

Week 5: PCB design template

Q1: Github

<https://github.com/Carciax/EEE3088F-Project-CKR>

Q2: Power subsystem failure management

i) Going to put populate zero ohm resistance in series with each modular component and unpopulated zero ohm resistor over each modular component allowing me to bypass each modular component by desoldering the populated 0 ohm resistor and resoldering the unpopulated resistor. I'm also going to put lots of test points which are going to be through holes which give me the ability to connect wires later on. I will also put jumpers next to the battery and at the 3V3 out and ground allowing us to connect an external 3.3 volts either in place of the battery or just to the main circuit.

ii) I will ensure to use the correct size track to the appropriate current rating for each track to protect against track damage.

I will also make the ground track thick as well as making ground its own level on the board.

iii) I will provide other options to replace components or go and buy the components from other shops and solder them on. I will also ensure to use components with large quantities of stock decreasing the chance of the component running out.

iv) I will make the circuit as modular as possible allowing for changes to the circuit as well as providing lots of available connection points to make modifications post PCB development.

The plans for failure management for this subsystem were unchanged; the update schematic below implements these plans.

Q3: Sensing subsystem failure management

i)

Each I/O trace will be connected to two separate pins on the microcontroller. Each trace will have a 0Ω resistor between the sensor pin and the microcontroller pin, but one of these 0Ω resistors will be DNP. This will allow a different microcontroller pin to be used in the event that one fails. These connections are shown on the microcontroller schematic.

This was implemented as described.

ii)

The I main power line will also be populated with a 0Ω , to allow the entire module to be disconnected from power in the instance that a design error and/or circuit damage within the sensor module leads to a short.

Since the previous schematic revision, PMOSFET switches were added to allow each sensor's power supply to be individually controlled. Populated 0 ohm resistors were placed at the power supply of each PMOSFET, and DNP 0 ohm resistors that bypass the PMOSFETS and power the sensors directly were added, to account for errors or failure in the switching circuits.

iii)

If the 0Ω resistors have to be de-soldered, their pads can be used to solder a wire directly to another pin on the microcontroller.

Through-hole test points were added to the schematic; these enable easy testing of the circuit and are easier to manually solder wires to than 0 ohm resistor pads.

iv)

Finally, the design includes 2 analogue sensors (a potentiometer which can be connected to some small mechanical device, and a thermistor), and 2 digital sensors (an ambient light sensor and a temperature/humidity sensor). Best case scenario, this means that the HAT is multi-faceted and can serve a variety of different research purposes. However, in the event of component/design failure or inadequate component stock, the HAT will hopefully still have other functioning sensors and thus will still be useful. It is specifically useful that there are 2 different temperature sensors, given that our original examination of use cases for the HAT placed on a strong emphasis on the HAT's ability to sense temperature.

This was able to be implemented within budget and thus was kept. The PMOSFET power switches described previously minimise the additional power draw brought about by having additional sensors.

Q3: Microcontroller interfacing failure management

For individual component failure, components will be connected into the circuit by some means that it can be disconnected. Components that are likely to fail will get jumpers which can be easily remove and another component can be connected from an external board.

Components that are less likely to fail will get two pads soldered together in line with the component. These can easily be soldered and de-soldered, and the signal can be measured in that line. Lastly, components that are likely never going to fail will get 0Ω resistors which can be taken out and/or rerouted to other lines.

It was decided to consistently use 0 ohm resistors instead of jumpers or solder pads. The 0 ohm resistors were chosen with a large physical size such to make manual soldering and de-soldering easier.

Power and controls:

The power and ground of the microcontroller has solder pads in place to ensure that the microcontroller can function without the power and sensors. The debugger input is already a pin header and thus can be plugged – which also means that the NSRT reset pin has a pin header that can be easily accessed to reset the microcontroller. There are enough redundant push buttons that failure management is unnecessary. The BOOT0 or boot mode pin has a solder pad in case the board needs to be booted in a different mode.

Besides the decision described above to use 0 ohm resistors instead of jumpers or solder pads, this was implemented as described.

Sensors and EEPROM:

The digital sensors inputs and the EEPROM communication lines have 0Ω resistor pairs in place such that they can change between I2C1 and I2C2 communication protocol, and so that there are a pair of lines available for isolated testing of components. The analogue sensors have 0Ω resistor pairs such that they can change between GPIOA and GPIOB reading the value.

This was implemented as described.

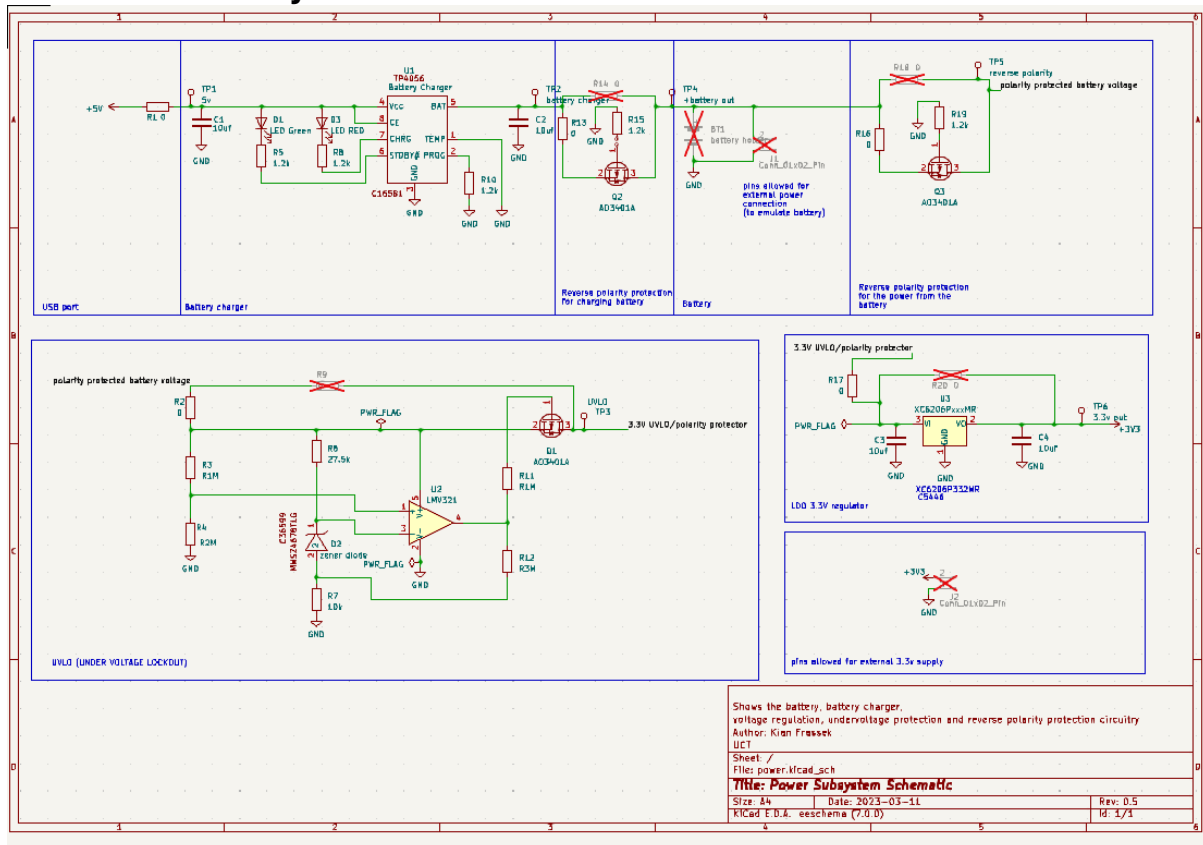
Output:

The output USB has a 3 pin header on both Tx and Rx lines such that the USB can be changed between UART1 and UART2 as well as the board can communicate with an external USB setup in case the onboard USB port or differential pair lines to UART chip are damaged. The LED has also been wired with a jumper such that an output signal can be jumped to an external board.

USB/UART connections were implemented as described. The LED connection was changed from a jumper to a 0 ohm resistor. A through-hole test point was added which will allow a wire to be manually soldered if the connection needs to be jumped to another board. A second LED was also added in case of LED failure.

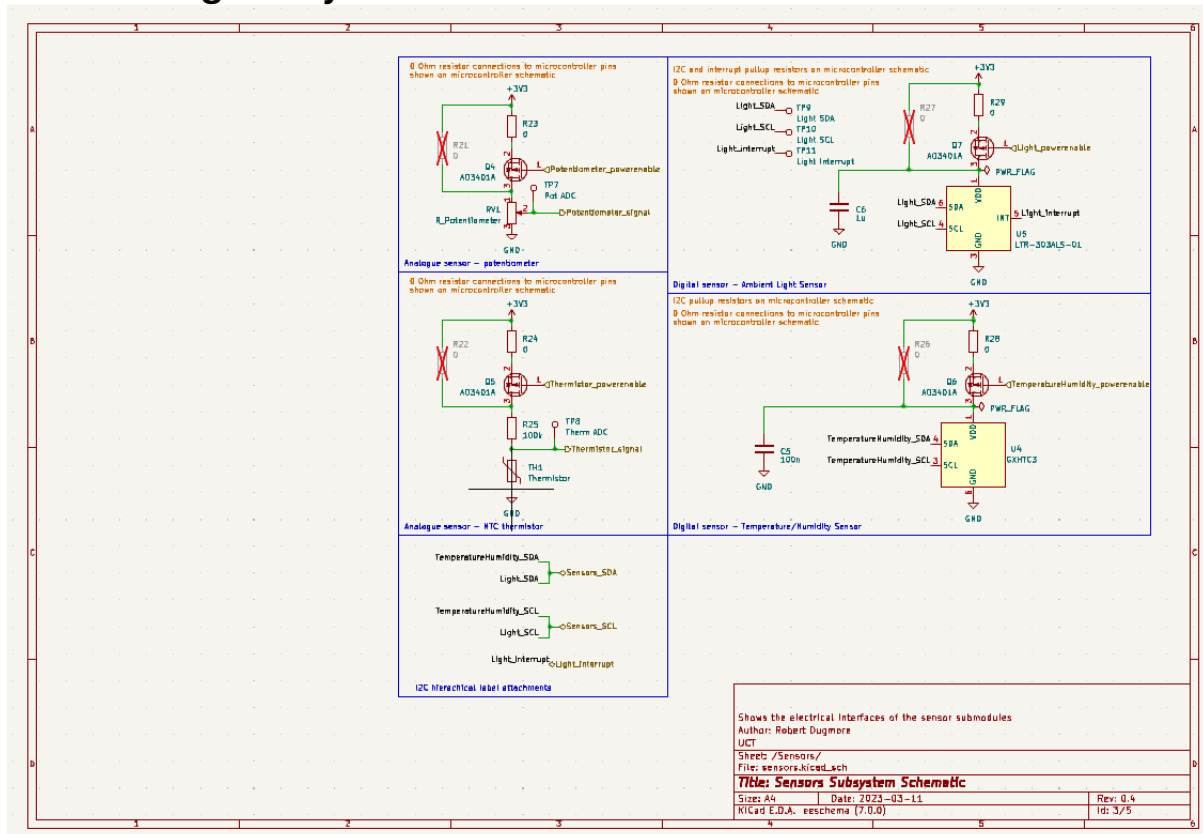
This system of failure management allows for most components to be isolated for testing, redirected in case of line damage, or have signals inputted or outputted to an external board for testing.

Q5: Power subsystem schematic



Some changes were made to resistance values as they were wrong. Test points were added throughout the circuit which will provide opportunities to test as well as solder on wires if needed. I have also added unpopulated and populated 0 ohm resistors over each modular component so that we can remove and add modular components as we want.

