

Backplane 2.0

Yoav Binyamin

1.17.2026

Contents

1	Introduction	1
2	Functional Overview	2
3	Voltage Sensing	3
4	Temperature Sensing	4
5	Indication Circuitry	4
6	Filtering	6
7	Grounding	6
8	PCB Design	6
9	Testing	9
10	Additional Documents and Links	10

1 Introduction

SOMTECH MIRAGE's Backplane 2.0 and corresponding power modules is the first iteration of a power distribution board for MIRAGE's URC rover. I wrote this for my personal sake, and as future documentation to aid future design and help non-electrical members interface with the power distribution system should it fail.

Mostly Aaron Moy (my project manager) and I worked on this design fairly equally, though there were some other contributors, including Haoze Wang and Reed Clardy with some additional tasks like research and help on the schematic with hatching planes and filtering. Eric Pham and Joaquin Zapanta helped verify footprints for the BOM.

2 Functional Overview

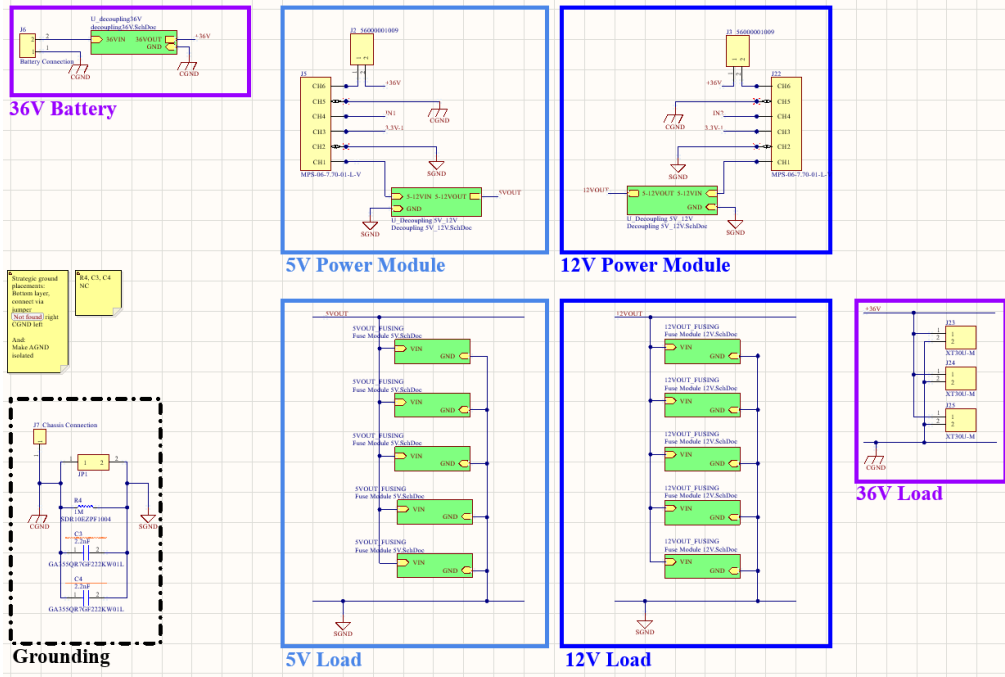


Figure 1: Main project schematic, part 1

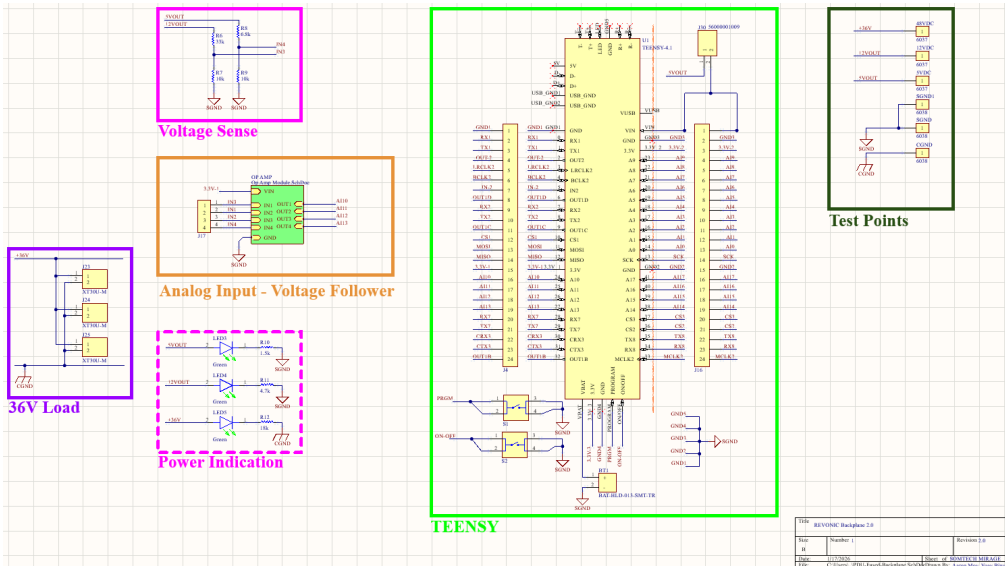


Figure 2: Main schematic, project part 2

The Backplane 2.0 acts as a backplane for our 36V-12V (FLEX PKU5511ESI 4.2A) and 36V-5V (FLEX PKU5513ESI 10A) fuse box with a 36V battery input. These are placed into PCBs and connected vertically from the backplane with power connectors. Should a power module fail, it may be easily replaced. To actively monitor the system's power and health of the buck converters, the Backplane 2.0 has voltage sensing, LED indication, and temperature sensing dedicated to each module but still placed on the Backplane 2.0. The Backplane 2.0 also acts as a fusebox with fuse-blown LED indication circuitry, and has a TEENSY4.1 that can support additional sensing

for the rover's sensing. Finally, it also has an LED indication for 36V power being connected. Should you need to interface with the design, here is a small overview:

1. Power input: Connect the battery, 18V-60V (notated as 36V), by soldering wires on the top left BATT+ and BATT- input. There is an auxillary battery for the TEENSY4.1 on the bottom of the board.
