EE102 – Digital Circuit Logic

Introduction to Altera DE2 and Quartus II Design Software

1. Objectives

- Learn basic operation and functionality of DE2 and Quartus software needed to implement digital circuits.
- Test a binary counter that has been pre-designed.

2. Altera DE2 Board

The laboratory assignments will use the Altera DE2 board shown in Figure 1. DE2 can be used to implement sophisticated digital circuits, and the Quartus II software that you will use is a complete design package used even by professional engineers.

Note:

Since you are new to digital design, do not be surprised if it appears intimidating at first. Do not despair; we will selectively introduce the features as needed while ignoring others that we do not need. As the semester progresses, you will understand more and more features, and by the end of the semester, you will be able to design complex circuits using many features of the Quartus II software.

You would need some pre developed files named **Lab0.zip** to complete the project. Download them from the course web page or ask from the TA.

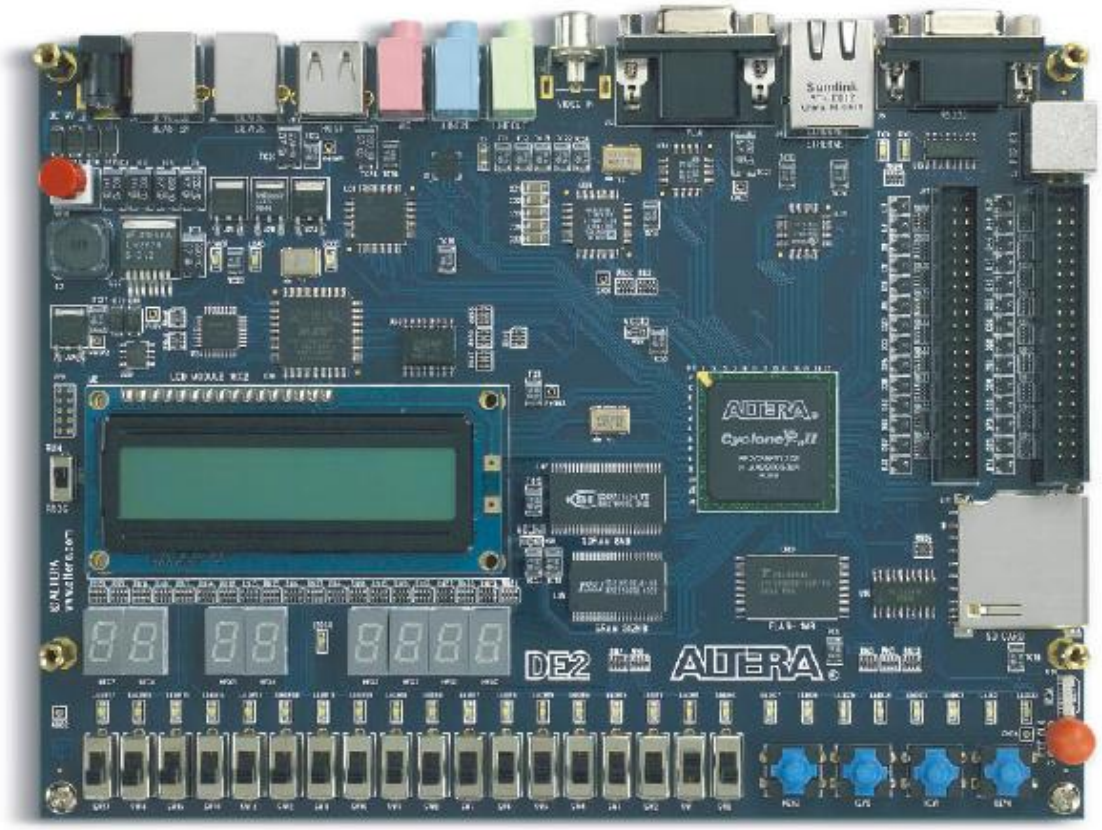

Figure 1 – Altera DE2 Development and Education Board

Note the set of switches SW0 – SW17, and push buttons KEY0-KEY3 that provide the inputs to circuits you will build, as well as light-emitting diodes LED0-LED17, eight 7-segment display units, and a LCD display unit that can be used for observing the outputs.