Q6: Sensing subsystem schematic



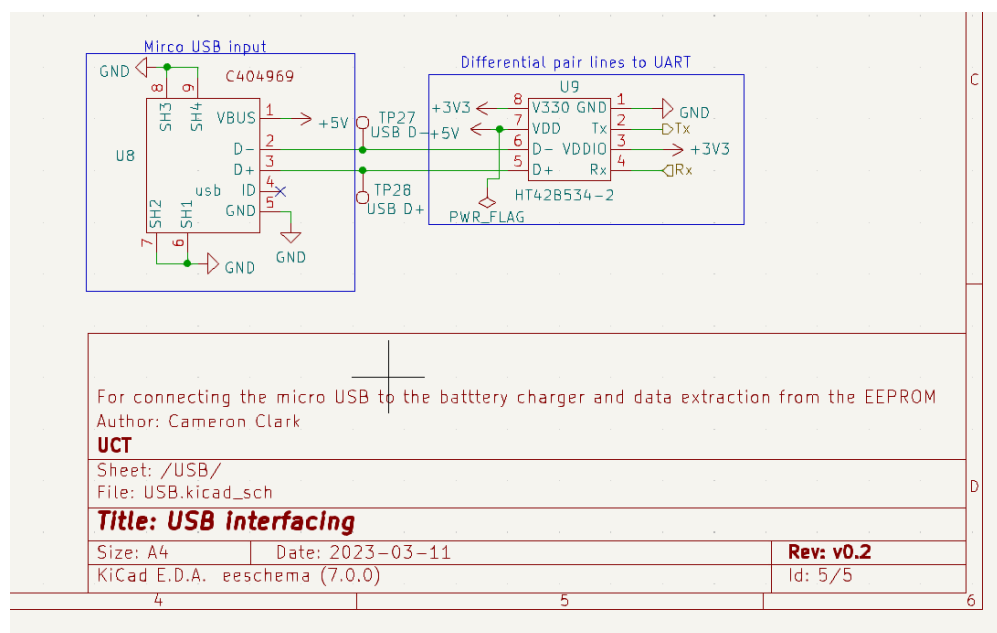
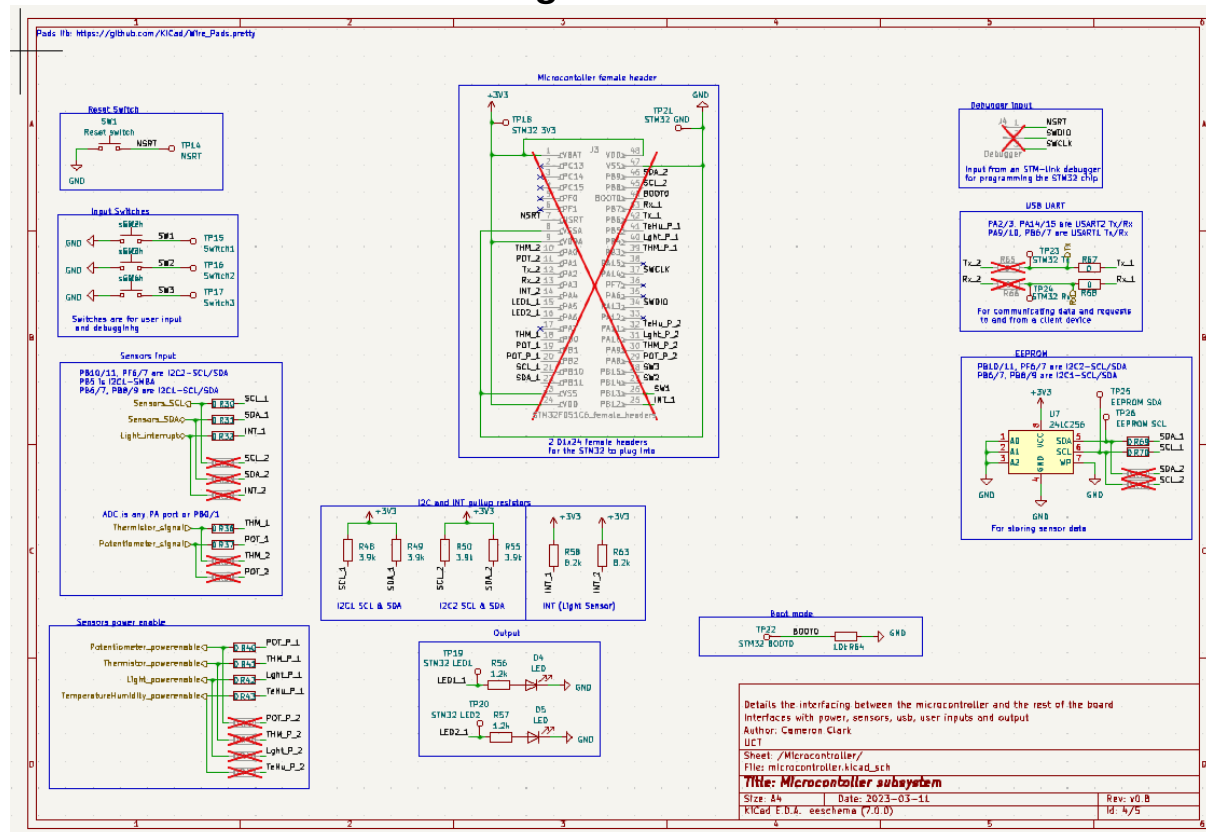
PMOSFET switching circuits were added to each sensor to allow the power to be enabled or disabled individually.

Test points were added to allow testing of the circuit or allow wires to be manually soldered between various components if necessary.

DNP 0 ohm resistors bypassing the PMOSFET switches were added to allow direct powering of sensors if necessary.

Net labels were renamed for clarity.

Q7: Microcontroller interfacing schematic



Connections to the PMOSFET switches on the sensors subsystem were added to allow the sensor power to be enabled/disabled via GPIO.

The failure management system was updated to use exclusively 0 ohm resistors instead of a mixture of these and jumpers/solder pads.

Test points were added to allow testing of the circuit or allow wires to be manually soldered between various components if necessary.

EEPROM was connected to I2C1 by default (instead of I2C2), and I2C pullup resistances were recalculated and changed.

Previously a component diagram of the actual microcontroller board had been included for visual reference – this was removed entirely.

A second output LED was added.

