

Installing and Testing Quartus

A. Install the Quartus II v13.0 Web Edition (Free) Program

NOTE: Quartus v13.0 is the last version that will support the Cyclone II chip that is on the DE1 board for 64-bit Windows. It will NOT work for the DE1 board if you use a newer version.

1. Download the two files in the folder from this link:

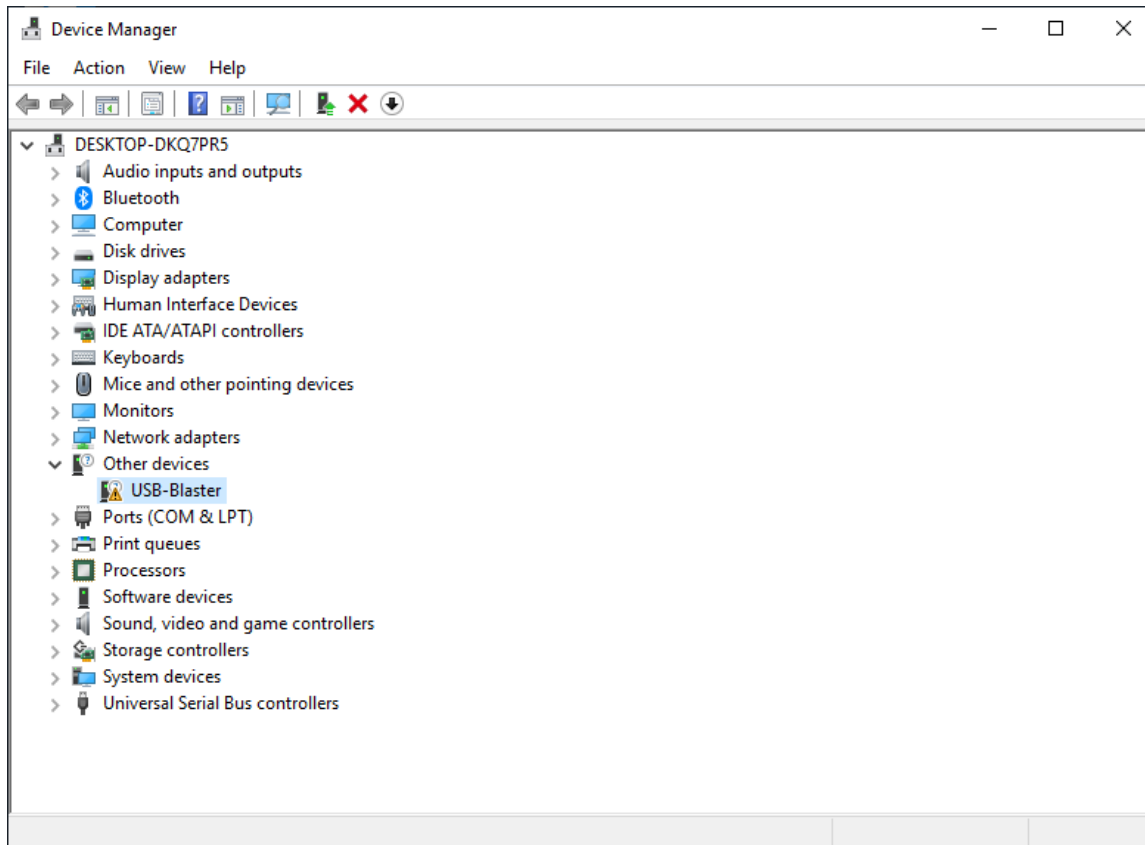
<https://drive.google.com/file/d/1kipCKjppdemW6vUfMND9aOn0TUPIT1FJ/view?usp=sharing>

They are **QuartusSetupWeb-13.0.1.232.exe** and **cyclone_web-13.0.1.232.qdz**. Download time about 10 minutes on a 80 Mbps network.

2. Run the setup program to install Quartus II. This is a 64-bit program so you must have a 64-bit version of Windows. Accept all of the default values during the installation. Installation time about 30 minutes.

