ECE 4723/6723 Embedded Systems Lab (section 2)

Team Scons Sucks

Task 2: ESOS and the ECE 4723/6723 Target Board

Alex Felder (af1460)
Josh McCown (jpm623)
Wes Yarber (way16)
Joseph Deladillo
Matt Beach

Build and Test Procedures

Preliminary Testing

Before beginning the build process, the schematic was examined closely for errors. Perhaps an intentional omission - the RTS pin on the FTDI header connection was lacking a connection to MCLR for easier programming. Other than that, no errors were found on the schematic. Next, the board was tested to ensure conductivity between different intended points shown on the schematic. At this point, various errors were discovered. It is possible that not all errors were discovered, but the errors discovered are as follows:

- Missing connection between RXD on SV1 headers to RF0 on H1 headers
- Missing connection between R7, R6, and V0 (V0 is still connected to R7)
- The SDA and SCL pins are not connected to the SCL1/RG2 and SDA1/RG3 pins.

These errors were noted to be fixed after the components were fully installed. Once preliminary testing was completed, we moved on to populating the board. As the board population went along, new connections and components were each verified for continuity with points where they should have connection and verified to not have continuity with points with which they shouldn't have connection. This is being mentioned now so that it does not have to be repeated for each new connection.

Board Population

Power

The power sub-section was the first to be populated. The LM2937 was soldered into its place first, then the 0.1µF capacitors C1, C2, C3, C4, C5, and C6. Next, the 10µF capacitor C7 was soldered into place, ensuring the correct polarity according to the silkscreen and verified with a continuity check in reference to the given schematic. Next, R1, the 1.5kΩ resistor was soldered into place. Next, the power jumpers, labeled JP1 on the silkscreen, were soldered into place, ensuring consistent spacing between the header pins to easily later connect the jumper. Next, the reset switch was soldered into place and tested for lack of continuity in the default state and continuity in the depressed state. Finally, the 3.3V power LED was soldered into place, ensuring the correct polarity with a continuity check in reference to the schematic. This completed the power portion of the build.

MCU

In this very important step, the dsPIC33EP512GP806 MCU was to be soldered into place. Before soldering, the proper orientation of the MCU was verified by testing for continuity to ground on three sides of the chip solder pads and lack of continuity to ground on the fourth side. This was verified with the datasheet for the PIC to ensure correct orientation before placement, which matched the placement suggested by the silkscreen. The PIC was put in place and soldered in. This was accomplished by holding the PIC in place with tweezers and running the soldering iron tip along the pins on one side of the chip at a time until the solder melted and the pins were able to connect to each of their corresponding solder pads. Extra care was taken to ensure correct alignment of the pins as well as no continuity between neighboring pins to avoid unintentional solder bridging.

MCU Output LEDs

After the MCU was soldered in place and rigorously inspected and tested, it was time to solder it's output LEDs in place. After verifying correct orientation of the LEDs as done for the previous 3.3V power LED, the LEDs were soldered into place. Then, their three corresponding 1.5kΩ resistors, R2, R3, and R4, were soldered into place.

MCU FTDI & ICSP Connections & IO Headers

Next, the six FTDI and six ICSP (SV1) headers were soldered into place. Then the three IO headers, H1, H2, and H3 were soldered into place. At this point, two of the jumper wires mentioned in the preliminary testing were ready to be soldered in - one between MCLR pin 1 on the SV1 headers to RTS pin 6 on the FTDI headers, and one between RXD pin 5 on the FTDI headers to RF0 pin 1 on the H1 headers. This allowed the board to be properly programmed through the FTDI headers after having the bootloader flashed over the SV1 headers.

MCU Inputs

Finally, the MCU inputs were put into place and soldered, starting with SW1 and SW2. These push button switches seemed to be a little oversized for their allotted space on the PCB, so one leg was bent under each of the switches before soldering them both to the solder pads. This fit the switch to the smaller footprint set aside for them on the board. Next the RPG1 rotary encoder and POT were soldered into place. This completed the building and testing of the necessary components needed to complete lab 2.