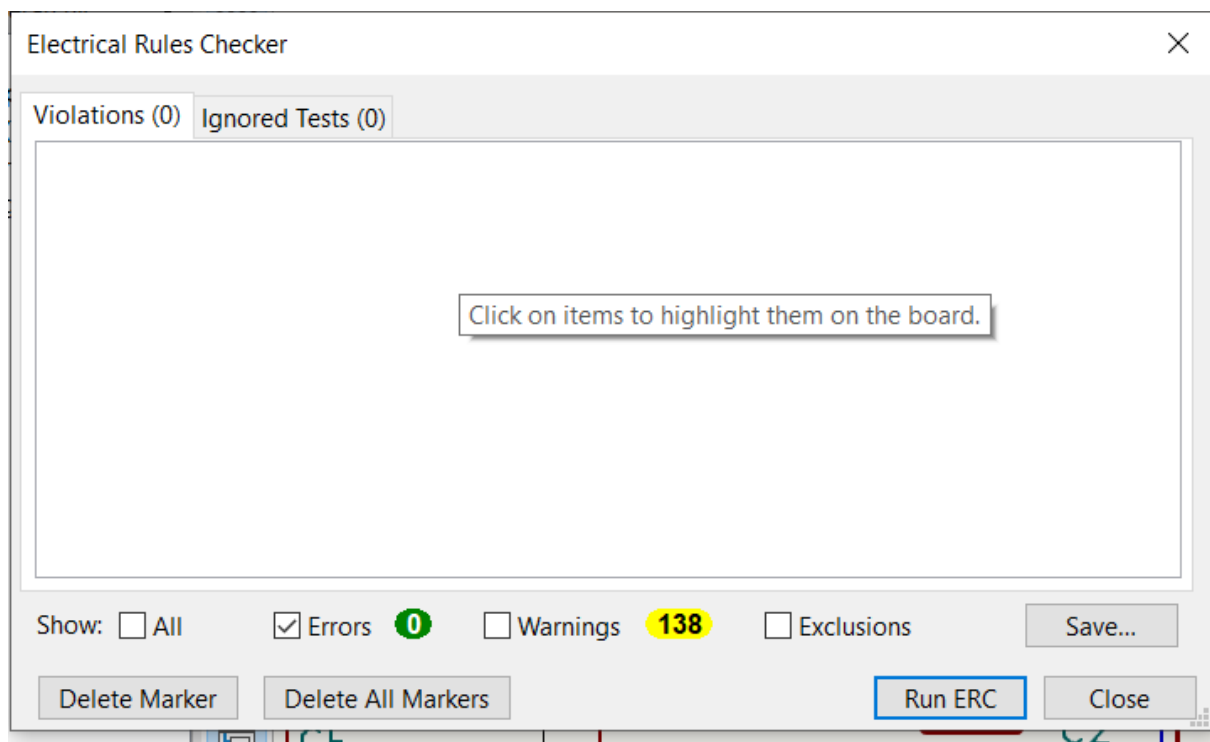
Net labels were renamed for clarity.

Q8: Updated ERCs

ERCs used were:

- Make sure the 5V and 3V3 and GND labels aren't short circuit to each other.
- Make sure there are no floating pins.
- Make sure the power running through resistor doesn't exceed their maximums.
- Make sure power lines do not go through traces that are too thin.

Screenshot of ERC report with no errors:



The 138 warnings are because a large number of wires in the schematic are off-grid.

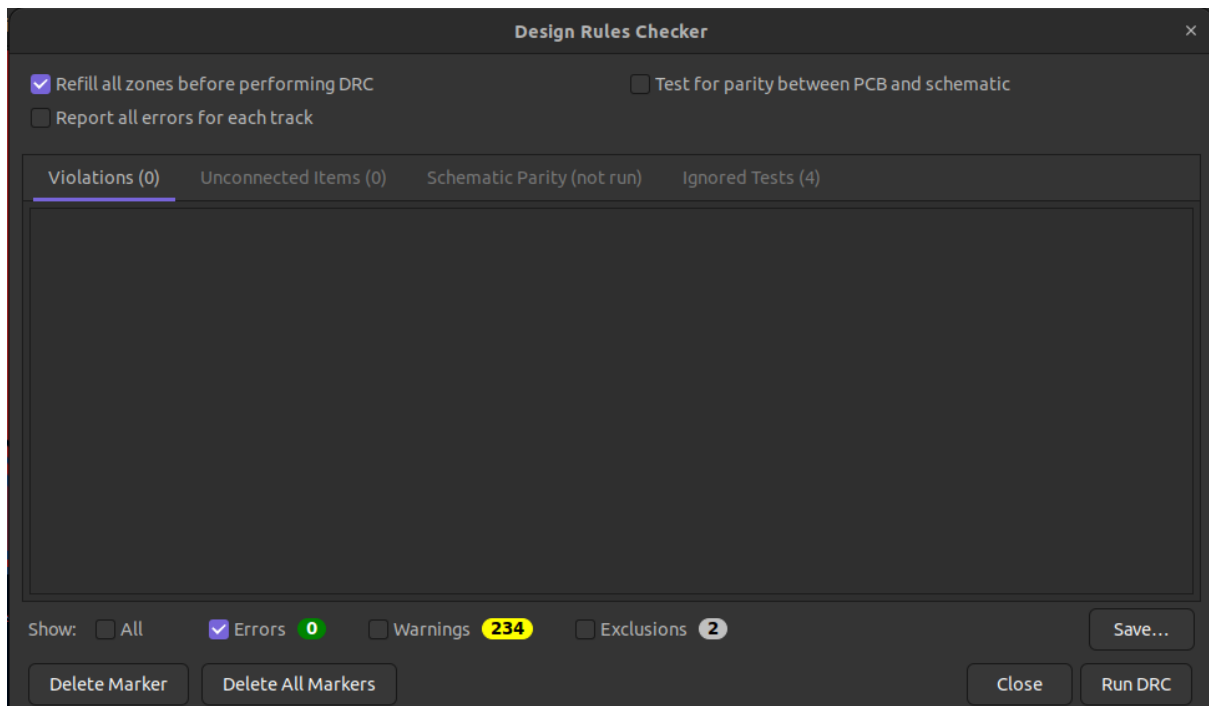
Q9: DRCs

DRCs used were:

Minimum distance between tracks

Check that all pads are connected to the correct nets

Check for thermal violations (GND pins must be connected to the GND plane with at least 2 spokes)



The 2 exclusions were:

- A GND pin on an IC that was unable to be connected to multiple GND spokes due to the physical layout of the IC pins
- A GND pin on the microcontroller header which was flagged for only connecting to 1 GND spoke on front copper even though it was connected to 2 spokes on the back copper.

The warnings were silkscreen overlaps and were not of concern.

