

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

OBJECTIVES

In this document you will learn how to solder and to debug a board as you are building it.

REQUIRED MATERIALS

- Website documents
 - *UF 68HC12 Development Board Manual (board v4.2, manual revision 0)*
- UF 68HC12 Development Board Kit
- Altera EPM7064SLC44- CPLD (which you will get in lab)

PRE-CONSTRUCTION (must be done prior to the lab in which you will start construction)

Print out and bring all of the documents in the required material.

Bring your USB or parallel-port ByteBlaster cable from EEL 3701 (or buy one). A USB version of ByteBlaster can be purchased at the UF bookstore.

The UF 68HC12 board kit can **NOT** be purchased. Your lab fee will purchase the kit as well as some other items used in lab. **A wire wrap tool should also be purchased.** Optionally, you might also want to purchase a soldering iron, a wire cutter and needle-nose pliers. Bring the board kit and your tools to your labs. (There are soldering irons available at every workstation in lab and a few wire cutters and needle-nose pliers that are available for sharing.)

You should have received the parts shown in Table 1 in your *UF 68HC12 Development Board Kit*. Print this page out and bring it with you to be sure you get all the required parts.

Table 1: Parts required for construction of the UF 68HC12 Development Board Kit.

Part Description	Quantity
32kB EEPROM (28C256) or 32kB EPROM (27C256)	1
EPM7064SLC44-10 CPLD	1
74HC373 Latch	2
28 Pin Wide IC Socket	1
20 Pin Narrow IC Socket	2
40 Pin Double Row Header	3
20 Pin Single Row Header	1
Pink Light Emitting Diode (LED)	1
1k Ohm Resistor SMD 0805	8
4.7k Ohm Resistor SMD 0805	2
1M Ohm Resistor SMD 0805	1
10uF Electrolytic Capacitor	1
1uF Electrolytic Capacitor SMD 0805	2
0.1uF Capacitor SMD 0805	7
Momentary Pushbutton Switch	1
44 Pin PLCC Socket	1
ZTT 4MHz Oscillator	1
USB connector	1
Jumpers	10
1-inch Male/Female spacer pair	4

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

Your kit also includes the parts listed in Table 2. These part listed that will be added to the board later in the semester.

Table 3 shows the parts you will need from your EEL 3701 lab kits.

Table 2: Parts that will be added to your UF 68HC12 Development Board later in the semester.

Part Description	Quantity
8 pin narrow WW socket	4
16 pin narrow WW socket	3
20 pin narrow WW socket	6
28 pin narrow WW socket	1
40 pin wide WW Socket	1
34 pin WW socket strip (for DB-9/pot/SIP)	1
330Ω DIP RPAC (8)	1
10KΩ DIP RPAC (8)	1
1kΩ SIP RPAC (8) bussed	1
74HC574 D-FF	2
74HC573 tri-state octal latch	2
32Kx8 Static Ram 7C199 (15ns)	1
16C550 UART	1
RS-232 Line Driver MAX232	1
Serial Cable with DB-9 male and female ends	1
DAC0800 Digital to Analog	1
5532 Op-Amp	1
16 key keypad	1
2x16 LCD	1
1.8432 TTL Oscillator	1
10kΩ potentiometer w/key	2
Piezo Speaker	1
0.1μF bypass capacitor	15
1.0μF Electrolytic capacitor	2
Green LED	6
8 position DIP switch	1
DB-9F w/ 5 pin socket	1
1x16 male header (long pins)	1
9V battery snap	1
40KHz IR detector (Remote Receiver)	1
USB cable	1

Table 3: Parts needed from your EEL 3701 lab kit.

Description of Parts needed from 3701	Number
5V regulated Power Supply (AC Adapter) (OPTIONAL*)	1
Digital Multimeter	1
IC (chip) puller	1
Dip Switches (8x1)	1
LEDs (10x1)	1
2.7k resistor pack SIP (9/package)	1
1k resistor package (RPack) DIP (8/package)	1
Toolbox	1

* The power supply is not needed since power can be supplied through the USB connection to your PC.

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

Figure 1 shows an unpopulated board (with only the 68HC12 and the USB bridge chip mounted).

