12-1-2023

Analysis of the Design and Manufacure of a Printed Circuit Board for a High Altitude Balloon Payload

Peyton Reade Crosby

Analysis of the Design and Manufacture of a Printed Circuit Board for a High Altitude Balloon Payload

by

Peyton Reade Crosby

An Honors Capstone

submitted in partial fulfillment of the requirements

for the Honors Diploma

to

The Honors College

of

The University of Alabama in Huntsville

1 December 2023

Honors Capstone Project Director: Dr. Michael P.J. Benfield

Deputy Center Director, Center for Modeling Simulation, and Analysis

Peyton Crosby 11/30/2023

Student (signature) Date

11/27/2023

Project Director (signature) Date

11/30/2023

Department Chair (signature) Date

Sean Lane Digitally signed by Sean M Lane
Date: 2023.11.30 16:02:24
-06'00'

Honors College Dean (signature) Date

Honors College

Frank Franz Hall

+1 (256) 824-6450 (voice)

+1 (256) 824-7339 (fax)

honors@uah.edu

Honors Thesis Copyright Permission

This form must be signed by the student and submitted with the final manuscript.

In presenting this thesis in partial fulfillment of the requirements for Honors Diploma or Certificate from The University of Alabama in Huntsville, I agree that the Library of this University shall make it freely available for inspection. I further agree that permission for extensive copying for scholarly purposes may be granted by my advisor or, in his/her absence, by the Chair of the Department, Director of the Program, or the Dean of the Honors College. It is also understood that due recognition shall be given to me and to The University of Alabama in Huntsville in any scholarly use which may be made of any material in this thesis.

Peyton Crosby

Student Name (printed)

Peyton Crosby

Student Signature

11/30/2023

Date

Table of Contents

Abstract

With the rapid advancement of technology, it becomes more important to harness the potential of Printed Circuit Boards (PCBs) as a building block in modern electronic systems. Their reliability, efficiency, traceability, and cost efficiency, combined with ease of mass production, make PCBs indispensable in today's modern electronics industry. Primarily, this project was designed to conduct a comprehensive study on the feasibility and cost-effectiveness of designing and implementing a PCB into the payload of a High-Altitude Balloon Payload (HAB).

The PCB schematic will be designed using a variety of widely used software, like KiCAD, EAGLE, and Fritzing. To ensure consistency, the PCB design will adhere to a standardized approach, employing the same set of components across all software. Following the completion of the schematic, the designs will then be uploaded to a variety of websites dedicated to producing PCBs, followed by a comprehensive cost analysis. This analysis will discuss various facets of the financial aspects of the project, including design, production, shipping, as well as any post-production modifications required to produce a fully working board. Through this project, I aim to provide valuable insight into the practicality of incorporating PCBs into engineering applications. This project will analyze not only the technical feasibility of the design process from a novice but also the financial aspects, offering a holistic perspective of the advantages and challenges surrounding the usage of Printed Circuit Boards.

Introduction

Nearly all modern technology, from the cell phones we use daily to the planes in the sky, rely on a basic building block: the Printed Circuit Board (PCB). PCBs allow electronic designs to shrink in size while growing in complexity. My research aims to reveal the important role that PCBs play in electronic designs and to outline the process of both designing and manufacturing your board, starting from scratch.

Background Information

PCBs are fundamental in electronic design, serving as a rigid platform where components are placed, establishing electrical connections. Normally, they consist of several conductive layers, usually made of copper or a similar material, sandwiched between layers of insulating material. Patterns are then etched into these insulating layers, exposing the copper layers underneath, which signify the electrical connections between each component. Components are attached to the board in two main ways: a “pad” or a surface-soldered connection, or a “through-hole”, where pins from the component pass through the board, and are soldered in place. The specific type of component mount used depends on the complexity of the design and the way that the components are manufactured. There may also be mounting holes, for mechanical assembly, grounding plane, and silkscreen, with labels and designators for the designer to identify components, orientation, or serial numbers.[1]

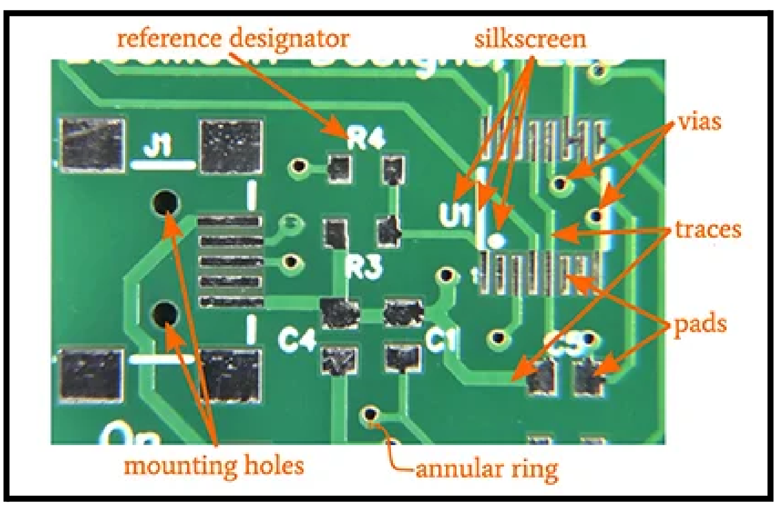


Figure 1: Labeled Example Printed Circuit Board, with Labels

[1] Keim, “What Is a Printed Circuit Board (PCB)?”

As the world advances towards using more technology in everyday life, PCBs will solidify their place in the industry. According to a study conducted by Brad Botwin of the U.S. Department of Commerce in 2017, the PCB market grew by approximately 4 percent every year over 8 years. Total sales totaled approximately 2.5 Billion Dollars in 2017, and sales were projected to continue that growth trajectory over the next few years. Notably, Botwin also unveiled a robust job market, with approximately 78 percent of the respondent companies studied were actively looking to hire new technicians and engineers.[2] Moreover, the study encompassed a wide variety of industries, including aerospace, communications, healthcare, and many others. As PCBs are establishing a larger global presence, their use and popularity are not only reflected in market growth but also influence the development of cutting-edge technology.

PCB Design Overview

The simplest way to start designing a PCB is to start with a simple wiring diagram. This is done by first selecting major components. Components are selected based on the needs of the project. This also includes a high-level view of power and ground connections and any signal connections that need to be made to the processing unit. It is mainly used as a graphical representation of the circuit and should be the main reference when designing the PCB. Next, the "footprint" of the PCB is designed, which defines how the components will physically connect to the board, whether with pads or through holes. The last major design step is to produce a PCB layout, which is when the overall PCB size, component placement, and connections are laid out. Once the PCB has been fully designed, advanced design considerations can be made, such as material selection, thermal management, and signal integrity can be changed, if needed.[3]

For this project, my team and I were tasked with designing a High Altitude Balloon Payload that records GPS coordinates barometric pressure, altitude, temperature, humidity, and gas quality. The payload also needs to wirelessly send data packets to a ground station using a radio transmitter module. Below is a table that summarizes the specific components that were selected to be used:

[2] Botwin, "U.S. BARE PRINTED CIRCUIT BOARD INDUSTRY ASSESSMENT:"
[3] Hamilton, *A Guide to Printed Circuit Board Design*.

Table 1: Selected Payload Components

Component Name	Purpose	Physical Dimensions	Required Power	Price
Arduino Nano	Microcontroller	44.8 x 17.6 mm	6-12 V	$19.99
BME680	Sensor for Barometric Pressure, Altitude, Temperature, Humidity, VOC Gas	25.5 x 17.6 x 4.6 mm	3.3-5 V	$19.95
NEO-6M	GPS Sensor	27.75 x 25 x 9.9 mm	3.3 V	$10.99
Molex Antenna	GPS Antenna	10 x 10 x 4.25 mm	N/A	$6.69
RFM69HCW	Radio Transmitter	25.3 x 29.3 x 3.5 mm	3.3 V	$ 9.95
Battery Holder	Holds 9V Battery	67.6 x 32 x 21.1 mm	N.A	$1.59
9 Volt Battery	Power Source	2.64 x 1.26 x 0.83 inches	N/A	$3.99

The main criteria for the sensor selection were cost, availability, and documentation. Following part selection, an electrical wiring diagram is made, showing the specific connections that need to be made from each component to the Microcontroller, which is what controls the operation of the payload once programmed. A larger version of this image can be found at the end of this paper, in the "Detailed Schematics" section.

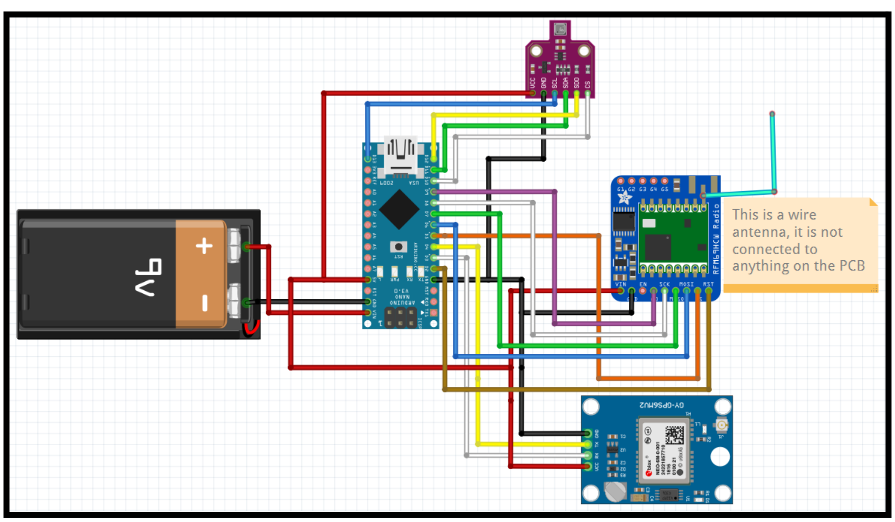


Figure 2: Payload Wiring Diagram

Red wires in the diagram above denote a power supply, black wires are grounding wires, and any other colors indicate a digital signal being sent between the Arduino Nano and each component. Note that the above diagram does not necessarily indicate where the components will be placed on the PCB itself, but it is used to draw the traces between each component pin. We can adapt this wiring diagram into a PCB schematic and footprint, which is explored fully in the next section.

