Arnav Gupta

arnav11@ualberta.ca   Portfolio   LinkedIn   +1 (403) 899-2495

Embedded Hardware & PCB Design Portfolio

This portfolio provides a visual overview of some of the hardware projects I have completed so far. It includes selected schematics, PCB layouts, and summaries of the design goals and key implementation details. While not exhaustive, the sections aim to highlight the overall system architecture, layout approach, and my personal thought process when designing each board.

Projects Covered

- High Power Battery Management/Monitoring/Levelling and Power Distribution Unit
- Custom Quadcopter Electronic Stack
- Custom Ethernet Switch and Injector
- Custom USB 3.0 Hub

Skills and Software Used

- PCB Design (Altium Designer, KiCad)
- STM32 Embedded Systems and Firmware Integration
- High-Speed Routing (USB 3.0, Gigabit Ethernet)
- High Power Electronics Design (Battery Management & Power Distribution Unit)
- Fusion 360 and SolidWorks (For chassis and controller shell design)

High Power Battery Management/Monitoring/Levelling and Power Distribution Unit

Role: UUARG Solo Project **Date:** March 2026 – Present
Tools: Altium Designer, LTspice

Battery Management/Levelling/Monitoring and Power Distribution Layouts

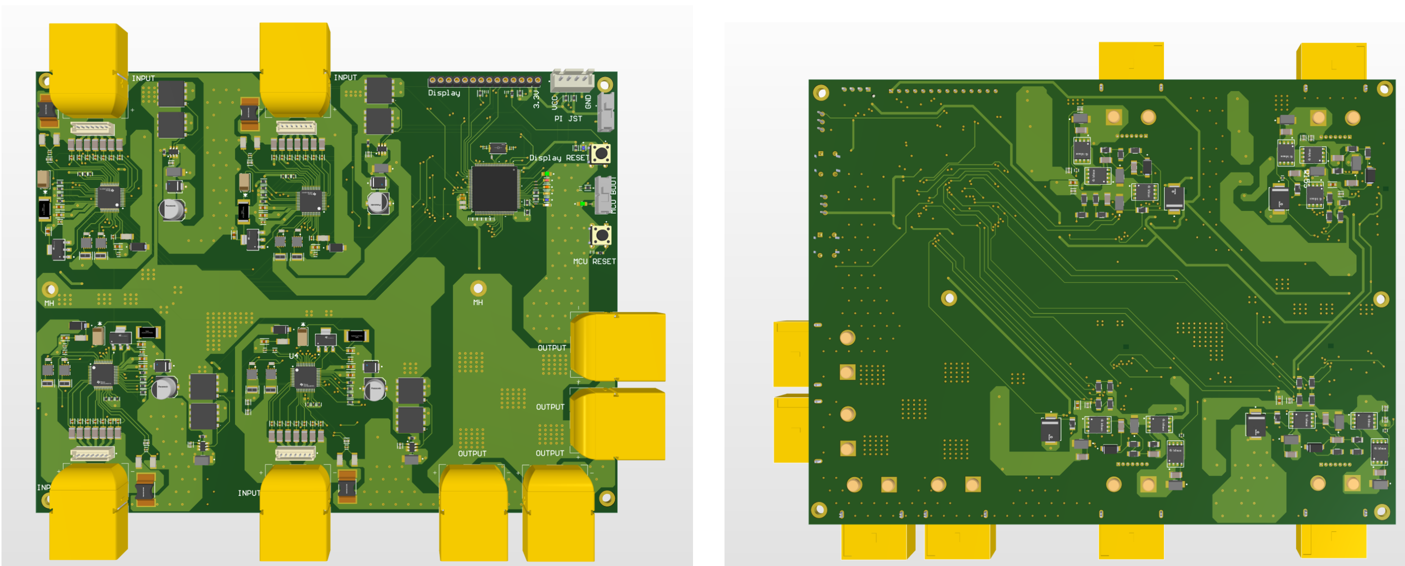

Figure 1: BMU & PD - Layout Top and Bottom

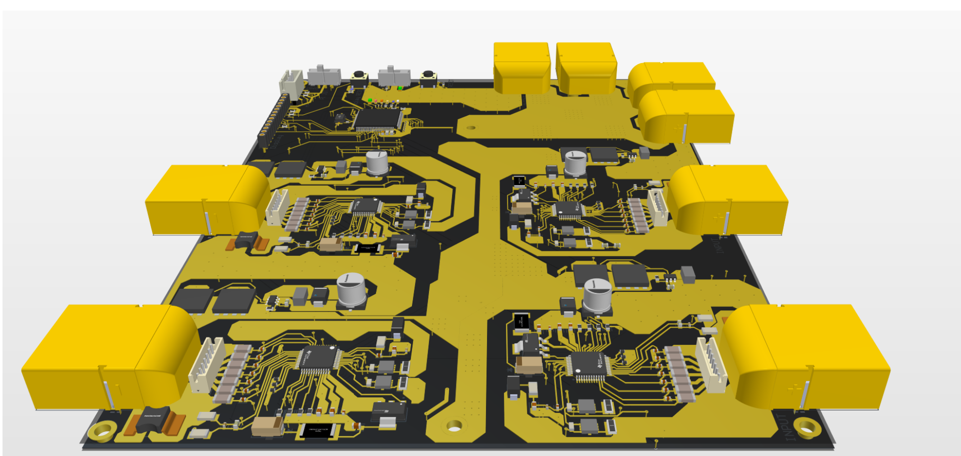

Figure 2: BMU & PD - Layout Isometric Side View

Design Requirements (Given From UAARG (University of Albert Aerial Robotics Group)

- Max PCB size of 150mm x 200mm
- Keep cost as low as possible while using reliable components
- System should be able to monitor/level up to 4 LiPo 6s batteries
- System should be able to output 6s as well as up to 170 Amps (Max draw from ESCs and FC on the main drone)
- System should be able to run on 1,2 or 3 batteries

- System should have a JST output (I2C) for communication with Pixhawk and Pi Configuration
- System should have a set of female header pins to be able to attach an ILI9341 display
- System should have 4 output connectors (2 for ESCs and 2 for miscellaneous)

