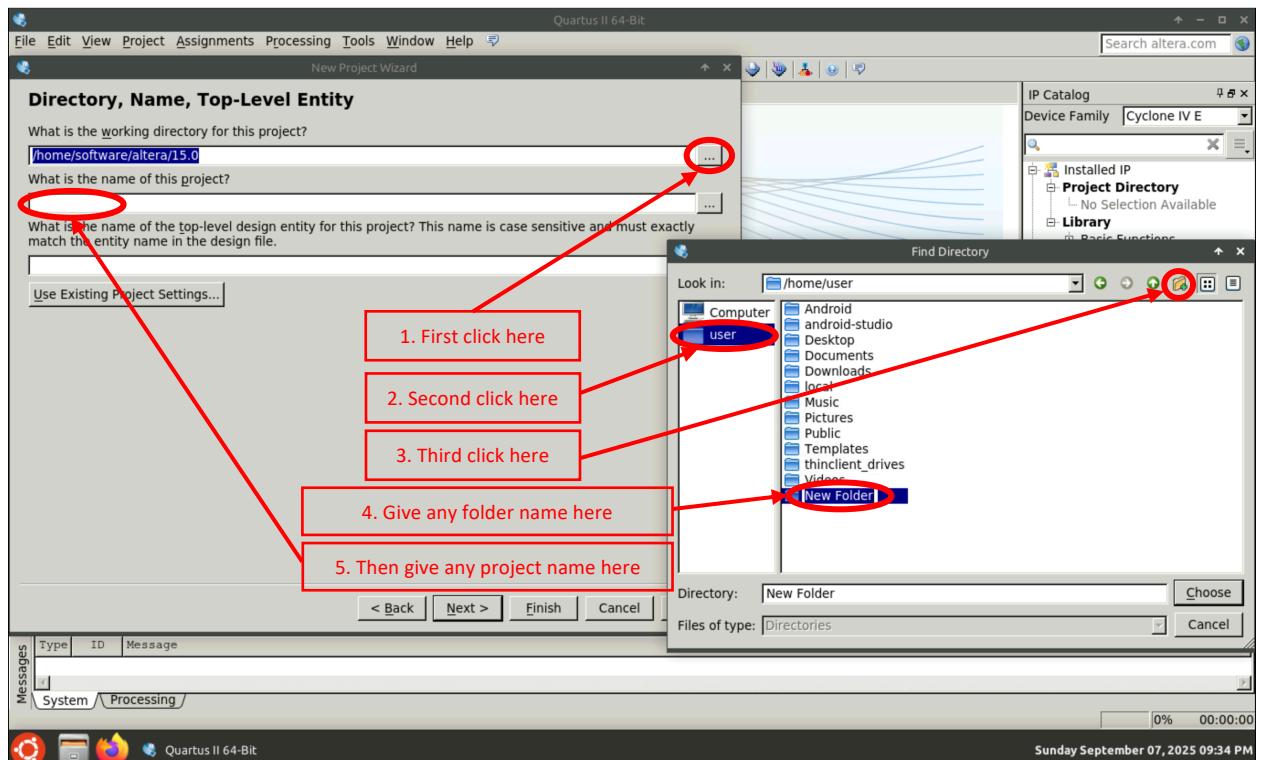
3. Once Quartus loads up, create a new project by clicking on “New Project Wizard”.



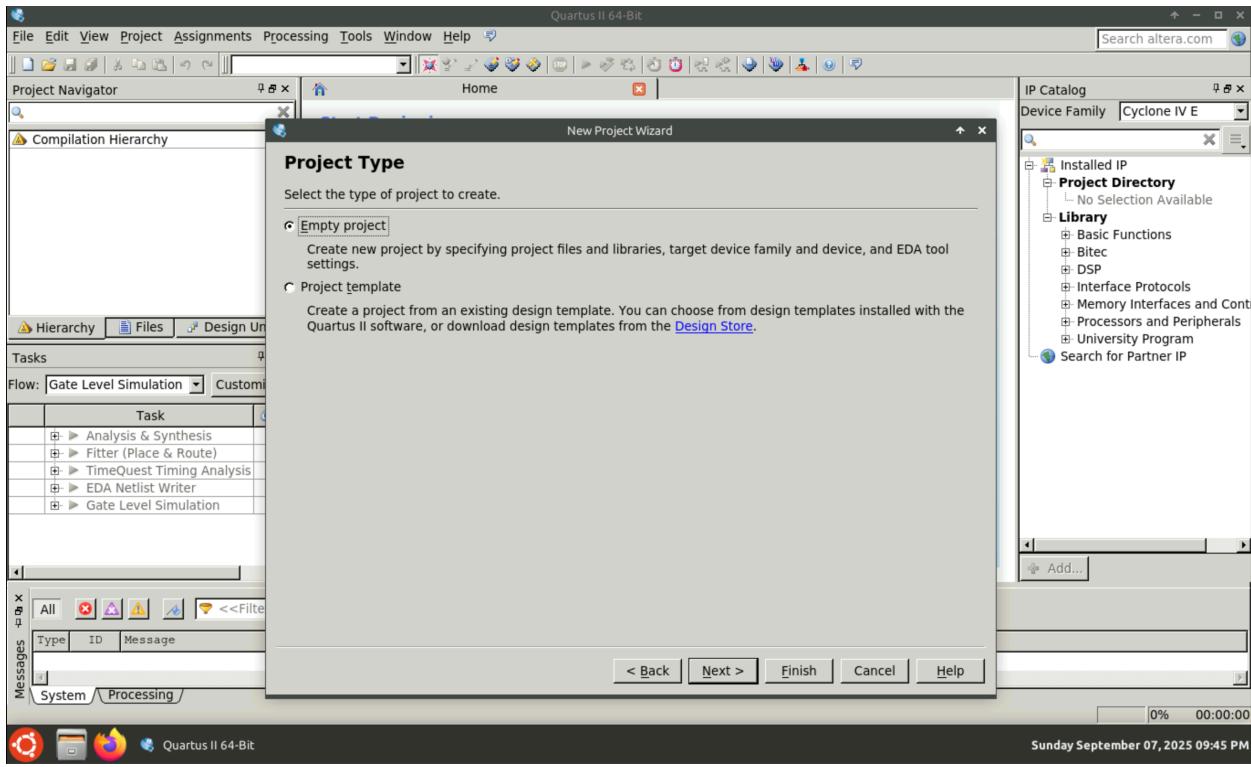
4. You will see the introduction page. Click on next.



