

I made some big changes to my keyboard design. I thought about going with an 84 key keyboard but I decided to go with a 60% keyboard for a few reasons

1. makes it easier to make the case for it in the future. 2. 60% is a more common keyboard form so things like plates and cases are easier to find
2. Also to have more room for my mouse on my desk.

I still need to work on the rgb of my circuit. This part has also been held up by programming considerations. I ended up asking about this to some keyboard makers and now I am a lot more confident in being able to use qmk to configure the keyboard I make.

So next term I think I can finish the PCB. I might even have it finished by spring break. Then get on with things like assembling the pcb, making the case, and buying or designing a keyboard plate.

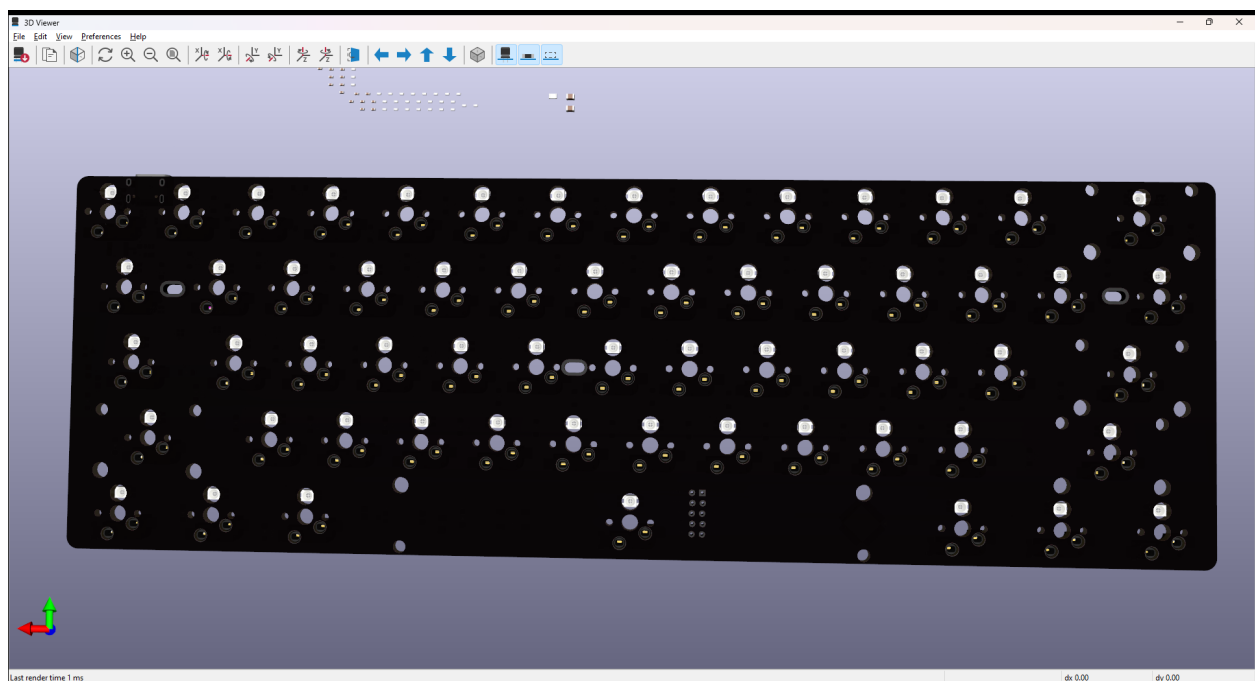


Figure 1: Front side of PCB 3D Viewer

