

Microprocessor-Based Systems (E155)

Harris & Wang

Fall 2011

Lab 1: Utility Board Assembly

Introduction

In this lab you will assemble and test your utility board containing an Altera Cyclone III FPGA, a PIC microcontroller, a configuration PROM, LEDs, switches, a clock oscillator, and a D/A converter for VGA output. You will be using the board for the remainder of the semester, so it is very important to assemble it properly. But don't worry! If you damage parts on your board, you may obtain parts for another one from the stock room.

Objectives

- Learn how to solder
- Assemble your utility board
- Write a Verilog module to control LEDs and a 7-segment display
- Program the FPGA with your Verilog code
- Test and debug the utility board
- Interface the 7-segment display to the utility board

Requirements

Follow the steps in this guide to become familiar with and assemble your board. Write some Verilog code to exercise the FPGA using the switches, LEDs, and a 7-segment display to ensure your board is operational. Simulate and synthesize your code, then burn it onto the Flash memory and test the board. Hook up a 7-segment display and demonstrate that it works.

Background

In the 1980's and 1990's, digital design projects were built from a truckload of chips, each containing a few logic gates such as 74xx series logic gates or simple programmable array logic chips (PALs). Such projects involve placing and wiring together dozens of chips on a breadboard. It was easy to make a wiring mistake or burn out a chip and spend hours tracking down the problem. Now you can perform all of your digital logic on a single field-programmable gate array (FPGA) to greatly reduce the necessary wiring and number of chips. Later in the course, you will use a PIC microcontroller to write programs in assembly language and C that can interface with external hardware and the FPGA.

You will need to connect your FPGA to the real world to get inputs and outputs. In particular, you will often find it useful to have a clock oscillator, switches, and LEDs. Moreover, the FPGA comes in a 144-pin thin quad flatpack (TQFP) package. This is a surface mount (SMT) part so the pins of the chip do not go through the printed circuit board (as with many of the other through-hole parts you'll be working with). SMT components are mounted on pads on top of the PCB. Their pins are so small that special soldering skills and tools are needed to reliably attach them to the board. Your board comes with the FPGA and its Flash memory already attached, but you will get to solder the rest, including a few easier SMT components.

The FPGA you will use in this course is the Altera Cyclone EP3C5, which contains 5136 Logic Elements. The microcontroller is a Microchip PIC32MX675F512H with 64 KB of RAM and 512 KB of Flash program memory and many I/O ports. You can find the databook for these in the lab notebook and on the class web page. You will need to become familiar with the internals as you progress in this class.

The board was designed to maximize your access to the capabilities of the FPGA and PIC from a breadboard. Therefore, as you will notice, the board has a single row of 58 header pins (on one of the long sides of the board) that give you access to most of the FPGA pins and all of the PIC I/O pins (except for reserved programming pins). The other pins in the FPGA are power, ground or unused. Furthermore, the pins are labeled for your convenience. If interested, we advise you to look at the Cyclone III and PIC databook. It contains full explanation of the functions of each pin. The other characteristics of the utility board will be explained later, in the Discussion section of this guide.

Figure 1 shows the schematic for the utility board. It is advisable that you study this schematic to understand what goes on in the board because you will need this information to use and debug your board and are likely to be asked about some subsystem during your checkoff. The parts in the schematics are labeled using R for resistors, C for capacitors, and U for units (chips and other large parts).

Discussion

Figure 1 shows a schematic of the utility board. Major components include the power supply, header pins, clock generator, DIP switches, reset pushbuttons, USB ports and selector switch, VGA connections, the debugger/programmer connectors, and LEDs.

Read through and understand this section that describes each of the board components. Identify the components in the schematic and think about how it operates. The subsequent section will guide you through the assembly process.

