

EE / CPrE / SE 492 – sdmay21-07

Small Rotor-Craft Obstacle Avoidance Radar System

Week 3 Report

Fall 2020 – Spring 2021

Feb 22 – Mar 1

Faculty Advisor: Dr. Al Qaseer

Team Members:

Matt Bahr – Antenna Designer

Joshua Welton – RADAR Firmware/PCB Design

Felipe Varela Carvalho – Signal Processing

Matt McDermott – PCB Design

Mike Ostrow – Pi Data Flow

Leonardo Bertoncetto Machado – PCB Design

Weekly Summary

The antenna model has been handed off to the ETG for use in creating a physical antenna through use of a CNC machine. The creation of the antenna will be monitored to look for any problems in the created antenna that may need to be solved in the modeling software. We are currently waiting on a specialized tool to arrive at the ETG to help aid in the antenna cutting process.

The printed circuit board design is close to being completed, as the most crucial traces for the board have been laid out, and we are currently routing the traces for the other connections. The main board design should be done by the next status report. We were tasked with making an alternate board for the analog to digital converters, and we are done with the schematic for this auxiliary board. The printed circuit board design for the ADCs is expected to be done by the next status report as well.

Past Week Accomplishments

There are no accomplishments made during the last week, as design work is still in progress for the PCB and the antenna is still in the progress of being machined. Hopefully we will have accomplishments to note on the next status report!

Pending Issues

There are no issues that the group is facing at this. We are steadily working through the required work for this project.

Individual Contributions

<u>Name</u>	<u>Individual Contributions</u>	<u>Hours worked this week</u>	<u>Hours Cumulative</u>
Matt Bahr	<ul style="list-style-type: none"> Communicated with ETG on status of antenna creation Monitoring entire process to look for any potential processes 	2	27
Joshua Welton	<ul style="list-style-type: none"> Completed schematic for ADC circuit Started layout of PCB for ADC Circuit 	7	17
Felipe Varela Carvalho	<ul style="list-style-type: none"> Started communication to set up communication between PC and microcontroller Studied protocols to do so (ethernet, usb, UART or bluetooth) 	4	13
Matt McDermott	<ul style="list-style-type: none"> Continued research for the PCB along with any extra assistance needed to finalize the layout. Coordinated with group members for a progress report concerning the manufacturing of the antenna. 	4	13
Michael Ostrow	<ul style="list-style-type: none"> Searched and downloaded SPI Libraries for pi. Initiated variables to hold PCB register values. Mapped out pins on pi to match the pins leading to the PCB. 	2	9
Leonardo Bertoncello Machado	<ul style="list-style-type: none"> Continued work on PCB, finalizing most of the components Added the antenna connectors and reference clock to the project (only components missing) Began the work on assimilating PCB and microcontroller 	8	16

Note: Hours Cumulative is for this semester of class only.

Plans for the Upcoming Week

Antenna – Matt Bahr

- If antenna is built incorrectly, the model will be remade to fix any mistakes found in the creation process.
- If antenna is built properly, will begin testing the electrical properties of the radar to ensure proper results are obtained.

Signal Processing – Felipe Varela Carvalho

- Write code to communicate between microcontroller and PC

PCB Design – Joshua Welton, Leonardo Bertocello Machado, and Matt McDermott

Raspberry Pi Environment – Mike Ostrow

- Start building code to startup the board and set register values on the PCB through SPI.

[Summary of Weekly Advisor Meeting](#)

The main focus of this week's meeting was in regards to PCB and making sure we can finalize its design and order it as soon as possible. The main topics touched on had to do with the high frequency RF lines and how those should be laid out and prioritized. Furthermore, going over manufacturer restrictions on via sizes and overall board thickness ensure that our design was complying with their specifications.

For software, we discussed which method we are going to use for communicate between the PC and the microcontroller. A couple of possibilities are communicating through USB, ethernet or UART. It was decided that the students are going to be the one deciding which method works best for the project.

Appendix – Screen Shots of Progress

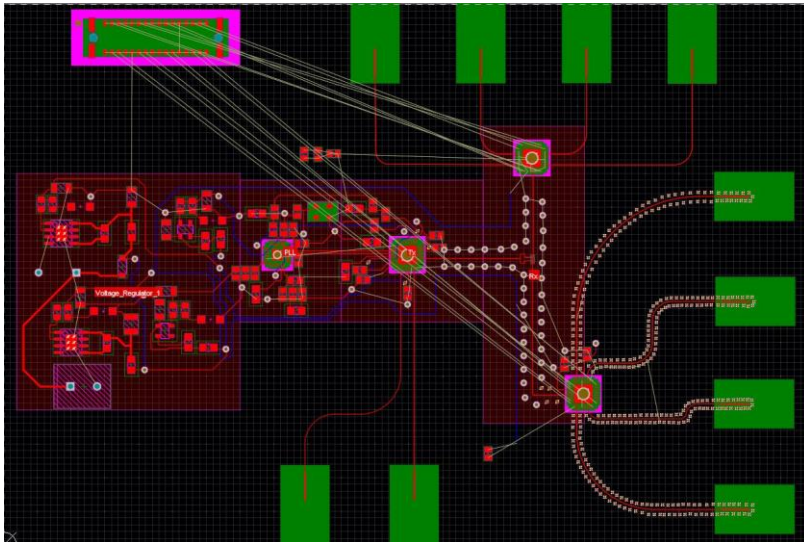


Fig 1. Main PCB Progress

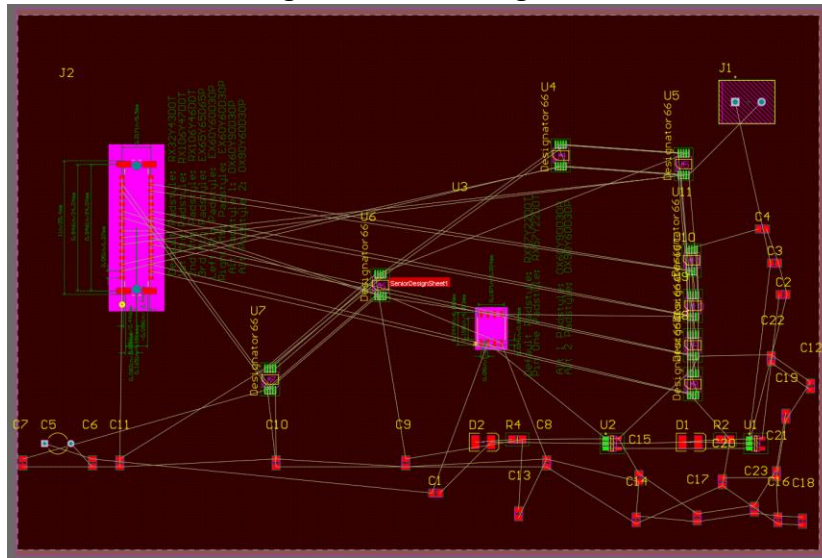


Fig 2. ADC PCB Progress