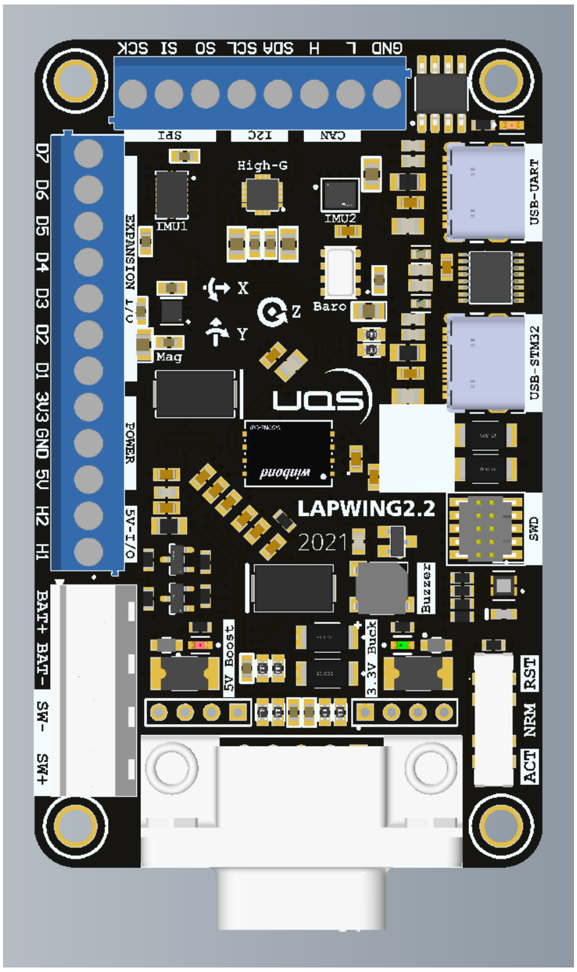
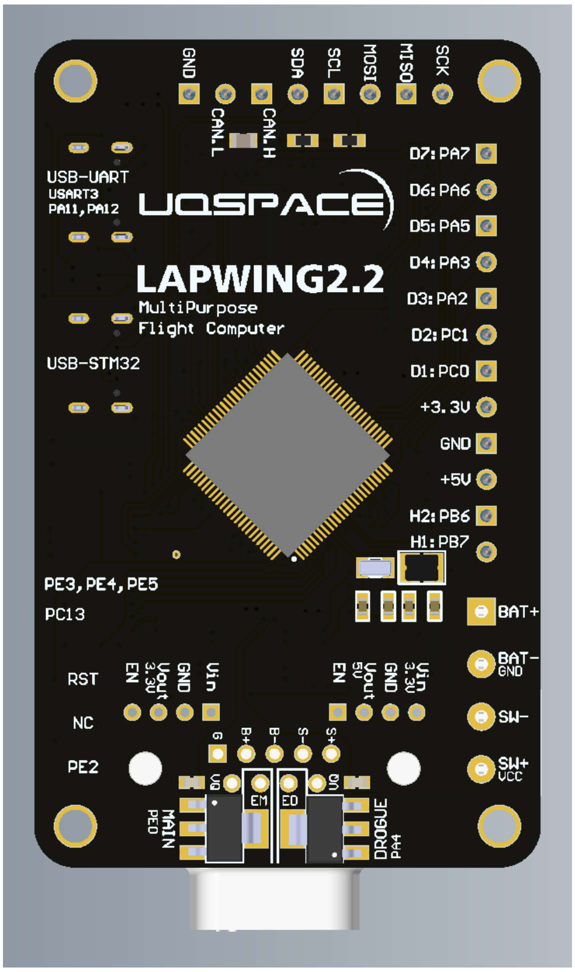
LAPWING 2.2 Hardware Documentation

Created	@September 25, 2021 11:27 AM
Last Updated	@December 2, 2021 1:58 AM
Owners	
Status	Current 👍



LAPWING 2.2 Top View



LAPWING 2.2 Bottom View

Overview

LAPWING 2.2 is a multi-purpose flight computer custom-built by the Avionics team of UQ Space.

Among a plethora of features, it has onboard data storage, a range of sensors equivalent to 10-DOF (accelerometer, gyroscope, magnetometer and barometer), 3.3V and 5V power, expansion capability through the GPIO terminals and 2 high-current low-side MOSFET switches (ie. the e-match circuit).

