Zane Gyorko
5 August 2021
5193-6096

FINAL PROJECT – Report

Introduction

The final project for this course is an open-ended design project based around the PIC18F47K40 microcontroller that was utilized in earlier design modules. The project shall meet a set of specifications that will be outlined in the design section. These requirements include mainly several analog and digital inputs and outputs. The functionality of the end design is left to the student to decide.

A number of design techniques and skills built earlier in the semester will culminate into this one design. These skills include PCB design and microcontroller programming, among others. The main difference between earlier modules and the final project, other than the larger scope, is the requirement to design and fabricate a PCB on which the project will be built.

Design

The requirements given for the final project are as follows:

- Display status of the system to an LCD
- Input an analog signal using internal ADC of PIC
- Output an analog signal via serial to the external DAC IC
- Input a digital signal
- Output a digital signal
- Place 85% of components on a PCB
- Add an extra input or output for full credit

My design will ultimately consist of buttons, switches, and potentiometers to control the appearance and state of an LCD display and LEDs. One each switch and potentiometer will be used to control two respective LEDs.

The potentiometer value will be read in using the built in ADC of the PIC. That value will correspond to a brightness level of LED1. The position of the potentiometer will feed a value to the external DAC via a SPI interface. The DAC will then send a signal anywhere from 0-5V to LED1. This brightness in voltage will be displayed on the LCD.

LED2 will be in either an on or off state based on the position of the toggle switch. The state of LED2 will also appear on the LCD. A pushbutton switch will be added as an extra input to override the potentiometer value and force LED1 to receive 5V regardless of the position of the wiper.

All components will be soldered to a PCB either directly or via a header connection to meet the 85% PCB placement requirement.

The hardware was designed in accordance with the following hardware block diagram, to be used as an outline for creation of more detailed schematics.

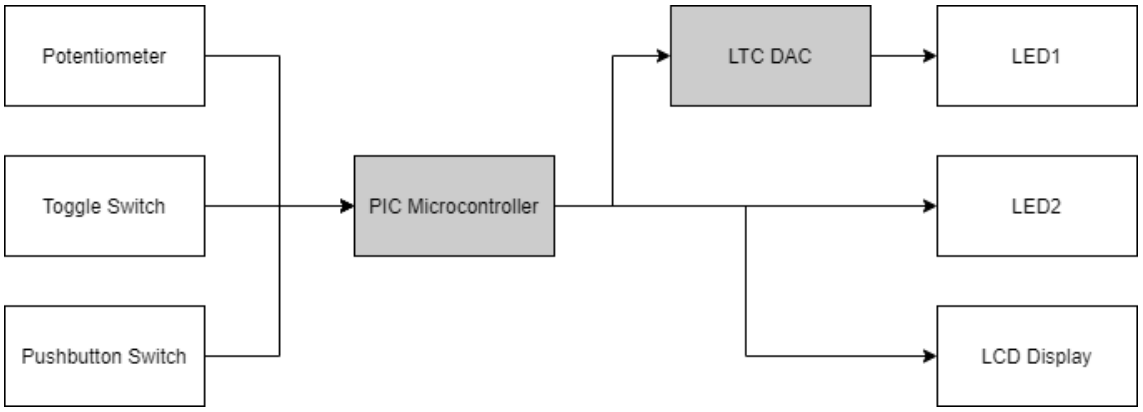

Figure 1. Hardware block diagram

As shown in the block diagram, requirements for inputs and outputs are met, along with the use of the liquid crystal display, and LTC DAC. A more detailed schematic is given below.

