Designing a Data Acquisition Board for Cosmic Ray Muon Detection

Justin Mulé,
Electronic Detector Group, Suffolk County Community College, Selden, NY 11784
David Jaffe
Department of Physics, Brookhaven National Laboratory, Upton, NY 11973
Raul Armendariz
Department of Physics, Queensborough Community College
08/7/2019

ABSTRACT
Cosmic rays are high energy particles of unknown origin that collide with atomic particles in earth's atmosphere generating a shower of secondary cosmic rays, such as muons. The secondary particles produced in these collisions hold clues about the primary cosmic ray. The goal for our group is to create an array of muon detectors across New York State to gather information about subatomic particle showers. This project entails the utilization of a computer software program, Altium Designer, for the creation of a two-channel data acquisition (DAQ) printed circuit board (PCB) to be used in the detection of cosmic ray muons. Altium Designer will be used to create schematic blueprints that detail the layout of the circuitry and position of the electronic components on the circuit board and create the necessary files for manufacture. The development of the PCB is essential to the project because it affords the capability to deploy multiple muon detectors in schools across New York. Implementation of the new design will be more cost-effective and provide additional specific functionalities needed for the ongoing studies. New features added to the PCB include signal strength outputs, noise counting circuits, and the automated upload of data files. As a result of my contributions to the project, I have become more proficient with hardware design and learned the functionalities of specific electronic components and integrated circuits. The knowledge, skills, and expertise that comes from working with senior electrical engineers and physicists has contributed to my personal and professional growth; thus, this experience has made me a more confident, competent, and competitive student with a passion for tackling upcoming challenges.

I. INTRODUCTION

When high energy protons interact with heavy ions in the earth's atmosphere, a shower of secondary cosmic ray particles are produced. Some of these particles are muons; an elementary subatomic particle that accounts for most of the cosmic radiation at earth's surface. These particles are detectable using special plastic scintillator that creates a flash of light when a charged particle passes through it. This flash of light is picked up by a photomultiplier tube which outputs a signal into our circuit board.

A data acquisition (DAQ) board is being designed for the project to replace an existing board with enhanced functionality and signal information. This DAQ board will be a robust alternative and afford additional capabilities that include noise counting circuitry, signal strength outputs, and a variety of deployment options. Furthermore, the new design is a more cost-effective and adaptable approach to collecting and analyzing cosmic ray muon data. The goal for our group is to create an array of muon detectors across New York State to gather information about subatomic particle showers.

II. METHODS

A computer software program, Altium Designer, has been used for the creation of the circuitry schematics, printed circuit board (PCB) files, and manufacturer files; it has also assisted in the layout, placement, routing, and wiring of components. Components for the DAQ front end were selected based upon functionality and cost effectiveness. When selecting components, it is important to ensure accurate representation of the part in the 2-dimensional footprints and 3-dimensional models as these are what are used to layout the PCB. The main source of components came from suppliers such as Digikey and Mouser. Component footprints and models were created if they could not be downloaded or procured through supplier websites. The layout of the components were designed with the traversal of the signal in mind. Items are positioned so that wiring traces can be made as short as possible, and all supporting parts are neighboring each other.

