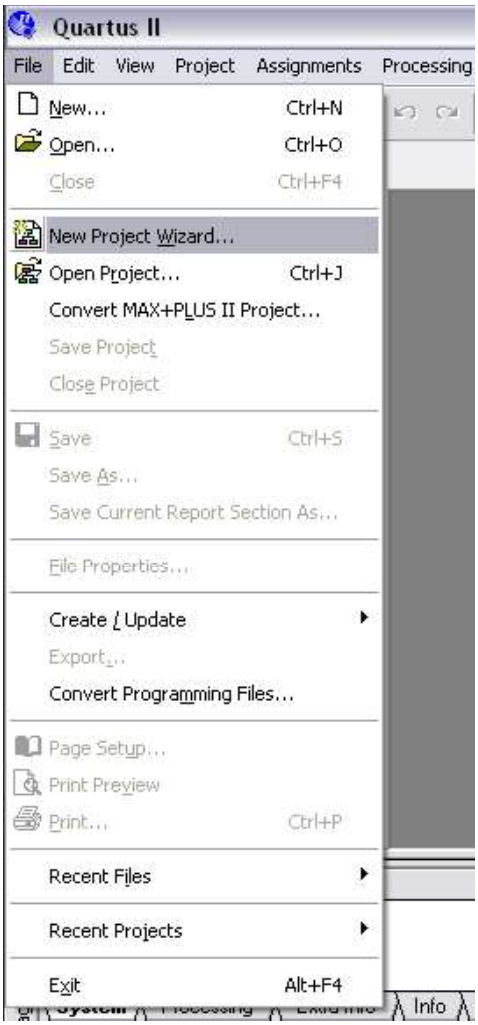
3bit Binary Counter for the Altera DEnano Development Kit

There are four basic steps to using the development kit. One, set up the directories to hold the project. Two, design the project. Three, simulate the project. And four, load the project to the development kit.

3bit_counter.zip - Zip file of all files from this example.

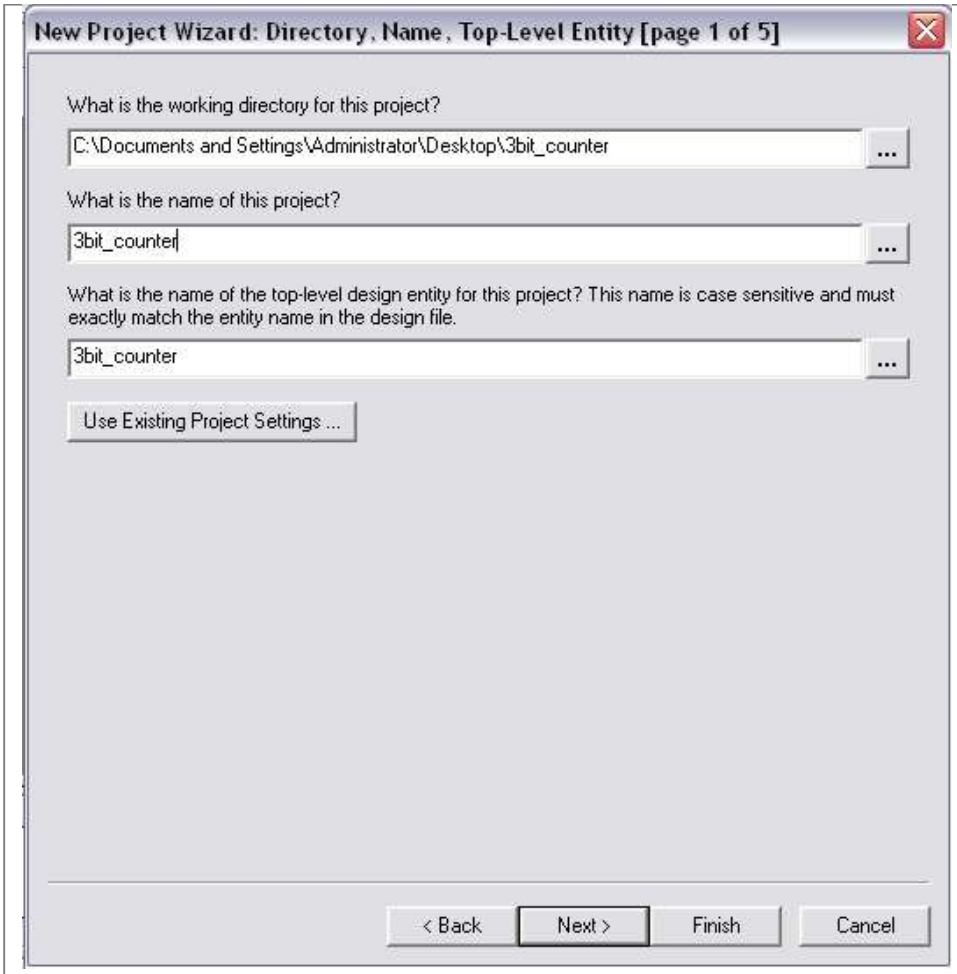
Setting Up the Project

Start Quartus and go to File -> New Project Wizard.

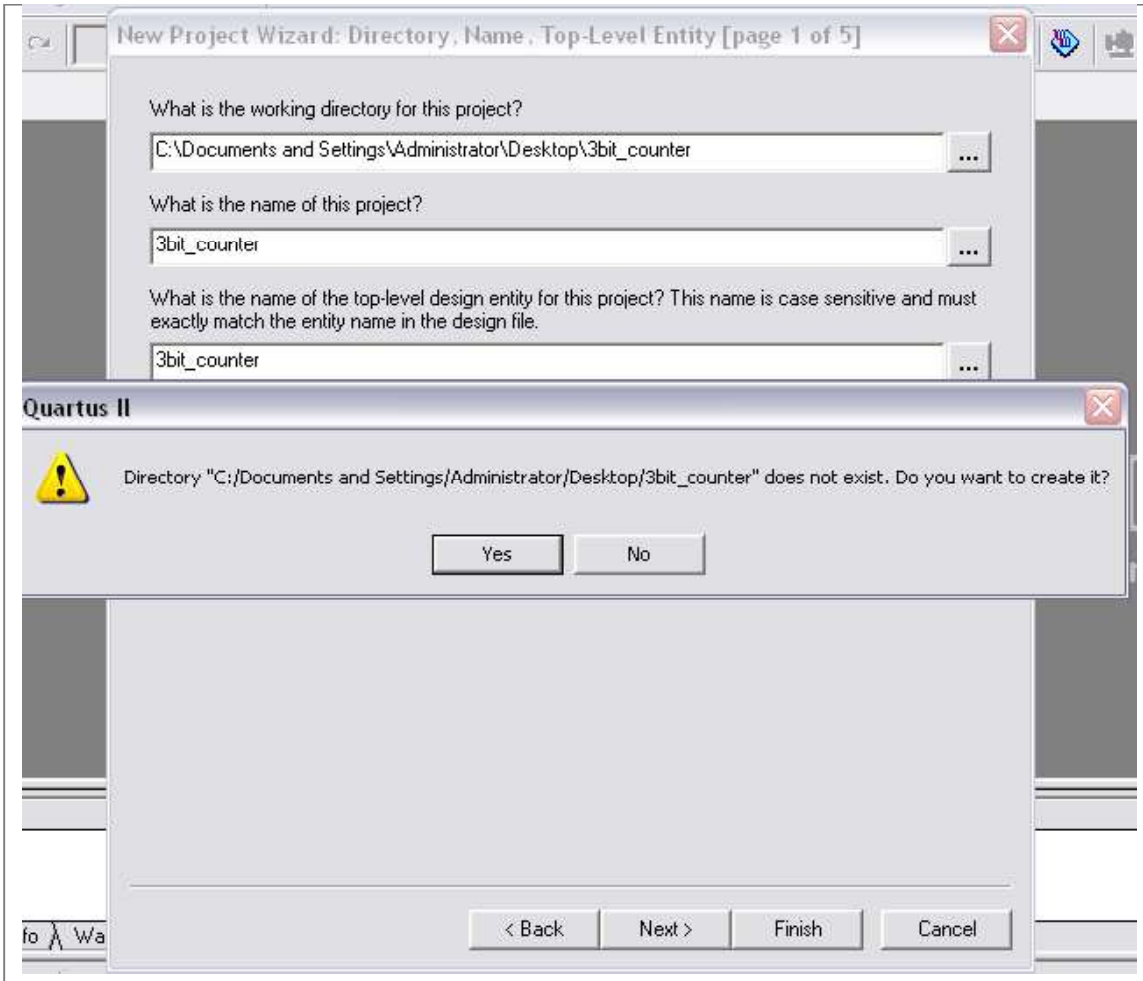


Click Next on the first screen.

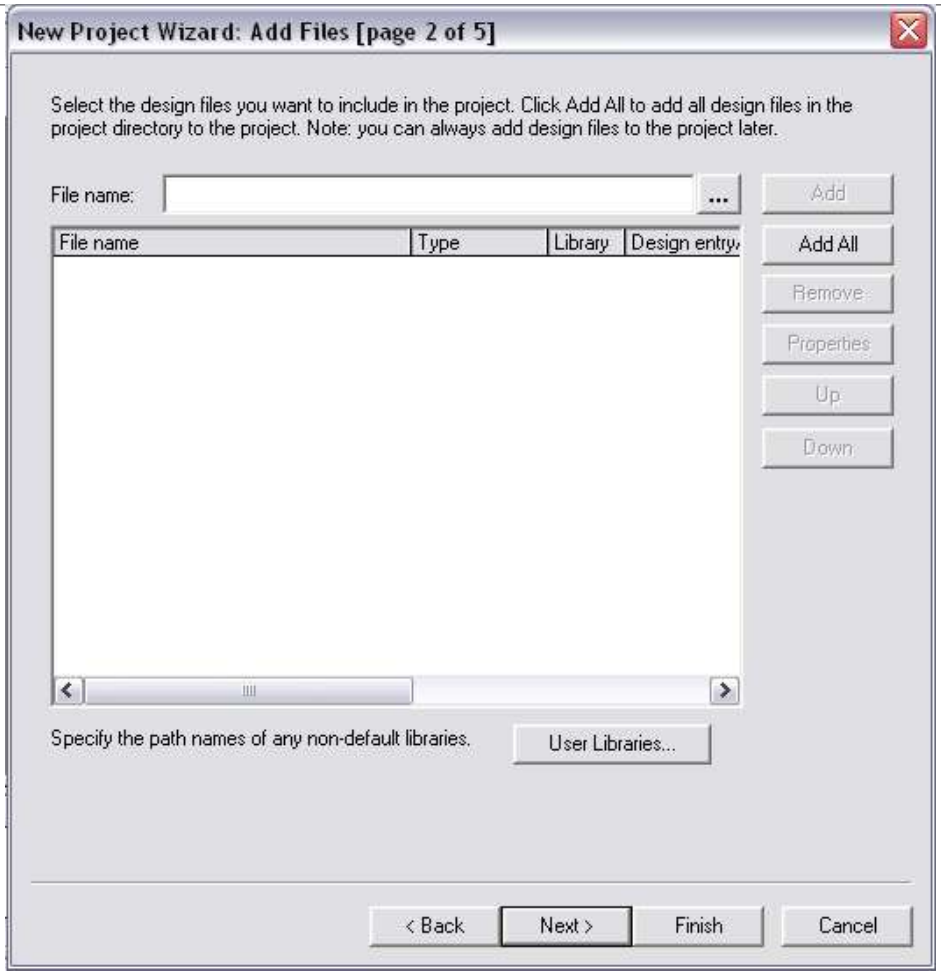

New Project Wizard: Introduction
The New Project Wizard helps you create a new project and preliminary project settings, including the following:
Project name and directory
Name of the top-level design entity
Project files and libraries
Target device family and device
EDA tool settings
You can change the settings for an existing project and specify additional project-wide settings with the Settings command (Assignments menu). You can use the various pages of the Settings dialog box to add functionality to the project.
Don't show me this introduction again
< Back
Next >
Finish
Cancel