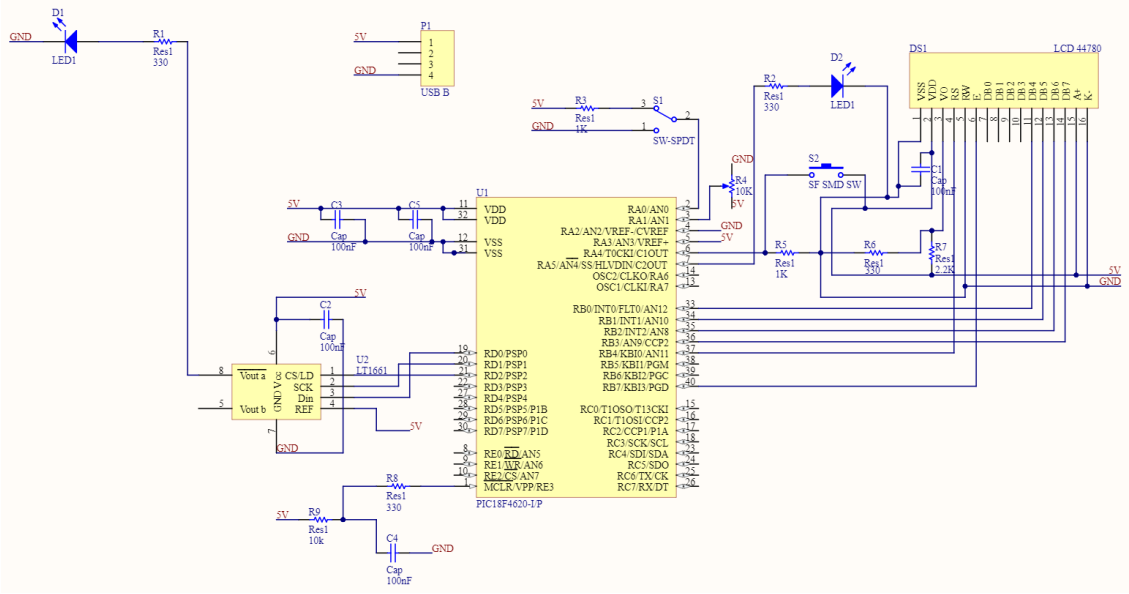

Figure 2. Altium schematic

From this schematic, a PCB was designed. Decoupling capacitors were added along with other power related circuitry to keep the device stable in operation. A USB type B header was added to

the board for easy and convenient power delivery. The following figure shows the layout of the PCB that was used in the final design.