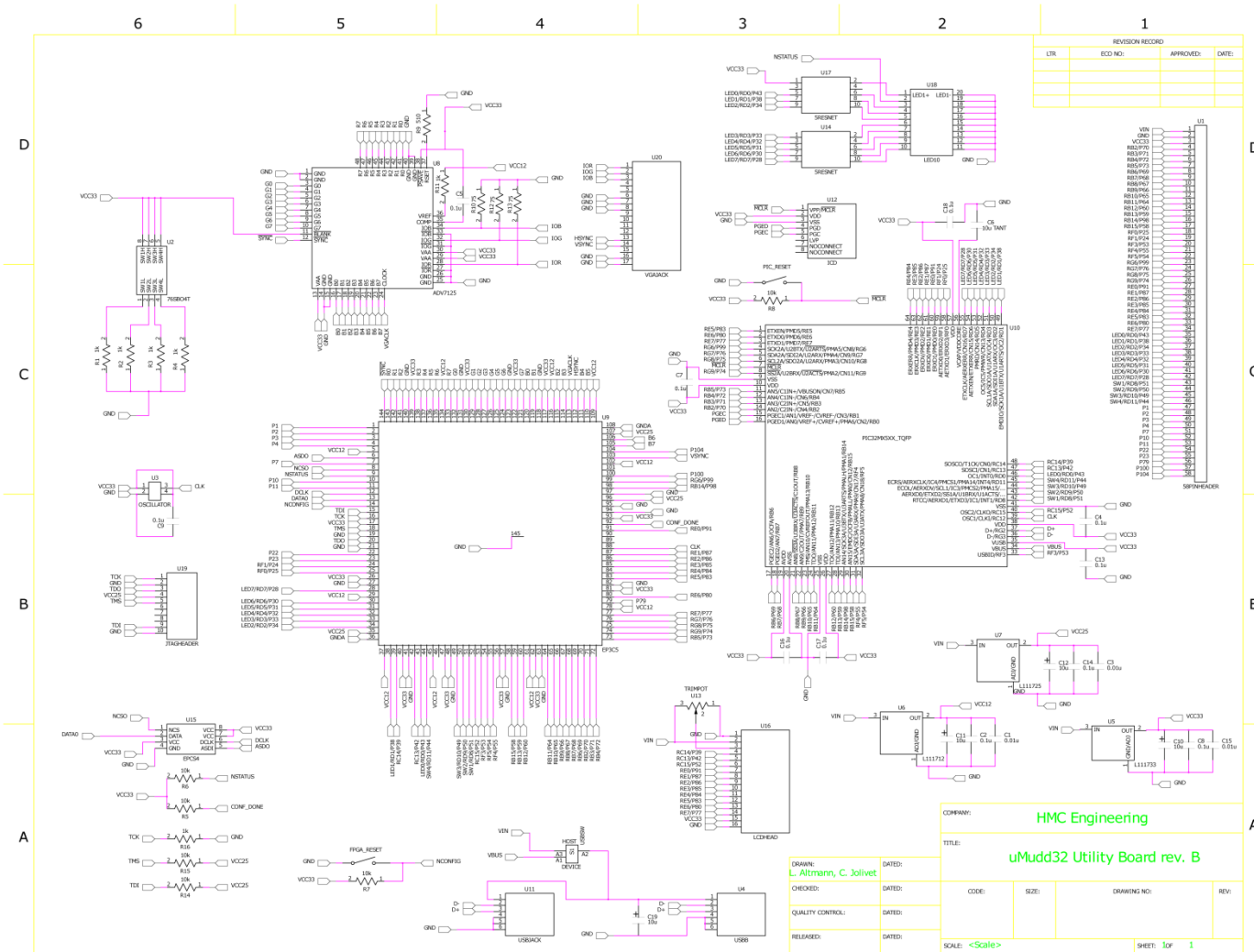


Figure 1 – Utility Board Schematics

Table 1 lists the bill of materials (BOM) with more information about each of the components on the schematic.

µMudd32 Bill of Materials								
Last updated 17 August 2011 David_Harris@Hmc.edu								
Component	Description	Manufacturer	Mfr Part #	Supplier	Sup Part #	Quant	Unit Price	Total Cost
	PCB	Advanced Circuits	P75294			1	\$15.18	\$15.18
U10	PIC	Microchip	PIC32MX675F512H	Digikey	PIC32MX675F512H-80I/PT-ND	1	\$7.14	\$7.14
U9	Cyclone III FPGA	Altera	EP3C5E144C8	Digikey	544-2431-ND	1	\$12.80	\$12.80
EPCS4	FPGA flash	Altera	EPCS4	Digikey	544-1243-5-ND	1	\$13.00	\$13.00
U3	40 MHz clock oscillator	Epson	SG636PCE 40 MHz	Digikey	SE2851CT-ND	1	\$4.38	\$4.38
U2	Quad DIP switch	Grayhill	76SB04	Digikey	GH7170-ND	1	\$0.92	\$0.92
FPGA/PIC_RESET	SPST pushbutton switch	Panasonic	EVQ-PAE04M	Digikey	P8008S-ND	2	\$0.29	\$0.58
L111712	1.2V low-dropout regulator	ST	LD117DT12TRI	Digikey	497-1231-1-ND	1	\$1.16	\$1.16
L111725	2.5V low-dropout regulator	ST	LD117DT25CTR	Digikey	497-1233-1-ND	1	\$1.16	\$1.16
L111733	3.3V low-dropout regulator	ST	LD117DT33CTR	Digikey	497-1235-1-ND	1	\$1.16	\$1.16
U18	10-segment LED array	Lumex	SSA-LXB10IW-GF/LP	Digikey	67-1010-ND	1	\$2.85	\$2.85
U14, U17	330 ohm resistor network	Bournes	4610X-2-331LF	Digikey	4610X-2-331LF-ND	2	\$0.29	\$0.57
R1-4, R11, R16	1K resistor	Yageo	CFR-25JB-1K0	Digikey	1.0KQBK-ND	5	\$0.01	\$0.05
R5-8, R14-15	10K resistor	Yageo	CFR-25JB-10K	Digikey	10KQBK-ND	7	\$0.01	\$0.07
R9	510 ohm resistor	Yageo	CFR-25JB-510R	Digikey	510QBK-ND	1	\$0.07	\$0.07
R10, R12-13	75 ohm resistor	Panasonic	ERD-S2TJ750V	Digikey	P75BACT-ND	3	\$0.05	\$0.15
U8	VGA DAC	Analog Devices	ADV7125	Digikey	ADV7125KSTZ50-ND	1	\$6.30	\$6.30
C2, C4-5, C7-9,								
C13-14, C16-18	0.1 uF capacitor	Panasonic	ECQ-V1H104JL	Digikey	P4525-ND	11	\$0.08	\$0.90
C1, C3, C15	0.01 uF capacitor	Panasonic	ECQ-V1H103JL	Digikey	P4513-ND	3	\$0.08	\$0.25
C10-12, C19	10 uF capacitor	Panasonic	ECA-1CM100	Digikey	P5134-ND	4	\$0.05	\$0.20
C6	10 uF tantalum capacitor	AVX	TAP106K025SCS	Digikey	478-1841-ND	1	\$0.63	\$0.63
U4	USB jack A	EDAC	690-004-621-013	Digikey	151-1080-ND	1	\$1.10	\$1.10
U11	USB jack B	EDAC	690-004-221-023	Digikey	151-1121-ND	1	\$0.83	\$0.83
U1	36-pin header	3M	929400-01-36-RK	Digikey	929400E-01-36-ND	2	\$1.32	\$2.65
U12	ICD connector	Tyco	5520250-3	Digikey	A31406-ND	1	\$0.61	\$0.61
U20	VGA connector	Tyco	1-1734530-1	Digikey	A35116-ND	1	\$1.63	\$1.63
U19	FPGA download connector	Tyco	5103309-1	Digikey	A33160-ND	1	\$0.76	\$0.76
U16	2x8-pin female header	Sullins	PPPC082LFBN-RC	Digikey	S7111-ND	1	\$1.30	\$1.30
U13	10k trimpot	Copal	CT6EW103	Digikey	CT6EW103-ND	1	\$0.26	\$0.26
S1	SPDT microswitch	Tyco	SLS121PC04	Digikey	450-1598-ND	1	\$0.91	\$0.91
Total:								\$79.57

Table 1 – Bill of Materials

Power Supply

The utility board is supplied power by the header pins V_{in} and GND. The input supply should be at least 5 volts; for example, an unregulated 9 V DC transformer suffices. It is regulated down to 3.3 V, 2.5 V, and 1.2 V to serve various components on the board. The 3.3 V regulated voltage is also available at another header pin to serve other components you might want to connect to your board.

