EEE088F 2022

Initial Design Template
Group 7

JNSRYA006 TRFDEV001 CRGALE002

Q1 Github [1]

[GitHub Repository Link here]

Q2 Power Subsystem Failure Management [2]

Component Shortage:

Should there be a component shortage, alternatives for all components are available on JLCBPCB as basic parts and in similar price ranges except the AMS1117-3.3 (The voltage regulator), LM393DRG (Comparator in the UVLO), MICROXNJ (USB connector), 1N5819WS (Schottky Diode) & TP4056-42-ESOP8 (Battery Charger)

For these 5 ICs, the following alternatives are available:

Unavailable Component	Alternative Component	Alternative IC Part Number	Alternative IC Price	Price Increase per board
AMS1117-3.3	AMS1117-3.3V	C173386	$0.6858	$0.5003
LM393	LMS393DR	C67470	$0.7862	$0.6396
MICROXNJ	MICRO-01-G1P1-A1T2	C2682504	$0.6752	$0.6256
1N5819WS	1N5819W	C1884550	$0.6146	$0.6043
TP4056-42-ESOP8	TP4056X-42-ESOP8	C2763448	$0.8511	$0.5933

For all other components, such as resistors and capacitors, alternatives are available, as basic parts, for similar prices on JLCPCB.

Component failure/destruction, Trace Damage & Errors Post Manufacturing:

For component failure, trace damage & errors detected after manufacturing has been completed, the… ~~following points of the power circuit are able to be isolated, and excluded using jumpers which connect to the following pins on the STMDiscovery Board:~~

~~**Figure 1:** Diagram showing pin locations for jumpers in power module~~

…following Test Points have been included in the schematic design and correspond to the following points in the power module:

- TP1 - VCCBus (5V Supply from USB)
- TP2 - USBDETECT
- TP3 - USB (D+)
- TP4 - USB (D-)
- TP5 - Output of Input Polarity Circuit
- TP6 - Output of Battery Polarity Protection Circuit
- TP7 - Output of the UVLO
- TP8 - GND

Q3 Sensing Subsystem Failure Management [2]

In the event of a component shortage, three alternative temperature sensors with similar specifications and cheaper or equivalent unit price have been found: these have the reference codes: LM75BDP, GX18B20U, LM70CIMMX-3/NOPB.

Additionally three alternative light sensors with similar specs and cheaper or equivalent unit price have been researched and found. Their references are: BH1603FVC-TR,

BH1680FVC-TR, ALS-PDIC15-21C/L230/TR8

Testing and debugging will be crucial for the sensing unit since it is the source of all of the project's input data. For the sensing sub-unit each sensor's data line will have connect/disconnect points in the form of male pins on the board which can be connected/disconnected via a female jumper. In the event that there is component damage/failure and the sensors are giving incorrect readings this will allow us to isolate their data outputs from the rest of the PCB and identify the cause of the issue without the consideration of all the other components and their potential for loading/distorting the sensor's readings . Each sensor will also have a connect/disconnect point from the power source. The sensor will also be positioned such that their data line traces are as far from any high current traces as possible. This is to minimise any noise that may be produced by the electromagnetic fields around these traces.

The sensors will also be positioned with space around them on the PCB in case of errors only detected post manufacture. This will allow for an easier modification of the circuit around the sensor should it be necessary.

Additionally sensors with relatively large pins were chosen to facilitate the placement of probes directly on the part for troubleshooting. If the part gets damaged and needs replacing this will also make for an easier job when desoldering it and resoldering a replacement.

Updates:

To mitigate the risks associated with trace damage, the sensing sub-module has been placed far from the other modules with generous spacing between traces. The sensors have also been placed in close proximity to the data pins which connect to the stm board, this was done so that in the event of trace damage, these sensors could be manually jumpered to their respective pins without needing long untidy jumpers.

Q4 Microcontroller interfacing Failure Management [2]

(i) I have chosen an EEPROM and USB To UART chip that are surface mounted devices with long leads and therefore will be unsolderable if a device fails. The same applies to resistors and capacitors in the circuit.

(ii) Double traces with jumpers will be added in order to allow for backup traces that can be used in the case of trace damage. Also I added Leds in order to test whether traces are damaged or not.

(iii) There is a large family of CH340 chips that can be used as USB to UART chips. The CH340G was chosen for its price and so there are a number of alternatives that are only slightly more expensive.

(iv) I have mentally prepared myself for the possibility. But apart from that I have inserted as many jumpers as our budget will allow in order to rewire connections. Furthermore, I will draw traces to unused pins in case they are needed once the board has been printed.