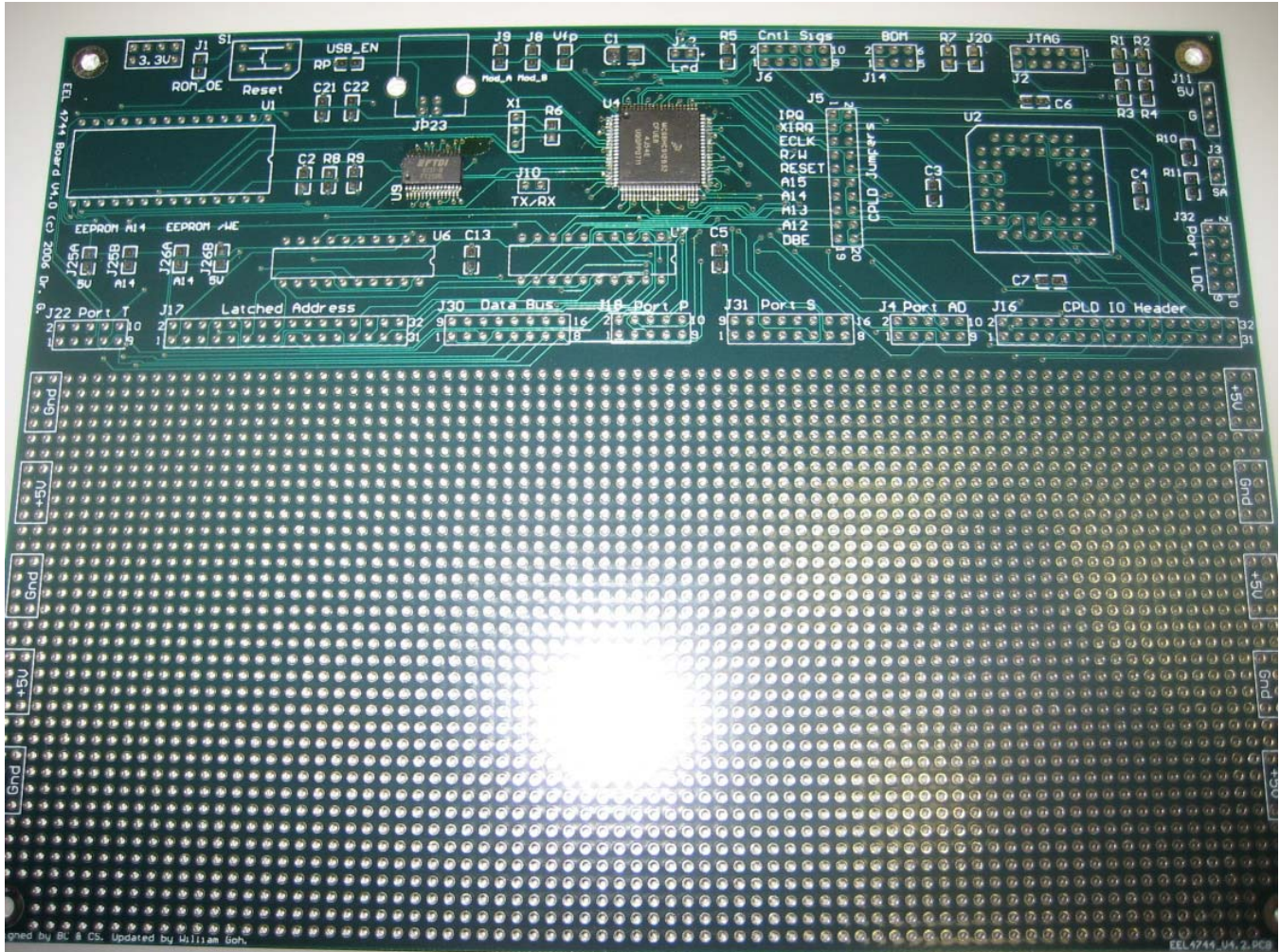


Figure 1: Unpopulated board.

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

Figure 2 show a partially populated board (missing only the headers). Figure 3 and Figure 4 show magnified views of Figure 2.

