Investigation and Verification of can2\_revF14\_FTDI schematic and PCB

Initial review of the schematic revealed a central pad for the MCU, a power delivery subsystem, two programming options, communication breakouts (I2C, SPI, and UART), multiple headers for easy I/O interface, various built-in project components (3 LEDs, a potentiometer, 2 SPST switches, and a rotary encoder with push switch), a DAC converter, support for the MCU’s onboard ADC converter (LM4128 Voltage reference), LCD display, and a thermometer.

There is a capability for CAN bus driver IC with two jacks that will not be implemented.

Confirmation of the PCB layout and functionality proceeded from major to minor subsystems.

The orientation of the 64 bit MCU was confirmed by checking continuity between expected GND pin locations and the board’s GND using the audible continuity function of a multimeter. In other words, does the dot on the board’s MCU position match the IC’s dot. It does. Connections between all VDD to 3V3 locations and well as VSS to GND were confirmed next. Using the appropriate Microchip datasheet, all required smoothing capacitor locations, connectivity, and values were confirmed. Special attention was paid to the 10uF capacitor on VCap as the datasheet emphasized it’s importance.

The power distribution system was verified next. The MCU’s 3V3 needs can be supplied by the picKit programmer on header SV1 but there must be a jumper on JP4. The JP1 header assembly can be thought of as three way power selector for the board’s other power supply options. The FTDI header deliveries 5V supply which can be connected to the VRaw line of the LM2937-3 voltage regulator. The other power supply option would be from the PWRCONN1 connector which is currently not implemented. Various resistor and capacitor values and connectivity were confirmed at this time via multimeter and datasheet reference.

The programming options were investigated next. In Chip Debugging/ Programming (ICDP) using the PICkit can be accomplished through the SV1 header. 3V3 power can be supplied to the board through the jumper on header JP4. Connectivity for all these connections was verified. As long as the bootloader program exists on the MCU, the UART connection on header FTDI can be used to download programs. The Bully Bootloader is used for this purpose. Connectivity of all pins was verified and some errors were found. The RTS# pin on the FTDI is used to single a reset via MCLR#. This connection does not exist and must be made using a jumper wire. Also the MCUTX line is currently connected to a reprogrammable pin RPI43 that can only be an input. The serial IO library lists pin RF0 as the proper connection for the F14 board revision. It can be found on the IO header H1 and must be wired externally. Power to the board is supplied from the FTDI cable. While the comm lines are the proper 3V3, the USB power line is 5V and must be passed to the LM2937-3 voltage regulator through the JP1 header and appropriate jumper. Connectivity and resistor/capacitor values were confirmed via the LM2937 datasheet. The C7 electrolytic capacitor polarity must be observed.

Onward to the serial comm options. The UART pins on the FTDI header have been discussed already. Those are the only UART pins that have been broken out to a useable position. The I2C bus pullups resistor values were verified by referencing a Texas Instruments Application Report (see link on parts spreadsheet). The connectivity between the I2C header pins on H2 and the MCU is absent. External wires will need to be used again. The DAC IC MCP49X2-S is communicated with via the SPI bus. The chip select pin CS# is not connected and must be hardwired also.

The LED pads are all oriented with GND on the right. The current limiting resistors will prevent exceeding the IO port maximum current sink/drain of 200mA. Calculated by measuring the LED forward voltage drop, subtracting that from 3.3V and dividing by the resistance of 1500ohm. The 2 switches, potentiometer, and rotary encoded were also checked for connectivity to ground and IO pins.

The NHD-NHD0208AZ display uses the parallel port E. Connectivity of IO pins to the display as well as VDD and VSS were tested and verified. The specified resistors should provide sufficient contrast but might be changed after observation.

In summary, there are 5 external wires that must be added for full functionality. The 2 on the FTDI header are of immediate concern unless the PICkit is used for all programming. This option will prevent utilization of the UART.

Addition for Lab 4.

IC 1 is a TI LM60CIM3 temperature sensor. It’s 3.3V power supply exceeds the minimum 2.7V expressed in the data sheet. There is a smoothing capacitor on the output that is not technically required but will only help. The output is hardwired to pin RB3 which will need to be configured as an analog input.

IC2 is a TI LM4128-3 3.0V precision voltage reference. It is hardwired to pin RBO which is the alternate Vref+ input for the ADC. The Enable pin can be either solder jumpered to VDD or RD1. We chose to hardwire the chip always on. Vin can be connected to VDD or VRAW. We chose the 3.3V VDD option because it is always available. There are the ubiquitous smoother capacitors on input and output voltage lines.

The previously mentioned potentiometer, with smoothing capacitor, has its tap connected to RB2. Again this pin will need to be configured as an analog input.

Addition for Lab 5

The New Haven Display NHD-0208AZ-FSW-GBW-33V3 is a 2 row by 8 character LCD display. It has 8 bi-directional data/address lines and 3 control lines. We wrote a test program that sequentially turned on each line so that we could verify proper PCB connectivity and solder effectiveness. No problems were found. There are two resistors that act as a voltage divider providing a low but not zero voltage on the Vo intensity line. These values were confirmed by the datasheet and through observation.