[1]

ARIANNA Multistage Amplifier

Josh Dodd, Jasper Kaur, Juan Moreno, Matthew Ruiz

Abstract—**The goal of our senior design project, and the topic of this paper, is the replication and enhancement of the low noise amplifier currently used by the ARIANNA collaboration. Currently, the amplifier used by ARIANNA is no longer in production, so a key aspect of our work is to to produce an amplifier design with similar function while still using new and available components from Mini-Circuits. To accomplish this goal, we have been working in KiCad, an EDA software suite, to produce a functional PCB design for the low noise amplifier using components chosen by the ARIANNA collaboration. Thus far, we have completely imported some key components of the new amplifier design into the KiCad software by referencing data sheets provided. These components are the Mini-Circuits PMA-5451+, TCCH-80+ and TB-501-1+. With these components fully imported into KiCad, we designed a 'unit-cell amplifier' to act as a generic building block to allow for easier modification in future PCB design work. Utilizing this unit-cell amplifier we cascaded two cells to create the main body of the new amplifier PCB. Moving forward we are going to be implementing the remaining stages of the PCB, such as power filtering, to finalize the design.**

Index Terms— **Amplifier, Antarctica, Component, Design, Detection, Low-Noise, Neutrino, PCB, Performance, Fabrication, Noise**

I. Introduction

The goal of this senior design project is to replicate and enhance the low noise amplifier currently being used by the ARIANNA collaboration. This low noise amplifier is a critical part of ARIANNAs work in detecting low energy neutrinos in Antarctica as it helps in producing experimental data. As of today, the components used in the original amplifier design are not easily obtained due to being out of production so any needed replacement board would be hard to source. For this reason, designing a new amplifier that uses easily sourced components is of great importance to the ARIANNA collaboration. The goal of this team is to produce this low noise amplifier using components that the ARIANNA collaboration has selected. For this amplifier design, the ARIANNA collaboration has selected various components sold by the manufacturer Mini-Circuits. Through testing various physical components, ARIANNA has been working on deciding which circuit elements are best used to produce a working amplifier. These testing results have been shared with us and give us a general idea as to what our amplifier will need to produce. Based on these decisions we are implementing those circuit elements onto the designed PCB. As stated above, this project is focused on replicating an already existing amplifier and so most of our design cues will come off the back of these decisions made by ARIANNA in terms of components and general PCB layout. With this context in mind, our goal is to use these Mini-Circuit components to produce final a low noise amplifier design that performs similarly or better than the original amplifier.

Throughout this project we have been working up to the amplifier final design in small steps. We have accomplished this by employing a component-oriented methodology in our design. Throughout this project we have been using a PCB design software suite to design the PCB of the amplifier itself. Using this software, we imported the components chosen by ARIANNA, by referencing the respective datasheets provided, to produce a 'unit-cell amplifier' as a means of generating a basic amplifying building block from which we can build upon. With this cell created, it is easier to change the PCB design on the fly depending on feedback from ARIANNA with regards to what circuit elements are required for the amplifier. We have tested the unit-cell amplifier in KiCad to ensure that essential function criteria are met. As of now, the design seems to be that there will be some filtering in between each unit-cell amplifier and a power filter for each stage. Previously in the old amplifier there was a similar build with filtering for each stage and a general power filter that distributes to each amplifier cell. For this reason, we are working towards following this general design layout. In this quarter we have finished the unit-cell amplifier and successfully cascaded it into a two stage amplifier.