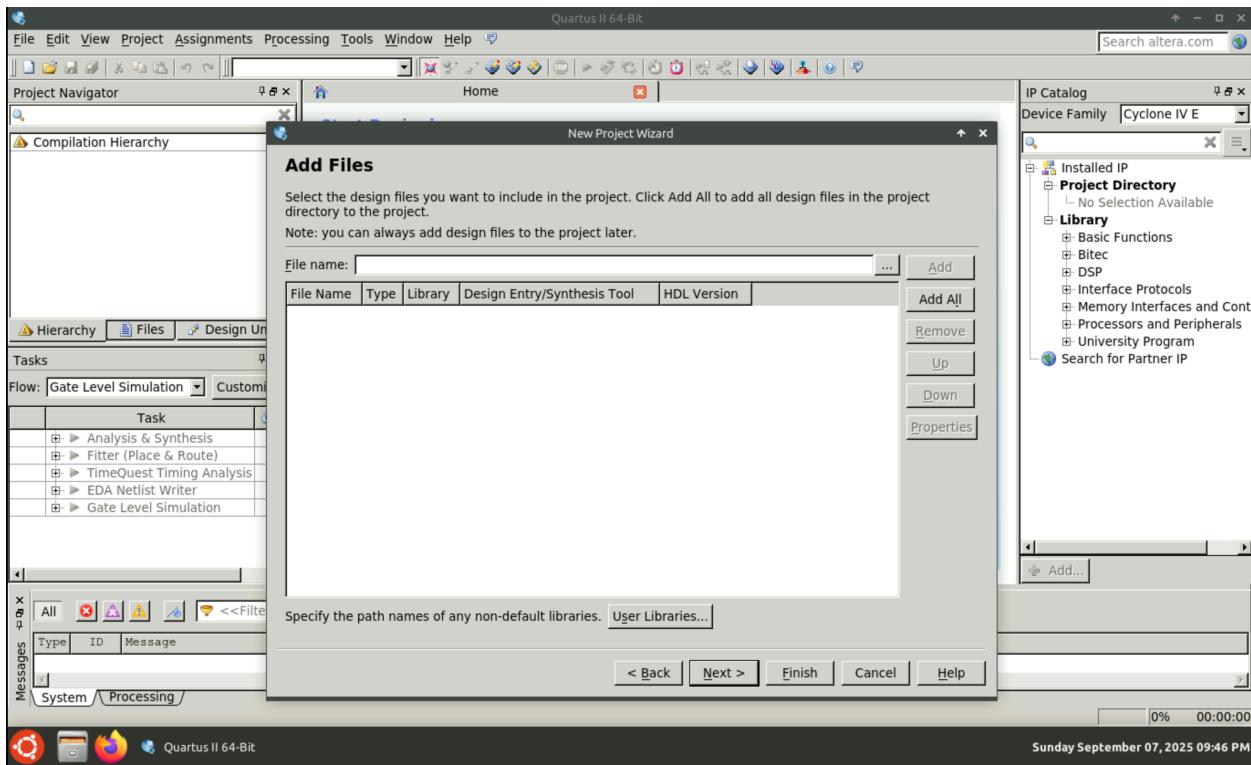
5. On the next screen, create a new folder for your project and name the project.



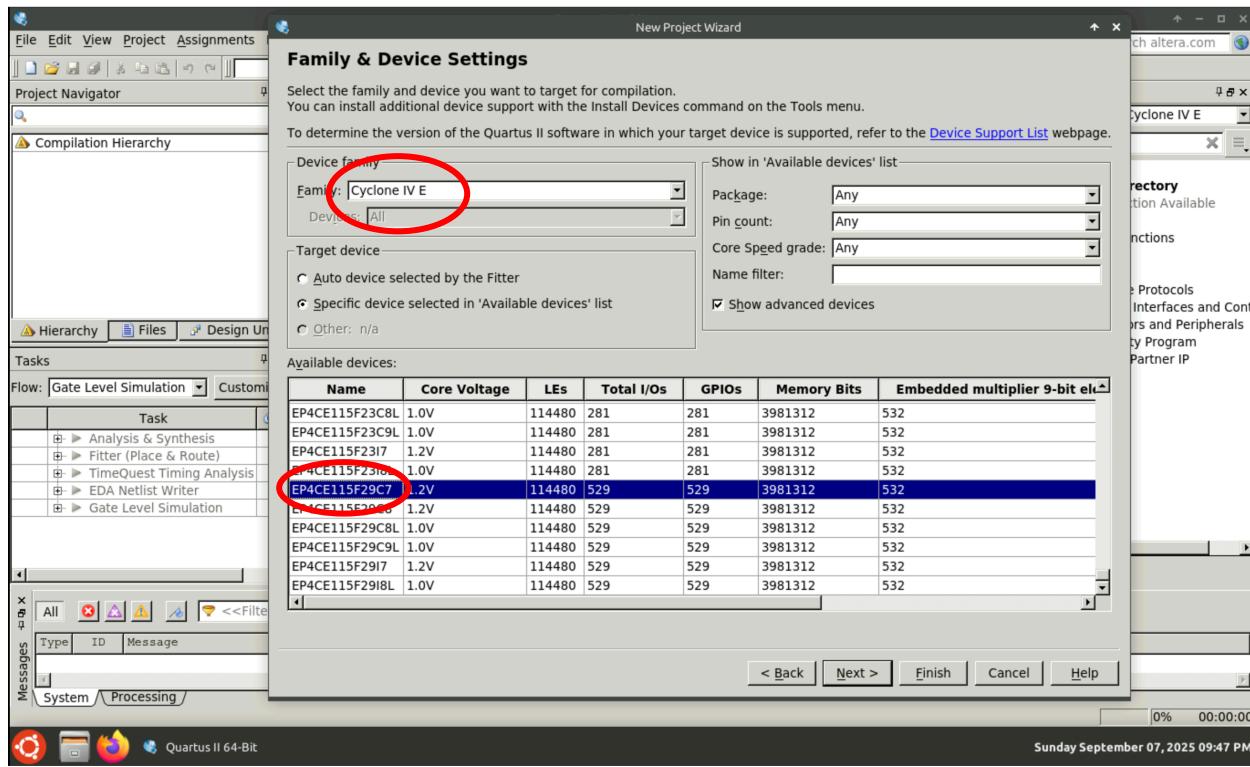
6. Then click next and select "Empty Project". Then click next.



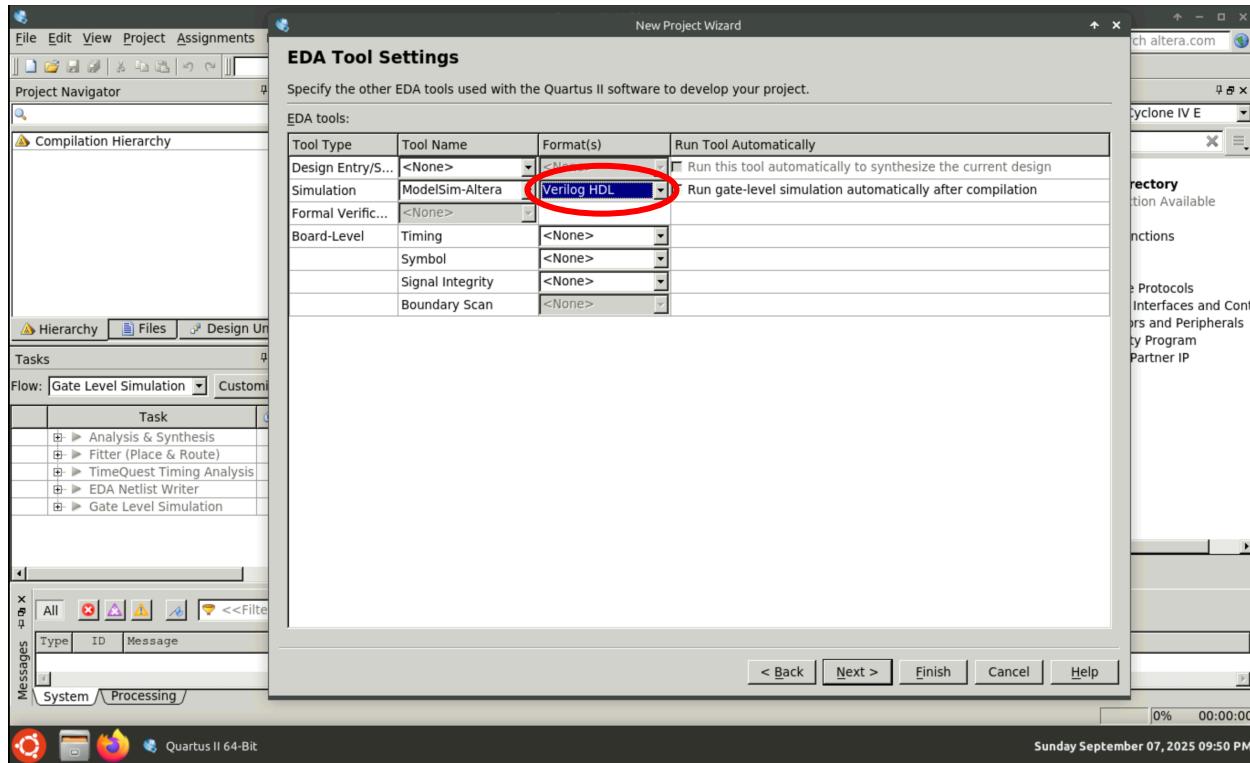
7. You will see the following screen to add files. You do not have to add any files. Just click next and proceed to next step.