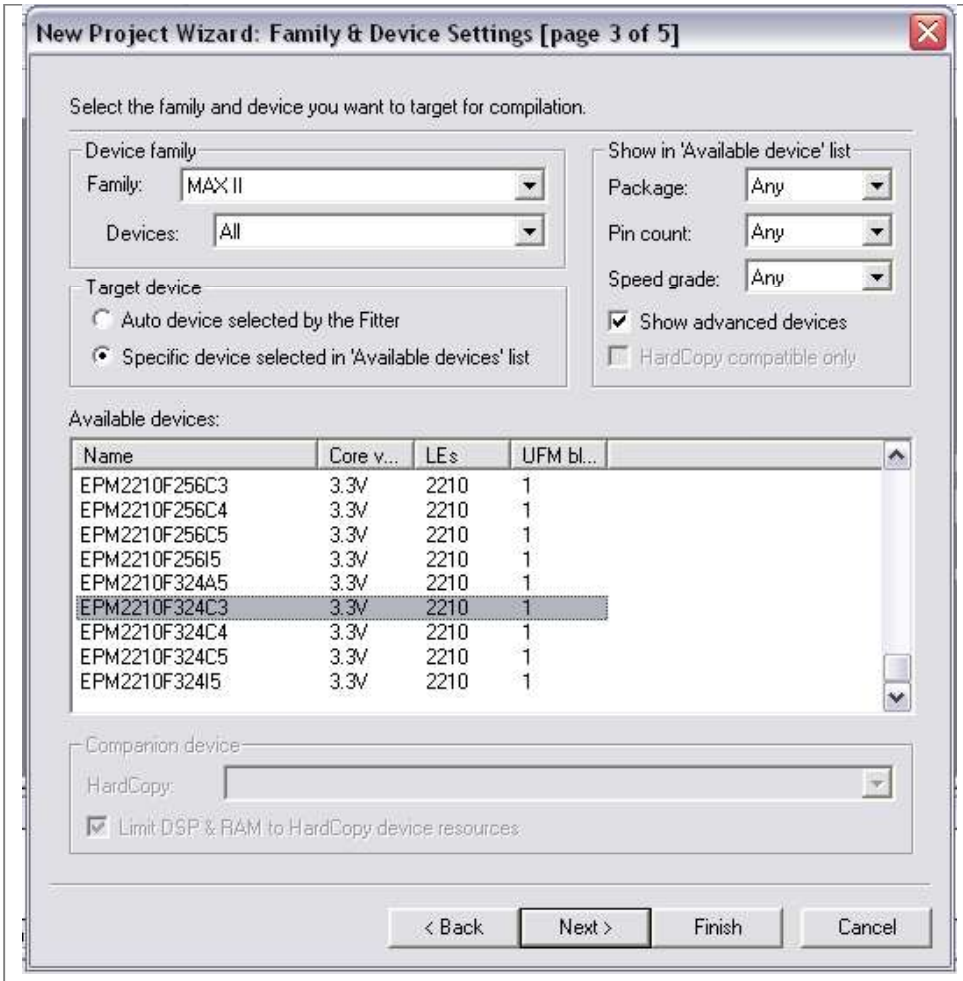
If the directory you specify is not present, you'll be prompted to create it.



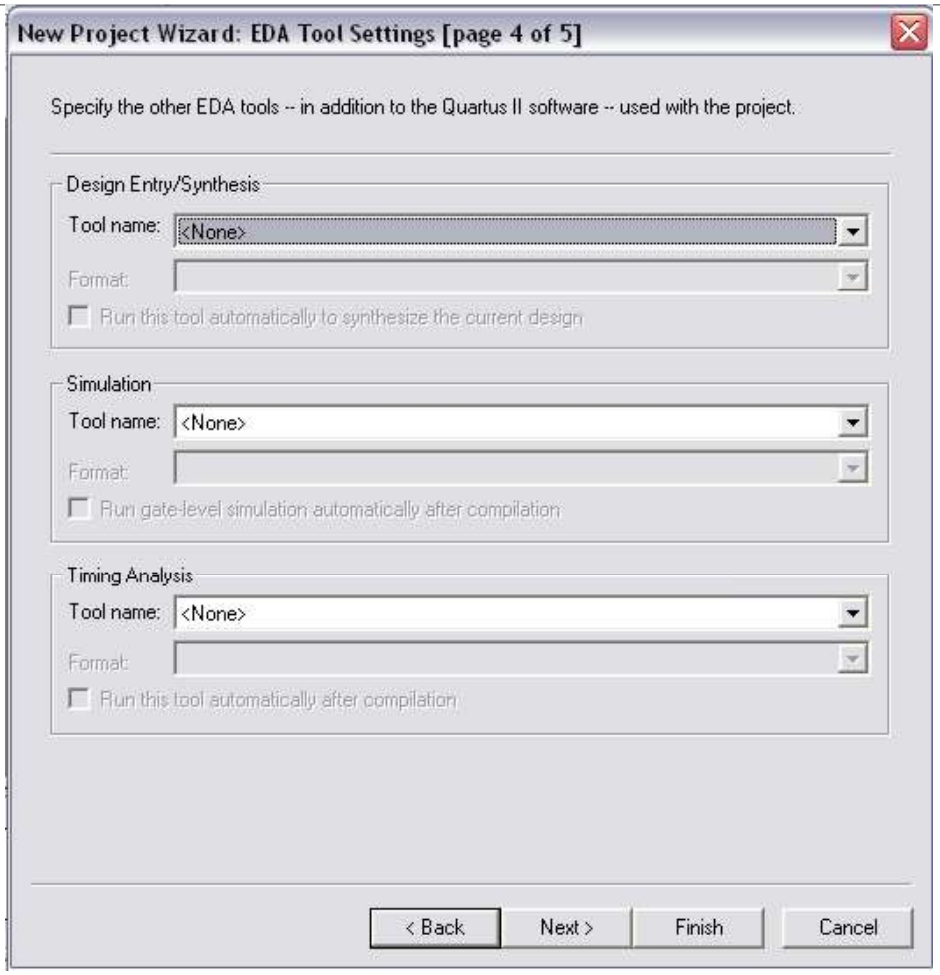
Click Next on the next window.



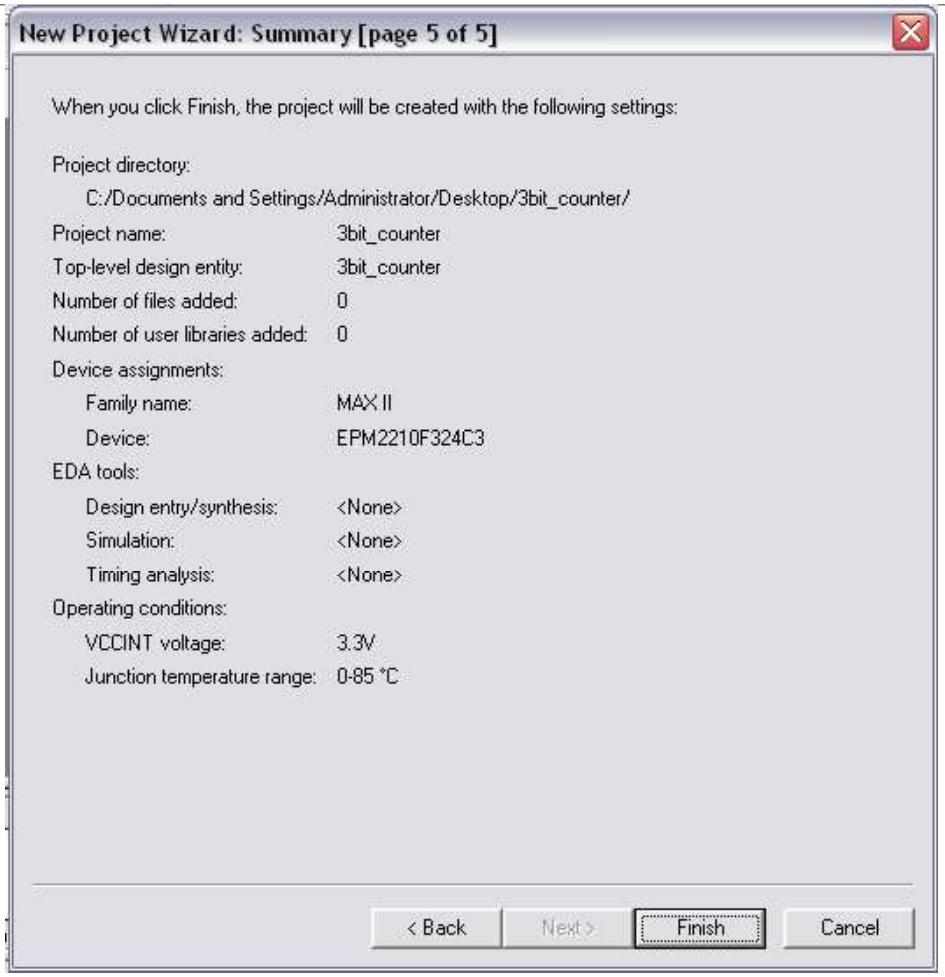
This window is very important. This is where we specify which chip we're using. If you look at our development kits, you should be able to read our chip, which is a MAX II EPM2210F324C3N, so select that chip (don't worry about the last N) in the list. Make sure that you have Family set to MAX II and Devices All.



Click Next in the next window.

