PIC24 System Schematic Creation in PCB Artist

2nd Laboratory Report for ECE 383 Microcomputers

Submitted by
Patrick Brooks
11650957

The University of Alabama
Tuscaloosa, AL 35487

1/29/21

Abstract:

The main objective of this lab is to introduce students to the creation of a partial PIC24-based schematic and printed circuit board layout in PCB Artist. A secondary objective of this lab was to increase student familiarity with the PCB Artist software. Students will learn how to design electronic circuits using this software. Task one asks students to research the voltage that will be outputted by the LM2937-3.3 component, as well as the max amount of current that it can supply. Their findings will be recorded and put into the final lab report as "deliverable 1". Task two instructs students to create a basic power circuit for the PIC24 system. This circuit consist of the PIC24HJ128GP502 microprocessor, the LM2937-3.3 voltage regulator, Vdd and GND components, and five capacitors. After creation of the circuit, students will screenshot their schematic to be used in the lab report. Task three is where students convert their schematic from task two into a PCB. This is done by using the "Schematic→PCB" tool. Students will follow steps from the first lab in order to produce said PCB. Once created, a screenshot will be taken of the PCB and used in the lab report. Task four instructs students to add two resistors, a SW_FSM2JH button and one LED to their existing circuit from task two. After completion, students will take a screenshot of the schematic. Task five requires students to create a PCB from the schematic they created in task 4. Students must screen capture the PCB to use in lab report.

After completion of the lab students learned how to create a PIC24-base circuit, convert that circuit into a PCB and researched the limits of the LM2937-3.3 voltage regulator.

Introduction:

The objective of the lab is to familiarize the students with PCB Artist while having them create two schematics, two PCBs based on those schematics, and getting students familiar with a PIC24 processor as well as a LM2937-3.3 voltage regulator. Task one focuses on introducing students to the LM2937-3.3 voltage regulator and the PIC25 microprocessor. They are asked to research the output voltage and the maximum current of the regulator. In task two students are instructed to create a basic power circuit for the PIC24 system. This basic circuit uses the LM2937-3.3 regulator, five capacitors, the PIC24HJ128GP502 microprocessor, Vdd, and GND where needed. Task three is where students convert their schematic from task two into a PCB. They use the software to automatically route nets and components. Task four asks students to add onto the schematic from task three by using resistors, a pushbutton and a LED.

Procedure/Results:

Task 1:

a. Go to the Digikey website to find the voltage output and max current that the voltage regulator can supply

i. **Voltage Output:** 3.3v

Max Supply Output: 125mA

Figure 1. Above are the values listed on the Digikey website

Task 2:

a. Bring in the correct components into the work space

b. Match the layout of the lab instructions

c. Attach the components in such a way that resembles Figure 1 in the instructions

