Technical note: Antenna Selection and Design

Alexander Coffman

For the antenna system my group initially decided that a balance of directivity and gain was of primary importance. This first led to the search for Yagi antennas for superior directivity while maintaining high gain. For these benefits the antennas required us to sacrifice weight. However all the Yagi antennas found were significantly heavier than hopped and after careful consideration it became apparent that weight should also be a factor in consideration of antenna choice. After coming to this realization (that many Yagi antennas weight were taken in units of multiple lbs when we hopped for a number of grams not exceeding 200) we changed our antenna focus to be one that was more lightweight.

This led us to the decision of a PCB antenna and most of the quarter was focused on selection and design of superior patch and/or Yagi antennas on PCB. Patch antennas are oftentimes smaller and much more lightweight but have poor directivity while Yagi are bulkier and heavier but with much better directivity. From the previous quarter coffee can horn antennas were kept, as backups for the system in case the Patch or Yagi PCB did not work out.

It is also important to determine if you will purchase the antenna system of design and built it yourself. The first step in designing a PCB Antenna is to simulate the antenna in a high frequency electromagnetic field simulation, HFSS. Once this has been done and we know an antenna will work according to simulations the antenna could be made in eagle PCB design software. Since experience with antenna design was needed to create the antennas from scratch pre-made designs were used as templates with the constraints that PCB types should match those being ordered from Bay Area circuits. Several of the requirements for the Bay Area Circuits PCB Design are that it may only contain two layers and the board is printed onto FR4 substrate. This are important points to note because they limit the designs for antennas on PCB's that can be used.

Because I had no experience using HFSS, as a group we decided it would be better to purchase a set of antenna systems or use pre-simulated designs and spend time figuring out which would work best for the competition. Initially, the plan was to submit a design simulated by Texas Instruments for a multi-layer Yagi PCB with supplied specs and dimensions shown below. It made designing the PCB in Eagle much easier and ensured they would work properly in functioning whereas if I had designed this it may not have worked this way.

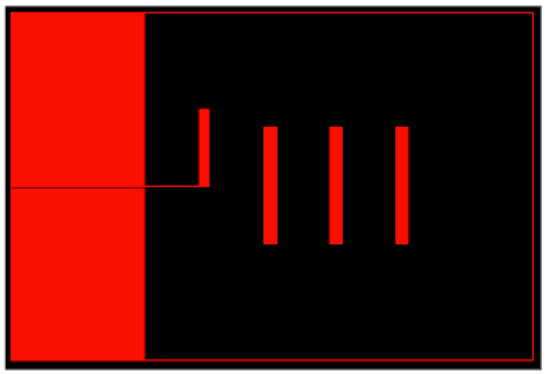


Figure 3. Layer 3 (Top, Antenna Top Layer)

