

ALTERA CPLD TUTORIAL

ECE 221

FALL 2003

I. OBJECTIVE:

The objective of this laboratory is to implement the 4-Bit Adder/Subtractor designed in Design Project #3 using the Altera Education Board.

II. PRELAB (15 points)

All three versions are compatible.

1. Download the Full Adder Plus schematic from the course web page and then using the Altera software open and print out the schematic. The Altera software has been loaded on the Sun Workstations and some of the PCs in room 185. You can also get a student version on the software by following the *Student Version Installation Instructions* under the interesting links on the course web page. Follow the instructions outlined in #2 of the **Procedure** section to start the software. To open a file select **Open** under the **File** menu and then when the pop-up window manager to find the file that you just saved and then click **OK**.

To get a good printout select the **Print Setup** option under the **File** menu. Change the **Orientation** to **Landscape** and hit **OK**. Then to print use the **Print** option under the **File** menu. When the print window appears clear on **All** under the **Print** area and then hit **OK**.

2. Print out the **fourbitaddsub.vec** file located in the above directory (/home/ejohnson/Public/ece222/altera). This is a stimulus file that will be used to verify the adder/subtractor is working correction via simulation. From the file, determine what the input test cases are and what the outputs should be. Write these on the printout beside each test case.
3. Bring a copy of your 4-bit Adder/Subtractor Schematic that you created in Mentor to lab.

III. LABORATORY PROCEDURE (70 points)

1. Log on to the PC at your station using your username and password. This will allow you to print from the PCs.

CREATING A SCHEMATIC IN ALTERA

Creating a schematic using ALTERA is similar to Mentor except that the commands within the graphical user interface (GUI) are a little different. The following information should assist you in creating your schematic. If you get stuck and are not sure what to do next, try using the help system or ask the instructor. For more documentation on the Altera Education Board, refer to the user's manual located located under the plastic box containing the board.

2. **Starting ALTERA:** To start the ALTERA tools, click on the **Maxplus II** icon located in the **Engineering Applications** folder on the desktop (If you are on the Sun Workstations you can start the Altera software but typing **max2win** in a terminal window). Once the GUI comes up, select **New** under the **File** menu and then when the New pop-up window appears, select **OK**. This will bring up the Graphical Editor where you can enter a gate-level design. Once you have brought up the Editor save your file as **fourbitaddsub.gdf** using the **Save** option under the **File** menu. Then set **Project->Set Project to Current File** under the **File** menu. This will set your overall project name to **fourbitaddsub**. You MUST do this step before performing the next steps.

Using the Mentor schematic from your prelab as a guide you are going to create a 4-Bit Adder/Subtractor in Altera. The Full Adder Plus block has already been implemented in Altera and is available for you to use.

3. **Schematic Size:** Change the size of your design by selecting the **Size** option under the **File** menu. When the box appears double click on the **C** option.
4. **Adding Parts:** All parts can be accessed from the **Enter Symbol** option under the **Symbol** menu. When the symbol box appears, you can select from a number of different libraries and components within the libraries. You can also select symbols of components you have created and use those in your schematic. Note: when you add a part if you don't give a location to place the part in schematic (simply clicking your cursor in the schematic window), the part will be added at the center of your schematic.
 - 4.a) Add a title block to your design by double-clicking on the **prim** library and scrolling down in the **Symbol Files** box until you see the component **title** and then double clicking on that component. Add the block to the bottom of your design and change the **Title**, **Company**, **Designer** and **Date** fields. To make a change, just select the current string in those fields and type over them. For title put **4-Bit Adder/Subtractor**, for the company change it to **ECE 221**, for the designer put in your names and change the date to the current date.
 - 4.b) Add the full adder plus component to your design by selecting the **fulladdplus** component under the **Symbol Files** box and hit **OK**. Once you have added one part you can either copy that part (using the standard windows copy and paste commands) or simply add three more parts.
 - 4.c) All of the lights on the Altera board used today are Active-0. That means you need to invert all of the outputs. The Sum outputs have already been inverted in the Full Adder Plus component so all you need to do is add an inverter to the output of your final carry out. The inverter is located in the **mf** library as the **7404** part.
 - 4.d) You will also need to add input and output ports to the design. These are located in the **prim** library under the names **input** and **output**.
5. **Adding Nets:** To add a net, move your cursor over a pin (on a part or net), the cursor should turn into a cross. Then hold the left mouse button (LMB) down, drag to your net end point and let the LMB up. A net should appear in your schematic. NOTE: any item in Red is selected, to unselect, click your cursor on a blank portion of the design.
6. **Naming Ports:** Once you have all of the parts and nets placed, update the port names by selecting the existing name and then typing in the new name. Use the following port names: **A0, A1, A2, A3, B0, B1, B2, B3**, and **MODE** for the inputs, **S0, S1, S2, S3**, and **COOUT** for the outputs.
7. **Saving your Design:** Save your design at this point.
8. **Assigning the Device:** Once you have the design created and the port names changed you need to assign your design to the proper device implementation. To do this select the **Device** option under the **Assign** menu. Choose as the family: **MAX7000S** and as the device: **EPM7128SLC84-6**. Then hit **OK**.
9. **Assigning Pin Numbers:** Once the device is chosen you can assign pin numbers to your input and output ports. Assigning pin numbers is done using the **Pin/Location/Chip** option under the **Assign** menu. To add a pin you put the port name in the **Node Name** field and its associated pin number in the **Pin** field and then click on **Add**. The port and pin names are given below:

MODE	33
A3	35
A2	37
A1	39
A0	41
B3	44
B2	46
B1	48
B0	50
S3	69
S2	67
S1	65
S0	63
COUT	61

Once all of the pin numbers have been added hit **OK**. The pin numbers should appear beside the port names in the schematic.

COMPILING AND VERIFYING THE DESIGN

1. **Compiling your Design:** Once you've created the design, the next step is to compile the design by selecting the **Compiler** under the **MAX+plus II** menu. When the compiler window appears select **start**. As the compilation completes you may get a couple of warning messages which you can ignore. If you encounter any errors, try to figure them out yourself before talking to the instructor.
2. **Setting up the Force File:** Once the design has been compiled you can use the simulator to verify that your design works correctly. A sample force file has been created which you can use to simulate the design (**fourbitaddsub.vec**). Select the **Waveform Editor** option under the **MAX+plus II** menu. When the editor appears select the **Import Vector File** under the **File** menu to bring in the forces. After you select the **fourbitaddsub.vec** file and hit **OK**, the waveforms should appear in the editor. Then select **Save** under the **File** menu and hit **OK** to save these waveforms as input.
3. **Simulating the Design:** Now that you have the force files set up you can run a simulation to verify your design is working properly. Select the **Simulator** option under the **MAX+plus II** menu. The simulation input should be **fourbitaddsub.scf** (or **fourbitaddsub.vec**) and the End Time should be set to 600 ns. To simulate, select the **Start** button. The waveforms in the **Waveform Editor** should be updated with the proper output values. You can see the entire set of waveforms by selecting **Fit in Window** under the **View** menu. Check to make sure they are correct (remember they will be inverted from answers you have in your prelab). If they are not correct, find the error in your design, fix it in the **Graphical Editor**, re-compile and then re-simulate. NOTE: You MUST re-compile your design anytime you make changes before you can perform simulation.
4. **Printing:** Once your waveforms are correct print them out along with the schematic. To get a good printout select the **Print Setup** option under the **File** menu. Change the **Orientation** to **Landscape** and hit **OK**. Then to print use the **Print** option under the **File** menu. For the waveforms, just hit **OK** (make sure the entire set of waveforms is displayed). For the schematic, follow the directions given in the prelab section.

