Power Distribution

General Description

The power distribution subsystem is centered around the power PCB. This is the PCB listed in the "Power Distribution" signoff. This PCB takes in an input voltage from the battery at about 12 V - 14 V, and outputs 3.3 V and 5 V along with a fused output directly from the battery to the motors. The 3.3 V and 5 V rails can output up to 5 A, while the 12 V rail can output up to 10 A.

Equipment, Parts, Software Used

Buck Converter	https://www.digikey.com/en/products/detail/texas-instruments/TPS565201DDCR/7732455
Flyback Diode	https://www.digikey.com/en/products/detail/rohm-semiconductor/RB070MM-30TR/5001862
LED	https://www.digikey.com/en/products/detail/qt-brightek-qtb/QBLP615-R/4814674
6 A Fuse	https://www.digikey.com/en/products/detail/bel-fuse-inc/0685P6000-01/10057380
10 A Fuse	https://www.digikey.com/en/products/detail/bel-fuse-inc/0685H9100-01/5466705
Screw Terminals	https://www.digikey.com/en/products/detail/molex/0397730002/9740839
Male/Female Pin Headers	https://www.digikey.com/en/products/detail/adafruit-industries-llc/5583/16688063
Emergency Stop Button	https://www.mouser.com/ProductDetail/IDEC/NWAR-27?qs=sGAEpiMZZMu9%252BhgzJMTM4pFw3ca8jvIJLNLdXi%252BpIOh5FfYzrHzz6g%3D%3D
On/Off Switch	https://www.digikey.com/en/products/detail/e-switch/ST141D00/2116289

10k Ohm Resistor	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/RMCF1206FT10K0/1759669
33k Ohm Resistor	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/RMCF1206JT33K0/1753821
56k Ohm Resistor	https://www.digikey.com/en/products/detail/stackpole-electronics-inc/RMCF1206JT56K0/1753864
0.1 uFarad Capacitor	https://www.digikey.com/en/products/detail/samsung-electro-mechanics/CL31B104KBCNNNC/3886675
10 uFarad Capacitor	https://www.digikey.com/en/products/detail/samsung-electro-mechanics/CL31A106KAHNNNE/3886733
22 uFarad Capacitor	https://www.digikey.com/en/products/detail/samsung-electro-mechanics/CL31A226KAHNNNE/3888705
2.2 uH Inductor	https://www.digikey.com/en/products/detail/tdk-corporation/SPM6530T-2R2M/1993513
3.3 uH Inductor	https://www.digikey.com/en/products/detail/tdk-corporation/SPM6530T-3R3M/1993514
Printed Circuit Board	https://jlcpcb.com/

Schematic

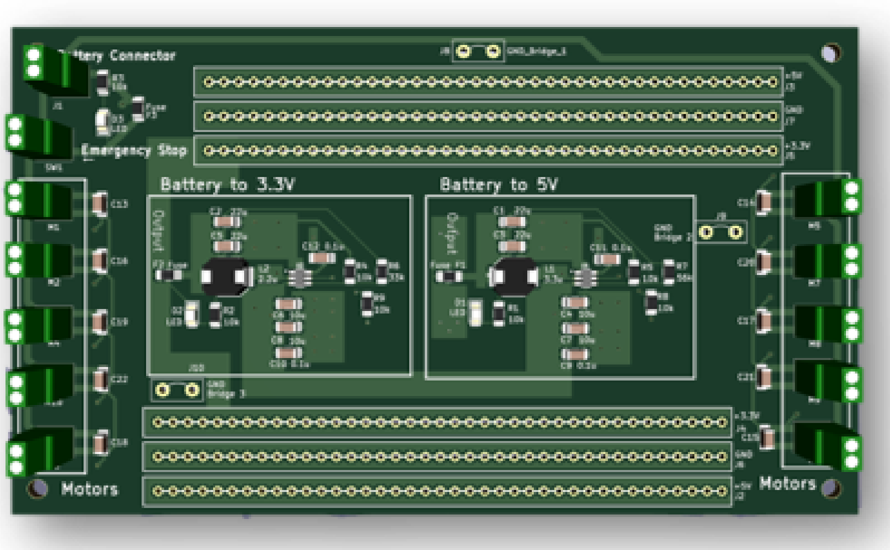

Figure 1. 3D Rendering of PCB in KiCad.