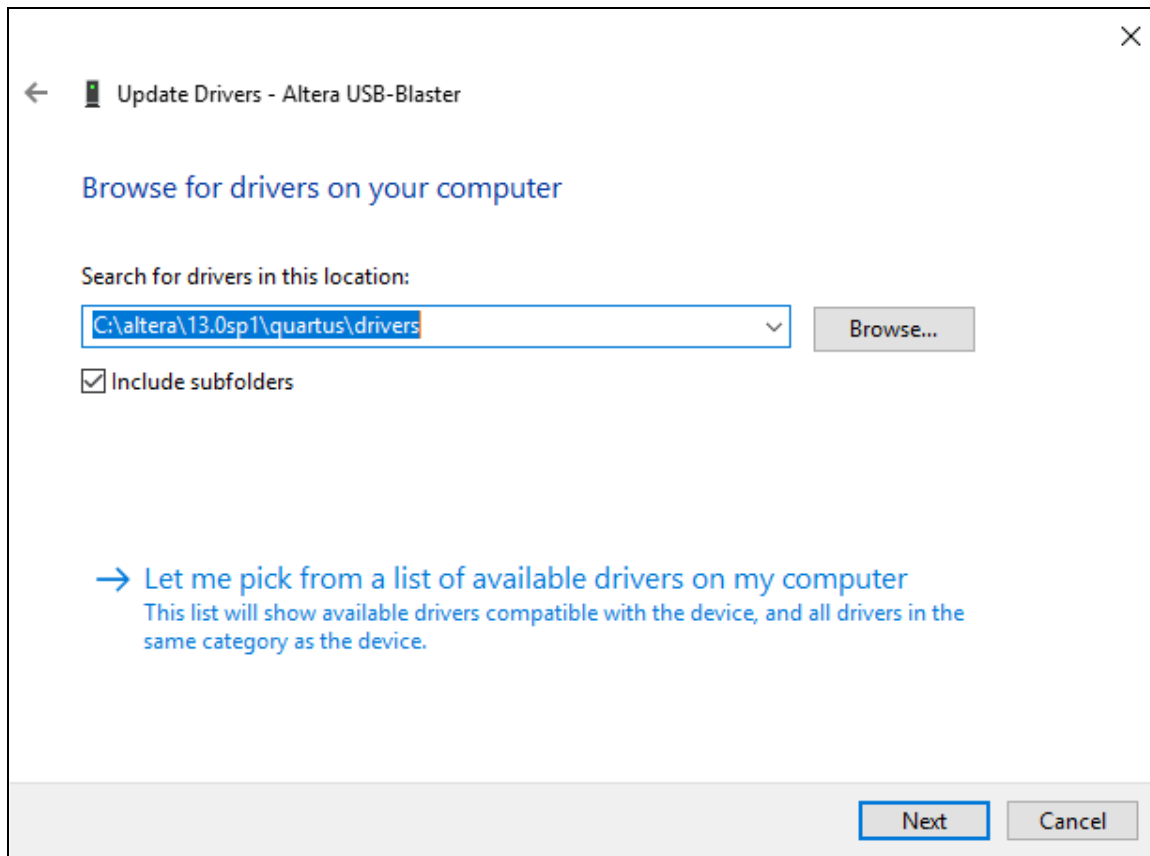
B. Install the USB-Blaster driver

1. Plug the DE1 FPGA board into your PC using only the USB cable. You DO NOT need to plug in the power adapter if using the DE1 board. For what we will be doing the USB can supply sufficient power to the board.
2. In Windows Search, type in **Device Manager** and open the Control Panel Device Manager program. You will see this



The USB-Blaster icon should have a yellow triangle exclamation mark. Right-click on this icon.

3. Select **Update Driver**
4. Select **Browse my computer for driver software**
5. Browse to the location where you have installed Quartus. The default location is at **C:\altera\13.0sp1\quartus\drivers**