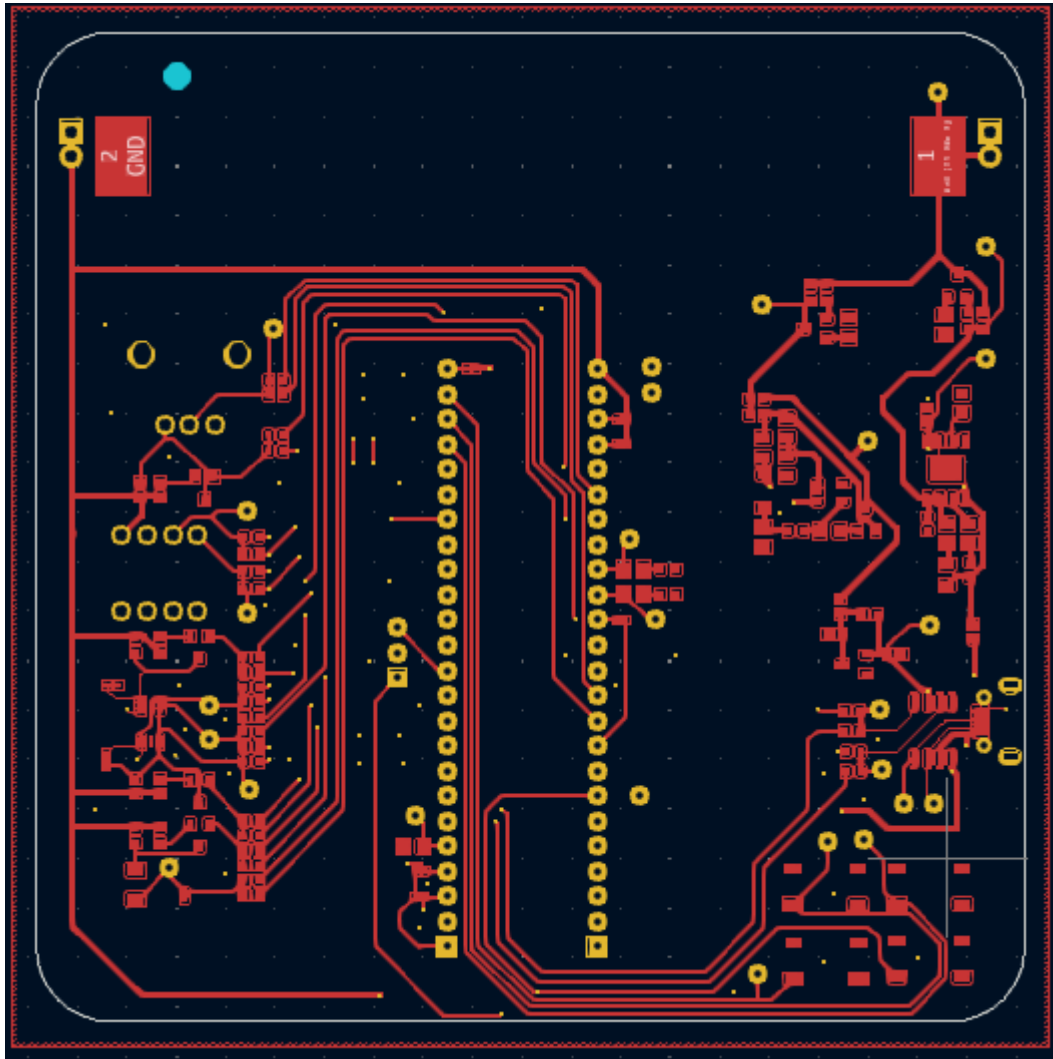
Q10: BOM

https://github.com/Carciac/EEE3088F-Project-CKR/blob/main/Budgeting/EEE3088F_CKR.csv

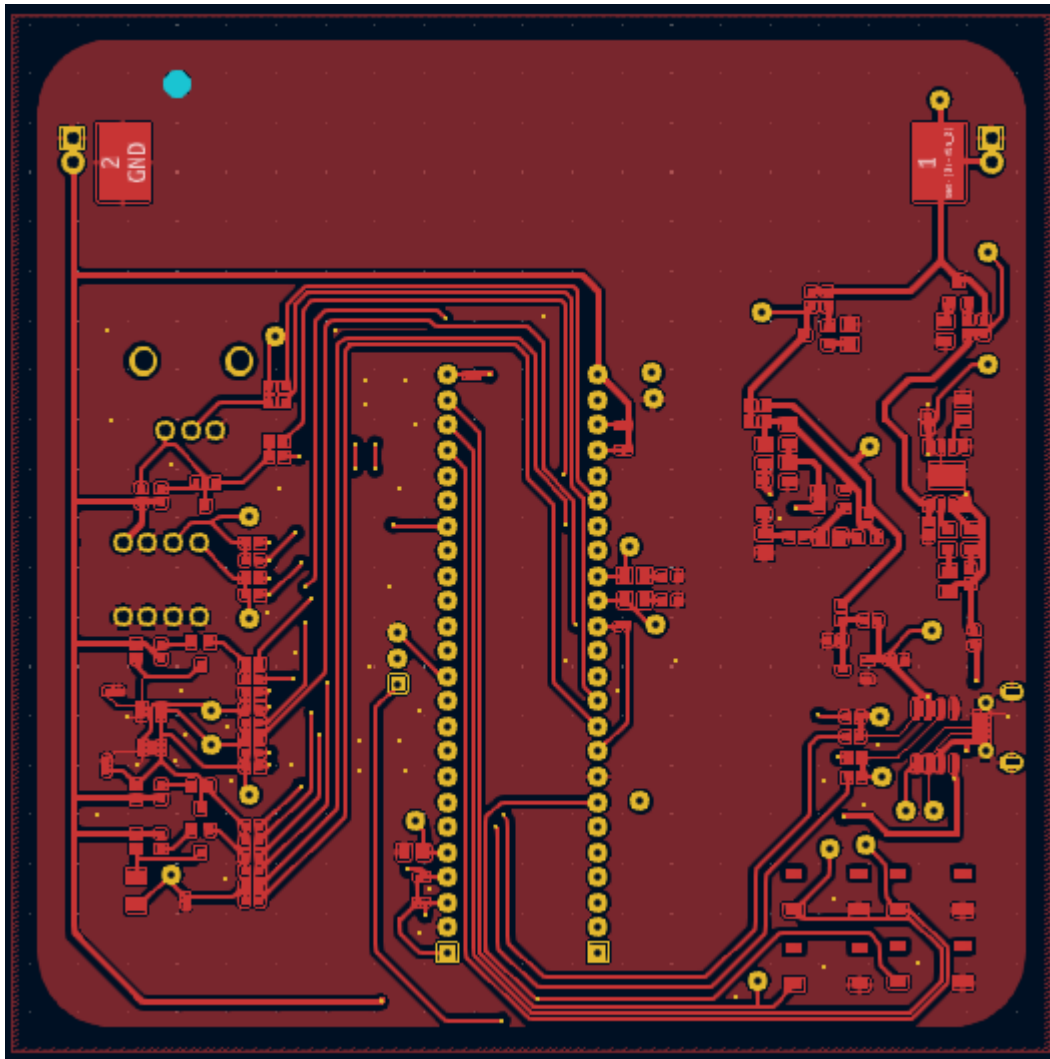
The estimated budget (excluding manufacturing cost) is \$56.50. We are account for an ~\$12.00 manufacturing cost (board and assembly), and thus are still within our budget.