2. Power-on indication: There are three (low power, 2mA) LEDs at the bottom left of the board indicating the power status of the 3 main voltage levels (exlucing 3.3V for the sensing)
3. Connectors/Interfaces: On the top left there are three 36V output connectors. On the top, five 12V output connectors. On the bottom, five 5V output connectors. There are many of them to provide high flexibility for extra electrical components on the rover. The TEENSY4.1 is connected at the right, the left power module is for 12V, and the right power module is for 5V.
4. Grounding: find the "CGND" pad on the bottom of the board to connect it to chassis. Find the "OPT_FLT" (optional filter) at the bottom of the board to adjust grounding after testing; currently, there is just a jumper cable connected. (See more in "Grounding.")
5. Test points: Locate the test points next to their respective outputs. There are extra test points for the voltage dividers located near the voltage follower, at the left of the TEENSY4.1.
6. Fusing: There are several fuses of different current specs placed throughout the board. There is one for the TEENSY4.1, one for each power module (on the power module boards), and one for each output connector of 5V and 12V. If a fuse is blown on the 5V or 12V output connectors, a red LED beside it will shine indicating it must be replaced (you should replace it with the correct one). As for those without indication, it will be obvious from the TEENSY4.1 and power indication whether they are blown or not.
7. Voltage and temperature sensing: Simple voltage dividers connected to the TEENSY4.1 will be programmed and calibrated to be able to sense the temperature of the critical point (MOSFETs) in the FLEX power modules as well as the output voltage.

3 Voltage Sensing

For the voltage dividers that sense voltage, the equation for voltage is:

$$V = \frac{R_1 + R_2}{R_2} U$$

Where V is the voltage to be measured, R_1 is the top (varying) resistor in the configuration, R_2 is the bottom (constant $10k\Omega$) resistor in the configuration, and U is the voltage measured by the TEENSY4.1.