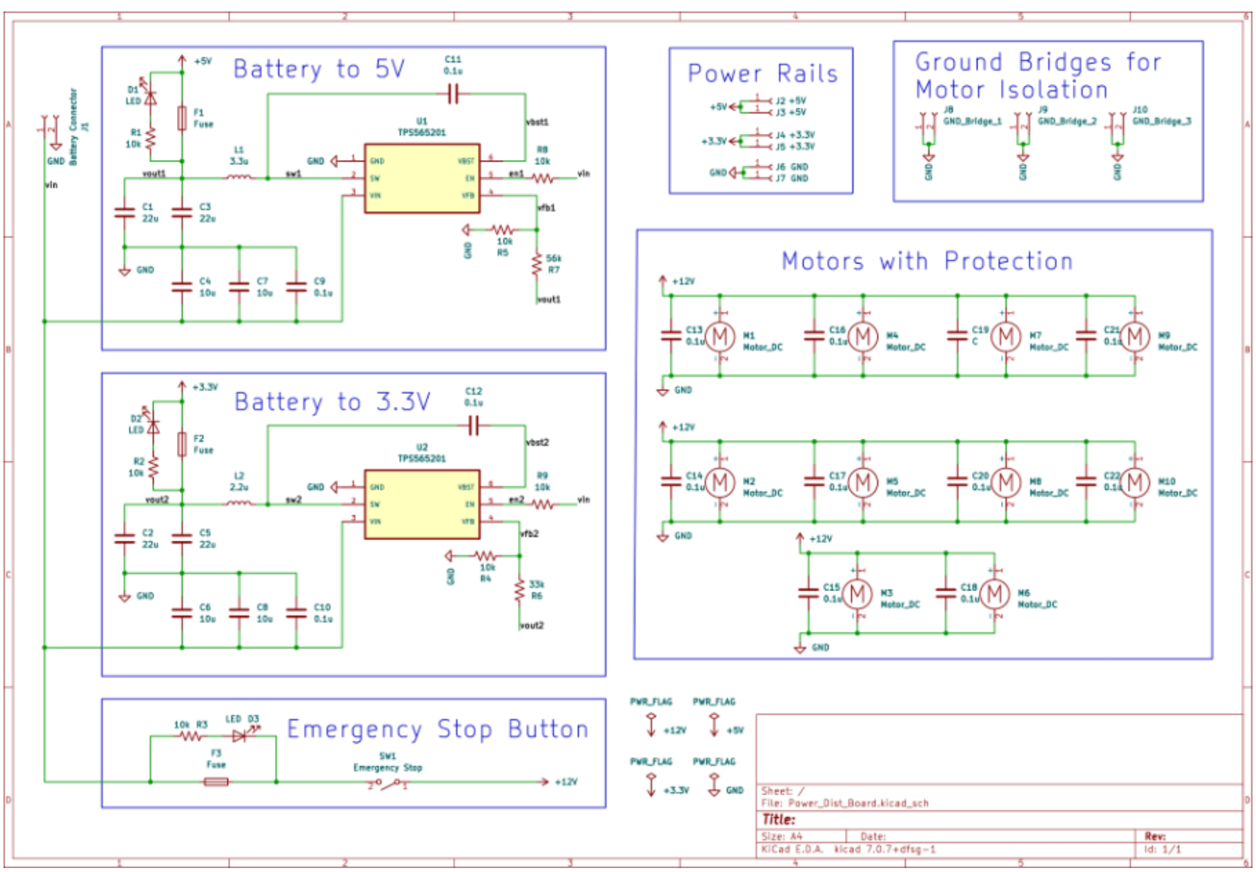

Figure 2. Schematic of Power Distribution PCB.

Logic, General Notes, Reasonings

PCB Design and Ordering

The design of the PCB was based around the voltage and current requirements of the components used in the robot. After making a list of components for testing, the worst-case currents for all of them were added up. The current requirements for each voltage rail are shown below.

Table 1. 5 V Rail Requirements.

Item	**Part Number**	**Quantity**	**Max Current Draw (mA)**	**Item Current (mA)**
Line Sensor	Pololu QTR-8RC	1	100	100
Ultrasonic Distance Sensor	SKU 101020010	4	8	32
Jetson Nano*	SKU 101020010	1	2000	2000
Arduino Mega 2560 Rev3	A000067	2	500	1000
Total Current				5132

Table 2. 3.3 V Rail Current Requirements.

