

Week 6:**Date:** 09/30/2021**Total hours:** 13**Description of Design Efforts:**

I spent many hours on the PCB design. Many hours on the PCB design. I also brought paddles from home because the ones that came with the table are not that good. Finishing up the PCB design, I want to clean up everything a tad bit before I go help with the computer vision stuff. The computer vision stuff was somewhat put on hold while we wait for orange balls and a long enough VGA cable to reach the ceiling. Most of my effort came in to finalize the PCB part placements before main lab, but I didn't do much work after as I had many assignments due Thursday and Friday.

- KICAD - Schematic

The schematic I had on my progress report last week was finalized. I showed it to Rohan, Joe, and Dr. Walters. There was not much I had to change. I had to make sure to send a decoupled and ferrite beaded signal to VDA. Both the capacitors and the ferrite bead act as an LC filter circuit. This cleans up the power signal to the analog pins on the microcontroller. I also had to add a 0 Ohm resistor that would be connecting both the two ground planes (one for analog and the other for digital). Besides that, nothing else major was added to the schematic. I did not know about this about KiCAD but for mounting holes, they need to be added in the SCHEMATIC VIEW and NOT the PCB EDITOR. For anyone that is struggling with adding mounting holes. Just add a mounting hole part and pick a footprint for the screw size you want to use on the schematic view.

Besides those additions, in the future (before design review), I want to add annotations and labels to the schematic. Label each section box a descript title. Potentially, if time permits, a document on specific design choices would be helpful. I need to also include the documentation I used to help design the UART to TTL circuit. It is the documentation for the FT232RL, but there is a specific portion I used to help with my decisions. And that was the section which specified how to share the power from USB with the FT232RL IC and the rest of the circuit. The original plan was to use the 3.3V out on the FT232RL but it does not supply enough current. So, we are going to use the 5V coming in through our USB-C connector to power the Serial to UART IC and a 5to3V regulator that will be supply power to the rest of the circuit.

Figure 1 attached on the next page shows the need for labels and the additions added after main lab. The 0 Ohm resistor in the bottom going from ground to ground is comical. The power to the analog power input of the microcontroller is debounced. There will be more adjustments that will be expected after design review. But we have something solid enough to start PCB traces.

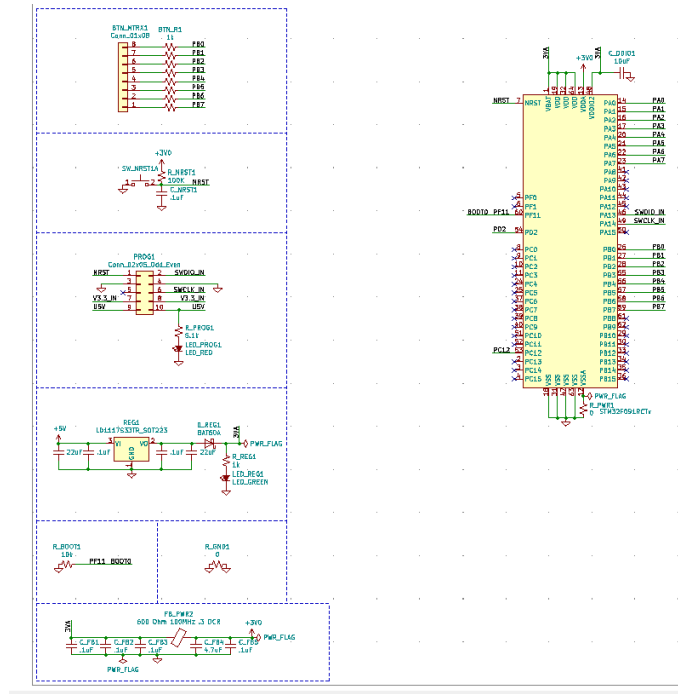


Fig. 1: Main changes made to schematic

- KICAD – PCB

Before main lab on Wednesday, I grinded out the PCB placements. I am familiar with placing parts on PCBs, so I did that the entire start of the week. Going with 8 microphones was annoying for this step. 4 microphones were not enough; 6 microphones left not connected pins on the amplifier; So, 8 microphones were needed. We may not end up needing 8 microphones. We know which side of the table the ball bounces on with just 2 microphones. It was slightly better with 4 microphones. So, who knows? We may end up with 8 microphones. The microphone circuit was placed farthest from the microcontroller because it needed an separate ground plane. The power for the PCB was put on the bottom right of the PCB which connects to the microcontroller in the center. Every other digital peripheral. From skimming the PCB, Rohan, Joe, and Dr. Walters seemed to be “ok” with it. So, by simple gloss over the PCB part placements are good enough to start the traces.