The TEENSY4.1 has 10 useable bits, e.g. 1024 different discrete values. Plugging in values, noting the imprecision and propagating uncertainty (resistors have 0.1% tolerance), the final values are:

1. for 5V:

$$V = 1.68U$$

precision $\approx 5.41mV$ uncertainty $\approx 4.17mV$. Maximum voltage before breakdown 5.544V.

2. for 12V:

$$V = 4.3U$$

precision $\approx 13.9mV$ uncertainty $\approx 16.9mV$. Maximum voltage before breakdown 14.19V.

Testing will help calibrate this sensing to the actual voltage.

4 Temperature Sensing

For the voltage dividers that sense temperature, the equation for temperature is:

$$T(R) = T \left(10,000 \frac{U}{3.3 - U} \right)$$

Where T is temperature based on resistance and U is the voltage measured by the TEENSY4.1. The values for temperature based on resistance are found by a table from <https://www.vishay.com/en/thermistors/ntc-rt-calculator/> and the temperature is linear interpolated from resistance. The resistance tolerance cannot be changed but the measured resistance should not change more than 3% which is reasonable with the 0.1% resistance tolerance and the TEENSY4.1's $\approx 0.1\%$ precision. The resulting uncertainty is about $2^\circ C - 4^\circ C$ for extreme values and about $1^\circ C$ for intermediate (room temperature and close) values. This uncertainty is a function of the temperature/resistance so it can be found dynamically for the temperature/resistance sensed.

Testing will help calibrate this sensing to the actual temperature.

5 Indication Circuitry

Red LEDs turn on when fuses are blown for those with indication circuitry implemented. These are any connector outputs for the 5V and 12V lines. The circuitry utilizing is the same but with different values (found by PSPICE simulation) for 5V vs 12V. Here are the PSPICE circuits used to determine the resistances for the circuit. Keeping the bottom right resistor as 1000 times the original resistance seems to work well.