Part of the LAPWING 2.x line (powered by the STM32L4 microcontroller), and the next revision succeeding LAPWING2.0, this board adds convenience modifications.

Compared to LAPWING 1.x, the LAPWING 2.x line of boards introduces major improvements and new features:

- Powered by the high performance and ultra-low-power STM32L4 microcontroller;
- GPIO / Expansion capability; and
- Improved rules compliance for Avionics' Safety Critical Systems.

General Specifications

To reuse functioning components, there are two variants to LAPWING 2.2 boards:

- (Black) Production, flight-ready board which follows the schematic / bill of materials to the letter.
- (Purple) Development / Refurbished, which may have some different components that are not always fully firmware compatible (ie. microcontroller STM32L476VET6 instead of STM32L486VGT6).

Features

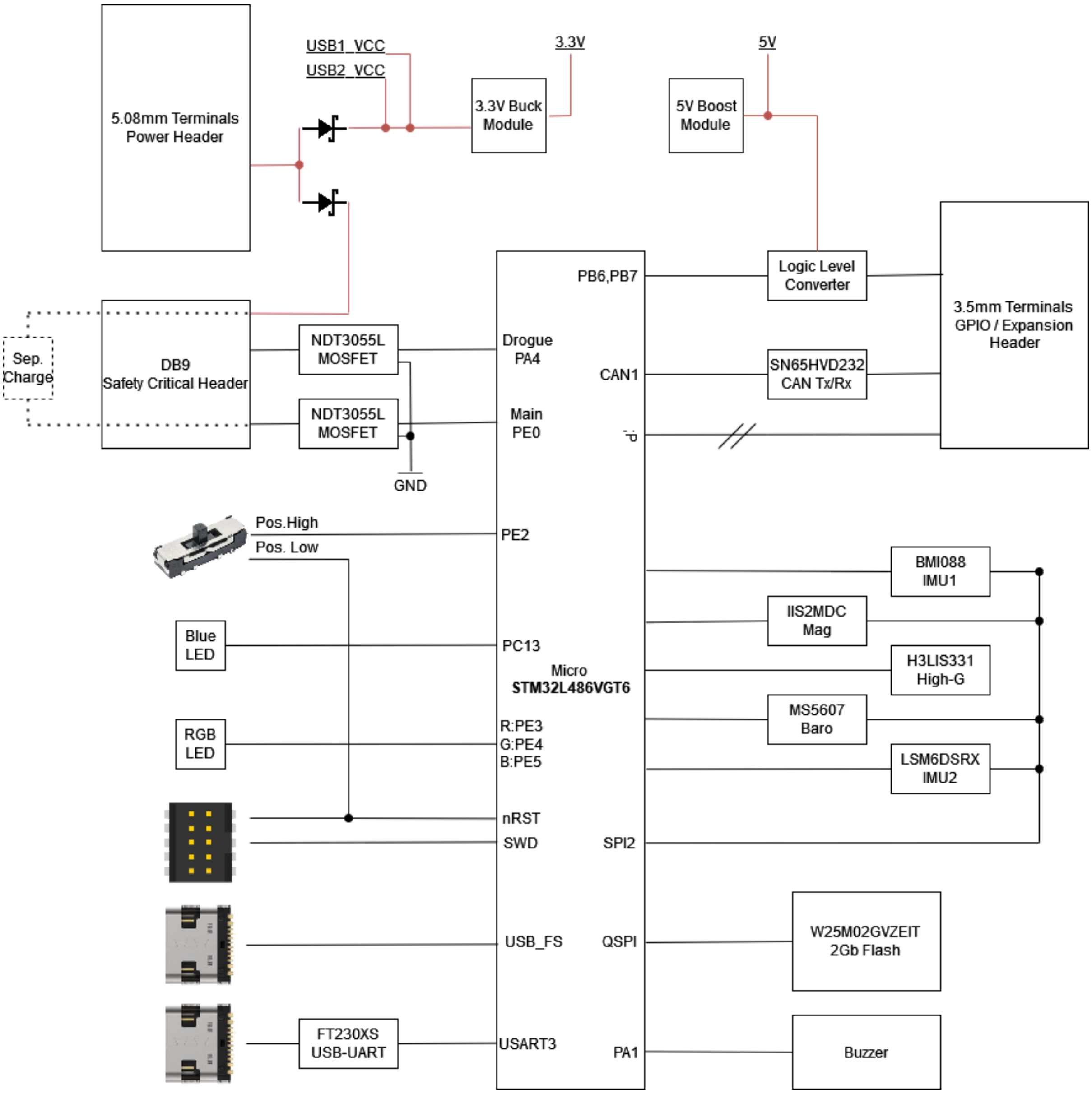
- STM32L486VGT6 Microcontroller
- On-board 2Gb SLC NAND Flash Memory
- USB-C (USB 2.0) to both UART and Native USB interfaces
- JTAG / SWD Connector
- RGB and Blue Status LEDs
- Reset / Mode Switch
- 3.3V and 5V power, delivered by daughter-board
    - *See specific module datasheets for specifics*
- Sensors
    - LSM6DSRX / BMI088 - IMU (accelerometer and gyroscope)
    - H3LIS331DL - High-G accelerometer
    - MS5607-02BA - Barometric Pressure
    - IIS2MDC - Magnetometer