Item	**Part Number**	**Quantity**	**Max Current Draw (mA)**	**Item Current (mA)**
Compass	LIS3MDL	1	0.27	0.27

Accelerometer	ADXL345	1	0.14	0.14
Total Current				0.41

Table 3. 12 V Rail Current Requirements.

Item	**Part Number**	**Quantity**	**Max Current Draw (mA)**	**Item Current (mA)**
Motor	Metal Gearmotor 37Dx73L mm 12V	4	720	2880
Motor Driver	L298N Motor Drive Controller Board	2	2000	4000
Total Current				6880

These current requirements shown above are the worst case, i.e. if everything were running at maximum power simultaneously. From this, it was suggested that the 5 V and 3.3 V rails be able to source 5 A and the 12 V rail be able to source 10 A to provide full worst-case current and some extra for future additions. A buck converter was chosen that could supply this amount of current at 3.3 V and 5 V and the PCB was designed around that chip.

The design of the PCB was based on the suggested schematic and layout given in the buck converter datasheet. The datasheet can be accessed on Digikey using the link provided in the item list above. The suggested schematic is given in the “Application and Implementation” section of the datasheet and the layout of the PCB is given in the “Layout” section. The PCB contains two of these circuits, one for the 3.3 V and one for the 5 V rails.

For these buck converters, the output voltage is entirely dependent on the resistors R1 and R2. If a different output voltage is required (e.g. 9 V), change the resistor values to the suggested values in Table 2 of the datasheet.

The PCBs are typically ordered through JLCPCB. To do the ordering, you first have to export a set of “Gerber” files from KiCad. These are put into a zip file and submitted to JLCPCB at JLCPCB.com. There is a working zip file in the github repository under the name “Power_Dist_Board.zip”, but any changes made will require generating new files. You can look up a tutorial on how to export gerber files, there is one on the KiCad website. It is not hard but there are several steps. You can also install the "Fabrication Toolkit” plugin in the Plugin and Content Manager on KiCad, which makes the process very easy.

For ordering PCBs, drag and drop the zip file onto the “Add gerber file” section on the JLCPCB website, and options will appear along with a rendered image of the PCB. Most of these can be left to default. All options can be left as default except for the following.

Required:

Outer Copper Weight: This **MUST** be set to 2 oz. This determines the thickness (height) of the copper traces, and in order to handle the required current this must be set to 2 oz instead of the default 1 oz. If 1 oz is selected, the current capacities of all traces on the board will be greatly reduced and may fail.

Optional:

PCB Qty: Default is 5, but more can be ordered if needed.

PCB Color: The color of the PCB can be changed, but the PCB will take longer to fabricate and ship. In my experience also it often seems that the silkscreen printing is not as high quality.

PCB Thickness: The thickness should not matter but may change the price of the PCB. 1.6 mm was used.

Board Assembly

When the board and components arrive, the board will need to be assembled. Most of the components are surface mount device (SMD). If you are unfamiliar with surface mount soldering, there are plenty of tutorials on how to do it. One thing of note is that a flux pen is very helpful with SMD soldering. A flux pen is not listed in the BOM but was used in board assembly. To use a flux pen, simply apply some of the flux to the pad before pre-tinning. It will clean the pad and help pull the solder onto the pad when heated. This is the flux pen that was used and it worked quite well: https://www.amazon.com/Liquid-Flux-No-Clean-10ml-0-34oz/dp/B07B53LNGX

The board is separated visually into the different subcircuits. The Battery-3.3 V and Battery-5 V circuitry have lines drawn around them and are labeled. The motor connectors also have a box surrounding them.

For most of the components on the board, their orientation does not matter. The only exceptions are the buck converters. The buck converter orientation is indicated by the dot on the chip. There is a matching dot on the board to indicate orientation and both should be on the same corner when the chip is added to the board, in the top-left corner. There are optional pads placed in parallel to the screw terminals on the 12 V rail. These can be used for capacitors or diodes (in the mono-directional motor case) for motor protection, though it is generally recommended to keep the capacitors as close to the motor terminals as possible.

There are several fuses on the board, one for each voltage output level. The largest fuse is between the battery. The resistors and LEDs in parallel with the fuses are optional. If the fuse is blown the LED will light up to indicate it.

There are three Ground Bridges on the board. These are for jumpers between the ground plane of the motors and the ground plane of everything else. The reason for this is that it keeps the grounds electrically connected but makes it harder for noise from the motors to get into the ground plane of other components. Only one of these jumpers should be used, as connecting more creates a full path for noise to flow into and out of the central ground plane. In our testing only the top one was tested and worked very well.