II. Materials

So far, we have not used any real physical materials for our amplifier. Everything we have done so far is designed in the software suite KiCad. However, in the spring quarter we expect to be able to have the board printed and the required SMD components shipped. At that point, our materials will be all the circuit components that we assemble into the physical amplifier. The physical amplifier board will be printed by a fabrication house which will be using the Gerber files created with our KiCad PCB design files. We expect this amplifier to be four layers with a bottom copper grounding plate and a solder mask as well as a top ceramic mask for use with colder weather. We will be using dielectric material RO4350 for our amplifier design. The design also required various passive components such as resistors, inductors and capacitors. We will have two Mini-Circuits PMA-5451+ amplifiers which were chosen for their low noise, making them ideal for amplifying the neutrino signal. The amplifier on the board will be connected to external components, like an RF choke (the Mini-Circuits TCCH-80+) and other passive components, for the desired performance. Before, after, and between the PMA-5451+ amplifier setups we will have low pass and high pass filtering to improve the noise performance of the

[1]Submitted December 11, 2019. "This work was done with the help of Professor Stuart Kleinfelder at UC Irvine."

amplifier as a whole. Power connections will be tied to both PMA-5451+ with power filtering in between to remove noise from the power supply itself. At the edges we will have edge mounted SMA connectors for external connection. If we do the surface mount soldering ourselves, we will need the necessary equipment and resources to solder the components to the board, namely a soldering iron and solder paste. Other than the physical assembly, everything has been simulated on the computer and tested with the PMA-5451+ evaluation board with SMA connected filters hooked up to equipment in Professor Kleinfelder's lab.

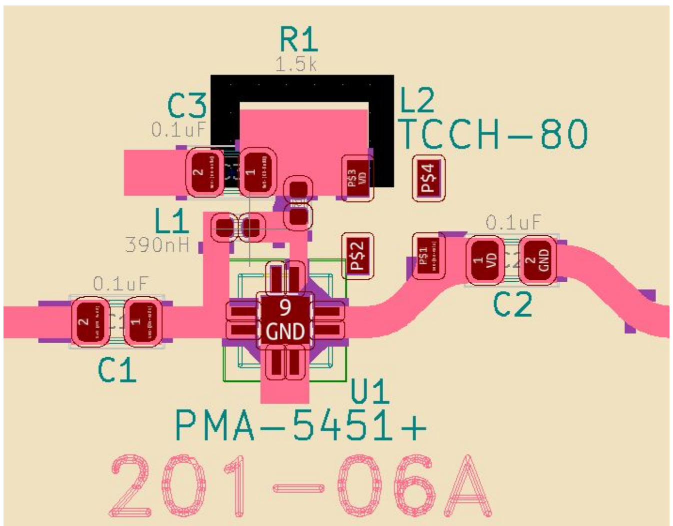

Fig. 1. Gerber files of the TB-501-1+

III. Hardware and Software

From a high level view of our project this project has been rooted mainly in software based work. We have been using the KiCad software suite to import all physical components and to create the PCB design of the amplifier itself. Our team was not familiar with any PCB developing software or hardware techniques so this project required a lot of research and self training. To learn the KiCad software suite we found various resources to help us become familiar. In particular, we relied heavily on a tutorial series hosted by a representative of the company DigiKey [3], a component distributor, that showed the entire process flow of producing a PCB design and some good industry practices to follow. This process flow taught us how most designs are assembled from the beginning in creating a schematic symbol using the Symbol Editor software in KiCad to its final phases in preparing Gerber files for production. To better understand Gerber file generation, as it is a key step in manufacturing an actual product, we also looked at some additional resources [4] dealing with the subject.

In terms of hardware, we will need to have the hardware ordered through UC Irvine as printing this kind of PCB will be expensive. For the most part, this project has been working around the expected behavior of hardware. For instance we have been designing the amplifier around what we expect each Mini-Circuit components to do and how we expect the materials of the PCB itself to work. We are also aware of the general behavior we expect and want from the final amplifier design [1] thanks to the research conducted by the team which designed the original amplifier. In terms of hands-on hardware work we have not used much hardware aside from testing an early test amplifier using reference amplifiers and filters sold by Mini-Circuits.