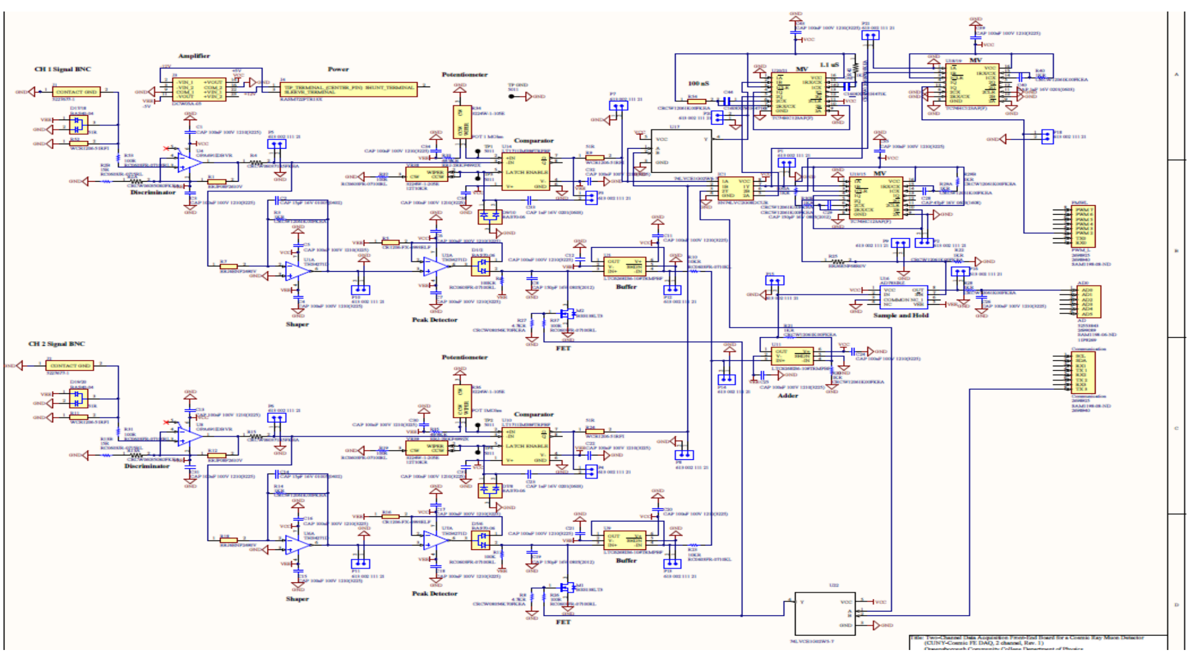

Figure 1: Circuitry Schematic

III. RESULTS:

The first obstacle to overcome was getting a schematic document wired adequately and to ensure all components used in the schematic could be properly imported into the PCB design files. Components that did not have 2D and 3D models associated with their schematic symbol could not be used in the layout process of the board and thus, the focus at the start of the project was to make sure all parts had these models via downloads from supplier websites or creating them manually. Once all schematic symbols have their respective model affiliations, it is possible to load the 2D and 3D version of the components into the PCB file and begin the placement as desired.

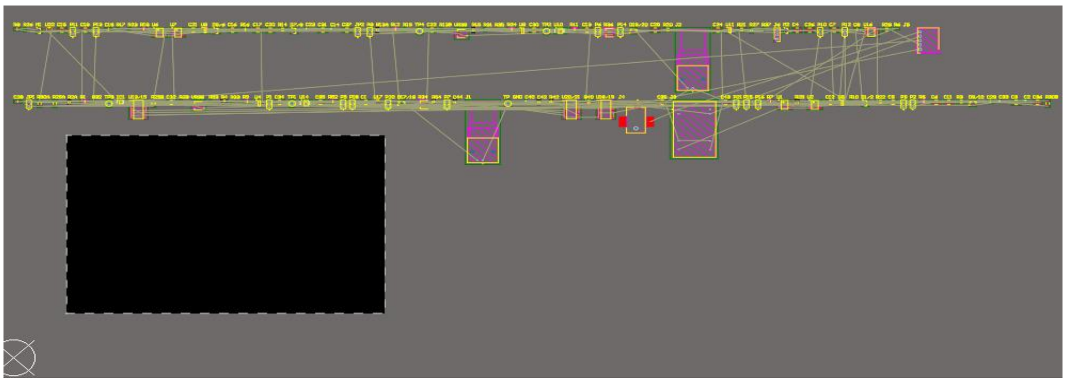

Figure 2: New PCB File

When components are first loaded into the PCB file, all items are laid out in a row in no particular order. From here, each component must be identified and properly placed on the board. Components are arranged at the 5mil level. 5 mil is equivalent to 5/1000 of an inch or approximately 1/8th of a millimeter. Components come with “rat nests” which show how each item is connected together in the circuit. The components will then be grouped depending on their connections.