Overview of Each PCB Design Software

KiCAD

KiCAD is a commonly used software in electronic industries, most often used by Electrical Engineers. KiCAD's mission is to "provide the best possible cross-platform electronics design application for professional electronics designers.[4] " It was first introduced in 1992 and is continuously updated to meet the needs of the design industry, based on feedback from users. Many large electronic companies such as Adafruit, Arduino, Raspberry PI, Digikey, and many others release KiCAD footprints and schematic files for their parts, showcasing its industry acceptance.

KiCAD offers many different design tools that can be used for a wide range of electronic design applications. The first is a schematic editor, for designing circuit diagrams. Next are symbol editors and footprint editors, for creating custom component electrical symbols and footprints. There is also a PCB editor, where a custom PCB can be made based on the electrical schematic for the project. Beyond these core design tools, KiCAD offers many other useful tools. For instance, a Gerber file viewer shows how the PCB appears during the manufacturing process. Additionally, a versatile calculator is included for computing essential design criteria, like voltage, current, impedance, and many others. Finally, KiCAD makes it even easier for users to customize their experience with a content manager, where users can import custom libraries and other content from third parties. KiCAD's toolkit ensures that designers have the necessary tools for an efficient design process.

When a user opens KiCAD, they are met with a navigator window, where they can select what they would like to work on:

[4] "About KiCAD."

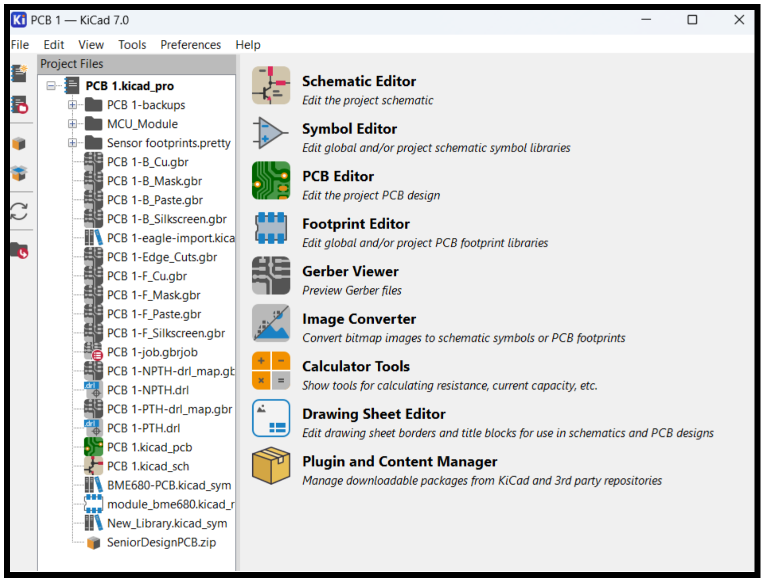


Figure 3: KiCAD Navigation Window

To design a PCB, you must first create an electrical schematic where you connect the pins of each electrical component. Each component is represented by a part symbol, shown in Figure 4 below. Many companies will provide a symbol that you can use, but you can also create your own if needed. In the symbol editor, geometric size is not the main focus. Rather, the designer focuses on the pin type, name, and orientation, as seen in Figure 3.

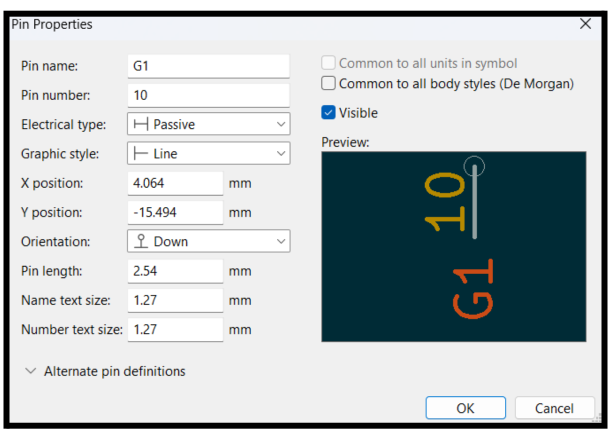


Figure 3: Pin Property Tab

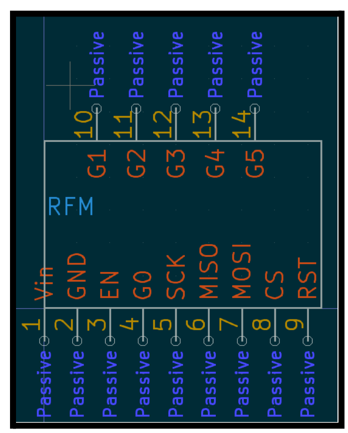


Figure 4: RFM Radio Module Symbol

Once each component has a designated symbol, you can then connect each pinas needed to create a full electrical system schematic. This is used as a reference to create proper connections for the PCB design. The schematic for this project is shown below in Figure 5:

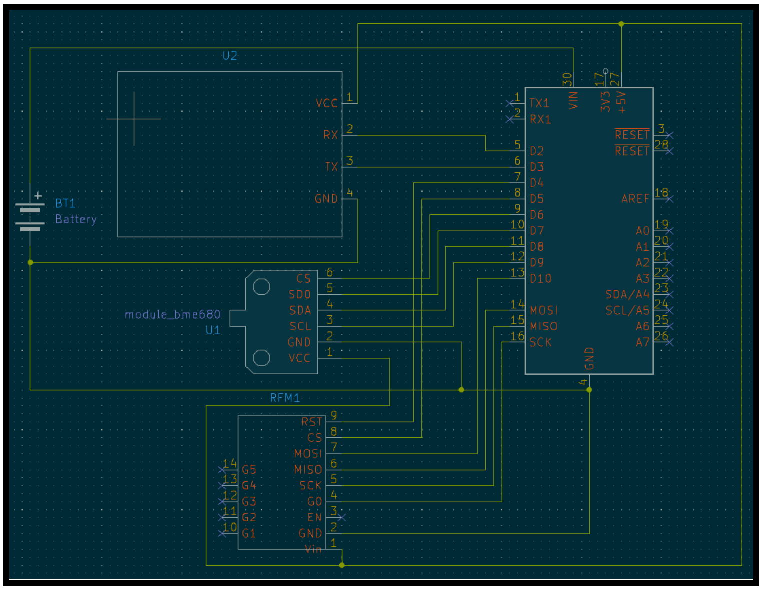


Figure 5: KiCAD Electrical Schematic

KiCAD simplifies the transition from schematic to PCB design with an "Export to PCB" function. However, for this to work properly, each of the components must have a footprint assigned to it. If a library from the manufacturer was used to import electrical symbols, then they will likely already have a footprint associated with it. User-created symbols require manual creation and assignment of the footprint in the Footprint Editor. Since the footprint describes how the component attaches to the PCB, geometric size and pin location matter the most in this phase of the design. Below is the footprint created for the RFM Radio Module:

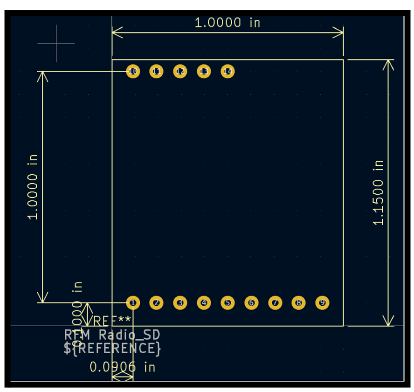


Figure 6: RFM Radio Module Footprint

The electrical schematic, once exported to the PCB design editor, opens with the components randomly placed:

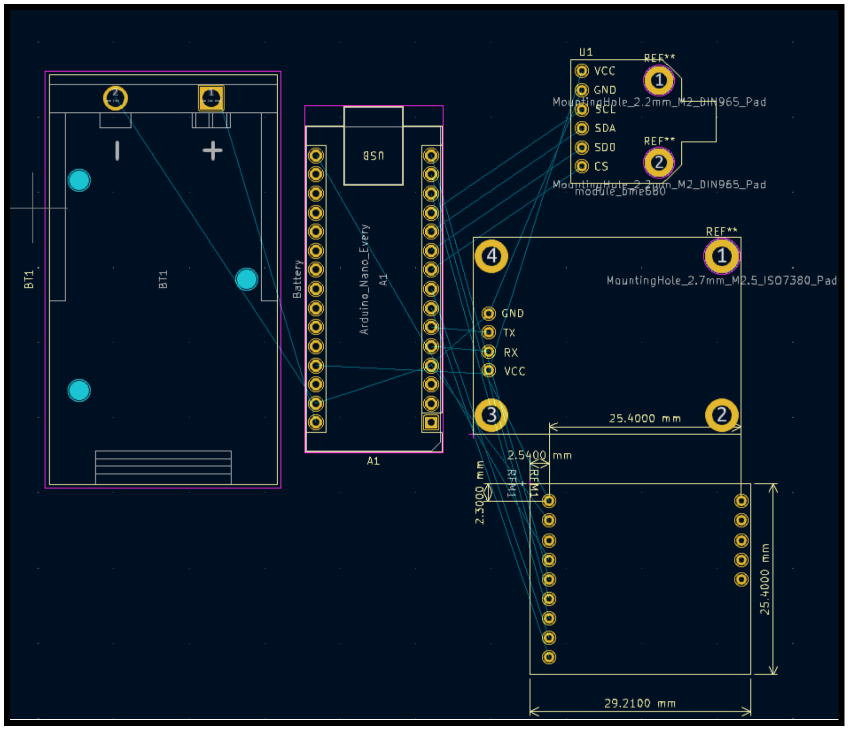


Figure 7: KiCAD Initial PCB View

After arranging components and drawing traces, the Final PCB Design is achieved:

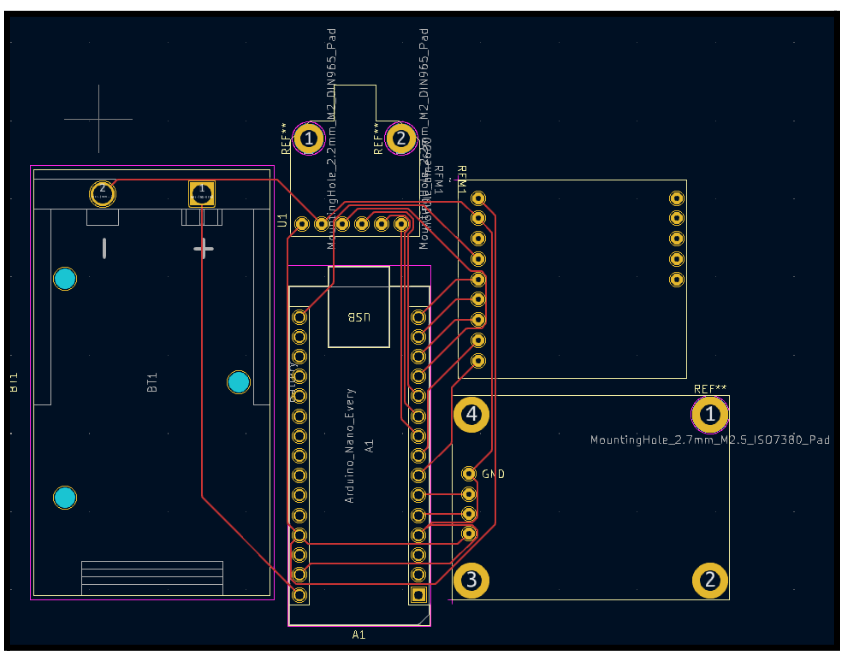


Figure 8: Final KiCAD PCB

Once you are satisfied with your design, you can export the PCB design as Gerber files and Drill files, which PCB manufacturing companies use to produce the boards.