Updates:

- In the event of a component shortage, the M24C64-RMN6TP can be used as a replacement chip for the AT24C256C-SSHL-T. It has lower storage, but still meets the overall project specifications and uses I2C to communicate. It is also slightly cheaper so it was a very close second choice for EEPROM initially. Using this chip would bring the budget down by 0.6$. The CH340G can be replaced by the CH340T. It is more expensive by 0.22$ so it will increase costs slightly, but it has the same features that are required for the USB to UART submodule.
- I have added silkscreens indicating what each test point is for. This will help during debugging.

Q5 Power Subsystem Schematic [2]

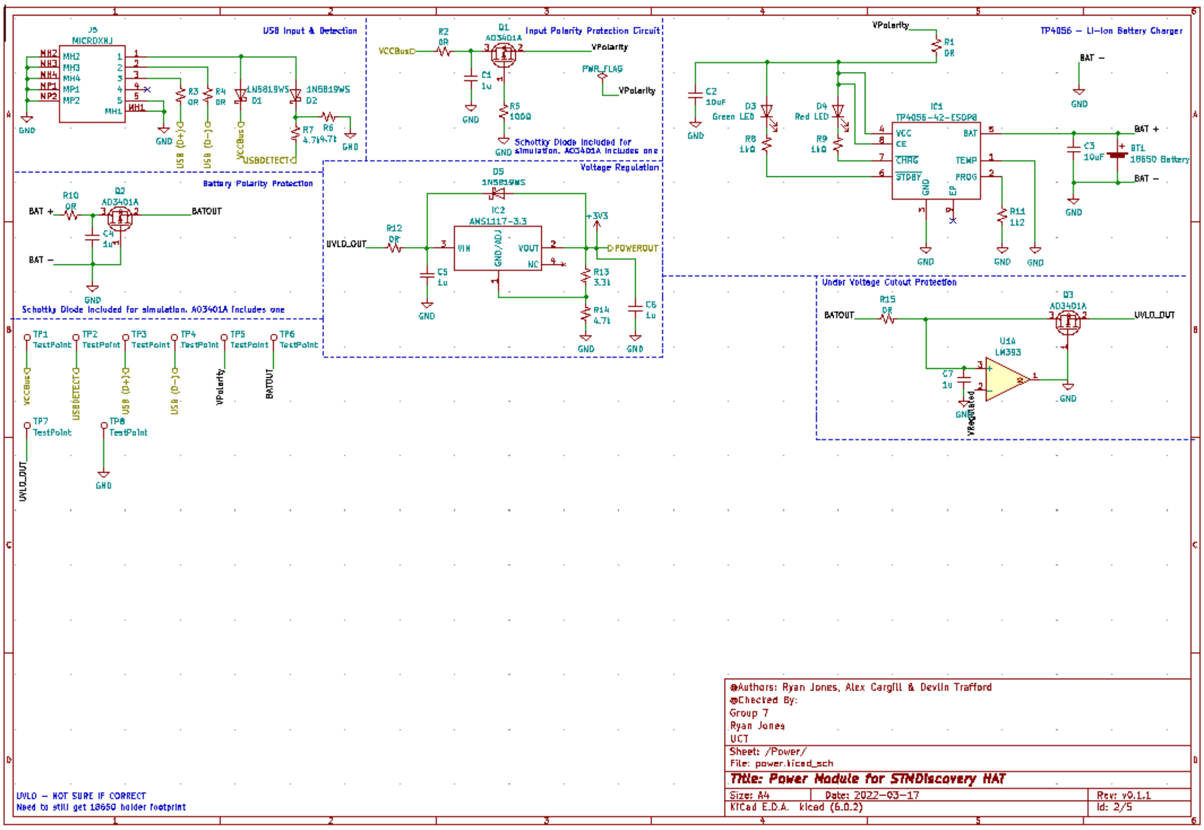

Figure 1: Diagram showing Power Module Schematic

[Please click here for a better quality image]

Updates:

- Added Test Points
  - Allow testing across two points in the module and against GND
- Removed voltage divider at Pin 1 of U1A (Comparator - LMS393DRG)
  - Not required, as the voltage at pin 1 (which is compared to reference voltage) is the voltage that needs to be compared
- Removed Schottky Diode (1N5918WS from polarity protection circuits)
  - The A03401A includes one - the Schottkys were only included for simulation

Q6 Sensing Subsystem Schematics [2]

