Caleb Froelich
ENGR 460

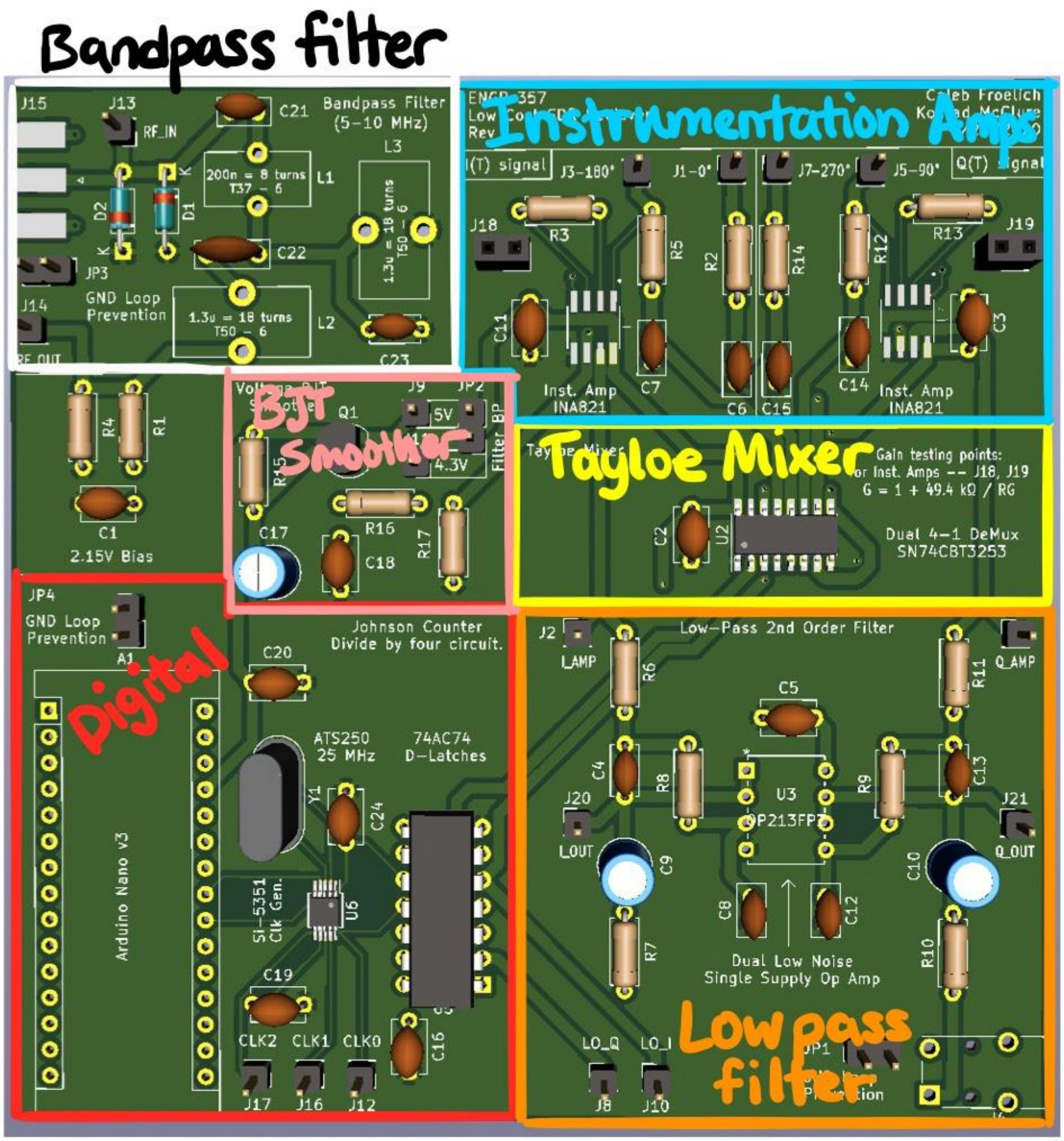
SDR Receiver – Build and Test

05/17/2020

Project Overview:

With this project, I am aiming to create an simple, low-cost software defined radio (SDR) receiver designed to operate in the (5.000 - 10.000 MHz) band. The software defined radio (SDR) is be used in conjunction with Quisk, a free open source SDR software. My project has been logically divided into five (5) sections as follows. The sections are depicted in the image below.

