**Goals**

During this revision of Headnode, I was actively working as the BPS hardware manager. After the last revision and summer testing, I decided that this revision would be used to clean up old redundancies and unused parts of the last board, while standardizing circuits to match our other boards. Some changes include updating our 12V power circuit, removing unused charging logic, updating the MOSFET buffers to the other battery pack boards, removing unused Moduleboard circuits, and updating the circuit that drives the battery pack relays to the new power switch part, instead of the old relays. The goal was to maintain meeting all the regulations of the old board while adding quality of life improvements that would help future development of Headnode.

**Design**

In general, the largest change that was made to the 12V power circuit was the standardization of power protection that was also used on boards like Buttonboard or HAL. If you would like to learn more about this, please visit those pages on this site. Connector ports were also removed for some of the charging logic that was now not needed.

On the interface page, we removed the XOR gate that was previously used for the external charging logic. This was handled by software in the previous revision so it is now removed to avoid confusion. We also decided to update the buffer circuits to powerboard and Precharge, so that they would now both use one PMOS to buffer a 5 V signal to each board. On each respective board now, that 5 V signal would get buffered, and in turn help reduce dependencies across the boards. We had to make sure to implement these changes to each of these boards during the same revision, or else we would not have been able to turn the car on or drive the motors.

On the Moduleboard interface schematic, we decided to remove the 0-ohm jumper that was previously used to start the 5 V logic line for the Moduleboard analog fault. Since we always needed this connected, it was one less part to manage and worry about on the working revision of the board. On the regulator page, we decided to change the 5 V regulator from the previous linear regulator to the standardized switching regulator circuit that was used on other boards. This is a push I made as both the manager and Librarian for parts standardization. We also removed the 2.5 V regulator since we did not use it anymore on Headnode.

The microcontroller page remained the same, but we did update the schematic block so that there was less text on it, which made it more readable.

The relay logic page removed the 2 0 Ohm resistor jumpers that were used to select between the relay and power switches that would be used to power 12 V to the battery pack relays. After we were able to receive and test the power switch part, we determined that they were more efficient and easier to implement, which is what we did on this revision. The circuit was created from the recommended parts and connections in the datasheet. We used to logic signal mentioned previously to turn them on, with a 5 V input. We used the 12 V supply and ground to power the switch, and connected the output to the control lines used to power the inductors on the battery pack relays.

In general, this update helped to clear up lots of space on the PCB. We did not fully redo the board this time, instead opting to clean up traces where we removed parts. We did move the power switches to the top of the board instead of the bottom, which enabled for easier testing since they were now the only way of closing the battery pack relays. We also added some extra parts that were included for our power filtering circuit in the bottom left-hand side of the board.

**What I Learned**

Overall, while this revision of the board did not include huge changes, that was a good sign for how well the board worked last year. We were still able to implement quality of life improvements to the board, which were seen at the next race during FSGP 2022 in the Summer. Overall, I was glad to be involved in two back to back revisions of one board in solar car, since I was able to see the continuous improvements our team could make to one custom PCB.