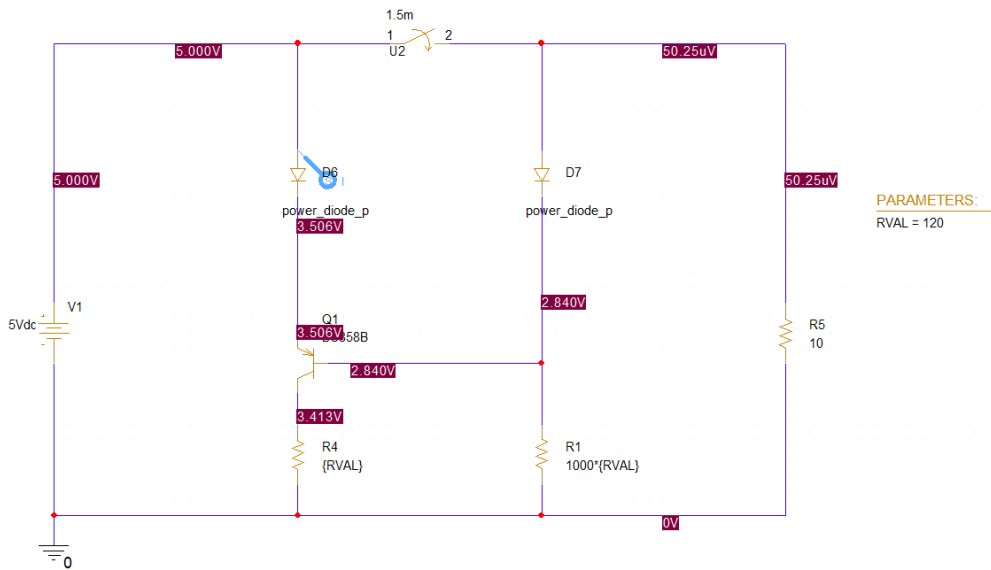


Figure 3: PSPICE Schematic for 5V Fuse-Blown Indication

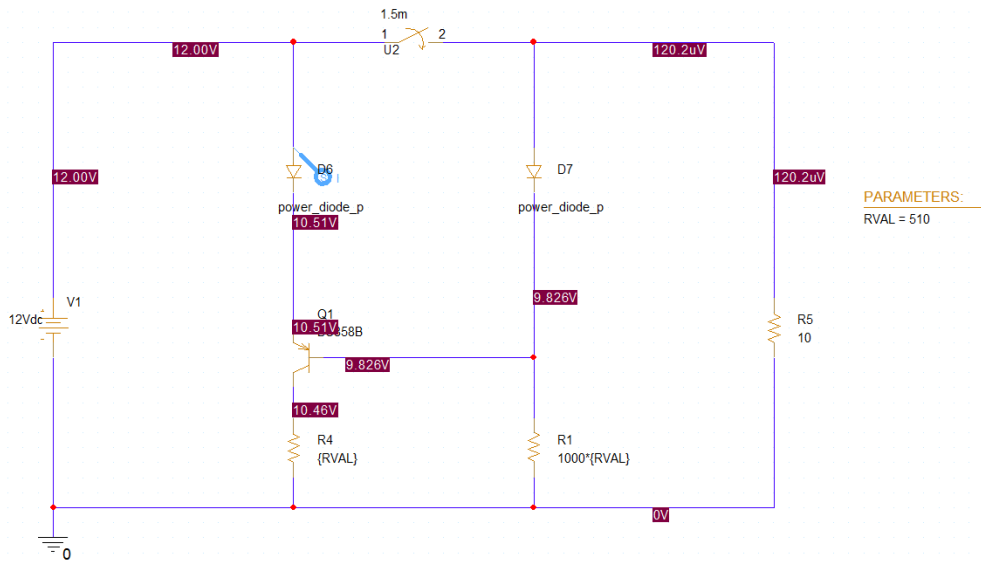


Figure 4: PSPICE Schematic for 12V Fuse-Blown Indication

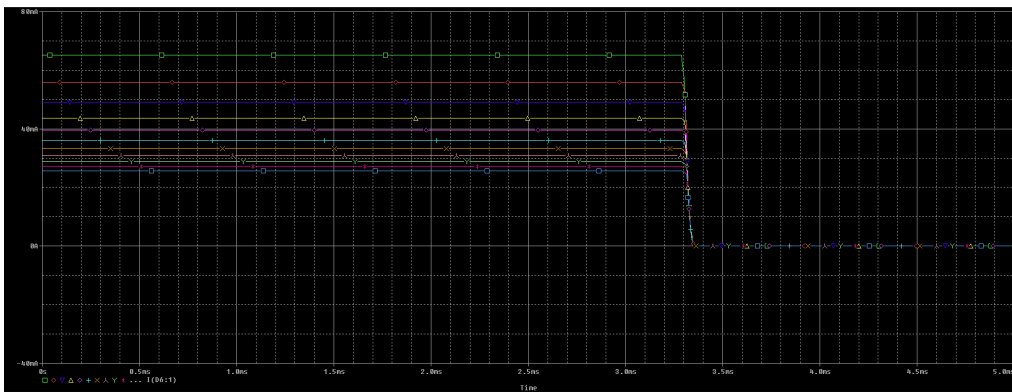


Figure 5: PSPICE Simulation for 5V Fuse-Blown Indication. Top: 50Ω. Bottom: 150Ω

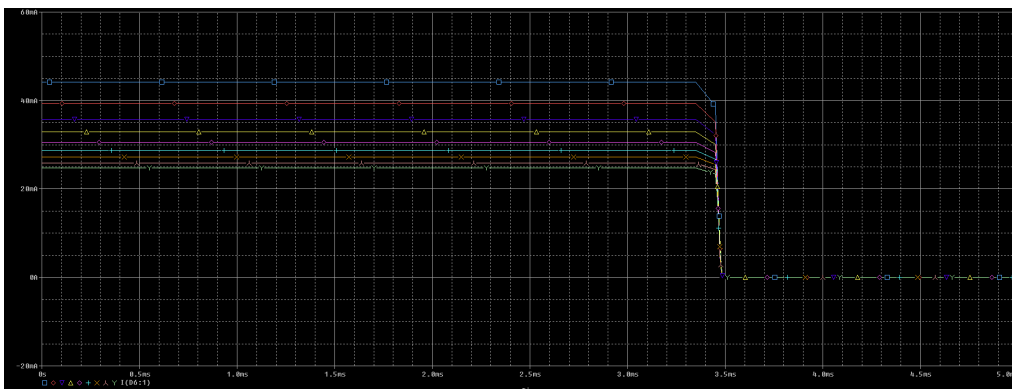


Figure 6: PSPICE Simulation for 12V Fuse-Blown Indication. Top: 300Ω. Bottom: 700Ω

