

## **App Note: Testing/Debugging RF System**

### **By Kevin Matsui**

It is inevitable that along the process of trying to build a working radar system, your group will run in to issues, hindering the performance of your system. Whether it is due to an incorrectly designed PCB, an unavailable part, or a burnt component, in one way or another, there will be setbacks throughout the quarter that your group will need to work around in order to get a working system. Hopefully this app note will help limit the amount of debugging needed and provide helpful tips to debug if necessary.

One important tip I wish I would have known at the beginning of the quarter is to use risers for all of the through-hole IC components on your PCB. Our group burnt out three or four DAC's on our PCB and had to de-solder and re-solder this component each time. Since the DAC we used had a maximum input voltage of 6.5 V, the component could easily be burnt out by slightly adjusting the potentiometer controlling our voltage regulator. Had we used risers initially, we would have saved hours of time not having to de-solder and re-solder new components since with risers all we would need to do is swap out the component from the riser with a new one.

With regards to debugging your system, test each block separately before putting everything together, similar to the self-guided labs in quarter 1. While it seems pretty intuitive to do so, sometimes we get ahead of ourselves and want to test everything at once and pray everything works. Instead, it is important to get each block working on its own before putting everything together. Working block by block makes debugging much easier. Say you are testing your baseband PCB by itself and the DAC output is not outputting what it should be. It is much easier to debug from here compared to hooking up your entire system and trying to figure out why it isn't working. Also, try to include test pins on your PCB to help make debugging easier. Specifically include test pins for important voltage references and outputs which would clearly indicate that your PCB is or isn't working properly.

### **Baseband PCB testing/debugging:**

Test each section of the baseband PCB separately starting with the voltage regulator and voltage reference output. Ensuring that your voltage outputs are correct to begin with provides a strong starting point to getting the rest of you PCB to work properly. An incorrect output from these reference voltages can cause permanent damage to other IC components on your PCB if the output is too high (the DAC in particular if you are using the MCP4921 which has a max input voltage of 6.5 V). If you are using risers for your through hole IC components, it is strongly recommended to remove these components from the risers when testing the voltage regulator and voltage reference.

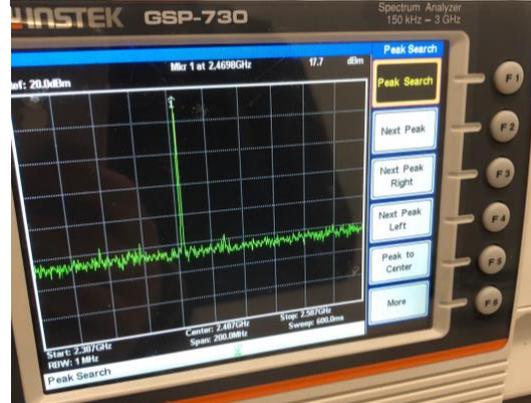
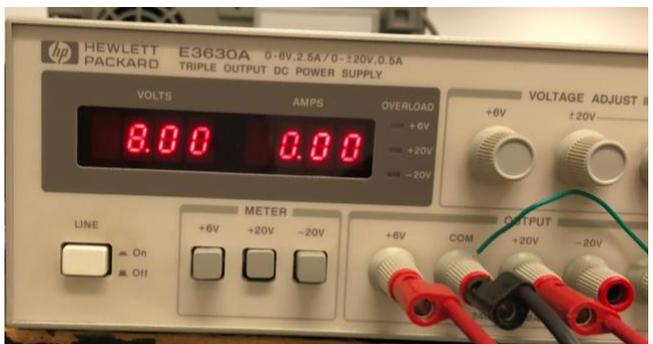
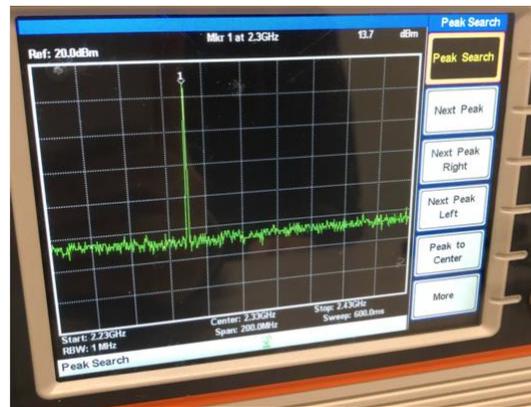
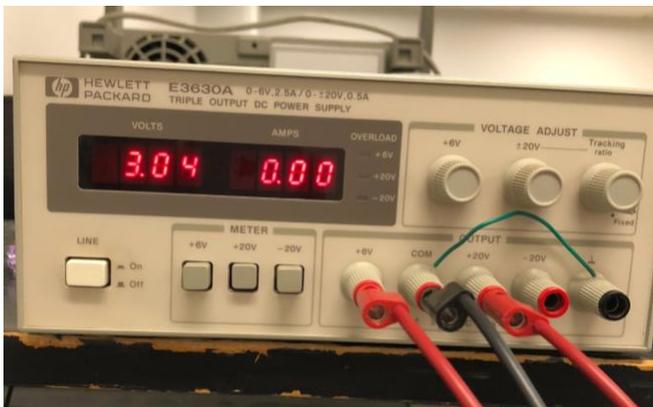
If your DAC output isn't outputting properly, it is likely that your DAC either burnt out from too large of an input voltage for an extensive period of time or it is not getting the proper reference voltages which could be an issue with your voltage regulator and voltage reference circuit.