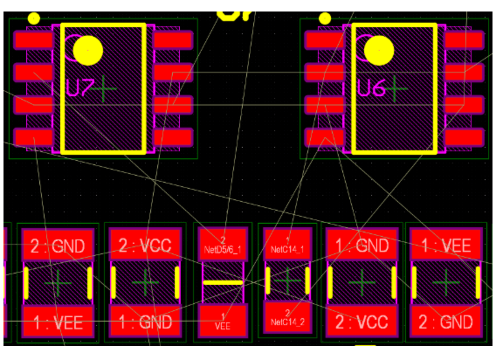

Figure 3: 5mil separation

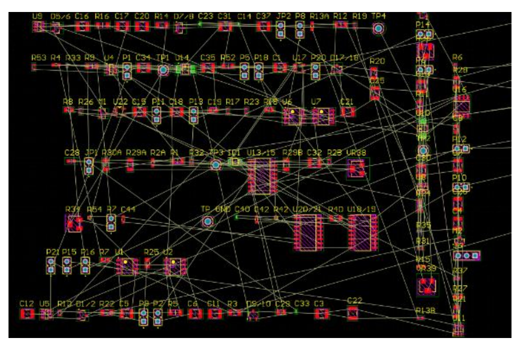

Figure 4: Rats Nest

The board is a six layer stack up consisting of 2 routing layers, 2 ground layers, and 2 power plane layers (±5 volts) totaling 62 mils in thickness. The size of the board was initially designed to be 2x4 inches with Arduino header pins so it could snap into an Arduino Mega 2560, as it will be communicating with it for specific functions. Mounting holes were placed at the corners of the board to offer alternative mounting options such as using PCB stand-offs to secure the

DAQFE on the inside of an enclosure. An enclosure is being designed in unison with the board that will house all components necessary for data acquisition including an Arduino, Raspberry Pi, GPS, LED counter, temperature and pressure sensor, and a laptop charger. With this in mind, threaded BNCs were selected for the board so that the signal connectors can stick out from the enclosure and be secured to the wall using a washer and nut. Because of the enclosure, we want our board to be as compact as possible, while still allowing all components to be properly spaced and labelled on the board.

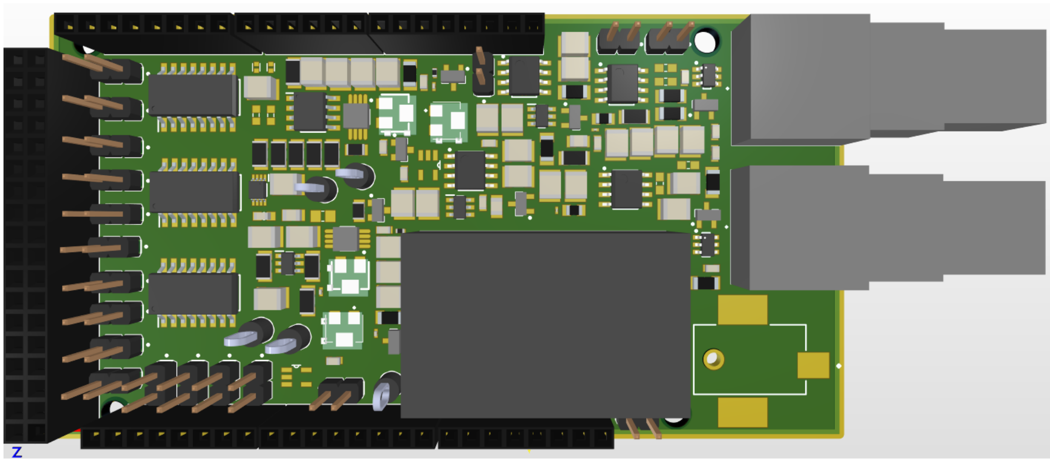

Figure 5: PCB with Arduino Headers

Although this is still a possible method of operation for our DAQFE, a more flexible approach was decided on, where jumper pins and a phoenix connector were included to provide additional power options. The board was then resized to 4x4 inches to accommodate a silkscreen overlay that will clearly identify each component and will make troubleshooting specific items on the board quick and simple. The overlay will also help guide the fabrication of the circuit in the event it will be soldered by hand instead of by a manufacturer. Test points were arranged throughout the board to allow a signal reading at different points along the circuit. This is an important feature because it gives us the ability to probe the circuit and pinpoint components that are working incorrectly.