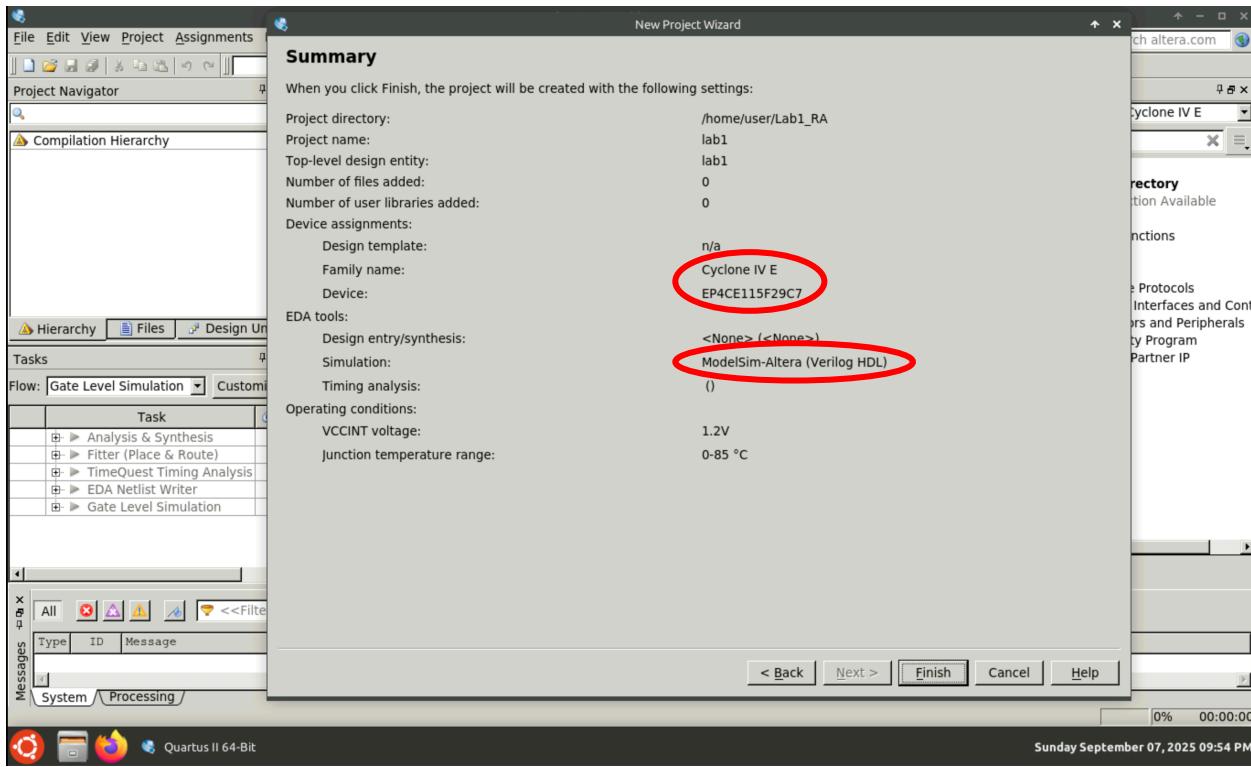
8. Please strictly follow the settings in the screenshot below. After making these selections, click next.



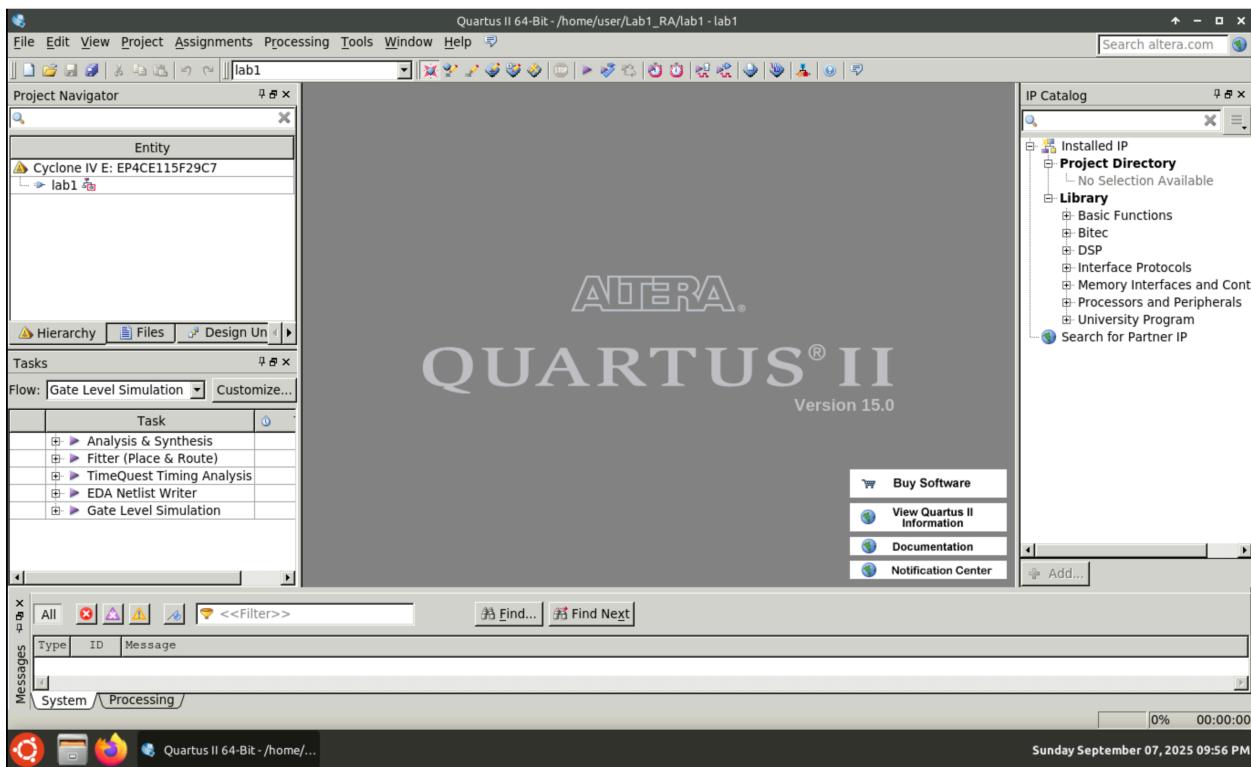
9. On the next screen, make sure to change the simulation format to Verilog HDL. Then hit next.



