Hydra Framework Design Notebook

|  |  |
| --- | --- |
| Rocket: | Gee-Force One |
| File Name: | -Framework |
| Sub Team: | Avionics: Hardware |
| Point of Contact: | Serban, Liam, Noah, Amanda |
| Purpose: | Document Hydra-level decisions that affect all Hydra boards to develop a general framework for supplemental hardware. |

Revision History:

|  |  |  |
| --- | --- | --- |
| Revision | Description | Date |
| 0 | Baseline updated (Hydra V1) | Jan 18, 2023 |
| 1 | Updated design following first design review | Feb 11, 2023 |
| 2 | Included Hydra V3 documentation | Sep 25, 2023 |
| 3 | Added requirements for Hydra V3 satellite PCBs | Oct 14, 2023 |
| 4 | Added HYDRA\_V3\_BEACON | Oct 17, 2023 |
| 5 | Added HYDRA\_V3\_GUIDANCE | Nov 20, 2023 |

Table of Contents:

[1 Introduction 1](#_Toc179395931)

[2 Documentation 2](#_Toc179395932)

[2.1 Hydra V1 (ignore this part of the doc) 2](#_Toc179395933)

[2.1.1 System Behaviour 2](#_Toc179395934)

[2.1.1.1 Arming 2](#_Toc179395935)

[2.1.1.2 Recovery 2](#_Toc179395936)

[2.1.2 Hardware Design 3](#_Toc179395937)

[2.1.2.1 Board Layout 3](#_Toc179395938)

[2.1.2.2 Inter-board Connections 4](#_Toc179395939)

[2.2 Order Specifications and Recommendations 5](#_Toc179395940)

[2.2.1 JLC PCB 5](#_Toc179395941)

[2.2.2 Digikey 5](#_Toc179395942)

[2.2.3 Hydra V1 Boards 6](#_Toc179395943)

[2.2.3.1 Recovery Board 6](#_Toc179395944)

[2.2.3.2 Communication Board 6](#_Toc179395945)

[2.2.3.3 Logic Board 7](#_Toc179395946)

[2.2.3.4 Power Board 7](#_Toc179395947)

[2.2.4 Board Hardware Requirements 7](#_Toc179395948)

[2.2.4.1 Recovery Board 7](#_Toc179395949)

[2.2.4.2 Communication Board 7](#_Toc179395950)

[2.2.4.3 Logic Board 7](#_Toc179395951)

[2.2.4.4 Power Board 8](#_Toc179395952)

[2.3 Hydra V2 9](#_Toc179395953)

[2.3.1 System Behaviour 9](#_Toc179395954)

[2.3.1.1 Arming 9](#_Toc179395955)

[2.3.1.2 Recovery 9](#_Toc179395956)

[2.3.2 Hardware Design 10](#_Toc179395957)

[2.3.3 Hydra V2 Boards 11](#_Toc179395958)

[2.3.3.1 HYDRA\_V2\_POWER 11](#_Toc179395959)

[2.3.3.2 HYDRA\_V2\_LOGIC 12](#_Toc179395960)

[2.3.3.3 HYDRA\_V3\_COMMUNICATION 12](#_Toc179395961)

[2.3.3.4 HYDRA\_V2\_RECOVERY 12](#_Toc179395962)

[2.4 Hydra V3 13](#_Toc179395963)

[2.4.1 System Behaviour 13](#_Toc179395964)

[2.4.1.1 Arming 13](#_Toc179395965)

[2.4.1.2 Recovery 13](#_Toc179395966)

[2.4.2 Stackable Board Design 14](#_Toc179395967)

[2.4.2.1 General Outline 14](#_Toc179395968)

[2.4.2.2 Inter-PCB Connectors 15](#_Toc179395969)

[2.4.2.3 Wire-to-board Connectors 16](#_Toc179395970)

[2.4.2.4 Debugging Interface 16](#_Toc179395971)

[2.4.2.5 Status LED Locations 17](#_Toc179395972)

[2.4.2.6 Voltage Sensing Dividers 17](#_Toc179395973)

[2.4.2.7 Silk Mask Layout 18](#_Toc179395974)

[2.4.2.8 Component Identifiers 18](#_Toc179395975)

[2.4.2.9 Component Sizing 18](#_Toc179395976)

[2.4.2.10 Vertical Clearance 19](#_Toc179395977)

[2.4.2.11 Production Checklist 19](#_Toc179395978)

[2.4.3 Satellite Board Design 21](#_Toc179395979)

[2.4.3.1 General Outline 21](#_Toc179395980)

[2.4.3.2 Wire-to-board Connectors 22](#_Toc179395981)

[2.4.3.3 Debugging Interface 22](#_Toc179395982)

[2.4.3.4 Status LED Locations 22](#_Toc179395983)

[2.4.3.5 Silk Mask Layout 23](#_Toc179395984)

[2.4.3.6 Component Identifiers 23](#_Toc179395985)

[2.4.3.7 Component Sizing 23](#_Toc179395986)

[2.4.4 Hydra V3 Boards 24](#_Toc179395987)

[2.4.4.1 HYDRA\_V3\_POWER 24](#_Toc179395988)

[2.4.4.2 HYDRA\_V3\_SPACER 26](#_Toc179395989)

[2.4.4.3 HYDRA\_V3\_NAV 26](#_Toc179395990)

[2.4.4.4 HYDRA\_V3\_LINK 27](#_Toc179395991)

[2.4.4.5 HYDRA\_V3\_RECOVERY 27](#_Toc179395992)

[2.4.4.6 HYDRA\_V3\_BEACON 28](#_Toc179395993)

[2.4.4.7 HYDRA\_V3\_GUIDANCE 29](#_Toc179395994)

[2.4.4.8 HYDRA\_V3\_CAMERA 30](#_Toc179395995)

[2.4.4.9 HYDRA\_V3\_SENSOR 31](#_Toc179395996)

[2.4.4.10 HYDRA\_V3\_ACTUATOR 33](#_Toc179395997)

[3 Software Design 34](#_Toc179395998)

[4 EMI/RF Considerations 35](#_Toc179395999)

[5 Arming and Safety Considerations 35](#_Toc179396000)

[6 Battery and SBG Considerations 36](#_Toc179396001)

# Introduction

This document contains an overview of the macroscopic design requirements and considerations for the first version of uORocketry’s new avionics system, Hydra. The goal of this system is to develop a modular avionics stack in which boards can be stacked in no particular orders and swapped/upgraded over time.

# Documentation

## Hydra V1 (ignore this part of the doc)

### System Behaviour

#### Arming

The power board includes a switch that is actuated by a rod protruding from the rocket and serves to disable power delivery to the avionics stack. No further arming procedures are included since no custom hardware is in control of any controllable mechanisms/elements.

The recovery electronics are controlled by two redundant COTS RRC3 boards with individual batteries which have a similar safety mechanism. Two switches are included on the ejection board, one for each RRC3 battery, to disable power flow to the recovery systems.

#### Recovery

The recovery portion of the first version of Hydra is composed of two RRC3s with separate power sources that control the dual event recovery. The presence of the second RRC3 is due to the competition redundancy requirement. The two altimeters function completely independent from one another, and in case of failure of the primary system, the backup system guarantees a safe recovery of the rocket.

### Hardware Design

#### Board Layout

All Hydra boards intended for mounting within the avionics bay will follow a circular shape, 140mm in diameter, with three 5.3mm mounting holes spread equally along a pattern of radius of 60mm. The mounting holes are specifically located at (0, 2362.2mil), (-2045.729mil, -1181.1mil) and (2045.729mil, -1181.1mil) on the circuit board. Each stackable board in the avionics bay has two 2x15-pin headers mounted with the following Altium settings, assuming the center is at the true center of the board (Figure 2). The right header uses the same values however the X value is positive (+50mm). IMPORTANT: the location is set by selecting the header component itself, not a pin within the header.