I know I was not supposed to use it but I did. I tried to do it by myself, but I ran into too many walls. I used an autorouter :(I used freeRouter. I had to install LayoutEditor and in their bins was a JRE executable for freeRouter. There wasn't anything wrong with the routing when I scanned everything manually. I removed all the ground traces and added the two separate ground planes. The routes are all correct and almost efficient. I will be trying to go over each trace individually and cleaning it up. There are a lot of errors that KiCAD is spitting out, but that's a question for Joe and Google for the upcoming week. Attached to this document is a screenshot of the image of the PCB design so far.

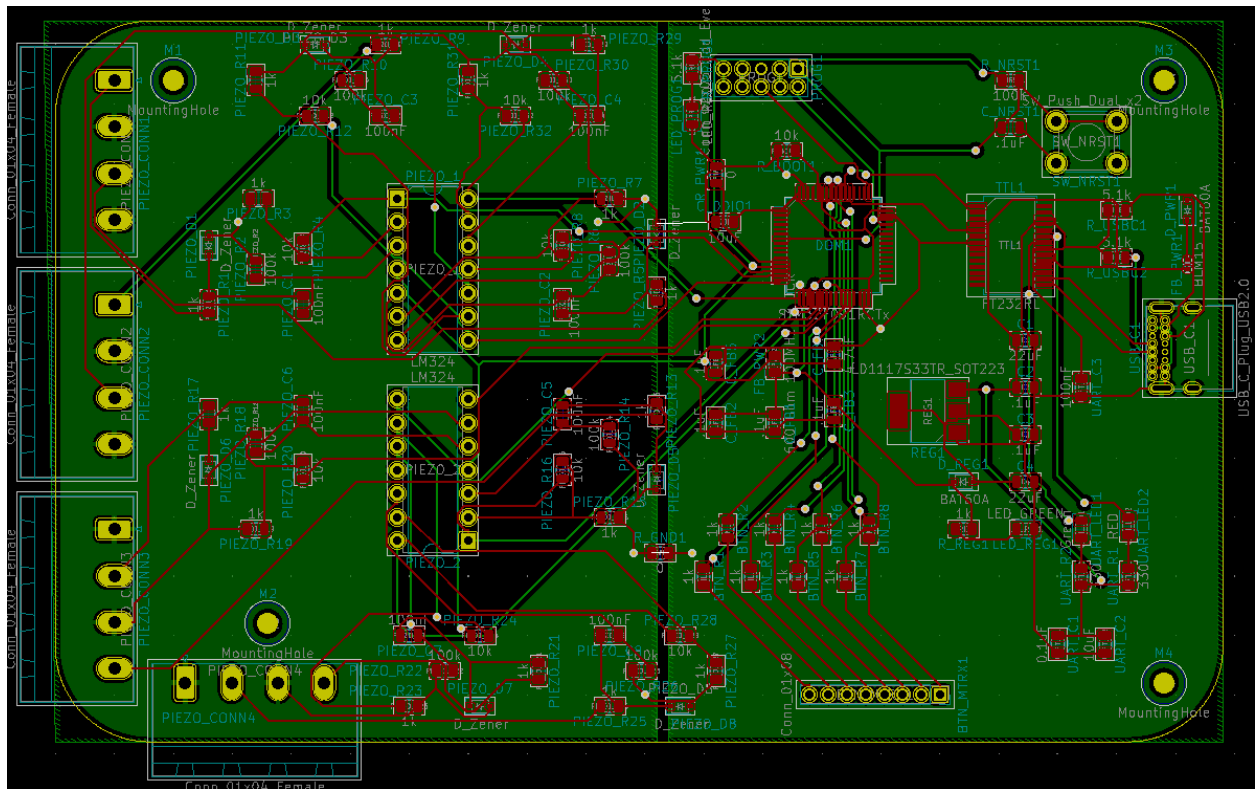


Fig. 2: The PCB with all its routing glory (left is analog, right is digital)

- Next week

I personally plan on finishing up the PCB. I do not believe it is ready for order, but it is close. Everything on the board is ordered so all we must do is order the PCB itself. I will be fixing the traces on the PCB and labeling the schematic to prepare for the design review. I plan on OpenCving the ball with the color sensor once I finish the PCB.