

AeroSimPCB

PCB design for the AeroSim system avionics interface hardware

Opening the Project files in Altium Designer

For those that use Altium `.DsnWrk` "design project group" files, open `AeroSim.DsnWrk` via **File > Open Project Group....**

Otherwise...

1. Open the project file `AeroSim.PrjPcb` in Altium via **File > Open Project**
2. Open the component library `AeroSim_CustomLib.LibPkg` in Altium via **File > Open Project**

Either way, compile the component library after opening the project by right clicking on `AeroSim_CustomLib.LibPkg` in the Projects panel and selecting **Compile Integrated Library AeroSim_CustomLib.LibPkg**.

TIP: Any time a change to something in the component library is made, the component library needs to be recompiled before the component update can be applied to schematics/PCBs.

PDF Schematics

PDF copies of the board schematics are in the `ExportedSchematics` folder.

Rev B Board

Fixed all errata from Rev A Board. Please note that RevB should be used for any future board manufacturing. Relative to Rev A, the microcontroller pinout has been modified to move VG1 (control signal for discrete output 1) from microcontroller pin PB4 to pin PA8. Pin PA8 was unused previously and pin PB4 is now unused. No changes have been made to the FPGA board pinout.

Rev A Board

Manufacturing Outputs

Manufacturing output files (gerbers) sent to JLCPCB for the Rev A board are saved as `AeroSimRevA_Gerbers.zip`.

Errata

The pinout and footprint of the 3V3 LDO regulator are incorrect. This was an error in the Ultra Librarian part files. Pins 1 and 3 are swapped; Pin 1 should be the input and pin 3 should be the output. This can be worked around by bending the legs of the LDO, U2, up so as not to be connected to their pads and soldering only the GND pad of the component. Then, solder on wires from each of the 2 legs, across one another, to 5V and 3V3 net points on neighbouring components/vias.

There is a minor error in the silkscreen pinout labelling for the 28V/open discrete outputs on the D-sub connector. The textual labels "28V", "DO5", "28V", and "DO6" are correctly placed, but the brackets

indicating the pairing of each output terminal, DO5 and DO6, to a 28V supply pin are incorrect. The 28V pin at the corner of the D-sub pin array is associated with DO6, while the 28V pin between the DO5 and DO6 pins is associated with DO5. This pairing is opposite what is indicated by the bracket markers.

Discrete Output 1 exhibits a brief ~7ms period of being asserted (short to ground instead of the default open state) immediately after the board is connected to power. This is believed to be due to a microcontroller errata related to overlap in the peripheral functions of pin PB4 with the USBPD functions of the microcontroller (see [this STMicroelectronics forum post](#)). As soon as the microcontroller firmware initializes the pin states this discrete output operates normally (returning to the default open state), but nothing can be done to correct this undesired power-on behaviour. The forum post recommends adding a pull-down resistor on pin PA10 to prevent this behaviour, but the revA PCB has such a resistor and this issue is still present. As such, the recommended solution is to change the MCU pinout to avoid using pin PB4.

The footprint for U1 was incorrectly designed as a SOT-23. It should be a SOT-416. It is still manageable to manually solder the SOT-416 component to the SOT-23 pad; the pad has the same general layout and correct pin assignment, but is a bit large for the component.

Board Preview