Graphical user interface, text, application, chat or text message

Description automatically generated

Figure 1. Left header Location in Altium

Finally, a pass through cutout is included at the rear section of the board as a 15mm circular cutout, tangent to the board’s outline, and along the vertical axis in Figure 3.

Graphical user interface

Description automatically generated

Figure 2. Hydra Board General Dimensions (all in mm)

#### Inter-board Connections

The goal of Hydra is to improve the avionics stack’s malleability and modularity thereby simplifying the process of upgrading the stack over time. As such, all Hydra boards intended to be placed in the avionics bay will have an identical connector pattern with identical pinouts such that various combinations of boards can be stacked in any order. All Hydra boards will have two sets of 2x15-pin 2.54mm pitch headers on opposing sides of the circular PCB with standardized pinouts across all boards as tabulated below:

|  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| |  |  |  | | --- | --- | --- | | POWER | V\_BATT | V\_BATT | | V\_BATT | V\_BATT | | +5V | +5V | | +5V | +5V | | +3V3 | +3V3 | | +3V3 | +3V3 | | I2C | GND | GND | | SDA | SCL | | SPI | GND | GND | | MOSI | MISO | | SCK | CS | | UART | GND | GND | | UART\_TX | UART\_RX | | CAN | GND | GND | | CAN\_HI | CAN\_LO | | |  |  | | --- | --- | | PYRO\_PWR | PYRO\_PWR | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | |  |  | | GND | GND | |

The left header transmits all power and data signals for the avionics stack whereas the right header is mostly free for future upgradeability. The data buses on the left header are segregated by ground pins to prevent crosstalk due to the proximity of these data lines.

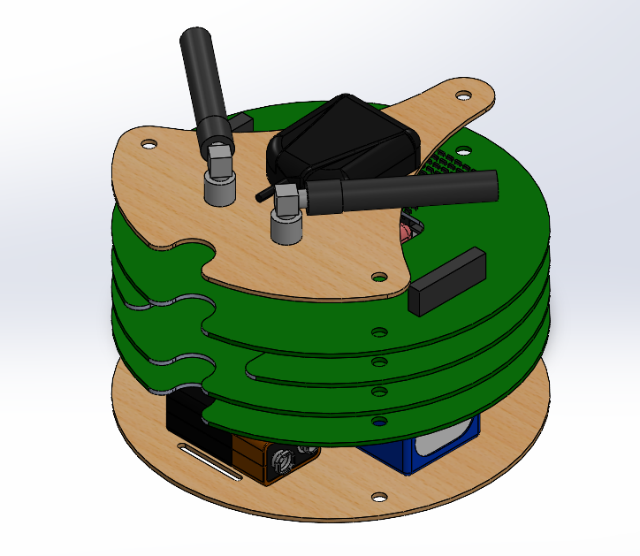
The power, logic, and communication boards stack closely in this design to enable the use of inter-board connectors and minimize free wires within the avionics bay. No additional inter-board connections should be required seeing as the ejection board runs independently of Hydra and utilizes separate power supplies. The only clearance issue lies with the SBG as its 24mm height will require a hole in the communication board to maintain the close stacking of the circuit boards.

## Order Specifications and Recommendations

### JLC PCB

### Digikey

### Hydra V1 Boards

A picture containing diagram

Description automatically generated

Figure 3. Hydra V1 Preliminary Render

The flight computer is currently split into four circuit boards each having a specific role, as depicted in Figure 4 above. The two beige plates above and below the stack only serve structural purposes and are not part of the computer itself. The top beige plate serves as a mounting point for the GPS antenna and the RFD900’s antennas whereas the bottom beige board serves as a mounting point for the lithium polymer battery as well as the two 9V batteries used by the RRC3s. The material for the beige plates is not determined, however we will likely use excess unpopulated PCBs since we must order a minimum of 5 for each PCB of this flight computer. The functional flight computer PCBs, depicted in green, from bottom to top are as follows: the power board, the logic board, the communication board, and the recovery board. Their specific functionality is listed below:

#### Recovery Board

Hydra V1 does not contain any flight-critical functionality. Therefore, this board functions independently and serves solely to house two RRC3s alongside individual power sources (9V batteries) for each and the SBG’s GPS antenna. The Rocktopus airframe stores the main chute in the nose cone of the rocket (immediately above the avionics bay) and the drogue chute immediately below the avionics bay. This means that two separate sets of wires are required on either end of the avionics bay to interact with both recovery systems. Furthermore, each recovery device is actuated by a set of redundant CD3s (CO2-based deployment charges) so four independent sets of wires are required (8 total).

Beyond providing mounting locations for the components this board serves to replace the wire-to-board connectors on the RRC3s with more reliable connectors. This eliminates the risk of electrical failure due to vibration while maintaining some level of maintainability in that the avionics stack can be conveniently connected and disconnected from the airframe’s hardware.

#### Communication Board

The Hydra V1 communication board serves to provide a mounting point for the RFD900 radio as well as the Featherlight backup GPS tracker. The RFD900 radio receives power and UART from the header with no additional hardware whereas the featherlight functions independently from the avionics stack. As a result, the featherlight uses a separate LiPo battery and does not communicate with the avionics stack.

#### Logic Board

Hydra V1 contains no flight-critical logic and therefore this board serves solely as a telemetry gathering system to log rocket attitude/location data from the SBG to an SD card and transmit it using the RFD900 radio. This is the only board containing an MCU in this version of Hydra. Data is gathered from the SBG using a CMOS-TTL to RS-232 converter linked to a UART port of the MCU and data is logged onto an SD card using an SPI port of the MCU.

In addition to the electrical hardware, this board provides a mounting point for the SBG that must be at its center to ensure accurate attitude data is collected.

#### Power Board

This board contains two switching regulators providing the other boards with a 5V and 3.3V rail from the integrated avionics battery. Current version includes reverse polarity protection as well as thermal and overcurrent protection.

### Board Hardware Requirements

#### Recovery Board

Used for redundant two-stage recovery system deployment and GPS antenna mounting.

* 2 RRC3 footprints to transfer I/O from RRC3 wire-to-board connectors to locking connectors on the recovery board
* Safety disconnect switches
* Cutout in center to provide clearance for SBG
* Passthrough for RF cables throughout stack
* Two 2x15-pin headers

#### Communication Board

Used to mount RFD900 and Featherlight GPS tracker.

* RFD900 footprint
  + UART and power traces
* Featherlight tracker footprint
  + Independent battery so no traces needed whatsoever
* Passthrough for RF cables throughout stack
* Two 2x15-pin headers

#### Logic Board

Used for data logging and ground station communication.

* Micro SD card holder
* 1 external crystal 32KHz
* 1 external crystal 8MHz
* Microcontroller ATSAME51J18A
* SBG mounting point
* UART link to SBG
  + CMOS-TTL to UART adapter
* LEDs:
  + Green: connected to power
  + Red: connected to chip, error
  + Yellow: connected to chip, communication
* SWD debugging port (<https://www.segger.com/products/debug-probes/j-link/models/j-link-edu-mini/>)
* Two 2x15-pin headers
  + UART link for RFD900
  + CAN link for future avionics boards
  + I2C link for future avionics boards
  + SPI link for future avionics boards?

#### Power Board

Used to step down battery voltage to 5V and 3.3V rails.

* 5V regulator
* 3.3V regulator
* LEDs to indicate operational state of each power rail
* Battery connector
* Safety power-disconnect switch
* Two 2x15-pin headers

## Hydra V2

### System Behaviour

#### Arming

The arming system is identical to Hydra V1 (see Section 2.1.1.1) and uses two arming rods: one for power and another for pyrotechnics. The power safety interacts with the power board and mechanically disconnects battery power from the avionics system whereas the pyrotechnics safety rod interacts with the recovery board and mechanically disconnects power from the pyrotechnics channels.