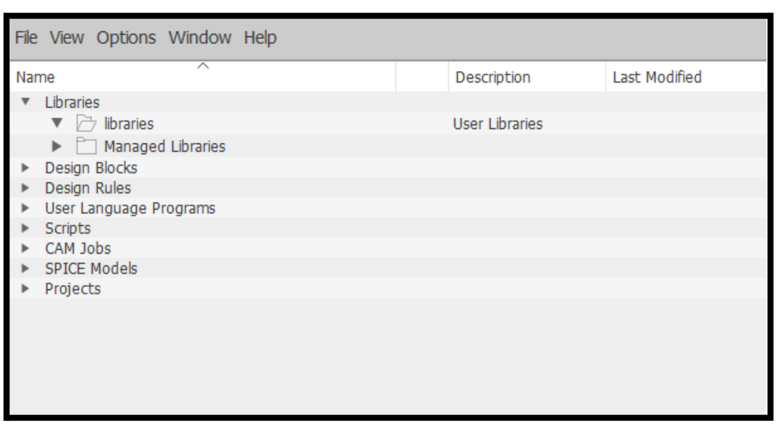
KiCAD's accessibility, being free to use, offers a significant advantage. However, the main difficulty in KiCAD lies in its lack of third-party parts libraries, making it necessary for users to design their own parts. While KiCAD supports this process in its software, it can be quite difficult and arduous to work through the process, since many third-party parts do not have official drawings with dimensions. This means that the design will rely on measurements taken by the designer, which could be inaccurate.

In conclusion, KiCAD, while daunting at first, becomes very user-friendly with online tutorials. Its widespread use among electrical engineers attests to its capabilities. The software's accessibility and continuous improvement make it a valuable tool in the electronic design landscape.

EAGLE

Easily Applicable Graphical Layout Editor (EAGLE) is an electronic design automation (EDA) tool with many features for designing electronic schematics, computer-aided manufacturing, and designing Printed Circuit Boards. Originally released in 1988 by CadSoft Computer GmbH, the software is now owned by Autodesk Inc.,[5] a company that has released many popular computer-aided design software for many disciplines, like AutoCAD, Fusion 360, AutoDesk Inventor, and many more.

Upon launching EAGLE, users are met with a navigation window where they can create custom parts libraries, manufacturing jobs, and project folders.



[5] "What Is EAGLE?"

Figure 9: EAGLE Navigation Window

To create a PCB or any parts, the user must first create a custom library. EAGLE offers many similar tools similar to those offered by KiCAD, such as symbol, footprint, and device editors. Notably, EAGLE boasts the 3D package tool, which connects to Fusion 360. Fusion 360 is a computer-aided design tool mainly used by mechanical engineers to build parts and create assemblies for the construction of prototypes. This integration benefits engineers in the industry by offering a united platform where both mechanical and electrical aspects are considered cohesively. This connected design ecosystem enhances prototyping, enabling the engineer to visualize the potential clashes between electronics and structural components, which enhances the communication between multidisciplinary teams.

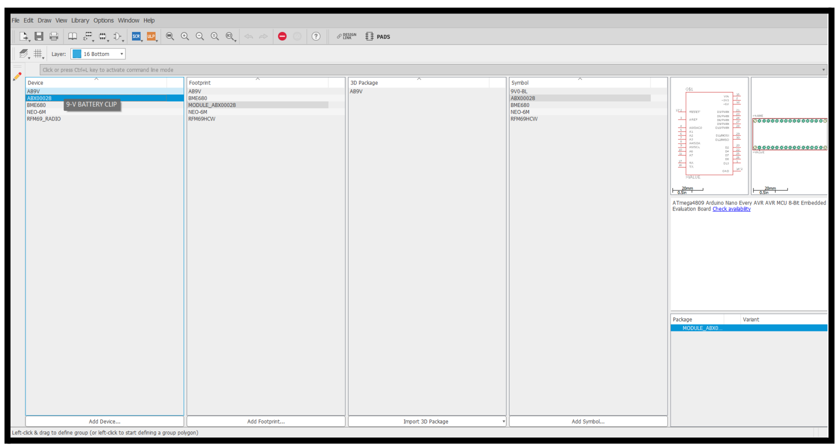


Figure 10: EAGLE Library Control Console

Following standard design procedure, the design must start with a component symbol and footprint. The schematic is a representation of the pin connections, and the footprint is a representation of the physical connections of the component.

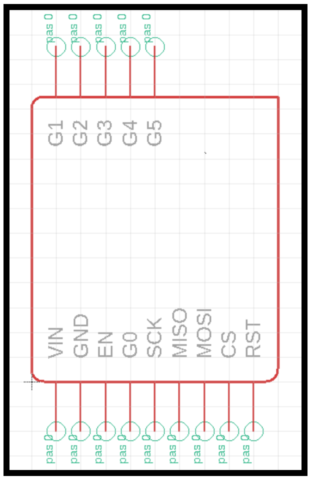


Figure 11: EAGLE RFM Radio Schematic

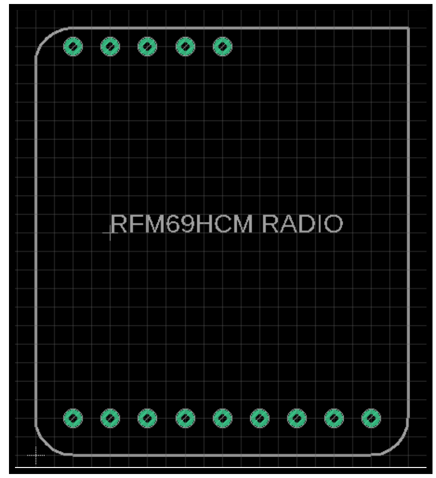


Figure 12: EAGLE RFM Radio Footprint

The next step is connecting the pins of the component schematic to the pads of the component footprint. This is done in the "Device" column of the library manager.

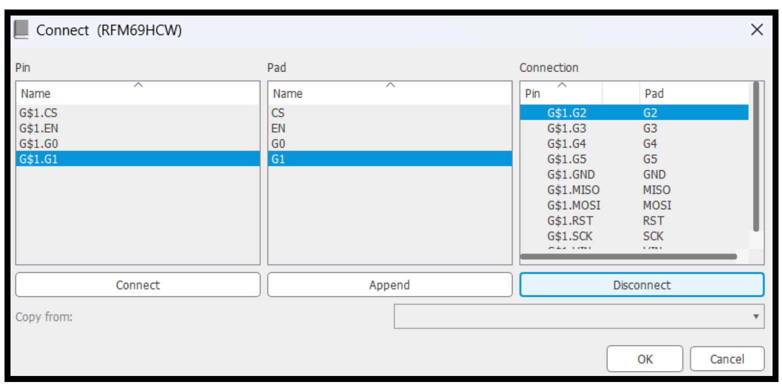


Figure 13: EAGLE Device Connection Manager

After each component has a schematic and footprint assigned to it, an electrical schematic can be created, which represents the electrical connections between the components.

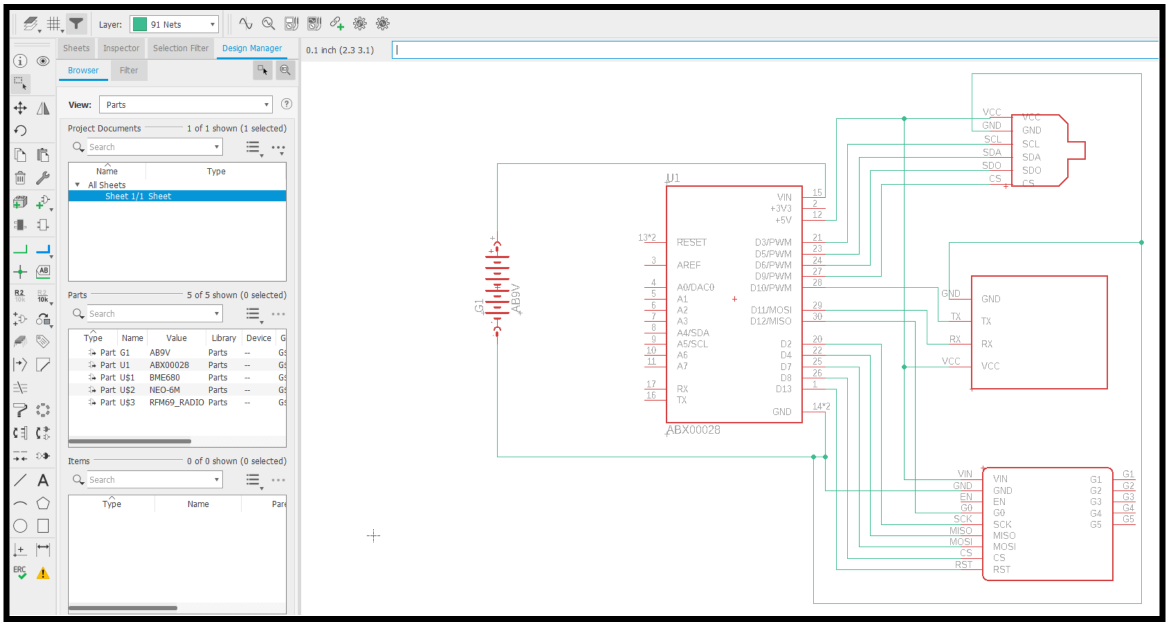


Figure 14: EAGLE Schematic Editor

Transitioning from the schematic, the PCB Board editor opens with yellow lines representing the connections made in the schematic editor.