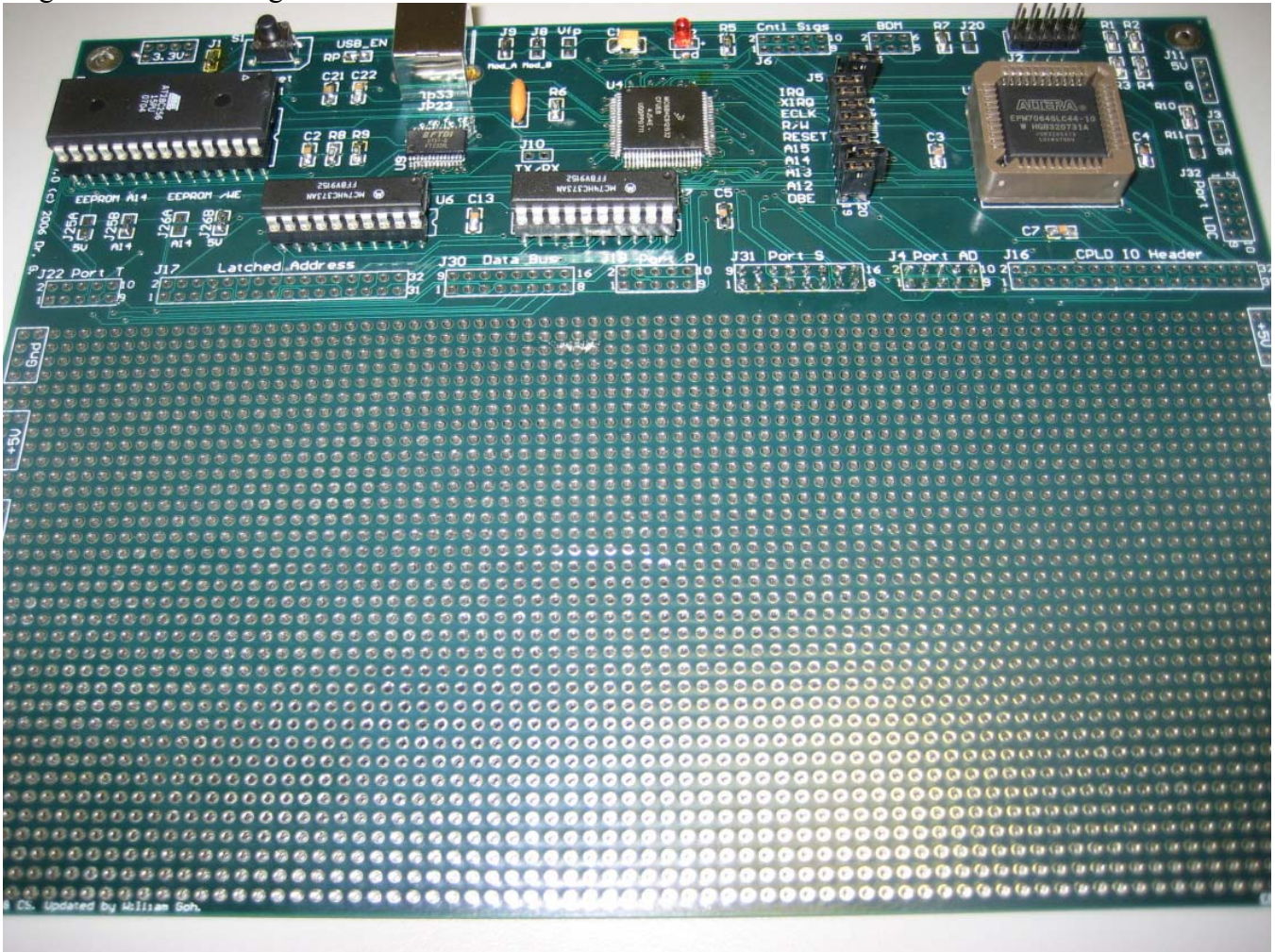


Figure 2: Populated board

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

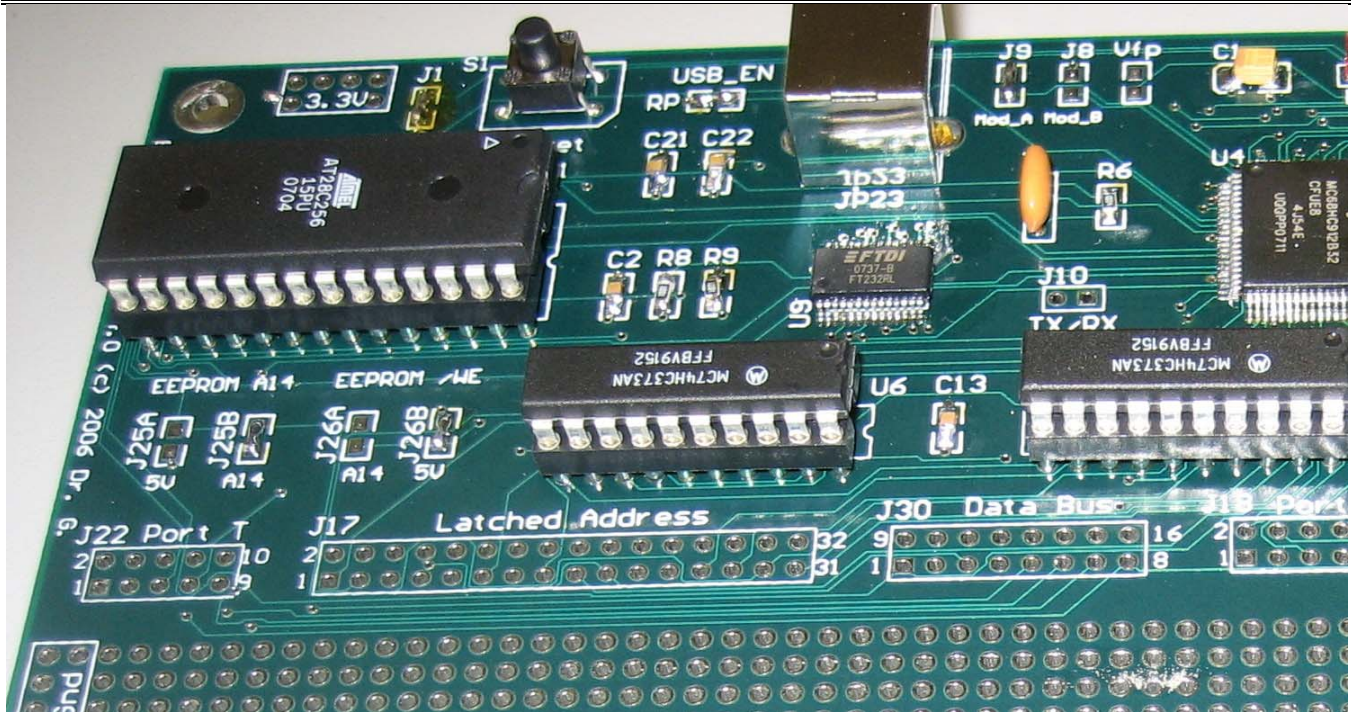


Figure 3: Left side zoom of populated board

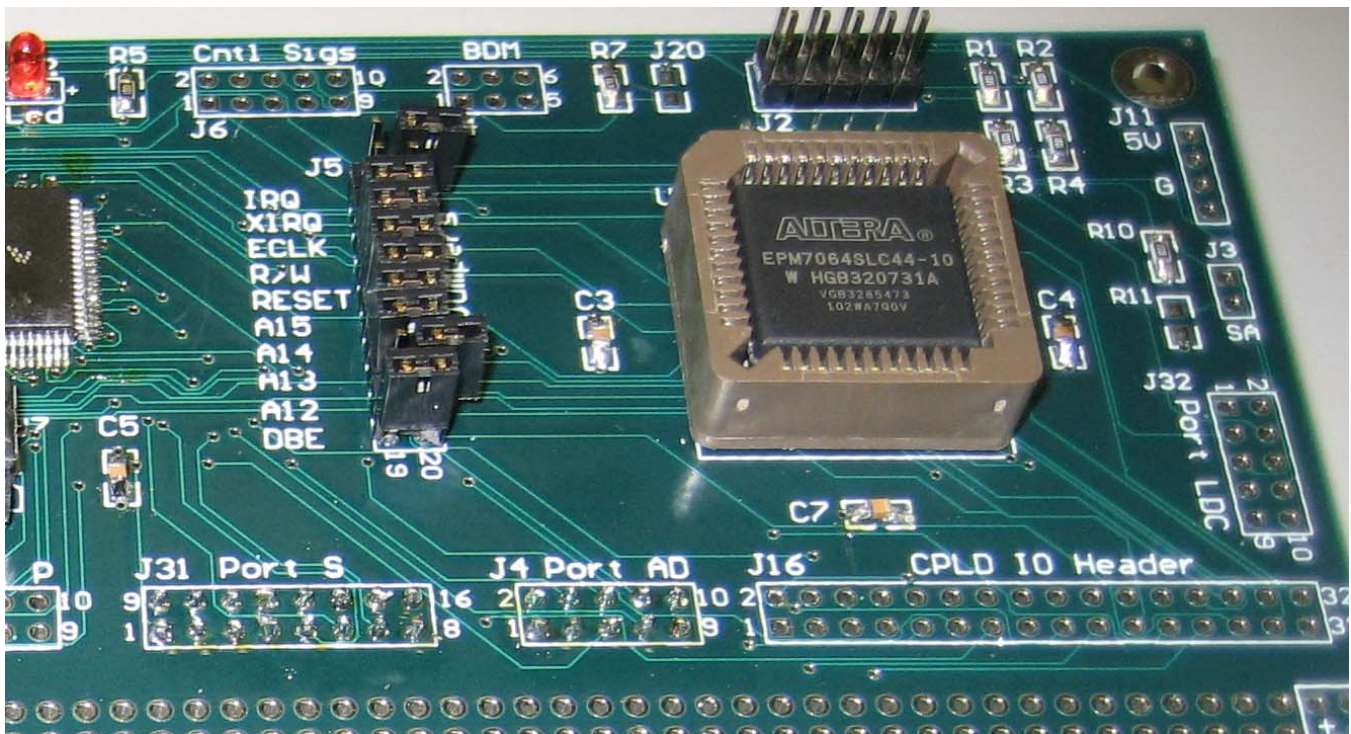


Figure 4: Right side zoom of populated board

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

PRE-CONSTRUCTION NOTES

(Read and keep in mind before beginning to solder)

- Be sure to place chips and sockets appropriately, i.e., line-up the notches on the sides of the chips/sockets with the notches on the silk screen (i.e., the printing on the top of the board).
- Make sure power is **NOT** connected when checking continuity and when adding parts to the board.
- The following parts should be pushed in from the bottom of the board.
 - J4, J6, J16, J17, J18, J22, J30, J31, (J32, if needed) headers are all placed from the bottom, with the plastic on the bottom of the board and the short pins going through the board to the top.