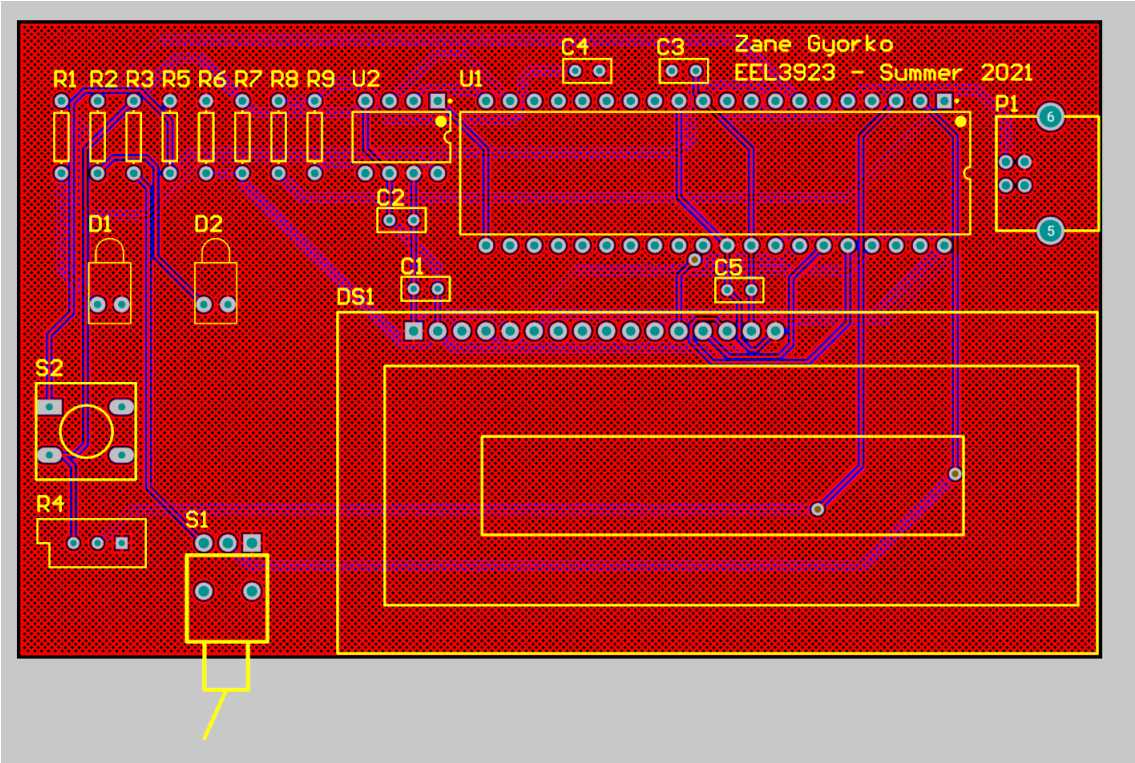

Figure 3. Altium PCB

Multiple considerations were taken into account when making the PCB. Decoupling capacitors were placed immediately next to the power connections for all ICs on the board. An ergonomic layout was desired, and as a left handed person I preferred to keep controls an the left side of the device. The controls for LED1 are aligned vertically, separate from the other set of controls for LED2, also aligned vertically. With the use of through hole components, a relatively large microcontroller, and a rather large LCD, the size of the board is quite big considering its functionality. However, with the board being quite large, routing was easy and placement could be organized logically to afford an easier assembly process.

On the software side, most of the code utilized had already been developed in earlier modules. The bulk of the code exists in initialization functions for the peripherals being used by the project (ADC, SPI, LCD, etc.). The following software flowchart explains the procedure by which the inputs are polled, and outputs are updated.