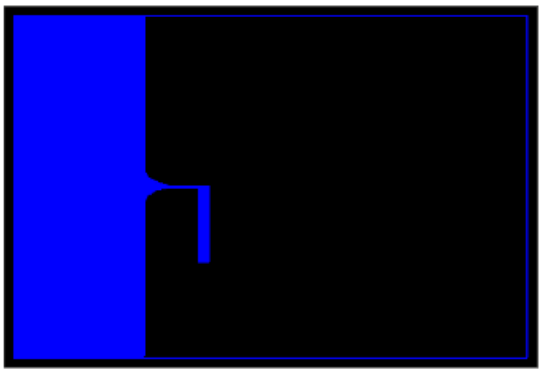
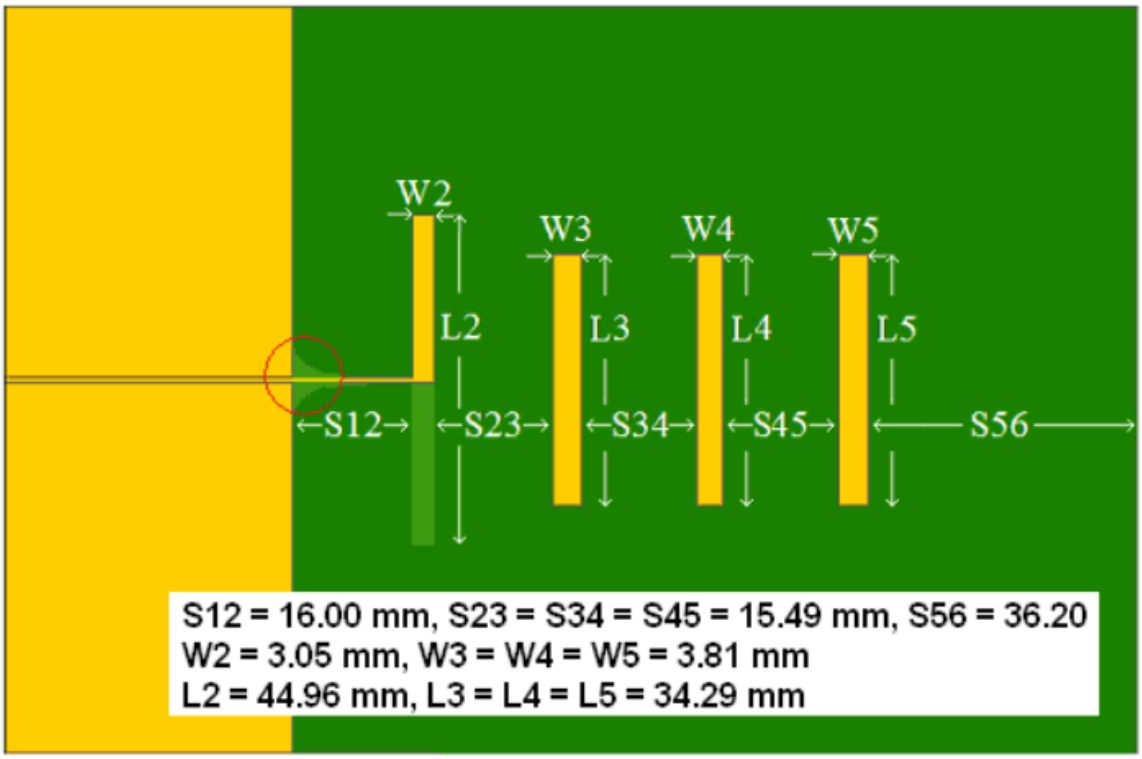