The red switch on the left side is the power switch, and the USB port on top left connects the board to a computer. Specifications and features of DE2 **

3. A Simple Binary Counter

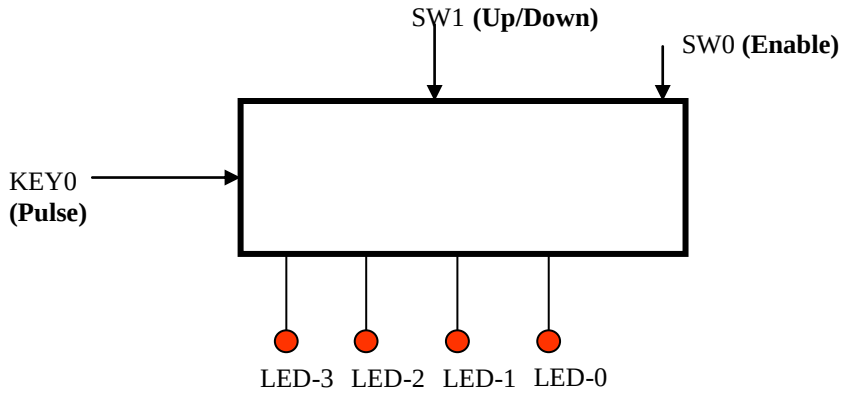

Figure 2 – Schematic diagram of a binary counter

For this laboratory session we have pre-designed a simple binary counter that you will use. A schematic diagram of the counter is shown above. It is designed to count and display the number of pulses that is applied by pressing **KEY0** input. The counter has to be enabled by setting **SW0** high. Setting of **SW1** decides whether the counter counts up or down.

4. Getting Started with Quartus II Software

This tutorial introduces few basic features of the Quartus II software. It shows how the software can be used to design and implement a circuit specified by means of a schematic diagram. It makes use of the Graphical User Interface (GUI) to invoke the Quartus II commands. Doing this tutorial, the reader will learn about:

- Creating a project and importing a pre-designed circuit
- Programming and configuring the FPGA chip on the Altera's DE2 board

4.1. Creating a project and importing a pre-designed circuit

Step 1: Starting the Quartus II software.

Click **All Programs → ECE → Altera → Quartus II 9.0 Web Edition → Quartus II 9.0 Web Edition** from the Windows **Start** menu to open the Quartus II software.

You should see a display similar to the one in Figure 3. This display consists of several windows that provide access to all the features of Quartus II software, which the user selects with the computer mouse. Most of the commands provided by Quartus II software can be accessed by using a set of menus that are located below the title bar. For example, in Figure 3 clicking the left mouse button on the menu named **File** opens the menu shown in Figure 4. Clicking the left mouse button on the entry **Exit** exits from Quartus II software. In general, whenever the mouse is used to select something, the **left** button is used. Hence we will not normally specify which button to press. In the few cases when it is necessary to use the **right** mouse button, it will be specified explicitly.