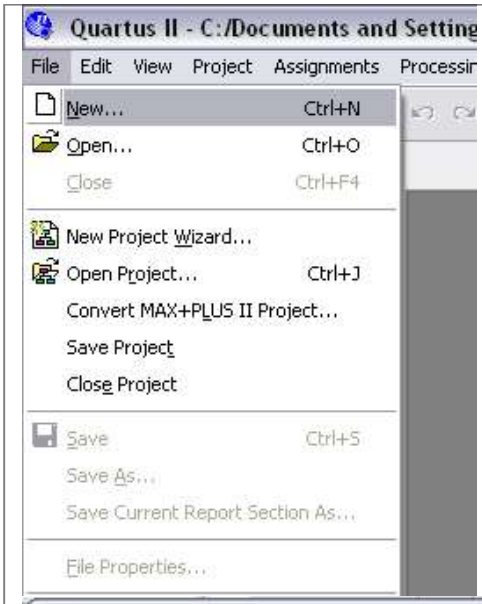


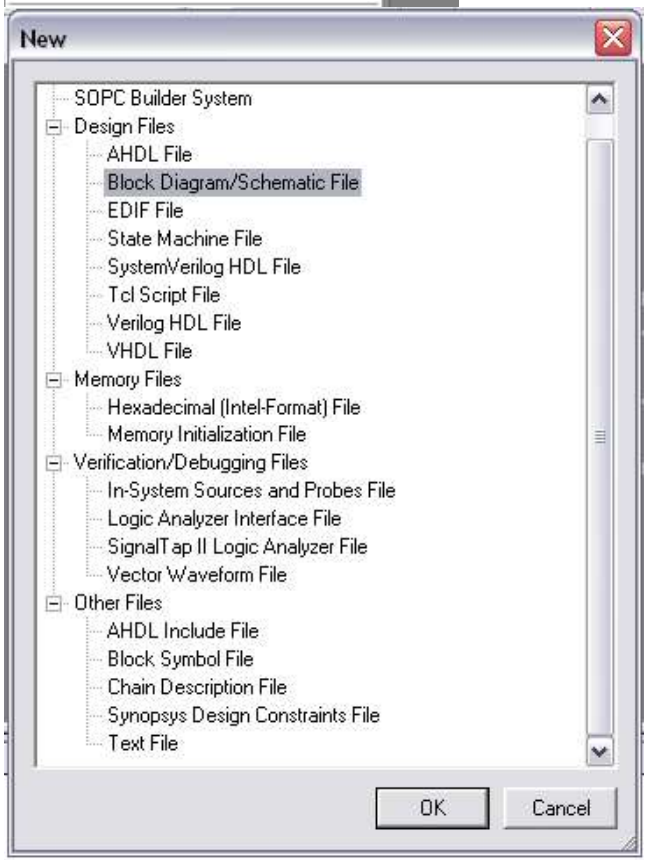
This last window is just the summary of the project that has just been setup. Click Finish.



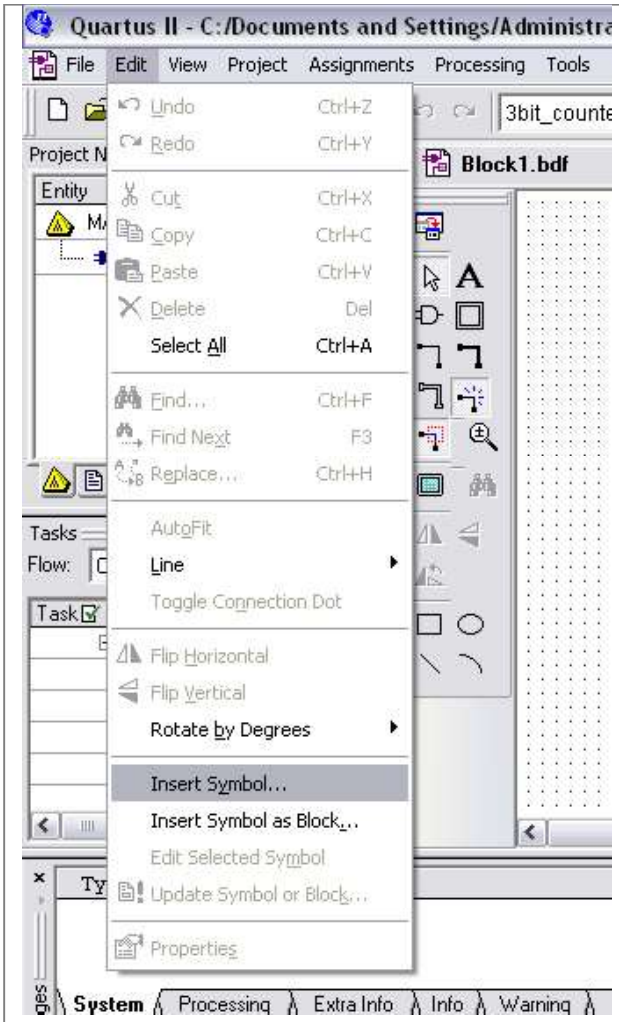
Designing the Project

Now we need to put the design the counter. In this example, we'll do it graphically. So the first step is to open a new schematic file. Go to File -> New and choose Block Diagram/Schematic File from the window that pops up.



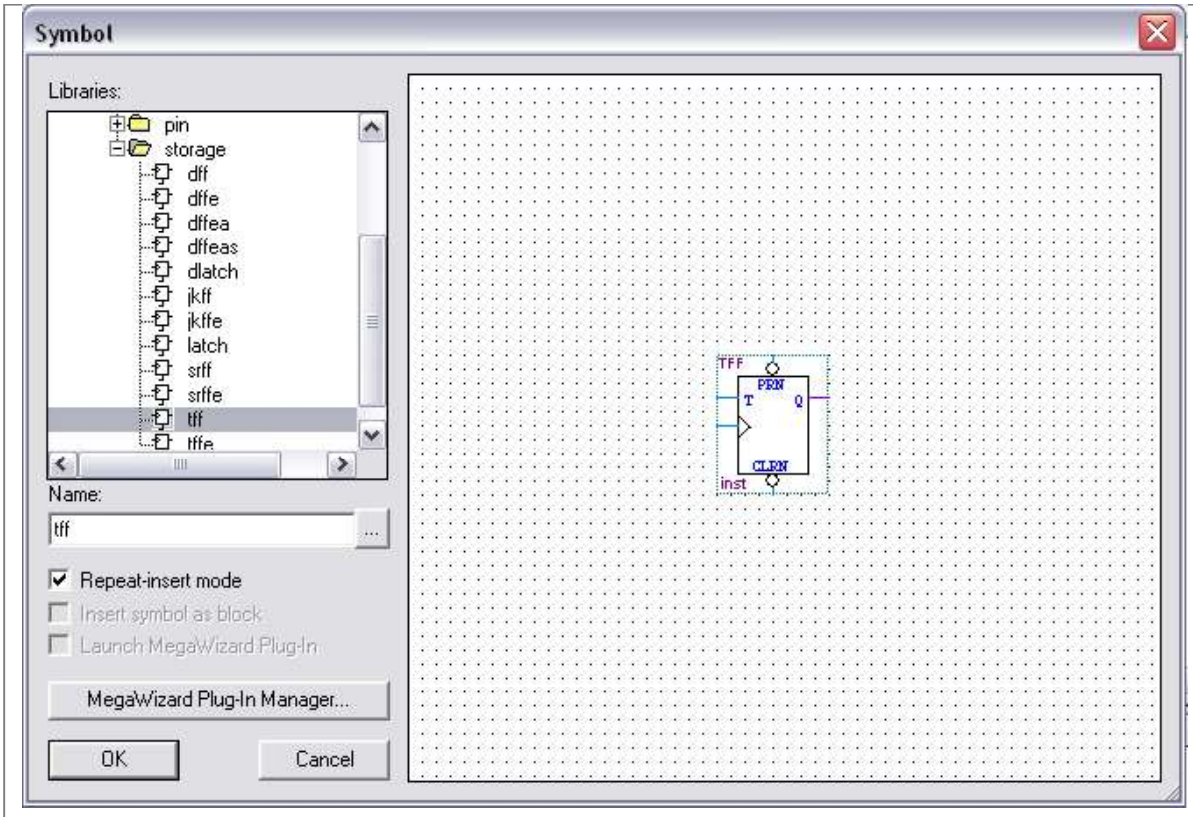


The next step is to insert symbols. Get to the symbol menu by going to Edit -> Insert Symbol or use the Symbol Tool.