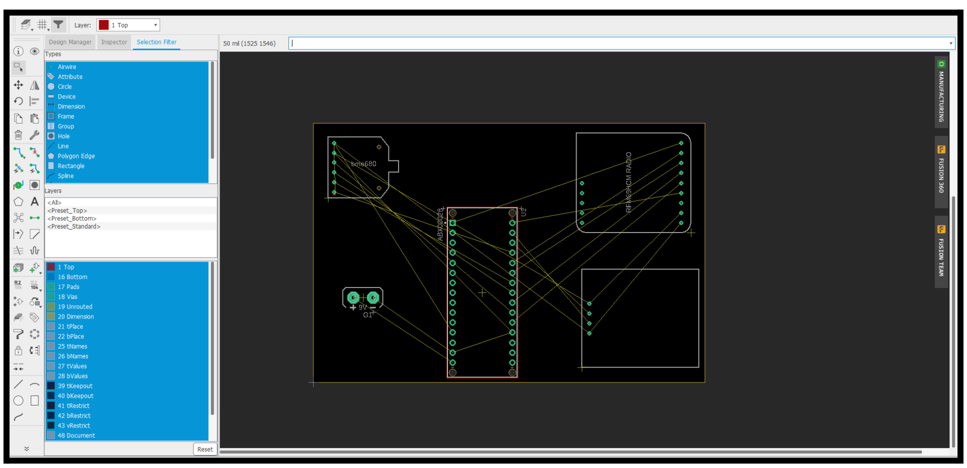


Figure 15: EAGLE Initial PCB Editor View

Once components are rearranged and traces are drawn, the final PCB takes shape:

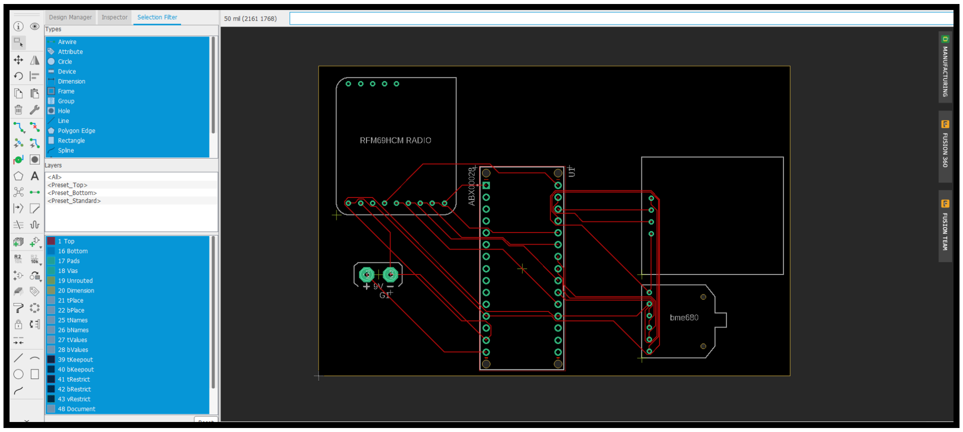


Figure 16: EAGLE Final PCB Design

EAGLE also supports exporting the PCB as Gerber and Drill files for manufacturing the boards. EAGLE is useful if creating a design that requires a mechanical design aspect in addition to the electrical design aspect. Unfortunately, EAGLE has a higher barrier of entry than the other software evaluated, as it requires a subscription to Fusion 360. Students and hobbyists can use the software for free, but these subscriptions only allow access to a trial version of the software, with some features removed. Mainly, the trial software has a maximum PCB size of 3.5 inches by 3.5 inches, you may only edit two layers of the PCB, the top and bottom. This means that more complex PCB designs are not possible unless you pay for the full version, which costs approximately $550 per year.

EAGLE's integration with Fusion 360 is a groundbreaking feature, expanding upon traditional boundaries of PCB design. The smooth progression from component symbol to footprint, then to PCB schematic and Board, while enabling a 3D model of the board, presents a distinct approach to design. While EAGLE and Fusion 360 both require a subscription, the collaboration proves invaluable to those who make more intricate designs. EAGLE's strength lies not only in the electronic design capabilities it offers but also in its holistic design capabilities.

Fritzing

Fritzing is a relatively new design software, with version 1.0.1 released on September 06, 2023. This open-source hardware initiative has fostered a thriving community of users who share their creations with the public. Unlike more traditional PCB design software, the program was designed to be used by non-engineers, especially educators and hobbyists. Due to this, very little technical knowledge is required to fully design the PCB. This makes it an ideal choice for first-time designers. The strength of Fritzing's software lies not only in its accessibility but also in its affordability. Currently, the cost of the full-release version of Fritzing is $8.59, and many previous versions are available for no cost.

Fritzing opens in a breadboard view, where users can create a wiring diagram easily. A base parts library is included, and the open-source nature of the software allows easy access to third-party part creations contributed by other users. The collaboration between the community adds to the versatility of the software. Fritzing features are easily accessed by users, including an easy way to switch between a breadboard, electrical schematic, and PCB view of the created diagram by simply clicking the appropriate section at the top of the diagram. Finally, there is a "code" section of the program, where Fritzing can simulate running code for the product. While useful for components that have easily predicted outputs, like LEDs, other components cannot be accurately simulated, like the GPS or the environmental sensor. Below is the opening view of Fritzing:

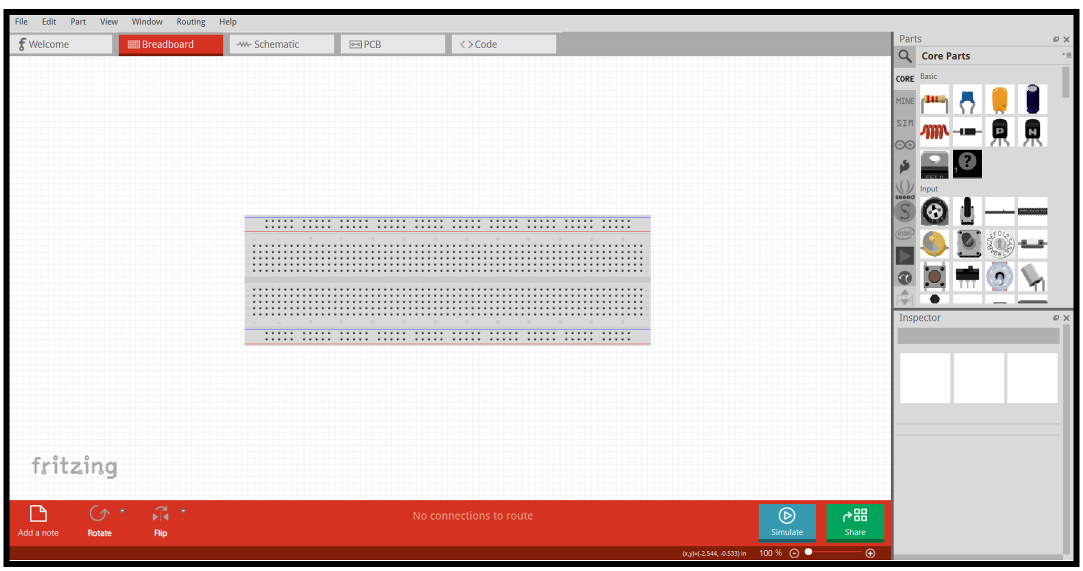


Figure 17: Opening View of Fritzing

To create a wiring diagram starting from the opening view, parts from the parts library can be dragged into the diagram, and wire connections can be made by clicking between component nodes.

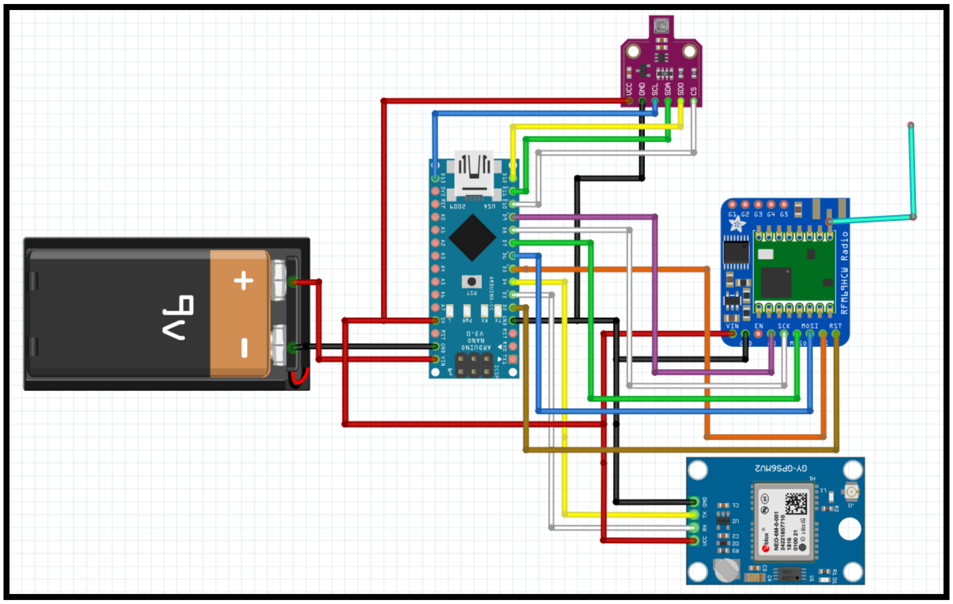


Figure 18: Fritzing Breadboard Diagram

Transitioning from the breadboard diagram is seamless, with connections automatically generated for you, based on the connections made in the breadboard view.