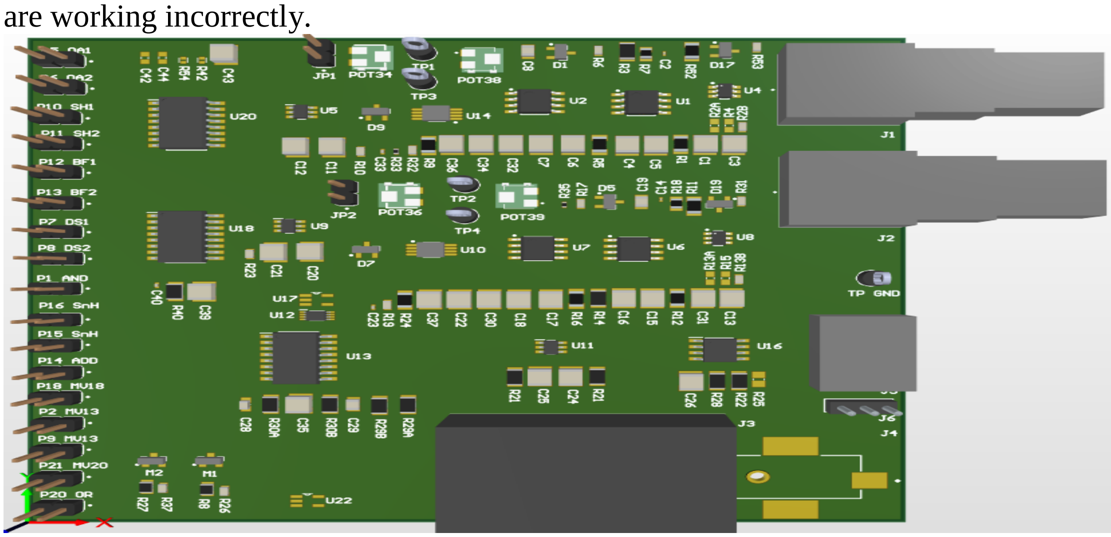

Figure 6: PCB File with silkscreen

Two mounting holes were placed toward the back of the design on this version to allow for more stability, as the front portion will be secured to the enclosure with nuts. Routing of the board was done using the auto-route feature and anything that could not be auto-routed was corrected manually. This allowed more time for other processes and ensured no violations would occur throughout the procedure. Although we do not have an extremely dense board, the auto routing feature greatly increased the efficiency of the design because it automatically follows any design rules that are already implemented and it will highlight any mistakes it made for quick fixes. Once the auto routing was completed, it was possible to create the necessary files for manufacture.

Creating the Gerber files needed for manufacturing is a simple process and mostly done automatically by Altium Designer. 14 Gerber files were created which has all information needed to fabricate the PCB in an industry standardized format. Some of the information included in these files are: drill hole drawings and guides, top layer and bottom layer routing information, pad outlines, Silkscreen designators, and solder mask information. A "read me" text file was also included in these files that details some parameters about the board including board size, board shape, board thickness, board type, the amount of layers in the board, and the width of traces. The file also describes the layer stack up of the board in an easy to read fashion.

