# Designing a two-channel data acquisition printed circuit board for cosmic ray muon detection

Justin Mulé, Suffolk County Community College, Selden, NY 11784
David Jaffe, Electronic Detector Group, Brookhaven National Laboratory, Upton, NY 11973
Raul Armendariz PhD, Dept. of Physics, Queensborough Community College
07/29/2019

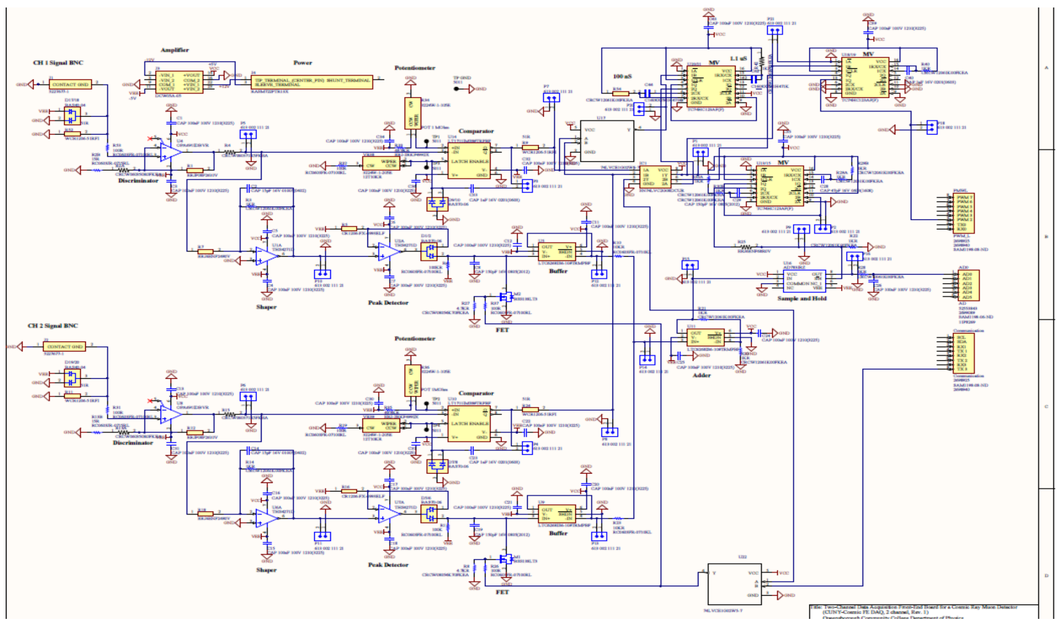
## Abstract:

Cosmic rays are high energy particles of unknown origin that collide with atomic particles in earth's atmosphere which generate 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 particles 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. Implementation of the new design will be more cost-effective and provide additional 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.

https://www.quantamagazine.org/ultrahigh-energy-cosmic-rays-traced-to-hotspot-20150514/

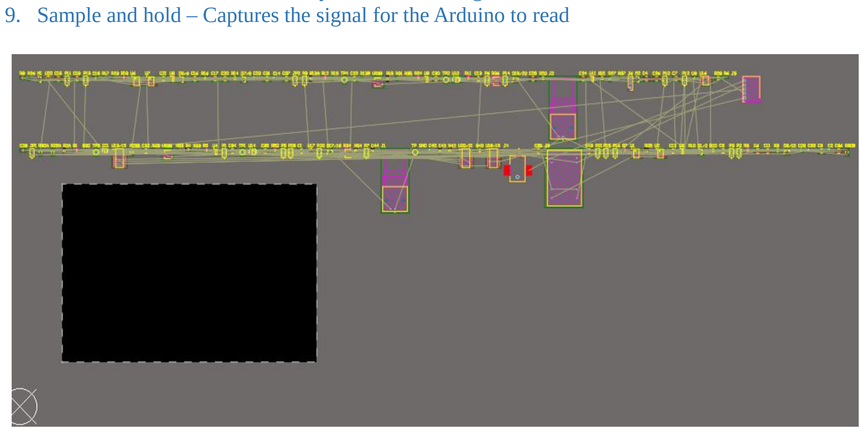
## Introduction:

When high energy protons interact with heavy ions in the earth's atmosphere, a shower of secondary cosmic ray particles are produced. Muons are charged particles; when they pass through plastic scintillator a flash of light is produced and registered by a photomultiplier tube. The photomultiplier tube inputs this signal into a circuit where a series of functions take place. The signal first gets amplified and compared to a user set threshold. If the signal is above threshold, the peak voltage is recorded and an output signal is sent to an AND gate. If signals from both channels arrive to the AND gate simultaneously (within 300ns) a coincidence is recorded which triggers an Arduino to time stamp the event. This is a cosmic ray shower.



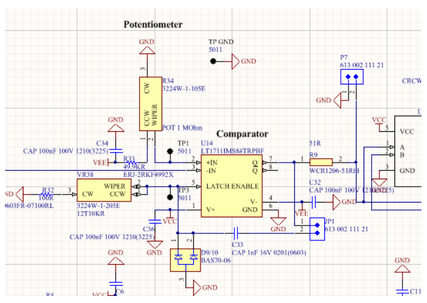

## System:

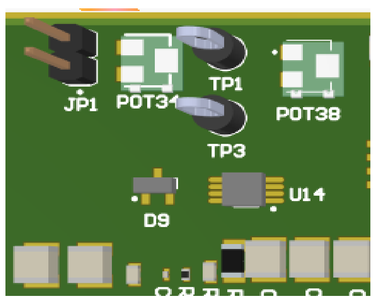
1. Operational amplifier - Multiplies the signal
2. Comparator - Compares the strength of the signal to a user set threshold (filters noise)
3. Shaper – Stretches and inverts the signal
4. Peak detector – records the maximum strength of the signal
5. Buffer - Protects the peak voltage from additional currents and voltages
6. Adder – Adds the averages of the two signals together
7. AND gate – Ensures both signals arrive simultaneously (within 300ns) for coincidence triggers
8. Monostable multivibrator – Delay/Stretch/invert signal
9. Sample and hold – Captures the signal for the Arduino to read

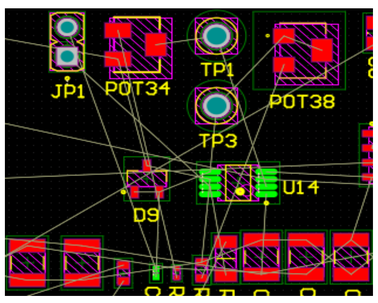


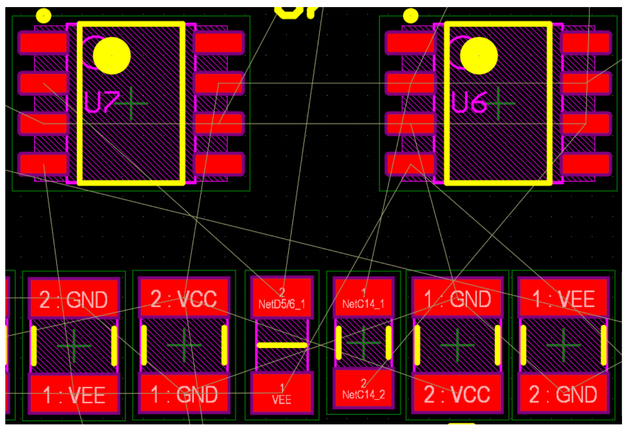
## PCB Design File:

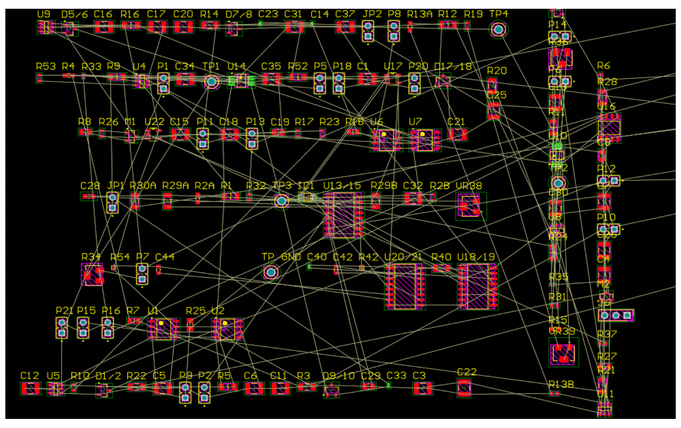
A new design file in the ECAD tool. All components are laid out in a row in no particular order. From here, each component must be identified and properly placed on the board. The Board has to be resized and shaped and a layer stack-up must be completed.



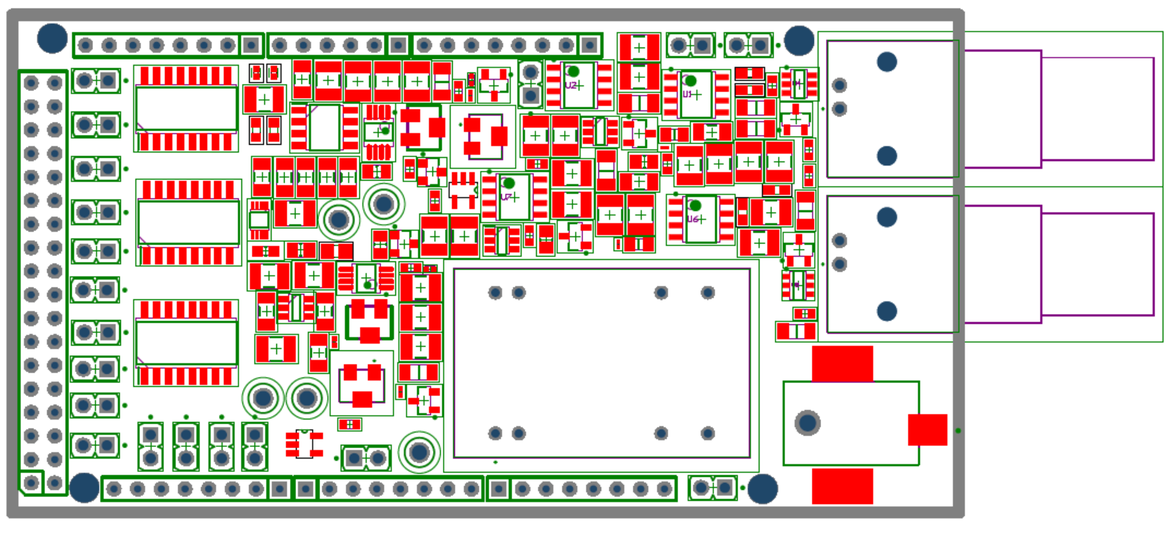



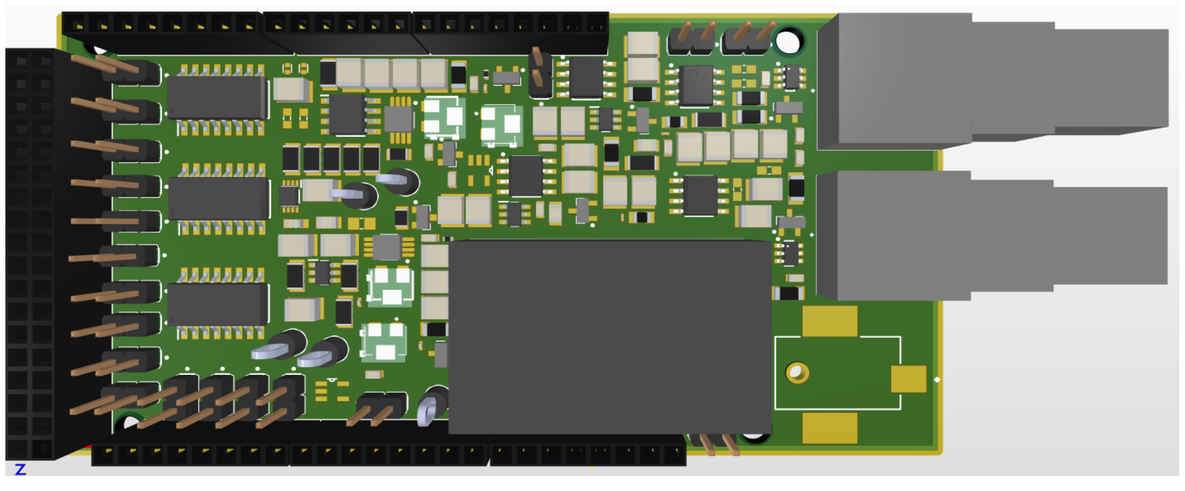
## Arrangement:

Components are arranged at the 5mil level. 5 mil is equivalent to 5/1000 of an inch or approximately 1/8$^{th}$ 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.



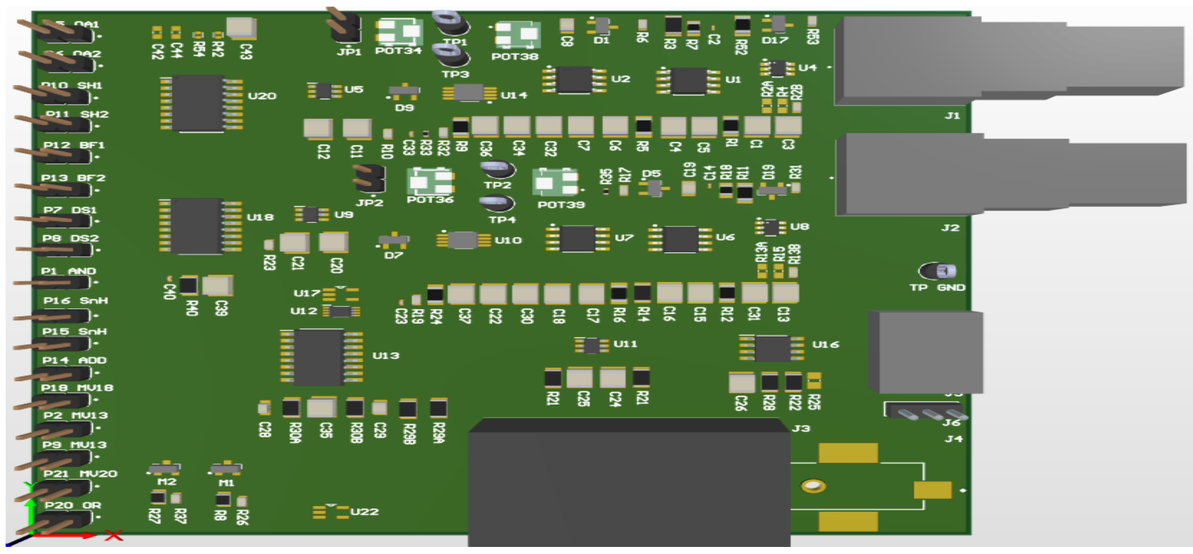
## Methods and materials:

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. Component footprints and models were also created if they could not be downloaded or procured through supplier websites. The main source of components came from suppliers such as Digikey and Mouser. The layout of the components were designed with the traversal of the signal in mind. Components are positioned so that wiring traces can be made as short as possible, and all supporting parts are neighboring each other. Mounting holes were created to secure the DAQFE to an enclosure and Arduino headers were added for the option to mount on top of an Arduino.



## Discussion and Results:

Although great progress has been made designing different iterations of the DAQFE, there is still much to do. Wiring and routing traces still need to be made, and a prototype is needed for testing the functionality of the circuit. Manufacturer files such as Gerber and bill of materials are also needed for a complete proposal of the board. The current design focuses on reliability, cost-effectiveness, and adaptability to multiple configurations. The board has been made larger to accommodate component designators for ease of reference and troubleshooting.




## 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.