Engineering Manual – Schematic and PCB Design

Design and implementation of mechatronic systems MR3002B

Grupo 501

Carlos Daniel López Montero - A01024473

May 29th, 2023

Instituto Tecnológico y de Estudios Superiores de Monterrey Campus Monterrey

Schematic and PCB Design

Objective

Throughout this entry to the Engineering Manual in the electronics section, the following goal is established, which was established in the document Activity 2.1. Definition of objectives, justification and expected results.

Goal 1.8. Make a preliminary design of the circuit/control board based on the previous results obtained. Manufacture a preliminary PCB and testing.

Goal 1.9. Based on the results obtained on goal 1.8, make the appropriate modifications to meet the portability, ease of use and easy maintenance objectives previously defined.

Expected results

a) First design of the schematic for the control board.
b) First design of the .brd file to manufacture the PCB.

Schematics design

It should be noted that a first mockup of the schematic layout was performed to have a general idea before designing the schematics for the final board. This mockup can be seen in the next diagram.

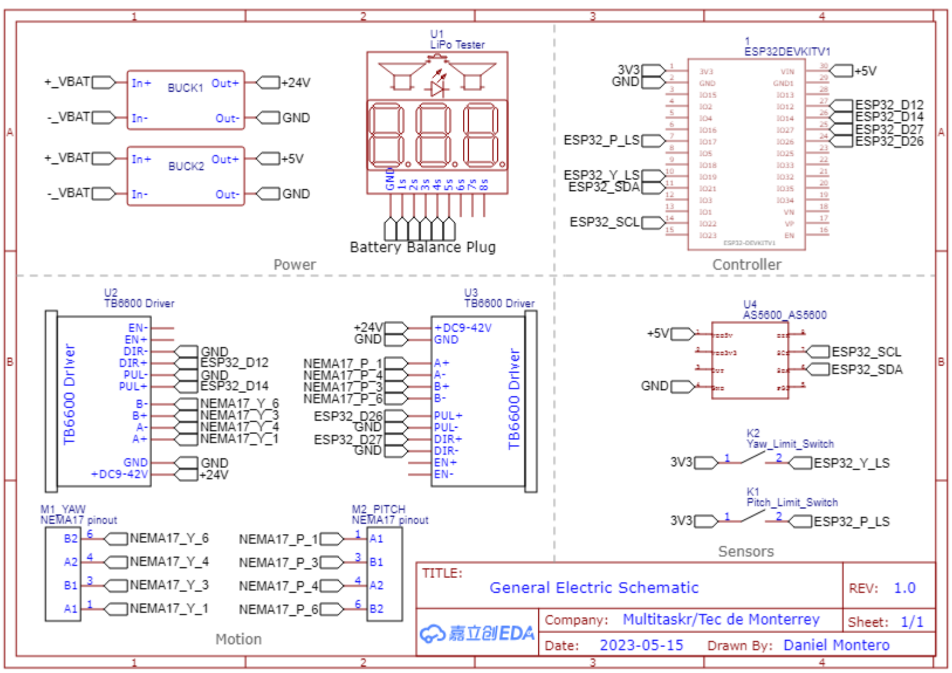

Diagram 1. General Electric Schematic for the project

Additionally, the only change that was made from the shown schematic was the addition of a second *Pitch_Limit_Switch* that's connected to the K1 switch in a parallel configuration, thus allowing to reduce the number ow wires needed on the slip ring to obtain full 360 range of motion on the yaw axis.

Once the General Electric Schematic was done, the following series of requirements were stablished and taken into consideration for the first design of the board schematic.

1) It should have an easy way of distributing/receive power from the various sources e.g.: Battery, Logic Voltage regulator and Motor Voltage Regulator.

2) It should have a way of limiting the current to be used in the circuits and provide an overcurrent protection.
3) It should expose the necessary pins/connections to be used on the microcontroller, while additionally exposing the unused pins to provide an easy way of making changes without the need of making another design.
4) All components used should be easy to obtain/replace.
5) It should incorporate a way of monitoring the health of the battery and notify the user if the charge is low enough to damage it.

The following diagrams are the first design of the board schematics and .brd file.