In the lab, the easiest way to supply power is from a benchtop power supply to the V_{in} header pin. If you do not have a power supply available, a DC transformer is an inexpensive alternative. For unteathered applications such as robotics, you can also use a battery pack providing at least 5 V.

The PIC and FPGA I/O's will operate at 3.3 V, which we will call V_{CC} . The FPGA contains tiny logic transistors that would burn out at 3.3 V, so the 1.2 and 2.5 V supplies are needed to operate the digital logic and the analog circuitry within the FPGA.

Header Pins

The board provides one 58-pin row of male header pins that tap out signals from the FPGA and PIC, LEDs, switches, and power and ground. The header pins will be installed along the left side of the utility board.

Clock Generation

In digital electronics, the term clock refers to a square wave that oscillates between GND and V_{CC} . Typically, clocks are used to sequence the action of a circuit. For instance, a flip-flop samples its input every time the controlling clock rises from logic 0 to logic 1. The clock signal is also sent to the PIC and the FPGA. The utility board provides 40 MHz clock from a crystal oscillator.

DIP Switches

The DIP switches provide a convenient source of inputs to circuits in your FPGA. When the switch is closed, it delivers a high output. When the switch is open, a pull-down resistor produces a low output.

Reset Pushbuttons

There are two reset pushbutton switches on the board. The pushbutton near the PIC resets the PIC, and the one near the FPGA resets the FPGA. Upon reset, the PIC will simply restart its program and begin as if it has just powered up. The FPGA, however, will have its memory cleared. If the PROM is programmed, the FPGA will immediately reprogram itself from the PROM. You will notice that these pushbuttons actually have 4 pins. However, only two pins are used, while the other two are left unconnected (and simply provide mechanical stability).

Debugger/Programmer Jacks

There are two programming connections at the top of the board. The double row of header pins (marked JTAG) are for programming the FPGA. The 6 pin jack (which resembles a telephone jack) is for programming the PIC microcontroller via the In Circuit Debugger (ICD 3) dongle. You will program and debug your PIC in Lab 4.

LED array

There is a 10 segment red light-emitting diode (LED) array on the board. The 'ON' LED (far left LED, with the pin header towards you) simply tells that the board is powered ON and receiving 3.3 V. The LED immediately to the right of the ON LED is unused. The other 8 red LEDs can be driven by the FPGA or PIC. Notice that there are 8 header pins labeled LED0 – LED7. These can be used in case you want the outputs of the LEDs for an external device. If a voltage is applied at one of the LED pins, the corresponding light is off for ground (GND) and on for voltage 3.3V (V_{cc}).

There are a few concerns that must be addressed whenever using LEDs. One is that LEDs glow nicely when given about 5 mA of current but may burn out when given more than roughly 20 mA. Therefore, it is important to include a current-limiting resistor to prevent LEDs from burning out. A $330\ \Omega$ resistor does the job nicely, allowing $\sim \frac{3.3V}{330\Omega} = 10\text{ mA}$ of current to flow. (In actuality, the current is about half of this. There is a voltage drop across the diode (about 1.7V for red), which makes the actual current $\frac{3.3V-1.7V}{330\Omega} = 4.8\text{mA}$. This is still around the desired 5-20 mA range and so still works).

FPGA

The FPGA contains a large array of configurable logic blocks (CLBs) and programmable interconnections. The configuration information is stored in static random-access memory (SRAM) within the FPGA. Therefore, the information is lost when power is removed and the device must be reconfigured each time it is powered up.

The FPGA can be configured via the Programmable Read Only Memory (PROM) on board. On powerup, the FPGA will serially configure itself from the PROM if the PROM has been programmed. It will also reprogram itself from the PROM when you push the reset button. Alternatively, designs can be programmed directly onto the FPGA in the Quartus software via the JTAG connection. This will overwrite the design already loaded from the PROM, but will only last until the FPGA is reset or the power is disconnected.

VGA Output

A digital-to-analog converter (DAC) is connected to dedicated pins on the FPGA and used to drive the VGA output on the board for control of a computer monitor. The DAC has three 8-bit channels, one each for the red, green and blue signals. 75-ohm termination resistors are used to prevent reflections in the transmission line (VGA cable).

Assembling the Board

This section of the lab guide will give you tips on how to assemble the utility board. Following these tips will help you complete this lab quickly and in a logical way that places the flattest components first to make soldering easier.

□ Identify the Component Side of the Utility Board

The utility board has two sides. The component side, with the white silk screen markings indicating component placement, should go up. All components except the 58-pin header are placed on the component side. The parts (except the 58-pin header) are soldered on the reverse solder side.

Using a multimeter, check that there is nearly infinite resistance between the pins labeled Vin and GND on the board. If the resistance is low, you have a short circuit on your bare board and should get a new one. As you assemble your board, occasionally check the resistance between power and ground. If it is ever less than 10 Ω , you've introduced a short and should debug it before continuing.

The PIC, FPGA, VGA DAC and PROM should already be soldered to your board. These are fine-pitch SMT components that require practice to attach without creating solder bridges between pins. If your board malfunctions and you suspect a bridged or bad connection on one of the SMT chips, check the connections under a microscope.

CMOS components such as the FPGA are sensitive to static electricity. Before you touch your board or solder anything, discharge your body by touching a large metal object. This is good practice when handling electronic components in any lab.

When you begin soldering, moisten the sponge. When the iron first heats up, tin the tip by applying a generous amount of solder all over the tip, then wiping off the excess on the sponge. Periodically repeat as you work to keep the tip looking silvery rather than black and blistered. When you make a connection, touch the tip of the iron to the pad on the board at the same time it touches the lead of the component. Apply the solder to the joint, not the tip of the iron. The solder should smoothly adhere to both the pad and the component pin rather than balling up on the component. The connection should appear shiny; a gray color indicates a possible unreliable "cold solder" joint. If you are in doubt about the quality of your solder joints, ask early on rather than doing all of them first and discovering that your connections are intermittent or unreliable. Some of the components will be soldered close to vias (the small holes through the board used to connect between wiring layers on the printed circuit board). Be sure excess solder does not bridge to the via, creating a short circuit. When you are done, tin the iron one last time to protect the tip before turning it off.