Design Overview & Main Components

- To monitor the 6s LiPo Batteries, a BQ76952PFBR BMU IC is used for its relatively low cost and good performance. IC can monitor the battery via the JST connector cells, and supports temperature and current sensing
- For current sensing, a 0.2 mOhm resistor was chosen with simple calculations due to its low power dissipation (good balance of "low" power dissipation and not super high cost)
- BQ76952PFBR IC also supports a PFET system, which allows for precharge of the ESC capacitors. (This is talked about in more depth in Key Design Considerations)
- NTMTS001N06CTXG Mosfets with a LM5050MK Controller IC to act as an ideal diode to allow batteries to supply power to a main bus; however, not receive any power from this main bus
- Bus Bar implementation from input to outputs for a low resistance path
- STM32G474 MCU for its good cost and speed
- General-purpose LEDs for indication
- Switches and buttons for resets (MCU and Display), and a boot loader switch for the MCU

Key Design Considerations

- Layer Stack: L1: Signal/GND — L2: Signal/GND — L3: GND — L4:Signal/GND
- (Note: Power will only be present on L1 and L4)
- For all high voltage or high current lines, thick copper traces or copper pours are used
- As this is a high-power board, thermal vias for heat dissipation are needed (spacing between vias cannot be to close, this would lead to delamination of copper or copper cracking during thermal cycling, as well as the increased cost)
- thermal vias should be near outputs, high power lines, and ideal diodes (as they get very hot)
- Keep MCU separate from high power lines
- Assure that the crystal oscillator has no nearby traces to maintain integrity
- Add stitching vias for low impedance returns for signal traces where integrity is important
- Utilize the ability to place components on top and bottom layers due to the number of components
- PFET stage (as talked about very lightly earlier) is to allow pre-charging of the capacitors on the ESCs, when plugging in power to ESCs the power sees uncharged capacitors, which acts like a short and sometimes creates a loud pop sound. Adding a pre-charge system it allows the difference in voltage to be minimal, therefore removing this "short."
- Main mosfet stage which can turn on and off the connection from any battery to main bus (placed in parallel with the PFET system)
- Mosfet + controller to act as a ideal diode to allow no reverse current back into batteries
- Fuses were something I considered; however, after further consideration, by chance, if there is an error and the fuses blow unintentionally, this could lead the drone to crash
- decoupling capacitors as close to MCU as possible and stitching VIAs to connect ground planes
- 4 oz copper is used due to the high power nature of the board
- Use net-ties for layout simplicity later on

Battery Management/Levelling/Monitoring and Power Distribution Top Level Schematic

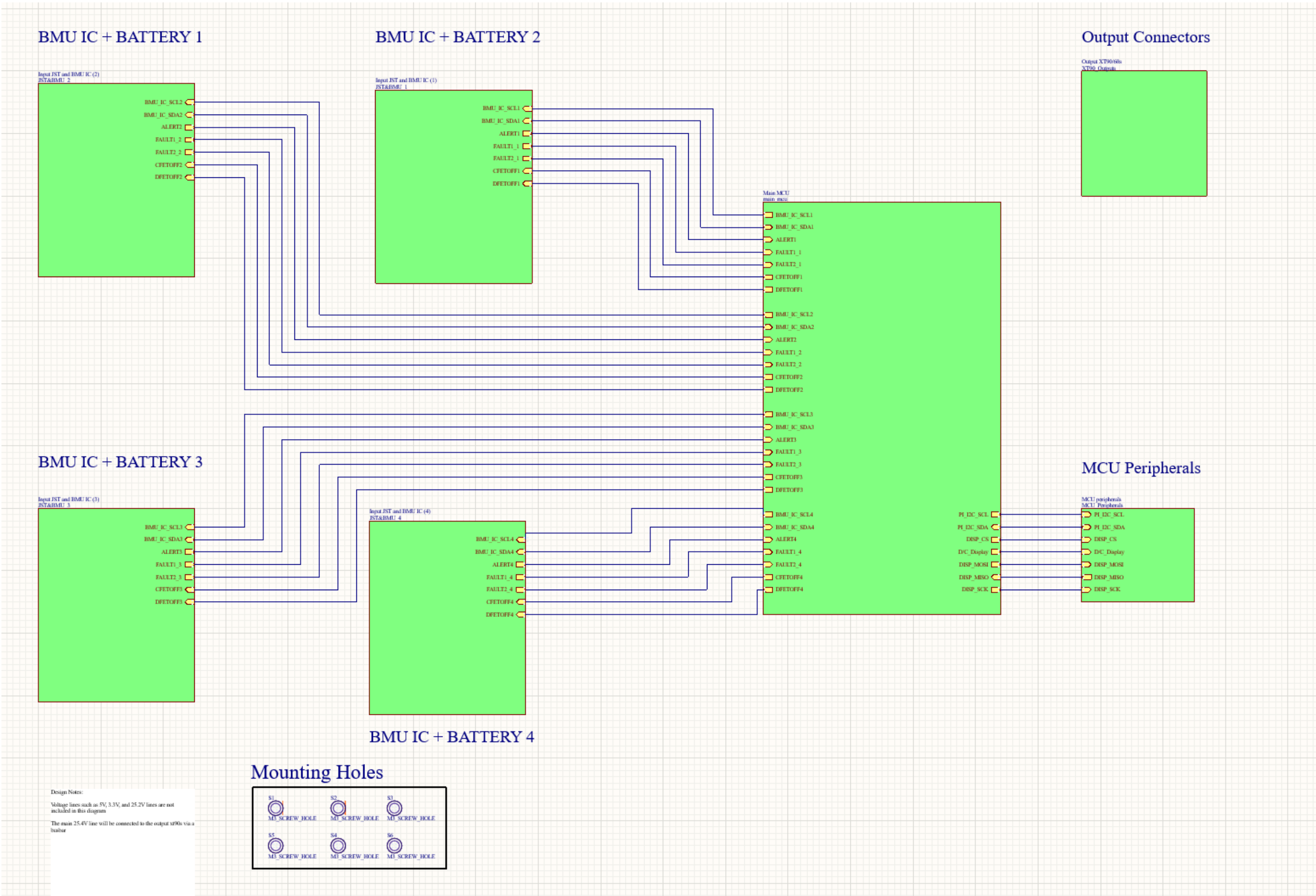

Figure 3: BMU & PD - Top Level Schematic

Note: (Bottom left of Image)