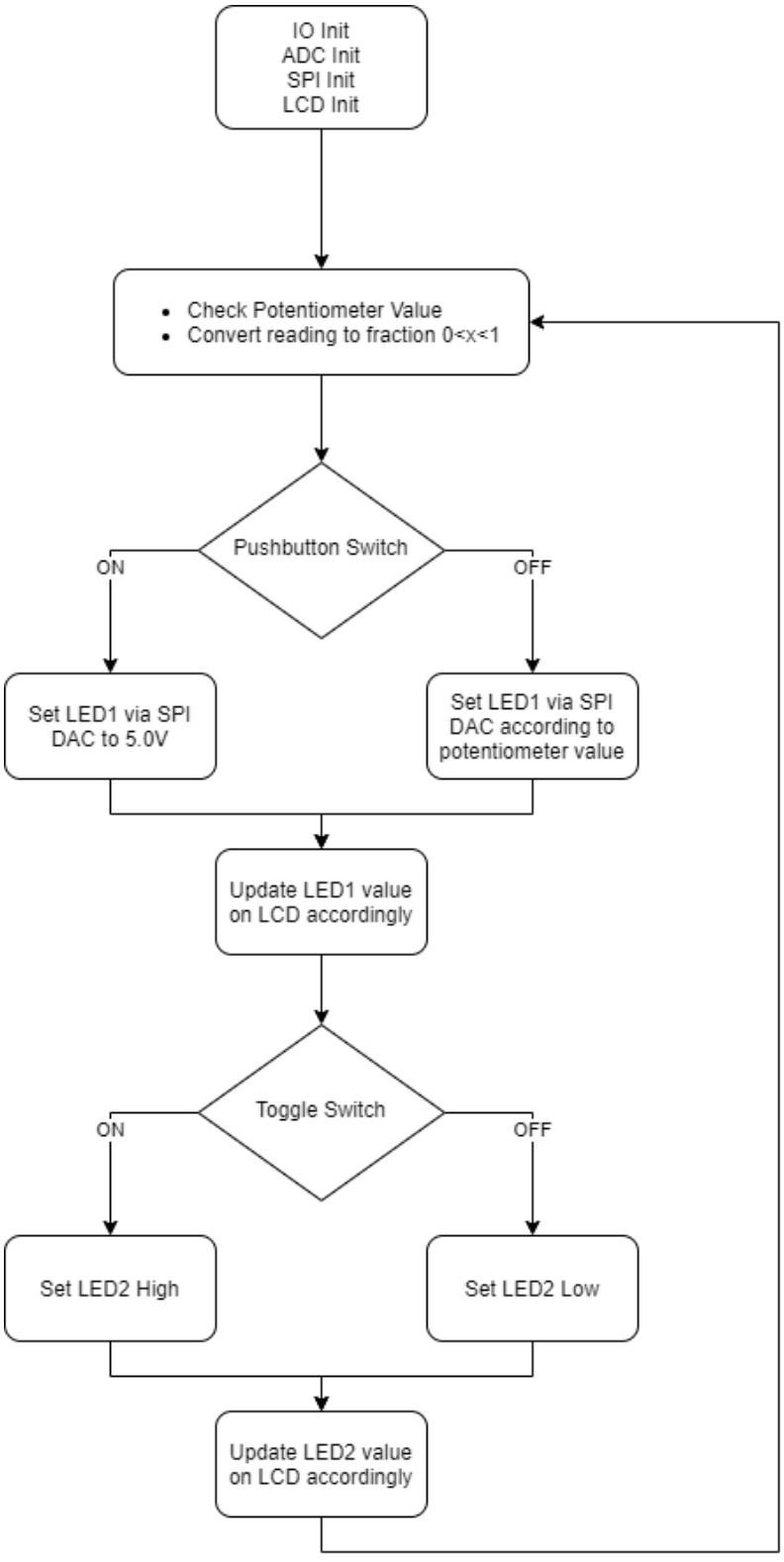

Figure 4. Software flowchart

With the design complete, circuit implementation could proceed. A breadboard prototype was designed and working before PCBs were put on order. A first set of PCBs contained some errors including a swapped pair of data connections and improper power distribution that rendered them useless. After redesigning the PCB and reviewing the design with a TA, the final design shown above was settled on. Those PCBs arrived before the deadline of the project, and assembly and testing moved forward. The final board performed flawlessly and passed the demonstration.