1. Bandpass Filter:
2. Digital Section:
3. 2^{nd} Order Low-pass Filter:
4. Instrumentation Amps:
5. Tayloe Mixer:
6. BJT Smoother:



Construction:

The board has been fully assembled except for connecting a few ends of resistors and soldering the 3.5 mm jack to the PCB. Construction of the board went fairly smoothly. I had a two issues in the construction as listed below:

- *Broke the T-37 toriod:* I don't know exactly how this happened, however, after discussion with Dr. Frohne, the problem was quickly remedied with some glue.
- *Bridge solder joint on the SI5351:* I was not as careful as I should've been when soldering the SI5351 and accidentally bridged two of the pins. This was remedied by heating some stranded wire and using it as a solder wick.

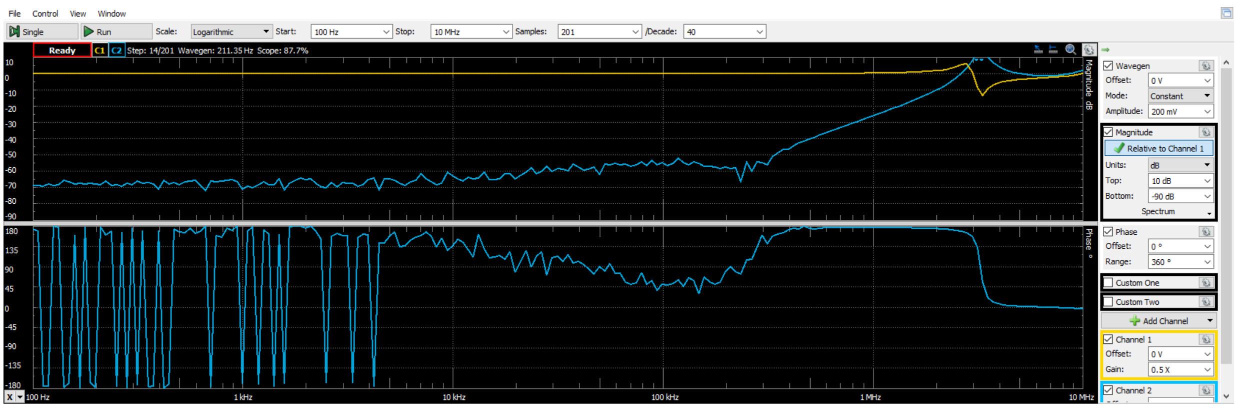
A picture of the assembled board is below. *Note: This picture was taken after testing. A few components had to be removed and some components added for the tests to pass. Pull up resistors (not shown) were soldered to the bottom of the board.*

Testing:

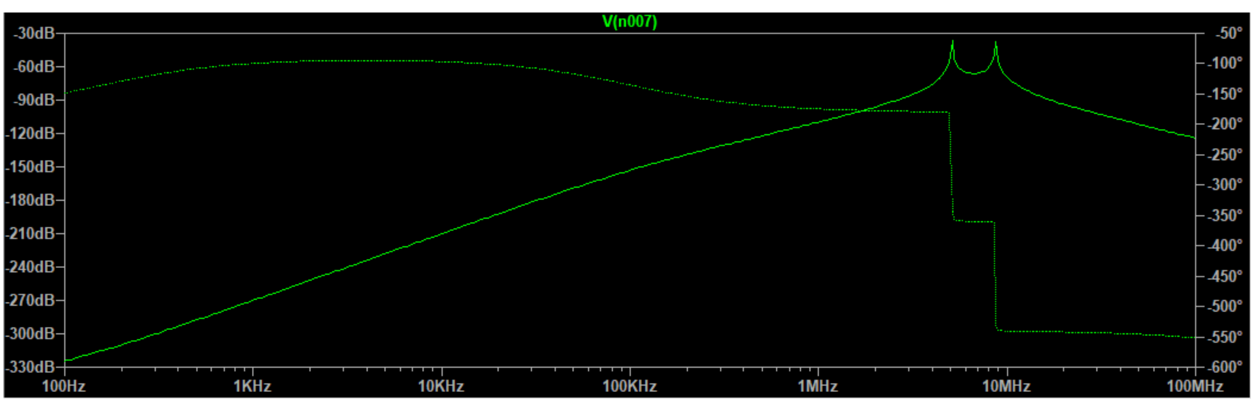
Testing of the PCB is mostly complete. I have tested my project according to the procedures described in the “Board Construction and Testing Plan” file. Below, I have provided notes as to tests performed, results of the tests and future work left on this project. I have organized tests by section as indicated below:

Bandpass Filter:

The bandpass filter tests were conducted using the Discovery 2. After verifying that there was no shorts in the power connection, I used the signal generator and scope to send a small signal in (roughly 5 mV) and measure the output voltage. I analyzed the noisiness of the signal at varying frequencies and compared the data to my LTSPICE simulations. Next, I used the network analyzer to view the transfer function of the bandpass filter. After some tweaking and fiddling, I obtained the following magnitude response.



I believe the noise in my frequency sweep was due to my low number of samples per decade. My partner sampled at a much higher rate and got a much cleaner looking Bode plot. This is fairly similar to the response obtained on LTSPICE, see below:



With these results, I concluded that the bandpass filter was working as desired.

Caleb Froelich
ENGR 460

SDR Receiver – Build and Test

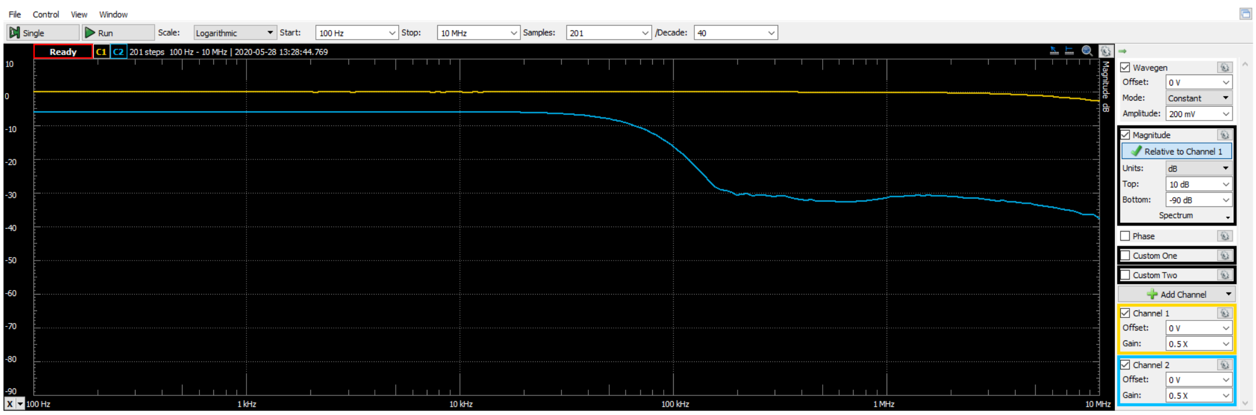
05/17/2020

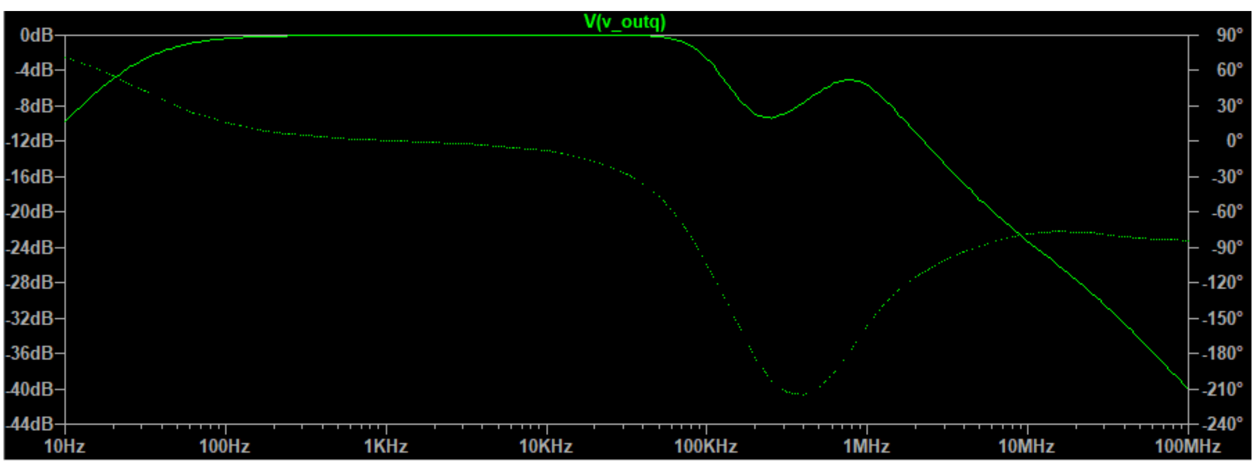
Digital Section:

The digital section has not yet been tested. Currently, I have flashed my Arduino and have removed the D-latches from the board to isolate just the Arduino and the SI5351. Tomorrow morning, I plan on running tests to see if I can get the clock generator up and running.

2nd Order Low-pass Filter:

The testing of the 2nd order low-pass filter was very similar to the bandpass filter tests. For these tests, I isolated the filter by removing the ends of R6 and R11 on the instrumentation amplifier side, such that I could send a signal directly through the low-pass filter and not have to worry about other parts of my circuit disrupting my analysis. I powered the OP213 IC directly from the supply on my Discovery 2. I again started by sending a single sinusoidal wave through and verifying that I could get an output from the filter. I then switched directly to obtaining a transfer function with the network generator. The result is compared below with the design in LTSPICE. Overall, the filter looked really good.





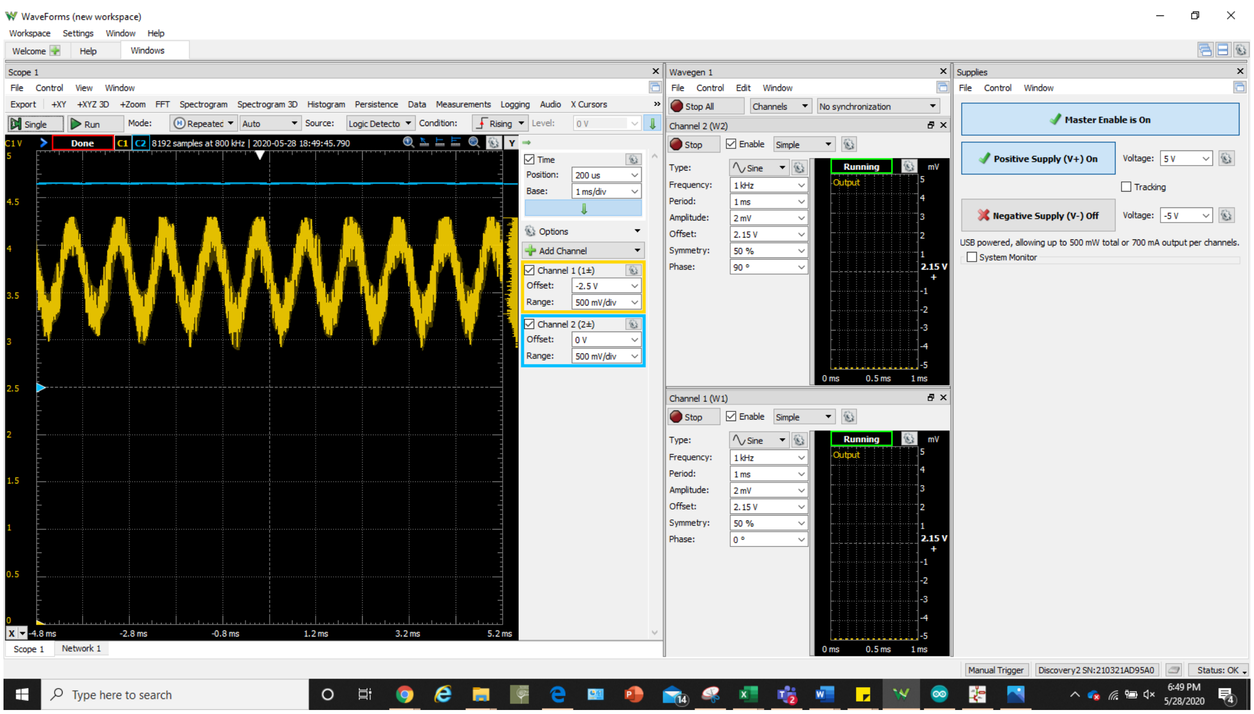
Caleb Froelich
ENGR 460

SDR Receiver – Build and Test

05/17/2020

Instrumentation Amps:

The instrumentation amps were tested to analyze the differential gain. By sending two signals through the I_IN0 and I_IN180 test points, I was able to view the differential output of one of the instrumentation amps. With a gain resistor of 120.5 ohms, I got a gain of 125 V/V. This was considerably less than the predicted gain. The gain equation provided in the data sheet predicted: 1 + (49.4*10^3) / 120 = 412.67 V/V. Interestingly, the actual results were closer to that measured in LTSPICE. The output from the differential amps were also quite noisy. At this juncture, I am not going to focus on the source of the noise. I will save this step until each of the individual section tests have been finished. The image below shows one of the two inputs (in blue) and the output (in yellow) of the instrumentation amplifier. The input signal is 2 mV in amplitude.



Tayloe Mixer:

The Tayloe mixer has not been tested yet. In our testing plan we didn't have very many tests defined and I couldn't think of anything that I wanted to try. Thus, I am waiting till I hook everything up to see if it works. Kind of sketchy I know, but there's not a whole lot involved and I'm not sure how to test just the mixer itself with my current equipment.

Voltage BJT Smoother:

I had a lot of problems getting the BJT smoother to work. Initially, I soldered the 2n2222 on backwards, giving me less than 4.3 volts on the emitter. I had been quite careful to check a data sheet for the 2n2222, however, the one I checked was not exactly the same as the one I had purchased and had a different pinout. Thankfully, I was able to unsolder the transistor and switch its orientation without too much effort. This solved the problem of not having 4.3 volts on the emitter, however, my AGND was not correct. I was measuring around 1.6 V. Unfortunately, this problem was caused by a poor design and was not as easily fixed. The main issue was that the reference pin of the instrumentation amps was drawing current that should have been flowing through the voltage divider. Analyzing the internal schematic showed us that the input impedance was not a large as anticipated.

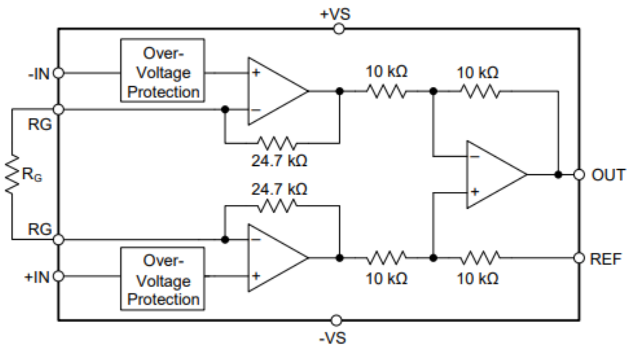
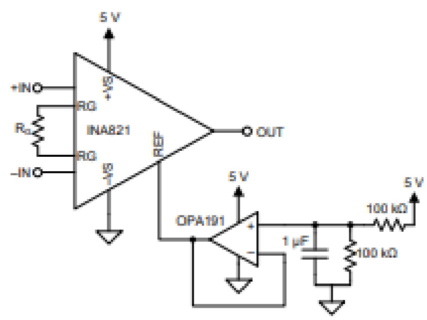


Figure 1 - Internal Schematic of IN821

Disconnecting the lower resistor in the voltage divider and routing directly to the IN821's reference pin, provided the correct AGND voltage as the resistance at the reference pin was in series with the internal 10-kΩ resistor. However, the datasheet warned that without a small source impedance, the common-mode rejection ratio would be degraded. Thus, I chose to buffer the divider with a voltage follower using the following circuit topology.



To accomplish this, I removed R16 and R17 and soldered some jumper wires to my board so that I could connect a voltage follower utilizing a spare LM741 that I had in stock. The addition of this buffer circuit worked well and solved my AGND issues. I plan on coming up with a solution that allows me to connect the op amp directly to my board, however, this will take place after testing has been finalized.

Future work:

Currently, the next steps in my project are as follows:

1. Test clock generator and verify correct operation.
2. Test Johnson Counter and verify correct operation.
3. Check multiplexer output while it is still disconnected from instrumentation amplifiers. Verify correct operation.
4. Connect all the sections and test with Discovery 2.

I have not completely thought through testing protocol for the whole circuit board. I'll think through this tomorrow and try to have a solid plan done before I start tests on the complete board.