#### Recovery

Hydra V2 uses SRAD and COTS recovery systems in parallel for redundancy. The SRAD system is in the form of the recovery board which also houses a RRC3 to use as its COTS backup.

### Hardware Design

Hardware design is highly similar to V1 and was not documented since so few people worked on the boards that there was no need.

### Hydra V2 Boards

A green and black device

Description automatically generatedA picture containing diagram

Description automatically generated

Figure 4. Hydra V1 Preliminary Render

The flight computer is currently split into four circuit boards each having a specific role, as depicted in Figure 4 above. The two beige plates above and below the stack only serve structural purposes and are not part of the computer itself. The top beige plate serves as a mounting point for the GPS antenna and the RFD900’s antennas whereas the bottom beige board serves as a mounting point for the lithium polymer battery as well as the two 9V batteries used by the RRC3s. The functional flight computer PCBs, depicted in green, from bottom to top are as follows: the power board, the logic board, the communication board, and the recovery board. Their specific functionality is listed below:

#### HYDRA\_V2\_POWER

The V2 power board uses identical regulators to the V1 power board albeit in parallel for redundancy. Each power rail (5V and 3V3) has two regulators whose outputs go through a load-sharing circuit based on the LTC4353IDE#PBF load-sharing controller. This ensures we have 5A of redundant current delivery per rail with a possibility of peaking up to 10A.

From a macroscopic perspective, the power is received through a Molex connector (J1) and passed through two cutoff switches before entering the global power protection circuit. This circuit handles reverse polarity protection as well as overcurrent and overvoltage protection (in this order). Finally, the power protection circuit provides the MCU with input voltage and current data. Following the protection circuit, power passes through two similar regulator circuits which power the 5V and 3V3 rails of the avionics stack. Both circuits consist of two identical regulators who largely share input and output capacitors and are combined using a load-sharing circuit. Finally, an MCU is included to manage the board and communicate with the remainder of the avionics. The MCU has its own internal regulators such that it remains online even in the case of a power failure on the main hydra power rails.

#### HYDRA\_V2\_LOGIC

This board is relatively identical to the V1 logic board with the exception of some added voltage sensing, a new MOSFET switch to turn the SBG on/off, and a secondary CAN network which enables us to interface with CAN devices without exposing the flight-critical avionics CAN network. Its purpose is to interface with the SBG, log data from the SBG to an onboard SD card and publish navigational data to the avionics CAN bus. This board is also intended for future use in a hybrid vehicle where it could interact with satellite PCBs throughout the vehicle to receive sensor data or control actuators.

#### HYDRA\_V3\_COMMUNICATION

The purpose of this board is to act as a black box with respect to all content going across the avionics CAN network, and to provide a radio link between Hydra and the ground station. Additionally, this version of the communication board contains a set of internal regulators which it can use in the event of a power failure on the main Hydra power buses. At that point, the communication board can revert to using pyrotechnics power and its own internal regulators which ensures that we can maintain a telemetry link with the avionics in the event of an accidental short circuit or other non-critical power failure.

This board is named the V3 power board since it integrates a blackbox and the internal regulator feature which was provisioned for V3 and not V2.

#### HYDRA\_V2\_RECOVERY

The Hydra V2 recovery board contains SRAD ejection circuitry in parallel with an RRC3 recovery device for redundancy. This version of the board uses a SAME51 MCU though this will be updated in the future since functions that rely on the ADC are unusable due to an issue with the chip’s clocks on the software side. The SRAD circuitry also contains a barometer and IMU that can be utilized in the event that no navigational data is available on the avionics CAN bus from the SBG.

## Hydra V3

Hydra V3 is the expansion of Hydra V2 that allows the avionics system to completely integrate with a hybrid rocket engine. Both the V2 and V3 systems are capable of this though hybrid hardware was not included in V2 seeing as the purpose of V2 was to test the avionics stack’s functionality.

Hydra V3 consists of revised and miniaturized Hydra V2 boards as well as two additional non-stackable ‘satellite’ boards: one for sensors external to the avionics stack and another for actuators external to the avionics stack. The stackable boards have many design constraints all of which are detailed in Section 2.2.2. Satellite boards have far less design constraints and their standard can be found in Section 2.2.3.

### System Behaviour

#### Arming

Hydra V3 uses a MOSFET-based switch on the power board which is actuated by some sort of external source such as a key switch. This is due to problems we faced with the arming rods and paddle switches used by Hydra V1 and V2. The arming rods failed to reliably interact with the switches which allowed devices to be powered with the rods inserted. Additionally, we believe the switches to have actuated due to vibrations in the airframe thereby resetting the stack mid-flight.

The MOSFET-based switch consists of two back-to-back MOSFET assemblies in parallel for redundancy and lowered resistance. These MOSFETs circuits are driven directly from battery power by an external switch and contain as little circuitry as possible to ensure reliability. This circuit is entirely analog and cannot be influenced in any way by any programmable hardware.

#### Recovery

Unlike Hydra V1 and V2, this system is too small to house an RRC3 recovery device and cannot therefore use our current COTS-based redundant recovery system. For this reason, Hydra V3 can only be redundant with the inclusion of two recovery boards within the stack running individual recovery algorithms and wired to independent initiators within the parachute deployment mechanisms. A redundant COTS solution can be included external to the stack itself if needed.

### Stackable Board Design

#### General Outline

The Hydra V3 ecosystem maintains the physiology of V1/V2 though at a much smaller scale such that it can fit smaller airframes. The wire cutout at the rear of the PCBs is also maintained though it is no longer circular to enable board-to-wire connectors to sit flush with the backside of the PCB making it easy to mate/unmate connectors without disassembling the stack. A sketch of the updated PCB layout is included below. Fundamentally, the system now has boards which are 50mm in diameter (roughly 2 inches), have a similar set of stacking connectors on all PCBs, and have the aft end of the PCB flattened for board-to-wire connectors. The boards are rated as Aerospace type with thickness of 1.6mm.

A blue circle with a blue line drawing

Description automatically generated with medium confidence

Figure 5. Hydra V3 Stackable PCB Layout

No mounting provisions are made for Hydra V3 to maximize usable board space for electronics. Instead, the system will be held in place using braces at the top and bottom of the stack which fix it to the airframe.

#### Inter-PCB Connectors

Hydra V3 uses Samtec TigerEye locking connectors between the PCBs (see part numbers below). The locking behaviour is important since Hydra V3 does not include any mounting provisions on the PCBs and therefore relies partly on the connectors to hold itself together. The set of connectors listed below mates together and provides 40 pins of combined I/O between all PCBs. Each pin is rated to 3.2A and the connectors themselves are rated for various data transfer protocols all of which far exceed our needs.

1. SFML-110-02-L-D-A
2. TFML-110-02-L-D-A

The SFML connector is the female variant and is placed on the bottom size of the PCB. Conversely, the TFML connector is the male variant and is placed on the top side of the PCB. This choice was made to mitigate the risk of short circuits by not having exposed pins on the bottom end of the stack which will be placed on various surfaces during testing. The left baset of headers is placed at (X: -20mm, Y: 0mm) whereas the right set of headers is placed at (X: 20mm, Y: 0mm). Both these dimensions are given with the center of the PCB’s radius (not the center of its area since it’s not a circle) as the reference (X: 0mm, Y: 0mm) and the connector footprints should have centers which perfectly align the pads. **A 0.7mm/0.5mm (via diameter/hole diameter) via must be included on each connector pad.** These connect the pads from the top and bottom connectors.