No changes were made

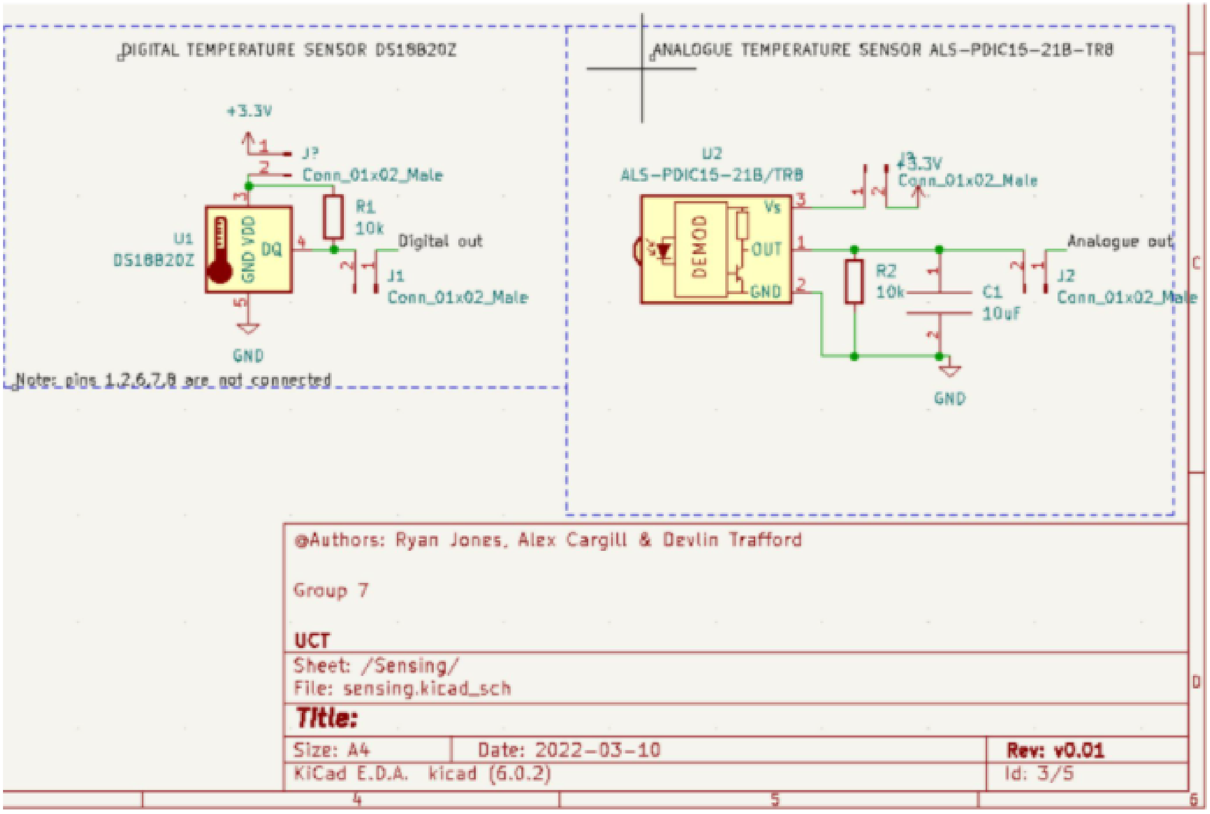

Figure 2: Diagram showing Sensing Module Schematic

Q7 Microcontroller interfacing Schematic [2]