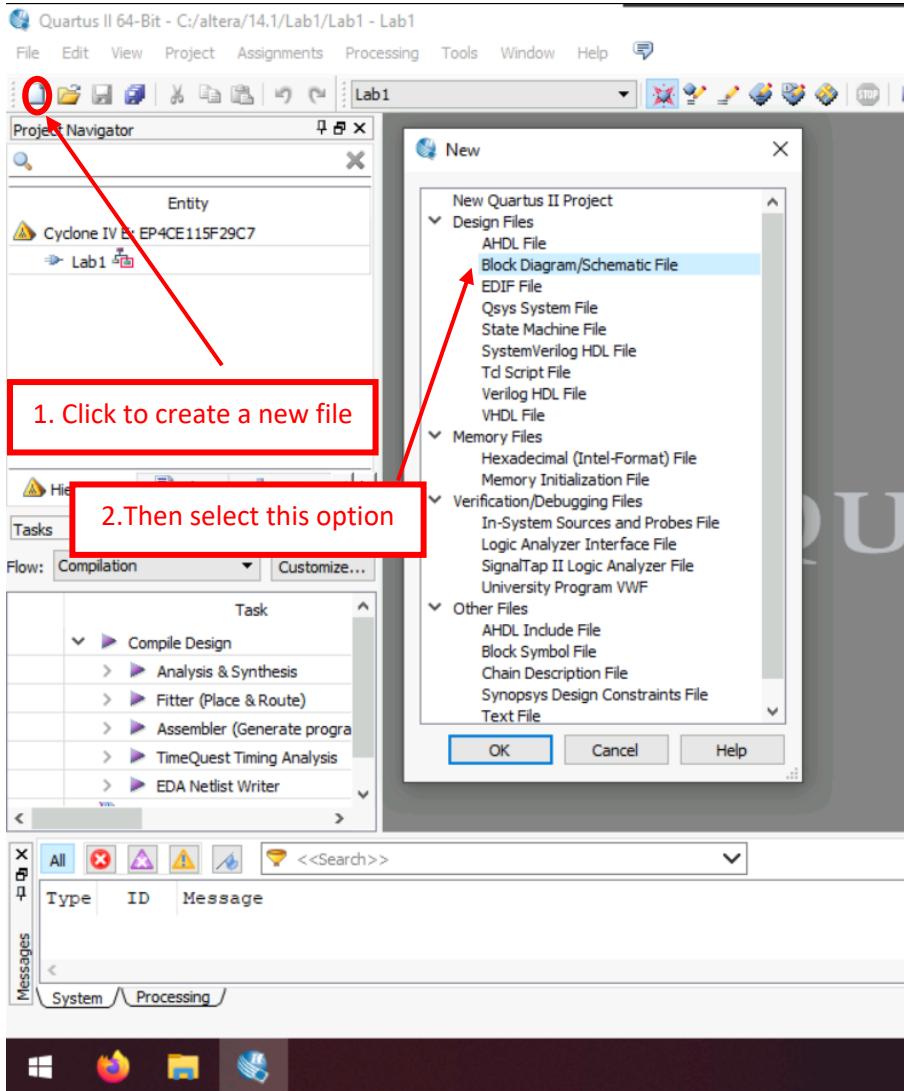
10. You will see the summary page next, verify that you selected the correct board (EP4CE115F29C7) and the correct simulation format (Verilog HDL). Once verified, hit finish.



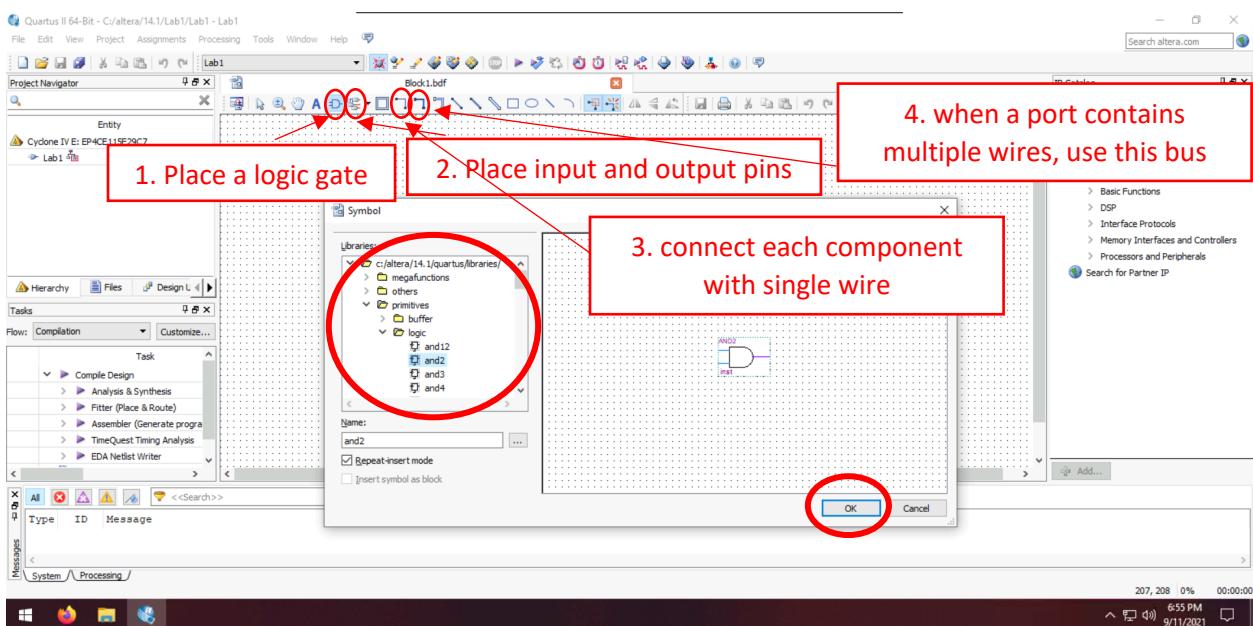
11. Quartus will take a few seconds to load. Finally, you will see the following screen. You have successfully created your first empty project.



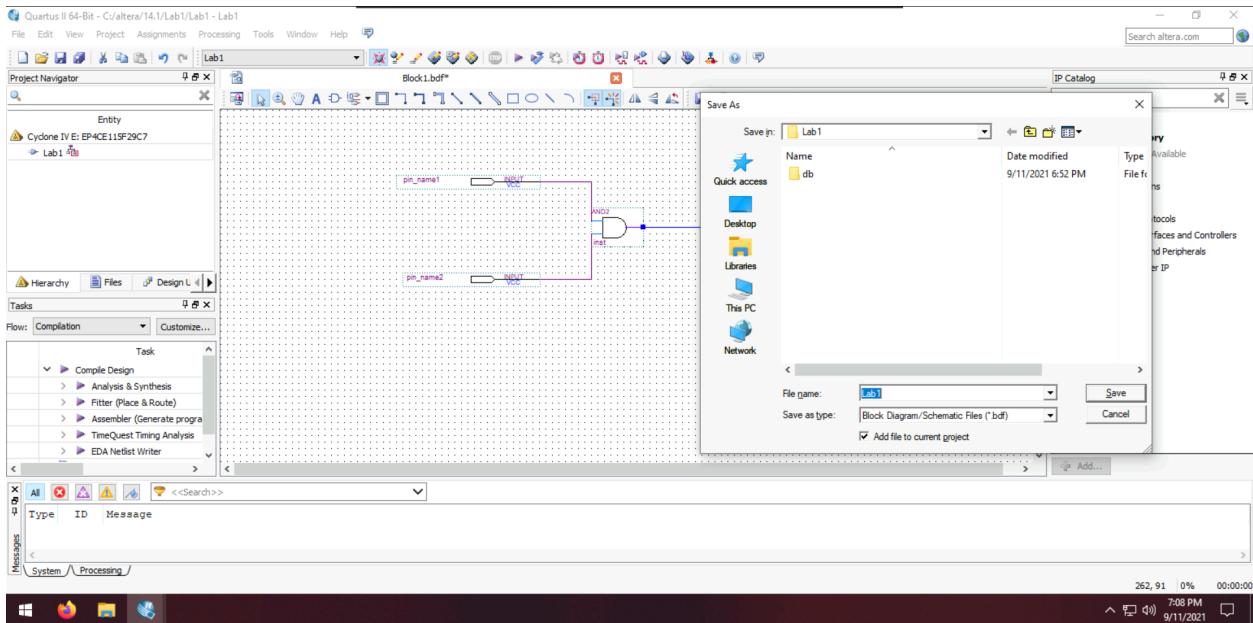
12. Now create a new file and select “Block Diagram/Schematic File” from the set of options that you get.