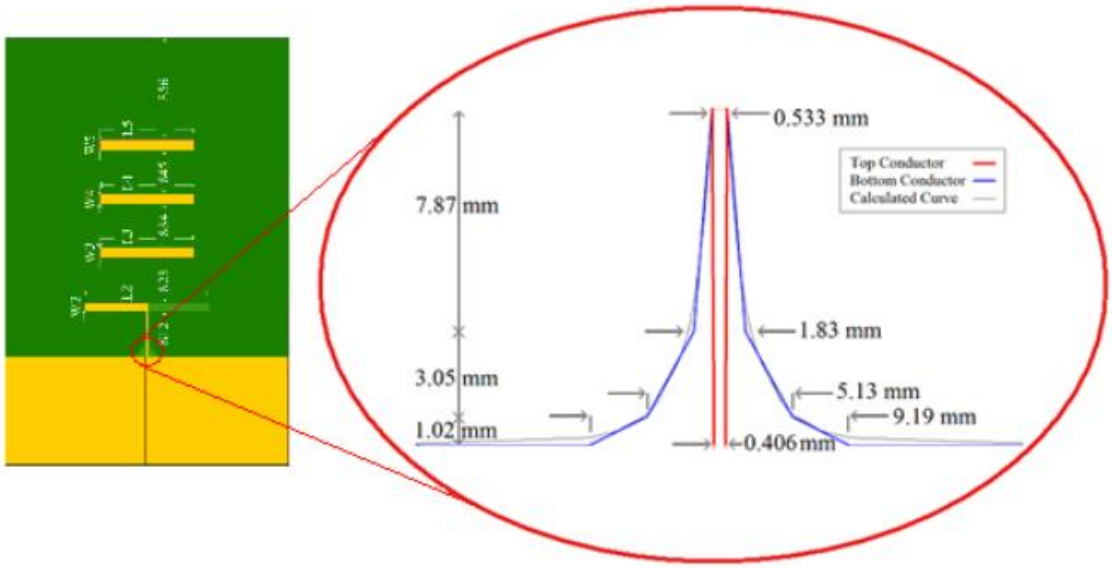
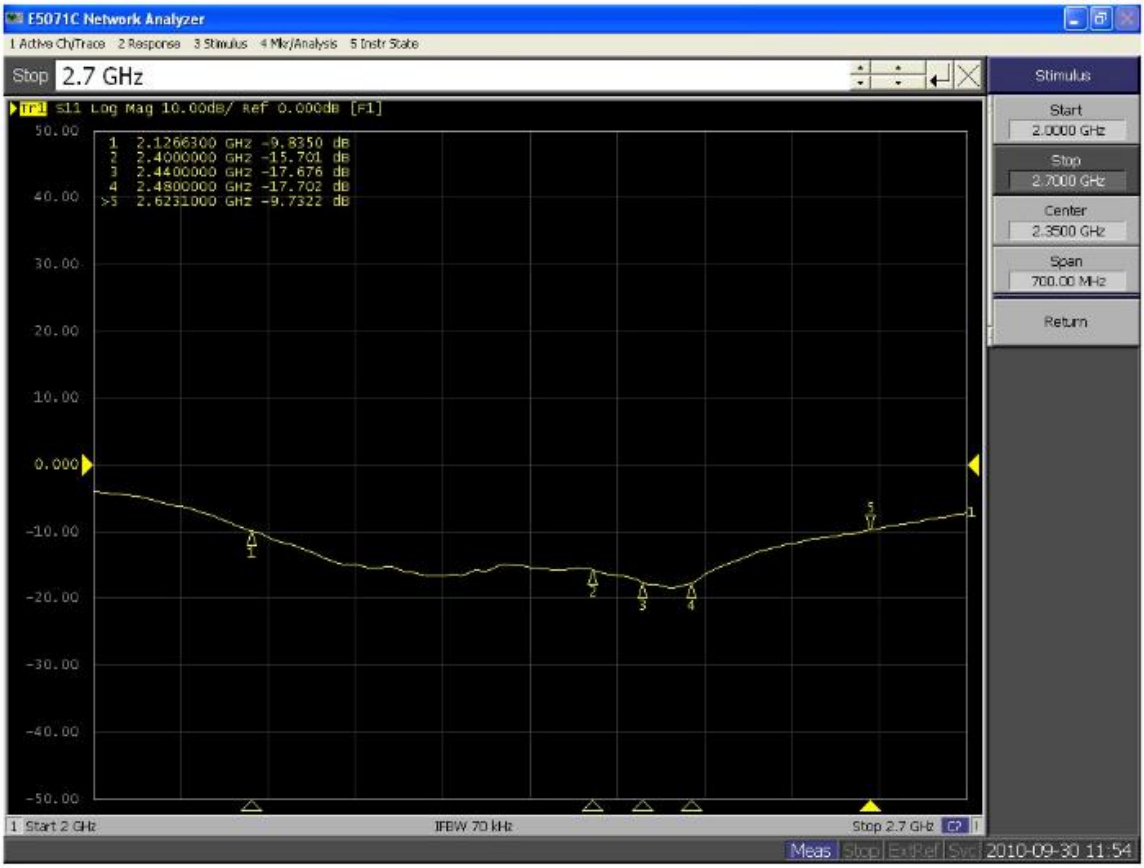


