

## EE 231L

### Getting Started with QUARTUS II

QUARTUS II is a software package from Altera which you will use to compile and simulate your PLD designs. Here is a brief tutorial for using QUARTUS II. For a more detailed tutorial, look in the which is available on the Altera website.

1. Find the QUARTUS II program. It should be a button in the lower-left corner of the screen next to the Start Menu. Open the QUARTUS II program.



2. From the **File** menu choose **New Project Wizard...** Enter the name of the working directory for the new project, or make a subdirectory for this lab if you haven't already done so. (Altera generates multiple file for each project. Having an organized file system will be VERY helpful when combining the separate elements for the final computer for EE 231 Lab). Enter the name of the **project** and the top level design file (**entity**). For your convenience, you can just type in the name of the project and ALTERA will use the same name for the top-level entity.

3. QUARTUS II will ask you for files to be included in this project. Leave this blank, and click **Next**.

4. Select which device you want to use for the implementation of your project. For this course we will use the EPF6016TC144-3 chip. First, go to the **Family** drop-down menu, and select **FLEX6000**. Click the button next to **Specific device selected in 'Available devices' list**. To narrow down the list of devices, go to the **Package** drop-down menu and select **TQFP**, then to the **Pin count** menu and select **144**, then to the **Speed grade** menu and select **3**. From the list of **Available devices**, select **EPF6016TC144-3**. Finally, click **Finish**.

5. From **File** menu choose **New**. For **Device Design Files**, select **AHDL**, then click **OK**. Now choose **File | Save As** and save with the same name as the top-level entity from Part 2.

6. Enter the AHDL program in the text editor window, and choose **File | Save**.


7. Choose **Processing | Start Compilation**. After the project compiles with no errors, you can check the design via a simulation.


8. You can verify your program by running a simulation in Quartus II. There are two types of simulation: Functional and Timing. A functional simulation is used to verify that the output of the circuit is what you want. A timing simulation will verify that the circuit will work at the speed you want. For now, you will run a functional simulation. To do this, you must first generate a functional simulation netlist. Choose **Processing | Generate Functional Simulation Netlist**.

You need to generate a file in which you tell Quartus II what inputs you are using, and what outputs you want to check. This is called a waveform file. Choose **File | New | Other Files | Vector Waveform File** then choose **File | Save As** and save with the same name as the project name (with a .vwf extension). Select **Edit | End Time...** to set the length of the simulation (the default 1.0  $\mu$ s is fine for now), and **Edit |**

**Grid Size...** to specify the time grid of your simulation. You may need to adjust the size of the window by clicking on **View | Fit in Window...** Click on **Assignments | Settings...** and under **Simulator Settings** choose the simulation mode: Functional.

9. Now that you have a waveform file, you need to add the inputs and outputs of your design. Choose **Edit | Insert Node or Bus** then **Node Finder...** Select **Pins: all** for the Filter setting, then click **List**. Highlight the variables you want to view on the simulation and move them to the **Selected Nodes:** window. Click **OK**. Choose the **Edit | End Time...** button and increase the time to 16 us. You now should have a diagram with inputs and outputs versus time. The inputs are all 0, and the outputs undetermined. Click the small magnifying glass (tool bar in the left side of the screen) several times to zoom out to the full picture.

10. There are two ways to enter simulation waveforms. You can draw the input waveform with the mouse, or you can use the **Edit** menu. For example a circuit with inputs called A, B, C: To make the A input high for the second half of the time, set the cursor on A near the center (8 us), press and hold the left mouse button, drag the cursor to the end (16 us) and release. Then click on the "1" menu option in the left-hand menu. A quick way to get alternating highs and lows on C is as follows. Click and highlight all of C by clicking in the **Name** column. Next from the vertical menu on the left-hand click on **Clock** (represented by the icon ) and enter the **Period** to 1 us. Repeat for input B, changing the **Period** to 2 us. Save this file.

11. Choose **Processing | Start Simulation** (represented by the icon  in the pull-down menu). If there were no problems during the simulation, there will be a message indicating that the simulation was successful and the output will change from undetermined to the desired values.

12. Now you should have a simulated output. Verify that the output is what you expected.