Connections

As labeled, the battery is connected to the board through the screw terminal on the top-left of the board. The polarity of the battery connection is shown by the plus and minus signs next to the openings. The same is done for all of the motor connectors to indicate the correct polarity of the motor connections.

On the top and bottom of the PCB are many holes for connections. The holes are spaced apart by 2.54 mm (0.1 in) (2.54 mm between the centers of each hole). This is a standard spacing for many connectors, and so any connector that has connections spaced apart by some multiple of 2.54 mm can be used. The female pin headers and screw terminals will both work for these connections or a combination of both. For higher current components, it is suggested that the screw terminals be used. The voltage and ground rails are indicated by the rectangle surrounding them and the label to the right.

To comply with SECON rules, a switch that turns the robot on/off needs to be connected. This switch must be able to handle potentially high currents and is connected in series between the battery and the board. When the switch is closed, power will flow from the battery to the PCB and then to the rest of the robot.

An emergency stop button is also required. There is a specific labeled screw terminal on the top-left of the board for the emergency stop button. Each end of the emergency stop button should be connected to one of the openings in this screw terminal. Orientation does not matter. When the emergency stop button is pressed power will cease to flow to the motors, but will remain connected to 3.3 V and 5 V buses. This is to keep the microcontrollers, sensors, etc. powered while the motors are not powered when the emergency stop button is pressed.

The motors are connected to the board using the screw terminals on either side. These terminals have boxes drawn around them and are labeled. It is also likely that the motor drivers will need to be connected this way. The voltage output at the screw terminals is fused but not regulated. This is because the voltage from the battery will range between 12 and 14.6 V, and the motors will function without issues over this range, so a regulated output is not necessary.

It is good practice to ground the chassis for safety. In the case of a broken or exposed wire touching the chassis, this will cause a short-circuit and a fuse/component to break if the chassis is grounded. If the chassis is not grounded, it will stay electrified and could shock someone if it is touched. To ground the chassis, connect a wire to the ground rail on the PCB and connect it in some way to the chassis. A screw terminal on the board would provide the best connection. Connection to the chassis could be done using a screw, copper foil tape, or some other way. Also note that the mounting holes for screws are not grounded. This was because of the position of the holes, which would cause motor noise to get into the screws and potentially into the chassis. In the case of our implementation, the extruded aluminum used for the chassis was coated in a non-conductive coating, and so the risk of electrifying the chassis was very low and so the chassis was not grounded.

Suggested Changes to the PCB:

Dedicated Battery Protection Circuitry

Upon further research into Lithium battery circuitry, I found that it is not good practice to have a load connected directly to a charger and battery at the same time. The charger tests if the battery is fully charged or not by taking current measurements, and if a load is continuously drawing current, then the charger will think that the battery is never fully charged and supply a little current to the battery continuously, which is not good for the battery. Additionally, because the battery is directly connected to the power board, some extra short-circuit protection could be added to the board and better insulated connections to the battery could be used. This would almost entirely eliminate the risk of short-circuiting the battery. Finally, some circuitry for some regulation could be done to keep the voltage into the board at a constant 12 or 13 V (or whatever is needed). This would make motor operation more consistent as the supply voltage would not change as the battery depleted.

Screw Terminals

Another suggested change is the selection of screw terminals used. The current screw terminals work fine, but the leads are prone to bending back when the terminal is hit from the front. Also, the openings are small enough that it becomes difficult to fit any stranded core wire of 16 AWG or larger, so larger openings would be helpful for high-current wires.

Reverse-Polarity Protection

Another suggested improvement is to add a series diode to the power board's input. This is to block any current if the polarity of a battery or power source is flipped, potentially causing components on the power board or the power source to fail. A little more about this is discussed here: https://www.allaboutcircuits.com/technical-articles/how-to-protect-your-circuits-using-only-a-diode/.

Fuses

It may be worthwhile to look into different fuses. The current fuses are simple one-use fuses that must be desoldered and replaced if they blow. There are types of fuses that can reset themselves after blowing that could be used. Also, it could be beneficial to have some kind of fuse-holder that is soldered to the board that holds the fuses for easier removal. The fuses have not yet blown during testing, so we never had to look into this.