Figure 4. Layer 2 (Inner, Antenna Bottom Layer)

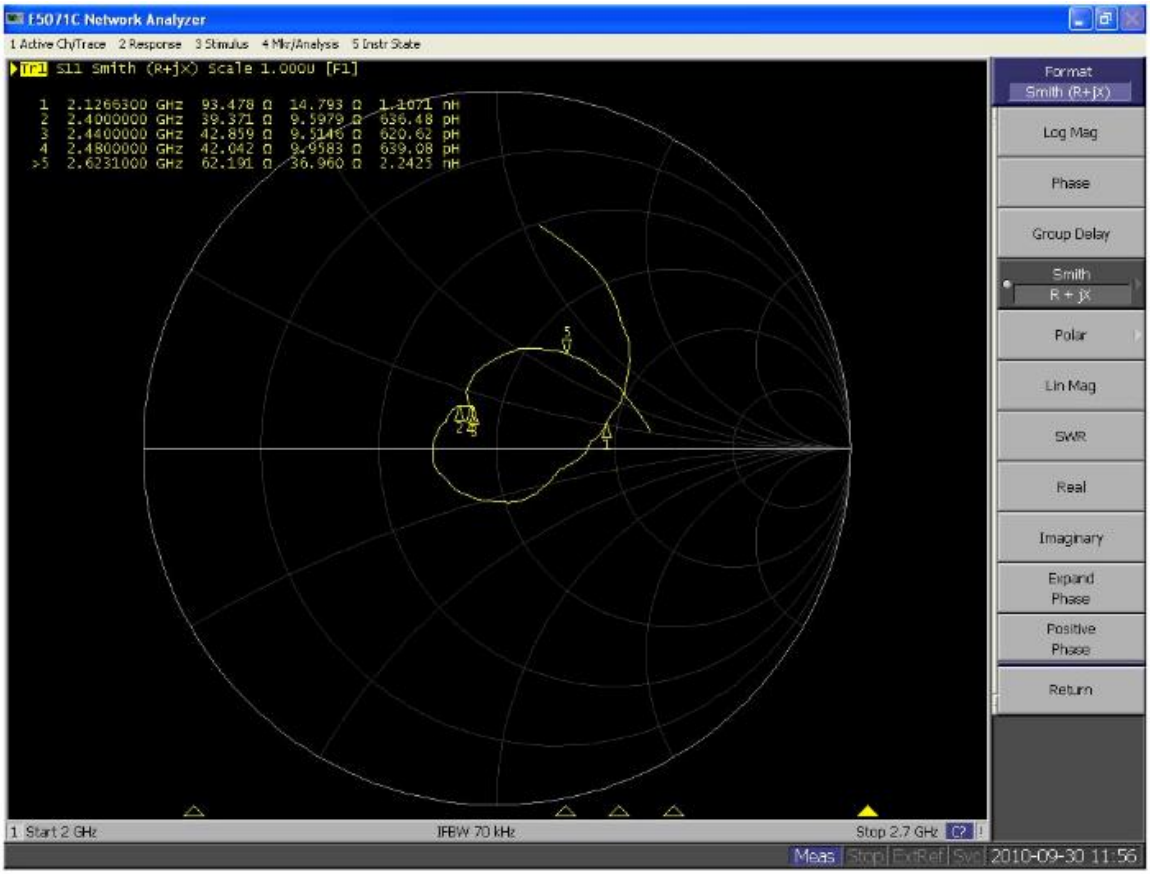




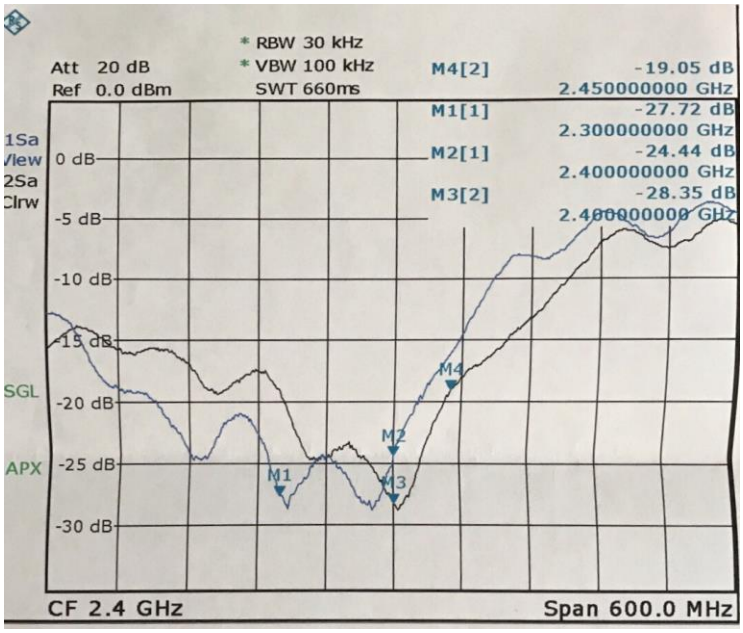
This antenna was ideal for our purposes because of the PCB it was lightweight and because of the Yagi 2.4GHz design it was highly directive and of the appropriate frequency. The bandwidth was also sufficiently large and met our requirements.