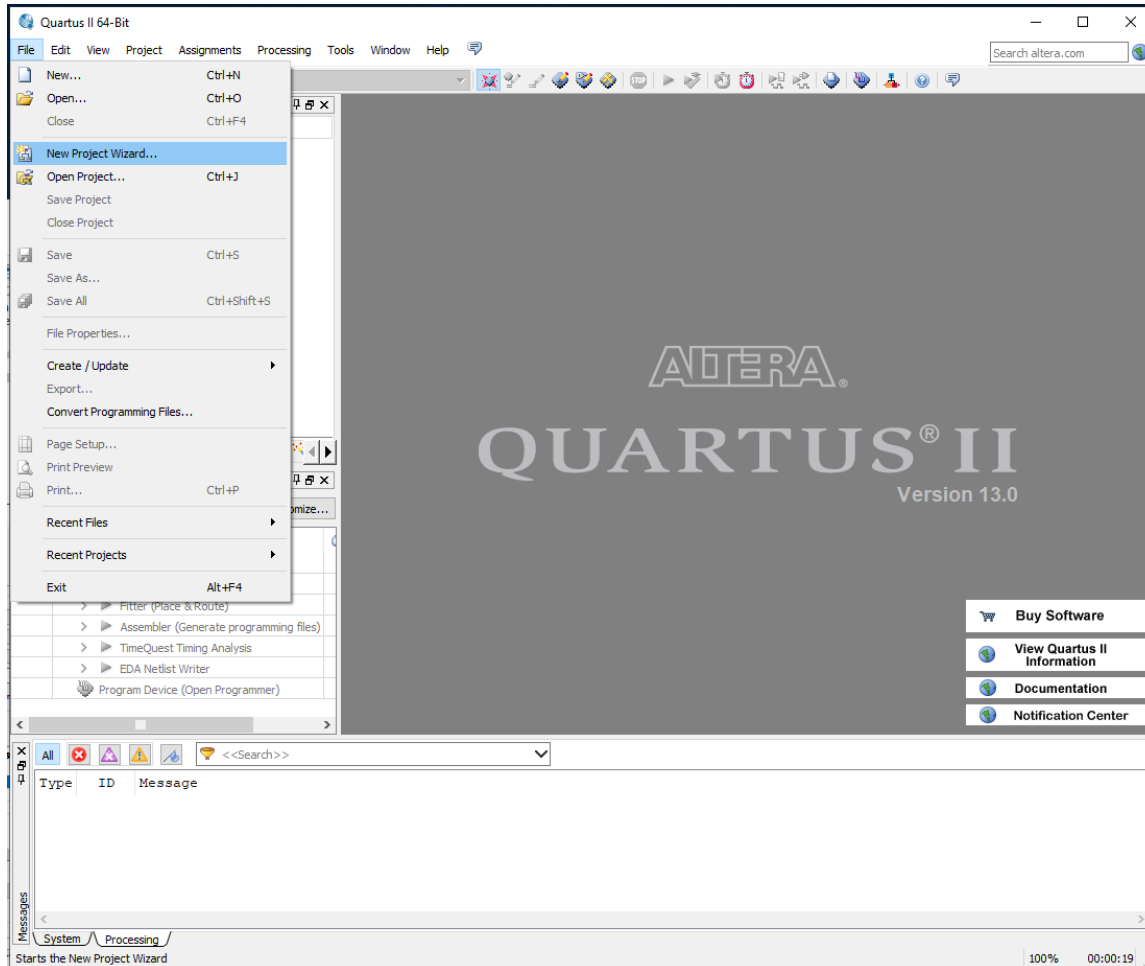


6. Click Next to continue. Driver should install successfully.

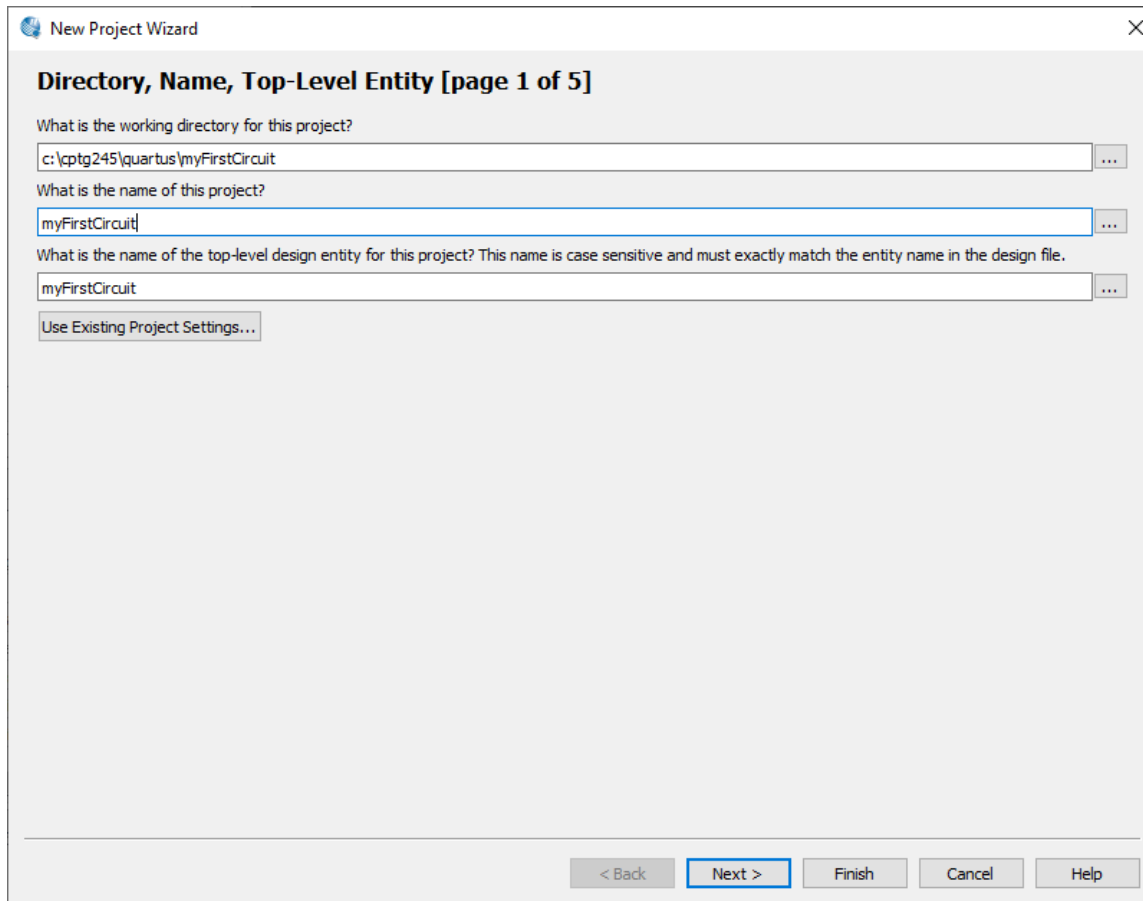
C. Create your first test circuit

i) Create the Project

1. Select from the menu **File | New Project Wizard...**



2. On page 1 of 5 in the New Project Wizard window, type in the location to where you want to store your project and the project name **E:\cptg245\quartus\myFirstCircuit**



Accept and create the location

3. On page 2 of 5 in the New Project Wizard window, you do not have any existing design files to add so just click Next.
4. On page 3 of 5 in the New Project Wizard window, it is very important that you select the correct device or you won't be able to upload your circuit to the FPGA board. For the DE1 board it is:

Family: **Cyclone II**

Device: **EP2C20F484C7**

To help narrow down the list, you can type **ep2c20f4** in the Name filter box.

New Project Wizard

Family & Device Settings [page 3 of 5]

Select the family and device you want to target for compilation.
You can install additional device support with the Install Devices command on the Tools menu.

Device family

Family: Cyclone II

Devices: All

Target device

Auto device selected by the Fitter

Specific device selected in 'Available devices' list

Other: n/a

Show in 'Available devices' list

Package: Any

Pin count: Any

Speed grade: Any

Name filter: ep2c20F4

Show advanced devices HardCopy compatible only

Available devices:

Name	Core Voltage	LEs	User I/Os	Memory Bits	Embedded multiplier 9-bit elements	PLL	Glob.
EP2C20F484C6	1.2V	18752	315	239616	52	4	16
EP2C20F484C7	1.2V	18752	315	239616	52	4	16
EP2C20F484C8	1.2V	18752	315	239616	52	4	16
EP2C20F484I8	1.2V	18752	315	239616	52	4	16

Companion device

HardCopy:

Limit DSP & RAM to HardCopy device resources

< Back Next > Finish Cancel Help

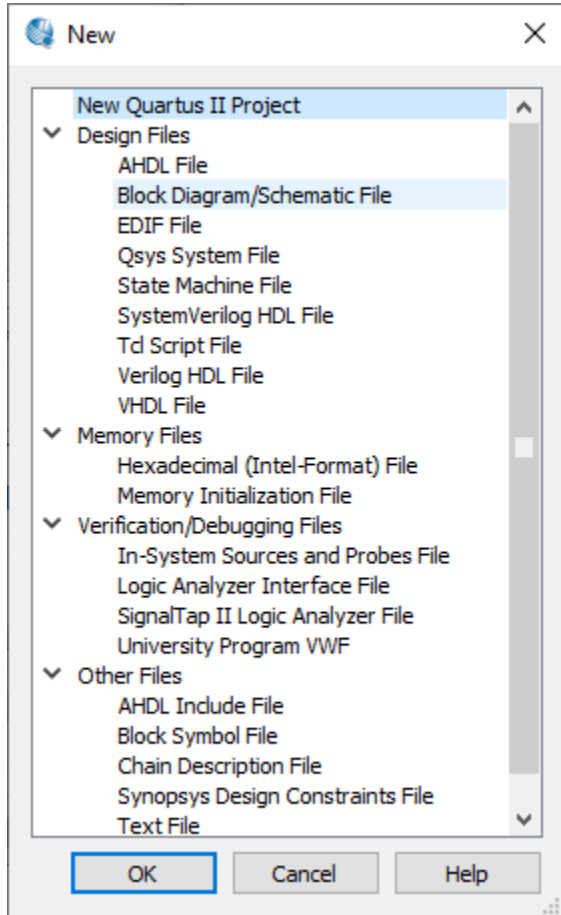
Click Next.

- On page 4 of 5 in the New Project Wizard window just click Next.
- On page 5 of 5 in the New Project Wizard window look at the summary and just click Finish.