d. Apply correct values to components

i. C_1 ,C_3 Will have values of 10uF

ii. C_2,C_4,C_5 Have values of 0.1uF

iii. Source will have a voltage value of +5v

e. Screenshot schematic

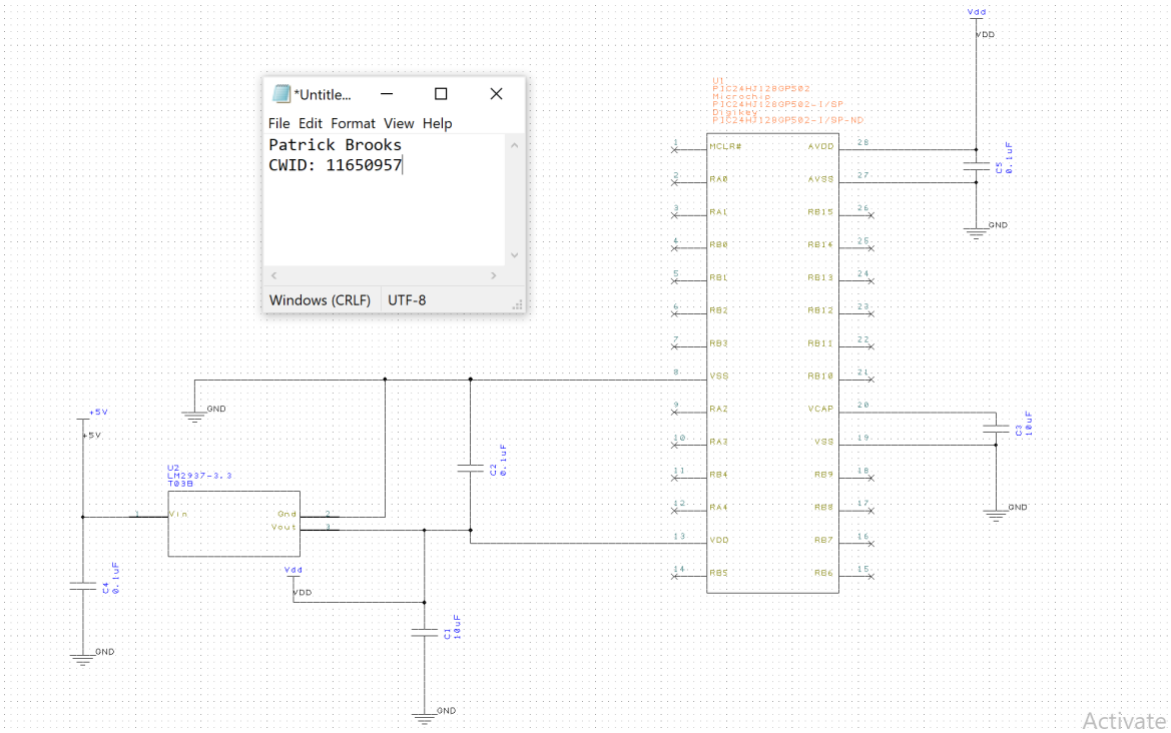

Figure 2. Schematic for Task 2

Task 3:

a. Click the “Schematic →PCB” option under tools

b. Follow the lab instructions when inside the PCB window

i. Change dimensions to 60mm x 60mm

ii. Change board part number to “ECE383-LAB2”

iii. Change the revision number to “001”

c. After creating the PCB Automatically route nets and components

d. Screenshot PCB

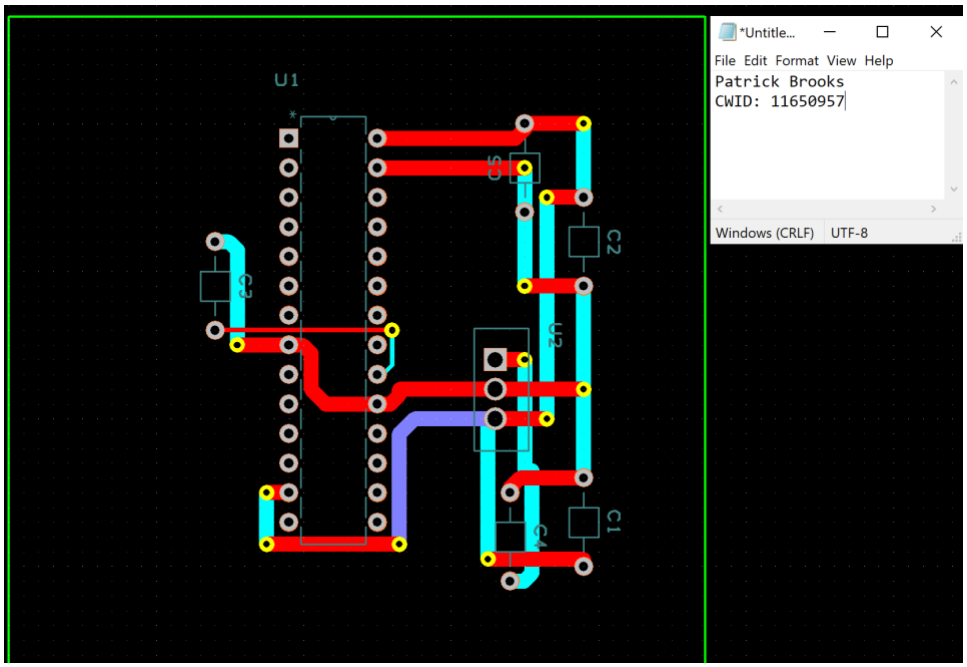

Figure 3. PCB created from task 2 schematic

Task 4:

a. Expand schematic from Task 2 by adding four new components

i. “SW1” switch

ii. “LED1” LED

iii. “RA” resistors labeled R1-R2

b. Arrange components in a way that resembles the schematic from the instructions