 - The Gnd and +5v headers on the edge of the board should also be placed in from the bottom.
 - The long ends of pins are on the bottom for wire wrapping.
 - All wire wrapping is done on the bottom of the board.
 - Solder to the short pins on the top of the board.
- The following parts should be pushed in from the top of the board:
 - J2, J5, J11 headers are all placed in from the top of the board, with the plastic on top of the board and the short pins going through the board.
 - The long ends of pins are on top for headers that are used with jumpers and connectors.
 - Solder to the short pins on the bottom of the board.
 - The power header (J11), the USB connector (JP23), the LED (J12), the JTAG header (J2) and all of the chips and chip sockets are all placed in from the top.
- All jumpers are on the top of the board.
 - Horizontally adjacent pins on J5 can be connected with female shunts (provided in your kit), when necessary.
 - Many locations on your board require only solder connections between adjacent pads. (Alternatively, pins can be inserted in these locations, and female shunts can be used.) These jumpers include: J1, J3, J8, J9, J10, J20, J25A, J25B, J26A, J26B. Only some of these solder connections will be made.
- Do **not** use boards from previous semesters to determine how headers should be mounted, since they may have been done incorrectly.
- The pinouts for headers are **NOT** like the pinouts for chips. For example, see header figures in the *UF 68HC12 Development Board Manual* in section 7.
- The following headers are not needed for our labs: J10, J14, J32
- There is no J7, J13, J15, J19, J21, J23, J24, J27-J29

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

LAB PROCEDURE

(Board Construction Procedure)

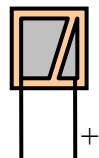
You should start this procedure **in the lab**. You may complete the procedure at home.