The updated connector pinout is as follows:

|  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| |  |  |  | | --- | --- | --- | | POWER | V\_BATT | V\_BATT | | +5V | +5V | | +5V | +5V | | GND | GND | | +3V3 | +3V3 | | +3V3 | +3V3 | | CAN | GND | GND | | CAN0\_HI | CAN0\_LO | | GND | GND | | CAN1\_HI | CAN1\_LO | | |  |  | | --- | --- | | PYRO\_PWR | PYRO\_PWR | | PYRO\_PWR | PYRO\_PWR | | PYRO\_PWR | PYRO\_PWR | | PYRO\_PWR | PYRO\_PWR | | GND | GND | | GND | GND | | GND | GND | | GND | GND | | GND | GND | | GND | GND | |
|  |  |

The left table represents the pinout for the left Hydra V3 connector (based on a top-down view of the PCB) where V\_BATT is the top left pin of the connector. **This is important because Altium inserts the connectors upside down and they must be rotated 180 degrees.** Likewise, the right table represents the pinout for the right Hydra V3 connector where the PYRO\_PWR pins are on the top half of the connector. Eight PYRO\_PWR pins are included to allow for enough ground pins such that there is a ground pin for every power pin (regulated or not).

Only two V\_BATT pins are included since V\_BATT was seldom used by Hydra V2 boards though it may be useful in the future and should therefore not be fully omitted. Hydra V2 boards sometimes tapped into the PYRO\_PWR rail for things like the logic board’s buzzer but **this should no longer be a thing**. The PYRO\_PWR rail should only be used by power boards, pyrotechnics channels, and the communication board to ensure that it remains a highly reliable power rail for the entire system.

The regulated 5V and 3V3 rails of the avionics system maintain 4 power pins each, providing each rail with a theoretical 12.8A current capacity (based on 3.2A/pin). The left header of the avionics stack also contains two separate CAN buses. This is for two reasons: firstly, the software team had issues sending commands over CAN as those packets would get drowned in sensor data and would not be received by target boards. Furthermore, this provides redundant CAN buses allowing Hydra V3 to persist in the case that one CAN bus fails. Of the two new CAN buses, one is reserved for sending time-critical command data and the other is reserved for sending non-time-critical sensor data.

#### Wire-to-board Connectors

Hydra V3 uses Molex PicoBlade connectors between its PCBs and external devices such as batteries and the SBG. These connectors are used for their small size, locking ability, and high current capacity. All connectors to external I/O will be located at the back end of the PCB (flat part of sketch in Section 3.2.1) and **must be labelled on the opposite silk mask**. The individual pinouts can be omitted from the silk mask on a case-by-case basis, but the general function of the connector should be clearly visible without needing to disassemble the stack. All wire-to-board connectors located at the rear of Hydra V3 PCBs should be located at a Y value of -16.25mm. Therefore:

* + - Wire-to-board connectors (general) à (X: varies, Y: -16.25mm)

#### Debugging Interface

The Samtec TigerEye connectors severely limit the vertical clearance available between Hydra V3 PCBs. This, in combination with the small size of the PCBs prevents the use of a dedicated full-scale J-Link connector. Instead, the Hydra V3 PCBs use a 6-pin Molex PicoBlade connector (part # 0532610671) for debugging which is located on the top layer of the PCB as follows:

* + - Debug connector à (X: 0.0mm, Y: 19.0mm)
      * Label 1 à (X: -7.0mm, Y: 21.0mm)
      * Label 2 à (X: 7.0mm, Y: 23.299mm)
    - Reset switch à (X: 10.0mm, Y: 18.7mm)
      * Label à (X: 8.9mm, Y: 20.8mm)

The pinout for the debugging interface (based on the Molex 0532610671 part available in our Altium cloud) is as follows:

1. SWO
2. SWDIO
3. SWCLK
4. RESET
5. VREF
6. GND

And both shield pins of the connector are also grounded.

#### Status LED Locations

Hydra V3 implements a standardized LED layout which enables all LEDs to be aligned across the entire stack. The V3 PCB layout provides space for 4 bicolor LEDs to provide status information about the board. From top to bottom the LEDs are as follows: power status, MCU status, custom status 1, custom status 2. The latter two LEDs are board-specific and do not need to be included on all PCBs. The LEDs are the Wurth Electronics 150066RG54050 and their locations are standardized as follows:

* + - Power status LED à X:13.6mm, Y:18.4mm
      * Label à X: 11.5mm, Y: 19.3mm
    - MCU status LED à X:16.5mm, Y:15.5mm
      * Label à X: 15.0mm, Y: 16.4mm
    - Custom status 1 LED à X: 19.0mm, Y:12.5mm
      * Label à X: varies, Y: 13.4mm
    - Custom status 2 LED à X: 20.5mm, Y: 9.5mm
      * Label à X: varies, Y: 10.4mm

All LEDs **must be labelled** as per the label locations provided in the list above. The X coordinate of the label may be adjusted depending on the length of the label’s text though the text must be kept as close as possible to the PCB’s edge. The dimensions above are provided with the center of the Hydra V3 PCB as the origin (X: 0mm, Y: 0mm).

#### Voltage Sensing Dividers

All Hydra V3 boards must have feedback for every voltage rail, internal and external, that they may interface with. It would also be ideal to have feedback for rails that the board does not interface with though this is not a requirement. The 5V and 3V3 dividers must consist of two 100k resistors (Part # ERJ-3EKF1003V) whereas the VCC and PYRO\_PWR dividers must consist of 100k and 10k resistors (Part # ERJ-3EKF100**3**V and ERJ-3EKF100**2**V). This change was made to eliminate the 3V3 Zener used to protect the MCU in older VCC/PYRO\_PWR voltage sense dividers and to increase the overall resistance size considering the amount of resistors used throughout HYDRA. In summary:

* + - 5V and 3V3 dividers à 100k/100k
      * Part # ERJ-3EKF1003V
    - VCC and PYRO\_PWR dividers à 100k/10k
      * Part # ERJ-3EKF1003V and ERJ-3EKF1002V

#### Silk Mask Layout

All Hydra V3 boards must contain the name of the board (HYDRA\_V3\_NAV, for example), the uORocketry logo, and the name of whoever designed the board. The uORocketry logo is standardized across all PCBs and is located at (X: 19.5mm, Y: -9mm. Furthermore, the PCB’s name should be located at (X: 0mm, Y: -12mm) whereas the designer’s name should be located at (X: 0mm, Y: -13mm). All the dimensions above are provided with the center of the PCB’s curvature being the origin (X: 0mm, Y: 0mm)

All text on Hydra V3 PCBs, with the exception of the designer’s name, has a text height of 0.75mm and a stroke width of 0.15mm. The designer’s name has a text height of 0.5mm and a stroke width of 0.1mm. To summarize:

* + - **Must include:**
      * uORocketry logo at (X: 19.5mm, Y: -9mm)
      * Board name at (X: 0mm, Y: -12mm)
      * Designer’s name at (X: 0mm, Y: -13mm)
    - All text except designer’s name:
      * Text height à 0.75mm
      * Stroke width à 0.15mm
    - Designer’s name text:
      * Text height à 0.5mm
      * Stroke width à 0.1mm
    - Test point label text
      * Text height à 0.25mm
      * Stroke width à 0.05mm

#### Component Identifiers

Individual component identifiers may be excluded from Hydra V3 boards as they were in Hydra V2 boards. This is to maintain clean-looking high density PCBs and should not be a concern seeing as Hydra V3 PCBs are solely for internal use by people with Altium access.