Based on the parameter sweeps and common resistance values, the final RVAL resistances were 120Ω for 5V and 510Ω for 5V. This keeps the current through the LED at 30mA, which is half of its maximum.

As for the power indication circuitry, we used simple pull-down resistor configuration to drive low-power LEDs at 2mA.

6 Filtering

The TEENSY4.1 contains a voltage follower for basic filtering from the voltage dividers.

All other filters are just basic decoupling capacitors.

Following FLEX power module suggestions, the initial decoupling capacitors for each power modules have about $420\mu\text{F}$ with added low-capacitance (2.2nF), low-impedance (small 201-402 imperial size capacitors) for any very high noise. Additionally, there are not-connected capacitors going to the signal (output) ground if necessary.

For power module output decoupling, we initially placed the full output filtering on the power module board. However, we instead just kept 2.2nF caps near the output to eliminate high-frequency noise (which cannot be eliminated with long path through the connector), and then placed the remaining filtering on the backplane side. On the backplane side there are again small (2.2nF), medium ($10\mu\text{F}$), and big ($680\mu\text{F}$, could be changed to $560\mu\text{F}$), for a total of about $3400\mu\text{F}$. This value is arbitrary; we just made it feasably large.

For the 36V input power, there are only electrolytic capacitors ($2200\mu\text{F}$) at a high amount, for a total of 11mF. This is arbitrary, and we did not know how to choose it based on our motors; we did not have information to decide whether the motor noise was already properly filtered or not.

7 Grounding

This sent us on a doozy. Initially we wanted to separate the 5V and 12V output grounds into "EGND," connect them to a start point with the battery ground, and connect that up to our actual metal chassis through y capacitors.

Eventually, we settled to connect 5V and 12V outputs into "SGND." This is because many sensors we used with 12V needed the same ground as the TEENSY, which has the same ground as the 5V's ground. We renamed the battery ground to "CGND." CGND and SGND are connected at a single point connection at "OPT_FLT" (optional filtering) at the bottom of the backplane PCB. This currently has a jumper cable connected. However, if testing shows it is necessary, one can connect rated Y-caps and a high-rated $1\text{M}\Omega$ resistor for a different grounding scheme. Finally, CGND has a large pad on the bottom of the board that may be connected to an external chassis if necessary.

As usual, the TEENSY has the SGND below it and connected to it, but the connection is made through a tinier segment to isolate analog ground from the typical SGND.

8 PCB Design

Planes are hatched for thermal dissipation. Small-capacitance (2.2nF), small capacitors are kept as close as possible to the originator of noise. Here are some pictures of the design.