c. Use default values on the components

d. Screenshot schematic

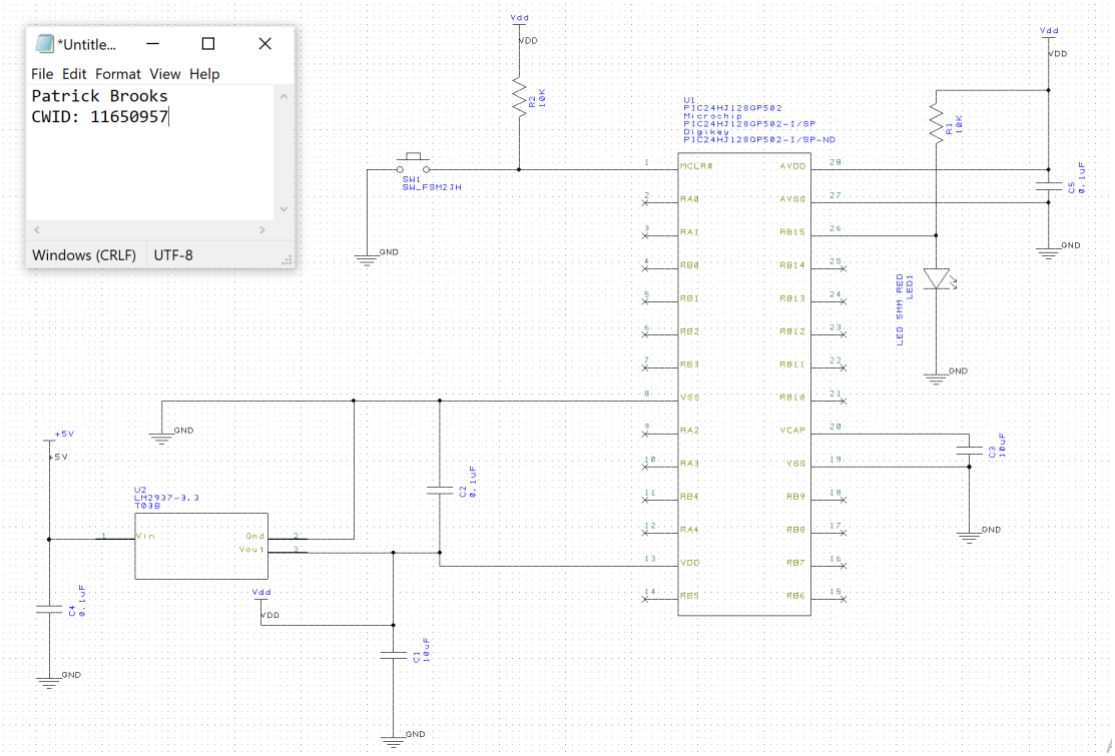

Figure 4. Expanded Schematic

Task 5:

a. Click the “Schematic →PCB” option under tools

b. Follow the lab instructions when inside the PCB window

i. Change dimensions to 60mm x 60mm

ii. Change board part number to “ECE383-LAB2”

iii. Change the revision number to “001”

c. After creating the PCB Automatically route nets and components

d. Screenshot PCB

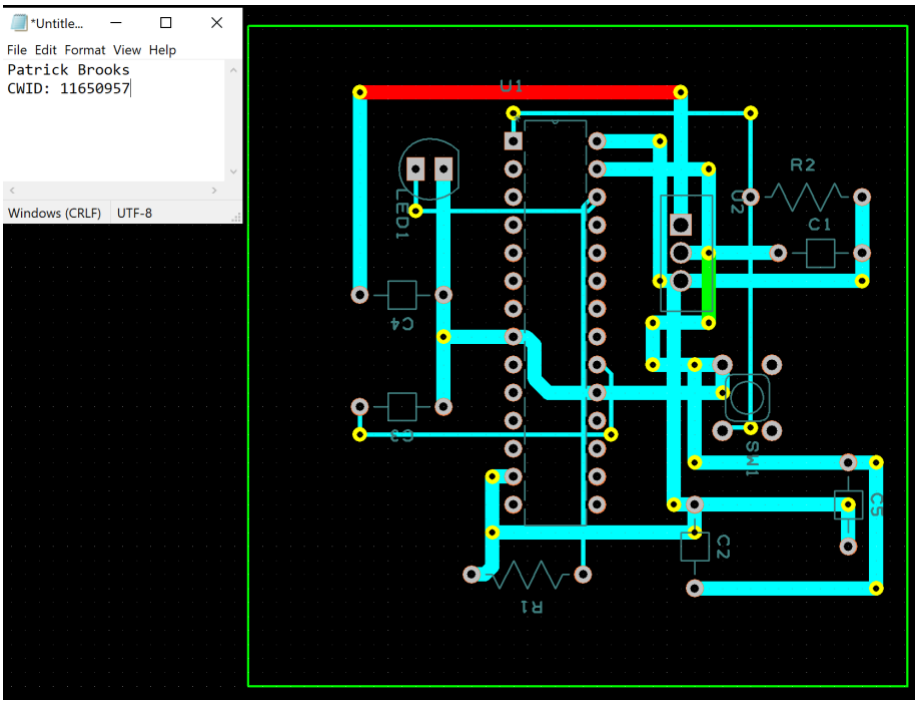

Figure 5. PCB created from task 2 schematic

Conclusion:

After completion of the lab students learned how to create a basic PIC24 circuit, convert a schematic to a PCB, expand the PIC24 circuit by adding components and increased their familiarity with PCB Artist. Students also deepened their understanding of the lab report format, which will be used throughout their academic endeavors.