You may wish to use safety goggles while soldering, especially if you don't wear glasses. Also, be sure to hold the ends of leads as you cut them after soldering so they don't fly

into the eye of the person working across the room. Solder contains lead, so wash your hands afterward.

□ Insert the Blue Wires

Rev B of the board accidentally let out the wires connecting the DIP switches to the header pins. Such PCB mistakes are traditionally corrected with thin (e.g. 30 gauge) blue wires joining the missing pins. Hence, the fixes have become known as “blue wires.”

Before anything else, you will need to cut and insert four short thin wires to correct this error. Cut the wires about 1.5” long and carefully strip $\frac{1}{4}$ ” of insulation off each end. Place them on the back side of the board to connect the bottom of resistors R1-R4 with the header pins labeled SW1-SW4, respectively. Bend the ends to hold them in place, but do not solder yet. When you insert the resistors and header pins, place them in the same holes and solder both the component and blue wire to obtain the missing link.

□ Insert the Resistors and Resistor Networks

The first components to install in the utility board should be the resistors. They will sit flat on the board. The polarity of the resistors does not matter, i.e., you can insert them whichever way you’d like. However, it is good practice to install neighboring resistors in the same direction, making it easy to read them. See Table 1 for resistor part values.

Next you should install the (isolated) resistor networks. ‘Isolated’ means that it contains a set of totally independent resistors. These particular resistor networks are sets of 5 330 Ω resistors and are installed at U14 and U17. Pins 1 & 2 are one resistor, pins 3 & 4 are another, and so on. Polarity does not matter. You may find it helpful to bend one pin at each end of the resistor network to hold it in place while soldering.

Part	Reference Designations
1k Ω resistor	R1-4, R11, R16
10k Ω resistor	R5-8, R14-15
75 Ω resistor	R10, R12-13
510 Ω resistor	R9

Table 1: Resistor component values.

□ Insert the Voltage Regulators and Oscillator

The next step is to insert the voltage regulators and oscillator. These parts are surface mount (SMT) parts. Soldering a surface mount device would be easiest if you had three hands: one to hold the part with pliers, a second to hold the soldering iron, and a third to apply the solder. One solution is to get a friend to hold the part, taking care not to apply the soldering iron to the friend. Another is to apply a blob of solder to one of the pads on the printed circuit board before the part is placed. Then place the part with the pliers and use the soldering iron to melt the blob onto the pin to hold the part in place. Solder the remaining pins to their pads. Then apply more solder to the first pin to be sure a good

joint is formed (a gray grainy appearance indicates an unreliable cold solder joint, while a silvery appearance covering the pad and pin usually means a good connection).

The oscillator is a 4-pin chip, and its location is near the PIC. This location is marked with U3. Be sure to align the notch on the board with the notch on the silkscreen. Solder the surface mount parts from the top of the board. Be sure not to fill neighboring vias.

Next add the voltage regulators. Be sure to put each voltage regulator in the correct location. If you do not, your board will not work and you may burn out the FPGA. The large pad on one end of the voltage regulator requires a substantial amount of heat to warm up, so you will need to apply the soldering iron for longer than usual. Be sure that you form a good solder joint between the large pad and the tab on the regulator; it is a common place for poor connections.

□ Install the Pushbutton Switches

There are two small pushbutton switches that come with your kit, which will be installed at PIC_RESET and FPGA_RESET. When installing one of these, you will have to align it in its holes and push until it snaps into location. Do not put your finger directly underneath the board because the sudden popping into position could stab your finger. A thin pair of pliers helps push the legs through. The switch will only align in two positions on the board; either alignment will work.

□ Insert the DIP Switches and LED array

The DIP switches are to be placed at the bottom of the board at U2. Align the writing on the DIP switches with the silk screen writing on the board.

Insert the LED Array at U18. Align the notch on the part with the square solder pad on the board for correct alignment.

□ Install the LCD Header and Trimpot

Now install the trimmer potentiometer (trimpot) directly at U13, near the voltage regulators. This allows adjustment of the LCD contrast. You may need to bend the pins slightly with a pair of needle-nose pliers so that they match the holes on the board. Install the 8x2-pin female header at U16, right next to the trimpot.

□ Install the VGA connector

The VGA connector has pins connected to the 3-channel DAC, to provide analog red, green and blue levels, and to the FPGA to provide synchronization signals. You will use the VGA connector in lab 7. Install it at U20. You may need to bend the outer rows of pins by a few mm in towards the center row before they align with the holes. Look from the bottom of the board to make sure all the pins are in their proper locations before pushing the connector on to the board.

□ Add the USB ports and mode switch

The PIC contains a USB module that can act in either host or device mode. Install the USB ports near the ICD connector. Each port fits in only one place. Solder the smaller electrical pins and the large board-lock connectors.

Install the SPDT slide switch by the USB ports. This switch selects between the host and device ports by either powering the VBUS signal from the board (host mode) or letting the PIC sense when the VBUS signal has been powered from another source (device mode).

□ Insert the Bypass Capacitors

The voltage regulators supply current sufficient to meet the average demand of the circuits on the board. However, they cannot respond quickly enough to deliver spikes of current at high frequency, such spikes would cause the voltage to briefly droop. Instead, this current is drawn from bypass capacitors on the board, keeping the voltage at the desired level. A typical digital system uses several sizes of bypass capacitors. Larger capacitors store more charge and can deliver more current, but react more slowly. Smaller capacitors can rapidly deliver charge to handle short spikes of demand. Overall, the bypass capacitors reduce the noise on the voltage supplies. Note that the chips and board have some inherent capacitance as well, so the voltage might be stable enough even without bypass capacitors. However, noise-related problems are so difficult to reproduce and fix that it is always better practice to put a generous number of inexpensive bypass capacitors on the board than to skimp on bypassing and hope the board works anyway.