Additionally, the impedance match for 2.4GHz of this antenna is within VSWR 1.4



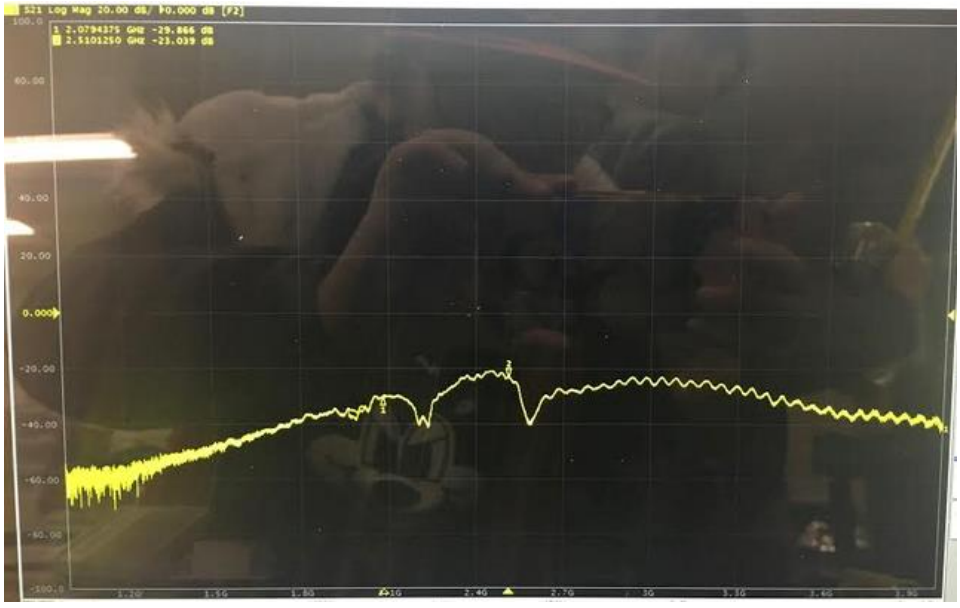
In the end unfortunately the antenna did not end up working for our purposes because several of the requirements for ordering from Bay Area Circuits student PCB were not met. Two other antennas were considered which we actually could acquire, one backup Yagi PCB antenna provided by the TA and one Commercial Patch antenna used by another group (the Alfa).



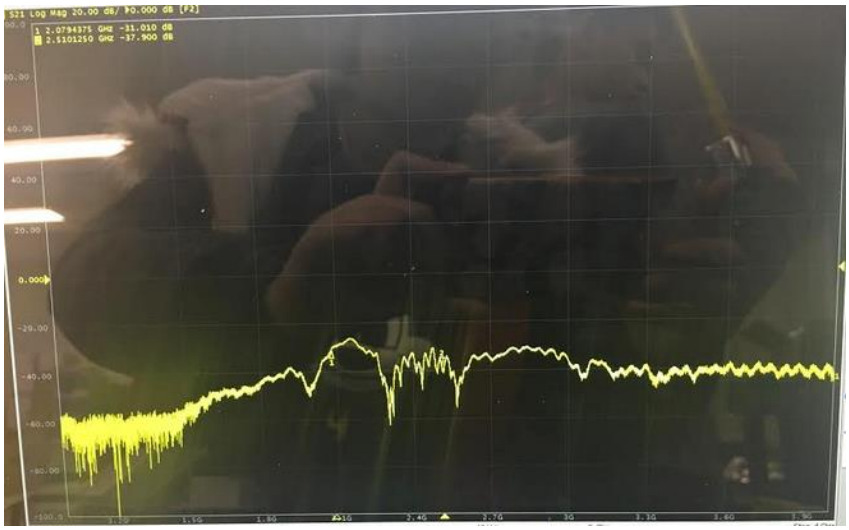
The purchased Yagi antenna had the above shown return loss parameters, which indicated it would meet our needs and was much more efficient than the TI antenna. In addition to the decreased directivity and gain of this antenna, coupling was also an issue for us, of which we measured in the lab with the help of the TA and another group. The following graphs show the coupling of the antennas at varying distances apart from one another.