0. Look at the 68HC12 and the USB bridge chip connections to the PCB under the microscope. Verify that they are properly soldered to the PCB.
1. Be sure to place chips and sockets appropriately, i.e., line-up the notches on the sides of the chips/sockets with the notches on the silk screen (i.e., the printing on the top of the board).
2. Make sure power is **NOT** connected when checking continuity and when adding parts to the board.
3. Make sure to solder all surface mount parts first before proceeding with through-hole parts.
4. Solder the 10uF SMD Capacitors first as seen like Figure 1 onto C1. The stripe is the positive polarity which should be aligned with the curved edge just like Figure 3.



Figure 5: 10uF Tantalum SMD 3258

5. Next, solder the 0.1uF SMD Capacitors onto C2, C3, C4, C5, C6, C7, C13 pads
6. Solder the 1uF SMD capacitor onto C22 pad. Solder another 1uF SMD capacitor onto C21 pad.
7. Look for the resistor with an imprint of “1004” on it. This is your 1MΩ resistor. The 1004 means 100×10^4 . Solder the 1MΩ resistor on R6.
 - a. Verify the resistance with your multimeter before soldering it
8. Next, solder the 4.7kΩ resistor on R7. It should have an imprint of “4701”
9. Solder the 1kΩ resistor on R1, R2, R3, R4, R5, R8, R9, R10 pads.
10. Solder jumper joints on J1, J9, J25A, J26A pads. A jumper joint is just some solder connecting two nearby pads. In this figure J26B has a jumper joint and J25A does not. (For EEPROM solder J25B and J26B; for EPROM solder J25A and J26A.)
 - a. J9 (MOD-A) jumper boots the board into expanded mode.
 - b. Check J1 (Rom OE) continuity J16-pin 24.
 - c. Solder jumpers on J25A and J26A. These jumpers are used to select EEPROM and not EPROM in U2. If you have EEPROM, use jumper J25B and J26B. (If you have EPROM, use jumpers for J25A and J26A.) We will use EEPROM this semester.
11. If you wish to power the board via USB, short solder across the USB_EN pad (RP). If you don't wish to use the USB power, the board can be powered using the power supply from EEL 3701. See Power Options below for more details.
12. Solder the USB connector on JP23. Also solder the support pins to provide a mechanical connection and strain relief.
13. Place the LED on header J12 (on top) with the + on the side as indicated in the LED figure and solder the LED in place. Push the LED in as far as possible.
 - a. Plug in your USB cable or power supply. Your LED should come on.
 - b. Unplug your USB or power supply!
14. Install the reset switch into S1. Solder.
 - a. Set the multimeter to beep when there is continuity (or to a small resistance value).
 - b. Hold a multimeter probe on the reset switch by the S1.



Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

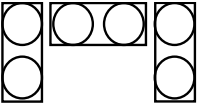
- c. Hold the other probe on any GND point on the board.
 - d. When the button is pressed, the beep should sound (or the resistance should go to almost zero).
 - e. The switch should not cause continuity unless the switch is pressed.
15. Insert and then solder the 28-pin socket into U1. Be sure to line up the notch on the silk screen and the socket before soldering. Do **not** solder the EPROM chip directly to the PCB. **Use the socket.**
16. Insert the CPLD socket into U2. Be sure to align the angled corner on the socket with the angled corner on the silk screen. (Check the alignment **again** before soldering!)
- a. Press the socket firmly while touch-soldering several corner pins.
 - b. Now solder all the other pins and finish up by re-soldering the pins that held the socket in place.
 - c. Test continuity on several of the CPLD pins to the CPLD I/O header (J16).
 - d. Observe a small curved edge on the CPLD and the CPLD socket. That curved should be aligned together before inserting it. This is very important. If you put your chip in wrong, you will destroy it. (See Figure 4.)
17. Insert and then solder the 20-pin sockets into U6 and U7. Be sure to line up the notch on the silk screen and the socket before soldering. Do **not** solder the chips directly to the PCB. **Use the sockets.**
- a. Verify continuity between power on each of the U6 and U7 sockets and one of the +5v on the side of the board. The power pin for these sockets is pin 20.
 - b. Verify continuity between ground on each of the U6 and U7 sockets and one of the Gnd on the side of the board. The ground pin for these sockets is pin 10.
 - c. Test continuity between the 6812 address pins (AD15-AD0) and the 373 input pins.