There are three 0.01 μF capacitors, eleven 0.1 μF capacitors, and four 10 μF capacitors to install. Install the 0.01 μF caps in C1, C3 and C15, and the 0.1 μF caps in C2, C4, C5, C7, C8, C9, C13, C14, C16, C17 and C18 (polarity does not matter). Next, install the 10 μF electrolytics in C10, C11, C12 and C19 (polarity DOES matter). There are two ways to determine polarity on an electrolytic capacitor. If it is new and uncut, the longer of the two leads is the positive terminal (+) and goes in the + hole on the utility board. The other way is to look at the capacitor and find the stripe going down one side. This points to the negative (-) lead. If you install an electrolytic with the incorrect polarity, it can leak or explode. Now install the 10 μF tantalum cap at C6 (polarity DOES matter). The polarity marker is missing from the silkscreen: put the positive terminal in the hole further from the header pins. Small capacitors are often labeled with a three-digit code similar to resistors. For example, 223 indicates $22 \cdot 10^3 \text{ pF} = 0.022 \mu\text{F}$. See Table 2 for capacitor part values.

Part	Reference Designations
0.01 μF capacitor	C1, C3, C15

0.1 μ F capacitor	C2, C4-5, C7-9, C13-14, C16-18
10 μ F electrolytic cap.	C10-12, C19
10 μ F tantalum cap.	C6

Table 2: Capacitor component values.

□ Install the ICD and JTAG connectors

Install the JTAG connector at U19. The connector is symmetrical, so rotation does not matter. The ICD connector looks like a telephone or Ethernet jack, and is installed at U12 next to the JTAG connector. When placing this part you will have to push gently until the jack is firmly in place. Then solder each of the pins.

□ Install the 58-pin Row of Header Pins

This is the *only* part that is installed on the bottom and soldered on the top (it will be inserted into the breadboard). Place the part so that the long pins extend below the board, with the black bumper up against the bottom of the board. The short pins go through the board and are soldered on top. Watch carefully that you do not use too much solder because it may bridge to one of the vias. When you place the header pins, try to keep all of them perpendicularly aligned so that placing the utility board in your breadboard isn't too much of a headache. As the header pins come in strips, you will need to use one long strip and cut one shorter strip.

Now hopefully you should be done with assembling your board. Before proceeding with the lab, you should check all of your solder joints. They should look reliable, filling their specific holes in the board. If some of your soldering looks more like a bubble on the top of the pin, you probably have a bad connection there. Also look for solder jumping across pins or to vias (solder bridges) and cold solder joints (gray, dull joints).

After you've checked your soldering and verified with a multimeter that there is no power to ground short on the board, your board is done!

□ Place the Utility Board on the Breadboard

Now you have to place your board on the breadboard. It should fit in the right-most side of the breadboard.

□ Test the Power Supply

After you've placed the utility board, connect a red wire from the red Vin banana plug connector to the upper power row. Then, from this row, connect a wire to the row of pins next to the Vin pin on the utility board. Then connect a black wire from the GND banana plug connector of the breadboard to the second power row (below the red wire). Connect

this row to the Gnd pin of the utility board. Make sure these wires do not touch each other.

Turn on a benchtop power supply and check the 0-6 V scale indicates 5 V. Adjust it if necessary. Please keep the supply set to 5 V. However, it is good practice to check the supply before connecting it to your board because the knobs occasionally get bumped and you don't want to connect an incorrect voltage that damages your board or causes erratic operation.

When the power supply is set correctly, turn it off. Get a red and a black pair of banana plugs and connect the power supply to your breadboard (never connect power circuits while the supply is turned on). Turn the supply back on. The power LED should turn on. Feel your PIC and FPGA and make sure they do not get very warm. Also check your other components, especially the voltage regulators. The current from the supply should be around 100 mA. Check the 3.3 V output pin and confirm that the voltage regulator is producing the correct value. Also check that the capacitors C11 and C12 next to the 1.2 and 2.5 V regulators have 1.2 and 2.5 V on one of their leads.

□ Test the LEDs and switches

You can test your LEDs and switches by connecting a wire in the breadboard between one of the switch outputs and one of the LED inputs. Toggle the switch and check that the LED toggles. Repeat to check all the LEDs and switches. Remove the wire when you are done so that it doesn't interfere with subsequent testing of the FPGA. If anything is amiss, turn off the power supply immediately before damaging any parts. Look for solder bridges on your board.

□ Test the oscillator

Verify that the oscillator was installed correctly by using the oscilloscope to view the oscillator signal. You should use the analog side of the scope. The probes are in the pouch above the oscilloscope. Make sure to attach the probe ground wire to your circuit ground. The clock signal is pin 3 of the chip. Adjust the voltage and time scales so you can see a nice signal. Use either the cursors or the Quick Measure feature to verify that the correct frequency clock is being produced.

□ Clean Up

Clean up your lab station. Discard the refuse you accumulated while soldering. Tin the iron and turn it off. Please try to keep the lab clean and neat as you work because you share it with many others.

FPGA Design

Your next goal is to write some Verilog modules to test the hardware on your board and operate a 7-segment display. By writing a module to control the LEDs based on the

switches and clock, you will show that these components work and also that the FPGA, PROM, and power supply are functional. The system should have the following inputs and outputs:

clk	input	the 40 MHz clock
s[3:0]	input	the four DIP switches
led[7:0]	output	the 8 lights on the LED bar
seg[6:0]	output	the segments of a common-anode 7-segment display

The following tables define the relationship of the LEDs to the switches and clock:

S0	LED0	LED1
0	OFF	ON
1	ON	OFF

S1	LED2	LED3
0	OFF	ON
1	ON	OFF

S2	LED4	LED5
0	OFF	ON
1	ON	OFF

S3	S2	LED6
0	0	OFF
0	1	OFF
1	0	OFF
1	1	ON

Table 3 – I/O Requirements

LED7

Blink at ~2.4 Hz

The 7-segment display should display a single hexadecimal digit specified by s[3:0]. Remember that you will be using a common anode display. The anode (positive terminal) of all of the LEDs is tied to 3.3 V through a single (“common”) pin. Each segment’s cathode (negative terminal) is connected to a pin. Therefore, you will need 7 separate control signals. Remember that a logic 0 applied to the cathode will turn on the segment. The segments are defined as shown below. Let seg[0] be A and seg[6] be G.