Integrating Other Functionality

The final addition to the PCB that has been discussed is to begin to integrate other parts of the robot's electronics onto a single board. This could include anything, such as the motor driver circuitry, I2C buses, etc. Connectors could be available for sensors, Arduinos, Jetson Nanos, and could allow for almost the entire robot to be contained within a single PCB. This would vastly reduce the amount of wires required and could make developing new addons for the robot very streamlined.

Testing

The power PCB has undergone several tests. The results of these tests are discussed here.

Table 4. Input and Output Voltages of Power Board.

Measured Input Voltage (V)	**Top 5 V Rail (V)**	**Top 3.3 V Rail (V)**	**Bottom 5 V Rail (V)**	**Bottom 3.3 V Rail (V)**
10.0013	5.0461	3.2792	5.0466	3.2792
11.0010	5.0491	3.2798	5.0490	3.2798
12.0014	5.0508	3.2802	5.0507	3.2802
13.0011	5.0524	3.2807	5.0524	3.2808
14.0008	5.0536	3.2811	5.0536	3.2811
15.0006	5.0547	3.2813	5.0546	3.2814

The table above shows the output voltages measured for all voltage rails at both the top and bottom voltage rails for a range of input voltages. The input voltages were chosen to be larger than the range of voltages output by the battery (The battery should not go out of the 11 V - 14.6 V range).

The current capabilities of the board have also been tested. By connecting the output of the power board to a power supply in the reverse direction (where the power supply will draw current out of the board), 5 A could be drawn from the 3.3 V and 5 V rails with no issues. The power board has also been able to supply current to all of the motors running simultaneously at maximum throttle with no issues. The largest risk of drawing too much current is the stall current. If a motor stalls for some reason, it can draw a very large current, which has not yet been evaluated.

Another test done on the power board was some (attempted) destructive testing to see how many power cycles the PCB could survive. To do this, a BJT was used as an inverter to either supply power from a DC power supply to the board or shunt it to a resistor. By cycling this BJT on and off using a microcontroller, we could emulate connecting and disconnecting the power board from a battery. The block diagram for this is shown below. The Arduino for each cycle measured the voltage of the 5 V bus and was set to write to an SD card how many cycles had passed if the voltage ever dropped below 4 V. In the testing done it never did, and performed fine for over 4,000,000+ power cycles over the course of many days. For this reason, we decided to not continue as we did not have enough time and it did not provide much additional useful information.

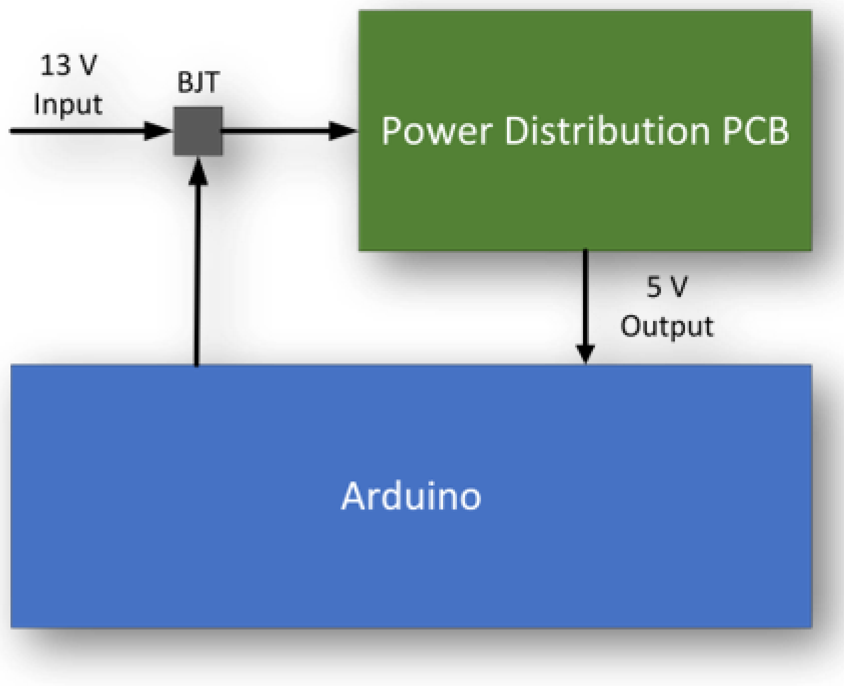

Figure 3. Experimental Setup for Destructive Testing of Power PCB.