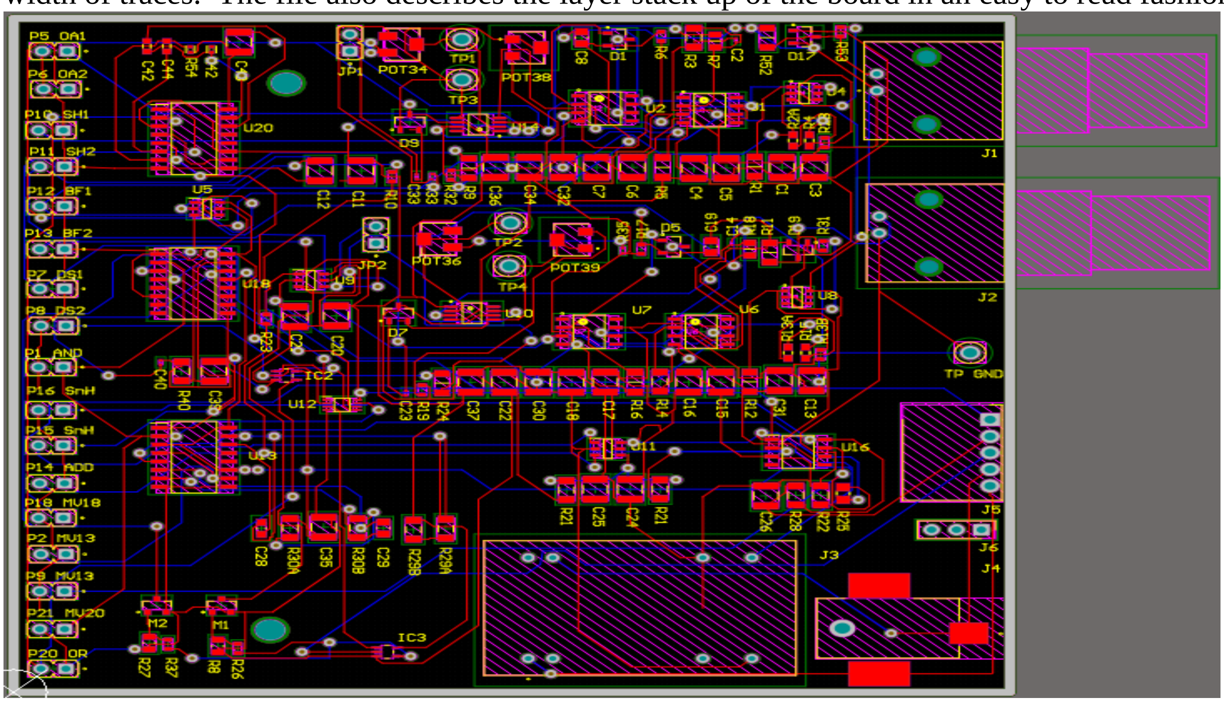

Figure 7: Routed PCB File

IV. FUTURE WORK:

Although there was insufficient time to produce a working prototype, progress has been made on the development of a data acquisition board for cosmic ray muon detection. I plan to keep working on this project to complete some additions to the circuitry that will be implemented at a later date. We are still in the process of choosing a manufacturer to fabricate the board. Once a working prototype is fabricated, I hope to begin testing the functionality and serviceability of the board and correcting any errors associated with it. When our design is complete, we can begin to conduct cosmic ray muon shower experiments in school across New

York State utilizing our own DAQ front end and uploading the files to a central data base accessible to seasoned scientists and new students alike.

V. CONCLUSION:

In conclusion, the development of the DAQ front end has come a long way. The schematic files have been designed and modified to represent the most up-to-date iteration of the board. All parts and components used in the schematic files have been imported into the PCB design files and are early in their life cycles. Obsolescence is an important factor to keep in mind when choosing which components to use because PCB components are constantly upgrading and changing in today's high speed world. Two different layout options have been designed for a more flexible approach to the finalization of the project. Although a myriad changes may still come to the design of the DAQFE, we finally have a visual representation of how our circuitry will look and function upon completion. The board has been completely routed and the Gerber files have been created, so once a manufacturer is decided on all files are ready to send to them. Communication via phone and email has begun between a few different manufacturers in order to get the best deals in the shortest time. A bill of materials file has also been created in an excel spreadsheet for all components used in the creation of the DAQFE. The spreadsheet shows manufacturers, part numbers, quantity of parts, designation of parts, and a description of each component in an easy to read format for fabricators and anyone new to the project.

VI. ACKNOWLEDGEMENTS

This project was supported in part by the U.S. Department of Energy, Office of Science, Office of Workforce Development for Teachers and Scientists (WDTS) under the Community College Internships Program (CCI). I would like to thank my mentor David Jaffe, my collaborators Professor Raul Armendariz (QCC) and Aiwu Zhang(BNL). I would also like to specially thank senior electrical engineer Steve Boose (BNL) for his continued assistance throughout the project.