PROGRAMMING THE ALTERA DEVICE AND VERIFY OPERATION

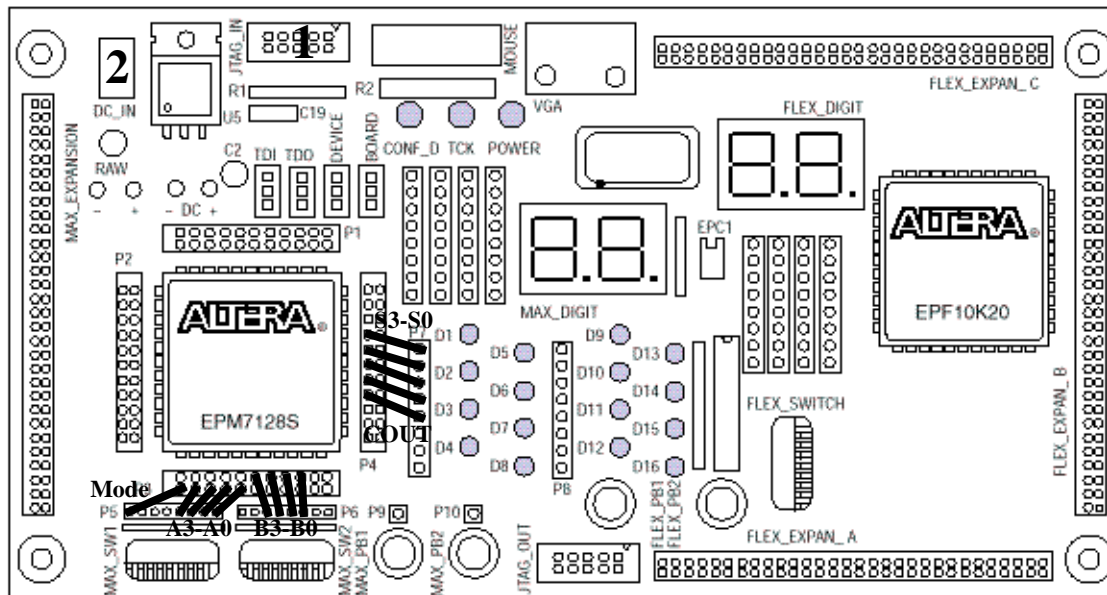
Now you are ready to program the device on the board and verify your operation. The following is a list of equipment you need for this project:

- ALTERA Education Board (EB)

- Serial cable with 25 pin male-to-female connector
- ALTERA ByteBlaster cable
- AC to 9VDC Adapter

1. **Wiring up the Board:** You will need to connect the wires from the device to the switches for the inputs and to the LEDs for the outputs. Use the Figure on the next page as a guide. Connect wires between the headers for the chip and the headers for the switches or LEDs. The left-most switch will be your **MODE** input. The right four switches on the left bank of switches will be the inputs **A3-A0**. The middle four switches on the right bank will be the inputs **B3-B0**. The first column of LEDs will be **S3-S0** outputs with **S3** being the top LED. The top LED on the second column will be the **COUT** output.

Figure 1. UP 1 Education Board Block Diagram



2. **Setting up the hardware:** The hardware should already be setup for you. Here are the steps that were taken to set up the hardware for programming the EB.
 - The parallel cable was connected to the LPT port on the back of the PC on one end and the ByteBlaster cable on the other.
 - The ByteBlaster cable was connected to the education board at #1 in the figure below. The red side of the cable should be on the right as you are looking at the board.

When you are ready to download your design, plug in the power adapter. The other end should already be connected to the power connection on the board (#2 in the figure). The green POWER LED should turn on when plugged in.

3. **Programming the Device:** Now you are ready to program the chip. Select the **Programmer** under the **MAX+plus II** menu. The programmer window should appear. If the Hardware Set-up window appears instead, make sure the **Hardware Type** is **ByteBlaster** and the **Parallel Port** points to **LPT1**. The file name listed in the programming window should be **fourbitaddsub.pof**. If it is not, select **Select Programming File** under the **File** menu, choose the file **fourbitaddsub.pof** and hit **OK**. If you have not set-up the hardware, choose the **Hardware Set-up** option under the **Options** menu. When the window appears make sure the **Hardware Type** is **ByteBlaster** and the **Parallel Port**

points to **LPT1**.

4. **Programming the Device:** Program the chip by selecting the **Program** button in the programmer. Programming the chip should take a couple of minutes. You can then use the switches to verify your design is working correctly. Check the six test cases that were given in the prelab (4+3), (-1+(-2)), (-6+5), (5-2), (2-6), and (-8-3). Verify the outputs are correct and have the instructor or TA sign off on your design. If you have problems, fix them and make sure you document what they were.

IV. DISCUSSION (15 points)

Write a paragraph summarizing the experiment and then answer the following questions.

1. What are the advantages to using Programmable Logic Devices instead of TTL logic? What are the disadvantages?
2. What are some advantages of using CAD tools to automate the whole design process? What are some disadvantages?