Introduction to Altera Quartus II Web Edition Software

CDR Charles B. Cameron

4 January 2005

Contents

1 Downloading the Software **1**

2 Creating a Project **2**

3 Compiling the Project **3**

4 Simulation of Your Design **3**

List of Figures

1 Sample Schematic 3
2 Input Waveform (left) and Simulator Output Waveforms (right) . 5

1 Downloading the Software

1. Direct your web browser to this URL:

   `https://www.altera.com/support/software/download/sof-download_center.html`

   Some of the steps that follow worked for me under Microsoft Internet Explorer but not under Netscape 7.2.

2. Under the heading Altera Design Software, select the link labeled "Quartus II Web Edition".

3. Select the single download file for the current version of the Quartus II Web Edition Software. As of the time of writing, the current version is Version 4.2.

4. You will be asked to provide personal information. Fill in at least the required fields. (If you have created an Altera.com username and password in the past, then it suffices just to provide them.)

5. You will then again see the page asking you to pick a file to download. If you are patient and do not select anything, you will see a File Download dialog box letting you save the software on your computer. Save the file in a convenient place so that you can find it again in the next step.

6. Execute the downloaded file. Follow the instructions provided by the installation dialog to complete the installation. At the end of the installation you will be permitted to examine the file readme.txt. You should do so because in that file is an explanation of how to obtain a license file, necessary for making Quartus II work.

   (a) The license file is available at URL `http://www.altera.com/support/licensing/lic-index.html`. Select the license for Quartus® II Web Edition Software.

   (b) You will need to provide personal information once again. You also need to provide you network interface card number. This prevents the license from being used on more than a single computer. You can determine it by issuing the command `ipconfig /all` at a command prompt. It is a 12-digit hexadecimal number. Enter it without any hypens.

   You will shortly receive an e-mail containing the license file as an attachment, along with instructions on where to save it. You will tell Quartus II about this file after you first run the program.[1]

7. After installation has been completed, you may launch Quartus II. The first time you do this, you will be asked whether you want to use the Quartus II interface or the older Max+Plus II interface. Everything that follows below is predicated on choosing the Quartus II interface.

   Quartus II should now present you with an Evaluation Mode dialog box. You can run the software using the evaluation license (good for 30 days) but it is better to specify the license file e-mailed to you previously. (If Quartus II does not present this dialog box, then you can ask for it explicitly. To do this, select Tools, License Setup. . . .) Enter the license file in the indicated space and select OK.

2 Creating a Project

To create a new project, select File, New Project Wizard. . . and follow the instructions in the dialog boxes.

1. If you do not already have files ready to be placed in the project, then when asked for their names, you can simply go to the next dialog box.

[1] You can defer this step and instead use a 30-day evaluation license. You will be asked if you want to do this each time you load Quartus II until you either install a permanent license or accept the 30-day evaluation license. Subsequently you can change your licensing choices in Quartus II through the Tools, Licenses Setup. . . menu dialog.

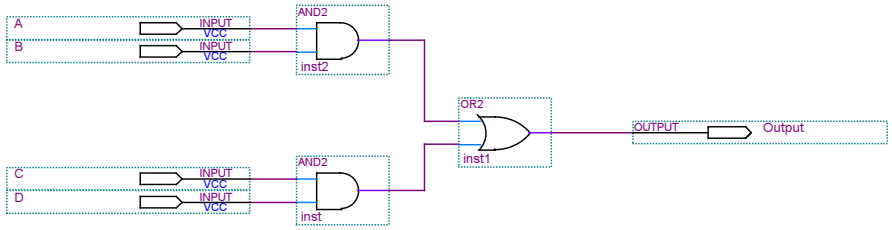

Figure 1: Sample Schematic

2. The device on the UP 2 board is in the MAX7000S family. It is the EPM7128SLC84-7 device.

3. You will not need the EDA tools.

To create a design file in the new project, select File, New... and choose Block Diagram/Schematic File under the Device Design Files tab. A graphics file is made available. Save it using the same name as the project itself.

You can enter symbols by double-clicking in an empty spot. You will see a hierarchy of symbols, although you can also just type the name of the symbol you want into the available space. Enter the schematic shown in Figure 1. The symbols contained in it are called and2, or2, input, and output.

3 Compiling the Project

To compile the project, select Processing, Start Compilation. (You may be asked whether you want to save the file.) This should result in no errors. If you only have an evaluation license, you may get a warning to the effect that you cannot generate programming files. Without a programming file, you cannot download your program into a real device, although you can simulate its behavior.

Messages are generated in several stages during compilation. You can examine them by looking in the Compilation Report under the headings Analysis & Synthesis, Fitter, and Assembler. Later, when you have done a simulation of the design, you can find messages under the heading Timing Analyzer, too.