ii) Draw the Circuit

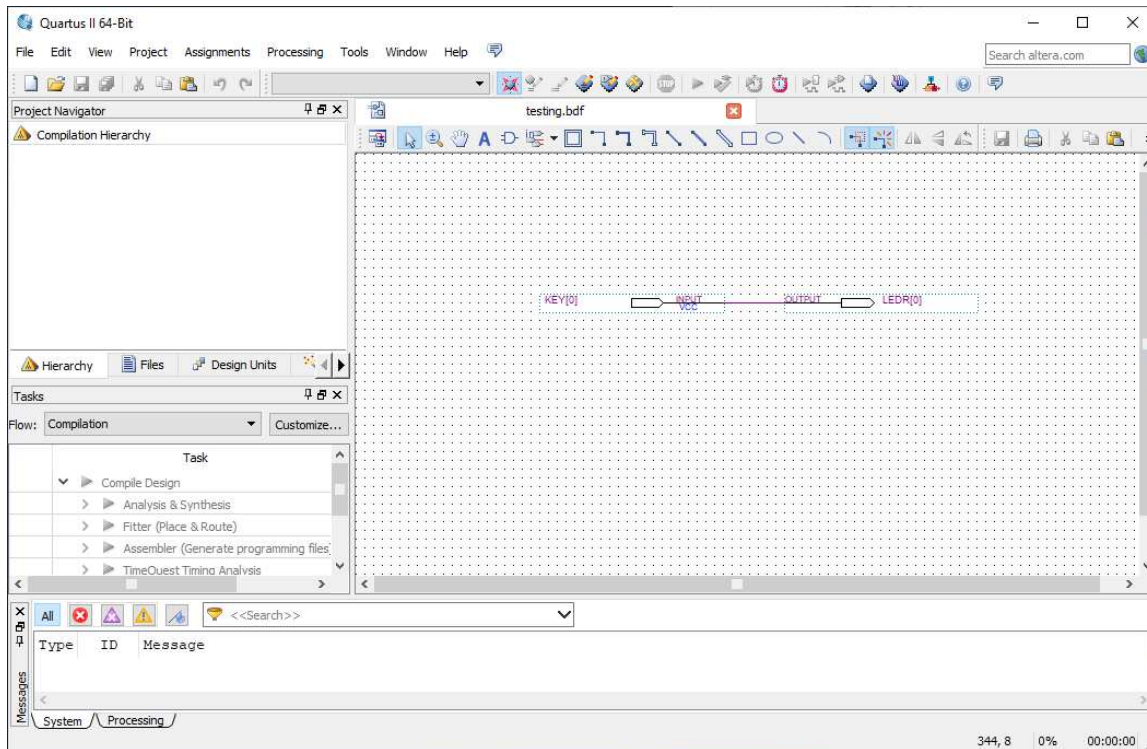
7. Select from the menu File | New or click .


Select **Block Diagram/Schematic File**

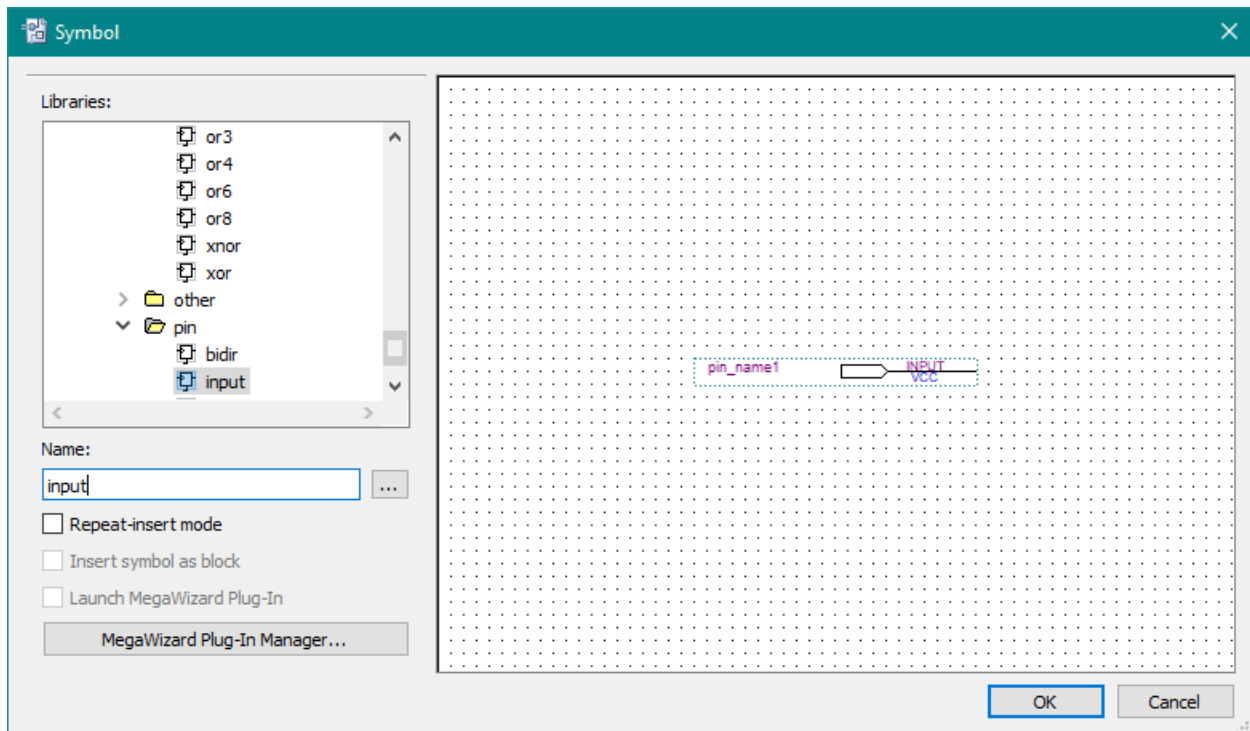


Click OK.

8. Draw this circuit with just one input pin and one output pin. Connect the input pin to the output pin.



To draw the circuit, either click on the Symbol Tool  or double click on an empty area on the drawing board to bring up the symbol library window.




You can either manually search through the library components or type in the name “input” to bring up


the input component.

Click OK and you are back in the drawing board window. Click your mouse on an empty area on the drawing area to place the input pin.

Do the same thing for the output pin.

Using the multipurpose tool , draw a connection line between the two ends of the input and output pins.