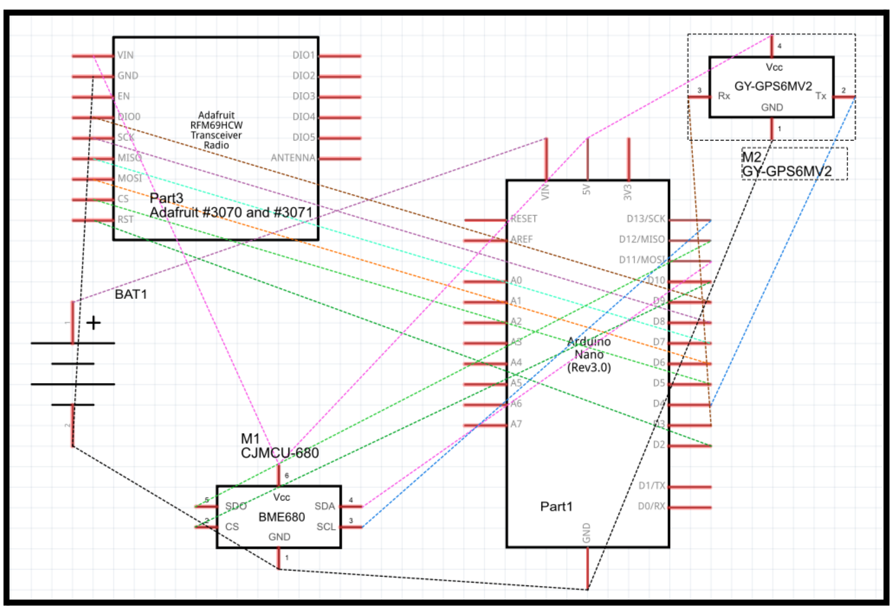


Figure 19: Initial View of Fritzing Electrical Schematic

Although the initial view of the schematic appears cluttered, Fritzing allows for easy modification and rearrangement, ensuring a clearer representation of the schematic.

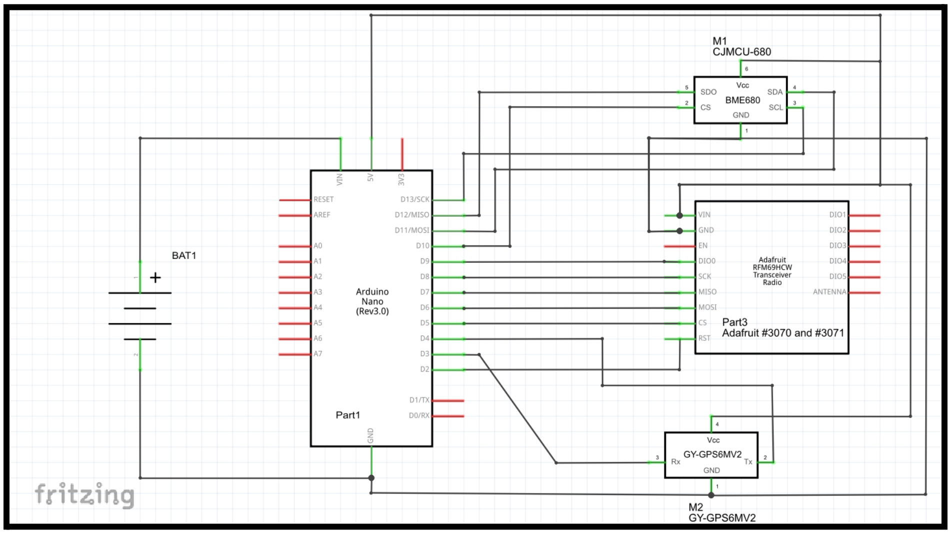


Figure 20: Organized Fritzing Schematic

While the schematics are not necessarily required to produce a PCB in Fritzing, they are useful to easily reference pin connections, and are often used in the electronics industry to show the logical electrical connections between each component. They are also used to explain electron sources, paths, and locations of switches, voltage dividers, or anything else that may affect voltage flow. Despite not being mandatory for PCB production in Fritzing, electrical schematics still prove themselves to be useful. Schematic diagrams offer a visual aid that enhances the understanding of the logical flow of electronics and are used by many engineers and technicians. Schematic diagrams are a valuable intermediate step, helping facilitate an insightful design.

Moving to the PCB layout, users can click the “PCB” button at the top of the software to start modifying the board layout.

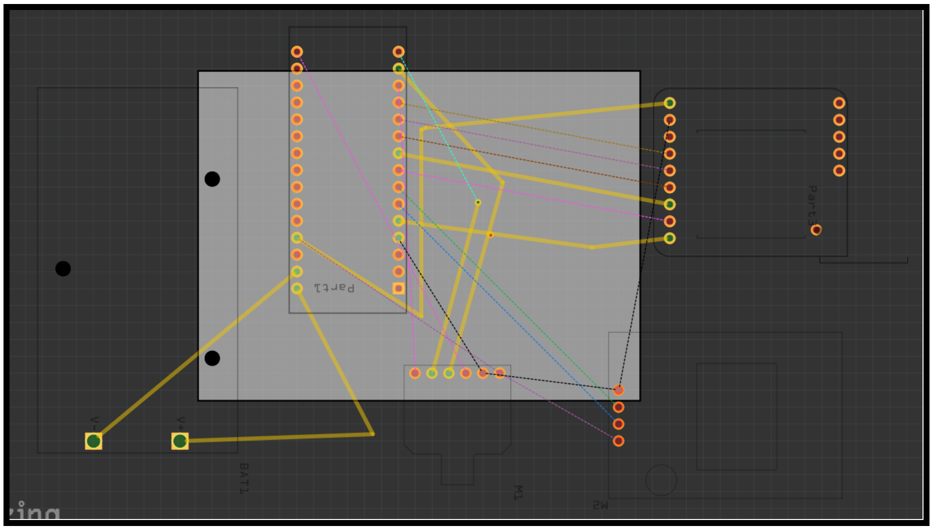


Figure 21: Opening View of PCB Layout

The light gray section of the page indicates the actual size of the PCB, and the components are outlined in black, with pin connections outlined in yellow. You can then organize the components and enlarge the board to create a more proper PCB. Traces are easily generated by clicking the "Autoroute" button at the bottom, which will generate a PCB schematic. Note that the orange lines indicate a route on the top layer of the PCB, while the yellow lines represent routes on the bottom of the PCB.

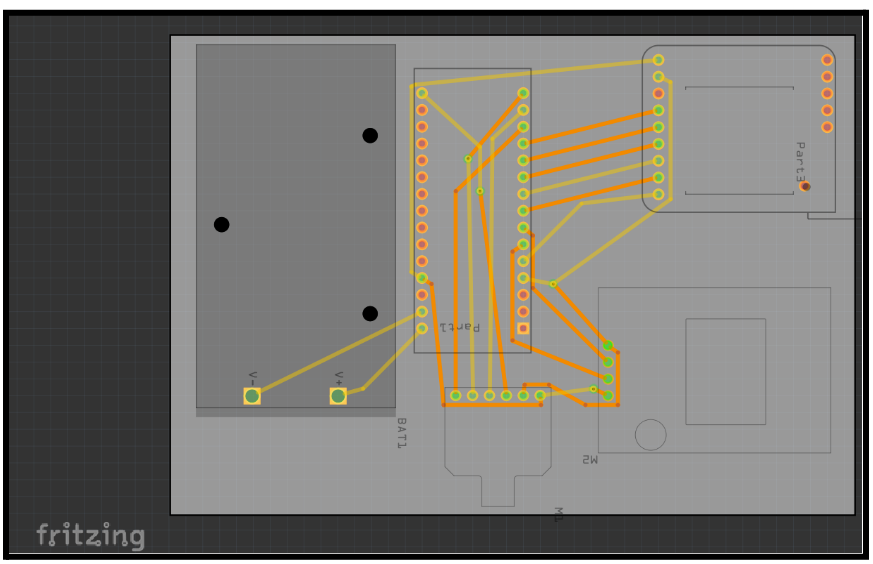


Figure 22: Final Fritzing PCB

Fritzing proves to be very accessible to the public, with an intuitive user interface that makes it simple to use, no matter the base experience of the user. The software is very efficient, taking approximately 30 minutes to go from a blank layout to a fully designed PCB.

Fritzing stands out as versatile and user-friendly, bridging the gap for new electronics designers. Its intuitive interface, affordability, and collaborative community make it a valuable resource for both beginners and experienced designers.

Comparison of Software

This section delves into a comparative analysis of three notable PCB design software solutions - KiCAD, EAGLE, and Fritzing. This comparative exploration of each software's strengths and weaknesses by investigating KiCAD's open-source versatility, Fritzing's accessibility to new designers, and EAGLE's Fusion 360 integration. By analyzing these key features, I aim to provide a nuanced understanding of each software, assisting new designers in making an informed decision of which software to use, depending on their specific project requirements.

KiCAD's software is useful for beginners who need the versatility of open-source software while offering many powerful tools to fully develop the PCB design. Its main strengths lie in its accessibility and flexibility, which allow professionals and hobbyists to cultivate a vibrant community of users. This enables collaborative development and a rich library of components, ensuring a vast collection of usable electronics in a variety of projects. Beyond this rich community, KiCAD's schematic editor, footprint editor, and PCB editor tools contribute to a comprehensive design process, making it an invaluable tool for design. KiCAD's free license, combined with its many design tools make it an amazing choice for new electronic designers.

Autodesk's EAGLE key feature is its integration with Fusion 360. As discussed, the integration allows for a multi-faceted design approach, letting users customize their PCB layout and mechanical design at the same time. The design journey is seamless, going from component symbol to footprint, PCB schematic, and finally a 3D model, which remains unique to EAGLE. This visualization tool makes it invaluable to interdisciplinary projects, meaning that many companies could benefit from using it to simplify the design of many prototypes. While I would not recommend this software for hobbyists due to its limitations on the free version of the software, the full version could be useful for those wanting to start their own company, where they could purchase the full version of the software.

Fritzing, the newest design software analyzed, positions itself as the most user-friendly solution, especially for new designers and educators. Fritzing boasts an intuitive user interface, which simplifies the PCB design process. Its accessibility is underlined by it requiring little technical knowledge. The affordable price for the full software adds to its appeal, with continuous updates offered at no additional cost. Over its development and release, Fritzing has cultivated a collaborative community similar to that of KiCAD, where users upload their creations for everyone to use as needed, and new users can request help easily on their forum. While its features may seem basic compared to other software, Fritzing's main strength lies in offering an uncomplicated way to get started in the world of PCB design.