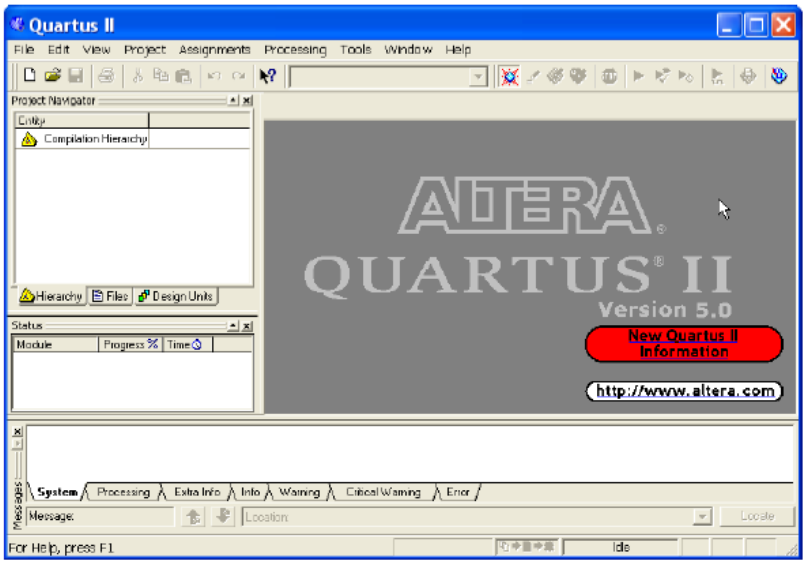

Figure 3 – The main Quartus II display

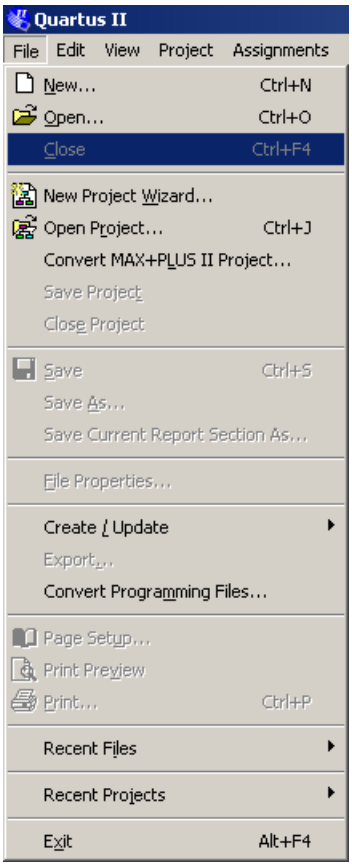

Figure 4 – An example of the File menu

For some commands it is necessary to access two or more menus in sequence. We use the convention **Menu1 → Menu2 → Item** to indicate that to select the desired command the user should first click the left mouse button on **Menu1**, then within this

menu click on **Menu2**, and then within **Menu2** click on **Item**. For example, **File → Exit** uses the mouse to exit from the program. Many commands can be invoked by clicking on an icon displayed in one of the toolbars. To see the command associated with an icon, position the mouse over the icon and a tooltip will appear that displays the command name.

Step 2: Starting a New Project

Each logic circuit, or subcircuit, being designed with Quartus II software is called a project. The software works on one project at a time and keeps all information for that project in a single directory (folder) in the file system. To begin a new logic circuit design, the first step is to create a directory to hold its files. To hold the design files for this tutorial, we will use a directory named **Lab_0**. The running example for this tutorial is a simple circuit that counts in binary.

To start working on a new design we first have to define a new design project. Quartus II software makes the designer's task easy by providing support in the form of a wizard. Create a new project as follows:

(a) Select **File → New Project Wizard** to reach the window in Figure 5, which indicates the capability of this wizard. Click **Next** to get the window shown in Figure 6.