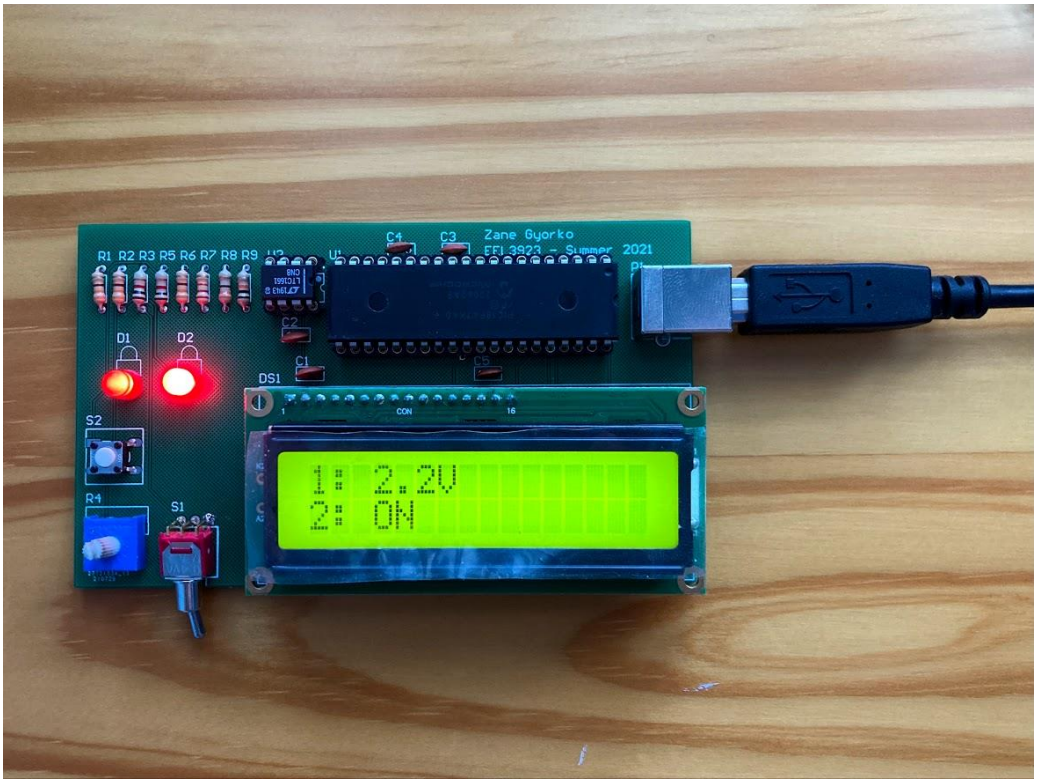

Figure 5. Final Project Assembly

A bill of materials was generated using the final design.

Design 1 - Final Project - Bill of Materials				
Description	Quantity	Price/Part	Part Number	Source
Resistor - 330 Ohms - Through-Hole	4	$ 0.10	CF14JT330R	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/CF14JT330R/1741399
Resistor - 1k Ohms - Through-Hole	2	$ 0.10	CF14JT1K00	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/CF14JT1K00/1741314
Resistor - 2.2k Ohms - Through-Hole	1	$ 0.10	CF14JT2K20	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/cf14jt2k20/1741321
Resistor - 10k Ohms - Through-Hole	1	$ 0.10	CF14JT10K0	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/cf14jt10k0/1741265
Potentiometer - 10k Ohms - Through-Hole	1	$ 0.47	500E-222	https://www.digikey.com/en/products/detail/nte-electronics-inc/500E-0224/11652015
Capacitor - 100nF - Through-hole	5	$ 0.45	K104K15X7RF53H5G	https://www.digikey.com/en/products/detail/vishay-beyschlag-draloric-bc-components/K104K15X7RF53H5G/1327984
LED - Red - Through-hole	2	$ 0.18	L513SRD-B	https://www.digikey.com/en/products/detail/american-opto-plus-led/L513SRD-B/13556894
SPDT Toggle Switch	1	$ 1.66	ANT11SF1CQE	https://www.digikey.com/en/products/detail/cit-relay-and-switch/ANT11SF1CQE/12503396
Pushbutton Switch	1	$ 0.34	B3F-1020	https://www.digikey.com/en/products/detail/omron-electronics-inc-emc-div/B3F-1020/44059?s=N4IgTCBcDaIEYGYBm
LCD Display	1	$ 6.39	CFAH1602B-NGG-JTV	https://www.crystalfontz.com/product/cfah1602bnggjtv-lcd-16x2-character-display
LTC DAC	1	$ 5.53	LTC1661IN8#PBF	https://www.digikey.com/en/products/detail/analog-devices-inc/LTC1661IN8-PBF/961738
PIC Microcontroller	1	$ 2.88	PIC18F47K40-I/P	https://www.digikey.com/en/products/detail/microchip-technology/PIC18F47K40-I-P/6208289
USB Type B Header	1	$ 0.63	U2J-BH-1-TH	https://www.digikey.com/en/products/detail/cui-devices/UJ2-BH-1-TH/6187914
	Total:	$ 21.31		

Conclusion

The design and implementation of this module was overall a success. A very simple design was chosen such that all requirements would be met while not calling for excessive complication. Given the time constraints near the end of the shortened semester, this decision proved to be beneficial as the project was finished before the deadline and met all requirements to earn a grade of 100%. A few issues were encountered along the way relating to PCB design; however, they were easily overcome.

With more time, a more efficient program could have been implemented. While the functionality of the device is complete, the speed is lacking. Interrupts could have been utilize dot further improve efficiency. With much more time available, a different device could have been designed with more practical use cases. That would not have been necessary, only more entertaining. Ultimately, though, the final design was fully operational and the device meets all design requirements.