PCB Layout Documentation for Rev 16

This document describes issues with the current PCB layout, and changes that should be made in the next revision. The design of the PCB started with the P08222 team using the PCB Artist software. Custom library files and all previous design files are available on EDGE. The design given to the P09222 team was not complete and had many issues, such as many connections that were not completed and parts that needed to be removed or changed. Due to the complexity of the PCB, the old design was kept but modified to meet the current design. Doing this allowed the board design to be completed more quickly at the expense of a quality design. Rev 15 of the design was fabricated by Advanced Circuits, however many changes have been made in Rev 16 (the most current schematic design). All issues in this document that have been listed as fixed in Rev 16 are correct on the schematic, but not in the Rev 16 PCB layout file. PCB Artist uses a different file for the schematic and PCB. Schematic files end with .sch and PCB files end with .pcb. When working on the schematic/layout, save often and make backups. Backup the libraries as well at regular intervals.

PCB Artist is very easy to use schematic and PCB design software. It can be downloaded from Advanced Circuits (www.4pcb.com). There are several tutorials available that show how to use the major aspects of the software.

The following are issues with the current board:
- The schematic and the PCB layout were developed in parallel. This caused many changes to be made the the PCB layout which complicated routing. To make the most efficient routing, the PCB layout should only be completed after the final schematic is known to reduce unnecessary route changes.
- Layer 2 of the PCB is a ground plane but there is no power plane. Ideally, a power plane should be created to reduce noise on the signal lines. Signals should also be isolated from the power plane and other traces that carry high current, such as the injector outputs. To make the ground plane, make a rectangular shape that covers the entire board on layer 2. Then right click on the shape, and select "Pour Copper" to fill in the shape with copper. Uncheck "Thermals on Pads", and "Thermals on Vias". When the program fills in the shape, it will automatically remove the minimum amount of copper around vias that are not connected to ground. The design files for Rev 15 show the ground plane, while the shape was removed in Rev 16 to make editing the design easier.
- The 4 transistors driving the injectors should be placed close to the Motec connector to reduce the length of the high current traces and reduce noise to other parts of the board.
- There are two traces from the microcontroller to the Motec connector that will be used for a USB connection in the future. This is desired so that the ECU can be controlled and data read from a standard interface readily available to the Formula team. Currently on the board, the two USB traces seem to go randomly around the board to their destination. If a USB bus is implemented using the traces as is, it will almost certainly not work. Due to the high speed, differential nature of the USB bus, the two data lines need to be right next to each other for their entire length. The traces must also be *exactly* the same length. Investigate the USB spec as well as layout recommendations, for example, http://www.cypress.com/?rID=12982
- JTAG has been surfaced in the Motec connector so that the case will not have to be opened to reprogram the processor. The pins routed to the JTAG are scattered on the connector and not clustered together. Ideally, the pins for the JTAG should be clustered in the same physical section of the Motec connector to make the cabling harness simpler. Investigate modifying the pinout on the Motec connector to make this wiring easier.

- The Motec connector was too close to the edge of the PCB for it to fit properly in the case in Rev 15. This has been corrected in Rev 16.
- The 0.5A fuse on the 12V input (part FS1 on the schematic) will blow when powering on the board. It is suspected that this is due to an in-rush current, but needs to be verified.
- Several zener diodes were removed from the design in Rev 15 as they are unnecessary. The Fan/Fuel Pump circuit has been simplified.
- Be careful when placing components near the JTAG connector, as it is larger than the silkscreen outline. The silkscreen should be modified to reflect the actual size of the component.
- LED's could be added to the voltage regulators to provide a visual indication that the board is powered.
- Adding a reset button to the processor would help in debugging the board.
- Place pads for a zero-ohm resistor on all unused pins to the microcontroller if possible. This will allow easy access to large pads, allowing a wire to be soldered on should the pin be needed for later use or debugging purposes. Soldering wires onto 0.5mm pins is hard.
- The pin numbering on the Motec connector was backwards in Rev 15 and earlier. The connector has a total of 60 pins which is divided into two groups (A1 through A34 and B1 through 26). The documentation from the formula team uses the A and B pin labels to specify to which pin a signal is connected. In the old schematic, the part for the Motec connector had the pins labeled incorrectly, such that pin A1 was incorrectly labeled A34, A2 was incorrectly labeled A33, and so on. The B group had the same reversal in pin numbering. The schematic part has been corrected such that the labels are mapped to the correct physical pin, however *the nets shown on the Rev 16 schematic have not yet been moved to the correct pin*. For example, according to the Formula team, the crank sensor needs to be on pin B1. On Rev 16 of the schematic, CrankIN is shown connected to pin B26. Rearrange the nets on the Rev 16 schematic such that CrankIN is connected to pin B1, CamIN connected to pin B2, etc. Use the datasheets from the Formula team to correct the schematic. Correcting the PCB layout may take a while to complete. To allow testing despite the incorrect pinout on the Motec connector, the test bench wiring has been altered such that the NIDAQ will work properly with the Rev 15 board.

**Rework Done to the Rev 15 Board:**
These are changes that have been physically made to the Rev 15 board. These changes were necessary for the proper operation of the board, and are reflected in the Rev 16 schematic.
- The processor was missing the required connections to the voltage regulator to control the RESET pin. This has been fixed in Rev 16 and wires have been added to the PCB. These wires are very fragile and care must be taken in handling the board. Specifically, pin 18 of U8 is now connected to pin 32 of U25. R14 is now a 10k pull-up, and pin 5 and 19 of U8 are connected. Pin 6 of U8 is connected to +5V through a 0-ohm resister.
- The RST pin on the JTAG needed a 10k pull-up resister to function properly (R185).
- The PLL of the processor was not operating properly, and it was discovered that a connection was missing on the microcontroller. Thus, R148 was removed from the PCB and a wire was added connecting pin 73 (PLLDIS) to ground.
- U3 was removed and the new injector circuit was added by connecting a perfboard to the PCB. This circuit was tested outside of the PCB on a breadboard, and should work correctly in the circuit. Rev 16 of the schematic does NOT reflect the new injector design.