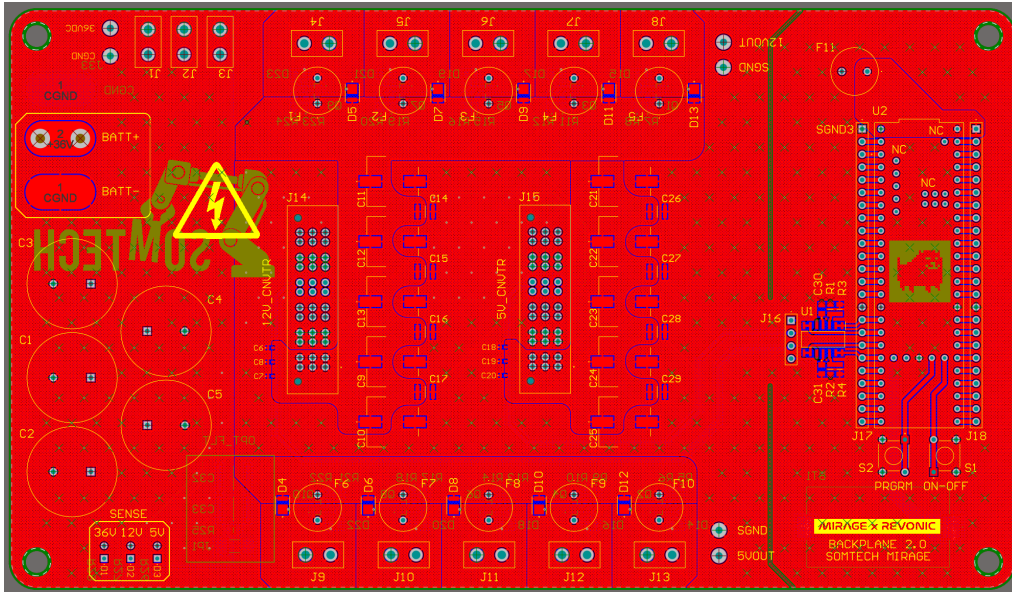


Figure 7: Backplane 2.0 main PCB flat view

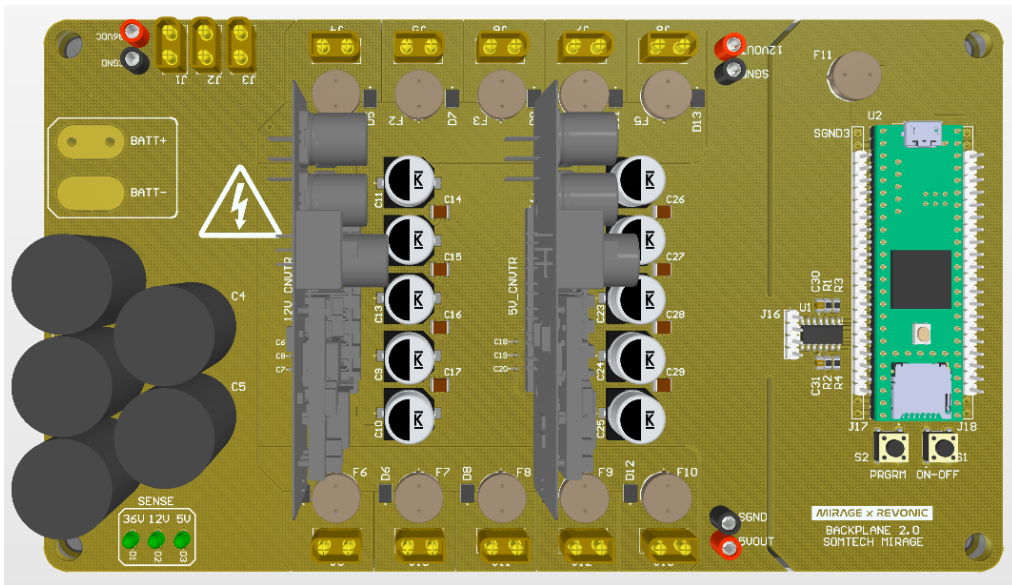


Figure 8: Backplane 2.0 main PCB top view (3D)

