EE 256 Final Project Report: Calvin Sokk

SECTION 1: Pre-Schematic/Layout

Coming into EE256, I had no idea what I wanted to do for my project. While there were many example projects to choose from, I felt that I had to determine a project which felt unique to myself, while at the same time achievable for someone with no prior experience in PCB design. I eventually was inspired by the LED ornament idea and wanted to change it slightly, converging on creating an interactive color organ.

My project idea was to design a PCB that can take in audio input from a jack and light up RGB LEDs when different pitches are present. I found many example circuits online; it seemed like the color organ is an easy project! The next step was to start flushing out the modular design and the requirements of the system. Looking back, the very first step should have been to breadboard the circuit to verify its functionality before designing my entire project around that specific prototype. Until I understand the crux of my project, it is difficult to picture the entire high level system.

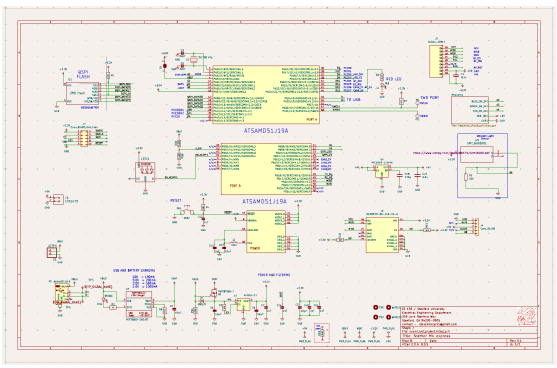
Link to Example/Reference Project:
https://www.youtube.com/watch?v=wJlglHS2w74&t=67s

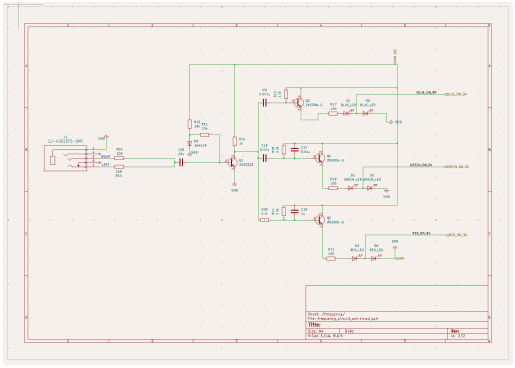
My requirements included functional and nonfunctional requirements to help distinguish between what is absolutely necessary for my project, and what can vary with implementation/hardware available. At this point, I was also realizing that I needed to figure out how I will interface with my sensors and microprocessor - what purpose will my communication module serve? I eventually decided that my project would not *just* be a color organ, but more of a general purpose lamp to satisfy the requirements of the project. I added a sound sensor and an LED sensor that interfaces with the microprocessor and a bluetooth module to communicate with the board and read from the microprocessor. I realized that as soon as I decided on these sensors, I should have ordered and breadboarded them to verify their functionality. Leaving this to last minute testing on the fully integrated system was a huge mistake that could lead to difficulties.

SECTION 2: Schematic/Layout

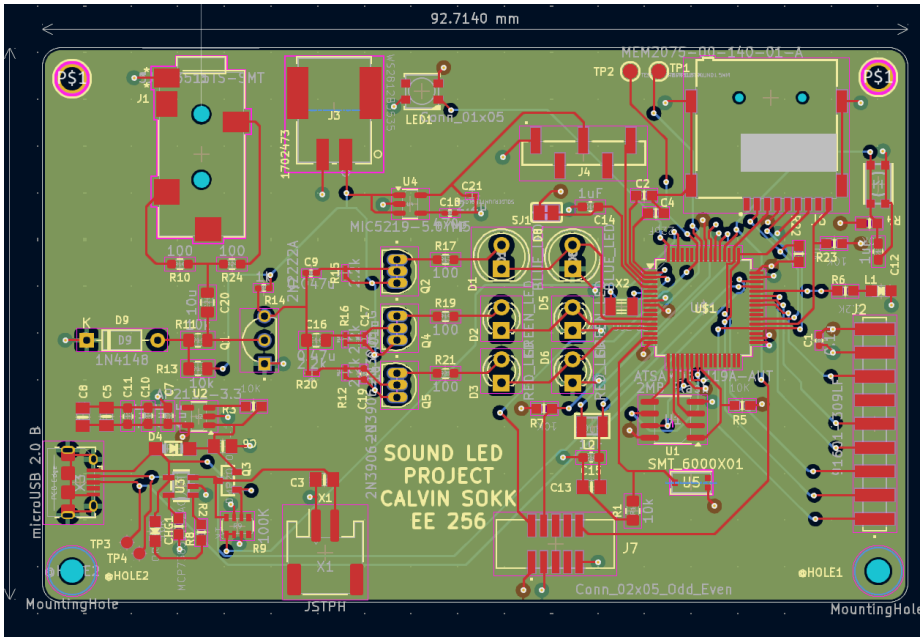
At this point, it was time to create the schematic and layout the design. The schematic was the most stressful part of this project for me. I had never had to design my own circuits from scratch! The core of EE usually entails analyzing a pre-existing circuit, which is much easier than making design choices yourself. I was also battling with new tools (KiCad), which I had minimal experience in from the previous labs. I felt like every single connection I made was wrong, and that I did not understand anything about the existing schematic for the featherboard. I also felt reluctant to start the project because I knew that any mistakes I would make would propagate to difficulties when the board is actually created. I realize now that starting the schematic as soon as

possible would have been the best plan of action, as I can make many drafts of schematics given that I have more time. Waiting until the last second after thinking about it a lot is worth less than creating a bad draft and getting constructive feedback from experts. Of particular difficulty was interfacing between all of the different modules and maintaining an organized flow to remembering which wires go where.





After creating the schematic, it was time for layout and preparation for order. While this was a lot of work, it was not as stressful or difficult as the previous part. I think the biggest contributor to my aversion to schematics was not understanding how to go about solving the problem/designing the circuit, but laying out the board on the other hand is a closed form problem to solve that just took some time. Of course, there was still the anxiety of wiring something incorrectly, and this did happen in several places on my fabbed PCB (I mixed up several pins on my connector to my sound module and I messed up the connections to my phototransistor.



SECTION 3: Post-Project Reflection

My biggest takeaway from this project is that engineers will never be on time. Because engineers will never be on time, if you do not set a deadline for yourself, you will be even later than late! Address aversion from a task immediately and get started even if it seems difficult or impossible - it will only become possible once you begin. This was the biggest challenge I faced in this project, and I am sure that I would have been able to do so much more had I dedicated pro-active time towards starting early.

On the more technical side, I now feel that given an arbitrary task, I could with enough time create a PCB that accomplishes said task. After understanding how to research and design a circuit on my own, use KiCad to organize my thoughts, design the schematic/PCB layout, and generate/send my order out, I can certainly replicate this process with more ease next time I need a PCB. I've learned not only how to use these new tools, but how best to go about solving problems when I have absolutely no idea how to start.

I also have become more familiar with the creativity required to solve actual hardware problems. When assembling my board, I had to battle ordering MICROSCOPIC resistors and capacitors, incorrectly wired connectors, missing/losing parts, and determining the source of user error with solder-bridges and component orientation mistakes. Solving all of these problems and having results to show is extremely helpful and rewarding. I hope to make another PCB sometime soon!