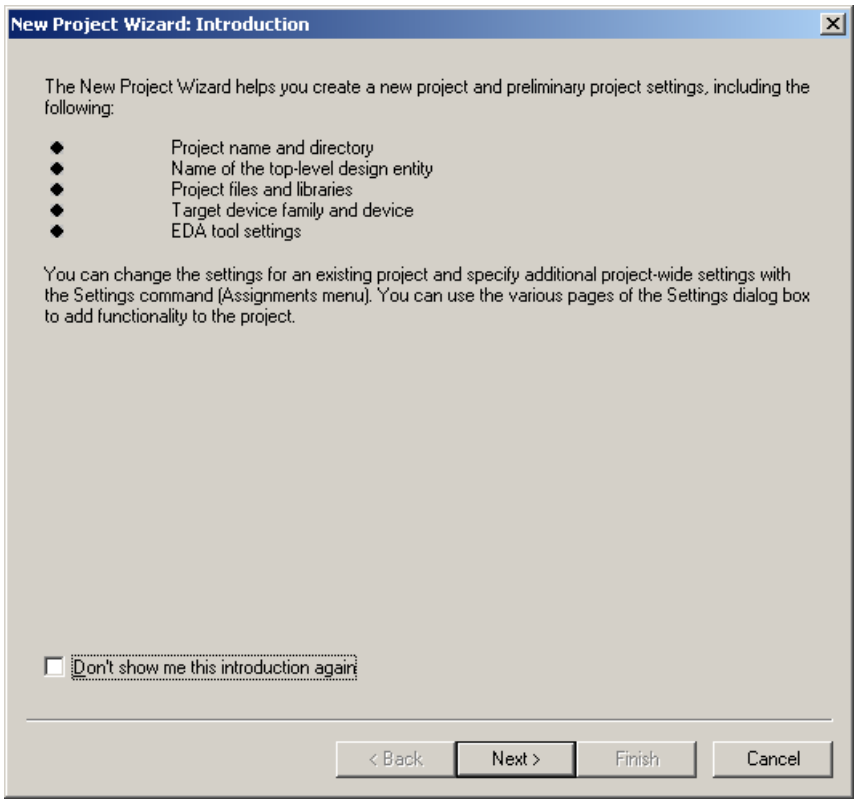

Figure 5 – Tasks performed by the wizard

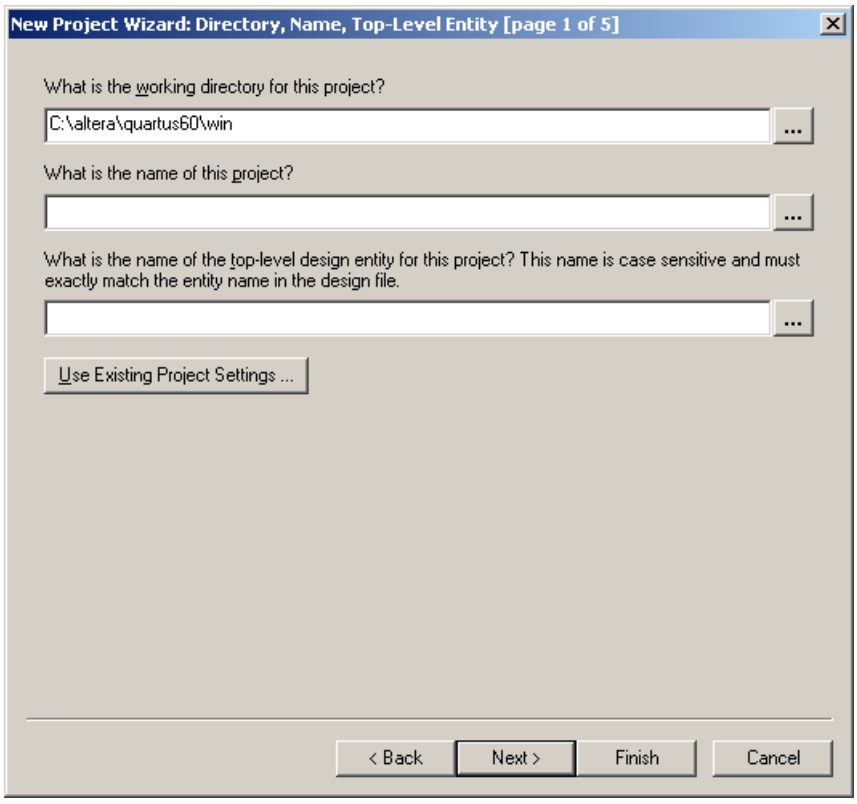

Figure 6 – Creation of a new project

B) Set the working directory to be **U:\ECE102\Lab0**; of course, you can use some other directory name of your choice if you prefer. The project must have a name, which is usually the same as the **top-level design entity** that will be included in the project. Choose **binary_counter** as the name for both the **project** and the **top-level entity**, in creation window shown in Figure 6. Then click the **Next** button.

Since we have not yet created the directory Quartus II software displays the pop-up message asking if it should create the desired directory. Click **Yes**, which leads to the window in Figure 7.

Note:

If you do not have an engineering account you may need to save your data on the Desktop.

Step 3: Importing a design

The wizard makes it easy to specify which existing files (if any) should be included in the project. These files will be added to your project folder but will not be copied into that.

Click the button labeled "**...**" in the **File Name:** field to open the **Select File** dialog box. The files that you need are in the **Lab0** folder that you download from the web.

