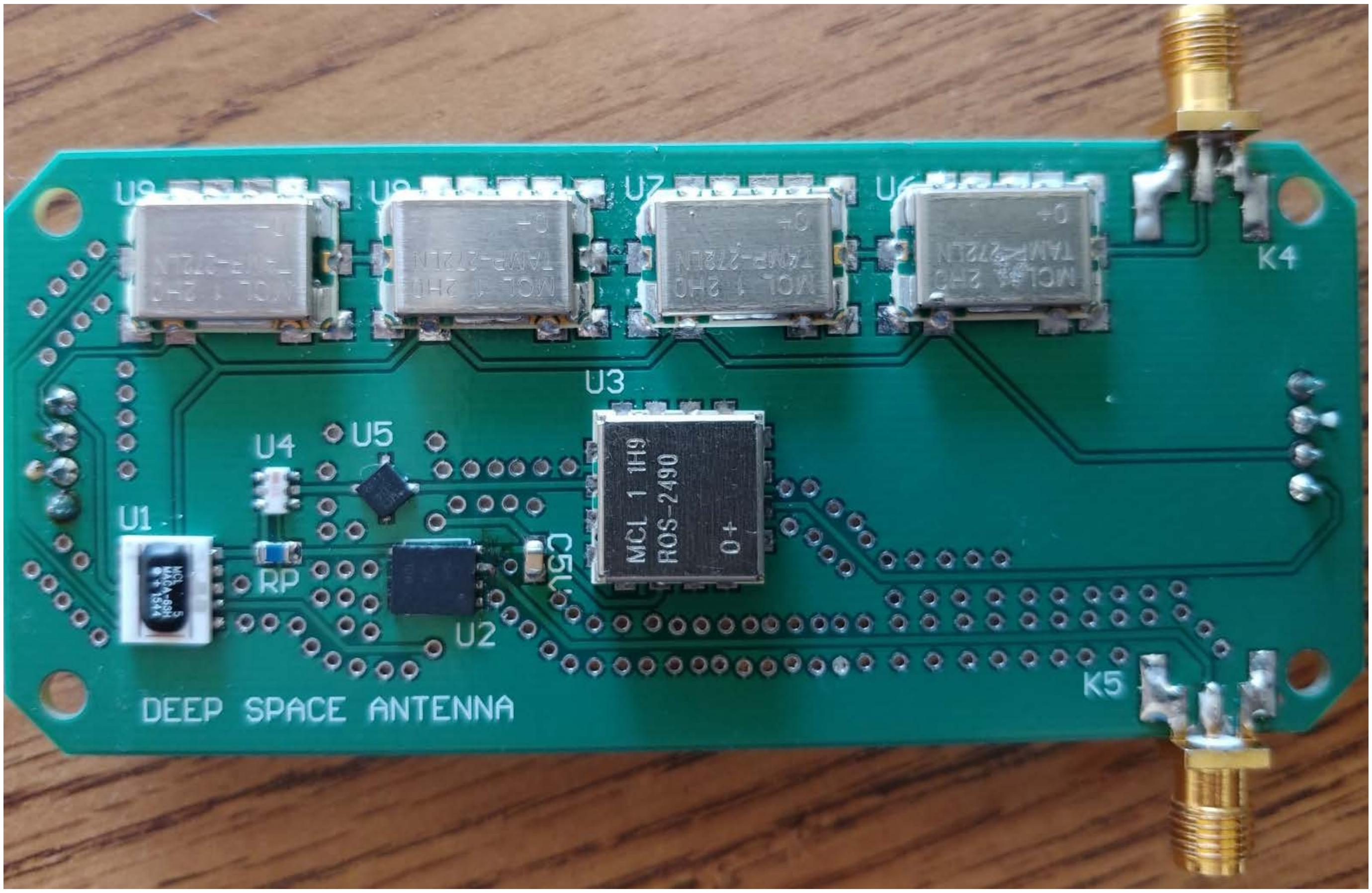
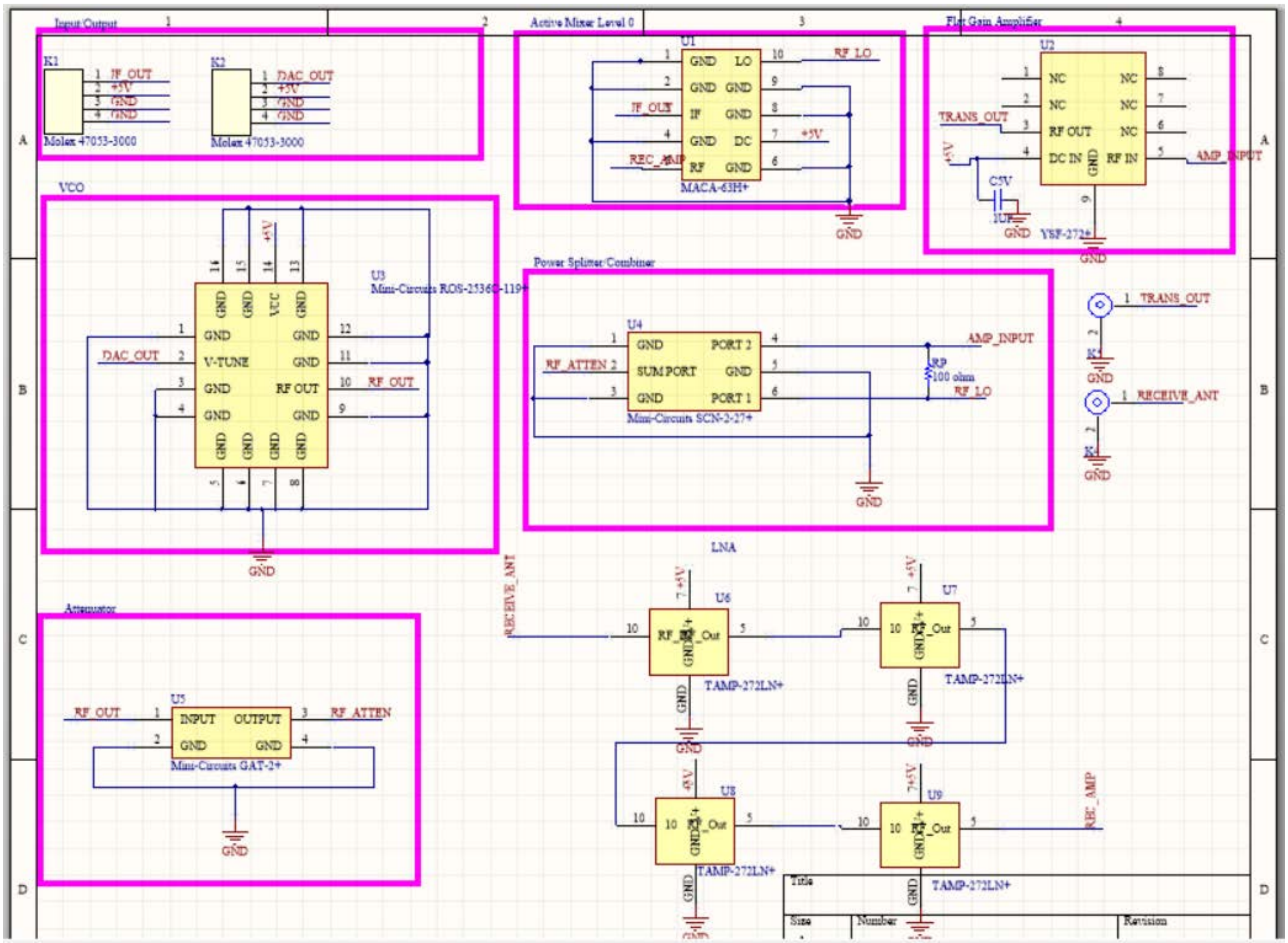
EEC 134 A/B
Application Note
RF Design
By: Christian Hernandez