- GPIO / Expansion Capability (CAN, UART, I2C, SPI, TIM, ADC, DAC)
    - *See below for specifics*
- Two max.4A low-side load switches (ie. e-match circuit)

Interfaces

System Architecture

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/e39623d4-189e-492f-81a3-2983cce495ae/lapwing2p2_systemdiagram.drawio



Electrical

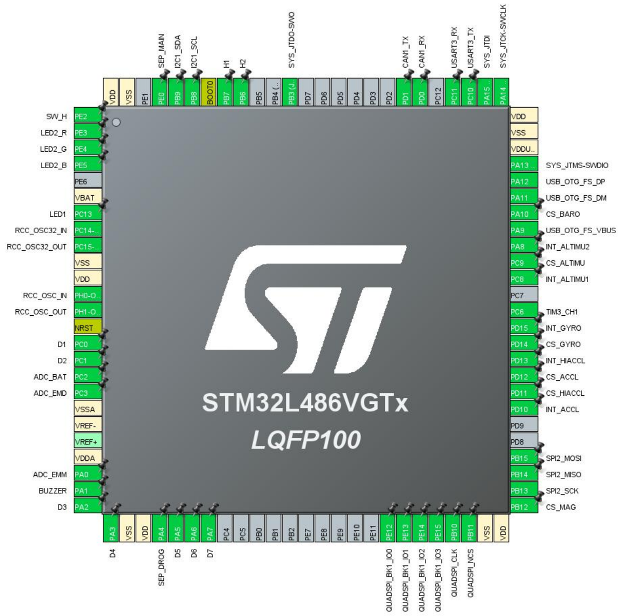
Lots of IO used on the micro.

Refer to following files (Open the .ioc with STM32CubeIDE).

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/b090e783-9042-4e7d-b8e2-c93511499c86/lapwing2p2.ioc

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/3f4c3cf4-75c5-41de-b76f-cc070c526314/cubemx_lapwing2p2.pdf

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/4c0d5810-46b7-4e83-8e6f-b2bbcd9410c4/cubemx_lapwing2p2.txt



Headers

Unless specified, order of pinout is from Left to Right given the standard orientation of the board (DB9 connector pointing vertically down).

Power Modules (required to power board)

- 3.3V Buck Regulator: Generic regulator with 3.3V output from an unspecified input (max 25V due to board capacitors).

> Modelled for Sparkfun BabyBuck, a 3.8V-32V input to 3.3V output 2A synchronous buck converter module.

  - Marked on board, Pinout is Vout [3.3V], GND, Vin, EN [NC]

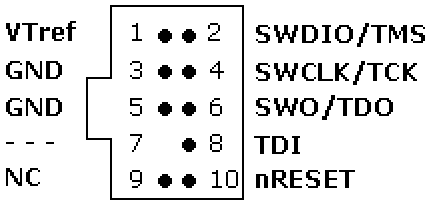
- 5V Boost Regulator: Generic regulator with 5V output from a 3.3V input.

> Modelled for Adafruit MiniBoost 5V, a 2V-5V input to 5V output 1A boost converter module.

  - Marked on board, Pinout is Vin [3.3V], GND, Vout [5V], EN [NC]

Debug

- miniSWD: SWD / JTAG Programming Interface
  - Specified in figure below, Connected to SWDIO [PA13], SWCLK [PA14], JTDI [PA15], SWO [PB3] and nRST.