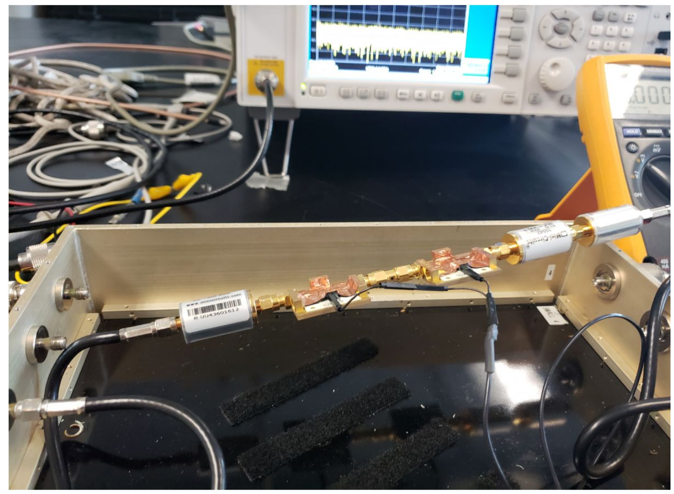

Fig. 2 TB-501-1+ cascade with external filters on the input and output ends. Background has a spectrum analyzer displaying the gain with respect to frequency.

Once we have finished the software design we will be ready to send the design off for physical production. When we have the assembled board, we will be testing the design in Professor Kleinfelder's lab to ensure that the noise performance and gain are to the specifications and standards of the ARIANNA collaboration.

IV. Methods

So far all our methods for producing this low noise amplifier have been rooted in developing the board layout in the KiCad software suite. Using information gathered from the Digi-Key tutorials and other online tutorial resources [2] we have been able to apply a streamlined workflow to the milestones given to use by Professor Kleinfelder. First, we created a netlist outlining how all our components are connected on the board itself. This netlist is crucial as we can update it whenever there is a change in the amplifier design. This is very useful moving forward for the multiple changes that will have to take place in finalizing the design. In a more forward thinking sense, this is also useful for when the amplifier itself gets resigned once more. For the actual board design, we began by placing component PCB footprints associated with their symbolic circuit elements in the netlist into the PCB layout tool. Important to note that we have been importing footprints and gerber files of the PMA-5451+ and TCCH-80+ RF choke from MiniCircuits to place on the design board. As KiCad does not store footprints for some of these components, we are making use of custom schematic symbols and footprint associations to better control and modify our design files in case of any necessary modifications that might arise. These changes will be made depending on the needs of the larger ARIANNA project. From the footprint associations and netlist we generate an association of the two referred to as a ratsnest. Using this ratsnest we arrange components into a more manageable and readable format and began drawing traces to physically connect the components together. For our initial design of the single stage amplifier we followed the schematic of the TB-501-1+ reference board with regards to track sizing and placement.

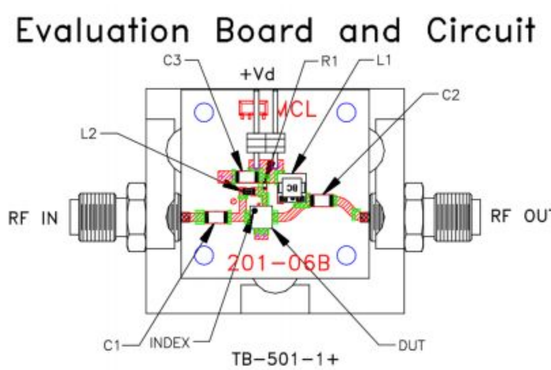

Fig. 3. Schematic of the TB-501-1+ evaluation board

From that initial replication we then modified the design to accommodate a more readily cascadable version of the design. We refer to this modified reference board as the ‘unit-cell amplifier’. Using this unit-cell amplifier we then cascaded two of them into a two stage amplifier using simple cascading methods in the KiCad software. Looking forwards, we have found filters that are optimal for the amplifier input, output, and intermediate between stages, but have yet to import the gerber files into our project to integrate them into our board design. After we have imported the filters, we will add power distribution to the board to power the amplifier chips. The power will be filtered with the same filter design employed by the previous amplifier used by the ARIANNA collaboration. Important to note that in the final version of this design the board will be using 50Ω traces and footprints for our surface mount components. The method for achieving this is by using a trace calculator to measure the dimensions of each trace for the datasheet specified dielectric material RO4350 Then external edge mounted connectors will be added for input and output. At that point, the board will be ready to be printed and tested.