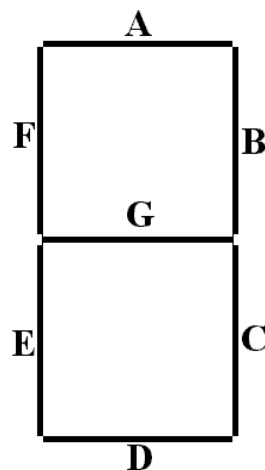


Figure 2 – Seven Segment Display layout

Launch the Quartus II software and start the New Project Wizard from the File menu. Your first decision is where to keep your files. Ideally, they would go on your Charlie home directory. However, Quartus can be painfully slow accessing files on the file server over the network. A better choice is to keep your files on the local C drive while you are working and then to copy them back to your Charlie account before you log out. Please clean up after yourself by deleting the project from the C file when you are done, and remember that it is an honor code violation to refer to somebody else's code that was left on a computer. Note that some CAD tools have trouble with filenames having spaces and other special characters, exceeding 8 characters, or in paths not starting with a letter drive. The best way to avoid these problems is to choose short alphanumeric filenames.

Create a working directory for the project such as `c:\e155\xx\lab1_xx` where 'xx' area your initials. Name the project `lab1_xx` and click Next.

Click Next at the Add Files page, since we will not be adding any existing files to the new project. Now we will tell Quartus which FPGA we are using. Under Family, select Cyclone III. Under Available Devices, select EP3C5E144C8. If you ever forget which device to use, the part number is written on the FPGA.

On the next page, select Pick ModelSim-Altera as the Simulation tool and SystemVerilog HDL as the format. Click Next and verify the settings for the project. When you are satisfied, click Finish.

Choose File -> New and create a SystemVerilog HDL file. Save the file as `lab1_xx.sv` in your project directory and check the box to add the file to the current project. Create modules to perform the functions described above. The 7-segment display decoder should be combinational logic. Use a reasonable amount of hierarchy. Name the top level module `lab1_xx`. The 7-segment display code, for example, will be reused in future labs, so it should be a module of its own. Every module should begin with a comment section that includes your name and email address, the date of creation, and a brief summary of its purpose, so that somebody else can understand what the module does and get ahold of you if they need help. Comment the modules as appropriate.

Logic Simulation

The next step is to simulate your logic with ModelSim.

Check that Altera has the correct path for ModelSim by invoking Tools → Options. Under EDA Tool Options, check that the ModelSim-Altera path is

```
c:\altera\11.0\modelsim_ae\win32aloem
```

and set it if necessary. EDA stands for Electronic Design Automation and is the industry name for computer-aided design (CAD) tools for electronic design.

For unknown reasons, Quartus wants you to compile your code before simulating. To do this, select Processing → Start Compilation to compile your design. Ignore warnings about missing pin assignments or timing violations.

Invoke Modelsim by choosing Tools → Run EDA Simulation Tool → EDA RTL Simulation. RTL stands for Register Transfer Level code (your SystemVerilog code). The ModelSim window will open. Get in the habit of watching the transcript window to look for errors and to familiarize yourself with what a good run looks like. If you see errors, close ModelSim, correct your Verilog code in Quartus, and reopen ModelSim.

In ModelSim, simulate your design by choosing Simulate → Start Simulation. Click on the + symbol next to the work library and select your code (lab1_xx).

If the wave pane isn't open, open it by choosing View → Wave. View all of the inputs and outputs of your design by selecting them in the Objects window and dragging them to the Waves window. In a more complicated design, you may wish to examine internal signals as well.

Manually test your design by forcing the inputs to specific values. In the transcript window, type:

```
force s 0000
run 100
force s 0001
run 100
force s 0010
run 100
...
```

You should see the led and seg outputs displaying appropriate values. Note that the clock is not driven and you have not reset any registers so led[7] will be an x.

Check the outputs against your expectations. If you find any discrepancies, fix the code and resimulate. A helpful shortcut to avoid restarting ModelSim is that you can edit the module by finding it in under “work” in the library pane, right clicking, and choosing Edit. Make your fixes, then right click again and choose Recompile. Then type `restart -f` in the transcript window to restart simulation without having to set up the waveforms window again. When you return to Quartus, you'll find your corrected code.

Pin Assignment

Next, assign pins to relate the signal names in your Verilog code to physical pin numbers on the FPGA. Launch Assignments → Pin Planner. A table listing all inputs and outputs for the project should appear. Under Value, type the pin number to associate with the given signal. For example, SW1 is mapped to pin P51, so enter PIN_51 as the value for s[0].

The FPGA pinouts are shown in the Utility Board Schematic. Most of the user input/output (I/O) pins are tapped out to the headers and labeled on the board silk screen. Some have special functions; for example, FPGA pin P43 is connected to LED0 and also to the PIC port RD0. The clock is connected to pin P88 and not to a header pin because a 40 MHz clock would be degraded by the parasitic capacitance and inductance of the breadboard.

The pin numbers for the LEDs and switches are marked on the board's silkscreen. For the outputs for the 7-segment display, you may select any I/O pins you'd like. For example, you could use pins 1, 2, 3, 4, 7, 10, and 11, since they are contiguous on the board.

Note that the Pin Planner defaults to assuming that each bank of I/O pins receives 2.5 V and uses 2.5 V outputs. However, our system uses a 3.3 V I/O. Fortunately, leaving the voltages at their default value in Pin Planner works and generates 3.3 V outputs.

7-Segment Display Circuit

The 7-segment display will be used throughout the class for general output of numbers. In this lab assignment, though, it will be used to output the hexadecimal number entered by the user through the DIP switches.

Each segment of the display works as an independent LED. Therefore, the same current-limiting concern with the LEDs applies to the display as to the on-board bank of LEDs. You can limit the current into each segment of the display the same way you did for the LEDs on board, adding a suitable resistor to provide roughly 5-20 mA of current. You can find resistors and other such components in the supply cabinet or in the stockroom.

Consult the data sheet for the pinout of the common anode dual seven segment display. All seven segments share the same anode, which should be connected to VCC (3.3 V). Each of the segments has its own cathode, which can be pulled to 0 to turn on the segments.

