Space to Space Advanced EMU Radio

EV2 – Data and Command Handling Branch

Andrew Maicke - Summer 2016

The main task for this project was the development of a prototype for the Space to Space Advanced EMU Radio (SSAER). The SSAER is an updated version of the Space to Space EMU Radio (SSER), which is the current radio used by EMUs (Extravehicular Mobility Unit) for communication between suits and with the ISS. The SSER was developed in 1999, and it was desired to update the design used in the system. Importantly, besides replacing out-of-production parts it was necessary to decrease the size of the radio due to increased volume constraints with the updated Portable Life Support System (PLSS) 2.5, which will be attached on future space suits.

In particular, it was desired to fabricate a PCB for the front-end of the prototype SSAER system. Once this board was manufactured and all parts assembled, it could then be tested for quality of operation as well as compliancy with the SSER required specifications. Upon arrival, a small outline of the target system was provided, and it was my responsibility to take that outline to a finished, testable board. This board would include several stages, including frequency mixing, amplification, modulation, demodulation, and handled both the transmit and receive lines of the radio.

I developed a new design based on the old SSER system and the outline provided to me, and found parts to fit the tasks in my design. It was also important to consider the specifications of the SSER, which included the system noise figure, gain, and power consumption. Further, all parts needed to be impedance matched, and spurious signals needed to be avoided. In order to fulfill these two requirements, it was necessary to perform some calculations using a Smith Chart and excel analysis. Once all parts were selected, I drew the schematics for the system in Altium Designer. This included developing schematic symbols, as well as layout. Once the schematic was finished, it was then necessary to lay the parts out onto a PCB using Altium. Similar to the schematic design, in order to accomplish this it was necessary to develop component land patterns and add component 3D models.

All of this was achieved, and the PCB is currently in review. After it is finished being reviewed, this board will be sent out for manufacture. All electronic components used in the PCB have been acquired, and once the board arrives they will be soldered onto the board using a machine in building 44. Finally, the board will be tested for performance on-site. This will likely be accomplished by the end of the internship.