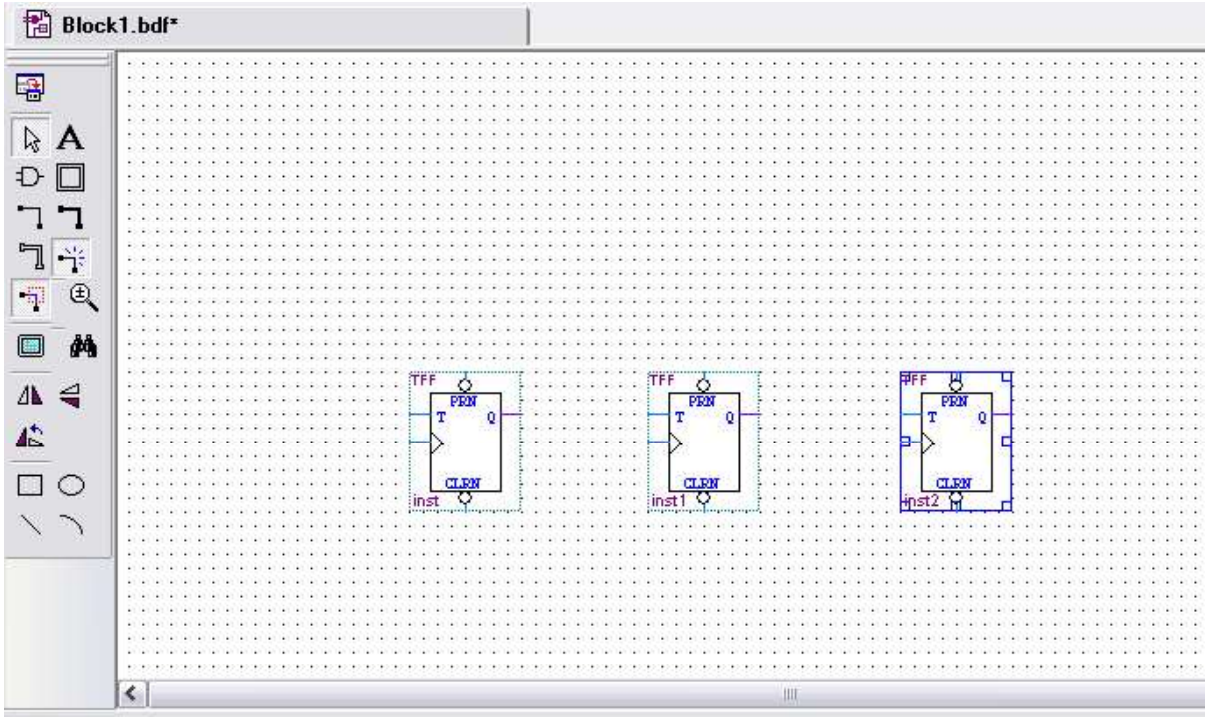


or

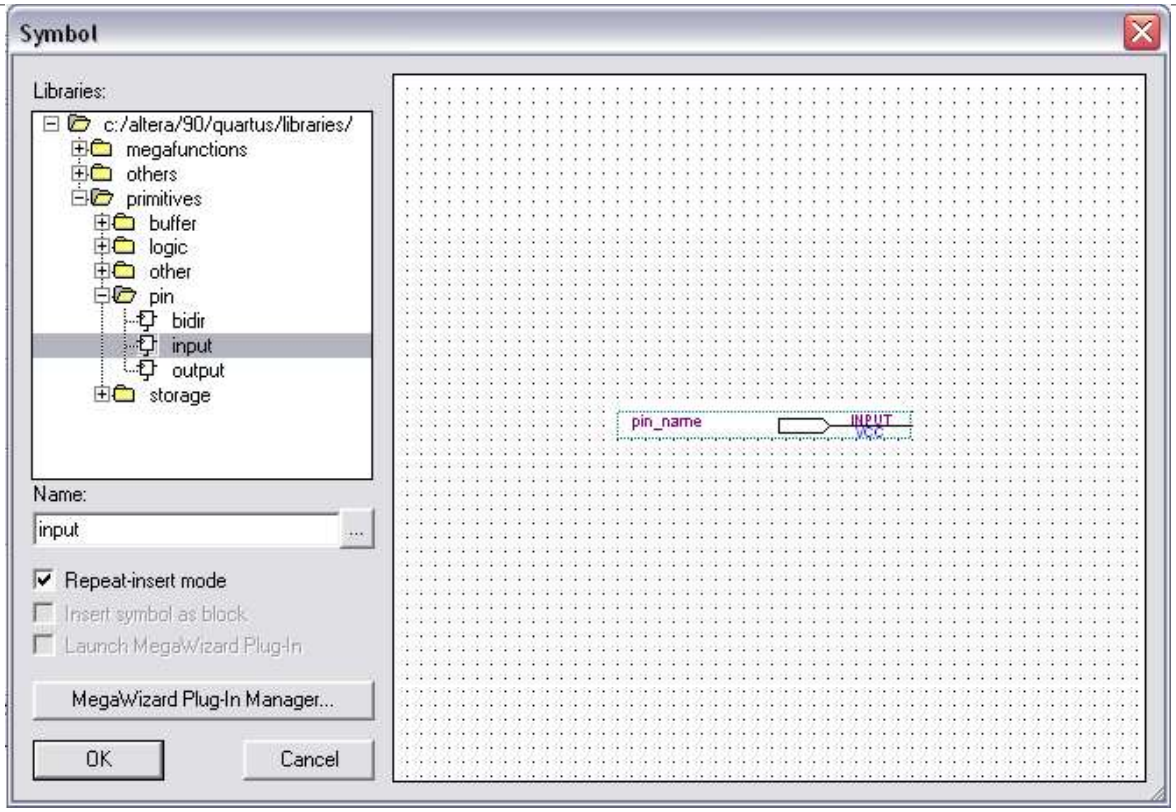
The first item to insert is a tff (toggle flipflop). It's located in the storage section. Check the Repeat-insert mode box, since we'll need three of these.



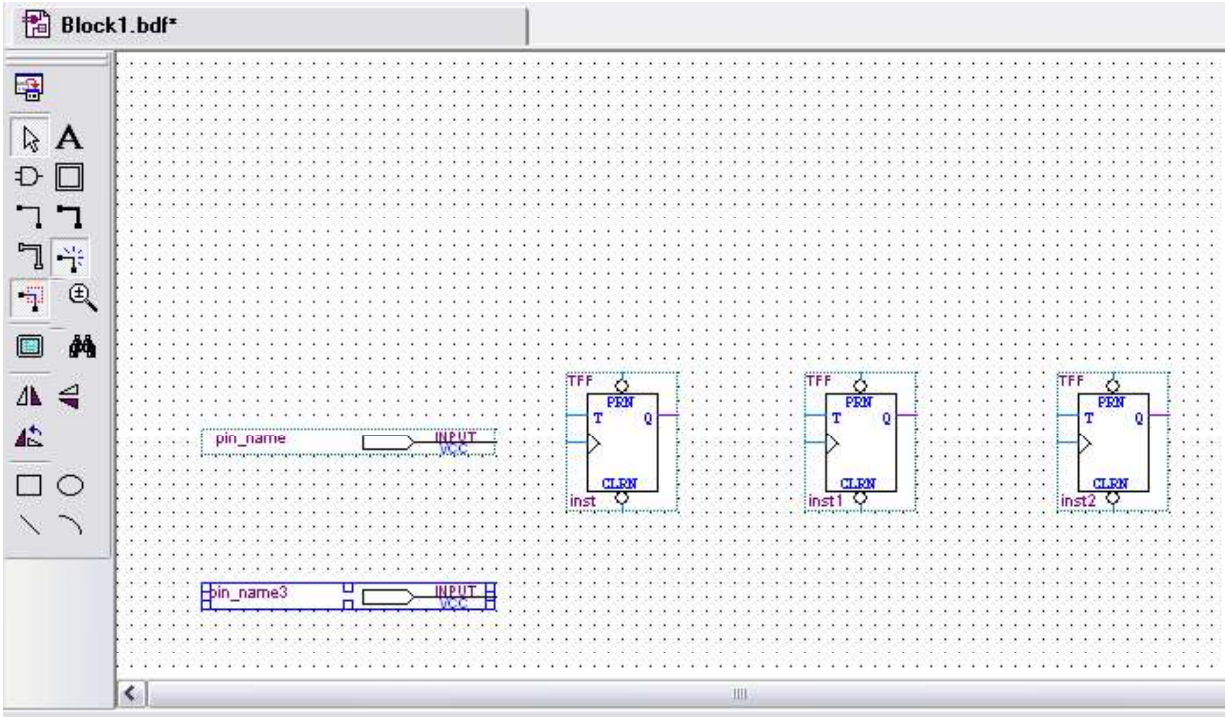
You should now have three tffs in your design.



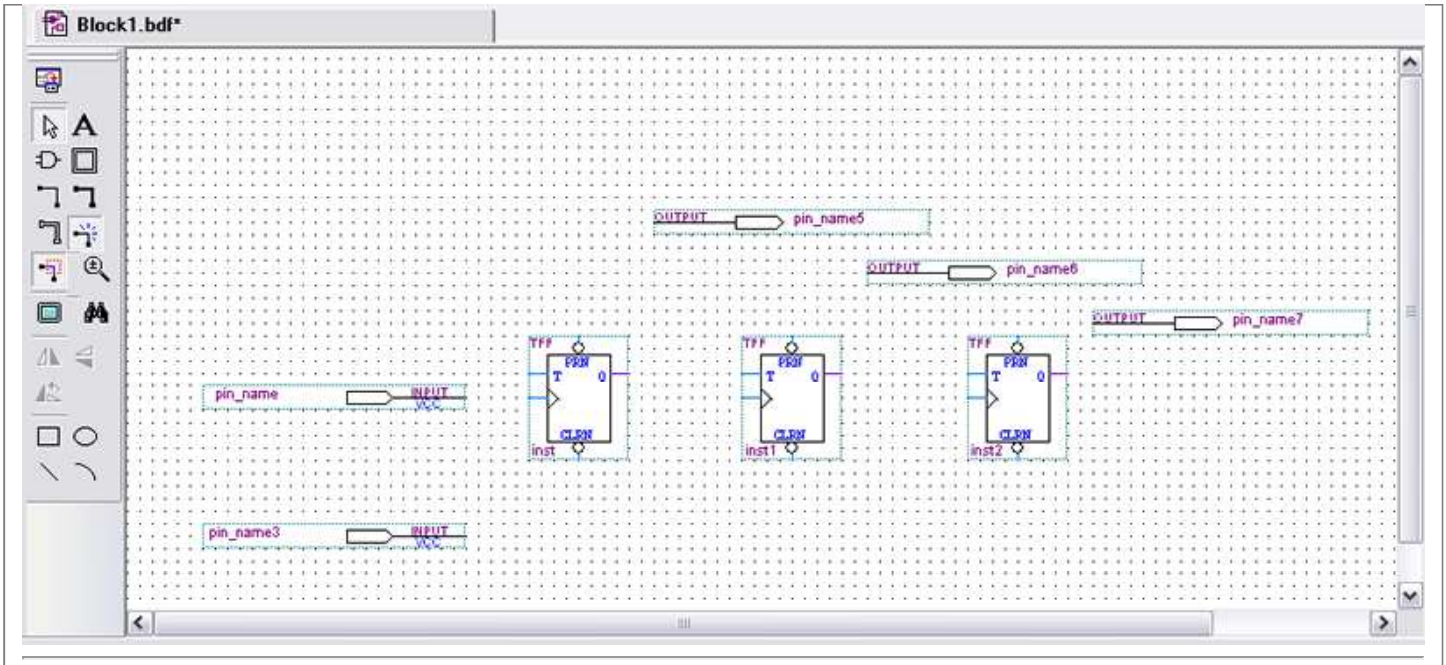
.