In conclusion, each software analyzed offers distinct advantages catering to different facets of PCB design. The choice of which software to use depends entirely on the needs of the designer. For a beginner, I would recommend either Fritzing or KiCAD. Fritzing would be preferable to a user who does not know much about electronics, mainly because of its easy transition to PCB starting from an easily readable wiring diagram. For users with prior knowledge of electronics, KiCAD would be my recommendation, even for those new to PCB design software.

PCB Manufacturing Services and Cost Analysis

Online PCB ordering companies offer two main services: a prototype PCB manufacturing service, and a "Pick and Place" PCB manufacturing service. A prototype PCB will be delivered as a blank PCB board, meaning parts will need to be soldered on by the designer, while a Pick and Place PCB will have components already assembled onto the board. Many companies offer the customer a choice between the two services, with the main difference being that, for the Pick and Place service, the customer must either supply the parts to be placed onto the board or pay for the manufacturer to supply the parts.

The choice between prototype PCB manufacturing and Pick and Place PCB manufacturing plays an important role in the production process, so understanding the different companies that offer each service becomes crucial.

Through research, the main candidates for analysis are pcbway.com, digikey.com, jlcpcb.com, rushpcb.com, and allpcb.com. Each of these companies offer different services that could be useful for PCB production, allowing the cost analysis to be as diverse as possible. PCB Way, Digikey, and JLC PCB all offer full assembly production and shipping, while Rush PCB and All PCB do not. Rush PCB specializes in rapid production, being the fastest but most expensive service offered. Digikey is mainly a mass production service, with many discounts offered for larger orders, and many parts offered in their catalog. PCB Way and JLC PCB seemed to be aimed at hobbysists, with a special price offering of $1 PCB prototyping, with a minimum order of 5 PCBs, bringing the total cost without shipping to $5, which is invaluable for hobbysists that only need a few prototypes.

There are many different online companies that provide each service, each with unique offerings and considerations. The main considerations noted in the following analysis are pricing structure, manufacturing time, ease of use, user reviews, and overall suitability for individual design needs. These analysis criteria are aimed to provide a comprehensive overview of the strengths and weaknesses of each service, so that new PCB designers can make an informed decision about the manufacturer they choose.

Results and Discussion

Once manufacturers were selected, they were sent the same PCB file to produce the PCB from, Gerber files from KiCAD. Each company then quoted how much it would cost for the PCB to be made as a prototype, then shipped to my address in Alabama. Below is a cost analysis for a prototype PCB from each company.

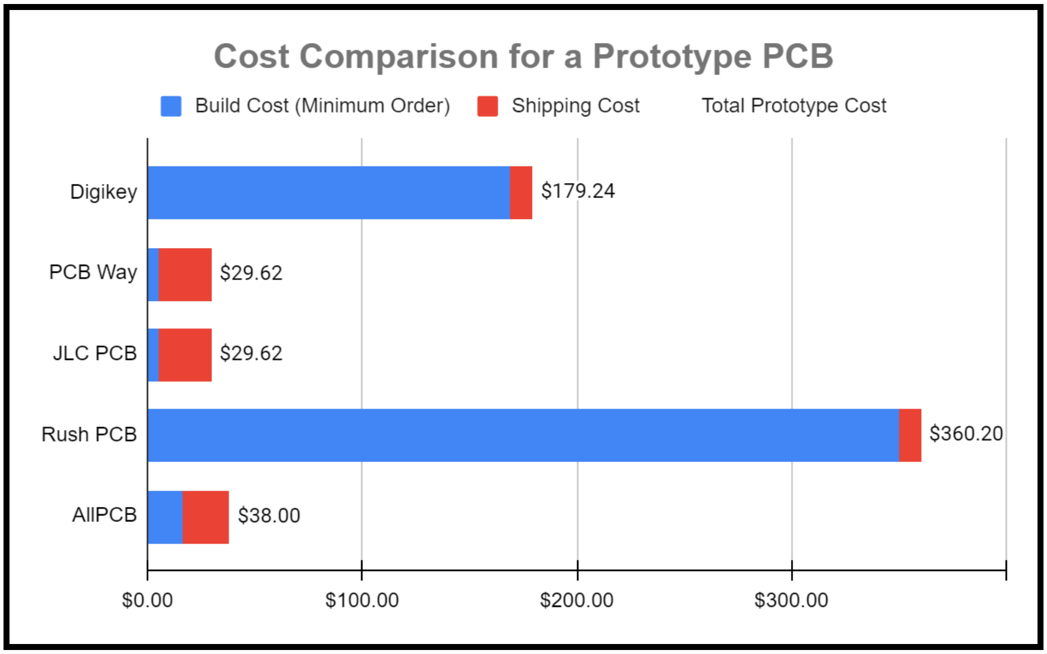


Figure 23: Cost Comparison for a Prototype PCB

For a PCB prototype, the total cost is a sum of the production cost from each company, at the minimum parts order, plus the shipping cost. Digikey, Rush PCB, and AllPCB are all United-States based companies, so their shipping costs are lower than that of the China based companies, PCB Way and JLC PCB. PCB Way, JLC PCB, and AllPCB have a minimum order of 5 boards, while Digikey and Rush PCB have a minimum order of 2 boards. It should also be noted that Digikey has a discount for larger orders. Next, a cost analysis was performed for the companies that offer a Pick and Place option, which are Digikey, PCB Way, and JLC PCB.

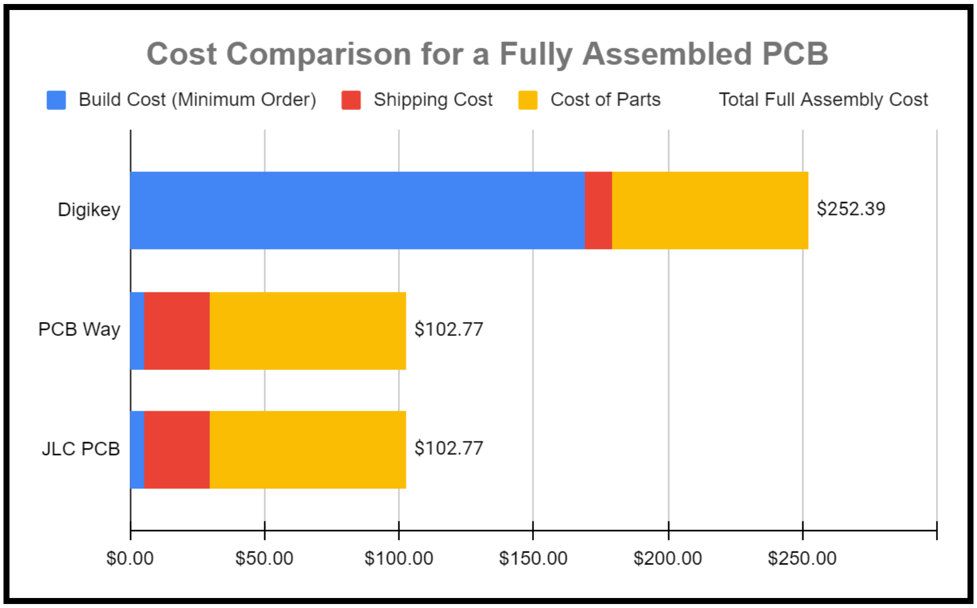


Figure 24: Cost Comparison for a Fully Assembled PCB

The above analysis assumes that the cost of parts will be the same, however, depending on the parts the the designer chooses, there may be extra costs for shipping/supplying the parts. Typically, parts supplied to companies outside of the United States will have a higher shipping cost, but parts made in the United States may cost more overall. It is also important to note that Digikey does not directly make PCBs themselves. Rather, they act as a distribution company for the seller. This means that Digikey has a higher chance of price fluctuation and lead time, leading to longer production times. They also will supply parts for lower costs, if the part is in their catalog.

Finally, the shipping and manufacturing time for each company, was considered for both prototyping services and Pick and Place services, if offered.