- Voltage lines such as 5V, 3.3V, and 25.2V lines are not included in this diagram
- The main 25.4V line will be connected to the outputs via a busbar

Battery Management/Levelling/Monitoring and Power Distribution Main MCU and Peripherals Schematic

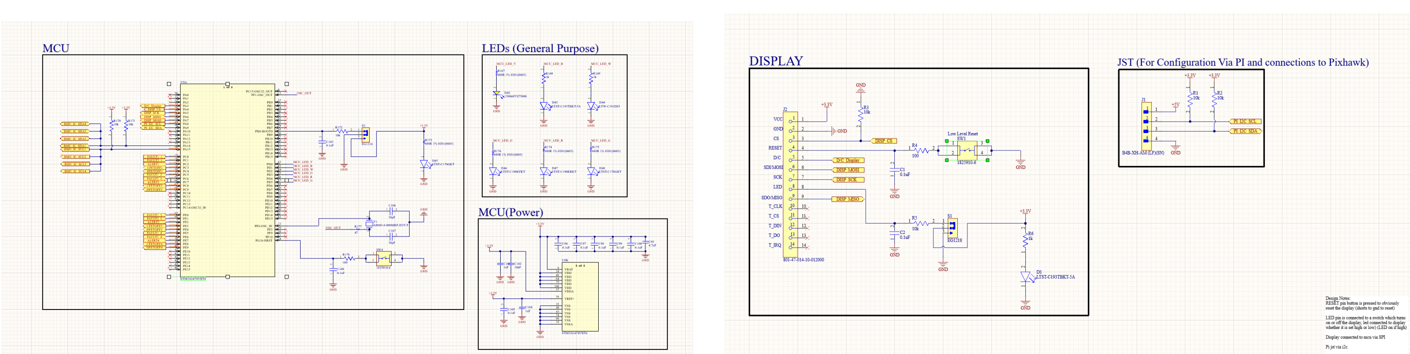

Figure 4: BMU & PD - Main MCU & Peripherals Schematic

Note: (Bottom Right of Image)

- RESET button (shorts to GND to reset) (For MCU and Display), Switch turns off the Display
- Display connected to MCU via SPI, PI JST via I2C

Battery Management/Levelling/Monitoring and Power Distribution BMU + Input Battery Schematic

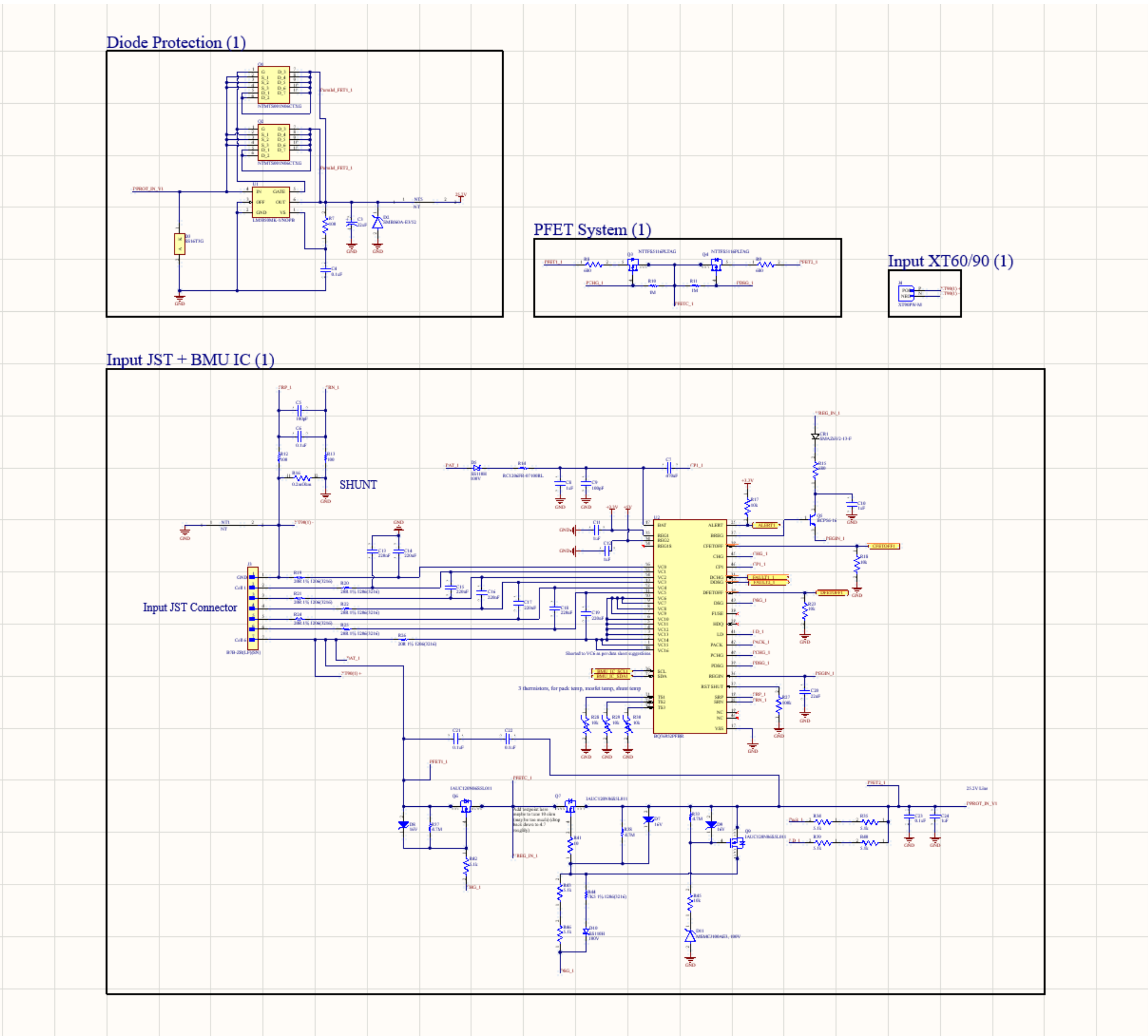

Figure 5: BMU & PD - BMU IC + Input Battery Schematic

Note: (Bottom left of Image)

- There are 4 of these in the design (Identical), obviously with different net names
- How the system is connected is as follows:

  Input Battery & JST Connector → Main MOSFETs (PFETs in parallel) → Ideal Diodes → Outputs

Custom Quadcopter Electronics Stack

Role: Solo Project **Date:** Feb 2026 – Present
Tools: Altium Designer, LTspice, SolidWorks, Fusion 360

Main Components

Flight Controller	ESC (Per Motor)	RF Controller
STM32H743 MCU	STM32G431 MCU	STM32G03 MCU
LSM6DSMTR IMU	DRV8323RS Gate Driver	Dual-Axis Joysticks
BMP581 Barometer	Shunt Current Sensing	External NRF24L01 RF Module
NEO-M9N GNSS	6 MOSFETs	MCP73871 BMU IC

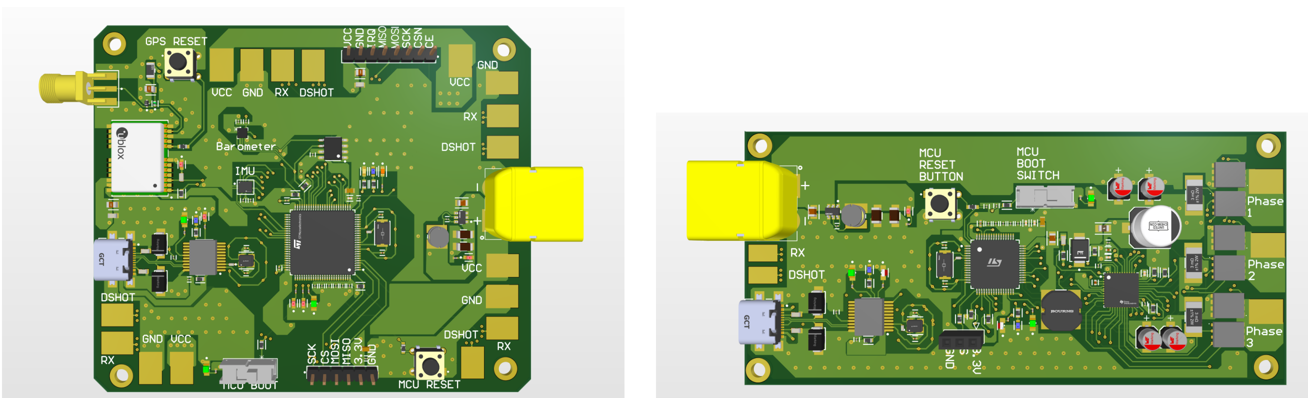

Figure 6: Flight Controller (left) and Discrete ESC (right)

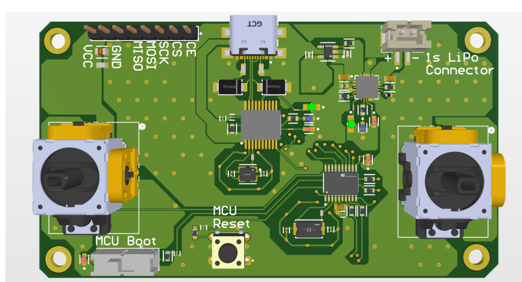

Figure 7: RF Controller Board

Key Design Considerations

- Designed such that ESC geometry was optimized to fit along chassis arms, while maintaining a square FC footprint for centralized mounting
- Orient the solder pads on the FC to allow easy connections to the ESCs on each wing
- Orient the RF board header pins in a way such that it's further away from the GPS Antenna to not allow minimal interference

Bring-Up & Validation

Implemented protection circuitry to allow for safe bring-up and buttons, such as reset buttons, for convenience. Also included header pins for an external RF board for controller inputs, as well as an additional set of SPI Protocol pins for convenience.

Flight Controller PCB (4 Layer Board)

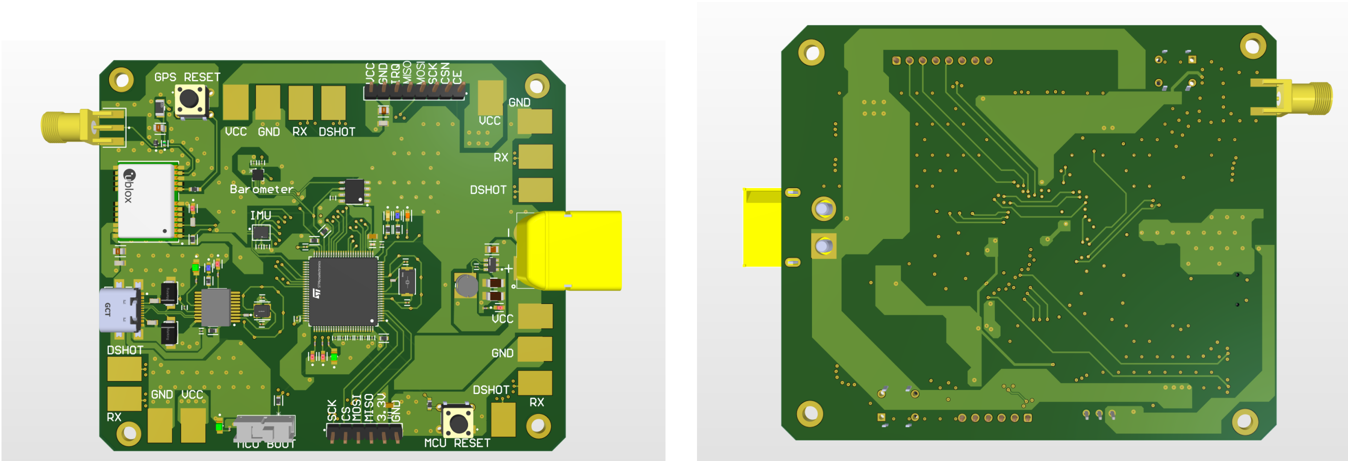

Figure 8: Flight Controller PCB — Top and Bottom Layers

Key Routing Considerations

- Create extra shielding for the GPS RF trace, compliant with the Data Sheet
- Make sure that Crystal Oscillators have no traces near them on any layer
- Assure that decoupling capacitors are placed as close as possible to pins
- Integrate stitching vias to link ground planes and stitching vias for low impedance return for certain lines
- Assure 90 ohm differential impedance for USB D+ and D- and 50 ohm impedance for other lines
- Assure Copper pours for GND and Thick copper pours or traces for Voltage and High current Traces