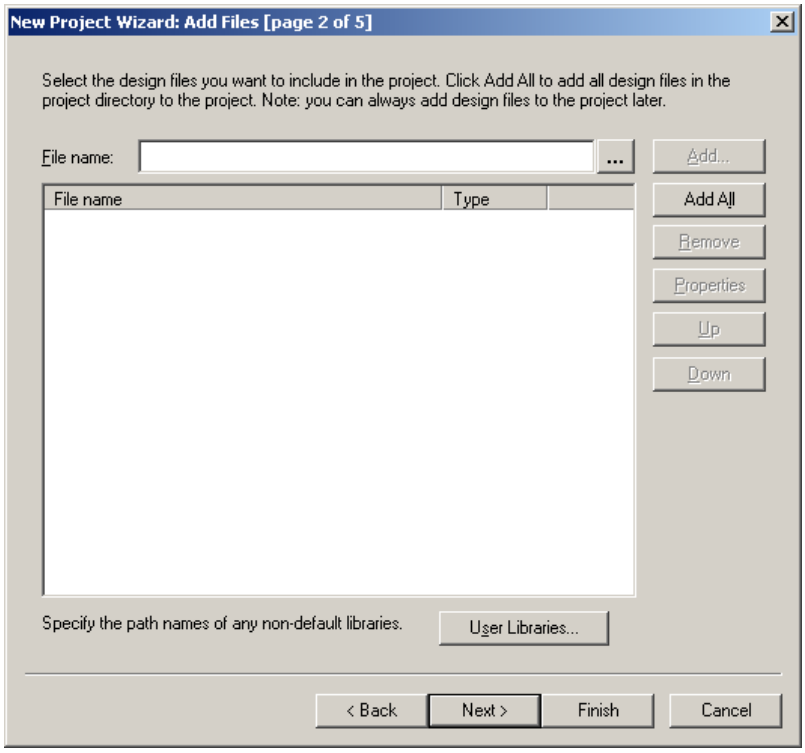

Figure 7 – The wizard for including user-specified design files

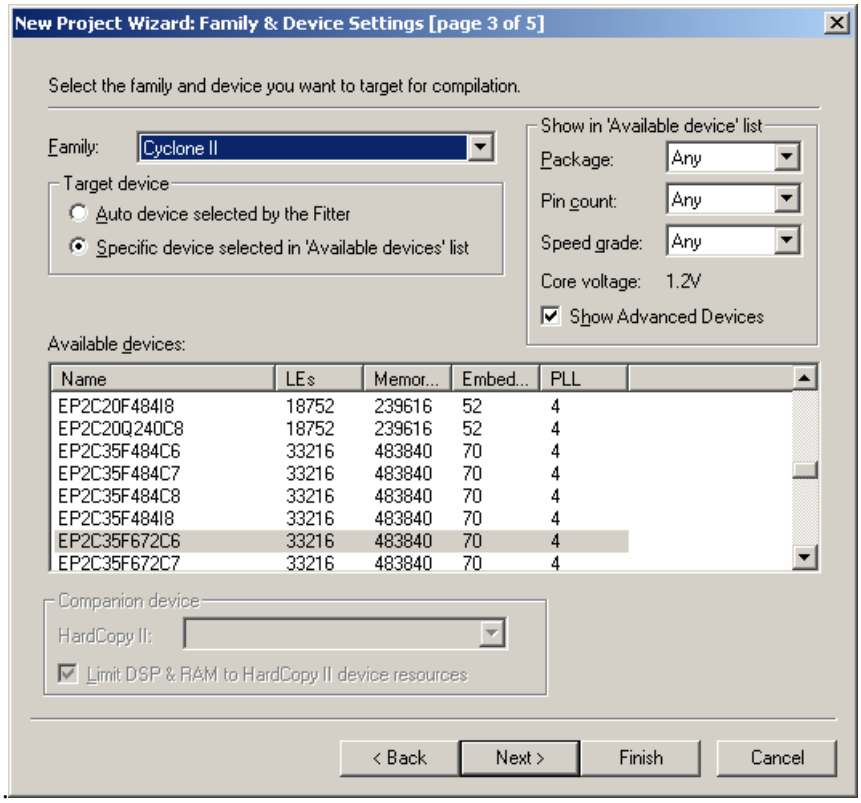

Figure 8 – Choose the device family and a specific device

Then from the **Select File** dialog box select the **binary_counter.bdf** file and then click the **Open** button. Then the path to the file will be displayed on the File name: text box. Click the **Add** and file will appear under the **File name** list.