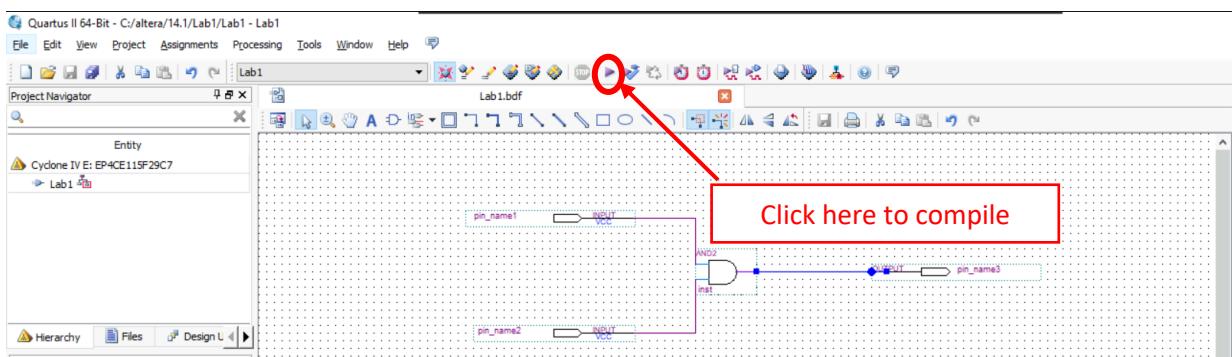
13. You'll see a canvas similar to what you might have seen before in LTspice. Play around with it and learn what the different buttons you can use and what they mean.



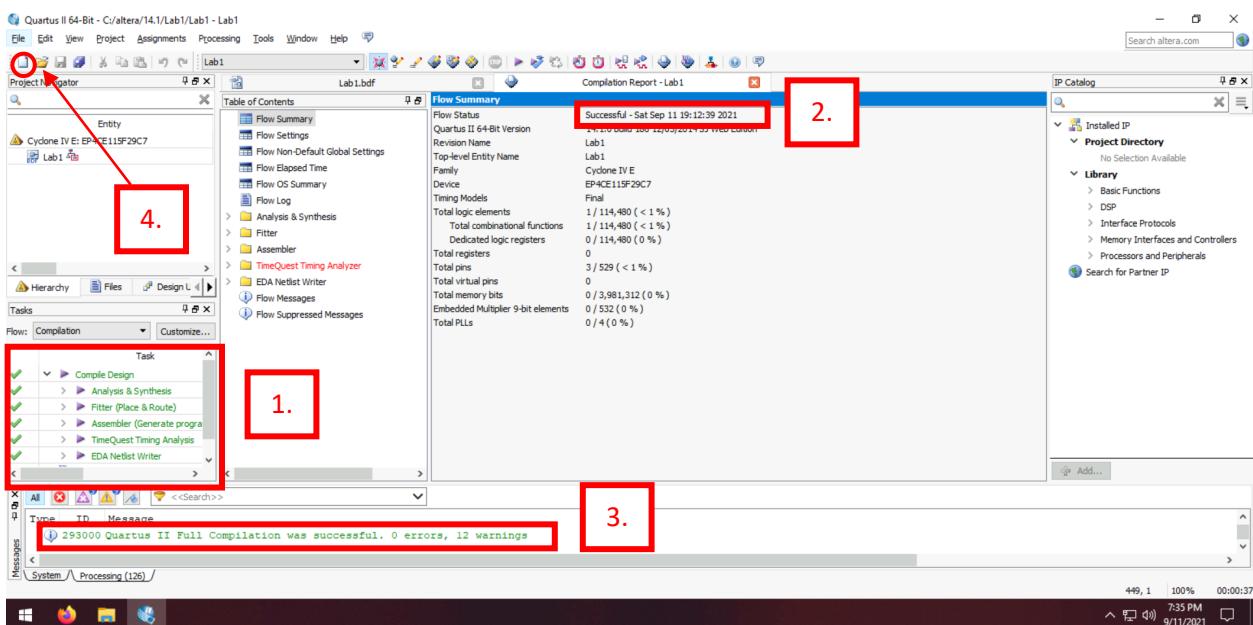
14. Save your source file (the block diagram you just created) with the same name as the project name (only for top level entity).