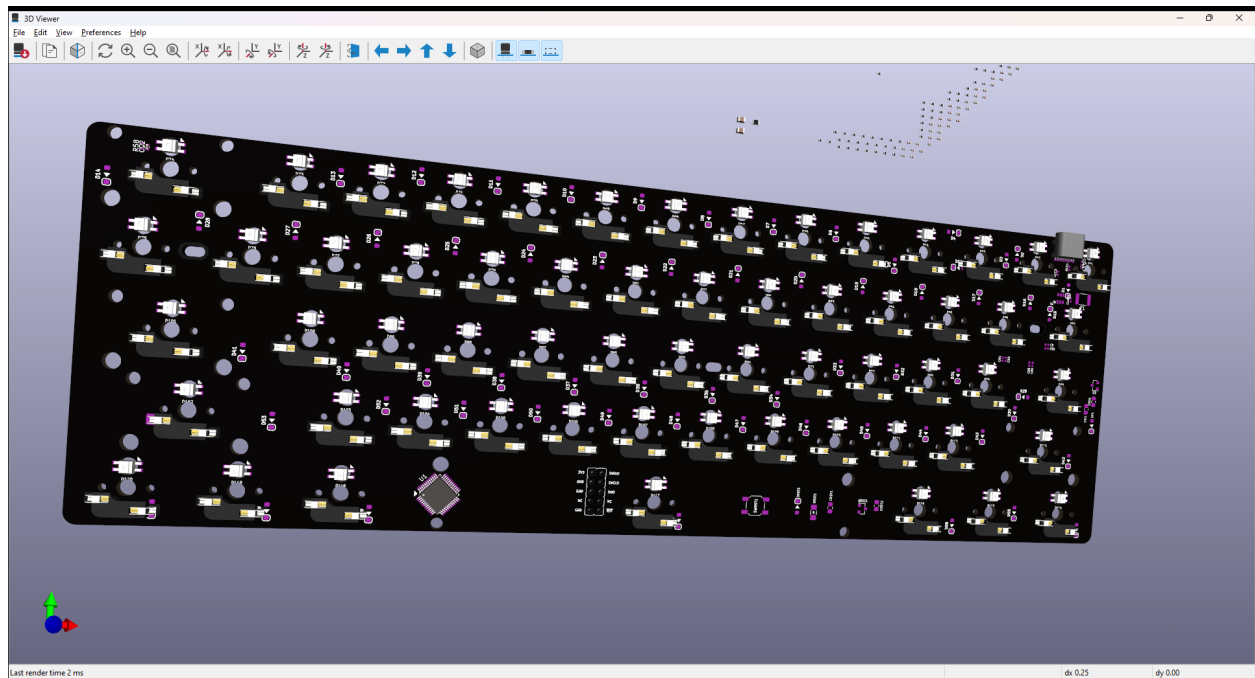
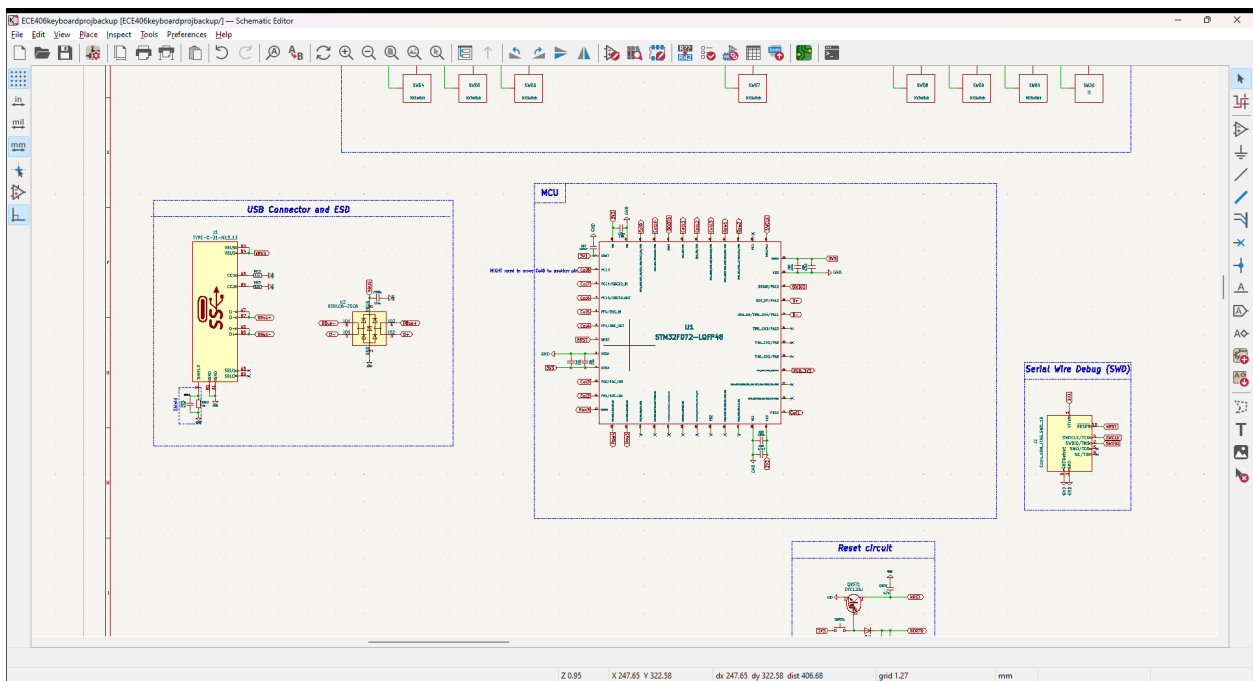
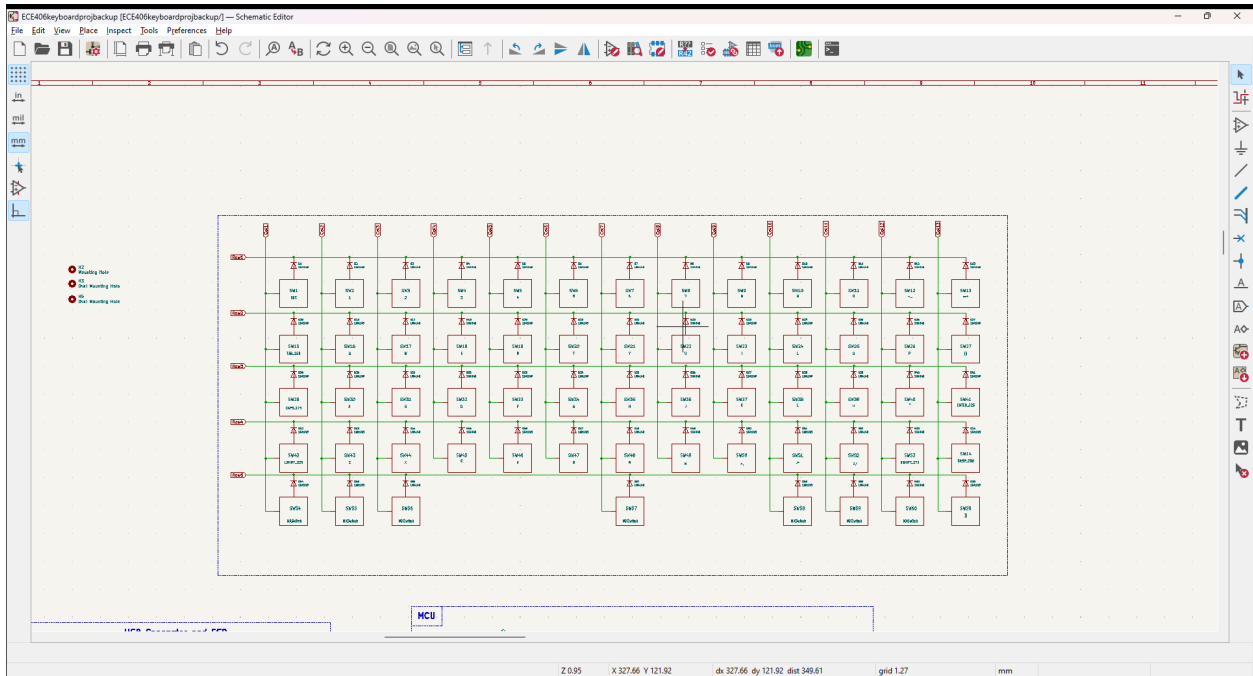
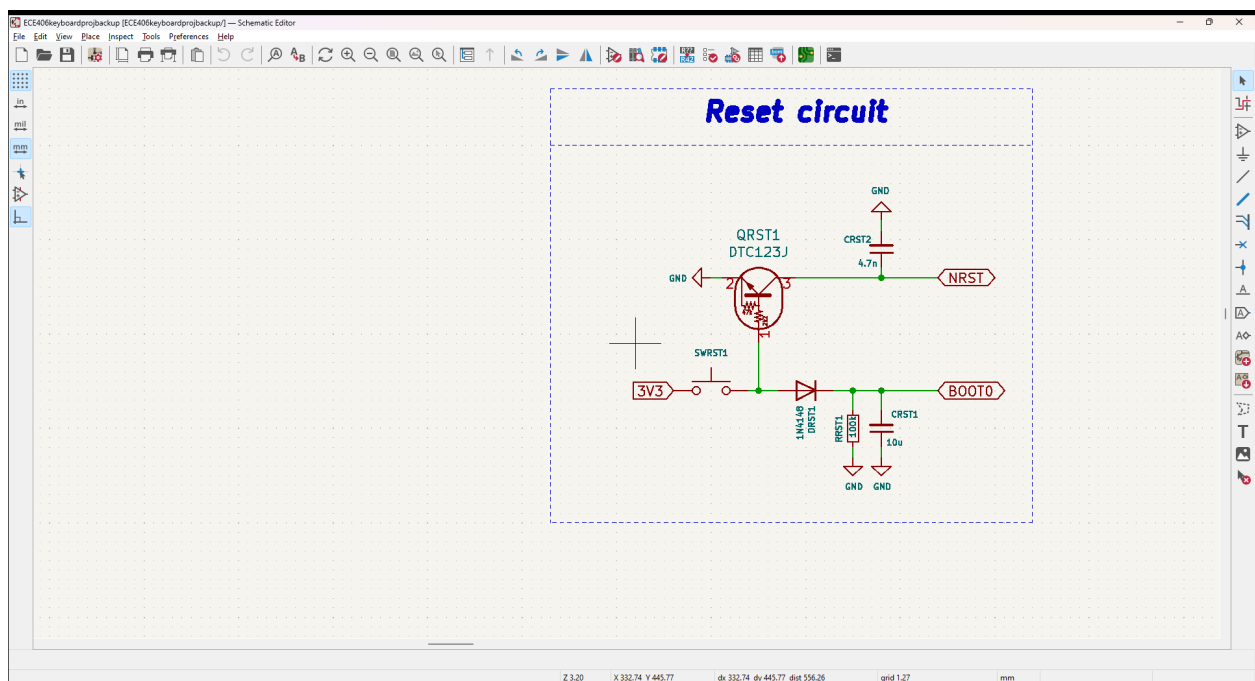
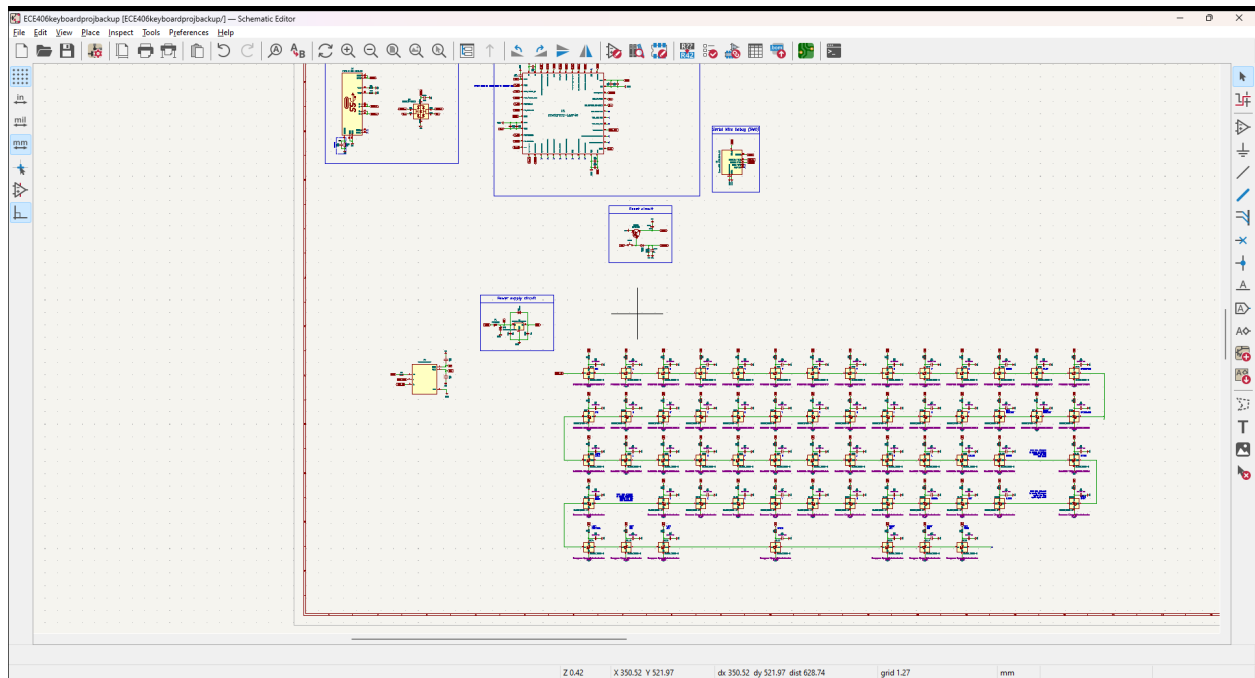
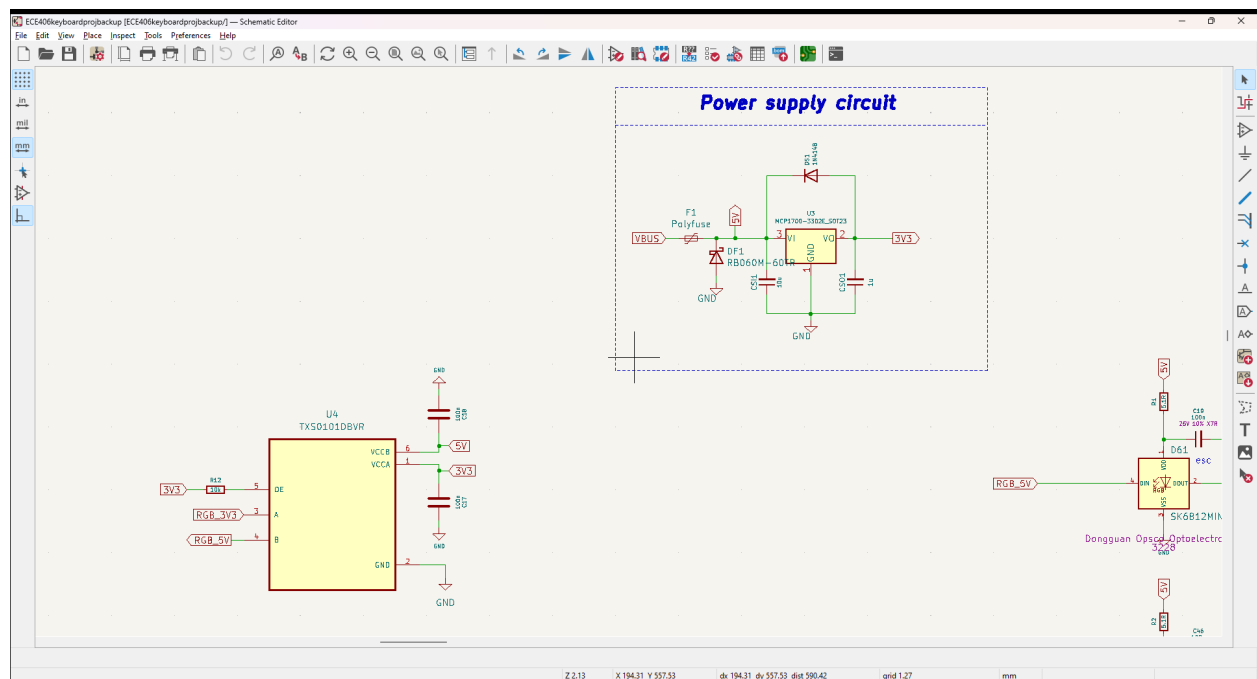