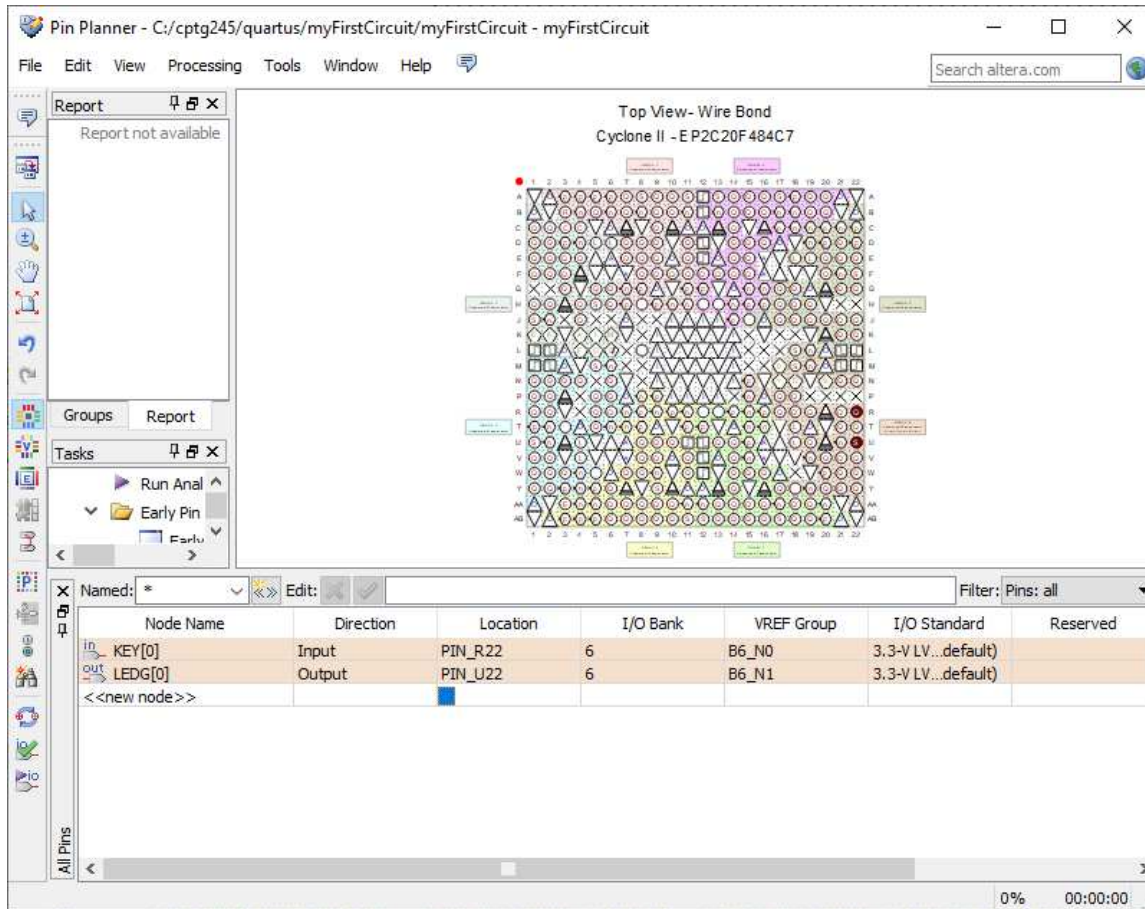
Refer to the Lab 1 document for more details on how to use the various drawing tools.

9. Double click on the input pin to change the input pin name to **KEY[0]**.
10. Double click on the output pin to change the output pin name to **LEDG[0]**.
11. Save the circuit using the default name **myFirstCircuit** which is the same as the project name that you gave at the beginning.
12. Select from the menu **Processing | Analyze Current File** or click . You should not get any errors.

iii) Pin Mappings

Do either step 13 or 14 to connect (map) the pins to the actual hardware devices. Step 13 uses a graphical tool and is slower. Step 14 uses a text editor and is much faster.

13. Select from the menu **Assignment | Pin Planner** or click  .

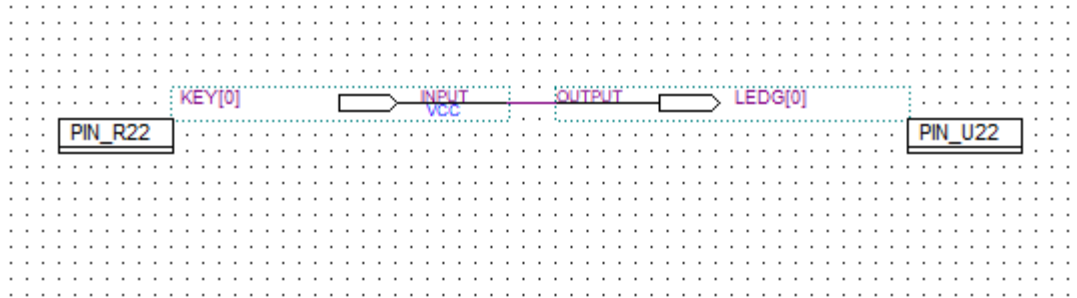


for the KEY[0] row under Location, type in **R22** (the PIN_R22 will pop up automatically) and press **Enter**.

for the LEDG[0] row under Location, type in **U22** (the PIN_U22 will pop up automatically) and press **Enter**.

Close the Pin Planner window.

You should now see the annotation PIN_R22 and PIN_U22 attached to the input and output pins respectively. If you don't see this then you did not assign the pins correctly.



A faster option. Instead of using the Pin Planner in step 13 you can use a faster and easier method described in step 14 below.


14. (This step replaces step 13 and is much faster for future pin mappings) Using Windows' File Explorer, browse to your project's directory at **E:\cptg245\quartus\myFirstCircuit**. There should be a file called **myFirstCircuit.QSF**. Make sure it is the one with the file type **QSF**. Right-click on this file and select **Open with...** This is a regular text file so select **Notepad** to open this file.

