Team APPLE

Wyatt Gamble

Andrew Hegman

JD Ford

James Tate

Populating the ECE4723 Lab Project Board revF14

Power Subsystem

The ECE4723 lab project board revF14 can be powered from 2 different sources. The first choice is a 26V max voltage supply using a 3mm connector. This connector is denoted as X2 on the F14 board. The second is using the 5V supply that is powered from a USB supply from the FTDI breakout connection. The 3mm connection will be our starting point. Thoughout the process of bringing the F14 board up we will use a multimeter to test components of each subsystem. For these test we will use the GND connection on the H3 breakout strip as a ground reference point.

After soldering the 3mm connector onto the F14 board, test the solder connections across GND and pin #3 of JP1. The voltage output should be exactly the output voltage of the power supply. **IMPORTANT**, after each test that involves having power connected to the board ALWAYS unplug and disconnect the power supply before continuing. With the power supply unplugged and disconnected, stuffing JP1 is the next step. This is where the decision to power the board from the 3mm connector, or the 5V FTDI supply is made. Header pins can easily be snapped to fit and soldered in place for this connection.

With JP1 stuffed and soldered, use a female header wire to jump pin 1 to pin C of JP1. This will allow for voltage from X2 to pass to VRAW onto the voltage regulator IC3 (LM2937). After the jumper wire is attached ensure the voltage that is going to the solder pad of pin 1 on the LM2937 voltage regulator matches the input voltage from the power supply. The pin assignment can be found on page 2 of the datasheet.

At this point we have a raw power supply for the board. Since the DSPIC33FP512GP806 processor has an operating voltage of 3V-3.6V, a voltage regulator must be used. IC3 will take the raw input voltage and supply the board with 3.3V. Page 9 of the datasheet for the LM2937 shows the schematic for connecting the voltage regulator. The capacitor assignments for the board are as follows: Cin = C6, Cout = C7. IC3 will also have a power LED attached to pin3's node with R1 as a driver for this LED. Once the capacitors, resistor R1, IC3, and the power LED has been soldered in place, power up the board again and ensure the LED illuminates. Test the voltage on pin 3 of IC3 to ensure the output is 3.3V. Also ensure that voltage is being passed to

the MCU solder pads. An assignment of VDD and VSS can be found on page 6 of the MCU's datasheet.

MCU

The PIC33 has minimum connections that are needed before it can be powered up. Page 31 of the PIC33 datasheet instructs that a decoupling capacitor must be used across each set of VDD and VSS supply pins. Page 32 shows a diagram of this requirement. Five .1uF capacitors are required to meet the decoupling requirement. A single 10uF capacitor is required for the VCAP pin. A push button switch will be used for bringing MCLR low in the instance that the PIC33 needs to be reset.

Start by stuffing and soldering C0 (VCAP capacitor), and the decoupling capacitors C1-C5. Next stuff and solder the RESET push button switch and R0. After these components have been soldered in place, the MCU can be soldered onto the board. The MCU has small pins so time and patience will need to be taken during this task. It might be best to do this when you are calm and have not had a lot of caffeine recently. Ensure the MCU is orientated correctly by aligning the dot on the chip, with the dot on the silkscreen. Use your own technique for soldering this in place.

Once the MCU is soldered in place visually inspect the solder joints for bridges. The use of a microscope can aid in finding solder bridges. Next test the integrity of the soldered connections by using a multimeter. Look for shorts between the pin and connections to the pins breakout location. A couple of methods will need to be done to check all the pins. Some pins can be check for shorts (for instance pins 33 and 34) by checking across there breakout connections on H1. Another method for checking for shorts is by checking a pins breakout location, and the pin next to the assign location. To do this place one lead on the breakout location assignment, then place the other lead on top of the MCU pin assignment where it comes out of the MCU's package. This will ensure the solder connection was successful. Next to check for a short place the lead on the adjacent pins. Before continuing test each pin to ensure no solder bridges have been made, and all breakout locations are connected to the MCU. Next power up the board and ensure that the MCU is receiving 3.3V across its VDD/VSS pins. If this is not the case power down the board and inspect for solder bridges.

LED's

Place and solder the three MCU output LED's and the corresponding resistor drivers for these LED's. Ensure that the resistors being used are large enough to drive the LED's safely. Before placing the LED's ensure the correct polarity by checking page 1 on the datasheet for the orientation. A multimeter in diode mode can also be used to check the polarity. A correct lead orientation should illuminate the LED.

FTDI breakout

Stuff and solder the FTDI breakout by using header pins. Test the integrity of the solder connection by using a multimeter to test connectivity between the header pin and the corresponding MCU pin. The second power test can now be done. Move the jumper from JP2 pin 3 to pin 1. Plug in a FTDI cable and verify the raw power to IC3 is 5v and the MCU is receiving 3.3V.

ICSP breakout

Repeat the process of soldering and testing the ICSP breakout that was done on the FTDI breakout.

MCU INPUTS

Place and solder SW1, SW2, SW3, RPG1, and POT to the F14 board. SW3 and RPG1 is on the same component. Test the functions of the components by using a multimeter connected to the corresponding MCU pins. If the connectivity test to the corresponding MCU pins fails, check the integrity of the solder connections at the component.