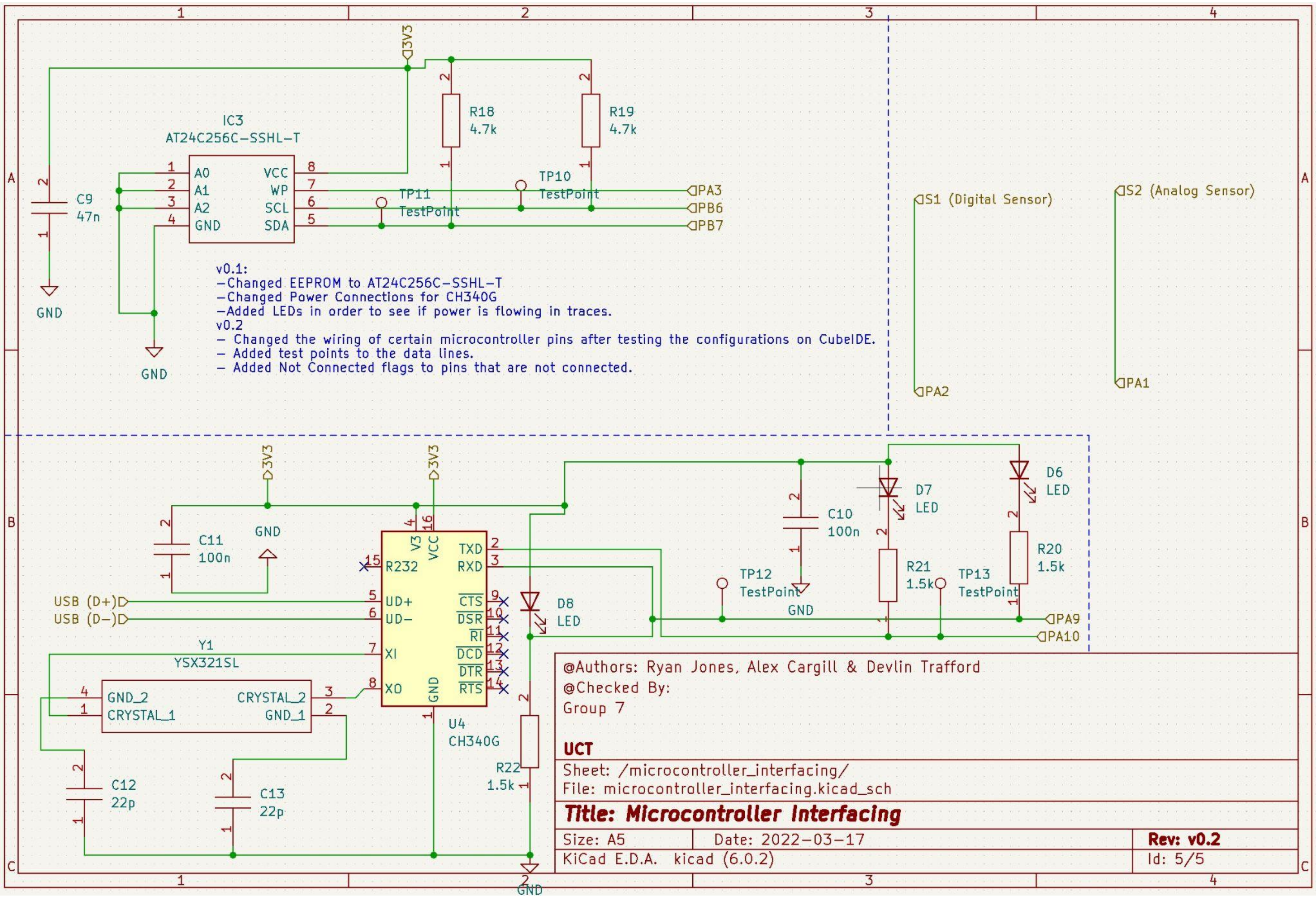

Figure 3: Diagram showing Microcontroller Interfacing Module Schematic

Updates:

- Added Test Points
  - Added test points to the RXD and TXD data lines of the CH340G as well as to the SCL and SDA lines of IC3.
- Changed the connections between the microcontroller and the UART to USB and EEPROM submodules.

    ○ I checked which pins were initialized in CubeIDE for the different communication protocols and found that they did not match those that I found online.

- Added Not Connected flags to unused pins.
    ○ It was giving errors during the ERC.

Q8 Updated ERCs [2]

ERCs are designed to check if a circuit is electrically correct and connected correctly. This consists of the following checks if:

- All schematic connections are valid,
- There are any inputs or outputs that conflict,
- Any ports or pins are open and unconnected,
- And if any ports or pins overlap

KiCad gives a ERC.txt file after running an ERC, which is available on GitHub here.

The output of the ERCs is also shown below as an image.

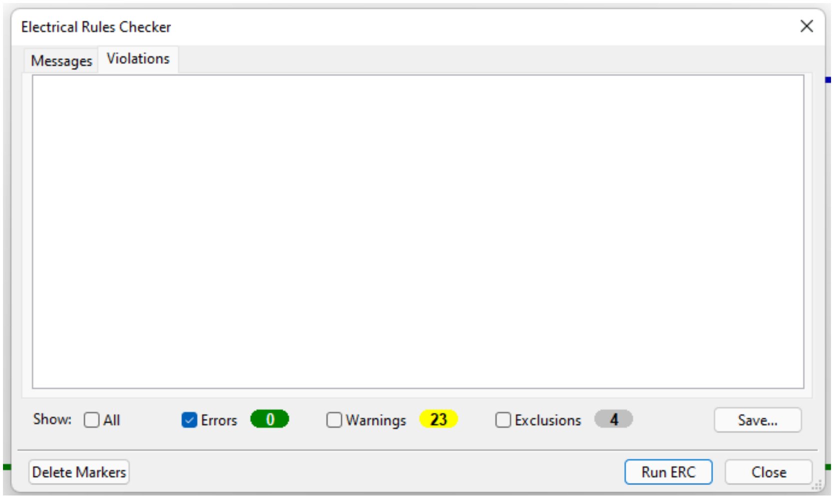

Figure 4: Screenshot after running ERCs in KiCad

The exclusions in the report are related to issues with symbol's downloaded from the internet and the way that their pins have been set up. These exclusions produced a *"Input Power pin not driven by any Output Power pins"* error. These can be ignored, as changing the input type of a schematic symbol will fix these checks. Furthermore, 2 more of the exclusions were due to connectors, which in the PCB operation, will have a connection between them, so this error was excluded.