V. Results & Performances

We do not have a prototype for our project printed yet, but we have tested the KiCad board layout using the rules check to ensure functionality of the cascaded mulit-stage amplifier.

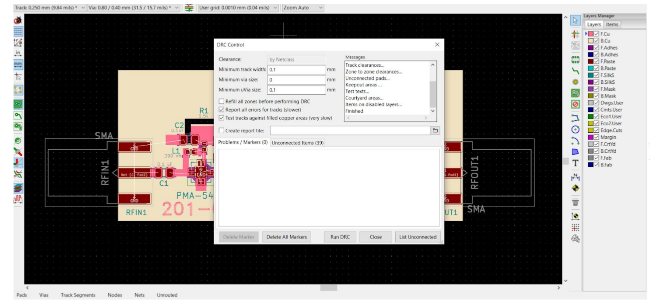

Fig. 4. Screen capture of the KiCad pcb layout with the cascaded TB-501-1+ with SMA connectors

We have also tested a proof concept board made up of two TB-501-1+ reference test boards connected in series to test the noise performance of the filter with filters added at various points. Using a spectrum analyzer, Professor Kleinfelder helped us determine that the optimal setup for noise performance is to have filtering at the input, the output, and between the two stages.This testing is key as it directly influences the design that we will be using in the final PCB.

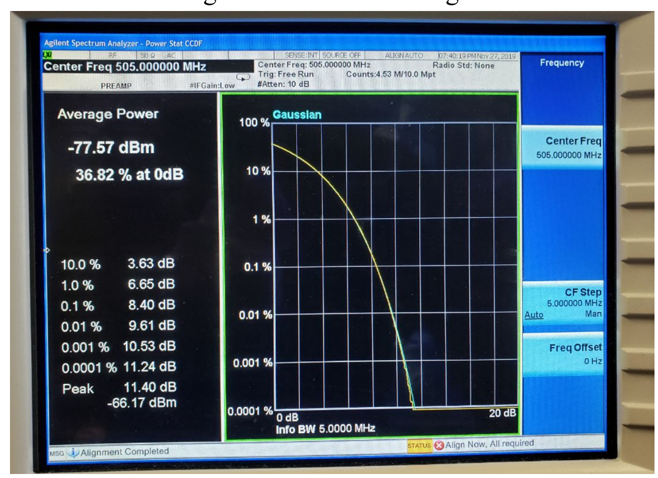

Fig. 5. Analysis of noise power of the TB-501-1+ cascade with filters

We expect to be able to print our circuit board this upcoming quarter and will then test the board’s noise performance and response to signals. If the design is functioning as required then we know that it will behave similarly to the response we obtained by using the concept setup. In the meantime, we are still preparing our kiCad model to be printable and up to the specifications of the ARIANNA collaboration.

VI. Conclusion

In conclusion, throughout this quarter we have fully imported various key components, the PMA-5451 amplifier, TCCH 80+ RF choke, and TB-501-1+ reference board, that into the KiCad software suite. These components are essential as they will be used the overall amplifier design. To import these components we used the datasheets provided by the manufacturer Mini-Circuits to ensure that we were using the components in an optimal setup. Importing these components included importing a digital footprint (these footprints were provided by SnapEDA, a website which specializing in maintaining a PCB footprint layout library for various circuit components) of the PCB layout and an electrical schematic equivalent for netlist generation. Using these components, we designed a unit-cell amplifier that is essentially the fundamental building block of the multistage amplifier as it is a cascadable single stage amplifier. Using this single stage amplifier. This netlist was then used when we cascaded two stages in series to build the bulk of the amplifier with careful attention placed in also including the proper track leads for additional filtering that will be done in the coming quarter. At the top of the cascaded amplifier there is a bare region that will accommodate future power filtering. We have also done some software testing in KiCad to ensure that our progress thus far does not have any glaring functional errors by performing a design rule check.