1 Inch

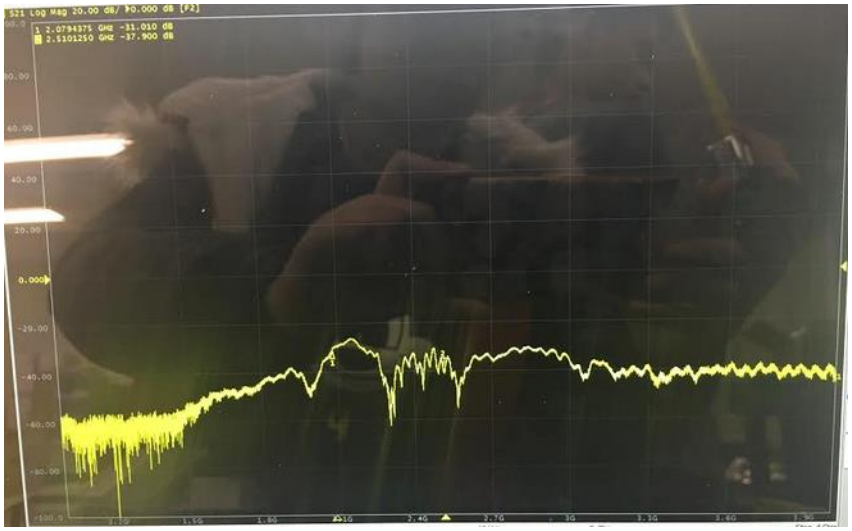
2 Inches



5 Inches



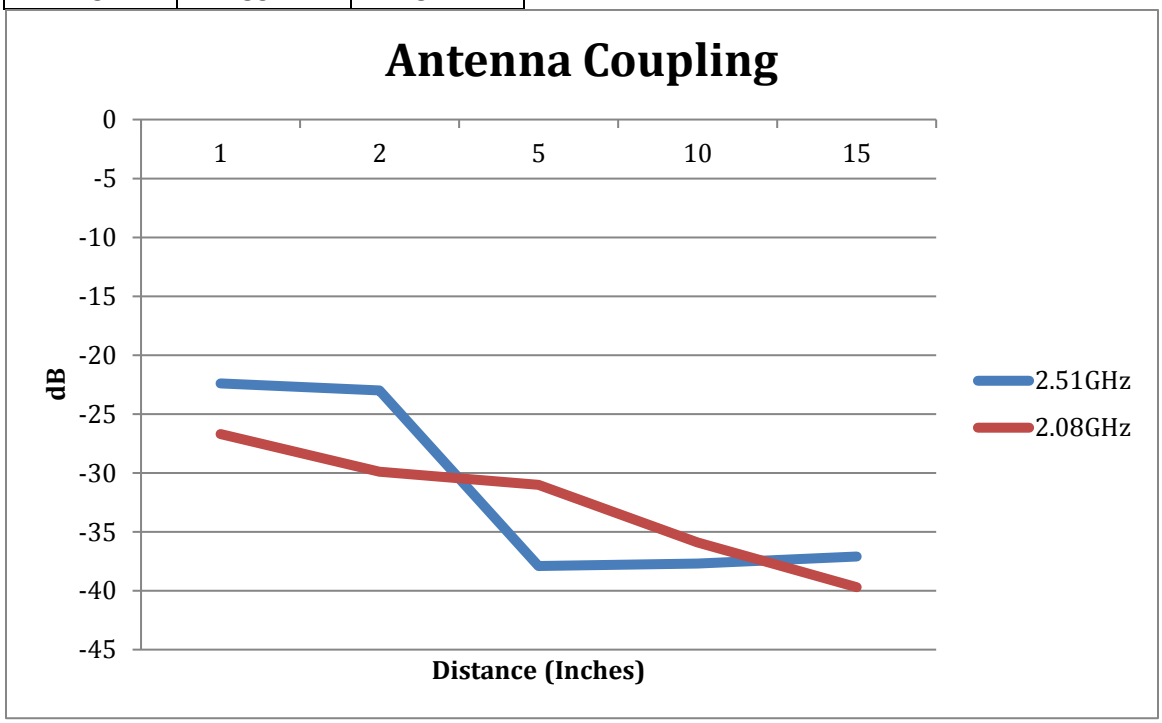
10 Inches



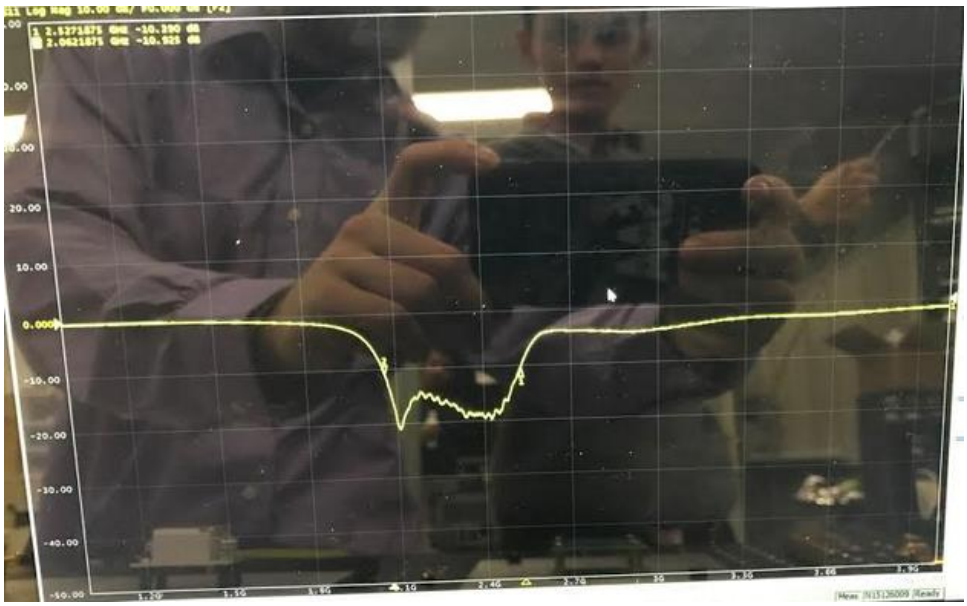
15 Inches

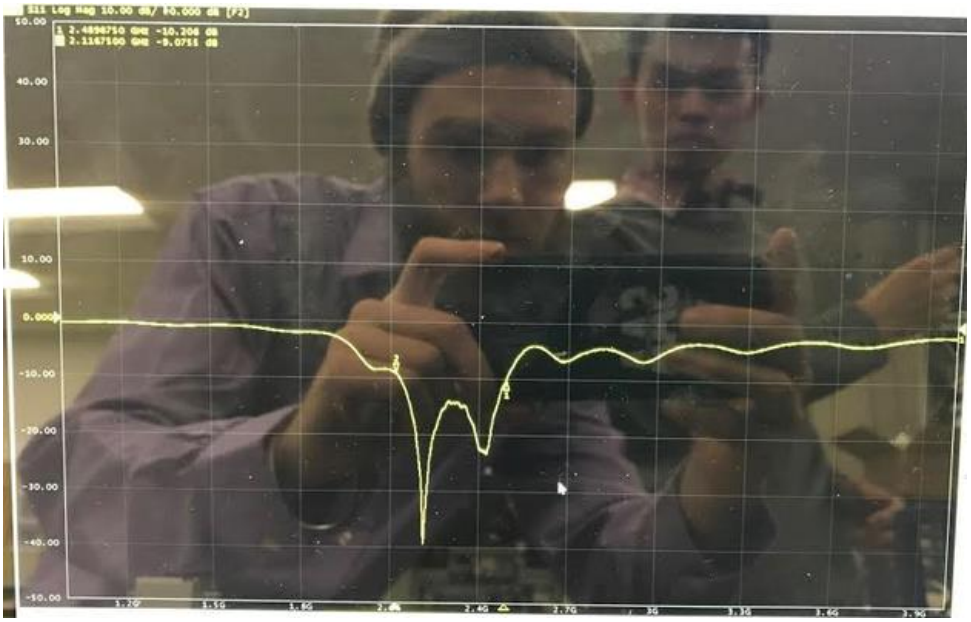
The above shown coupling for the Yagi antennas is summarized in the below table and plot

	2.08GHz	2.51GHz
Distance	Coupling(dB)	
1	-26.7	-22.4
2	-29.9	-23
5	-31	-37.9
10	-35.9	-37.7
15	-39.7	-37.1