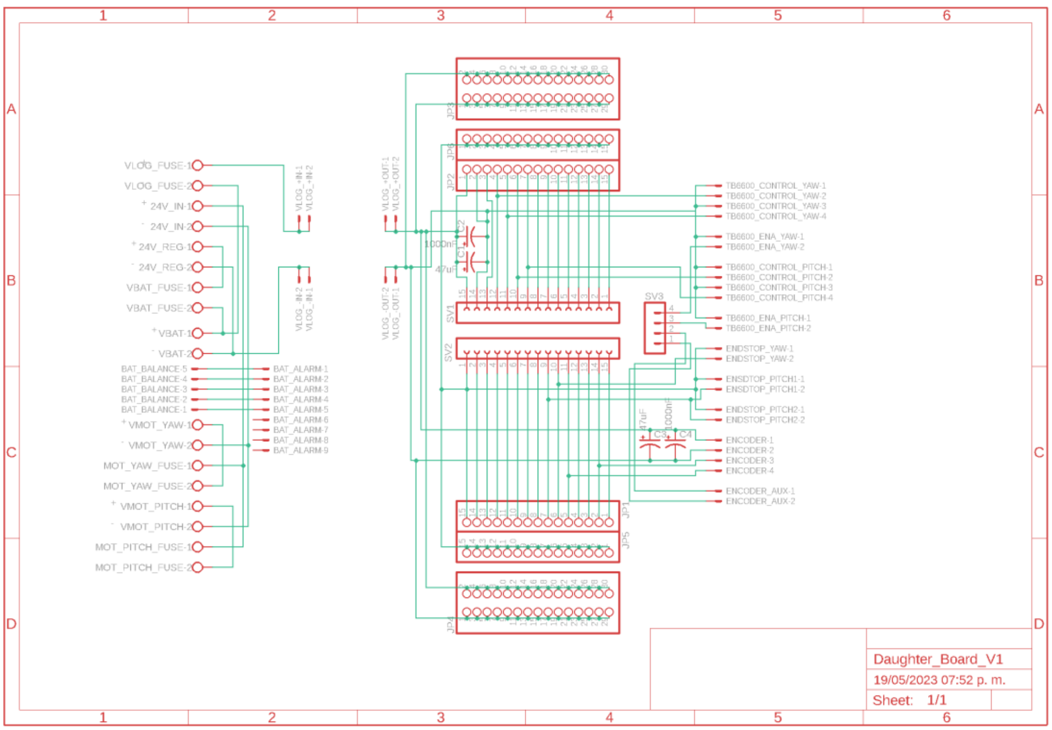

Diagram 2. First revision of the daughterboard schematics.

As it can be seen, this schematic suffices all the stablished requirements as follows:

- On the left side (coordinates 1-2) we can see the distribution power solution. This solution uses screw terminals to receive the different cables that will carry power from the battery to the voltage regulators (in the case of the 24V regulator; it also receives back the power to distribute it to the motor controllers) and adds a way to connect fuses to the circuit, protecting the voltage regulators and the motor controllers.
- On coordinates 3 and 4 we can see the microcontroller breakout (JP1 and JP2), as well as power breakouts offering 5V, 3.3V and GND connections (first row of JP4/3, JP5/6 and the second row ofJP4/3 respectively).
- On coordinates 4 and 5 we can see the designated breakouts to connect the motor controllers control pins, encoder pins, and limit switches signals.
- Lastly, we can see the battery monitoring circuit on coordinates 1 and 2, between the screw terminals of the power delivery system. This circuit takes the balance lead of the battery and offers a connection for the LiPo tester used to monitor the voltage levels of the cells on the battery.

From diagram 2, the next .brd file was designed.

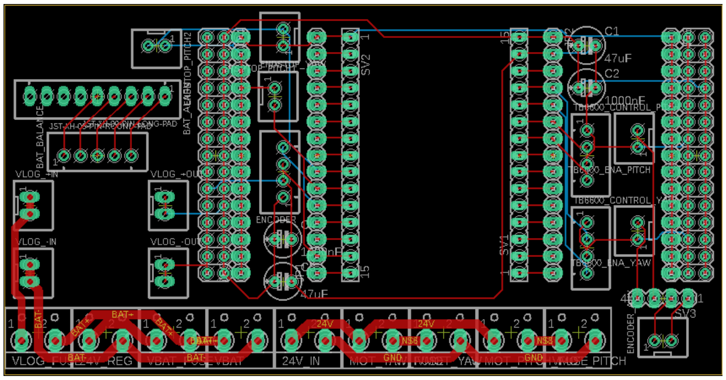

Diagram 3. First revision of the daughterboard PCB design.

It should be noted that, due to the complexity and number of connections to be made on the PCB, there are some traces that are highlighted on blue. These traces indicate traces that need to be on a second face of the copper board used to build the PCB; this not only means that manual manufacturing of the board becomes a bit trickier (it shouldn't affect the process as all connection would've been made with cables) but also means that we would need to use a specialized service to obtain a more professional result.