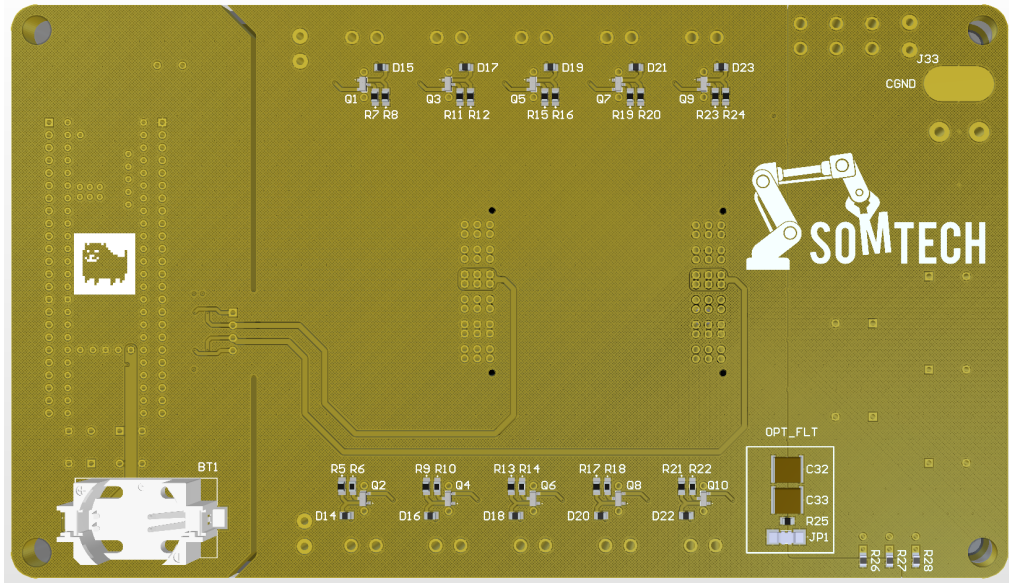


Figure 9: Backplane 2.0 main PCB bottom view (3D)