Q9 DRCs [2]

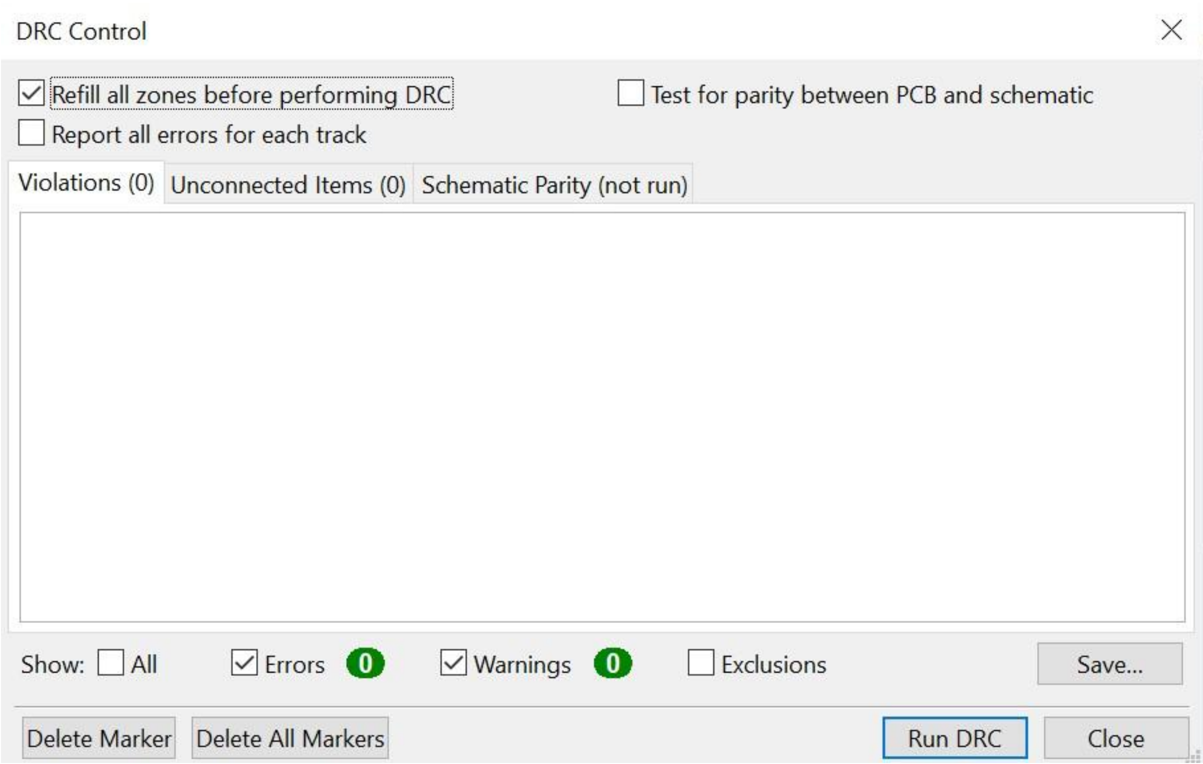

Figure 5: Screenshot after running DRCs in KiCad

No warnings or errors given. KiCad gives a DRC.txt file after running an DRC, which is available on [GitHub here]

Q10 Updated BOM [5]

[Link to BOM on GitHub Here]

Budget Total: $8.8033

Q11 PCB [15]