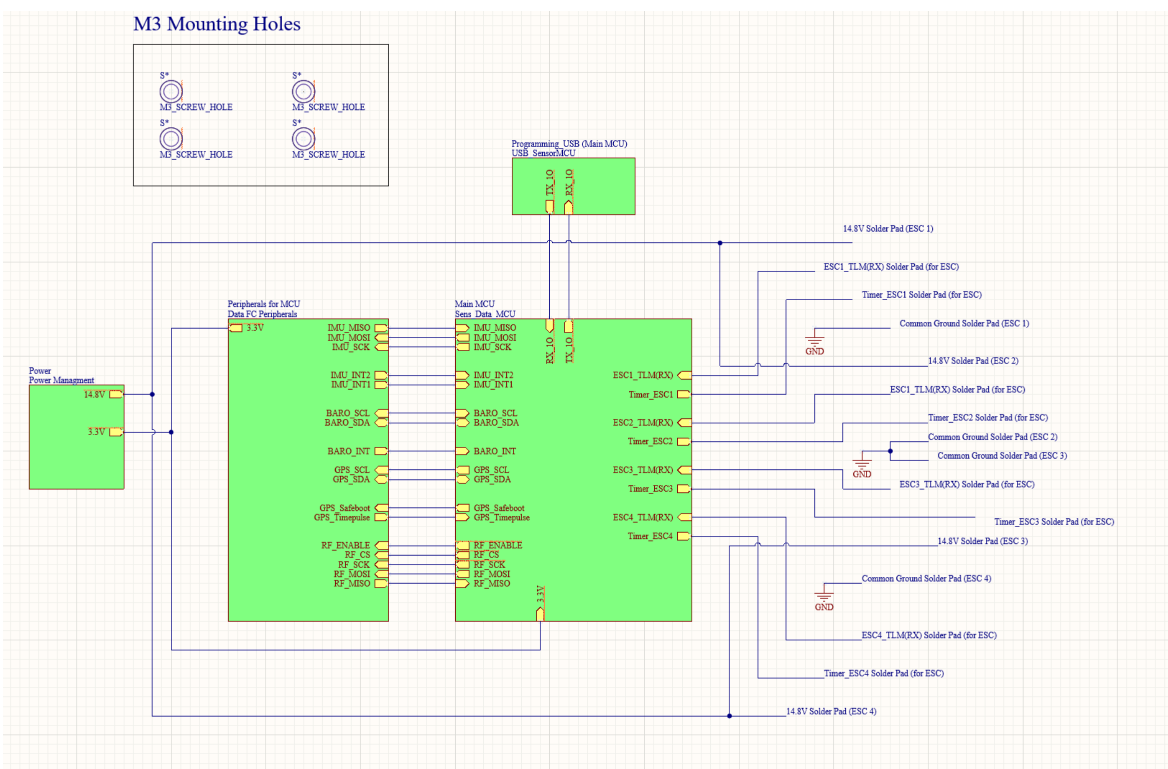

Figure 9: Flight Controller Top Level Diagram (Labels on the right symbolize Solder Pads)

ESC Power Stage (4 Layer Board)