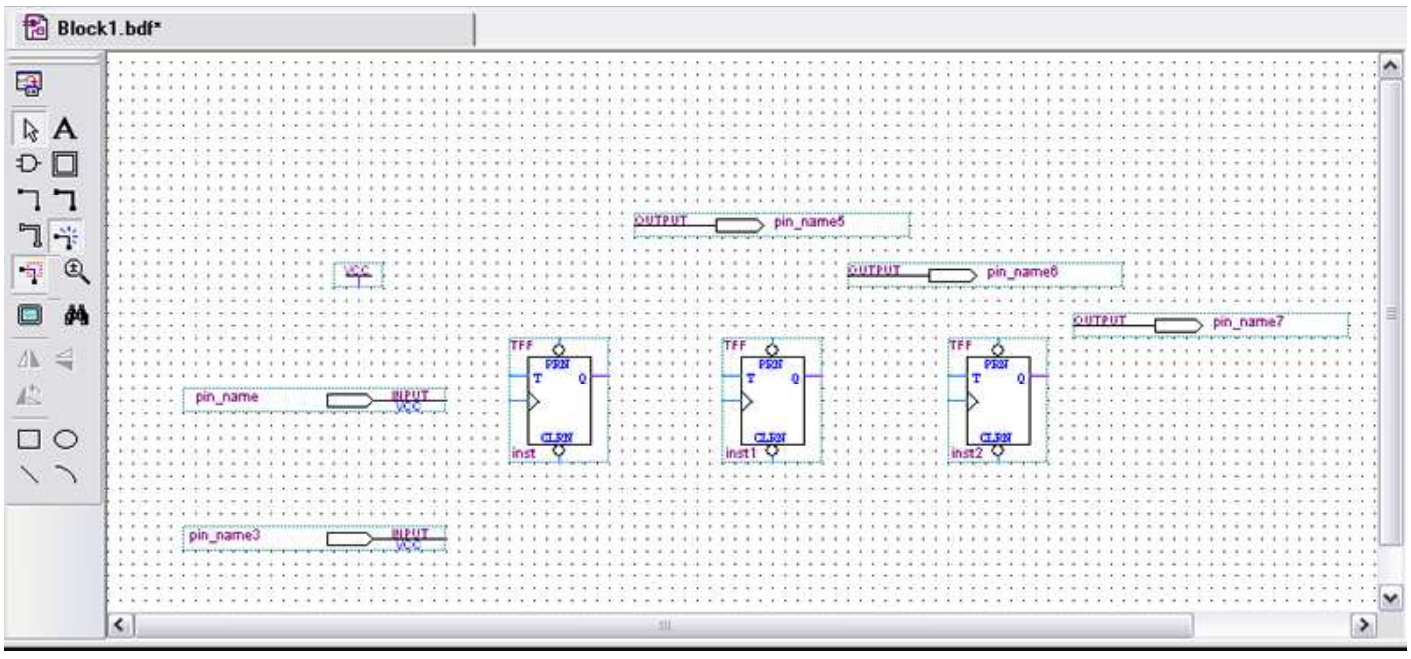
We'll need to input pins.



Now do the same thing and insert three output pins.

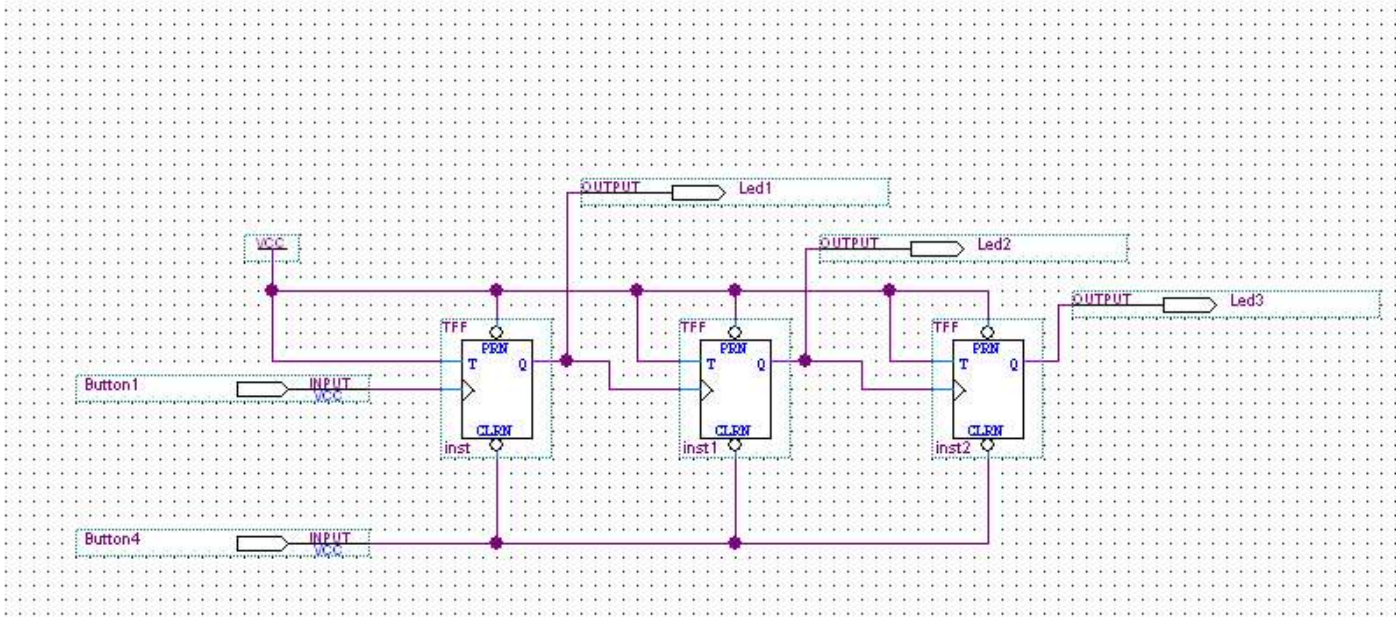


Repeat again and put a vcc in. The vcc will be found under primitives, then other in the symbol menu.

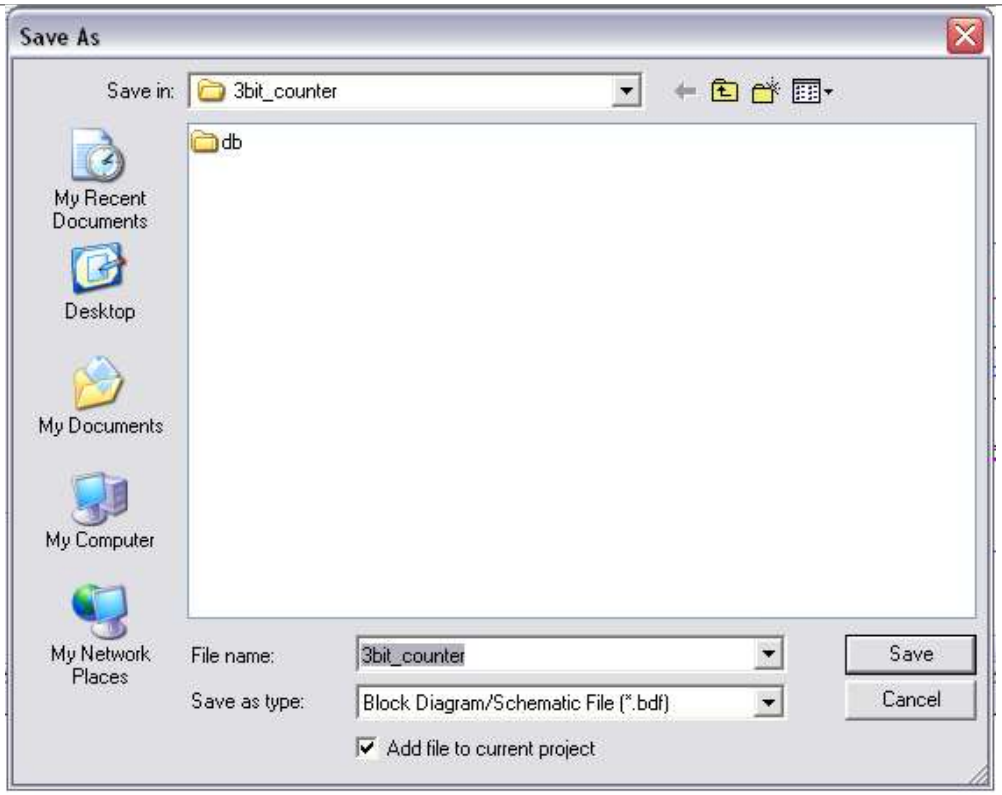


Those are all the parts that you need. Now you need to connect them according to the following diagram. Use the Orthogonal Node Tool to draw the connections.

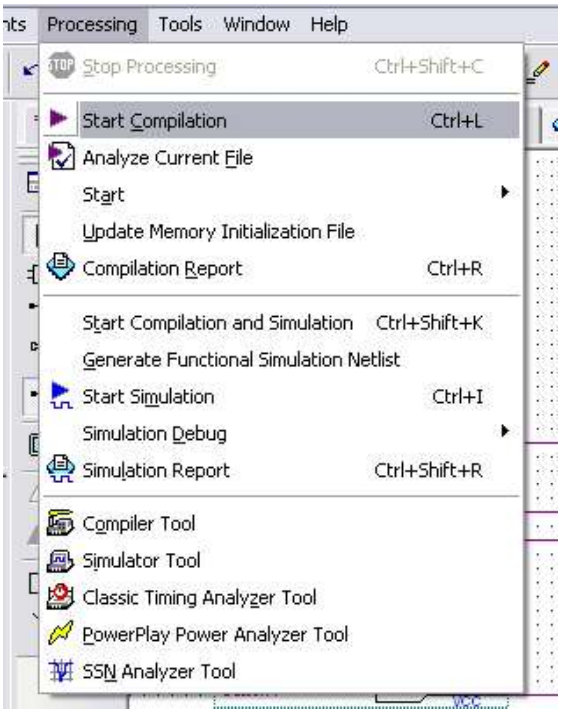
And after connecting all the parts, change the names of the input and output pins. Follow the names in the image below.



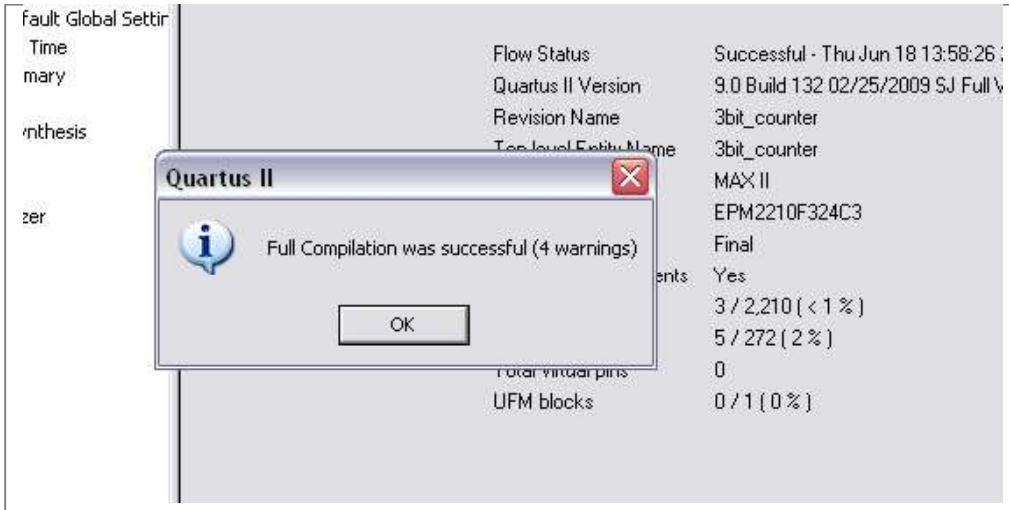
Save the project.