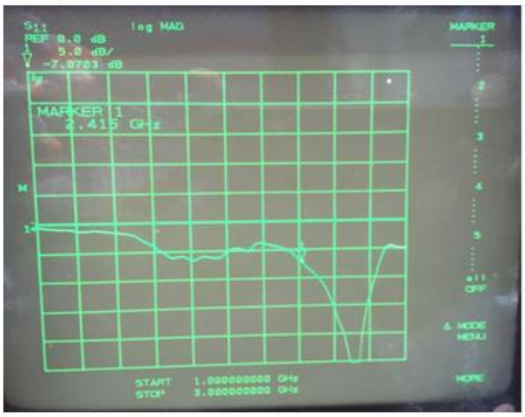
In addition to measurements of the coupling issues we measured return loss parameters for the Yagi PCB antennas in the lab. Two antennas were measured in the lab with 1 being of longer cable length and one being shorter. This gave us one that could be held further from the board to reduce coupling and one that would need to be closer to the board but have much lower loss. The measurements are shown in the pictures taken below.





In addition to the Yagi PCB antenna we ordered and tested the Alfa patch antenna. The purchased patch antennas worked reasonably well with good return loss results when measurements were taken, results shown below.

TX port

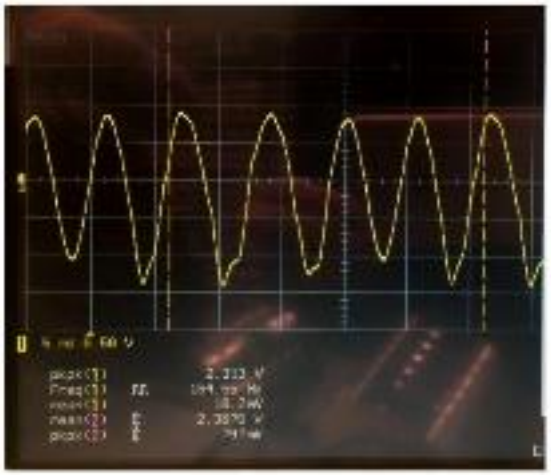


RX port

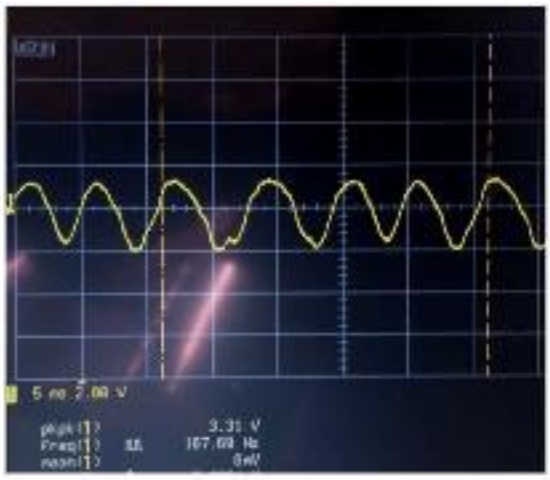
Unfortunately, the patch antennas were not Directive enough for our purposes and as a consequence could not be used for reliable results.

For testing we found that 2 Yagi antennas did not have the required power for a clear useful signal to be received. This required us to use the backup Coffee cans which have results shown below.

Radar System with 2 Coffee Can antennas



Radar System with 1 Coffee Can and 1 Yagi PCB Antenna



As you can see above the system with 2 Coffee Cans works much better than 1 coffee can and 1 Yagi PCB. It was in part for this reason that in the end my group decided to compete with our system using 2 coffee cans. Another key factor was that use of the Yagi PCB antenna provided un-reliable results.

Advice I would give to other students is to choose antennas that work well for your system and are also reliable. We got good results with the Yagi PCB and Coffee can antenna combination but the system was not as reliable as we wanted and in the end our result was simply to use the coffee can antennas exclusively. Although weight is an important factor and should be considered when selecting and if necessary designing antennas the functionality should be considered first and foremost. Furthermore if you are designing your own antenna keep in mind time constraints. Designing an antenna requires multiple stages, simulation in HFSS, Layout in Eagle, ordering and testing of physical part. And sometimes it does not always work the first time (this is usually the case that several re-iterations are required) It was nice in this class learning how to design an antenna as well as the process for testing but in the end we just used the coffee can antennas again which was a little demoralizing considering the time spent on researching, finding, and building antennas for our system.