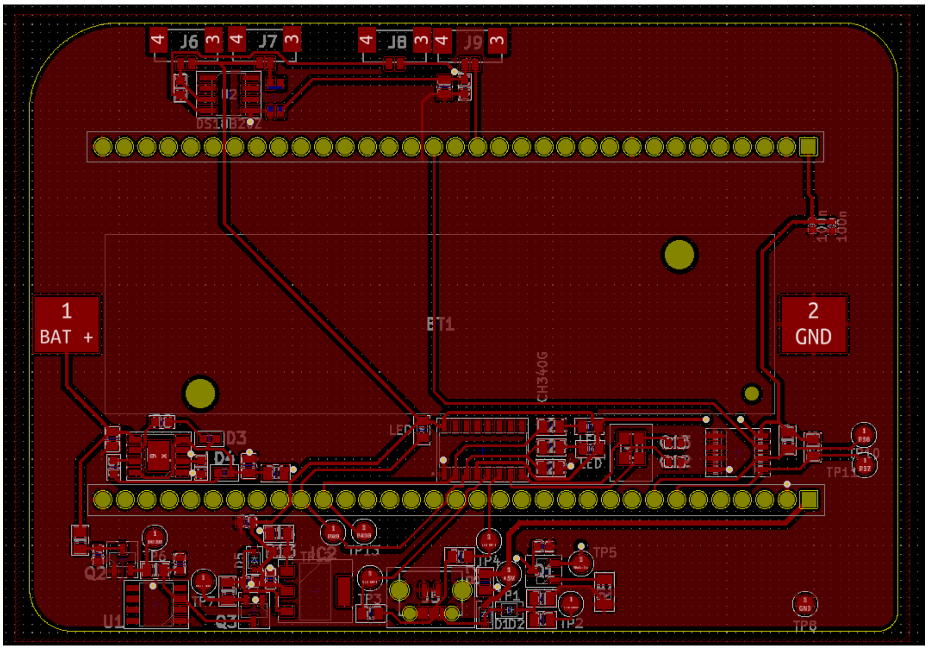

Figure 6: Screenshot Of the Front Copper Layer of the PCB

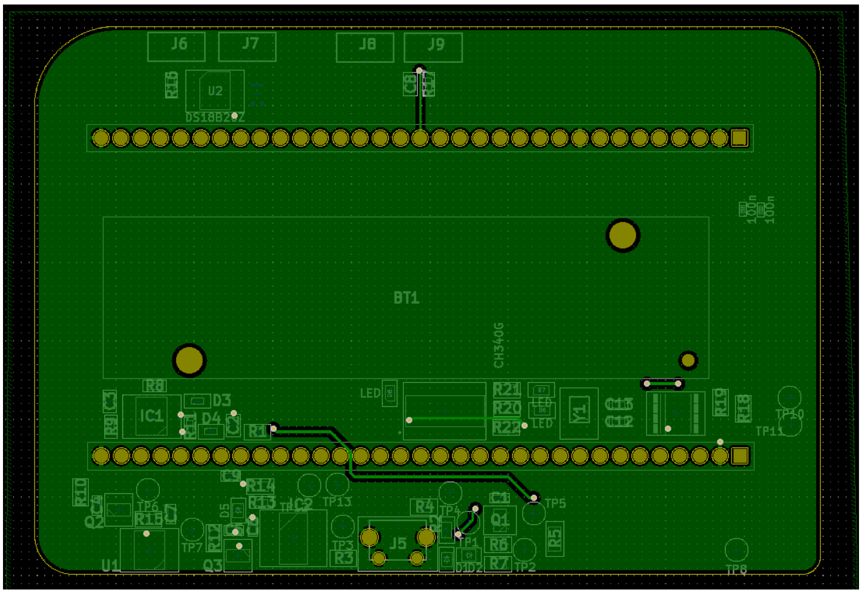

Figure 7: Screenshot Of the Back Copper Layer of the PCB