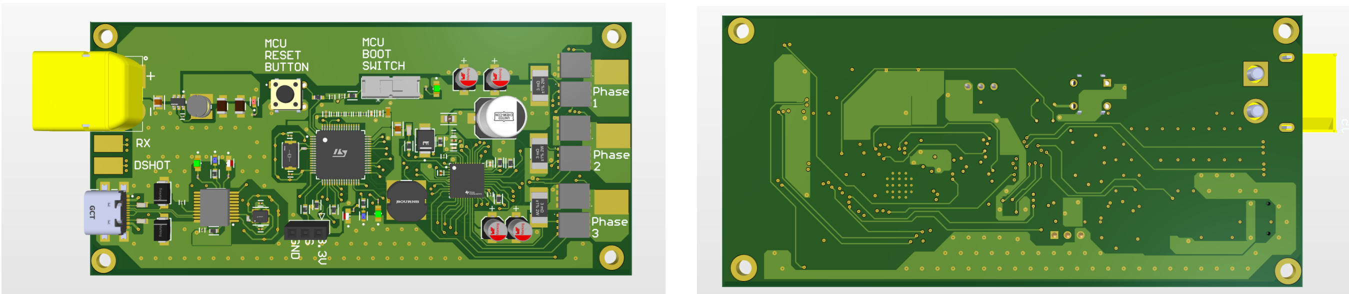

Figure 10: Discrete ESC PCB — Top and Bottom Layers

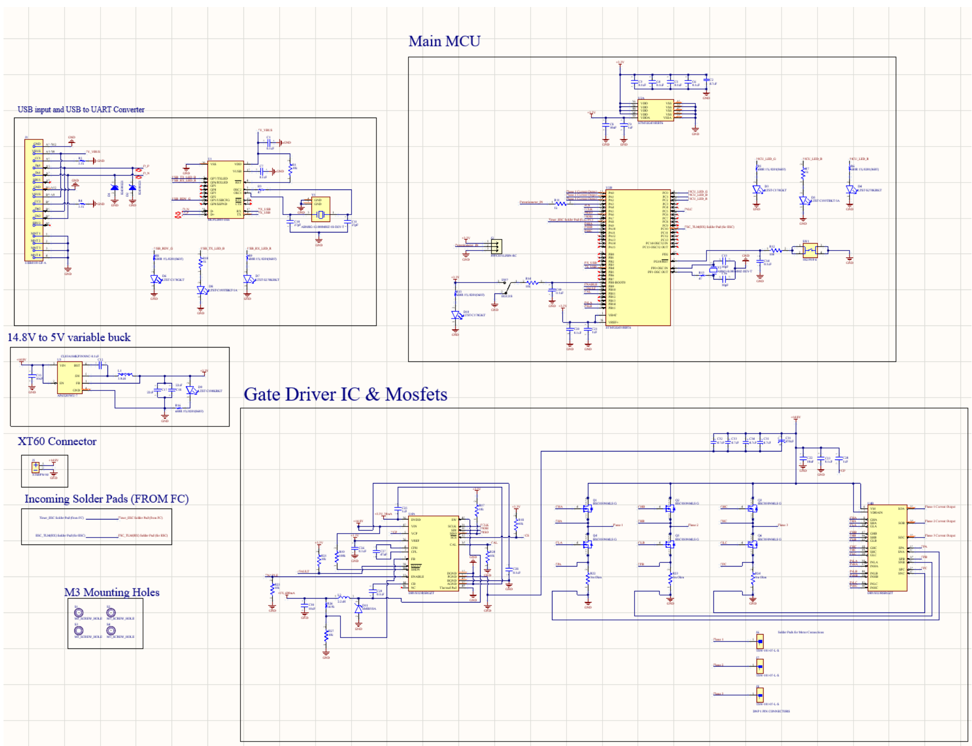

Figure 11: DRV8323RS Gate Driver and MOSFET Bridge Schematic

Key Considerations

- Routing strategy follows similar mixed-signal and power integrity principles as the flight controller, with additional focus on high-current switching loops and bulk decoupling placement near MOSFETs.
- Key design considerations include assuring that all solder pads are placed in a way that connections are convenient

Custom Controller (4 Layer Board)

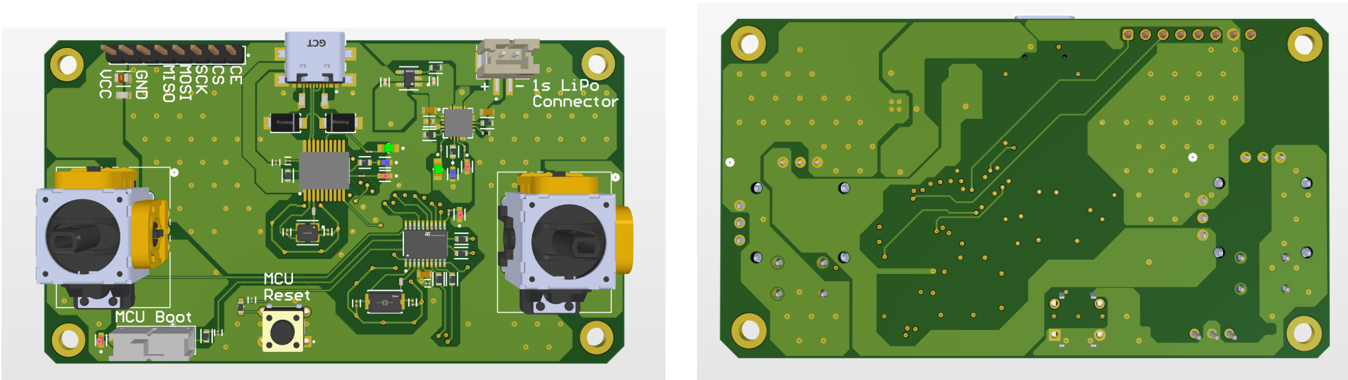

Figure 12: Custom Controller — Top and Bottom Layers

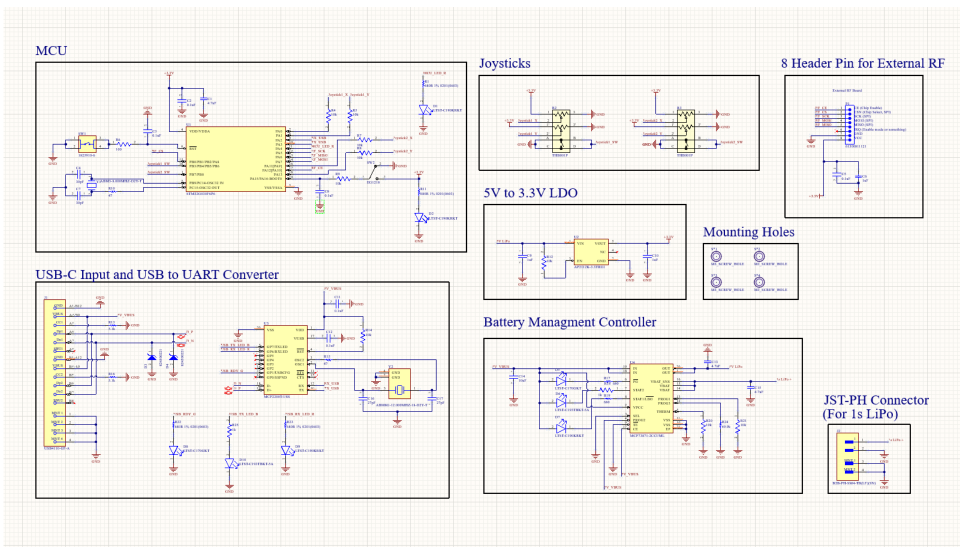

Figure 13: Custom Controller Schematic

Key Considerations

- Routing Considerations are essentially identical to what was covered earlier. However, I used inspiration from a standard PS4 controller for the distance between joysticks and location of USB Port

Currently Working On

- Hardware development
- Chassis Design and Iteration
- Controller Shell Design

Gigabit Ethernet Switch and Injector

Role: Solo Project **Date:** Febuary 2026
Tools: Altium Designer, LTspice

Overview

Designed a custom Gigabit Ethernet switch with one upstream port (Non PoE) and four downstream PoE+ ports compliant to IEEE 802.3 standards (Up to 30W per port), designed featuring a KSZ9567S Ethernet IC and TPS23882B PSE PoE controller. This design is a 6 Layer board to allow for much easier routing.

Main Components

- KSZ9567S Ethernet IC
- TPS23882B PSE PoE IC
- STM32G07 MCU
- 749022310 and 749023310 Transformers
- TVS Diodes for protection