Figure 2: Backside of PCB 3D viewer









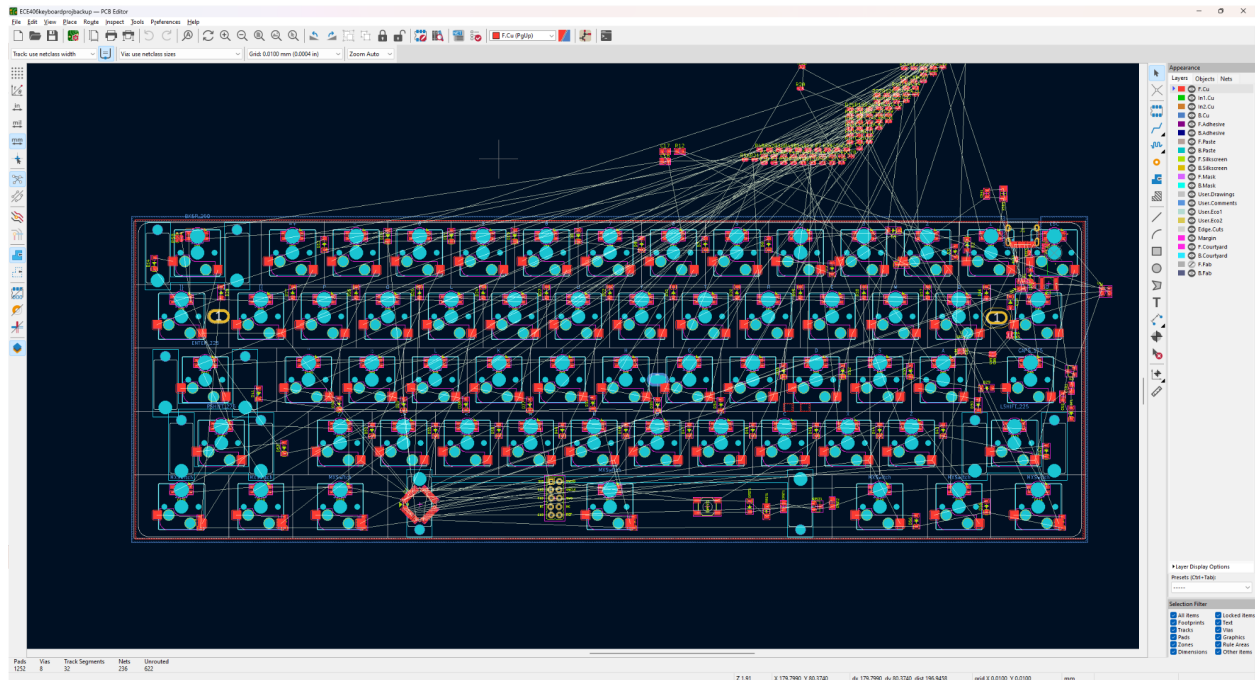


Figure 4: Keyboard pcb

There were several very helpful tools that helped me get this far. This includes open source keyboard examples from Acheron, keyboard layout editor (<http://www.keyboard-layout-editor.com/#/>), plate and case builder (<http://builder.swillkb.com/>), and keyboard firmware builder (<https://builder.mrkeeb.com/>).

As you can see there is plenty of work to do when it comes to routing. Getting a PCB done within 40 hours of work was very optimistic. I am confident that I put at least 80 hours of work into it this term alone. Not all of it was PCB work but a lot of it was. There was a lot of time and consideration given to the schematic first and foremost but there were other things like contemplating the keyboard format, learning QMK to program the keyboard, and the case I plan to make. The original keyboard I was working on had a lot more keys but I decided to go with a 60% keyboard to make the project more feasible and I also needed a smaller keyboard than what I have now. I also considered designing a laser-cut brass plate but that is pretty expensive so I decided to just make sure the PCB is compatible with a typical 60% brass plate. I think you get the idea that a lot of consideration and planning has gone into this.