4 Simulation of Your Design

To simulate your design, you must first create a collection of input waveforms. For the schematic in Figure 1, these are the pins marked A, B, C, and D. To create the file, choose File, New.... Under the Other Files tab, select Vector Waveform File.

In the resultant waveform file, right-click in the left-hand column, under the heading Name. In the pop-up menu, select Insert Node or Bus.... In the Insert Node or Bus dialog, select Node Finder.... In the Filter, select Pins: all and

then select List. You should then see all the pins in your design, including the input pins already mentioned and the output pin called Output. Highlight them all and select the > button to get them listed under Selected Nodes. Select OK and then select OK again. All five pins will now appear under the heading Name.

Select the minimum increment of time by selecting Edit, Grid Size.... For this design, 10 ns is a suitable choice. Also specify the maximum simulation time by selecting Edit, End Time.... For this design, 500 ns is a suitable choice.

In the waveform window, highlight input pin D and select the Overwrite Clock tool from the toolbar to the left of the waveform window. Establish signal D as a square wave with a period of 10 ns.

Use the Overwrite Clock tool on signal C, making it a square wave with period 20 ns. Similarly, give signal B a 40 ns period and give A a 80 ns period. The resultant waveform should look something like the one on the left in Figure 2. Save the waveform file using the same name as the project itself.

Select Processing, Start Simulator. You should get a message box saying you had no errors. The Simulation Waveform should look something like the one on the right in Figure 2.

If you study them, you will see the effects of delay: the output does not become true until roughly 7.5 ns after signals C and D are true and roughly 2.5 ns after signals A and B are true. Whether this will create problems depends on how you use the signals. In this project it is of no consequence, but in some applications it might dictate the minimum update time you can tolerate. The maximum delay was reported in the Compilation Report under Timing Analyzer Messages, too.

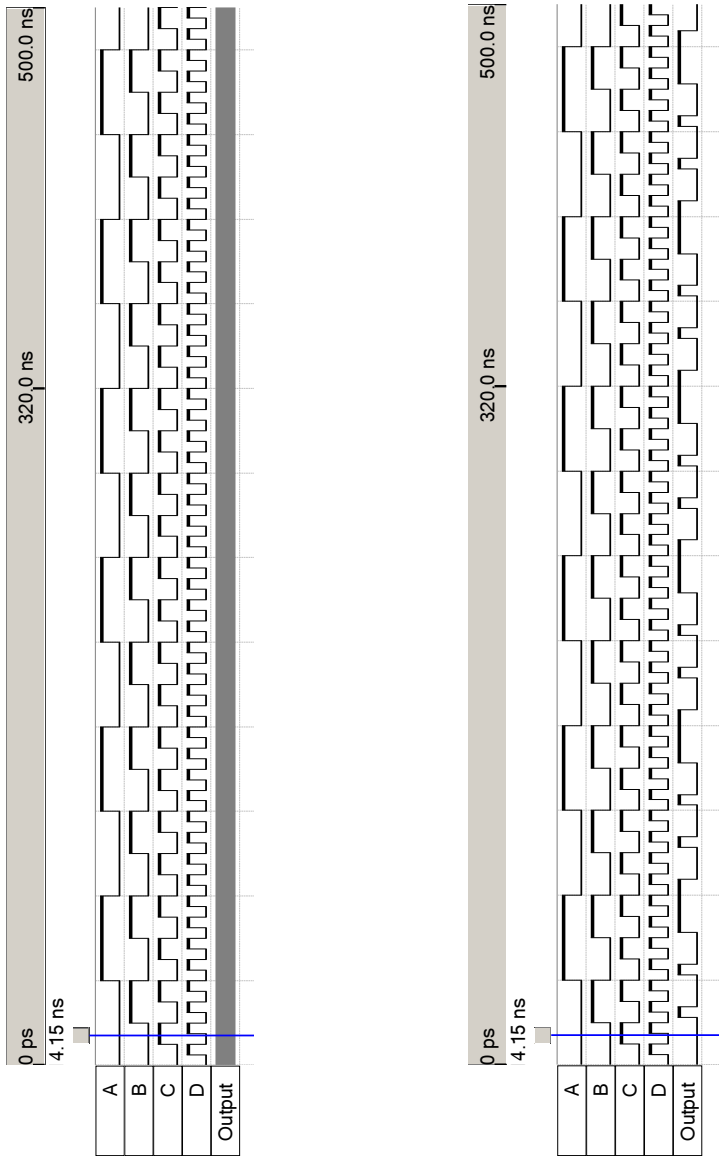

Figure 2: Input Waveform (left) and Simulator Output Waveforms (right)