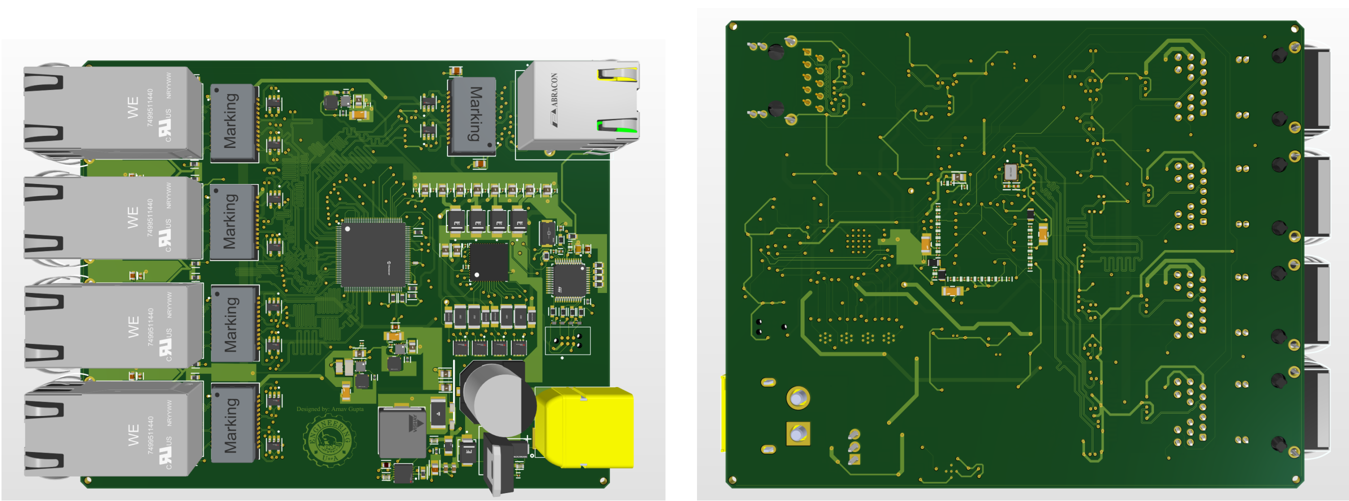

Figure 14: PoE+ Switch PCB — Top and Bottom Layers

Key Routing Considerations

- Large number of differential pair lines, for ethernet 100 ohm differential impedance is used to maintain signal integrity
- Multiple stages of signal "travel" (for differential pairs), from input to input transformer, input transformer to ethernet mcu, ethernet mcu to output transformers, output transformer to ethernet outputs. Where each of these stages of travel must be length tuned to be the same length of traces of other traces in the same stage (length within 5-10
- When making accordions for length tuning, higher amplitude accordions over amount of accordions is preferred due to impedance drop at each turn
- Made sure decoupling capacitors are close to pins
- Add stitching vias for low impedance (Implemented for necessary signals)
- Polygon pours for ground and voltage routing (or very thick traces)
- No copper pours (that are on same layer) near differntial pairs to maintain signal integrity

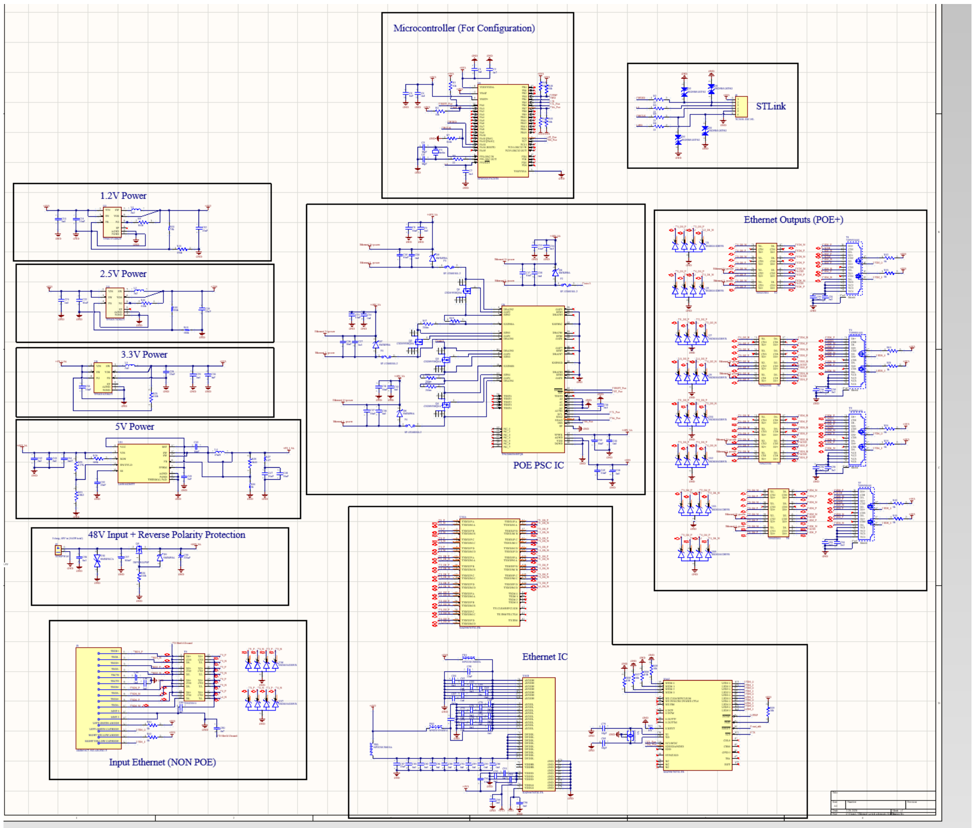

Figure 15: PoE+ Power Delivery and Ethernet Switch Schematic

Custom USB 3.0 Hub

Role: Solo Project **Date:** 2026
Tools: Altium Designer, LTspice

Overview

Designed a custom USB 3.0 hub PCB focusing on high-speed differential pair routing, impedance control, and robust power + ESD protection. Board features one USB-C 3.0 Upstream port, 2 USB-C B 3.0 Downstream ports, 2 USB-A 3.0 Downstream ports, 2 HDMI 1.4 Downstream ports. This design is a 6 Layer board to allow for much easier routing.