Q11: PCB design

Front copper layer:



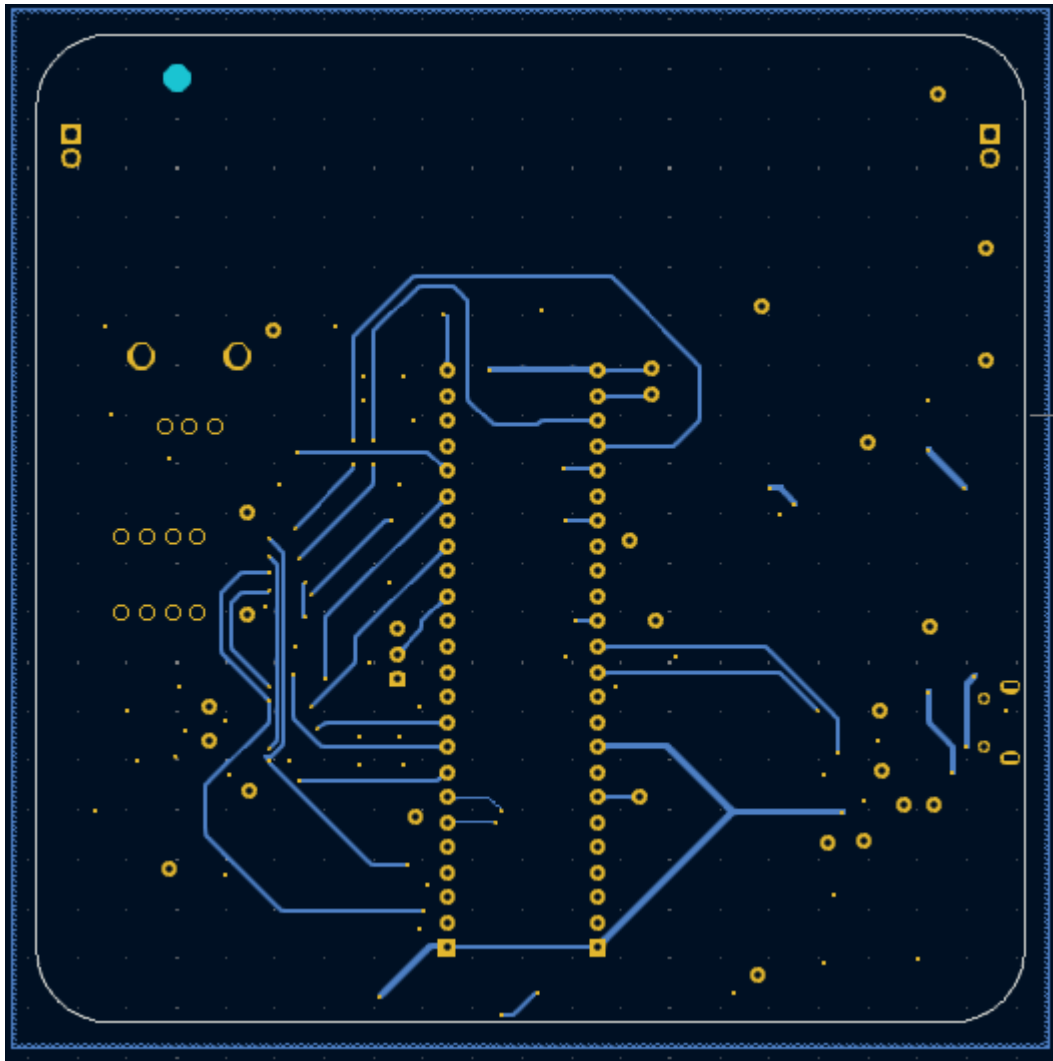
Front copper layer with GND plane shown:



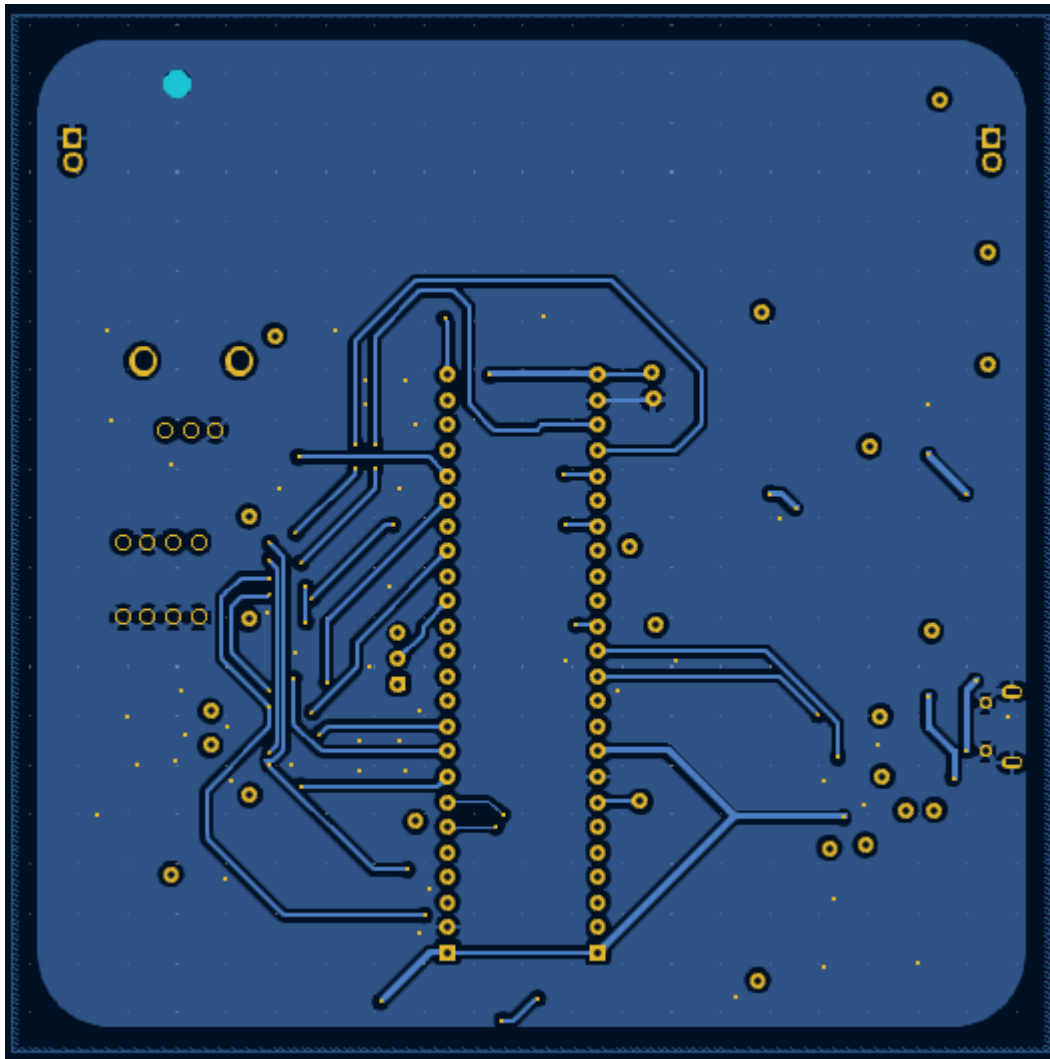
The lightly-shaded area is the GND plane.