Similarly add the **binary_counter.sof** file.

Click **Next**, which leads to the window in Figure 8.

Step 4: Since Quartus II software is used with different types of boards and devices, we have to specify the type of device in which the designed circuit will be implemented. Choose **Cyclone II** as the target device family. We can let Quartus II software select a specific device in the family, or we can choose the device explicitly. We will take the latter approach. From the list of **Available devices:**, choose the device called **EP2C35F672C6** which is the FPGA used on Altera's DE2 board.

Click **Next**, which opens a window to select other tools that a designer may use.

Click **Next** again, since we do not use such tools in this class.

Step 5: A summary of the chosen settings appears in the screen similar to that shown in Figure 9 but with binary_counter specified as the new project.

Click **Finish** button, which returns to Quartus II window of Figure 3, with **binary_counter** specified as the new project in the display title bar.

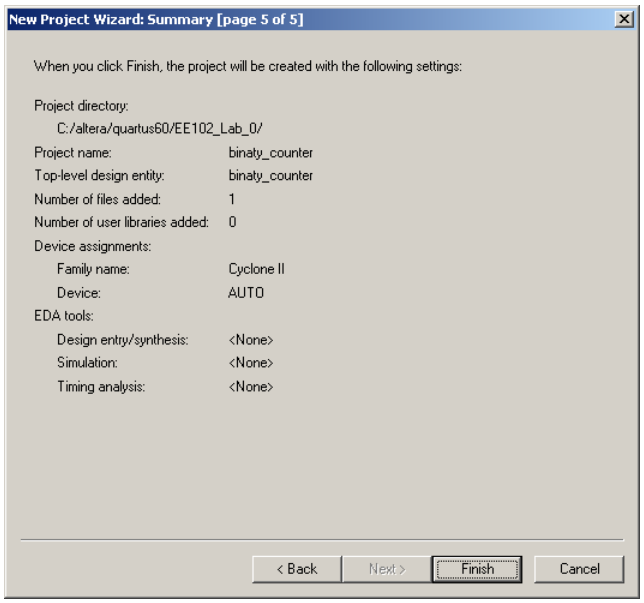

Figure 9 – Summary of project settings

4.2. Programming and Configuring the Device

The FPGA device must be programmed and configured to implement the designed circuit. For this lab, we will be using a mode called 'JTAG Programming' to program the device.

Step 1: Flip the RUN/PROG switch into the **RUN** position.

Step 2: Select **Tools → Programmer** from the menu to reach the window in Figure 10.