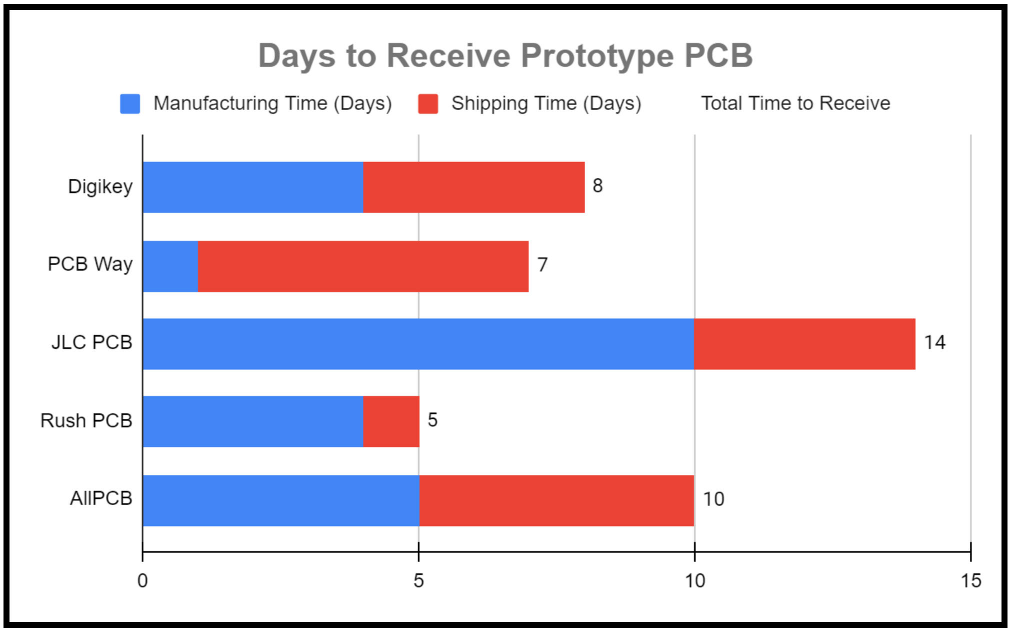


Figure 25: Days to Receive Prototype PCB

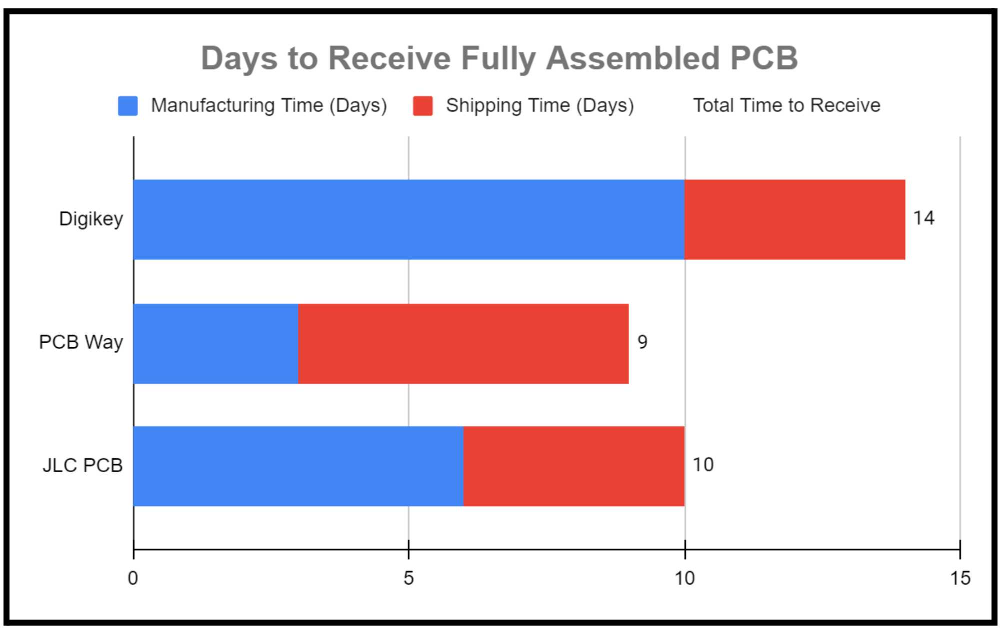


Figure 26: Days to Receive Fully Assembled PCB

Shipping time was assumed to be the maximum number of days as listed for each company. This means that some companies may deliver parts sooner, depending on the number of orders they have. It should also be noted that PCB Way guarantees a 24 hour production time for prototype boards maximum, and a 3 day production time for fully assembled boards, once parts are delivered. This makes PCB Way the fastest low-cost option for both prototyping and Pick and Place manufacturing.

For new PCB designers, I would recommend either PCB Way or JLC PCB. The websites are easy to navigate, only requiring the upload of Gerber files for each PCB layer, which all of the discussed software is capable of producing. Their low-cost prototyping is invaluable, and if you need to opt for the Pick and Place option, they offer that serice as well. Each website also offers a point of contact that reviews the design, ensuring that all requirements are met for the design, and will be produced as expected. For those looking for mass part production, Digikey is the best choice. Its discounts for large orders and vast electronic catalog are useful for both prototyping and full assembly solutions.

Conclusion

In conclusion, the comprehensive analysis of the design process of PCB design and manufacturing services in this study provides valuable insights in to the production process of a PCB for both new and experienced electronic designers.

The comparison of each software led to the conclusion that Fritzing should be used by designers that want to design a PCB with little experience with electronics, offering a highly graphical representation of the PCB design process. KiCAD should be used by designers that have experience in electronic design, but need access to a large library of footprints and schematics for a variety of parts. EAGLE, while powerful, is most useful for designers that need the versatility of the multidisciplinary design integration with Fusion 360. EAGLE is difficult to use without a large amount of guidance, but can be used extremely effectively by professionals with prior experience.

An analysis of several popular PCB manufacturers revealed that there are two standout options, each catering to a different need. PCB Way is an excellent choice for small-scale projects, particularly due to its low production cost of $1 per board and quick turnaround times. Additionally, the user-friendly interface simplifies the design-to-production process, providing a solution for low-volume production. Digkey excels in mass production, making it a popular choice for orders of 100 boards or more. While the cost-per-board is higher than that of PCB Way, the discounts offered for large orders and electronic component discounts make it an ideal choice for high-volume production.

As technology continues to evolve, staying informed about how PCBs are designed and manufactured will become essential. Embracing software like Fritzing, KiCAD, EAGLE, and other tools, along with knowing the different manufacturing services offered by companies like PCB Way and Digikey empowers designers to bring their visions to life. There are many considerations that must be taken into account when designing and producing a PCB, so it is important that the designer considers the best route to take for their project, based upon their experience and production needs.

References

Autodesk.com. “What Is EAGLE?,” n.d. https://www.autodesk.com/products/eagle/overview?term=1-YEAR&tab=subscription&plc=F360.

Botwin, Brad. “U.S. BARE PRINTED CIRCUIT BOARD INDUSTRY ASSESSMENT:,” 2017.

Hamilton, Charles. *A Guide to Printed Circuit Board Design*, 1984. https://books.google.com/books?hl=en&lr=&id=qQ4BBQAAQBAJ&oi=fnd&pg=PP1&dq=what+is+a+printed+circuit+board&ots=UWWefI98it&sig=XQqaVItIY6VCb0noiZGkHiZTqmA#v=onepage&q=what%20is%20a%20printed%20circuit%20board&f=false.

Keim, Robert. “What Is a Printed Circuit Board (PCB)?” Blog. allaboutcircuits.com, April 10, 2020. https://www.allaboutcircuits.com/technical-articles/what-is-a-printed-circuit-board-pcb/.

KiCAD.org. “About KiCAD,” n.d. https://www.kicad.org/about/kicad/.