Be sure to turn power off before wiring circuits on your board. You can choose either side of the display to use in this lab. After deciding on which side to use, you will need to connect the V_{DD} pin of that side (either V_{DD1} or V_{DD2}) to 3.3 V. Then connect the input pins of the same side of the display to the header pins you chose. Remember to add suitable resistor between each of the inputs to the display and the header pins. These LEDs are common anode LEDs. That is, all the anodes from the LEDs are connected to a single V_{DD} (V_{DD1} or V_{DD2}). You are driving the cathode of each LED. Given this information, you might need to modify your Verilog file. Do so in the simplest way possible.

Generating the FPGA Configuration Files

Now you will synthesize your HDL into a programming file to be transferred onto the FPGA. This outputs an SRAM Object File (.sof) in your project directory that can be used to program the FPGA directly over JTAG. Be sure your SystemVerilog files are saved, and choose Processing → Start Compilation. To help sort the many messages that the compilation process generates, click a tab under the Message area to see only that type of message. If compilation is successful but generates warnings, check the Warning and Critical Warning tabs for errors relevant to your design. Warnings about incomplete I/O assignments may be ignored if you have in fact assigned all relevant I/O pins. Missing Synopsis Design Constraints file warnings and timing analysis violations may also be ignored.

Note that Quartus seems to crash from time to time while compiling. If it does, restart and try again. If you find a consistent pattern of what causes the crashes, please let the instructor know.

Launch Tools → Netlist Viewers → RTL Viewer and examine the RTL schematic of your design. This shows the logic synthesized from your Verilog design. Ensure the hardware matches your expectations.

Look at the Compilation Report tab. In the Flow Summary, you should see a total number of registers and pins that match your expectations. Under Analysis & Synthesis, you can see how the logic blocks and registers are broken down in each module. Under Fitter, the Pin-Out File should match the pin assignments you intended.

Programming the FPGA via JTAG

Next, you will load your design onto the FPGA and PROM with an Altera USB Blaster programming device and the chip's JTAG¹ interface. Run Tools → Programmer to bring up the Programmer window. In the top left corner, check that USB Blaster is selected and use Hardware Setup to choose it if necessary. In the top-right corner, set Mode to JTAG. Your lab1_xx.sof file should appear already in the programmer window, and list EP3C5E144 as the associated device. If it does not appear, click the Add File... button and find the file by hand in your project directory. Be sure the Program/Configure box for your .sof file is checked.

You are now ready to program the FPGA. Connect the USB Blaster to the JTAG port with the ribbon cable leading off to the right of the board. Turn on the power supply. In the programmer window, press Start to begin the process. Check the Messages pane for programmer output, and verify that there were no errors. Look for “Configuration succeeded” and “Successfully performed operation(s)” messages.

¹ JTAG stands for the *Joint Test Access Group*. The JTAG interface is a serial interface used to communicate with chips on a board using a few pins during testing and configuration. JTAG is also called Boundary Scan.

At this point, with your design loaded, you should see the hexadecimal digit currently set on the switches displayed on the 7-segment display. Debug your design until it is fully functional. Good luck!

If you have problems, the following checklist may help:

1. Check the voltage outputs on each of the voltage regulators. Also check it on the adjacent capacitors (C10-12) to detect a bad connection between the regulator output and the PCB. This is one of the most common soldering problems.
2. Check that the power supply is hooked up correctly. Measure with the volt meter that you have 5 volts between the V_{in} and GND pins on your FPGA board. Check that the current out of the power supply is less than 200 mA. If the current is high, look for power/ground shorts on your board. Otherwise, if the voltage is incorrect, adjust the power supply to produce the correct output.
3. If the JTAG programmer indicates “Program Failed,” first, make sure your board is powered, close the Programmer window and try again. Sometimes programming fails if you power on your board after you open the Programmer window. If programming still fails the problem could be in your board or in the programmer. If programming works on a classmate’s board using the same programmer, you may have a bad solder joint or other board error. If the programmer doesn’t work on multiple boards, please report it to the instructor.
4. If your power is good but the FPGA still refuses to respond, check the NCONFIG signal (on the FPGA_RESET button). It should be 1 when the button is not pressed.
5. If the LEDs flicker on and off when you tap on the board or toggle switches, you probably have a loose connection or marginal solder joint somewhere. Two common problems in the past are poor banana plugs that are too small for the recepticals and inadequate solder joints on the voltage regulators.
6. If the LEDs do something but do not function correctly, the problem is likely in your Quartus schematics or implementation. Check the implementation report file that the pin mapping is what you want. Check for any logic bugs.
7. When exiting the programmer window, don’t save anything. That way the design programmed to the PROM or FPGA is always the most recent build.

Programming the PROM with the Serial Flash Loader

Programming over JTAG only changes the volatile SRAM of the FPGA itself and does not write to the non-volatile PROM. The design will only stay on the FPGA until it is reset or power is disconnected (try it!). Programming the PROM directly would require adding an additional programmer port to the board, connected to the Active Serial (AS) interface between the PROM and the FPGA. Instead, it can be programmed over JTAG through the FPGA, using the Serial Flash Loader function in Quartus. JTAG is used to write a temporary design to the FPGA that effectively bridges the JTAG and AS interfaces, allowing the USB Blaster to talk to the PROM without being physically connected to it.

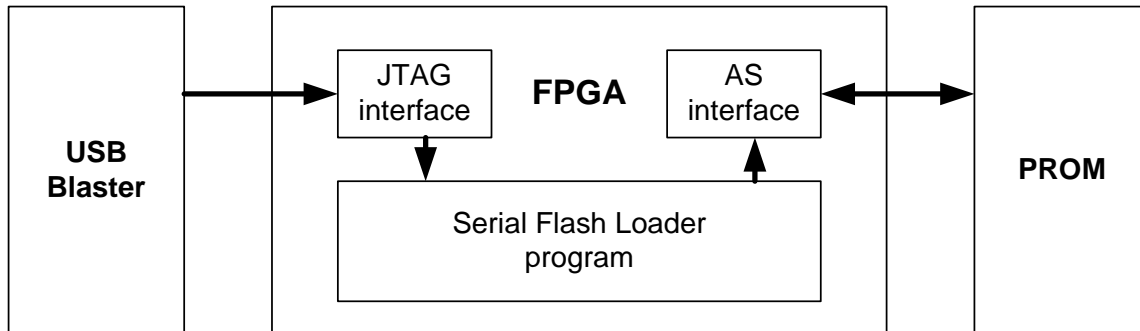


Figure 3 – Serial Flash Loader block diagram. A program bridges the JTAG and Active Serial interfaces to let the USB Blaster talk to the PROM.