Our goals moving forward in this project regarding the software design are to import the various filtering components that are needed (a low-pass and high-pass filter are both used in addition to possibly a bandpass filter) to finalize the design. In addition, we are also going to be implementing the power filtering for each amplifier stage. Following the completion of the software design, this project will also aim to find a fabrication house in order to physically produce the amplifier. When selecting this fabrication house there is a great importance on finding a manufacturer that is compliant with the general PCB manufacturing standards in order to produce a reliable and functioning amplifier. Ways to expand this project could be to do the main physical testing of the produced amplifier in the next quarter. Using a network analyzer we can test the noise and gain of this amplifier to confirm that the design is fully functional and usable by the ARIANNA team.

Appendix I

We are making use of technical standards in our project by relying on the technical standards used to determine PCB component sizing. For our project, we have opted to use the imperial standard of component sizing in accordance with JEDEC JEP95 sizing guides (we have also seen these shown as EIA sizing). We chose to use this as our sizing standard because the manufacturer of the TB-501-1+ has provided data sheet references that make use of imperial sizing standards (in the test board data sheets provided the dimensions are listed following imperial sizing). One example of this sizing standard in action would be the size of one SMD resistor being used on the test board. In the reference diagram one resistor is listed as being size 0402, which means It has a length of 0.04 inches, a width of 0.02 inches and a height of 0.014 inches. These kinds of measurements are important as they help us determine what are the constraints and requirements for our PCB design. For this reason, we opted to remain faithful to the original units used in the design for our replication. Moving forward this standard might change depending on what specific sizing is requested by the ARIANNA project. In general, the design of PCB boards will vary in conventions and sizing standards depending on the company (or group) that is working on the design. Some designs will use metric conventions and others will make use of imperial sizing. There are sizing standards setup for both so in either case we will be making use of one set or the other. In terms of hardware standards, we are also aware that the fabrication house we choose to produce our physical implementation of the PCB design should be in accordance with IPC standards to ensure a reliable and proper production. The fabrication house we elect to use for our physical production of the amplifier will be one that is properly following the IPC fabrication standards for instance they should realize IPC-A-600, the Acceptability of Printed Boards. It is important to have standards in general because they lay the foundation of all product development. For this reason the development of our project has been conscious of the various standards that exist with regards to producing a functioning product. This means we have taken into account various IPC standards as well as the conventional sizing standards. Important to note that our design might also change depending on any additional standards imposed by our chosen fabrication house.

Appendix II

The main constraints that we have encountered in our project have been working with limited financial resources, limited design references and limited PDB design experience for our replication. As the ARIANNA collaboration is funded in part by the NSF there are financial constraints imposed on the project as we have to make a conscious effort on minimizing the spending of ARIANNA funding. One of the ways we are helping to promote wise spending of the NSF funding is by working on our amplifier in the least inexpensive way possible through our selection of software to get the maximum value possible. To achieve this, we are using KiCAD, an open source PCB software suite, because it is completely free and multiplatform while still maintaining all the functionality necessary to make a functioning amplifier. By bypassing the need for expensive PCB software we are conserving funds from the ARIANNA project that can be allotted to more pressing issues. In addition to this software choice, we are also working to make the amplifier boards as inexpensively as possible by only spending on components that are absolutely necessary. Taking these precautions ensures that money allocated for the amplifier is used as prudently and efficiently as possible so as to make sure that limited funding is used to its absolute greatest potential. These financial constraints due mean that some aspects of the project have been impeded. One of the biggest effects on the project has been small issues with the KiCad software, for instance, we had some issues with doing the traces found in the references given by the amplifier manufacturer. This has to do with some of the ways that KiCad deals with traces. It allows the user to do certain shapes and sizes but overall the trace design is very rigid and so getting the more creative shapes and curves can be a bit of a hassle. These kinds of issues are lessened in fancier and paid software suites. To overcome these kinds of constraints we have been working on finding creative solutions. To solve some of the issues for example with the traces we found that the user can either make PNG files with the shapes and import them into a trace or use the simple freehand shaping tools that KiCad offer. We also have had to work somewhat in the blind with regards to the amplifier design as there are no concrete design reference that we can draw inspiration from. As this project is intended to be a replication of an existing amplifier the lack of concrete design reference has been a big constraint. To overcome this constraint ARIANNA has been testing the low noise amplifier to decide what kind of subsequent stages are needed in the amplifier. We are using this feedback and incorporating it into our design to get a functioning design. Lastly from a software perspective we have never worked with the KiCad software prior to this project so we haven't had any experience with it. In a larger sense, we have never worked on designing a PCB before so this entire endeavor has been a learning curve with regards to have to work around our limited knowledge of which design practices are best. To overcome these constraints we have been following design practices commonly used by design professionals such as the people over at Digi-Key with regards to our software design.