Once everything is connected, you can start compilation of the project.



If you didn't make any mistakes, you should have a successful compilation. If it's not successful, check the error messages to see what went wrong.

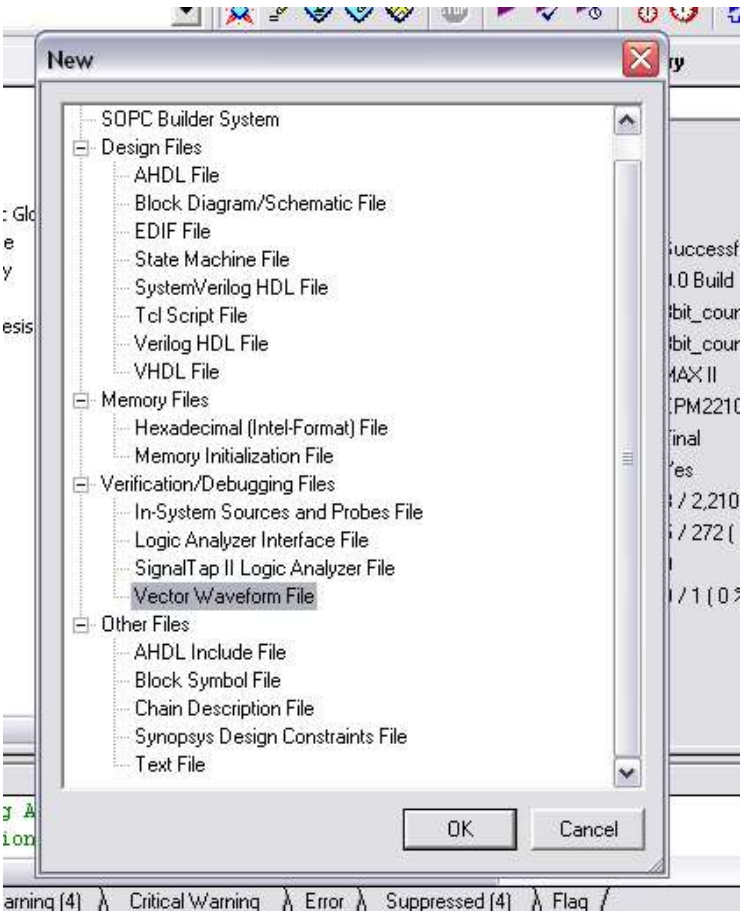

fault Global Settir
Time
mary
nthesis
zer
Flow Status
Successful - Thu Jun 18 13:58:26
Quartus II Version
9.0 Build 132 02/25/2009 SJ Full
Revision Name
3bit_counter
3bit_counter
MAX II
EPM2210F324C3
Final
Yes
3 / 2,210 (< 1 %)
5 / 272 (2 %)
0
UFM blocks
0 / 1 (0 %)
Quartus II
Full Compilation was successful (4 warnings)
OK

Assign pins according to the following diagram and then RERUN THE COMPILATION.

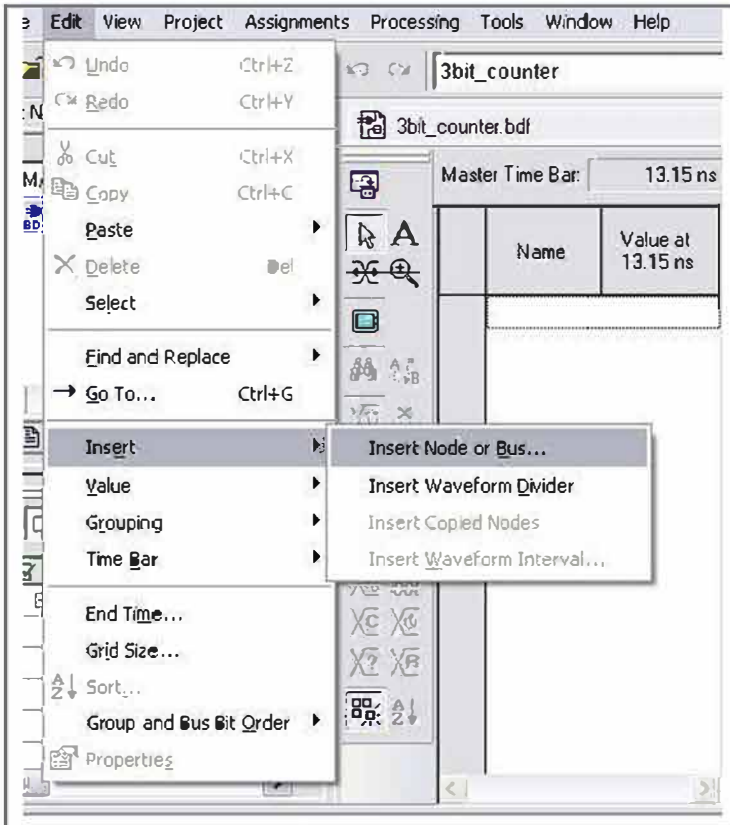
Simulating the Project

Now that we have our project, we can set up some inputs and see what the outputs would be to see if our design is correct.

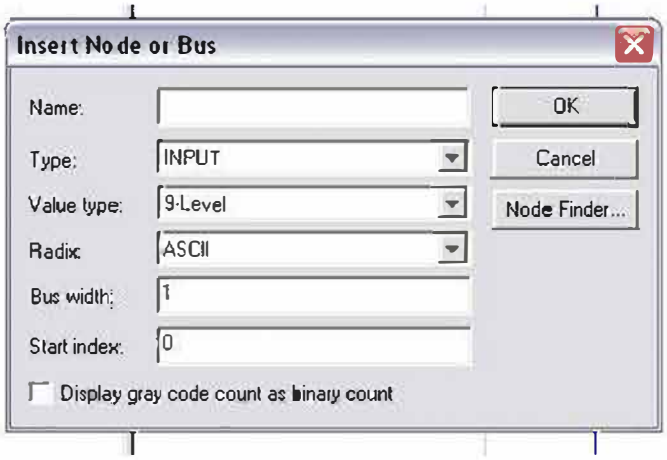
The first step is to set up a vector waveform file. Go to File -> New and choose vector waveform file from the list.

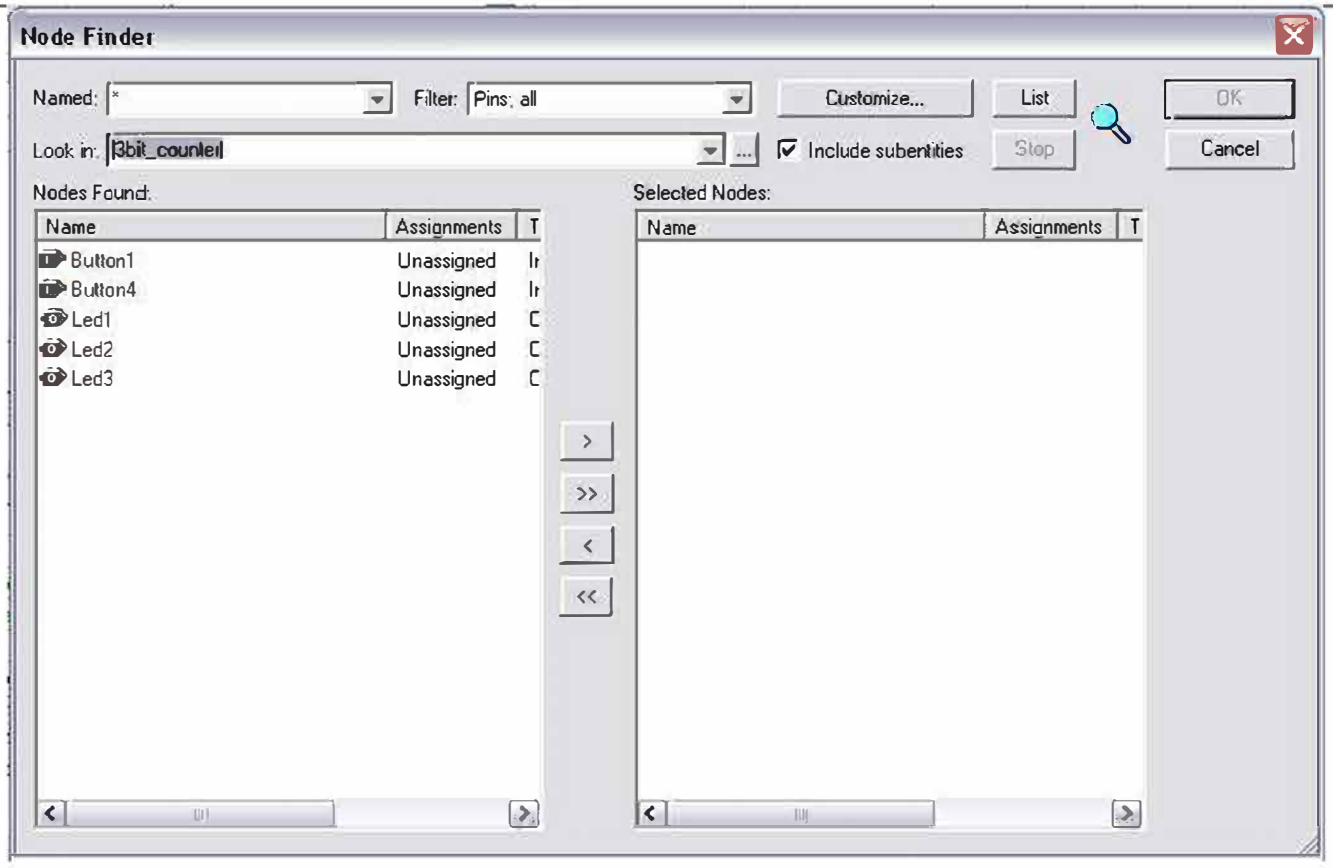


Now we need to add our pins to the waveform file.