0.6mm tracks were used for power lines, 0.4mm tracks were used elsewhere. For some ICs with small pads, 0.25mm tracks were used in small sections.

Back copper layer:



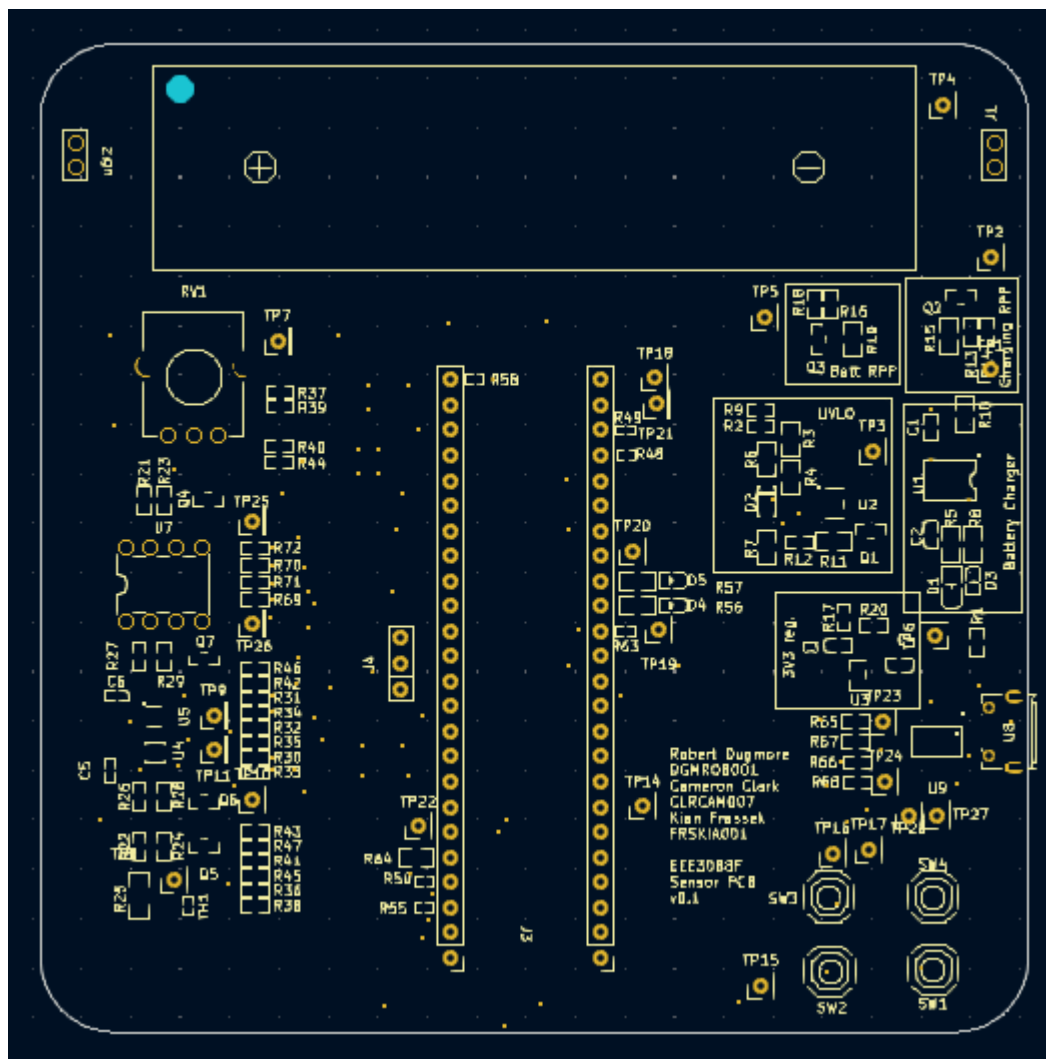
Back copper layer with GND plane shown:



The lightly-shaded area is the GND plane.

0.6mm tracks were used for power lines, 0.4mm tracks were used elsewhere. For some ICs with small pads, 0.25mm tracks were used in small sections.