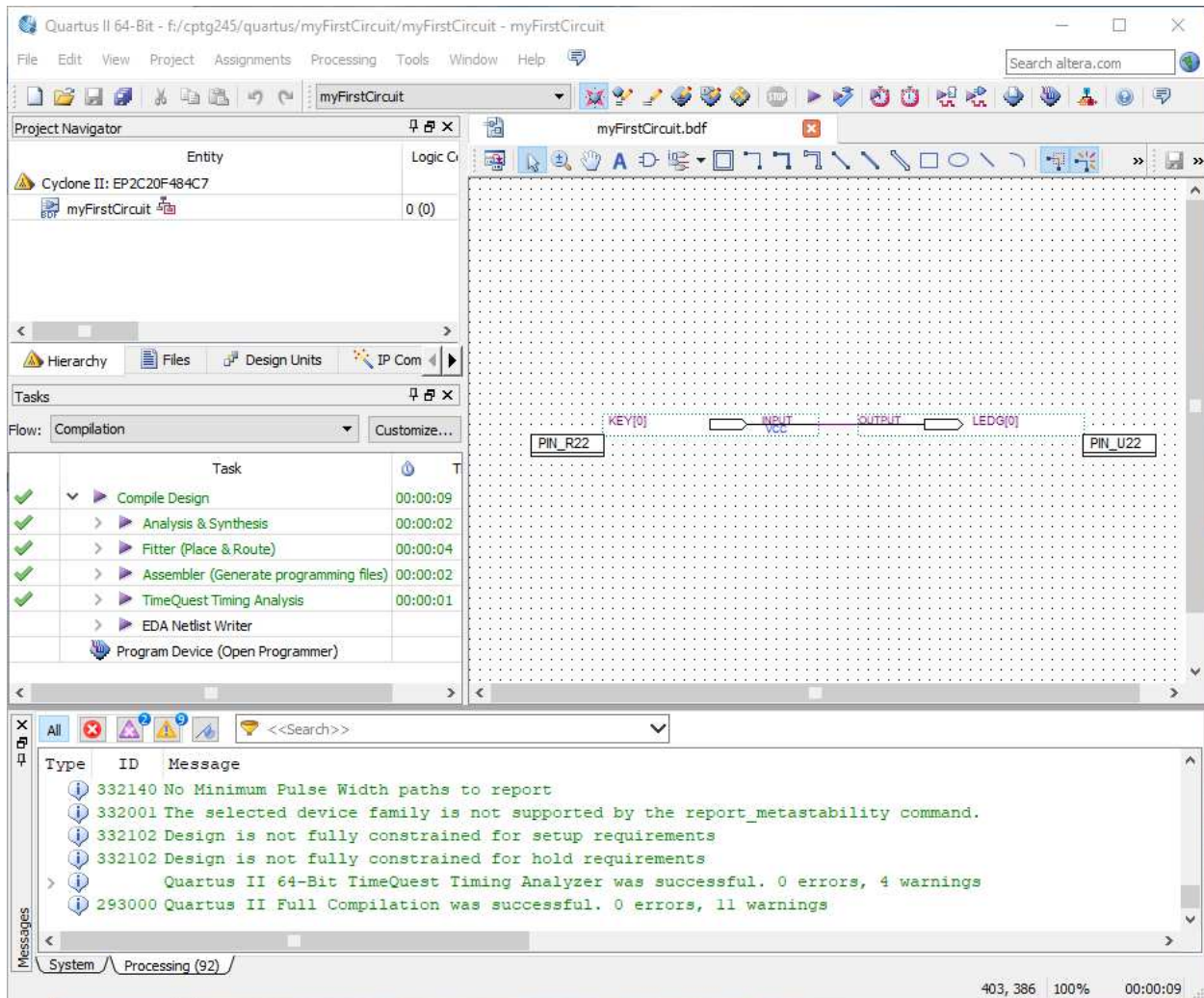
Copy the following two lines to the end of this file

```
set_location_assignment PIN_R22 -to KEY[0]  
set_location_assignment PIN_U22 -to LEDG[0]
```

Save the file and exit out.

iv) Synthesis

15. Select from the menu **Processing | Start Compilation** or click . You will get many warnings. You can ignore them. But you should not get any errors. You should see five green check marks and the message "Full Compilation was successful" in the message window at the bottom.



Quartus II 64-Bit - f:/cptg245/quartus/myFirstCircuit/myFirstCircuit - myFirstCircuit

File Edit View Project Assignments Processing Tools Window Help

myFirstCircuit

myFirstCircuit.bdf

Entity Logic C
Cyclone II: EP2C20F484C7
myFirstCircuit 0 (0)

Hierarchy Files Design Units IP Com

Tasks

Flow: Compilation Customize...

Task	Time
Compile Design	00:00:09
Analysis & Synthesis	00:00:02
Fitter (Place & Route)	00:00:04
Assembler (Generate programming files)	00:00:02
TimeQuest Timing Analysis	00:00:01
EDA Netlist Writer	
Program Device (Open Programmer)	

Messages

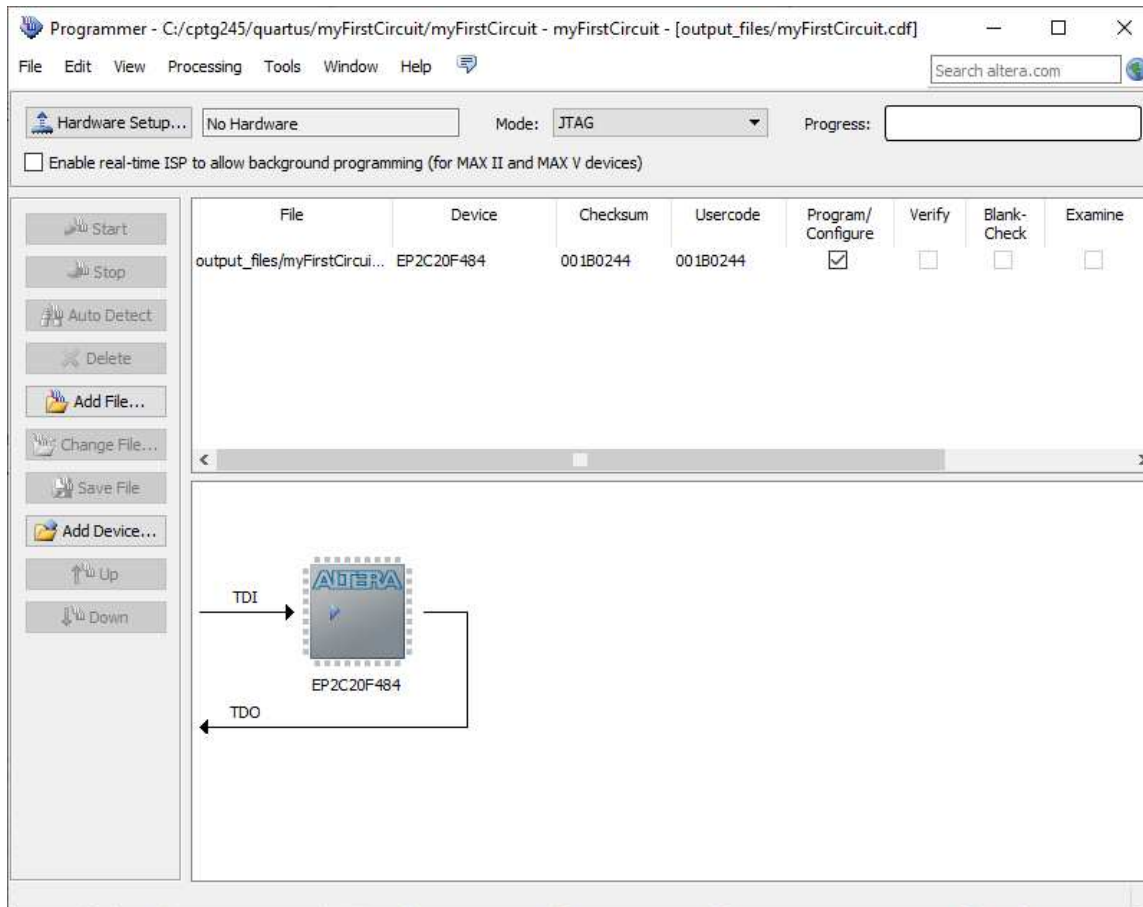
Type	ID	Message
Warning	332140	No Minimum Pulse Width paths to report
Warning	332001	The selected device family is not supported by the report_metastability command.
Warning	332102	Design is not fully constrained for setup requirements
Warning	332102	Design is not fully constrained for hold requirements
Success		Quartus II 64-Bit TimeQuest Timing Analyzer was successful. 0 errors, 4 warnings
Success	293000	Quartus II Full Compilation was successful. 0 errors, 11 warnings