Main Components

- TUSB8041-Q1 Hub IC
- STDP4320BA Dp Alt Mode 1:2 Splitter
- TPS65987D USB-C PD Controller
- HD3SS460 Mux Controller
- 2x DP to HDMI Converters
- Multiple LDOs to supply power

Key Routing Considerations

- Large number of differential pair lines, for USB-C and USB-A 90 ohm differential impedance, 100 ohm differential impedance for HDMI and 50 ohm differential for other signals is used to maintain signal integrity
- Multiple stages of signal "travel" (for differential pairs), where each of these stages of travel must be length tuned to be the same length of traces of other traces in the same stage (length within 5-10
- When making accordions for length tuning, higher amplitude accordions over amount of accordions is preferred due to impedance drop at each turn
- Made sure decoupling capacitors are close to pins
- Add stitching vias for low impedance (Implemented for necessary signals)
- Polygon pours for ground and voltage routing (or very thick traces)
- No copper pours (that are on same layer) near differntial pairs to maintain signal integrity

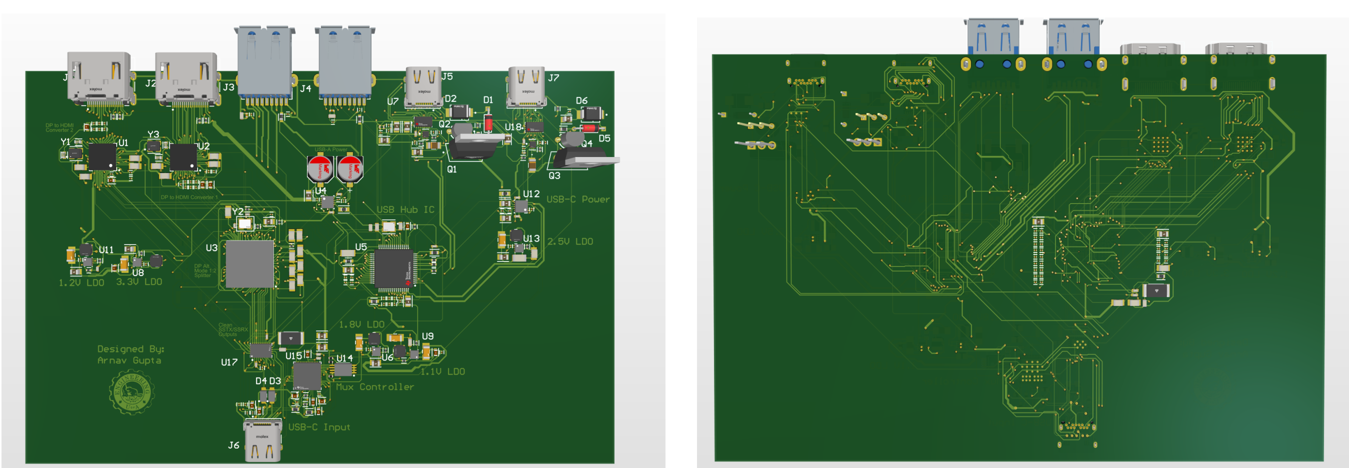

Figure 16: USB 3.0 Hub PCB — Top and Bottom Layers

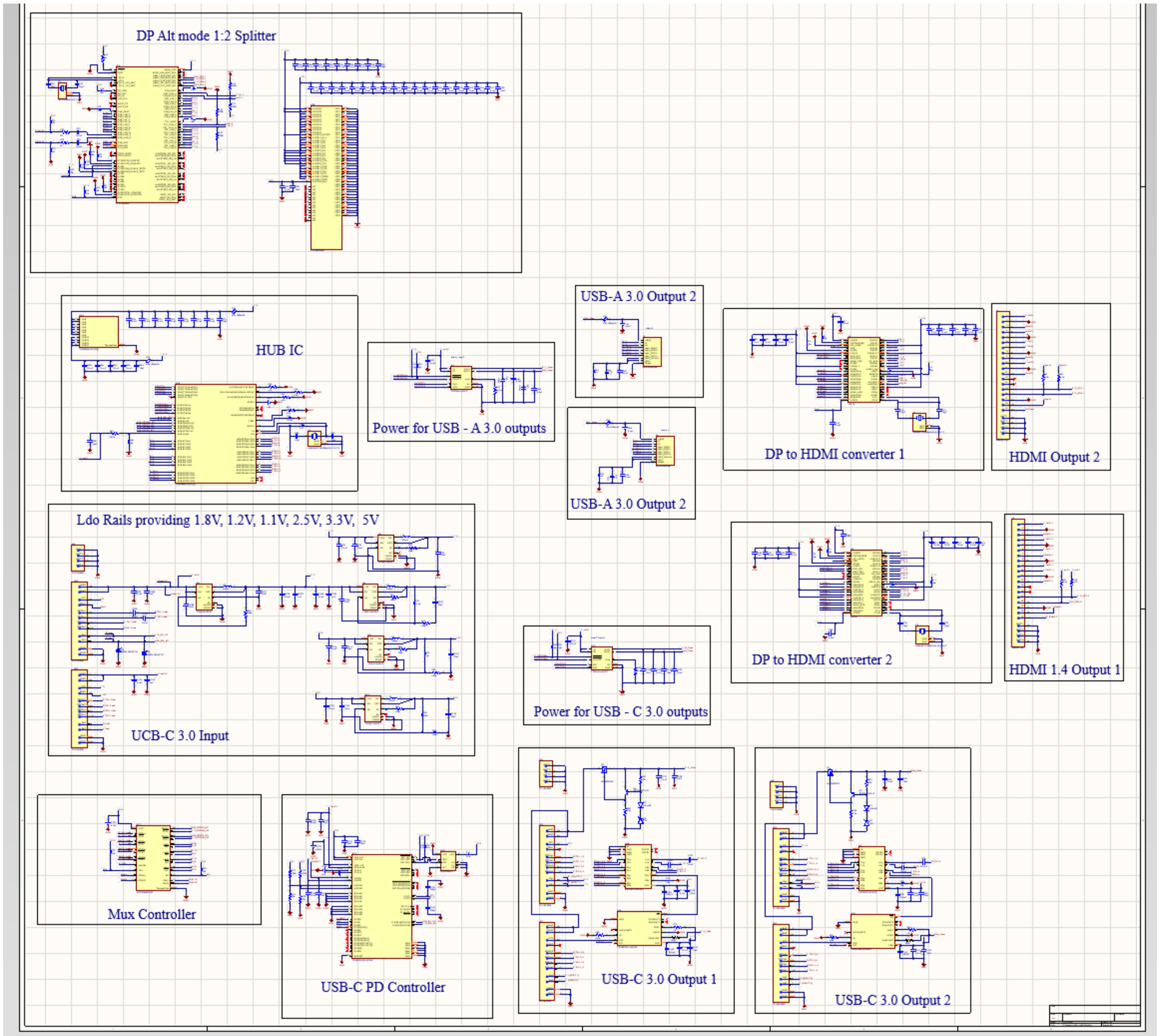

Figure 17: USB 3.0 Hub Schematic Overview