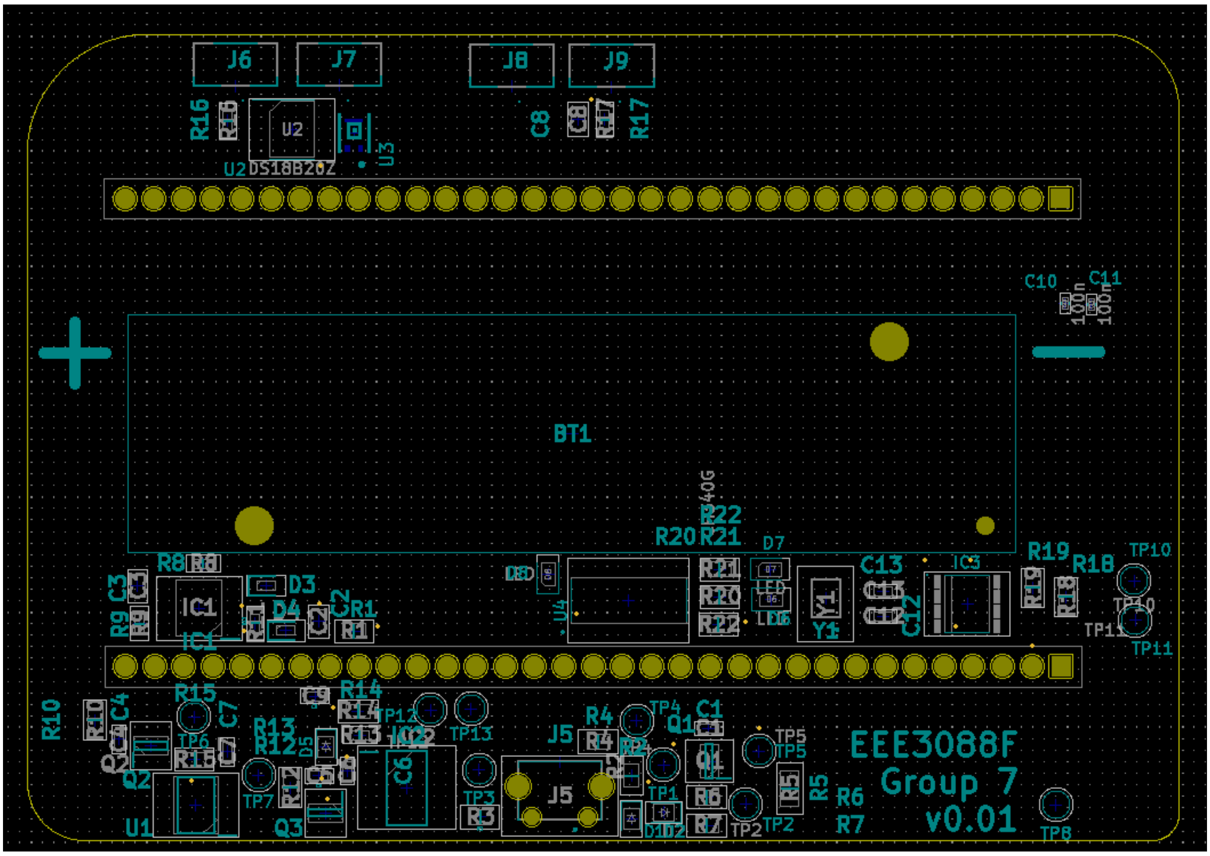

Figure 8: Screenshot Of the Front Silkscreen & Fabrication Layers of the PCB

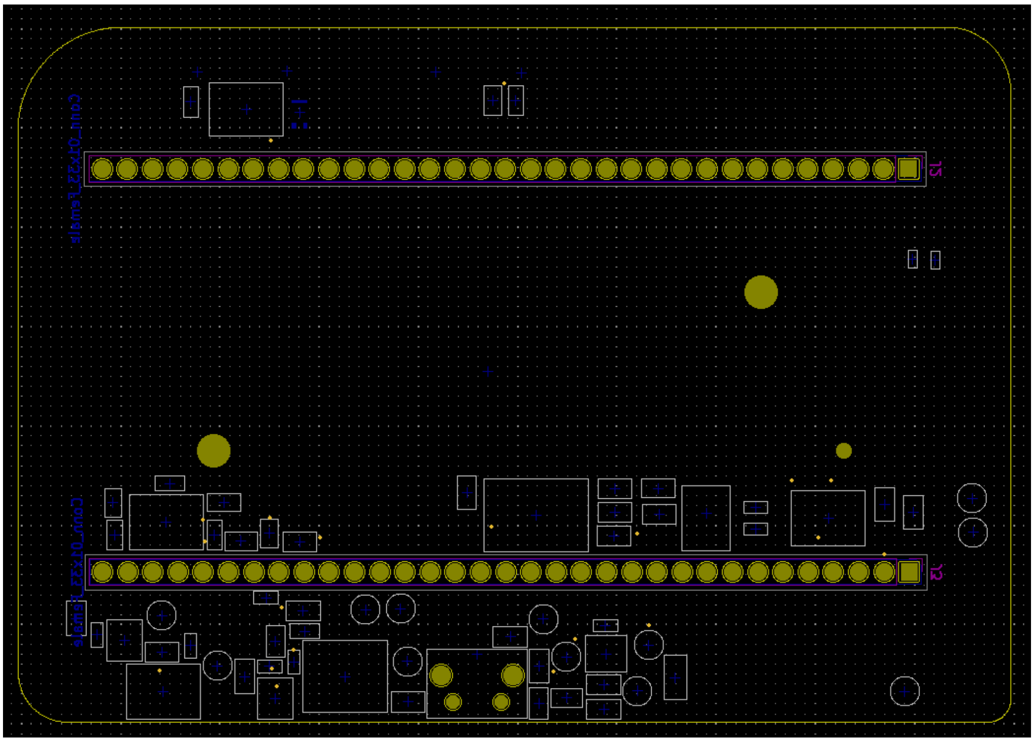
Figure 9: Screenshot Of the Back Silkscreen & Fabrication Layers of the PCB