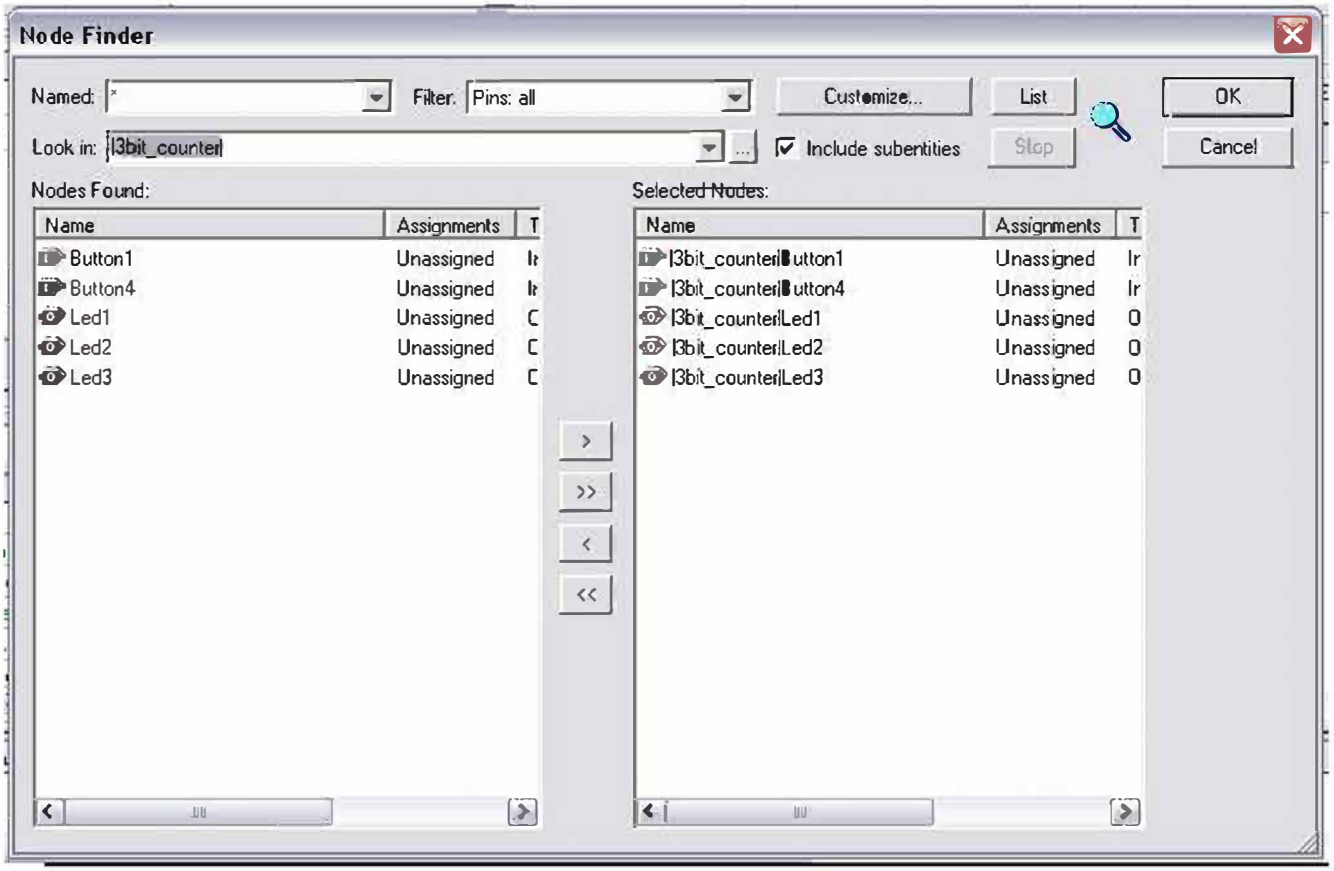


In the "Insert Node or Bus" window, click the Node-Finder button.

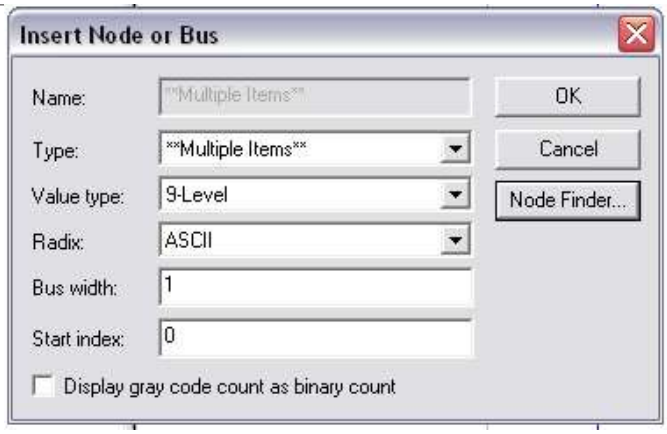




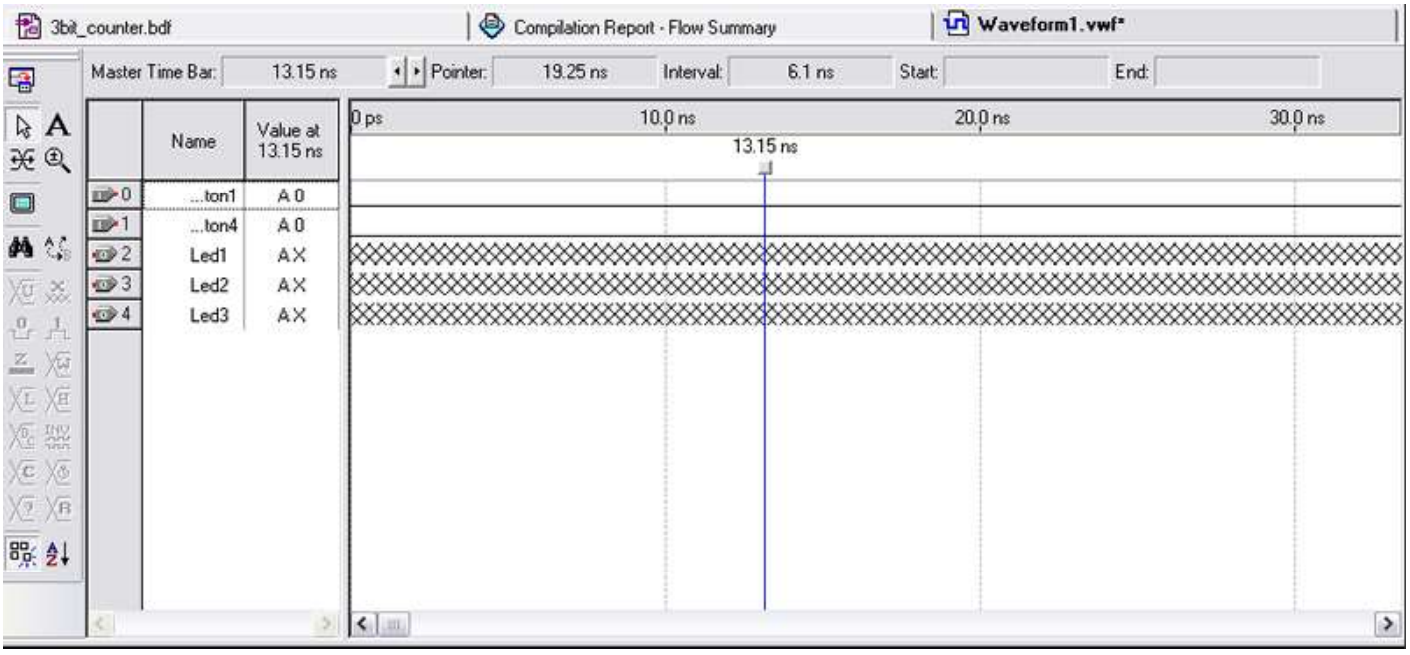
Select all the pins in the Nodes Found window and then click on the > button to put them in the Selected Nodes window.



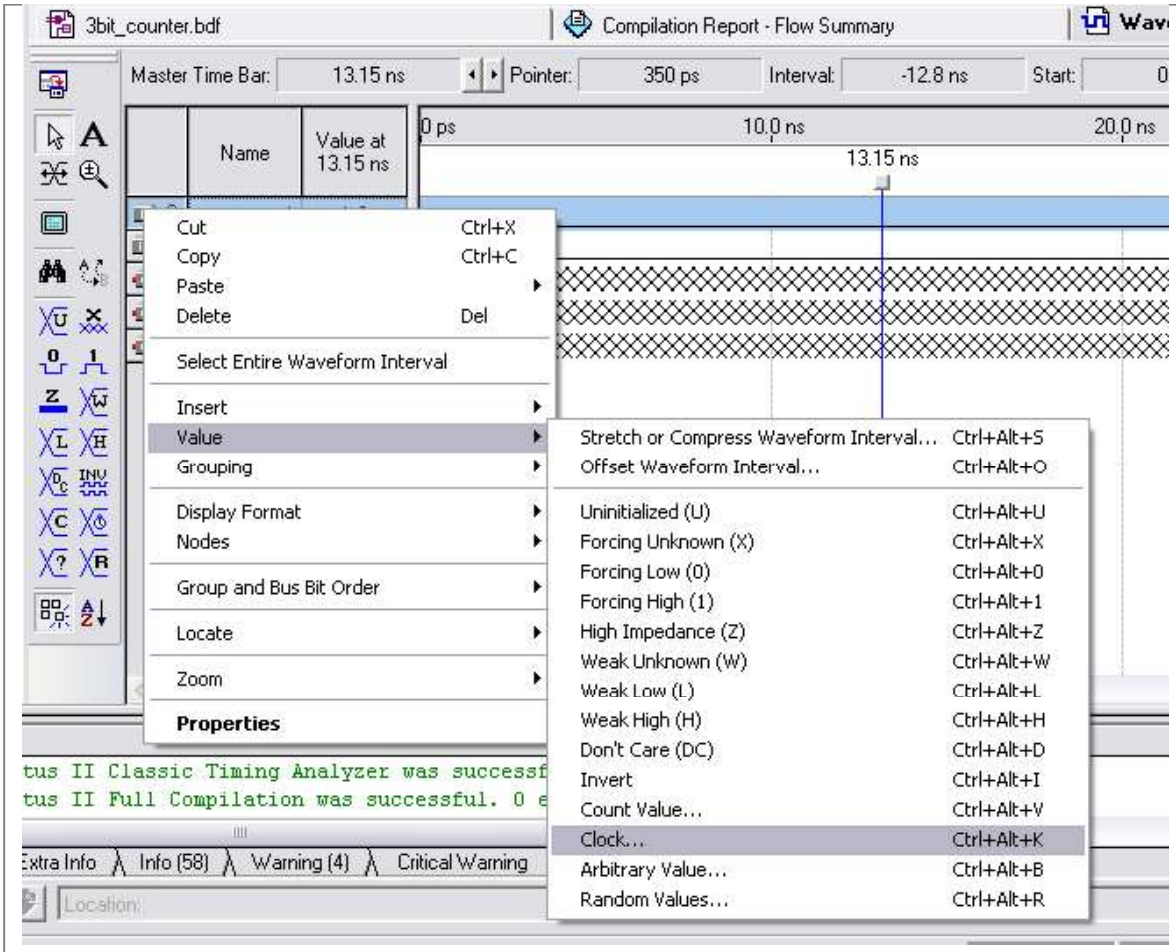
After you press OK, you should be back at the Insert Node or Bus window and it should look like this. Just press OK.