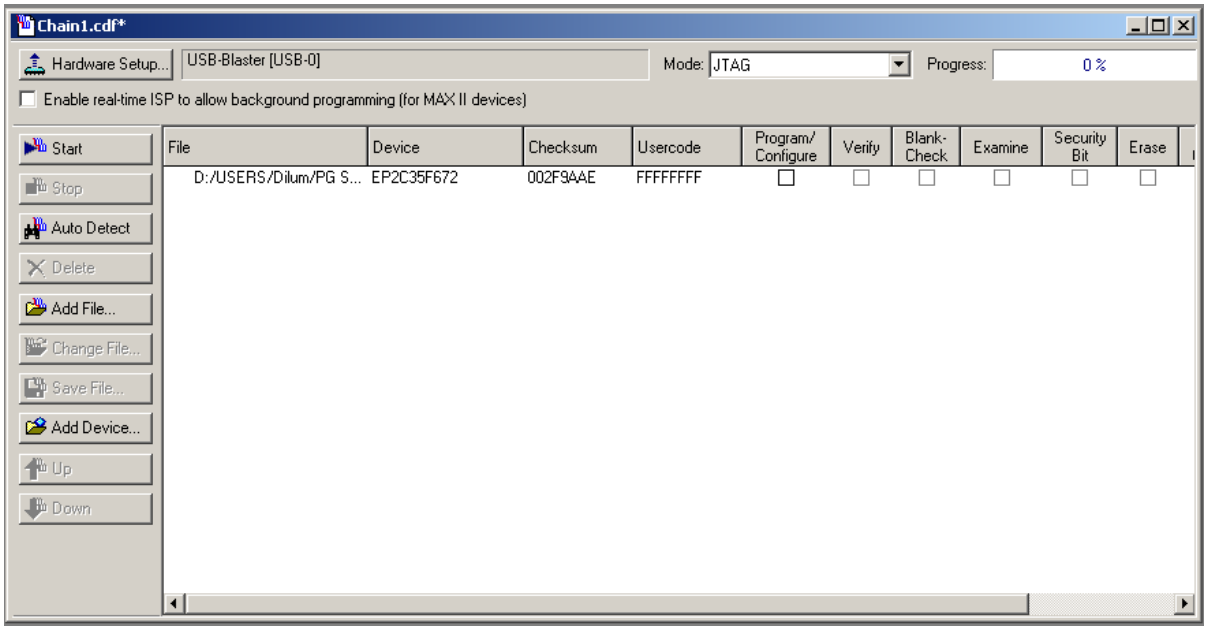

Figure 10 – The Programmer Window

Here it is necessary to specify the programming hardware and the mode that should be used. If not already chosen by default, select **JTAG** in the **Mode** box. Also, if the **USB-Blaster** is not chosen by default, click the **Hardware Setup...** button and select the **USB-Blaster** in the window that pops up, as shown in Figure 11.

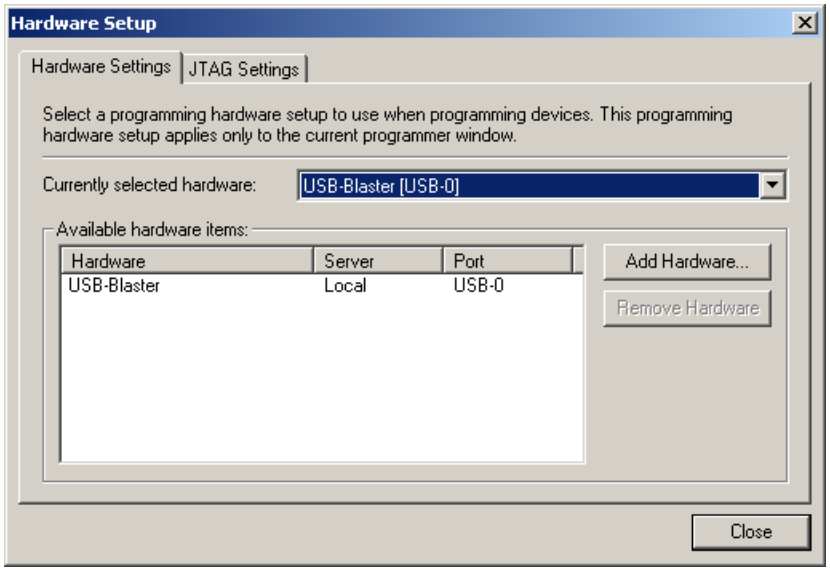

Figure 11 – Hardware setup window

Observe that the configuration file **binary_counter** is listed in the window corresponding to Figure 10.

If the file is not already listed, then click **Add File...** and add the **binary_counter.sof** file.

Click on the **Program/Configure** check box.

Make sure that the device is connected to the computer using the USB cable, is powered on, and the PROG/RUN switch in on the **RUN** position.

Step 3: Now, press **Start** button in the window in Figure 10. A Blue LED on the board will light up when the configuration data has been downloaded successfully. If you see an error reported by Quartus II software indicating that programming failed, then check to ensure that the board is properly powered on.

5. Testing the Designed Circuit

Having downloaded the configuration data into the FPGA device, you can now test the implemented circuit. Flip the RUN/PROG switch to the **RUN** position. Set **SW0** to 1 to enable the counter. Press **KEY0** to verify that the counter counts as intended.

Make a Table indicating the pattern of LEDs with each pulse applied. Change the position of **SW1** and repeat.

Note:

You do not need to submit a report for this lab session. However you need to demonstrate your circuit to the TA before leaving the lab.

Acknowledgements

This tutorial is based on material (from CDs, website, publications, etc.) published by Altera.