	AD7	AD6	AD5	AD4	AD3	AD2	AD1	AD0
68HC12 pins	25	24	23	22	21	20	19	18
U6's 373 pins	18	17	14	13	8	7	4	3

	AD15	AD14	AD13	AD12	AD11	AD10	AD9	AD8
68HC12 pins	46	45	44	43	42	41	40	39
U7's 373 pins	18	17	14	13	8	7	4	3

18. Insert and then solder the oscillator (marked 4.0 something) into X1. Polarity does not matter.
19. Put on the header for J5. The header has to be placed on from the top of the board, with the plastic on top of the board and the short pins going through the board.
- a. Place jumpers on the following pin sets: DBE, A12, A13, A14, A15, Reset, R/W, ECLK, IRQ. Do **not** place a jumper on the XIRQ pin set.
 - b. Verify the continuity between the jumpers and the corresponding pins on the CPLD I/O header (J16).
 - c. Move the jumpers at A12, and IRQ to the "off" position, as shown in Figure 4 with XIRQ and IRQ. (The reason A12 and IRQ jumpers were first placed added and then removed was to test them for continuity. In future labs you may reinstall these jumpers.)
20. Solder the JTAG header (J2). This header has to be placed on from the top of the board, with the plastic on top of the board and the short pins going through the board.

Building a UF 68HC12 Board (Version 4.2) from a Development Board Kit

21. Align the notch of a 74HC373 with the notch on the U6 socket (and silkscreen) and insert it into the socket. Repeat this process with another 74HC373 and U7.
22. Add +5V and GND headers (as needed) throughout the semester on the sides of the board. The easiest thing to do here if you want to use all six pins is to use three 2-pin headers as shown in the figure.
23. For the headers J4, J6, J16, J17, J18, J22, J30, J31, J32, insert short side of the header from the bottom of the board and solder from the top of the board.
24. Use Quartus to create the following logic equations:
$$\text{ROM_OE} = \text{A15} * \text{A14} * \text{A13} * \text{RW} * \text{/Reset} * \text{DBE}$$

(Some have also used: $\text{ROM_OE} = \text{A15} * \text{A14} * \text{A13} * \text{RW} * \text{/Reset} * \text{ECLK}$)

$$\text{ECLK_POUND} = \text{/ECLK}$$

Note that Reset, DBE and ROM_CE are all active-low. All other signals above are active-high. See your UF 6812 Board Manual for more information about the equations.
25. Wirewrap a connection between PAD7 on J4 (also known as ANA7) to ground. I suggest that you use the ground on pin 16 of J31. The PAD7 pin is monitored during boot-up by the D-Bug 4744 monitor program, version v2.4 or later. (Pin 7 on the A/D port, PAD7, is pin 2 on J4.)

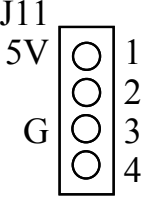
POWER OPTIONS

UF 68HC12 board can be powered via 2 options. It can be powered through the USB cable or with a 5V regulated power supply connected to J11. If you have set the board to be powered via USB, do **NOT** connect the 5V power supply to J11.

USB Power

Solder a short across the RP (USB_EN) jumper.

5V Regulated Power

1. Solder a header onto J11 from the top.
 2. Make a power supply connector (using your **5V regulated** power supply from 3701)
 - a. The connector should have +5V on pin 1 and GND on pin 3.
 - b. The power connect (J11) pinout is shown here, where G represents ground (Gnd). The cable should mate with this pinout. Be **absolutely sure** that you wired this connector correctly **before** plugging it into your board.
 - c. Plug in the power supply and verify that the power to the board is +5V. Measure the voltage between two labeled points on the side of the board with the positive terminal at +5V and the negative terminal at Gnd.
 - d. Unplug the power supply!
- 

You have now completed the board construction. See lab 1 to further test your UF 68HC12 Development board.