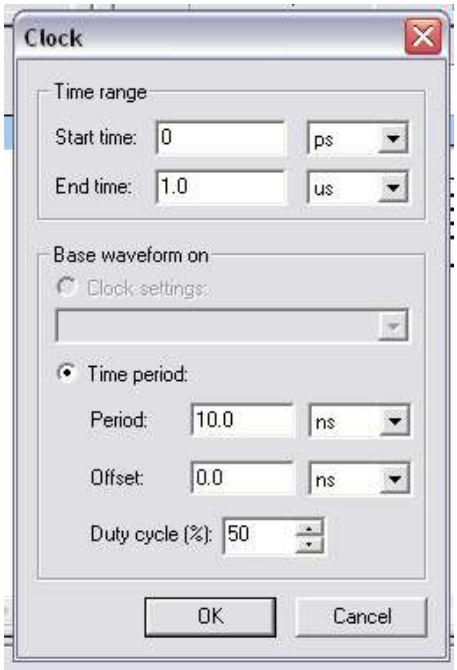
Your waveform file should now look like this.



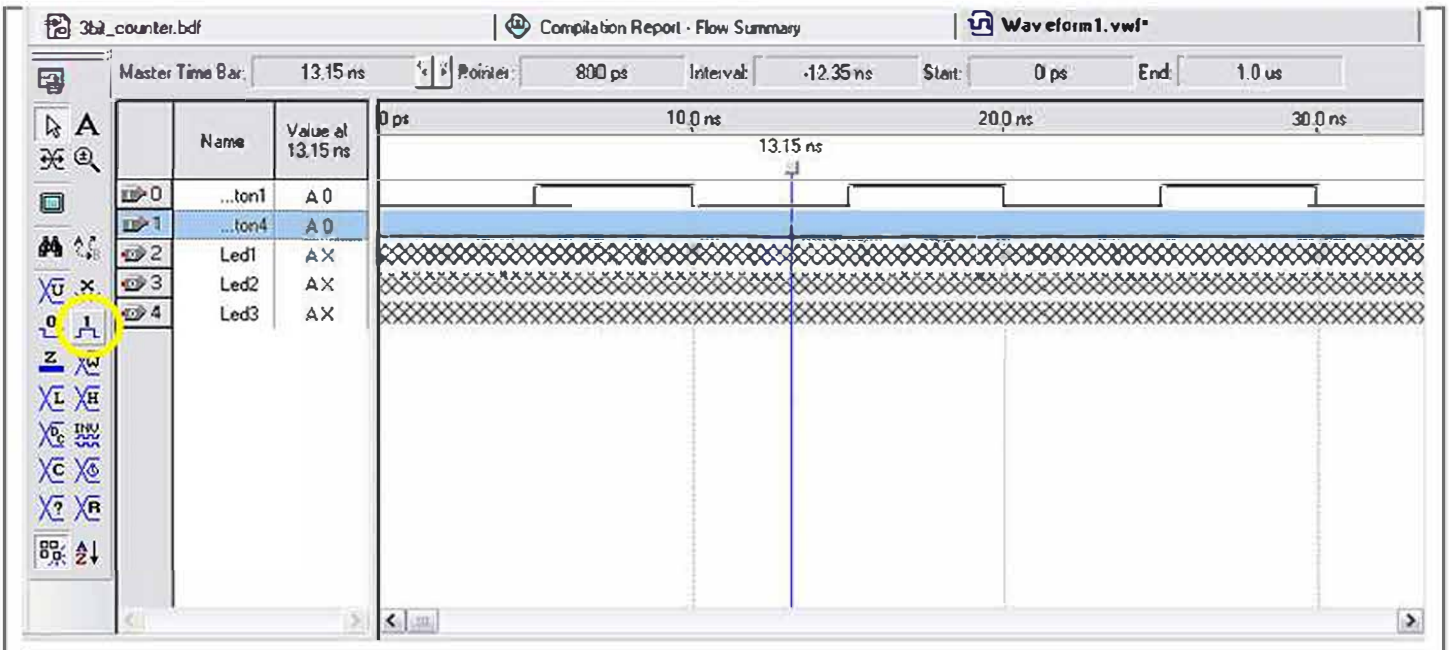
You now need to assign values to your inputs. For button 1, just assign a clock to it. Click on the symbol with the 0 next to it to highlight the entire line. Then assign a clock signal to it.



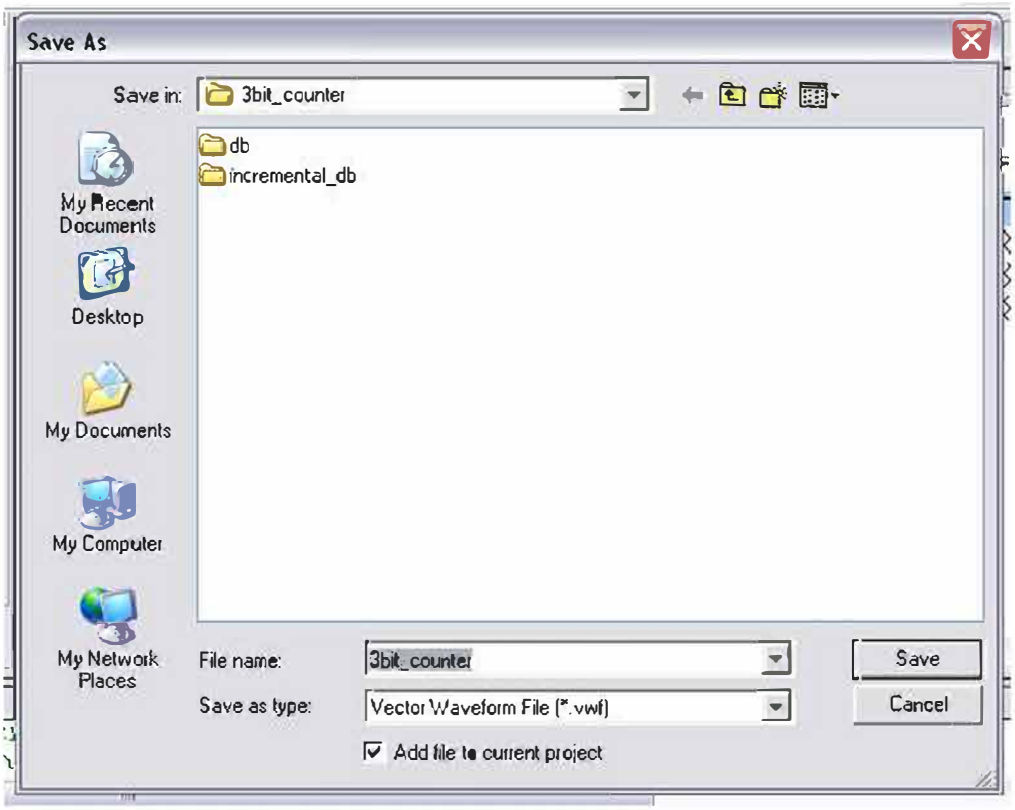
Take the default values for the clock that come up.



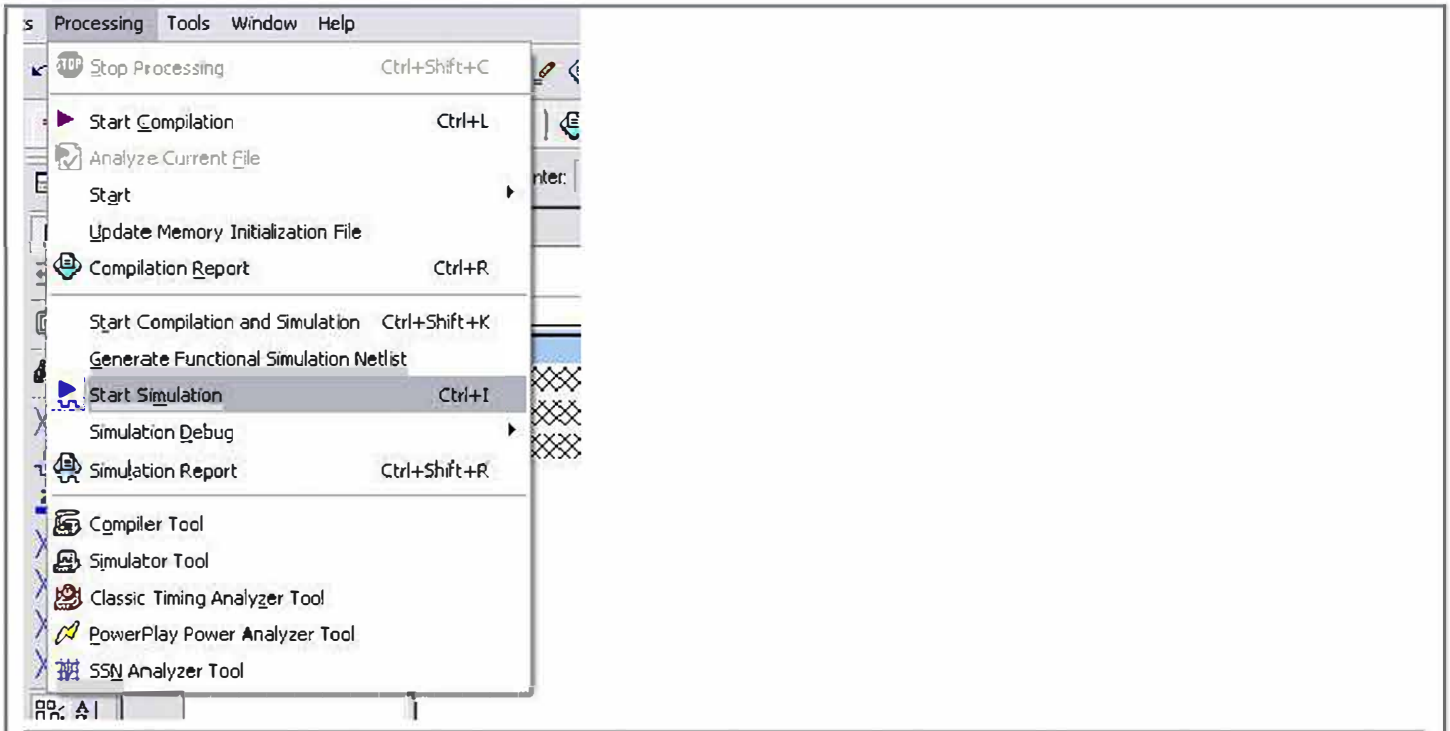
Next highlight the line for button 4 and click the icon to set the signal to 1. (Note the screenshot was taken before the button was pushed.)



Save the file.



Start the simulation



If everything worked, you should get a success message.

The simulation results.