Introduction:

The RF design was crucial to the system working since it was based off high frequency to get a high resolution. The RF system design needs components that behave optimally, or linearly, within a specified bandwidth. To help with this, ADISimRF can calculate the power consumption and whether or not it received RF system has passed the compression point.

Schematic Design:



To simplify the design, components that required minimal external components were prioritized. As was seen by other teams having issues, choosing parts that had complicated external schematics can complicate the PCB design to a point where it might be the wrong schematic can accidentally be implemented. If it was drawn wrong, then the PCB had to be redone which will delay testing. So to bypass all that, it is strongly recommended that components with the simplest if any, external components be chosen.

PCB layout:

As for the PCB layout, it's essential to have via's surround the traces so that coupling is minimized. The theory and reason behind adding the vias's is discussed in Professor Leo's notes. Another practical design implementation was separating the RF and Baseband layouts. The RF board should only receive the DAC out and 5V from the Base Band board. It should just send the IF signal.

Component Selection:

Using ADISimRF was helpful when determining the components. MiniCircuits also has a great section and browser when looking for the parts. Professor Leo's notes give great insight into how these components work with another and should be looked at if there is any confusion into how to select them.

Testing:

When assembling the PCB, it is a lot easier to use a heat gun when soldering the components. After all the solder has been completed make sure there are no dry sockets.

To test the transmit section, send a constant voltage to the VCO, the DAC signal, and use the spectrum analyzer to see what signal is being outputted from the VCO. It's possible to measure the signal using an exposed RF cable. When probing the output of the VCO, there will be a lower power being observed because of the mismatch between the PCB trace and the exposed RF cable. The spectrum analyzer should measure a frequency that is close what datasheet states. That measurement can determine whether the VCO is working.

The next step is connected spectrum analyze to the SMA that is supposed to connect the transmit antenna. The spectrum analyzer should accurately read something close to the output power that was calculated in ADISimRF. It should also read a frequency close the output frequency.

To test the receive portion of the RF board, input a frequency, using the TPI synthesizer, into the SMA connection that should be connected receive antenna. This frequency should be within the frequency the system is designed for. Probe the SMA connector, using an exposed cable and the spectrum analyzer. The spectrum analyzer should read a power and frequency close to what is expected. The next step would be to probe the output of each LNA to see if it is functioning. This process should be repeated for every high-frequency component. ADISimRF should give the expected power gained, or loss, after every component to get accurate measurements.

Finally, If the high-frequency components seem to be outputting the correct measurements, then connect the entire system and observe the output of the filer/gain stage. There should be a noticeable difference on the oscilloscope when a metal sheet is moved in front of the antennas.

Conclusion:

There should emphasis put into soldering the components. It seems that two of the LNA's used in the system were not working. We think it was caused by the grounds not being correctly soldered on to the board. To see if the PCB would work we sent a high frequency through the receive SMA and noticed that there was still a signal at the end of the cascaded LNA's. So even though we were not getting the gain that was expected, we were able to get the signal through which was ultimately want we wanted. Also, there probably shouldn't have been four LNA's; two would have been more than enough.

Another important thing to keep in mind is to have the SMA connector as close the edge of the board of the PCB as possible. If its not, the signal can be grounded. A way to overcome this mistake is to cut into the board exposing the substrate so that it is no longer grounded.