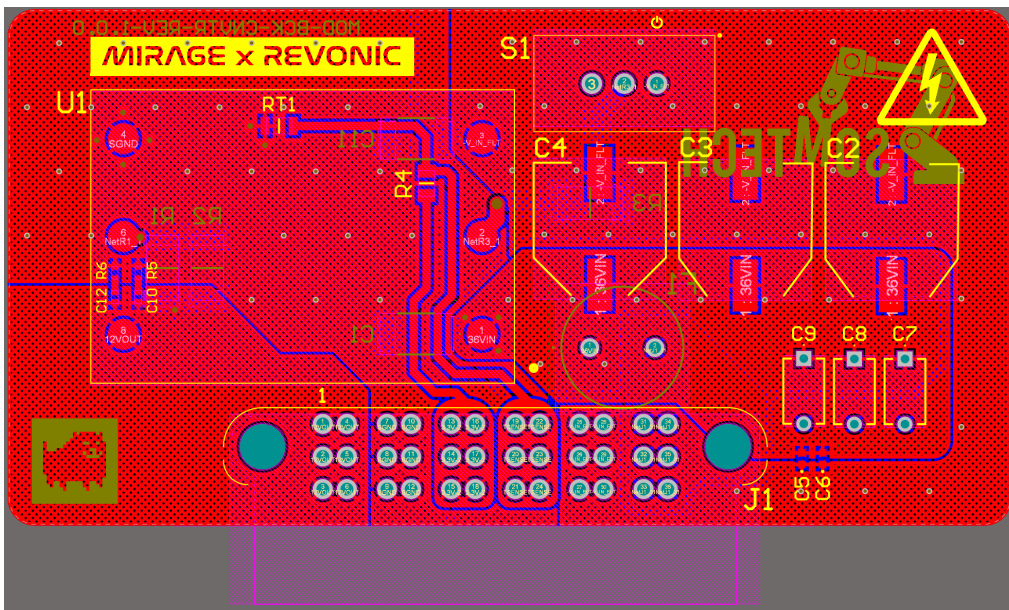


Figure 10: Power module flat view

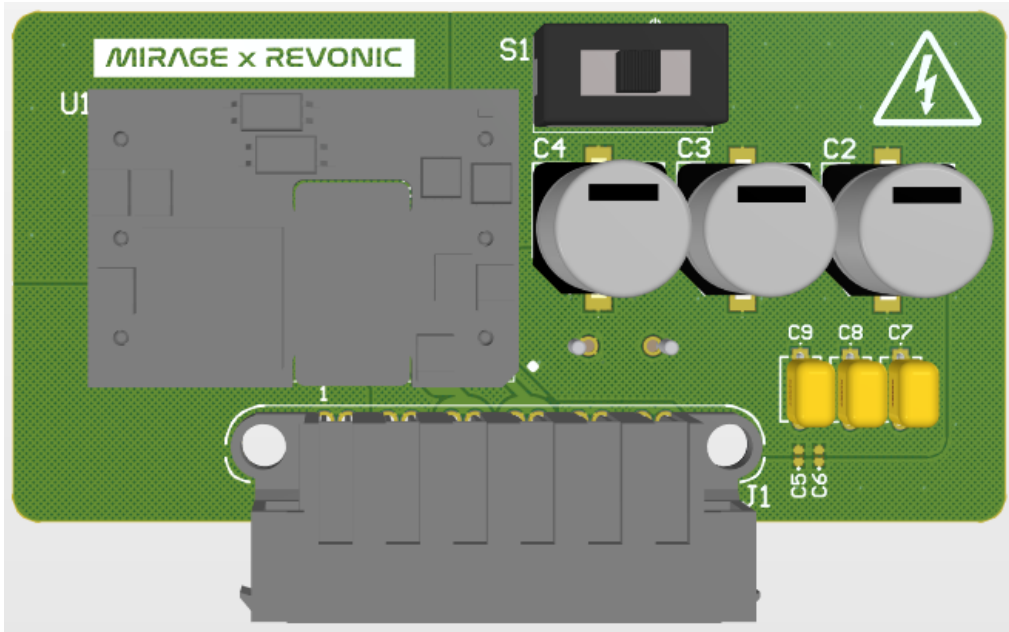


Figure 11: Power module top view (3D)

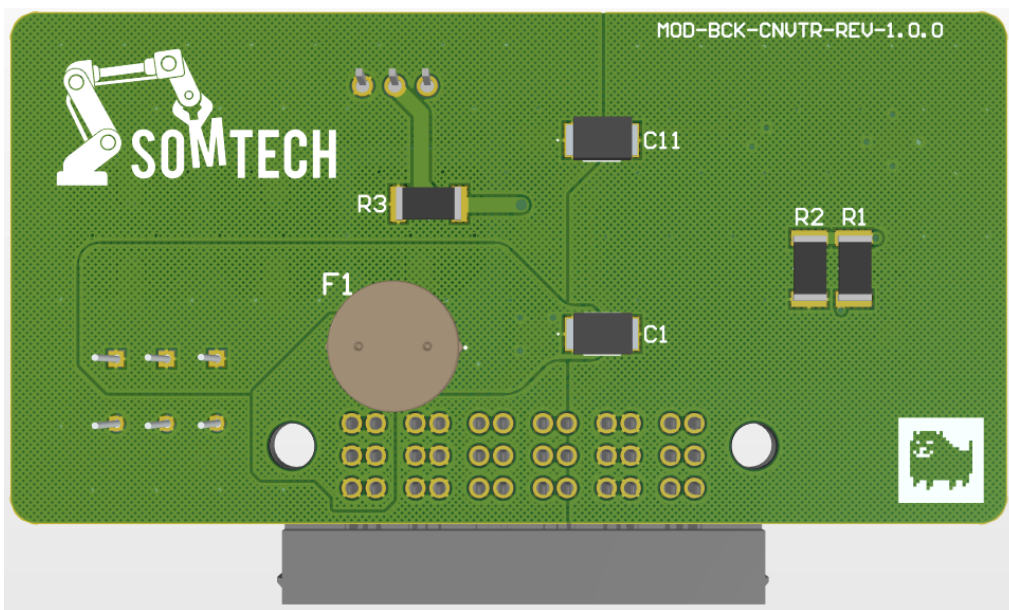


Figure 12: Power module bottom view (3D)

9 Testing

TBD.

10 Additional Documents and Links

- Backplane 2.0 Smart PDF output job - https://yoavbinyamin.github.io/Backplane_2.00OutputJob.pdf
- Power module Smart PDF output job - https://yoavbinyamin.github.io/FlexPM_PKU5513ESI_2.00OutputJob.pdf
- Society of Mechatronics Engineers - <https://somtech.mx>
- University Rover Challenge - <https://urc.marssociety.org/>