If you are using a similar baseband amplifier/low pass filter as quarter 1, the gain stage can easily be tested separately by inputting a 1kHz small signal from a function generator to the input of the gain stage and reading the output of the gain stage with an oscilloscope. Tuning the pot and observing the amplitude on the oscilloscope changing is a good indication the gain stage is working properly.

When you run into issues with your baseband PCB regarding incorrect voltage readings or outputs, think logically and try to diagnose the issue instead of jumping to the conclusion that your PCB is broken. Many times, these issues may arise due to a shorted solder job or misplaced component and can be fixed easily. To confirm that your components are properly placed and soldered, try doing a continuity test with a multimeter across resistors and capacitors. If the multimeter beeps when the probes are placed across the component, the solder likely bridged across the pads and formed a short circuit. This is particularly common when working with closely placed SMD resistors and capacitors in the gain stage section of your PCB.

### RF PCB testing/debugging:

To test the transmit section of the RF PCB, input DC voltages to the input of your VCO and use a spectrum analyzer to measure the output power and center frequency. Compare the output power and center frequency at each DC voltage you test to the datasheet of the VCO and to your expected transmitted power calculated when designing your system. Repeat with at least 5 voltages within your expected DAC output voltage swing to make sure your VCO and power amp is working properly.



To test the receive section of the RF PCB, use the TPI Synth program on the lab computers to output a low power signal with a center frequency within the expected frequency swing of the VCO output to the input of your Receive section of the RF PCB; this signal will represent the reflected signal received by your Receive Antenna. Next, use an SMA cable stripped at one end to probe the output of your LNA and the other end attached to a spectrum analyzer to probe the input and output of your LNA and measure the output power at the amplifier output to make sure the LNA is working properly. If multiple LNA's are used, probe after each LNA to make sure the gain is constant and not saturating too early.

If you want to test the entire RF PCB at once, you can input a DC voltage within your expected DAC output voltage swing to the input of the VCO while also inputting a low power signal with a center frequency near the frequency output from the VCO at the chosen DC input and measure the output signal from the mixer with an oscilloscope. To test if the RF PCB is working, try changing the DC input to the VCO slightly and notice the change in the output frequency on the oscilloscope. Or try adjusting the center frequency of the TPI Synth slightly ( $\leq 1$  kHz) and observe any changes in frequency of the output signal. However, this method of testing may be very difficult to see clear results, as it is hard to choose a frequency for the TPI Synth to match closely to the output of your VCO (within a couple kHz) which is what is expected when the whole system is being tested. Once you have got the transmit and receive section of the RF PCB to work on its own, it might be easier to test you entire RF system with the PCB to make sure it is working properly.