- USB-STM32: Native Interface on USB_OTG_FS
  - Directly connected to USB_OTG_FS_DP [PA12], USB_OTG_FS_DM [PA11] and USB_OTG_FS_VBUS [PA9].
- USB-UART: UART Interface on USART3
  - Connected through *FT230XS* to USART3_TX [PC10] and USART3_RX [PC11]

Power

- 5.08mm Terminal Header

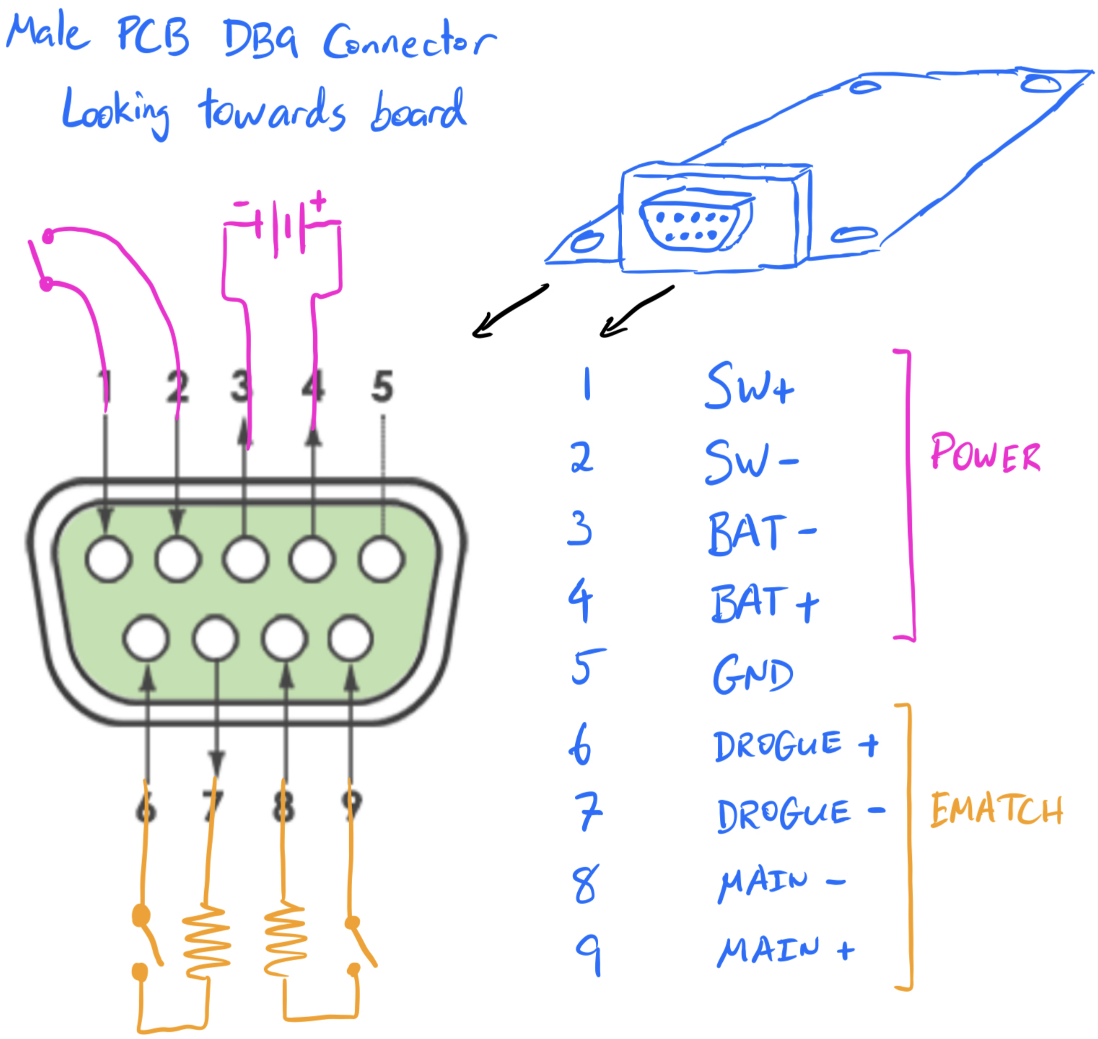
> Do not use if Safety Critical Header is also supplying power (power backfeeding).