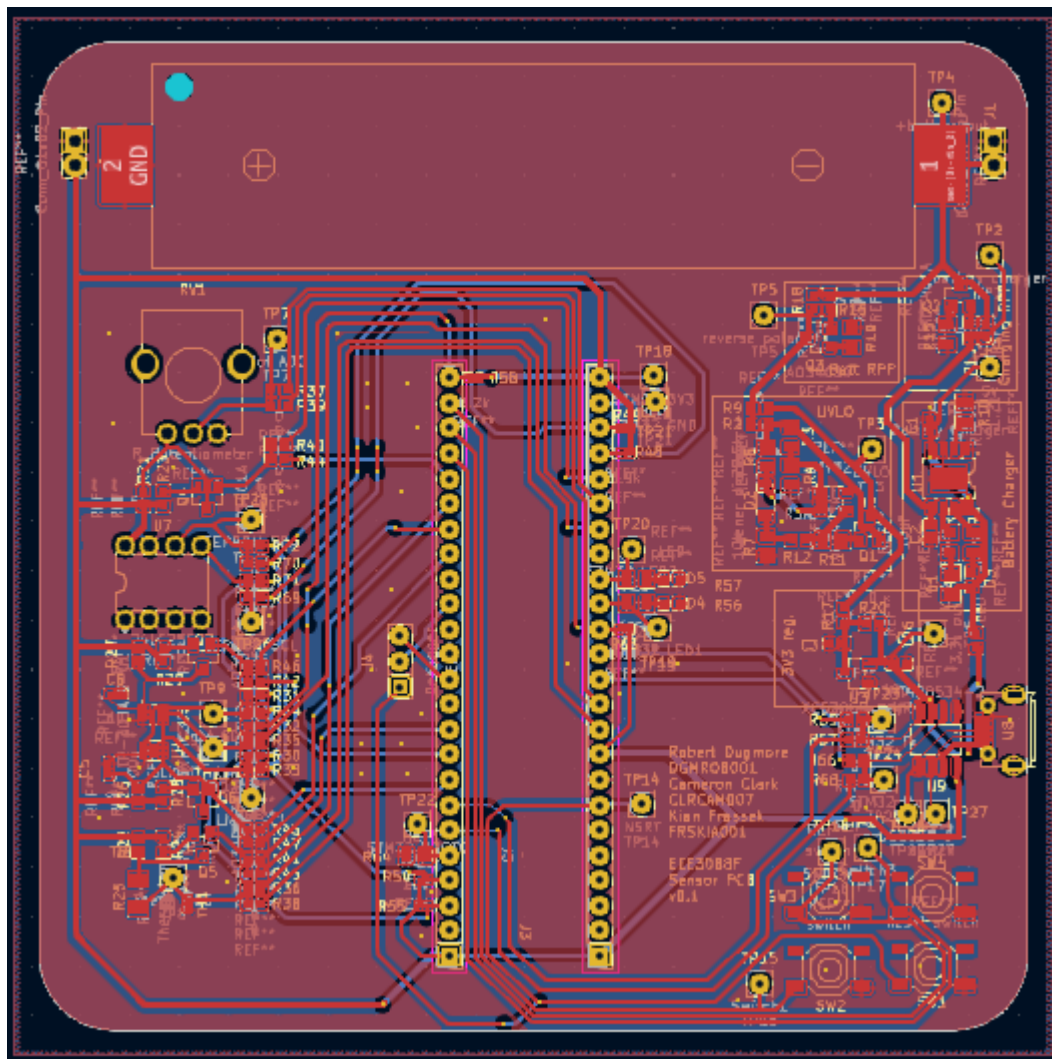
Front silkscreen:



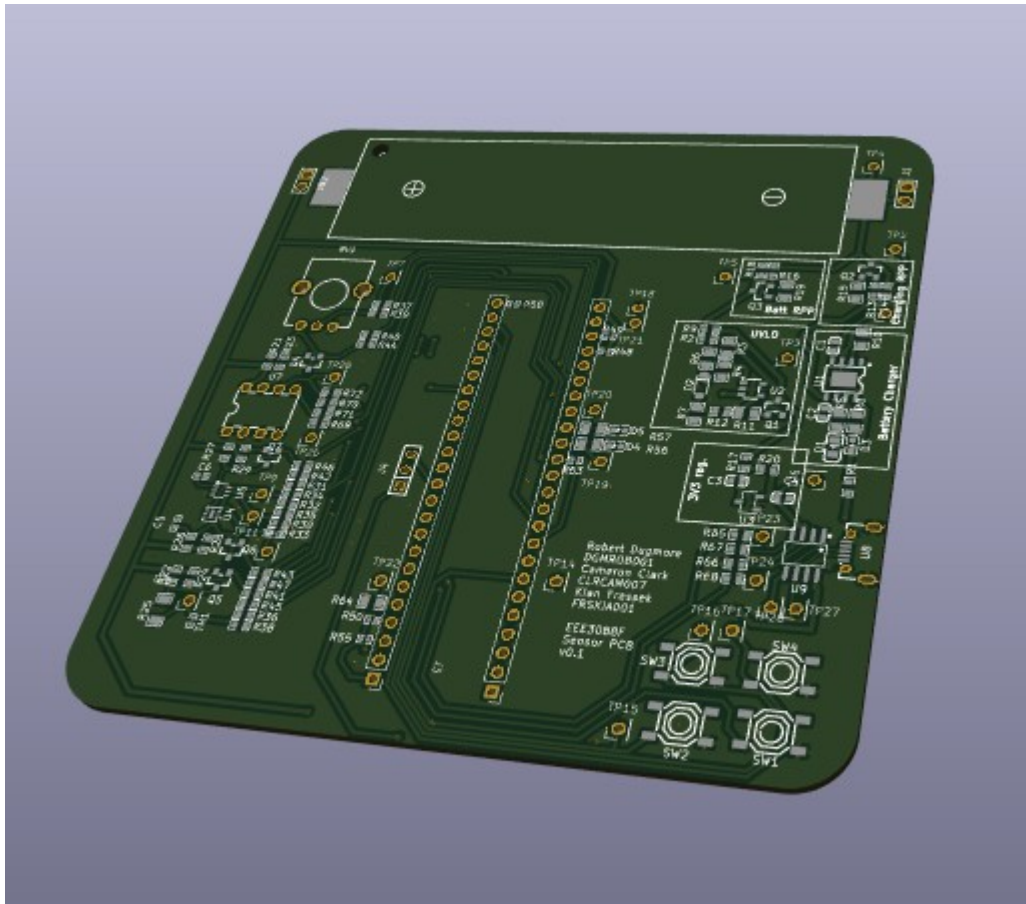
Note that the USB was intentionally allowed to stick slightly off the edge of the board to make it easy to plug in to.

Back silkscreen has not been shown as it was left empty.

All layers shown:



3D view:



Link to gerber files:

<https://github.com/Carciax/EEE3088F-Project-CKR/blob/main/Gerbers>