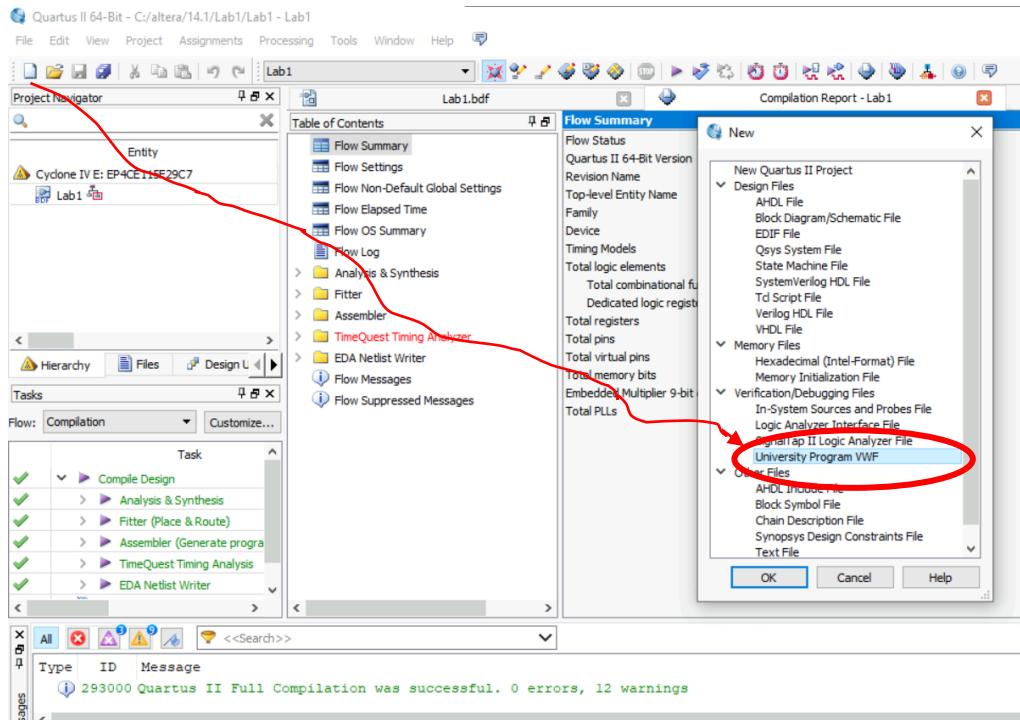


15. Now compile it by clicking on the play button on the top bar.



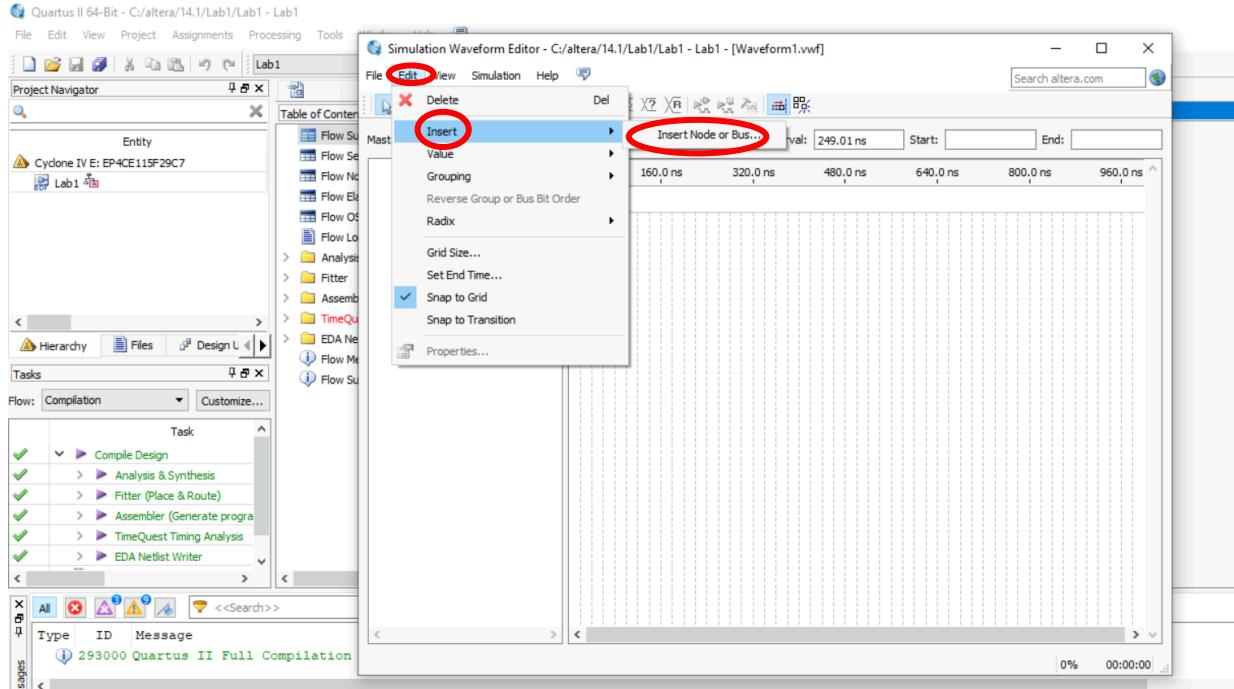
16. Make sure that compiling is successful by checking at following three locations numbered 1, 2 and 3. Then, create the simulation file by clicking at number 4 and choosing the following options.