    - BAT+ [connected to SW-]
    - BAT- [GND]
    - SW- [connected to BAT+]
    - SW+ [VCC]
    - Measurement with ADC PC2 (VCC voltage *0.161)

Expansion - Safety Critical Header

SEP_DROG: Low-side switch on PA4

- Alt function LPTIM2_OUT and DAC1_OUT1 (variable resistance)
- Continuity checked with ADC PC3 (EMD- voltage *0.161)
- SEP_MAIN: Low-side switch on PE0
    - Alt function TIM16_CH1
    - Continuity checked with ADC PA0 (EMM- voltage *0.161)



LAPWING 2.2 Safety Critical Header

Expansion - GPIO Terminal

- H1: PB7
  - GPIO, I2C1_SDA, TIM4_CH2, USART1_RX, EXTI7
- H2: PB6
  - GPIO, I2C1_SCL, TIM4_CH1, USART1_TX, EXTI6
- +3.3V: Non-isolated 3.3V output (same power domain as microcontroller, sensors, etc.).
- GND: Common Ground.
- +5V: 5V output (same power domain as Logic Level Converter)

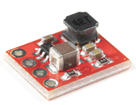
- D1: PC0
    - GPIO, ADC, I2C3_SCL, LPTIM1_IN, LPUART1_RX, EXTI0
- D2: PC1
    - GPIO, ADC, I2C3_SDA, LPTIM1_OUT, LPUART1_TX, EXTI0
- D3: PA2
    - GPIO, ADC, TIM15_CH1, TIM2_CH3, TIM5_CH3, USART2_TX, EXTI2
- D4: PA3
    - GPIO, ADC, TIM15_CH2, TIM2_CH4, TIM5_CH4, USART2_RX, EXTI3
- D5: PA5
    - GPIO, ADC, DAC1_OUT, TIM2_CH1, SPI1_SCK, EXTI5
- D6: PA6
    - GPIO, ADC, TIM16_CH1, TIM3_CH1, SPI1_MISO, EXTI6
- D7: PA7
    - GPIO, ADC, TIM17_CH1, TIM3_CH2, SPI1_MOSI, EXTI8

Power Consumption

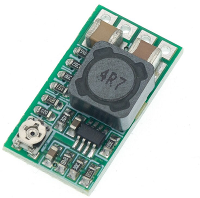
Power Modules

Input power to the board is dictated by the 3.3V Buck Regulator Daughterboard. There are two modules that we're using:

- (Preferred) Sparkfun BabyBuck, based on the AP63203

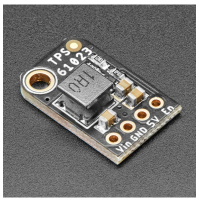


- HW-613, based on the MP2315 (unsure, AliExpress module)



5V isn't really used by the board itself; but rather to enable 5V I/O (H1 and H2 of the expansion header). The two modules that we're using:

- (Preferred) Adafruit MiniBoost 5V, based on TPS61023



- HW-626. 2-5V input, 5V 0.5A output.

Absolute Maximum

- Input Battery (25V, 4A+2A)
    - Forward-facing are tantalum cap (**25V**), diodes (40V 5A) and e-match's NDT3055L (60V 4A)
- USB (5V, 500mA)
    - Forward-facing are ferrite bead (1.5A), although USB spec will limit current to **500mA**.

- SEP_MAIN and SEP_DROG (Battery Voltage (max 25V) 4A).
- CAN_TX and CAN_RX conform to SN65HVD232 (4V to 16V).
- H1, H2, D1, D2, D3, D6, D7 are 5V tolerant pins.
- D4, D5, SCK, SI, SO, SCL, SDA are 3.3V ONLY pins.