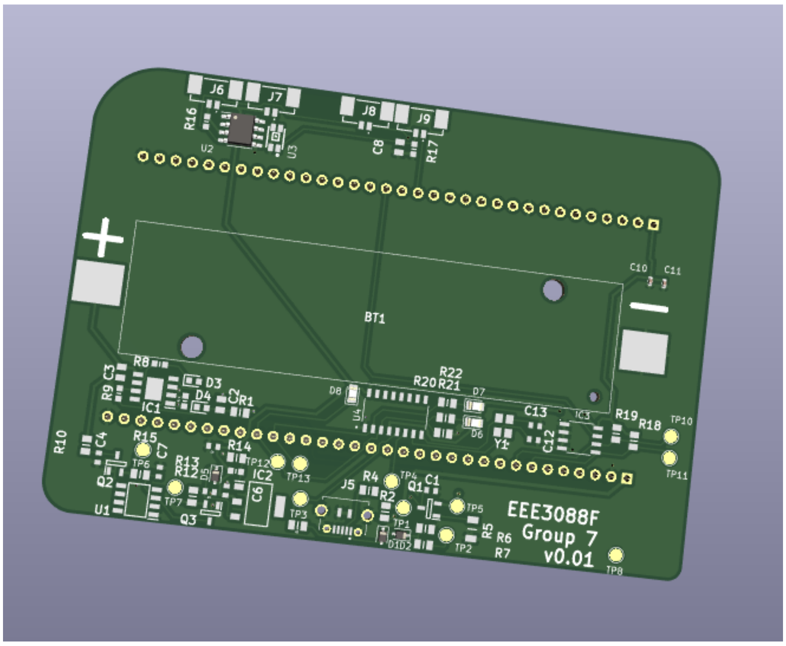

Figure 10: Screenshot Of the Front of the PCB in 3D Viewer

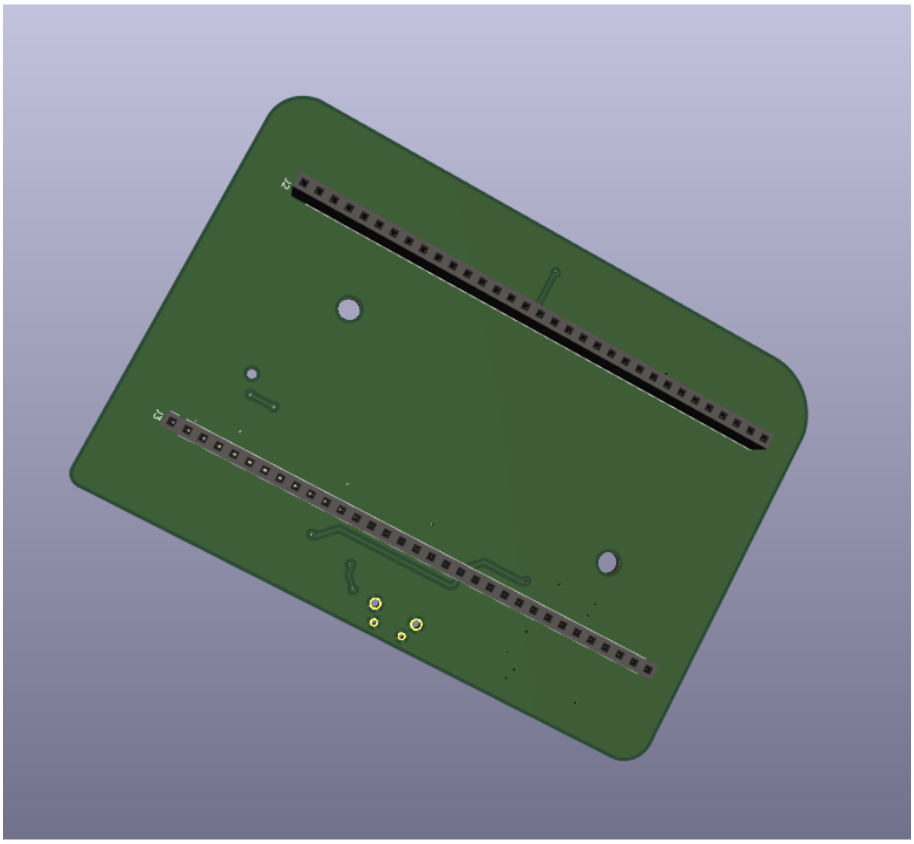

Figure 11: Screenshot Of the Back of the PCB in 3D Viewer

[Gerber Files in GitHub Here]