While manufacturing the PCB using stripboards, the method used was to first place and solder the necessary components (mainly the screw terminals, female/male headers, capacitors and JST-XH plugs) and the solder the cables. This method proved to be the least efficient and adequate as it was difficult to remelt the solder joints and add cables, as well as adding unnecessary stress on the joints and board as they were being exposed to heat again. To alleviate this problem, it was decided to simplify the board by eliminating the power and microcontroller pins breakouts, leaving only the breakouts needed for the electrical components; as well as starting on a new stripboard to obtain a cleaner and more reliable prototype of the PCB board. Below are shown the updated revisions of both the schematic and .brd file.

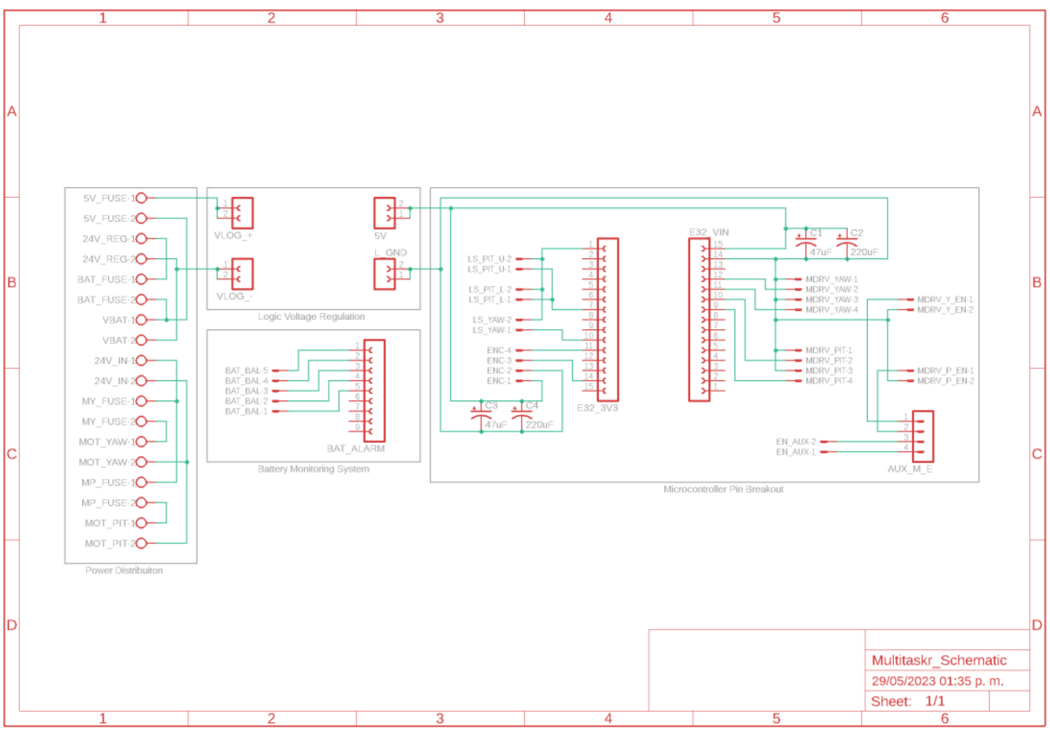

Diagram 4. Second revision of the daughterboard schematics.

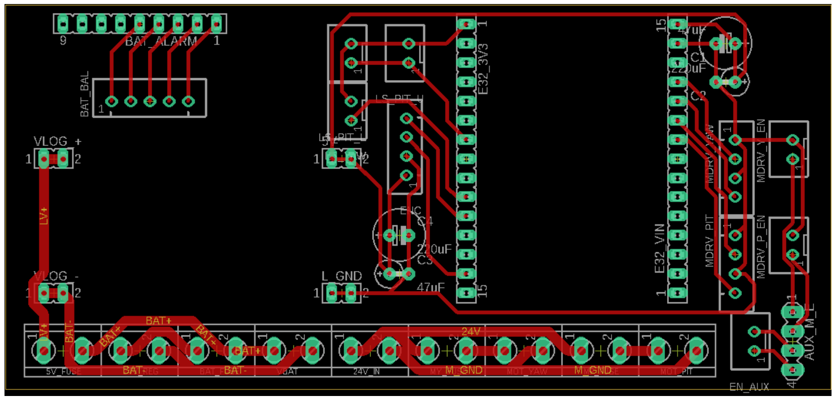

Diagram 5. Second revision of the daughterboard PCB design.

This revision aims to simplify the circuit by eliminating the power and pin breakouts present on the first revision of the schematics while maintaining the needed breakouts for the components to be used. As a result of this simplification the .brd file was also simplified, allowing us to manufacture a single face PCB and thus providing us with more options to manufacture the prototype board (mainly allowing us to use CNC and chemical based manufacturing options as well as the "manual" by cable method).