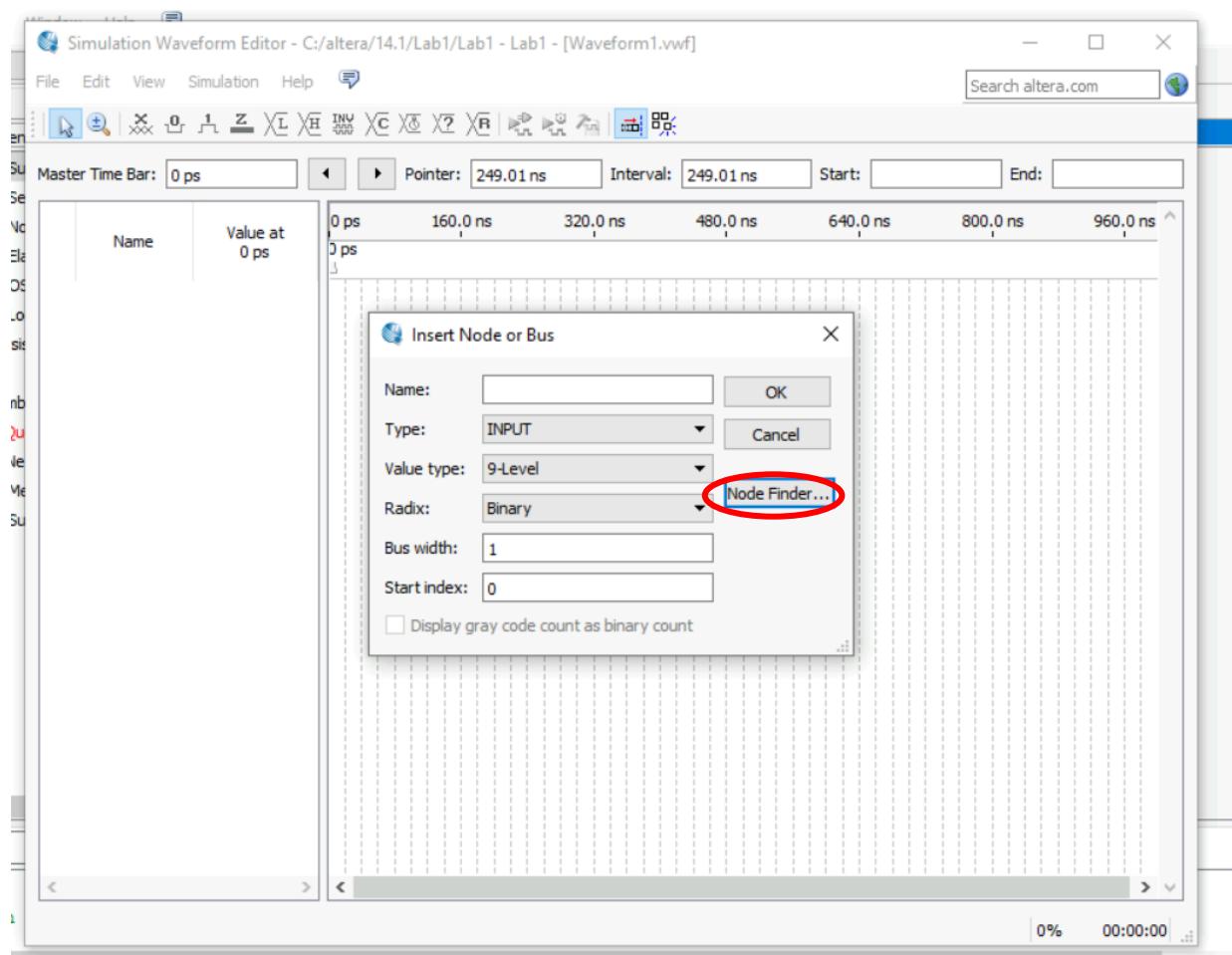


17. Add the input and output nodes for simulation by following steps 1 to 4.

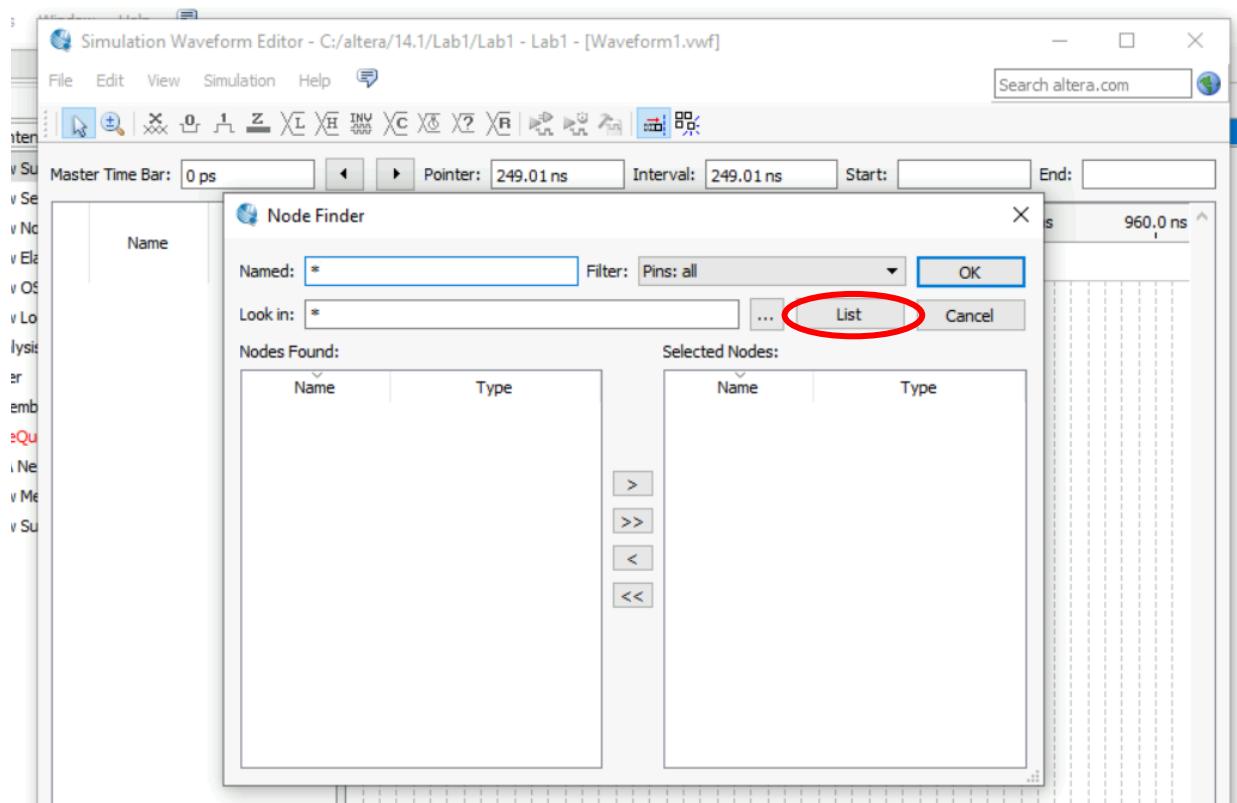
Step 1



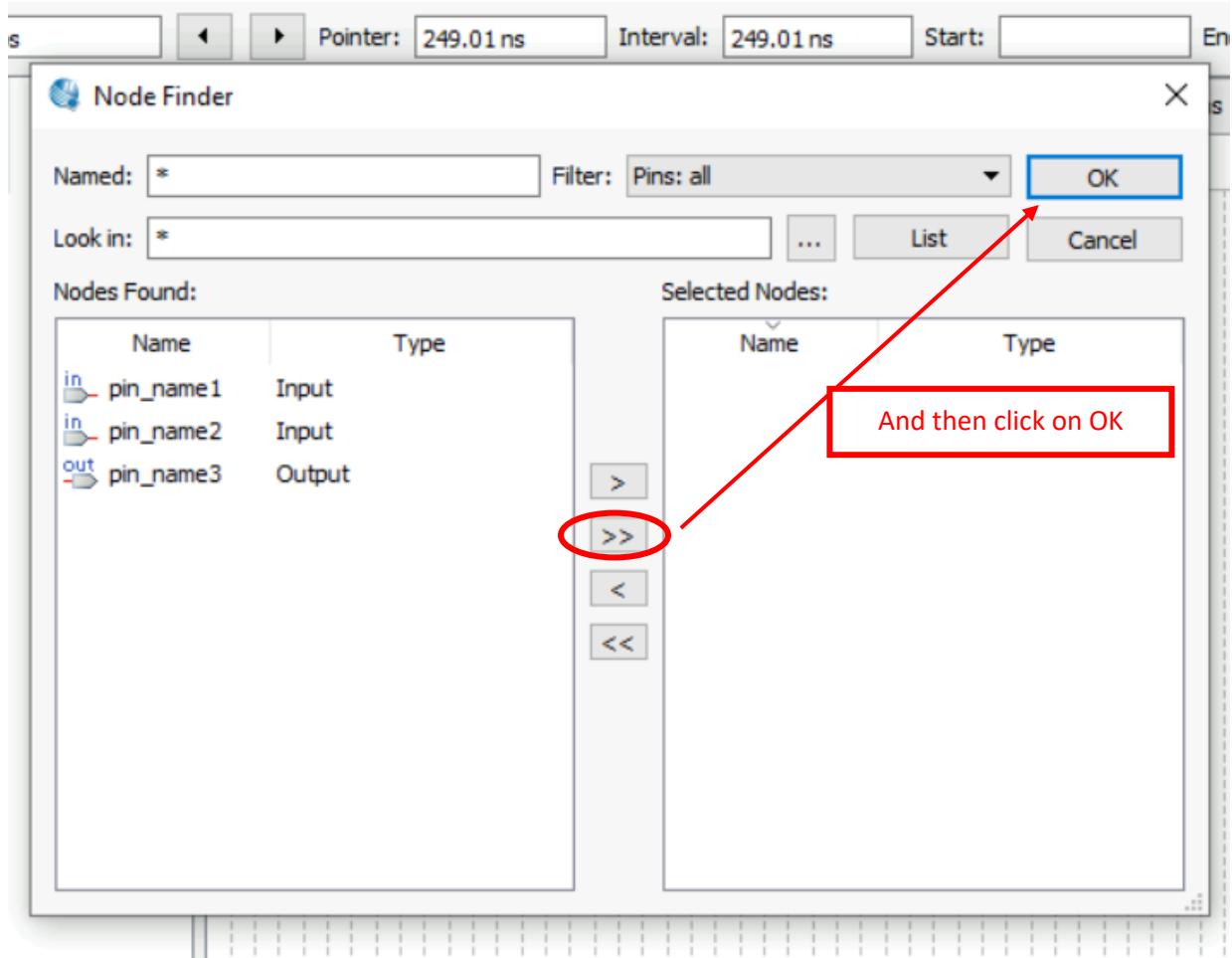
Step 2



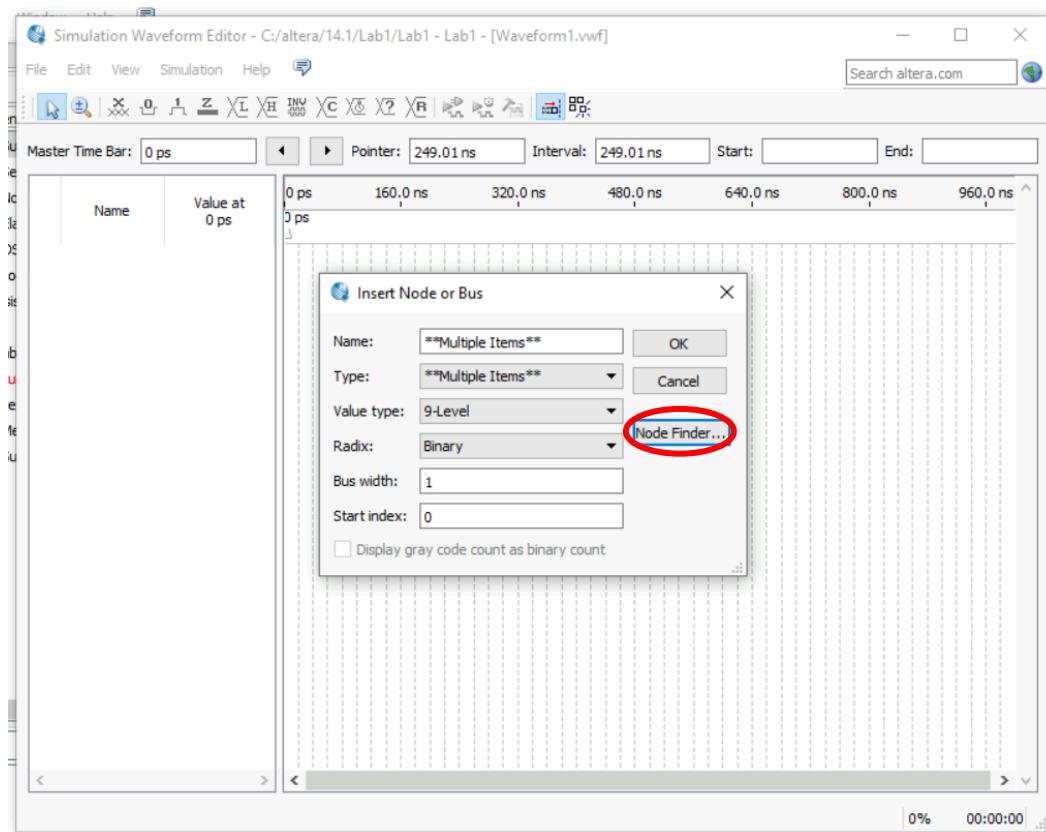
Step 3



Step 4

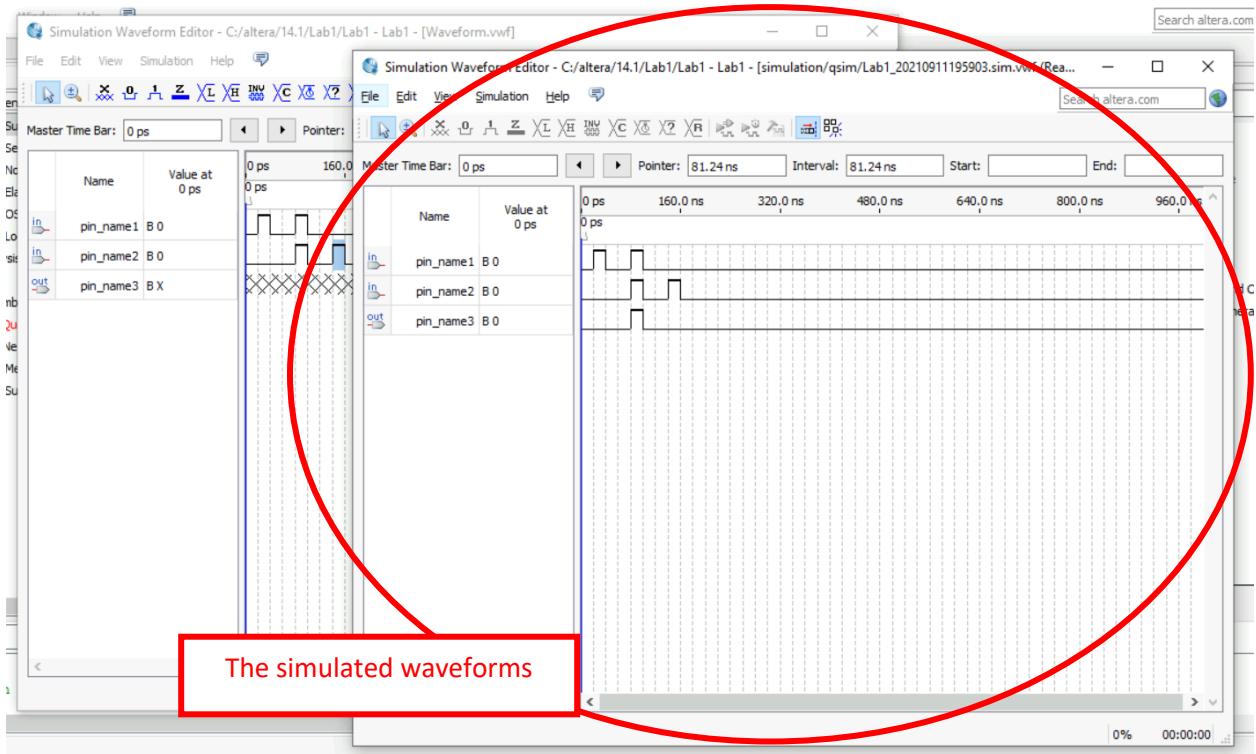
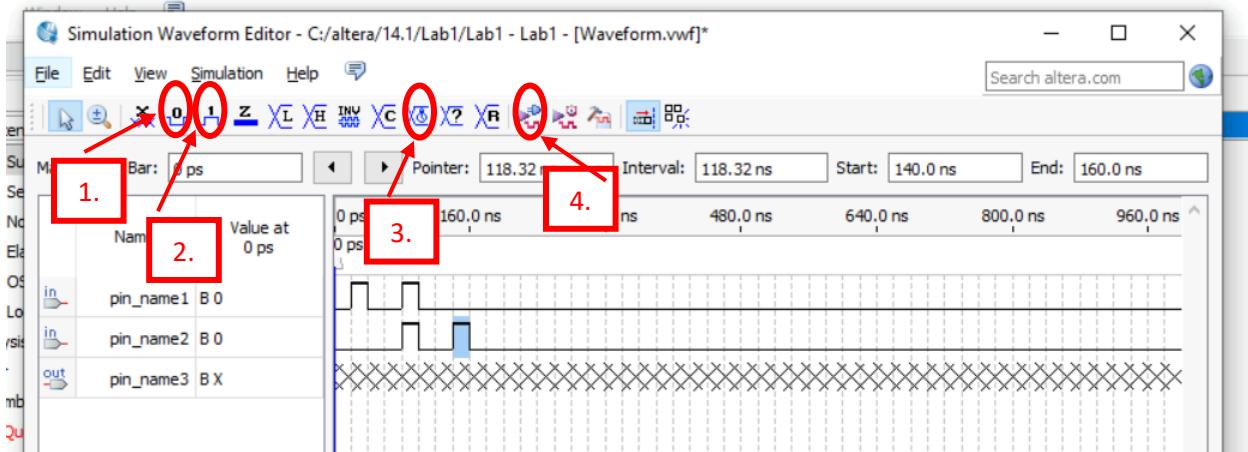


Step 5



Step 6

Use the tools numbered 1,2 and 3 to generate the input signals and then hit 4 to save and start the simulation.



Some Tips

1. Keep saving your work frequently.
2. If you want to access your source code later on, copy it to your google drive or some other cloud storage. The ECE machine may not keep your work saved for later.