KiCAD PCB Project

Abraham Lopez

Github Directory: https://github.com/ALcmos5780/class/tree/main/CPE301D

Main Directory: https://github.com/ALcmos5780/class.git

Goal:
The goal of the final project in CpE301D will be to design an Arduino Shield for your X-plained Mini Atmega328p/pb.

Deliverables:
Students are required to submit the schematic, PCB and Gerber files of this project. The entire project folder along with the libraries used and created should be uploaded to the github page under the folder CpE301D.

1. Components

A. Two LEDs with appropriate resistors connected to PORTD pins 5&6 in reverse logic.
Description: First part of the design was to create a schematic that had two SMD 0603 LEDs connected in reverse logic with 2 1k 0603 SMD resistors, to reduce the amount of current to the LEDs, to PORTD pins 5 and 6. Then layout their footprints in the PCB design.

B. Two push buttons working in internal pullup mode connected to PORTD pins 2&3.
Description: Second part of the design was to create a schematic that had two SMD tactile pushbutton switches connected in internal pull-up mode. Meaning PORTD pin 3 and 2 are on one side of the switches where a 5V source with 2 250 ohm 0603 SMD resistors are located, while on the other side of the switches is ground. Then laying out their footprints in the PCB design.

C. A 10K Potentiometer with a filter connected to PORTC pin 0.
Description: Third part of the design was to create a schematic that had a 10k through hole bourns potentiometer resistor connected to PORTC pin 0, where there is also a 100 nF SMD 0603 capacitor to help filter the voltage. The potentiometer is connected to 5V and ground. Then laying out their footprints in the PCB design.

D. Female header pins to interface to LM34/35 to PORTC pin 1.
Description: Fourth part of the design was to create a schematic that had a 3 pin female header that connects to the LM34/35 sensor. Two of the pins would serve as connections to 5V and ground while the third pin connected to PORTC pin 1. Then laying out the footprint in the PCB design.

E. Female header pins for SPI0 interface with VCC and GND to accommodate the ICM20948 sensor.
Description: Fifth part of the design was to create a schematic that had a 6 pin female header that connects to the SPI interface side of the ICM20948 sensor. The pin connections would include PORTB pins 2, 3, 4, 5, 5V and ground. Then laying out the footprint in the PCB design.

F. Female header pin for I2C interface with VCC and GND to accommodate the ICM20948 sensor.
Description: Sixth part of the design was to create a schematic that had a 6 pin female header that connects to the I2C interface side of the ICM20948 sensor. The pin connections would include PORTC pins 4, 5, 5V and ground. Then laying out their footprints in the PCB design.

G. Interface to DRV8833 DUAL MTR DRIVER CARRIER board and terminal block connectors to the side of the shield.
Description: Seventh part of the design was to create a schematic that had two 8 pin female headers that would connect to the DRV8833 board, while including connections for the two screw terminal block connectors. The pin connections would include pins PB1, PC2, PC3, PD1, 5V and ground. Then laying out their footprints in the PCB design.

H. Male headers to attach to X-plained Mini
Description: Eighth part of the design was to create a schematic that had two 8 pin male headers, one 6 pin male header, and a 10 pin male header. These male headers would be used to connect directly to the female headers of the X-plained Mini board. The headers were separated into Digital, Analog, and Power sections as these are easy to follow and are the same as the Arduino uno layout which the X-plained mini follows as well. Then laying out their footprints in the PCB design.

I. Female headers for additional connections
Description: Ninth part of the design was to create a schematic that had two 8 pin female headers, one 6 pin female header, and a 10 pin female header. These female headers would be used to connect additional wires to the shield as it will block the X-plained Mini, so if for example a logic analyzer needs to be used to measure the output of a pin, it can be done from the shield. The headers were separated into Digital, Analog, and Power sections as these are easy to follow and are the same as the Arduino uno layout which the X-plained mini follows as well. Then laying out their footprints in the PCB design.

I. SCHEMATICS

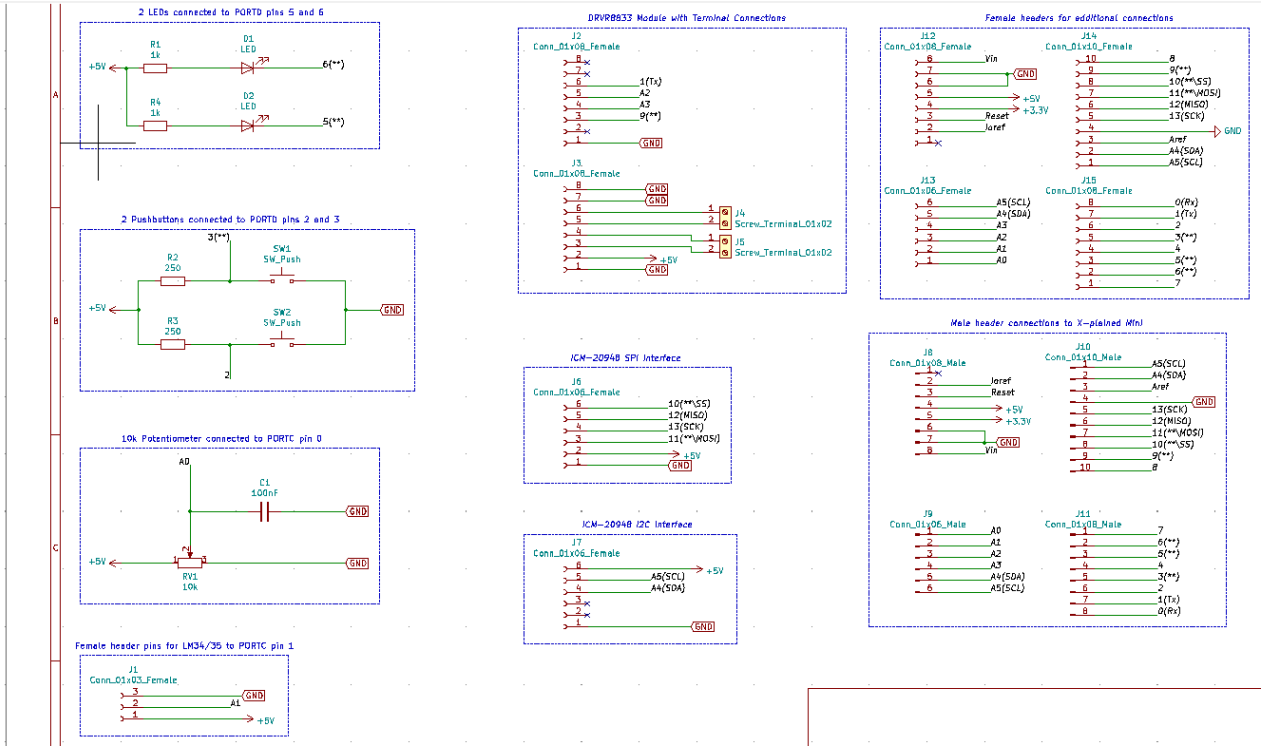

Figure 1: Schematic of Shield

II. Implementation

The Project was implemented with KiCAD software, specially with the schematic editor Eeschema and PCB layout software PCBnew. First the schematic sheet was made and placing down all the components with their proper connections to pins and nets. Then, assigned each component their respective footprint and then making to DRC the schematic to be sure that no errors were made. Following that, a netlist was generated and then switching to PCBnew and uploading that netlist to the software which generated all the footprints and the ratline connections. Afterwards, the footprints were moved to achieve the right dimensions and proper spacing for the components. Then traces were used to connect the footprints and then a placing an edge cut around footprints to make the shape of the shield. Then filling in the board with the ground plane using the fill-in area tool and then DRC the design to make sure there were no errors. Then, the Gerber files were exported for future use.

III. Screenshot of PCB layout

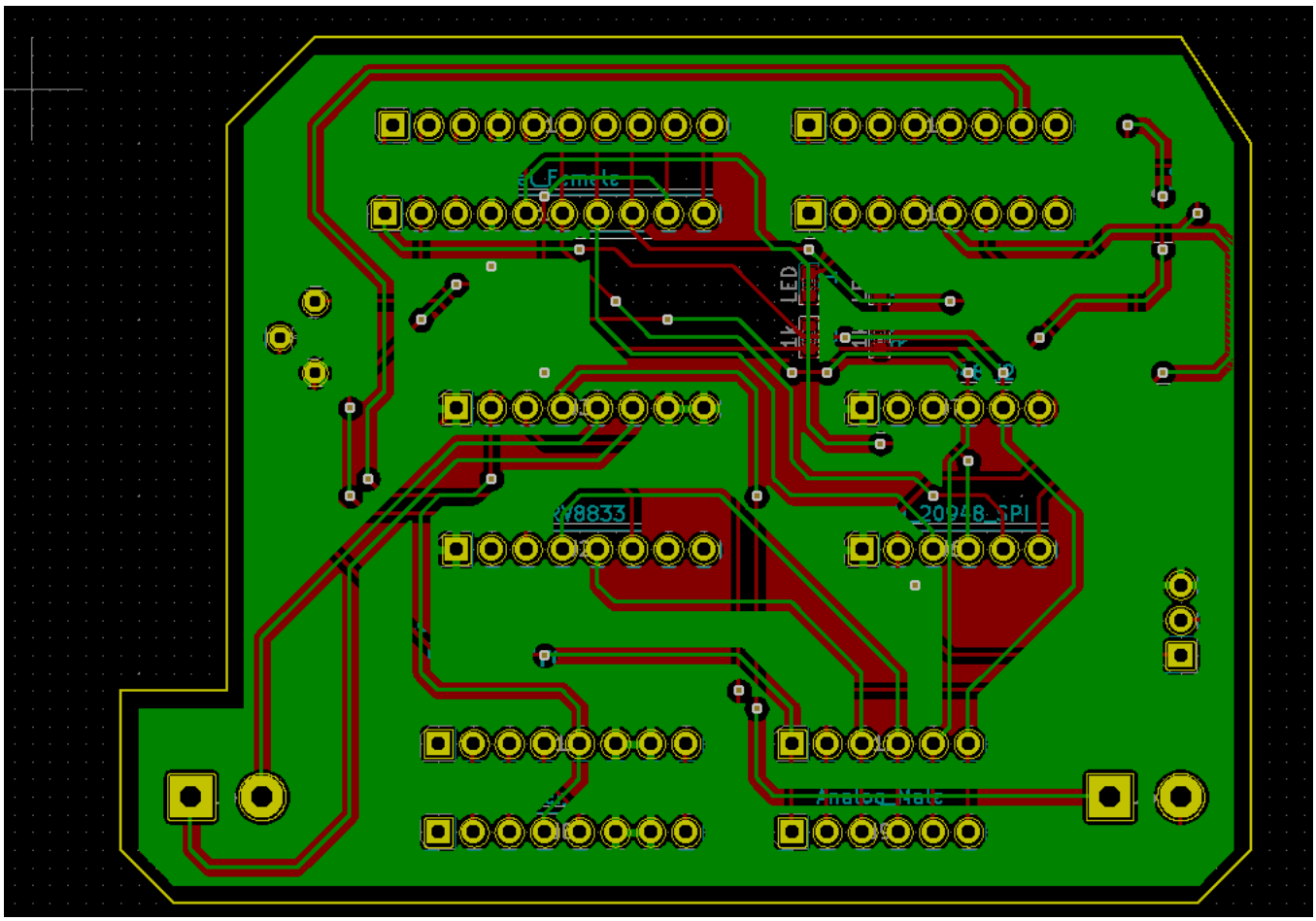

Figure 2 – PCB layout of Shield

IV. Screenshot of PCB 3D view

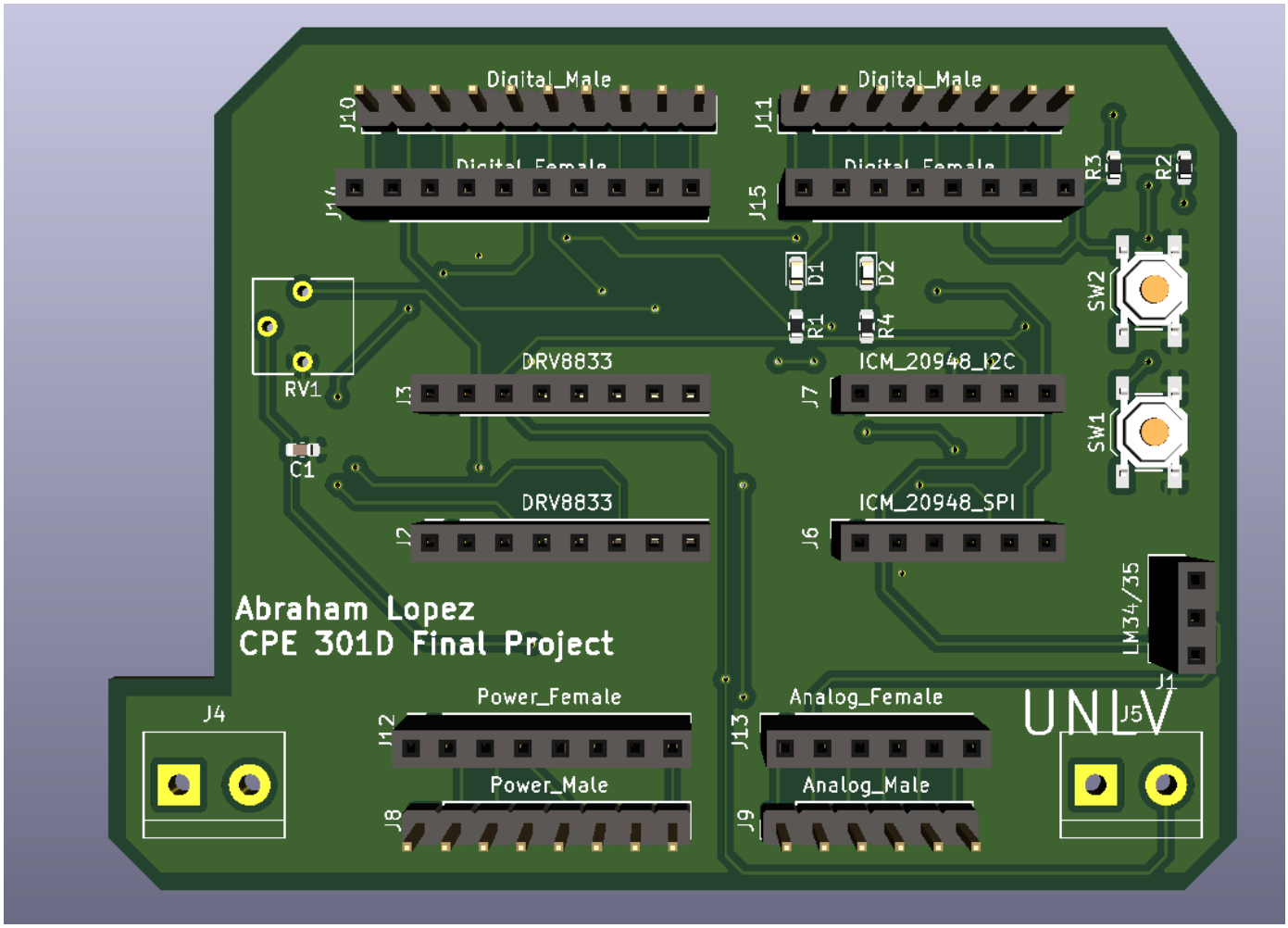

Figure 3 – 3D view of the shield

V. Conclusion

The PCB project was useful in learning about the basics of PCB design and layout. Resulting in enhanced knowledge and an actual working PCB that can be used for future classes and have practical applications in use with the X-plained Mini. This project also helps with the possibility of making a PCB for senior design.