#### Component Sizing

Hydra V2 followed strict guidelines with regards to component sizing in which all resistors were 0603 and all capacitors were 0805. Due to the size limitations of Hydra V3 and the general pain that is soldering 0402 components, Hydra V3 will use 0603 for all resistors and capacitors unless PCB constraints require the components to be smaller. Decoupling caps will remain 0402s because they are universally used by all ICs on Hydra PCBs and are therefore used in large numbers on all boards. To summarize:

* + - Resistors and capacitors à 0603 unless larger size is required.
    - Decoupling capacitors à 0402

#### Vertical Clearance

The Samtec TigerEye connectors provide 5-7mm of clearance between two mated PCBs which is assumed to be exactly 5mm for design purposes. Therefore, no components with a height greater than 5mm can be used on Hydra V3 PCBs. Additionally, some high-density PCBs may benefit from locating components on the bottom side of the board; this should generally be avoided as it may prevent the system from truly being stackable in any order/combination thereby compromising Hydra’s design philosophy. Some exceptions may be made to this rule in which case no component with a height greater than 1.5mm (solder included) should be included on the bottom side of the PCB.

#### Production Checklist

The purpose of this section is to specify a checklist to test whether a finalized stackable Hydra V3 board adheres to the system guidelines and is manufacturable/wired correctly. All boards should strictly comply with the conditions listed below unless deviation is strictly necessary (such as vertical clearance of the power board, where the inductors are 1.5mm too tall but smaller inductors would cause thermal issues).

1. ELECTRICAL CHECKLIST
   1. Hydra headers in correct position/orientation with correct pinout
   2. Ensure adequate via stitching
   3. Peripheral headers positioned, labelled, and pinned out reasonably
   4. Double check all power protection circuitry
      1. Fuse sizes
      2. Resettable OC protection latch trigger currents (check component values to ensure proper trigger current)
   5. Check MCU pinout
      1. All serial interfaces have correct pinouts and IOSETs
         1. CAN
         2. SPI
         3. I2C
         4. UART/USART
      2. Software agrees with serial interface pinouts
      3. Analog signals are connected to ADC pins
      4. Programming interface wired and pinned correctly
         1. Connector pinned correctly
         2. MCU programming pins wired correctly
      5. Proper components are standardized across all boards (should add a DN section for this)
         1. Voltage sense dividers identical across all boards
         2. CAN interfaces identical across all boards
         3. Reset SW interface identical across all boards
   6. Check impedance of differential pairs where relevant
      1. Altium automatically manages impedance if you ask it nicely
      2. Ensure proper grounding/shielding/distance around differential pair
   7. Check impedance and routing of any sensitive traces (RF or high precision ADC)
      1. Ensure proper shielding where needed
      2. Ensure enhanced via stitching where necessary
      3. Ensure no unnecessary copper pours
   8. Check routing of high-speed interfaces (parallel buses, etc)
      1. Ensure no problematic/unnecessary copper pours
      2. Ensure adequate length matching where necessary
      3. Ensure adequate shielding where necessary
2. THERMAL CHECKLIST
   1. What components generate heat and where are they?
   2. Ensure adequate number of vias included to transfer heat to PCB
   3. Heat sink/active cooling?
   4. Run thermal analyses where necessary
      1. Convection analysis of AV bay to predict time before overheating
3. COMPLIANCE CHECKLIST
   1. Meets all guidelines stated in this section, most importantly:
      1. Components on top layer do not exceed 3.5mm in height
      2. Components on bottom layer do not exceed 1.5mm in height
      3. All LEDs, connectors, and text is placed as per the guidelines in this section
      4. Correct components are used for common features such as debug and I/O connectors
      5. Component sizing adheres to standards defined in this section unless deviation is necessary
4. MANUFACTURABILITY CHECKLIST
   1. Ensure traces and vias are within JLC specs
      1. Simple Altium rule check should be good
   2. Ensure adequate spacing between QFN and QFN-similar packages
   3. Verify no component interference (resistors under larger IC)

### Satellite Board Design

A general guide is provided for satellite boards below though **it is not imperative that all PCBs adhere to these guidelines**.

#### General Outline

The general outline of satellite boards will vary based on the purpose and intended location of the satellite board. Generally, these boards should be as small as possible while being as convenient to mount as possible. Some boards may evolve to have eccentric sizes but the V3 satellite boards will target a Raspberry Pi Zero-like format. This entails a generally rectangular shape with screw holes and fillets at every corner. A sample diagram is provided below:

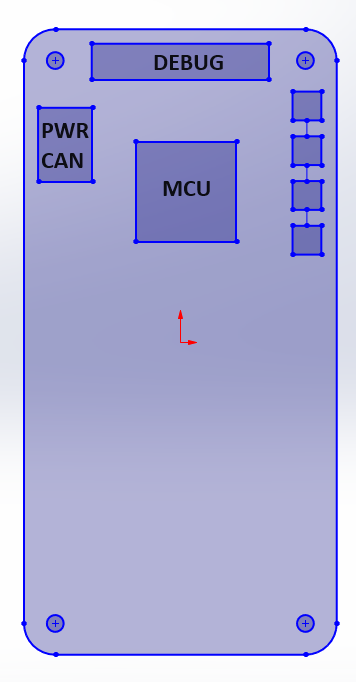


Figure 6. Hydra V3 Satellite PCB Sample Sketch

As previously mentioned, there are no set dimensions for this board, the sketch above simply serves to provide a general idea of where components should be. The debugging connector should be at the top of the PCB (the wider rectangle in the figure above) with the MCU directly under it. The four squares on the top right of the PCB outline represent status LEDs. There are 4 LEDs on this sample sketch but only two are required: one for the MCU status and another for power status. These are included in the MCU pinout that is common to all Hydra V3 PCBs. The power status LED should be the top LED with the MCU status LED directly under it.

The connector on the top left of the PCB represents the power/CAN input of the board; More information about this connector can be found in Section 2.3.3.2. Beyond this, general I/O connectors may be placed at the designer’s discretion though it would be ideal to have all I/O connectors on the left side of the PCB.

**The following is my personal opinion (Serban):**

For the sensor board, I’d likely keep all the I/O on the left but place the connectors closer to the bottom left corner of the board such that there’s a clear separation between the power/CAN input and general I/O connectors. For the actuator board, I’d likely place the motor/servo outputs on the bottom edge of the PCB and only have feedback inputs on the left side of the PCB (potentiometer/limit switches). The feedback inputs would ideally be placed close to the bottom left corner, but it may be difficult to do so in which case I’d place them halfway up the left side or something along those lines.

#### Wire-to-board Connectors

As with the stackable board design, the satellite boards will use Molex PicoBlade connectors to interface with hardware external to the PCB itself. Placement of these connectors is up to the designer and a general guideline is provided at the end of Section 2.3.3.1. **It is critical that all connectors, aside from the power input, be clearly labelled.** The individual pinouts should also be included on the silk mask and the general function of the connector should be clearly noted on the bottom silk mask, directly below whatever connector is being labelled.

The power/CAN input connector is a Molex 0430450606 due to its higher current capacity and the fact that we’ve already flown these successfully. This 6pin connector is an upgrade from the 4pin connector used by the old power/recovery boards so we can maintain 2 power and 2 ground pins in addition to the CAN\_HI and CAN\_LO lines. The connector pinout, as per the Altium component (in the Altium cloud) is as follows:

A diagram of a circuit board

Description automatically generated

1. CAN\_LO
2. GND
3. VCC
4. CAN\_HI
5. GND
6. VCC

#### Debugging Interface

Hydra V3 satellite boards maintain an identical debugging interface to the stackable boards though its placement may vary per board. Refer to Section 2.3.2.4 for connector pinout and part number. Suggested placement is detailed in Section 2.3.3.1.

#### Status LED Locations

Suggested placement for status LEDs is found in Section 2.3.3.1 but they are not strictly standardized like those of the stackable Hydra V3 PCBs. Only two LEDs are required: power status and MCU status. The power status LED should be the top-most LED with the MCU status LED directly beneath it.

#### Silk Mask Layout

All Hydra V3 boards must contain the name of the board (HYDRA\_V3\_SENSOR, for example), the uORocketry logo, and the name of whoever designed the board. No strict standards exist with regards to text placement on Hydra V3 satellite PCBs but the text size remains as it was in the stackable PCB design standard:

* + - All text except designer’s name:
      * Text height à 0.75mm
      * Stroke width à 0.15mm
    - Designer’s name text:
      * Text height à 0.5mm
      * Stroke width à 0.1mm

It is likely most convenient to include the board name, designer name, and uORocketry logo on the bottom layer silkmask and these should be included towards the bottom side of the PCB but may be moved as per the designer’s discretion as long as they are in similar locations on all satellite boards.

#### Component Identifiers

Individual component identifiers may be excluded from Hydra V3 boards as they were in Hydra V2 boards. This is to maintain clean-looking high density PCBs and should not be a concern seeing as Hydra V3 PCBs are solely for internal use by people with Altium access.

#### Component Sizing

Hydra V2 followed strict guidelines with regards to component sizing in which all resistors were 0603 and all capacitors were 0805. Due to the size limitations of Hydra V3 and the general pain that is soldering 0402 components, Hydra V3 will use 0603 for all resistors and capacitors unless PCB constraints require the components to be smaller. Decoupling caps will remain 0402s because they are universally used by all ICs on Hydra PCBs and are therefore used in large numbers on all boards. To summarize:

* + - Resistors and capacitors à 0603 unless larger size is required.
    - Decoupling capacitors à 0402

### Hydra V3 Boards

The Hydra V3 ecosystem will be identical to the Hydra V2 ecosystem albeit with the inclusion of two new satellite PCBs (not part of the avionics stack) for interfacing with external avionics hardware. The satellite PCBs do not have many size constraints and will therefore be built as small as possible while also maintaining reasonable mounting provisions.

#### HYDRA\_V3\_POWER

The Hydra V3 power board is a revamped version of the Hydra V2 power board which allows multiple power boards to be stacked within the avionics system. Each power board contains a single regulator per voltage rail alongside a load sharing circuit which allows all power boards to share the load of the avionics system. Furthermore, the Hydra V3 power boards are based around a 3-way MOSFET junction which allows the MCU to actively rewire the PCB as it sees fit. A basic system diagram of the Hydra V3 power board is included below.

A group of squares with text

Description automatically generated

Figure 7. HYDRA\_V3\_POWER System Diagram

The 3-way MOSFET junction allows the board to connect any combination of its power input, the avionics-wide pyrotechnics power rail, and the board’s internal regulators. Furthermore, a battery balance connector is included on each power board to monitor the status of the battery as the 3-way MOSFET junction now provides us with the ability to charge the avionics battery.

The hardware requirements for this board are as follows:

* + - MCU
    - Voltage sense on all rails and temperature sense on regulators as well as current sense wherever possible (at least have current sense at board input but maybe also at regulated rail outputs and on pyrotechnics power rail)
    - Separate regulators for the onboard MCU
    - Battery cell voltage sensing circuit
    - XT30 for battery (could change if you like)
    - Fuse at input
    - MOSFET-based reverse polarity protection
    - MOSFET-based safety switch mechanism
      * Prevent bidirectional current flow à back-to-back MOSFETs
    - Large 5V and 3V3 switching regulators (ideally 2+ amps)
    - Two load sharing circuits, one for each of the regulators mentioned above
      * Load sharing circuits should enable regulators to share the load on the Hydra 5V and 3V3 rails
    - One bi-color status LED per regulator
      * Green à pwr ok
      * Red à pwr prot trip
    - 3-way MOSFET junction
    - Resettable OC and OV protection between 3-way MOSFET junction and regulators
    - **Very important:** have very sensitive OC protection on V\_BATT rail since no other hydra boards have protection on these pins and a short on this rail will kill the system

**Edit Feb 2nd 2024 – Amanda:** Because of issues with the space clearance of some components, we decided to do two power boards instead of only one. In the future, the plan is to have 3 boards, one of them with [idk yet]

*HYDRA\_V3\_POWER\_PROT*

Power input from the battery and power protection overlapped in multiple stages. We have two power subsystems: pyrotechnic and flight critical. We want to ensure that if a subsystem of the power stage goes off (for example, the pyrotechnic power rail), then the system will still be able to provide energy to the 5V/3V3 rails to guarantee that other avionics boards can still operate. An overcurrent/overvoltage protection system was also developed, being present in all boards, to limit the current to the system in the event of a higher current or voltage than initially determined.

*HYDRA\_V3\_POWER\_REG*

Power is then sent through a second board that converts the battery voltage into 3V3 and 5V through two regulators to power the 3V3 and 5V rails. This board is also equipped with current sensing and voltage sensing, as well as temperature sensing of the regulators that monitor their activity on telemetry. Each avionics board has an internal use 3V3 regulator. This is to remove dependency from the main 3V3 rail so that in the case of a failure on that rail, the individual MCUs will still be able to receive power and send data back to the telemetry system, also facilitating the debugging of the system.

*HYDRA\_V3\_POWER\_RECHANGING*

#### HYDRA\_V3\_SPACER

This is a blank PCB included in the system as a spacer. It contains no components on the top or bottom layers other than the stacking connectors included by all stackable Hydra hardware. Its purpose is to provide more clearance to the boards above and below for components and for thermal purposes. It exists because the inductors of the power board exceed the maximum height allowed by Hydra guidelines and cannot be swapped since no reasonable solution with a smaller footprint exists.

MAYBE MAKE THIS HAVE AN INTERESTING SHAPE WHERE IT’S NOT A SOLID PCB BUT RATHER A ‘RIBBED’ PCB. THAT WAY IT CAN PROVIDE MORE PATHS FOR AIR TO MOVE AROUND AND MAY THEREFORE BE ABLE TO ACT AS PART OF A COOLING SOLUTION. WE COULD PUT SCREWHOLES AND CAD A 3D PRINTABLE SHROUD THAT COULD MOUNT TO THIS BOARD AND DIRECT AIRFLOW TO THE PCBS AROUND THIS BOARD FROM A MOUNTED FAN OR EVEN AN AIR SCOOP.

#### HYDRA\_V3\_NAV

The Hydra V3 navigation board is very similar to the Hydra V2 logic board albeit with an upgraded MCU and solder-on eMMc memory. The upgraded MCU provides the board with more processing power in hopes of it keeping up with the high data rate emanating from the SBG Ellipse2-N-G4A3-B1 sensor. The solder-on eMMc memory is included since mechanically affixing the SD card onboard the PCB has proven to not be reliable enough for our application. Lastly, the V3 logic board does not include the buzzer from Hydra V2 since it is a large through hole component that is difficult to add and was not used at all by Hydra V2.

The hardware requirements for this board are as follows:

* + - MCU
    - Voltage sense on all rails
    - Permanent SD card with onboard USB reader IC
    - RS-232 to TTL converter
    - MOSFET-based switch to turn off SBG power
    - Outward-facing CAN network
    - Three status LEDs
      * One for SD card read/write through USB
        + Green means card power
        + Red means card activity
      * One for UART to SBG
        + Green means Rx
        + Red means Tx
      * Second for outward-facing CAN network
        + Green means Rx
        + Red means Tx

#### HYDRA\_V3\_LINK

The Hydra V3 link board is highly similar to the Hydra V2 communication but contains fixed regulator logic, so the board no longer relies primarily on its internal regulators to power itself. Instead, the board will use the avionics power rails until they fail and will only switch to internal regulators at that point. Furthermore, the RFD900 is no longer mounted on the communication board due to its small size and therefore interfaces with the board using a connector at the rear of the PCB.

The hardware requirements for this board are as follows:

* + - MCU
    - Voltage sense on all rails
      * **Include voltage sense on internal rails as well**
    - Onboard regulators with failover logic
      * **Swap 5V regulator**
      * **Swap logic à use HYDRA power by default**
      * **Change bulk capacitors**
    - RFD900 interface
      * **Fix RFD900 footprint**
        + **Third screwhole is wrong**
    - eMMc memory?
      * Maintain SD card for first iteration
    - **Two status LEDs on this board**
      * **One for UART activity (OP-amp-based)**
        + **Green is Rx**
        + **Red is Tx**
      * **Second for regulator status**
        + **Green means using Hydra power**
        + **Red means using internal regulators**

#### HYDRA\_V3\_RECOVERY

The Hydra V3 recovery board is virtually identical to the Hydra V2 recovery board albeit with an updated sensor package that should hopefully be more functional. This board contains both an IMU and a barometer such that it can fully replace the navigational data provided by the SBG with the exception of the SBG’s GPS data. The sensors chosen for the Hydra V3 recovery board are as follows:

* IMU: TDK InvSense ICM-40609-D
* BARO: TE MS561101BA03-50

The hardware requirements for this board are as follows:

* + - MCU
    - Voltage sense on all rails
    - Separate SPI links to each of the two sensors listed above
      * **IMU must be centered on the PCB**
    - MOSFET-based pyro switches
      * Continuity and detonation detection
        + Go with current-based detection
        + Voltage-based continuity detection is great but doesn’t provide feedback on whether the charge drew enough current to ignite. We can detect both continuity and successful ignition by just measuring current going through the pyro channel.
    - MOSFET-based safety switch for pyrotechnics power
      * Add a 2-pin Molex Picoblade connector to the rear of the PCB that will interface with a keyswitch on the rocket. Once the two pins of the Molex connector are shorted, the MOSFET switch should turn on and allow pyrotechnics power to reach the actual pyro switches
    - Two status LEDs for pyro channels
      * One bi-color per pyro channel
        + Green means continuity
        + Red means ignition

#### HYDRA\_V3\_BEACON

This board is a last-minute addition for Hydra V3 intended to replace the featherweight tracker in future rockets. This board features a LoRa-based telemetry radio and a battery which charges from the Hydra power rails and provides HYDRA\_V3\_BEACON with a minimum of 3 days’ worth of power once the primary avionics battery has been exhausted. This board may additionally include a standalone GPS receiver to provide a reliable source of positional data that is not dependent on other hardware within the avionics system.

The buzzer output hardware requirement would effectively be a MOSFET switch that would turn on some external buzzer using an external supply. This board won’t have enough power to drive any meaningfully loud buzzer but it may be possible to power a buzzer from the primary avionics battery such that it will go off if that battery has not been fully drained yet.

The hardware requirements for this board are as follows:

* + - MCU
      * Possibly a different low power one but the SAME51 should be fine
      * Used STM32L051K8T6TR
    - Voltage sense on all rails
      * Also sense on all internal 3V3 rails and battery
        + 3V3 rail, batt 3V3 rail, Hydra 3V3 rail
    - LoRa radio or some other low power long range alternative
    - Standalone battery
      * But default to Hydra power
    - Charging circuit for aforementioned battery
    - Standalone GPS receiver
      * uBlox MAX-M10M
    - Buzzer output?
    - Could add some status LEDs for a ‘cool factor’ but these should turn off when the board is running off of its internal power
      * One status LED for internal battery status
        + Green power on
        + Red is charging
      * One status LED for UART activity
        + Green is GPS Rx
        + Red is LoRa Tx

#### HYDRA\_V3\_GUIDANCE

HYDRA\_V3\_GUIDANCE is a stackable PCB designed to interface with up to six servo or servo-like actuators in the context of a guided vehicle. It is single-handedly responsible for running all guidance algorithms and interfacing with actuators based on navigational data provided by other boards on the avionics CAN bus. Due to the potentially current-heavy load induced by driving these servos, this board must incorporate internal regulation and pull power from the PYRO\_PWR rail to not exceed the current capacity of the flight-critical, regulated, avionics power rails. Additionally, considering the significant computational power that may be needed for active guidance, this board must incorporate the STM32H733 MCU.

The hardware requirements for this board are listed below:

* + - STM32H733 MCU
    - Voltage sense on all rails
    - Internal adjustable regulator for servos pulling from PYRO\_PWR
      * No need to make it programmatically adjustable but could be cool if possible
        + Swapping feedback resistors on the PCB is good
      * Ideally multiple regulators for the servos
        + Ideally three, so one per pair of servos

Six may be too many

That way a regulator fault doesn’t take out all servos

* + - Power protection
      * **Individual** power protection **on each servo**
        + Simple fuse is perfect
      * Resettable overcurrent latches **on each regulator** from PYRO\_PWR?
        + Simple fuses are fine once again, but resettable latch would be better

Space will probably be an issue

Remove the HYDRA 5V overcurrent protection since this board shouldn’t need 5V

* + - Two status LEDs
      * Arming LED (connect to red and green to two separate MCU pins, rest is software)
        + Green if system disarmed
        + Red if system armed
        + Yellow if system error
      * Regulator status LED
        + Green if internal power ok
        + Red ? idk what to do with this

Could be used for displaying the status of the multiple regulators if that is implemented

Could also be used to represent the status of the over-current protection on each regulator

#### HYDRA\_V3\_CAMERA

HYDRA\_V3\_CAMERA is a stackable board designed to record up to four analog video feeds to an onboard SD card while simultaneously enabling the ground station to obtain still frames of any video feed through the vehicle’s telemetry link. Analog cameras are used to enable long wires between this board and the camera itself; a satellite board for this purpose would have been unnecessarily large and a stackable board makes more sense to have access to the high speed avionics CAN buses.

The Analog Devices ADV7280 IC is used for decoding the analog video feeds and converting them into a digital format. This IC has an integrated 4-channel camera MUX but **cannot** record more than one camera simultaneously. Lastly, for reliability reasons, the cameras are powered from the PYRO\_PWR bus as opposed to the flight-critical avionics power rails since they will be wired and housed outside of the avionics stack.

The hardware requirements for this board are as follows:

* + - STM32H733 MCU
    - Voltage sense on all rails
    - Standalone regulators for external cameras pulling from PYRO\_PWR
      * 5V
      * Maybe include two independent regulators?
        + One per bank of two cameras
        + Have some way to only populate/use one regulator because we’re gonna be broke with this many boards
    - SD card for recording video
    - ADV7280 video encoding IC
      * The -M (8-channel) version is cheaper than the 4-channel version **but** 8-channel version uses MIPI-CSI2 which the STM32H733 does not support whereas the 4-channel version uses ITU-R BT.656
    - Four camera interfaces (molex picoblade)
      * 3-pin interfaces
        + GND
        + 5V
        + VIDEO

#### HYDRA\_V3\_SENSOR

HYDRA\_V3\_SENSOR is a satellite board designed to interface with pressure transducers and temperature sensors within a hybrid engine. The propulsion team only wanted pressure transducers within the flight-ready engine, but this board is designed to be compatible with many more sensors such that it can be used for any sensor we may want to integrate down the line. This board only takes battery power and CAN; all additional voltage regulation must happen internally.