When debugging your design, simple JTAG programming has advantages, namely that it is a much faster process. Losing the contents of the FPGA with every reset is less important if your design is being changed with every test. The Serial Flash Loader is much slower than JTAG programming and requires extra steps to complete, but should be used whenever the design is unlikely to be changed in the short term.

Always write your finished labs to the PROM before checkoff. Also remember that whatever design you last programmed to the PROM will be loaded immediately when the FPGA turns on. Forgetting this could cause you massive confusion when debugging, especially in Labs 4-7 when old code on the FPGA might interfere with pins you are trying to use on your PIC.

To set up the Serial Flash Loader, run Tools → MegaWizard Plugin Manager. Choose ‘Create a new custom megafunction variation’ and press Next. At the next dialog, expand the JTAG Accessible Extensions folder and select Serial Flash Loader from the options below it. In the output file name box, append ‘sfl’ to the path already there. Click through the rest of the wizard with the default options, and click Yes to add the file to your design.

The MegaWizard step need only be done once per Quartus project. The subsequent steps must be completed each time you program the PROM.

Compile your project to generate an up-to-date .sof file. Next, you will convert the .sof to a JTAG Indirect Configuration (.jic) file for use with the Serial Flash Loader. Choose File → Convert Programming Files. Set the programming file type to JTAG Indirect Configuration File and the configuration device to EPCS4. Pick a simple file name, such as lab1_xx.jic.

Under Input Files to Convert, select SOF Data and click Add File. Find your lab1_xx.sof and press OK. Now highlight Flash Loader and click Add Device. Select Cyclone III in the left pane, then EP3C5 in the right pane, and click OK. Click Generate and verify the .jic was generated successfully, then close the window.

Launch the Programmer and remove your old lab1_xx.sof from the programming file list. Now add the lab1_xx.jic file you just generated. Check both Program / Configure boxes and click Start. Watch the Messages pane for any errors. When the process completes, your design is loaded on the PROM. It will automatically be loaded onto the FPGA after reset or power-on until the Serial Flash Loader is used again to replace it.

If you want to erase the EPROM, change the EPCS4 option from Program/Configure to Erase and click Start again.

When you are finished, save you files somewhere (such as your network drive or a USB key) and remove the files from the local machine. Good luck!

You will test the PIC microcontroller in Lab 4 and the video output in Lab 7.

What to Turn In

Turn in a lab report with the following sections:

- **Introduction:** Briefly explain what was done.
- **Design and Testing Methodology:** Explain how you approached the design of this assignment from both a software and a hardware standpoint (as appropriate). Include how you tested your design. These test should convince the reader that the requirements of the assignment have been met.
- **Technical Documentation:** Include your Verilog code and schematics of your circuits.
 - Quality and clarity of your code is import. Make sure it is well commented.
 - Schematics of your breadboarded circuits should be sufficient for another engineer to understand and reconstruct the circuit on the breadboard. Always use standard symbols for standard components such as resistors, switches, transistors, diodes, etc. The reader shouldn't have to open Quartus to relate your Verilog code to the schematic. Don't assume that the reader has memorized the pinouts of any chips. There is no need to draw any of the circuitry on the board; just refer to it by the pins number and name.
- **Results and Discussion:** Did you accomplish all of the prescribed tasks? If not, what are the shortcomings? How might you address them given more time? As appropriate, how did the design perform (ex. How fast/accurate/reliable was it?). Is there anything you would do differently if you were to redo the lab? Is there anything else interesting worth mentioning?
- **Conclusions:** Briefly summarize what was done and how it performed.
 - How many hours did you spend on the lab? Any comments, suggestions, or complaints about the assignment? This will not count toward your grade, but will help refine the lab for the future.

The report does not need to be long, but should be complete. Some individual questions may not apply to this particular lab, but are listed to give you a general idea of what is desired. Future reports will follow a similar format and the questions may be more applicable in these instances.

Have your lab checked off by the instructor. You will need to demonstrate that the board and 7-segment display operate correctly. You will also be asked a question about some part of the lab or your board. You should be thoroughly familiar with all of the lab and the components of your board to be able to answer the question. The oral exam is typically in the form of a “Fault-Tolerance Question.” What would happen if a particular wire is broken or a pin is shorted to V_{CC} or GND? Be prepared for any other questions about your lab, however.

Acknowledgments:

This lab manual and previous utility boards have been developed by Profs. Matthew Watkins, David Money Harris, and Sarah Harris, Leo Altmann '11, Christian Jolivet '11, Elizabeth Reynolds '02, Marty Weiner '02, Fernando Mattos '00, and Eliot Gerstner '99.

Appendix: FPGA Initialization Sequence

If your FPGA board is assembled incorrectly or your PROM is programmed wrong, you may find that your FPGA fails to initialize. This sheet documents the initialization process to help you track down your problem. The initialization sequence is described in more detail in pages 9-9 through 9-14 of the Cyclone 3 Device Handbook (vol. 1).

The MSEL[2:0] pins on the FPGA are used to select a configuration scheme. They are connected to GND, VCC and GND respectively to indicate Active Serial configuration at 3.3V levels.

When the FPGA powers on or you release the reset switch (pulling nCONFIG from low to high), the device begins initialization, pulling nSTATUS and CONF_DONE low while clearing internal memory and preparing for configuration. In < 200ms, nSTATUS is then pulled high to indicate the start of configuration.

During configuration, the FPGA generates a 30 MHz clock via an internal oscillator and transmits the signal via DCLK to the EPCS4 PROM. The PROM responds by sending serial configuration data back over the DATA connection. The FPGA has 3,000,000 bits of internal state, so serial configuration via PROM should complete in about 100ms. When configuration is complete, CONF_DONE is pulled high again, and the device enters user mode < 0.5ms later. If CONF_DONE is never asserted or nSTATUS is pulled low after initialization completes, a configuration error has occurred. Check the soldering on the FPGA and PROM.

The configuration waveforms are shown below:

