Final PCB Fabrication

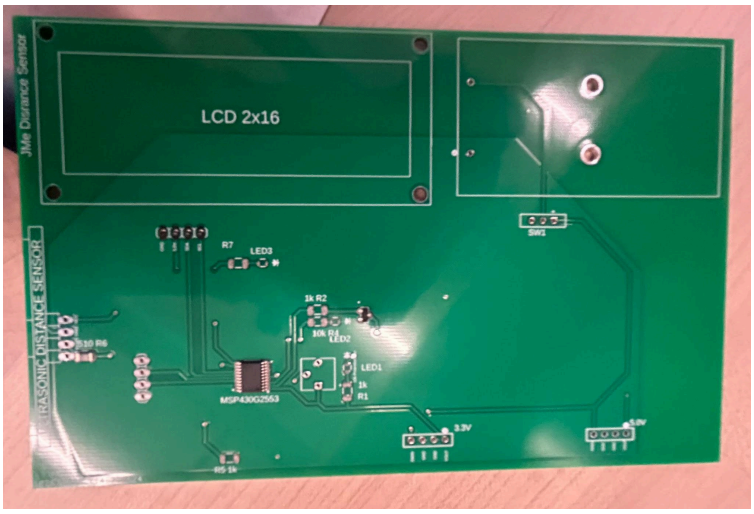
James Eyler

Design Process:

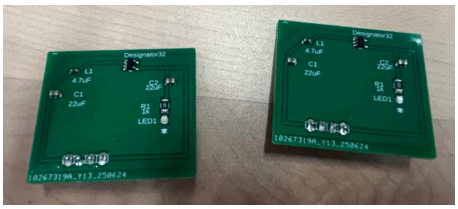
My first steps were applying solder paste to the Regulator Boards as well as the main PCB board. I used the stencil as well as masking tape and spare boards to achieve a smooth even application. The spare boards were positioned around the board that I was applying the solder paste to in order to add stability to the stencil so it did not bend when reaching the edges of the board.

Main PCB board : After Solder Paste is applied

After the solder paste applied, I then added the components by hand using forceps. I did this for the main PCB and one of the Regulator boards. For the other regulator board I used the Pick-N-Place machine that was available in the Junior Design Lab. This process was slightly tedious because the components would slide around or stick to the forceps when trying to place them on the board.

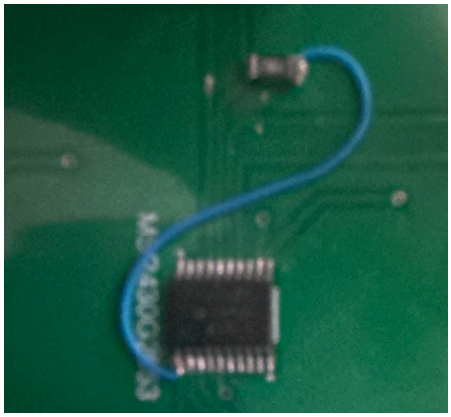


Main PCB Board : With components



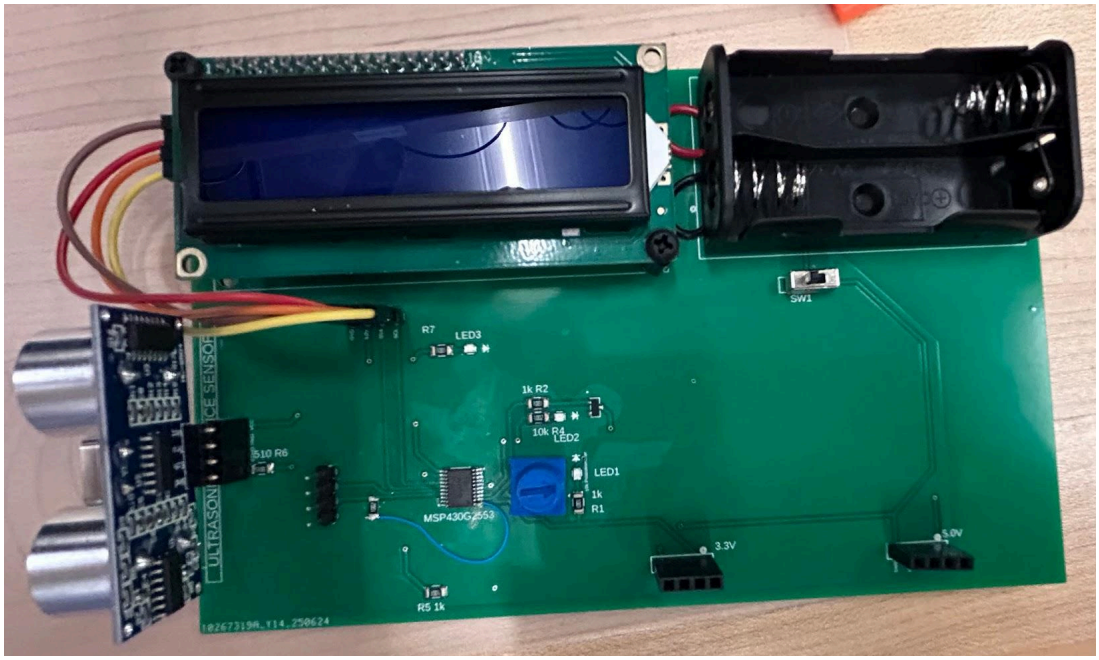
Regulator Boards : With Components

The board was then put through a reflow oven and the components were soldered to the boards. At this point I had noticed that I had missed a 10kΩ resistor that was supposed to be in the programmer pins. I was able to fix this with the help of a TA by using a smaller gauge external wire that he had soldered to the resistor and the board and then soldering the other pin of the resistor to the board via copper that he had exposed by scratching the board.



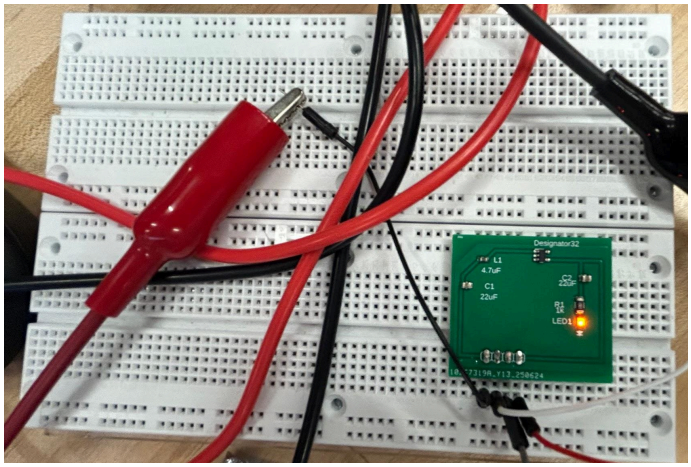
Main PCB Board : wire connecting from 3.3V to 10kΩ resistor to programming pins

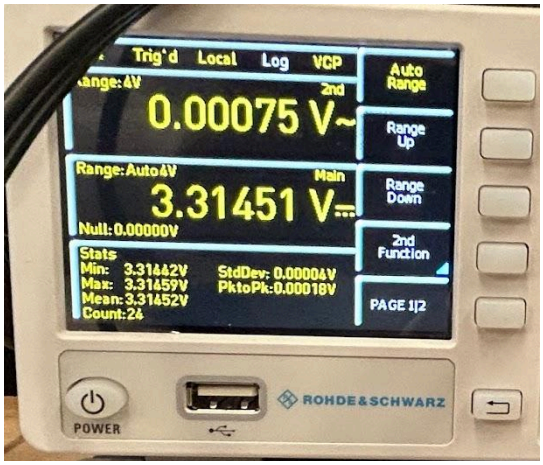
At this point I was able to begin the process of hand soldering the rest of the bigger components onto the board through the through holes. To do this I used the soldering iron provided in the lab as well as a breadboard in order to hold pins in place. This process was not very challenging because shortly before this I had been practicing soldering male header pins while my boards were in the reflow oven. I carefully added each component that needed to be soldered making sure not to have an excess of solder while maintaining a good connection between the components and the board.



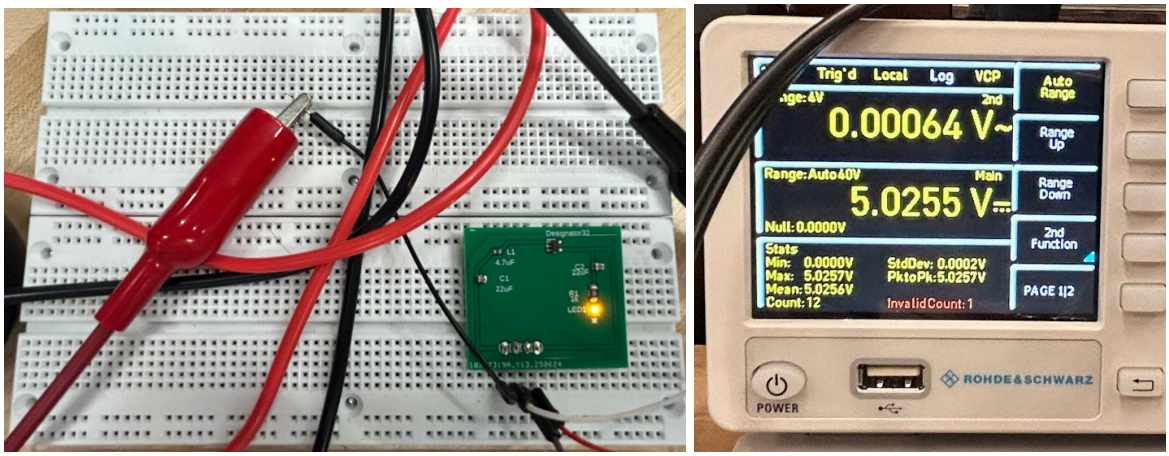
Main PCB Board : After all components are soldered to the board.

Now that I had all components soldered to the board, I needed to add the regulator boards. From looks, I could not tell which was which because they are identical in design and the regulator component itself is not branded. To find out which was which I used a power supply unit and a digital multimeter (DMM) to determine the output voltage of each board.



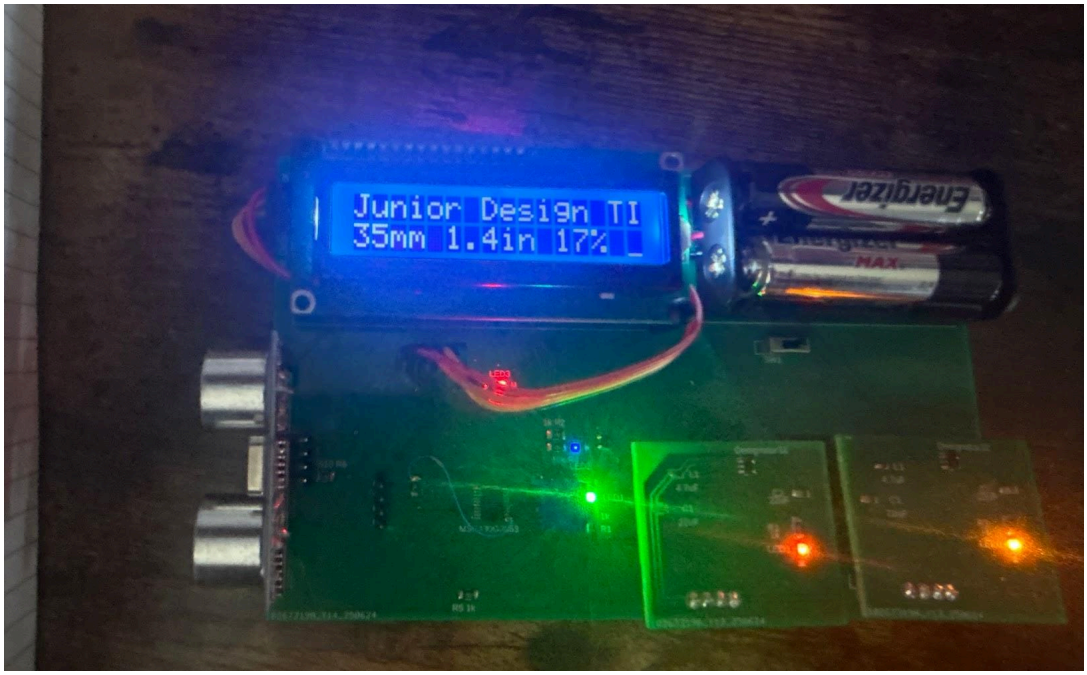


Regulator Board 3.3V : DMM showing 3.3V output



Regulator Board 5.0V : DMM showing 5.0V output

Now that all components are added to the board and the regulator boards are in place, I now need to code my board. I start by putting batteries into my board and turning the switch on. I then used an MSP430G2 board with the same chip as my main PCB to code my board. To do so I used the MSP430G2 board's built-in USB connection to plug the board into my laptop, then connected my Main board to the SWBTCK and SWBTDIO ports of the MSP430G2 board. Using this method I was able to successfully code my board.



Main PCB Board : Coded and functioning as expected

What I Learned:

In this part of the project I learned a lot about the fabrication process of PCB boards. Using both soldering methods, Surface and Throughhole, I was able to get a good idea of how to do both while maintaining a good connection. I also learned ways to fix design errors later in the process with things like missing components, like my 10k resistor that I had missed in my design process. I feel like making that mistake was very beneficial for me because initially I had thought that I would have to order a new board and delay finishing the project – I would not have known the method that the TA had shown me was an option. I also learned how to code the board when there is no USB connection or some way to connect it to a computer. In previous courses (embedded systems, digital systems, etc.) the boards had USB connections to the computers so you could code them directly. Obviously the board I had made did not so it was a good learning experience to see how it would be coded.