The list of sensors that HYDRA\_V3\_SENSOR must interface with, alongside the associated hardware requirements are listed below:

* + - **Thermocouples –** generate voltage based on temp difference between the connector and the thermocouple’s head
      * 3 ADC pins (two for sensing the thermocouple and another for a thermistor at the connector of the thermocouple
      * Add pullup/pulldown resistors to detect disconnect/burnout
      * Connect directly to an instrumentation amplifier
    - **Thermistor** – changes resistance based on temperature
      * 1 ADC pin
      * Needs pullup/pulldown resistor to form voltage divider
      * Instrumentation amp/ADC connected at junction between thermistor and pullup/pulldown
    - **Pressure transducers** – Output voltage or current based on pressure
      * 1 ADC pin
      * For voltage à instrumentation amp + voltage divider
      * For current à run output across resistor and measure voltage
    - **Load cell** – wheatstone bridge
      * 1 ADC pin
      * Instrumentation amplifier
    - **Strain gauge** – basically a resistor
      * 1 ADC pin
      * Instrumentation amplifier

Based on the sensor list above, HYDRA\_V3\_SENSOR should function as follows: four pins should be used for sensor input with two providing power and the other set of two being the sensing pins. Each sensing pin should have pads for pullup/pulldown resistors which can be left unpopulated, populated, or short circuited depending on the board’s application. Additionally, a resistor should be placed between both of these inputs to sense current-based devices. A thermistor should be placed as close as possible to the sense pins to provide temperature data of the connector’s pins for measuring thermocouples. Lastly, pads should be provided to create a RC differential antialiasing filter which can also be used in as a common-mode RC filter if the sensor being measured is not differential.

A black background with a blue circle and white hexagon with black text

Description automatically generated

Figure 8. Differential Antialiasing RC Filter Schematic

Multiple of the interface assemblies described above (3 or so) should be included on the HYDRA\_V3\_SENSOR board. Each interface assembly being a set of sensing pins with the accompanying pads for the resistors described above. These interfaces should then ender a MUX + PGA+ ADC IC such as the Texas Instruments ADS124S06. The board should not rely on the MCU’s internal ADC but rather some specialized, higher-resolution MUX + PGA + ADC IC.

In addition to the extensive sensor interface above, it could be useful to expose a serial communication interface to use SPI, I2C, or UART-based sensors. Lastly, this board should include some method to store data onboard at a high logging rate while simultaneously transmitting it over CAN.

Special attention must be paid to the regulators used on HYDRA\_V3\_SENSOR to ensure noise within the sensing circuitry is minimized. [This link](https://e2e.ti.com/blogs_/b/powerhouse/posts/how-to-enhance-power-and-signal-integrity-with-low-noise-and-low-ripple-design-techniques) may be useful but this will definitely require a bit of research.

The hardware requirements for this board are as follows:

* + - MCU
    - Voltage sense on all rails
    - Power protection
      * Reverse polarity and over-current (fuse is good enough)
    - Internal 3V3 regulation
      * Maybe 5V regulation as well if any hardware requires it
    - Exposed SPI interface
      * Also include exposed I2C and UART interface is possible but SPI is most useful
    - MUX + PGA + ADC IC with low noise supply
      * MCU power and the power for the ADC IC should be separate
        + Ideally linear regulator I think?
      * ADC IC should have a standalone low noise supply
    - Three sensor interfaces with plenty of exposed pads, as described above, to allow for the following features. Each sensor interface means a set of four pins used to interface with an external sensor, two of which are sense pins and an additional two pins for power and ground.
      * Pullup/pulldown resistors on both sensor input pins
      * Resistor between sensor input pins
      * Differential RC antialiasing filter
      * Thermistor near the sensor interface’s connector
        + Possibly shift thermistor off of the PCB if thermocouple adapter becomes a thing.
      * A set of pins for power
    - SD card slot
    - Status LEDs (optional)
      * One for CAN activiy
        + Green means Rx
        + Red means Tx
      * One for ADC activity
        + Green means power (since ADC power domain is separate from MCU power domain)
        + Red means Tx from ADC to MCU
      * Maybe more for exposed SPI interface activity and for SD activity but this may be starting to get excessive.

#### HYDRA\_V3\_ACTUATOR

The Hydra V3 actuator board is used to interact with servos and solenoid valves within the flight-ready hybrid rocket engine. This board only takes battery power and CAN; all additional voltage regulation must happen internally.

This PCB should contain a beefy programmable power supply to power any servos that may be attached to it as well as an H-bridge to drive solenoids/motors. In case the board is driving a feedback-less motor, it must also be able to interface with a minimum of two limit switches and a potentiometer all of which should be connected to ADC/interrupt-capable pins of the MCU.

The hardware requirements for this board are as follows:

* + - STM32H733 MCU, just the normal MCU used on all boards, maybe it’ll get updated some day but that day is not today
    - Voltage sense + Current sense on all rails
      * This shit is literally copy pasted from another boards schematic that Serban made, I blame him if anything goes wrong but seems like it works so I give a thumbs up
    - Power protection
      * Reverse polarity and over-current (fuse is ok but resettable OC protection may by better here)
    - Internal 3V3 regulation
    - Exposed analog/interrupt-capable I/O for limit switches and some form of position feedback like a potentiometer.
    - Servo interface
      * Programmable power regulator
      * PWM output
    - MOSFET/motor interface
      * H-bridge
    - Status LEDs (optional)
      * One for servo interface
        + Green for servo power
        + Red for PWM data
      * One for H-bridge status
        + One colour for forward
        + Other colour for backward
      * One for limit switches?
        + One colour per limit switch

# Software Design

Enjoy, cutie

# EMI/RF Considerations

Four antennas are used for the avionics stack’s various RF components: two quarter-wave 900MHz dipoles for the RFD900, one 900MHz dipole for the Featherweight tracker, and a GPS antenna for the SBG. The avionics stack may be surrounded with non-conductive RF isolating material depending on future testing which will be done once the boards are manufactured. A potential concern is the proximity of the RFD900’s antennas to the ejection circuitry though the RRC3s are likely perfectly capable of withstanding the radiation induced by the RFD900’s relatively low 1W transmission power.

The RFD900’s two dipoles will be placed orthogonally and perpendicular to the rocket’s axis above the avionics stack alongside the GPS antenna. The Featherweight’s antenna is placed at the top of the payload bay to limit interference since it transmits on the same band as the RFD900. As previously mentioned, the entire Featherweight unit may be transferred to the top of the payload bay pending testing.

# Arming and Safety Considerations

The power board has a cutoff switch that interacts with a rod protruding from the rocket’s fuselage which must be removed to allow power to flow from the battery to the voltage regulators. Furthermore, the ejection board has two separate switches, one for each RRC3, that utilize a similar mechanism to disconnect the 9V batteries from their accompanying RRC3 prior to flight. Both of these rods will protrude out of both sides of the rocket through holes drilled along a centerline of the fuselage. Additionally, each rod intended to be removed before flight will have a Clevis pin to ensure it cannot be removed accidentally. A sample photo of the proposed locking pin design is included below in Figure 5.

Text, whiteboard

Description automatically generated

Figure 9. Clevis Pin

3D-printed rod guides will be attached to any PCB containing safety cutoff switches such that a dummy rod can be inserted during assembly to maintain the switches in an open position. Once the rocket has been fully assembled, the real rod will be inserted into the fuselage hole consequently pushing out the dummy rod without ever closing any safety cutoff switches. These 3D-printed rod guides will also contain O-rings to provide additional friction for maintaining the safety rods in place.

# Battery and SBG Considerations

As depicted in Figure 4, the battery is placed horizontally below the PCB stack. The avionics battery will be a 3S 2200mAh lithium polymer pack as we found this to be lighter than using lithium-ion cells of a similar capacity while also providing plenty of battery life. Two additional 9V batteries are included for each RRC3. All batteries are attached using a combination of double-sided adhesive and straps going through slits in the battery mounting plate.

The SBG was included in this section as it lies on the second board from the bottom of the stack and is relatively large. As a result, the communication and ejection boards will have cutouts in the middle to allow the SBG’s body to protrude through their center thereby providing the required clearance. Due to the relative simplicity of the hardware, this cutout does not deprive us of any required PCB space. A few additional cutouts are included on the rear of the stack to allow RF and power cables to pass through the boards since clearance around the avionics stack is limited. These cutouts allow RF cables to pass from the RFD900 to the two antennas above the stack and the GPS antenna’s cable to connect with the SBG. Additionally, they allow power cables from the two 9V batteries on the bottom plate to pass through to the RRC3s on the recovery board.