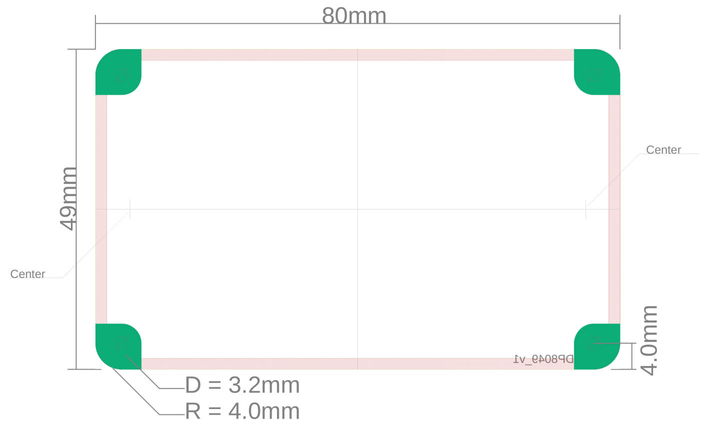
Power Budget

Aa Device Name	# Current	Description
STM32L486VGT6	10.3	80MHz, Run, 55degC
SN65HVD232	10	(driver) typ 10, max17
RGB LED	5	(red) 6, (green, blue) 1-5
Blue LED	5	roughly 1-5
+3.3V LED	5	VF=2.1
+5V LED	13	Vf=2V
LSM6DSRX (Alt BMI088)	1.2	LSM6: (high perf.) 1.2, (normal) 0.7, (shutdown) 3u BMI088: 5.15
H3LIS331	0.3	(normal) 300u, (low power) 10u, (shutdown) 1u
MS5607	1.4	(normal) 12.5u, (peak) 1.4
IIS2MDC	1.13	(high perf) 1.13, (low power) 25u, (shutdown) 1u
W25M02GVZEIT	25	(active) 25, (standby) 20u
I2C Pullup	3	2*4.7k
Logic Level Conversion Pullup	2	2*10k (3.3V), 2*10k (5V)
3.3 Regulator Quiescent	0.022	22u
5V Regulator Quiescent	0.02	20u
General Pullups (ie. mode switch)	2	5*10k

Mechanical

The dimensions of the board are 80x49x20mm.

The board size is based on the DP8049_v1 Sick of Beige v1 PCB Specification by Dangerous Prototypes, as defined at http://dangerousprototypes.com/d



Manufacturing Plan

Most soldering will be done by reflow oven. Use 60/40 solderpaste. Light cleaning with isopropyl wipes.

Requires:

- Full day (8am to 3pm) for 3 boards.
- Jaycar solderpaste (60/40) https://www.jaycar.com.au/solder-paste-smd-syringe-15g/p/NS3046
- Card to spread solderpaste
- Electropolished Stencil
- ETSG with manual solder paste printer (50-S309 tutorial room).

Procedure for applying Solder Paste.

1. Leave solderpaste out for at least 30 minutes, but not more than 4 hours.
2. Clean everyting with isoproyl wipes (stencil, pcb, card)
3. Prepare stencil printer with stencil. Align everything up until stencil is aligned and flush with PCB.

4. Add a significant amount of solderpaste to the left edge of the stencil (assuming right-handed).
5. on the left side of the solderpaste, touch the card lengthwise down and tilt top towards board at a 50 degree accute angle.
6. ONE PASS ONLY, wipe the solderpaste, gradually lowering the card as to push the solder into the gap. More passes will mess up the edges of solderpaste.
7. Temporarily place finished pcb into fridge. Repeat for a max of 3 boards at any one time.

Procedure for component placement

- Place components onto the board. Start with caps and resistors, the work way up to larger components. Leave complex alignment components like sensors and USB until last.

> If the solder paste gets too wet, place in the refridgerator covered for 10 minutes.

- Touch up components afterwards with a powerful >40W Temperature controlled soldering iron (Weller, Hakko). Work at 330degC or 350degC, and use small amounts liquid no-clean flux.

> DON'T USE TACKY FLUX

- Clean with isopropyl wipes, don't spray directly onto board.

Testing Plan

//TODO: Complete test firmware to be spec-ed and commissioned...

1. Visually check the board (preferably under microscope) to look for any shorts.
2. Resistance test 3.3V to GND and 5V to GND.
3. Connect 3.3V to power supply, Output as 3.3V and 0.02A limit. If it 'shortcircuits', voltage should be >2.5V, if not FAULT.

4. Connect power pins (bypass Switch) to 1V, then step up to 3.3V then 5V then 12V, checking each time. Power consumption at idle should be < 50mA, active less than 100mA.

Design Files

The latest files are available on the UQ Space `avionics-hardware` GitHub repository.

https://github.com/uqspace/avionics-hardware

PCBs were printed from JLCPCB. Production Files (Gerber and BOM) are provided here.

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/a1e52dcb-926d-4646-9124-b63ca7f57816/Gerber_Lapwing2p2.zip

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/171e724e-6ae2-47b6-a2a5-e9fc2d49a107/BOM_lapwing2p2.csv

https://s3-us-west-2.amazonaws.com/secure.notion-static.com/61d0a518-9ab0-45ba-a3b9-c8f7240e9bcc/lapwing2p2.pdf

Thanks to all those who contributed to this project... Your help made this possible! 😆