Appendix III

In today's world of hidden software vulnerabilities and hardware errors we have been conscious of the security and product reliability issues that are present in designing and producing any product in the modern age. This project has taken into account these reliability and security concerns by making a conscious effort to remain diligent in our product checks and software design. A major concern we have had throughout the process of making this amplifier has been ensuring that the design we make in KiCad is in fact an accurate and printable design, this means we have to ensure that the KiCad software itself is capable of running proper design checks and diagnostics tools. To ensure that we are in fact making a functioning design we have not only relied on the rudimentary eye check within KiCad but have made sure to use the deeper diagnostics tools which measure connections as being proper or not. In addition to this, we have worked to make not only the PCB design but also the electronic schematic for the larger amplifier itself. With this schematic we are able to export and import into a more robust design checker such as Spice. By having this schematic we are able to run tests in a variety of software suites in order to ensure that we do not rely entirely on one piece of software for our design check. With regards to security concerns and hacking we have not been concerned as this design does not use any kind of wireless interfacing and instead is a purely physical connection only, we do not have any kind of Wi-Fi or Bluetooth connection so those concerns do not apply. From a hardware perspective we are not concerned with any kind of vulnerability that can manifest itself in a meaningful way. We are aware that this finished product will only be used in a remote region of the world as the ARIANNA project is working in Antarctica and that our equipment will be handled by trained professionals. Our only concern with regards to the hardware aspect is that the board we produce is fully functional for its intended purpose.

Acknowledgment

We would like to thank all the work done by the ARIANNA collaboration in the name of science and for giving our team a functional and interesting design challenge. In particular we would like to thank Professor Stuart Kleinfelder for being our advisor on this project and guiding us towards the various milestones we should be working towards in addition to providing various resources for component work and funding. We would also like to thank the TAs for running the senior design lab which will allow us to test the amplifier in the coming quarter. In addition we would like to thank Shawn Hymel and the Digi-Key youtube channel for the fantastic work they have done in producing a tutorial series for budding PCB designers. This resource helped us immensely in working with the KiCad software and becoming familiar with proper working practice when designing PCBs. We would also like to thank SnapEDA for providing PCB footprints for Mini-Circuit components that were not registered in the standard KiCad footprint library.

References

[1] Barwick, S et al., (2015). Design and Performance of the ARIANNA HRA-3 Neutrino Detector Systems. IEEE Transactions on Nuclear Science, 62(5), pp.2202-2215.

[2] Ho, W. (2019). Application Note: PCB Design Using KiCad [online] Dart.ece.ucdavis.edu.

[3] Hymel, S. (2019). An Intro to KiCad – Part 1: How PCBs Are Made | DigiKey. [online] YouTube.

[4] Layne (2019). Kicad Tutorial: Gerber file generation | Wayne and Layne. [online] Wayneandlayne.com. Available at:https://www.wayneandlayne.com/blog/2013/02/27/kicad-tutorial-gerber-file-generation/