System Processing (92)

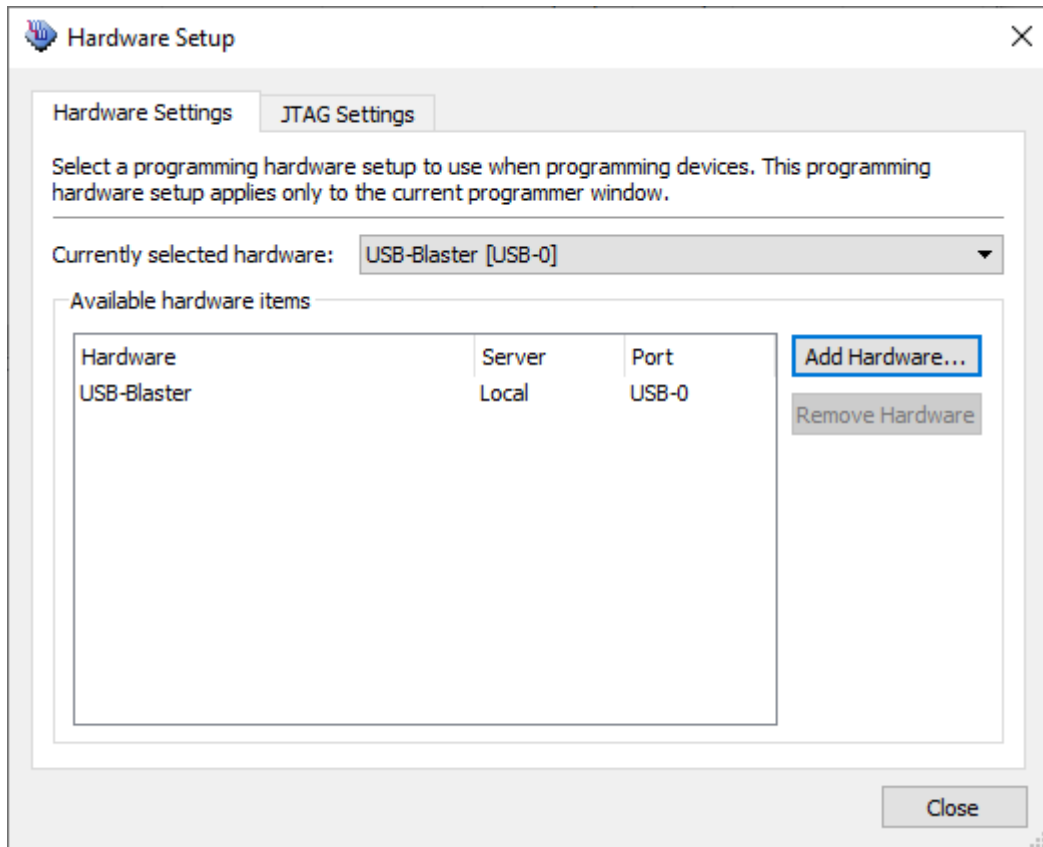
403,386 100% 00:00:09

D. Upload your circuit to the FPGA chip

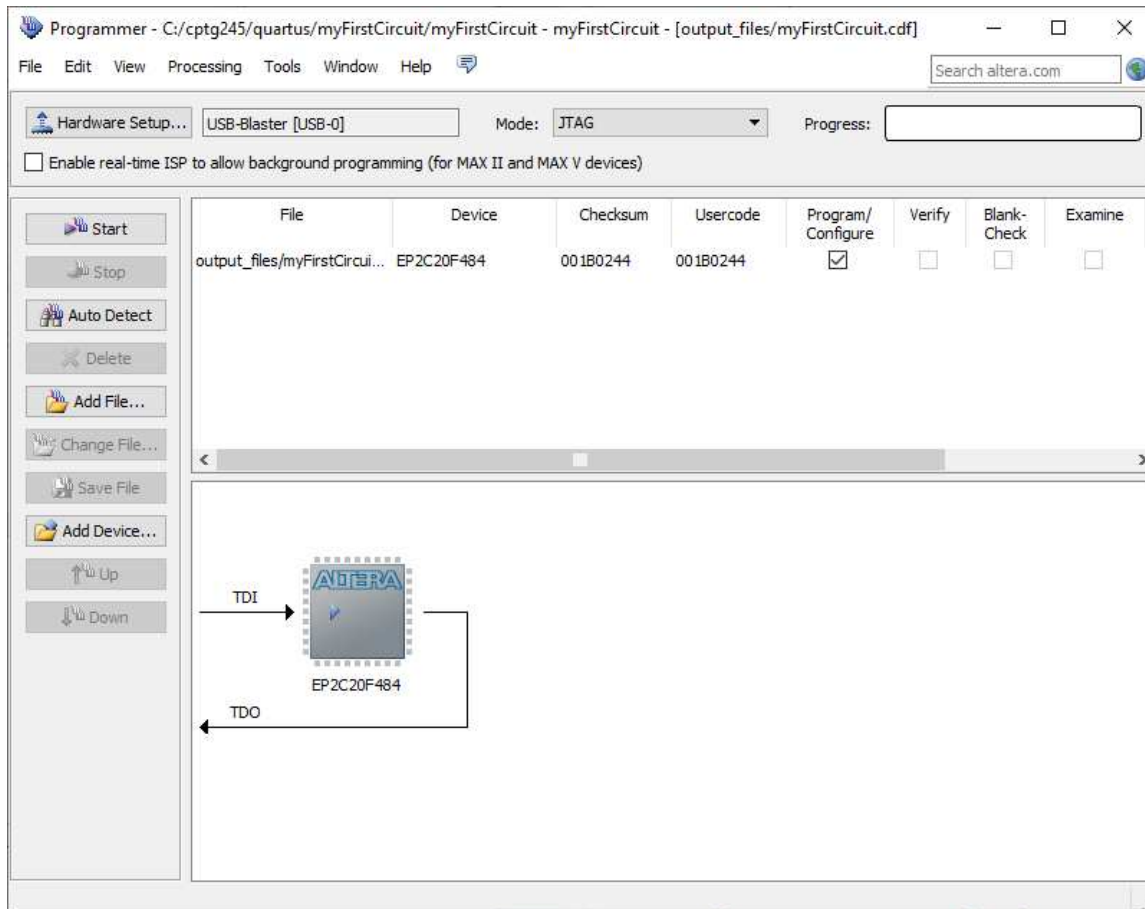
1. If you have not already done so, connect the FPGA board to your computer using only the USB cable. You DO NOT need to plug in the power adapter. For what we will be doing the USB can supply sufficient power to the board.
2. Select from the menu **Tools | Programmer**.



3. If it says "No Hardware" then make sure that you have installed the USB-Blaster device driver following the instructions in step B above, and then go to step 4 next. Otherwise go to step 6.
4. Click on "Hardware Setup" and select "USB-Blaster" from the Currently selected hardware



5. Click "Close". DO NOT click "Add Hardware"
6. In the Programmer window, it should now show



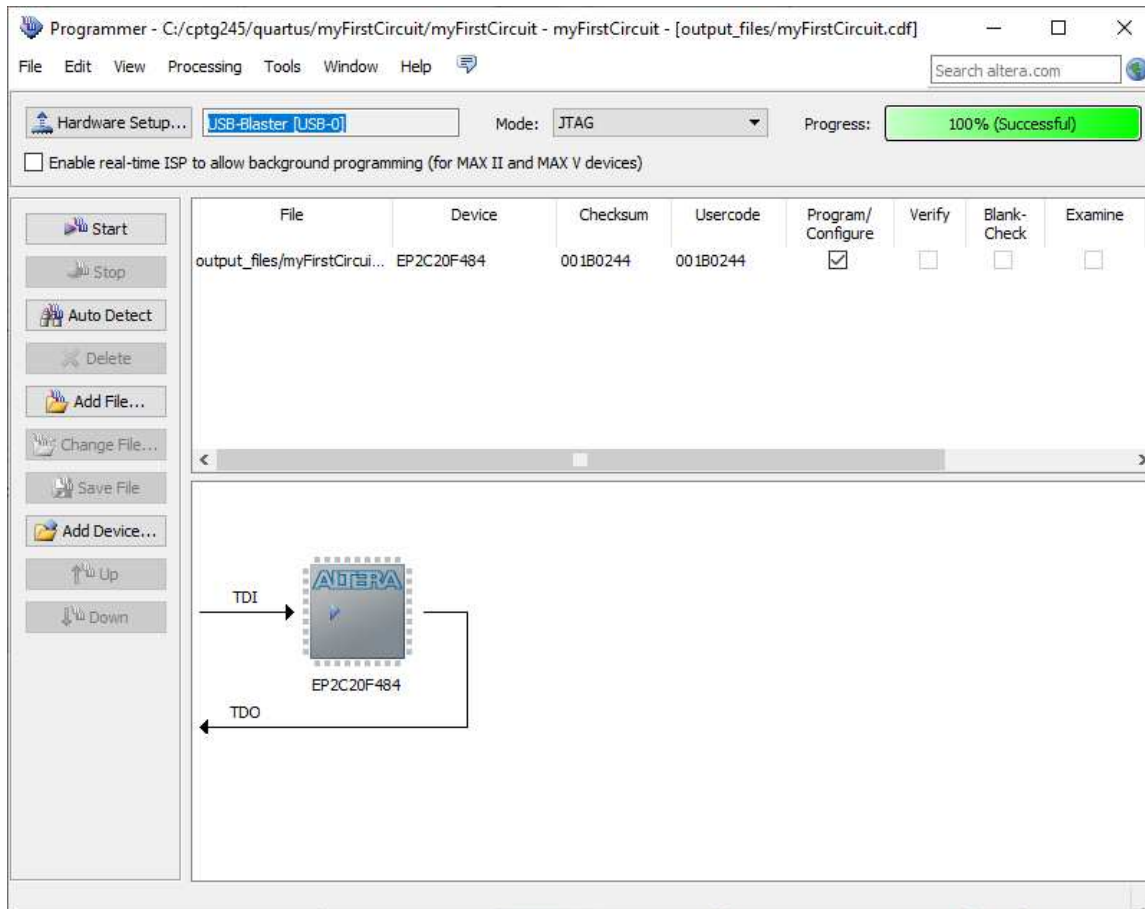
In the top portion of the window under the column File it should show the correct name of your project.

Under the Device column it should show the device name EP2C20F484.

The box under the Program/Configure column should be checked.

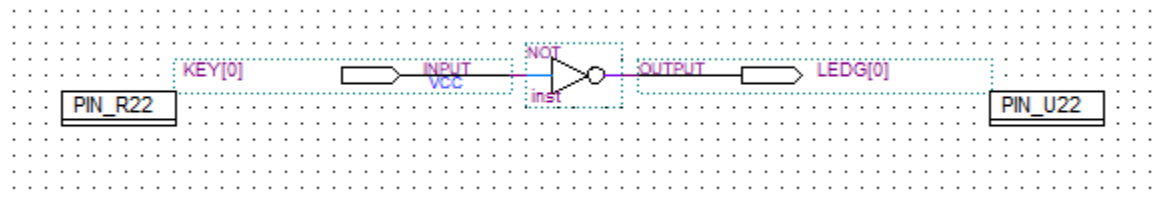
The bottom half of the window should show a picture of the EP2C20F484 chip.

7. Click the **Start** button. You should see the green 100% successful message bar when the circuit has been uploaded to the FPGA board.



E. Testing your circuit on the FPGA board

1. The green LED (LED0) on the FPGA board should be on. Press KEY0 and the green LED will turn off.
2. If you want it so that the green light is initially off and turns on when you press the button, you need to invert the signal going from the input to the output. In other words you need to add an inverter component between your input and output. The inverter component is called the NOT gate. Search for the NOT gate in the symbol library and add that to your circuit drawing like this.



Make sure you delete the original connection line between the input and output.

3. Do the full compilation again and then upload this new circuit to the board.
4. The green LED (LED0) on the FPGA board should now be initially off. Press KEY0 and the green LED will turn on.