Detailed Schematics

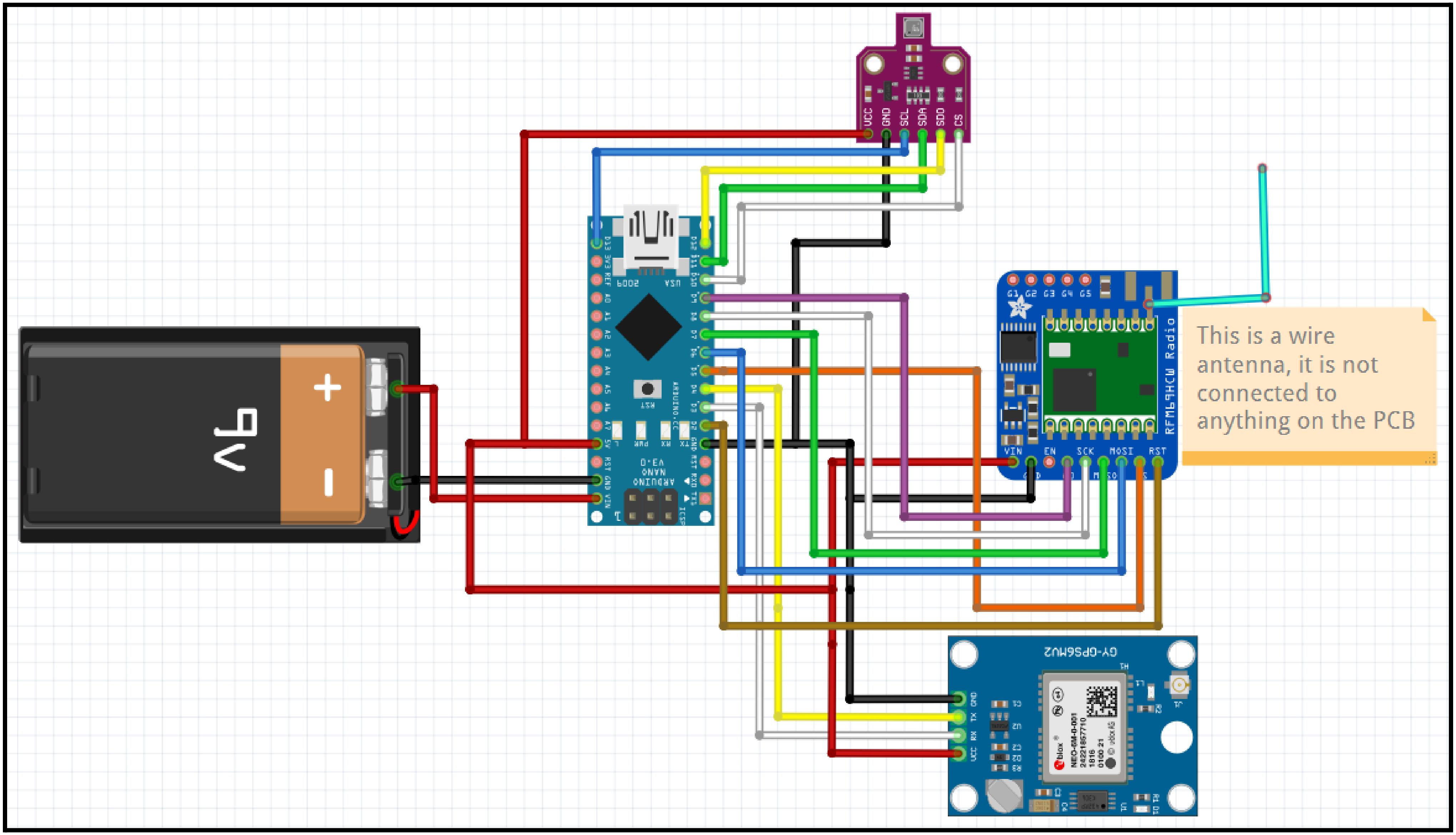


Figure 2: Wiring Diagram

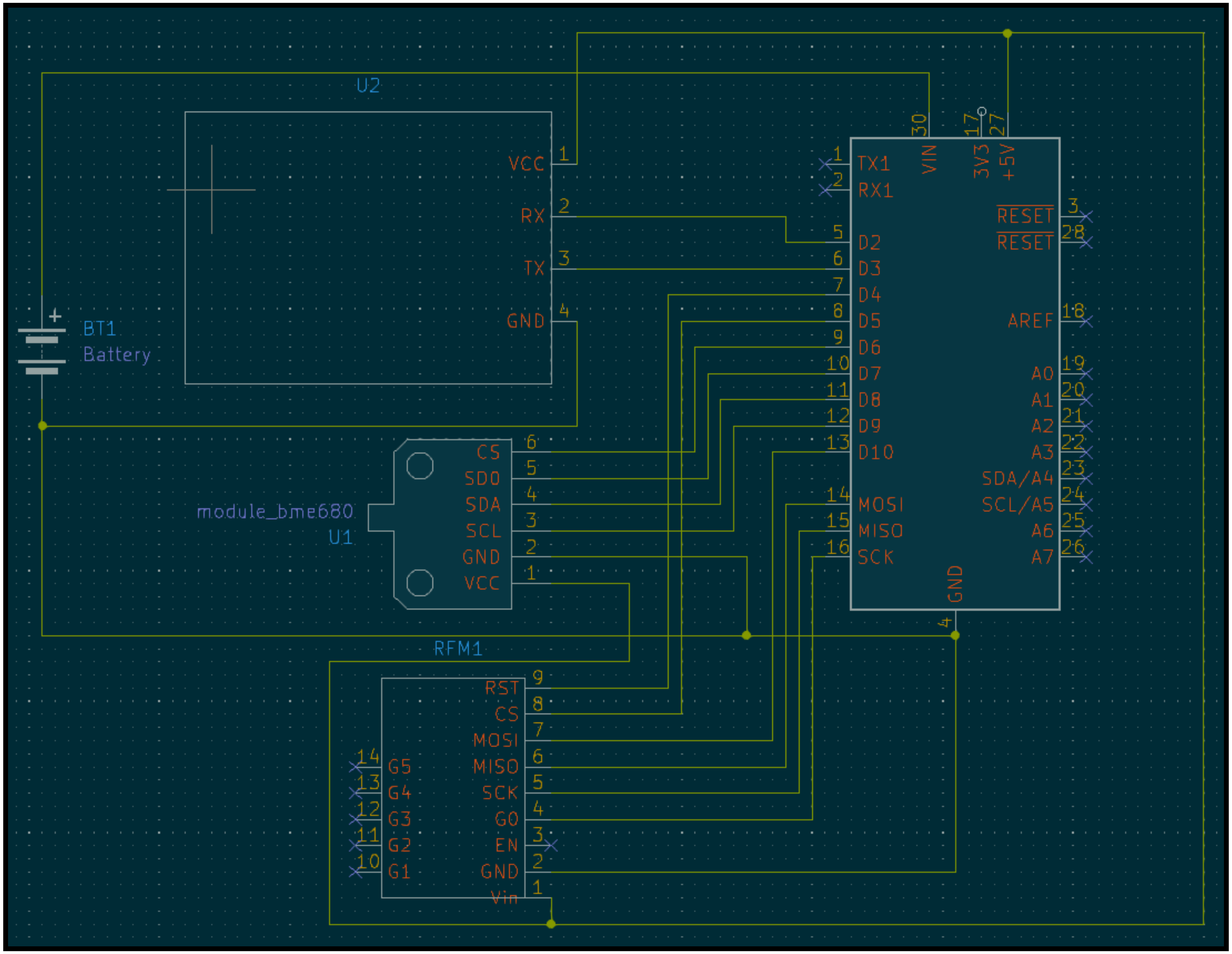


Figure 5: KiCAD Electrical Schematic

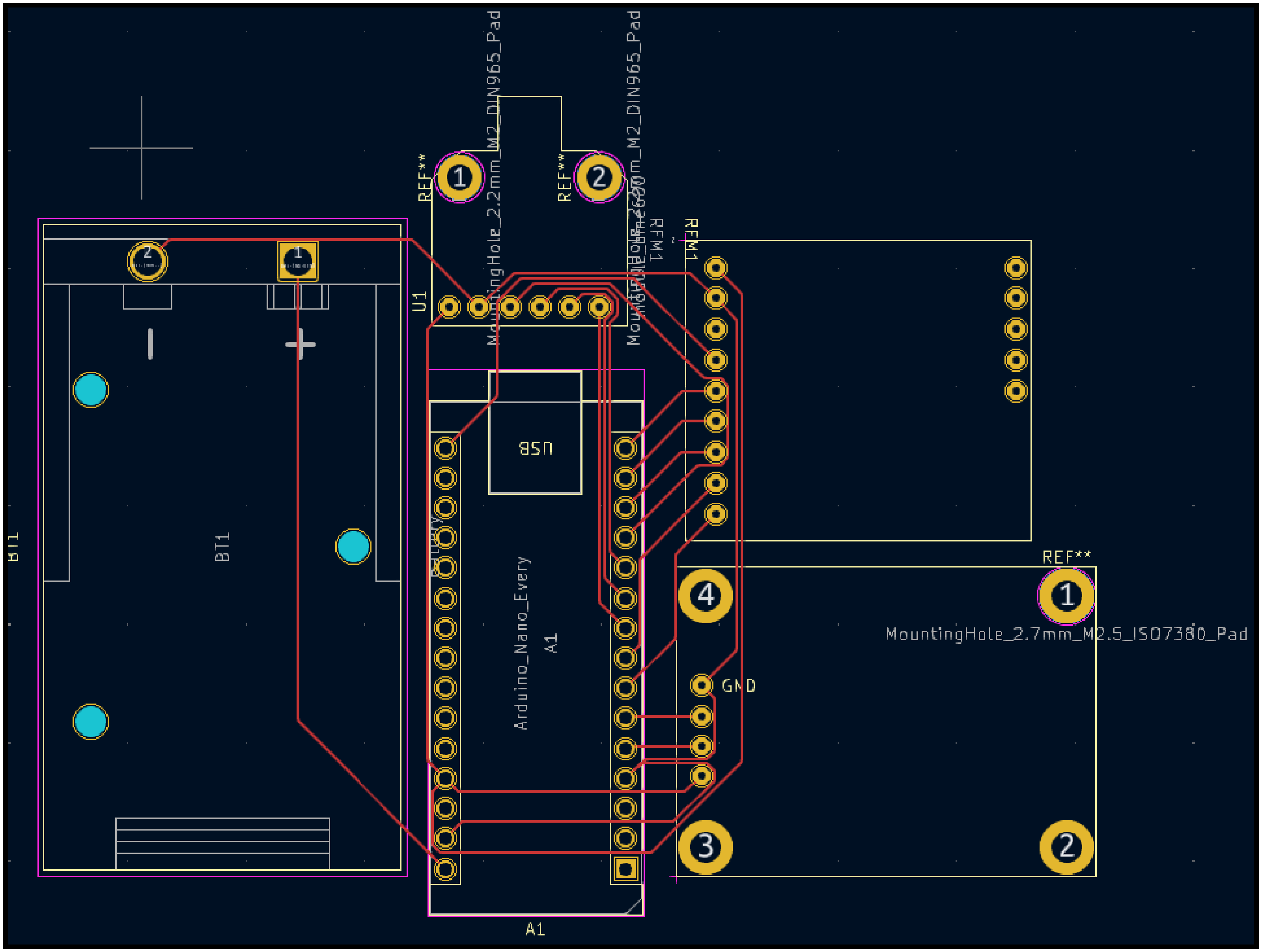


Figure 8: KiCAD Final PCB Layout

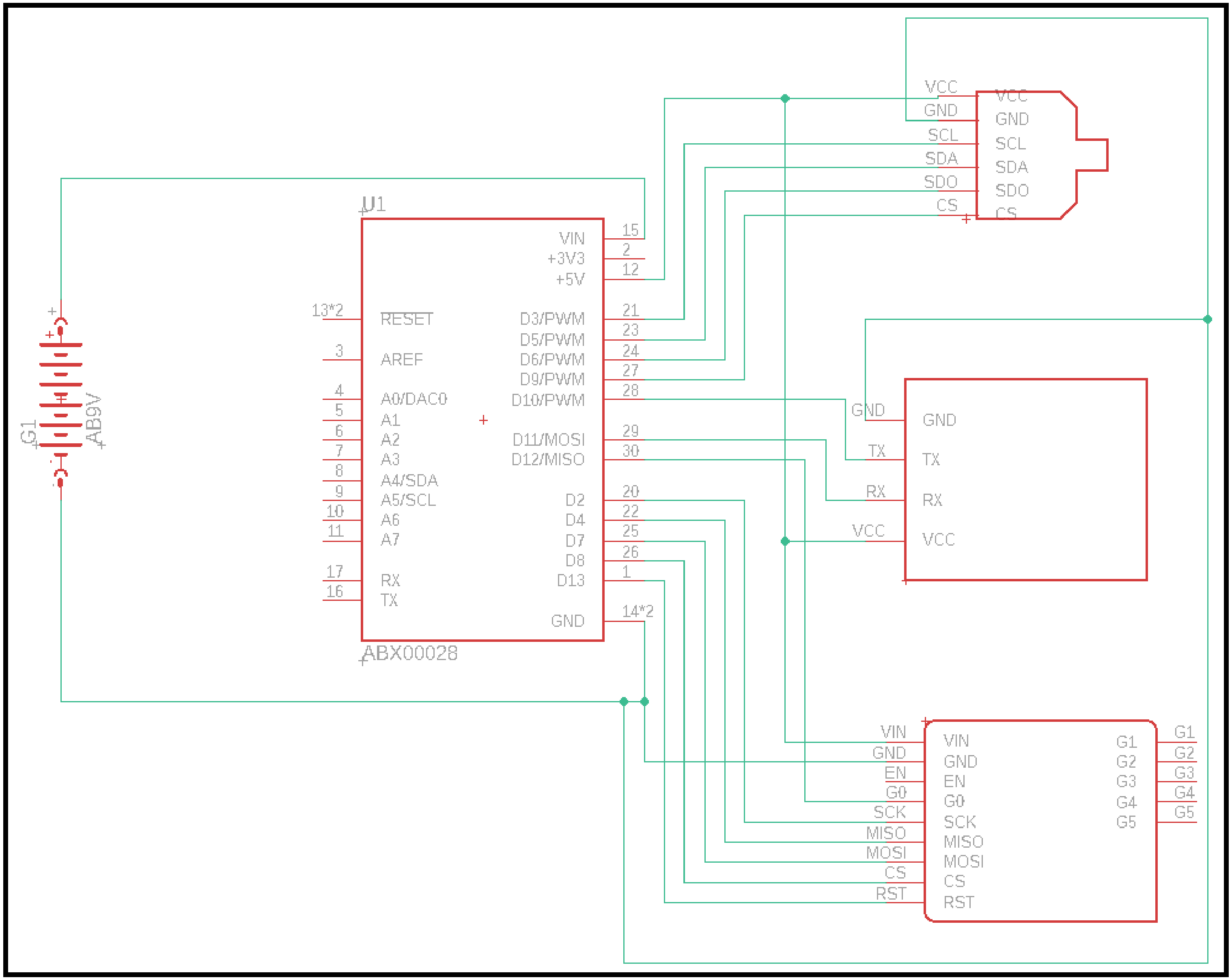


Figure 14: EAGLE Electrical Schematic

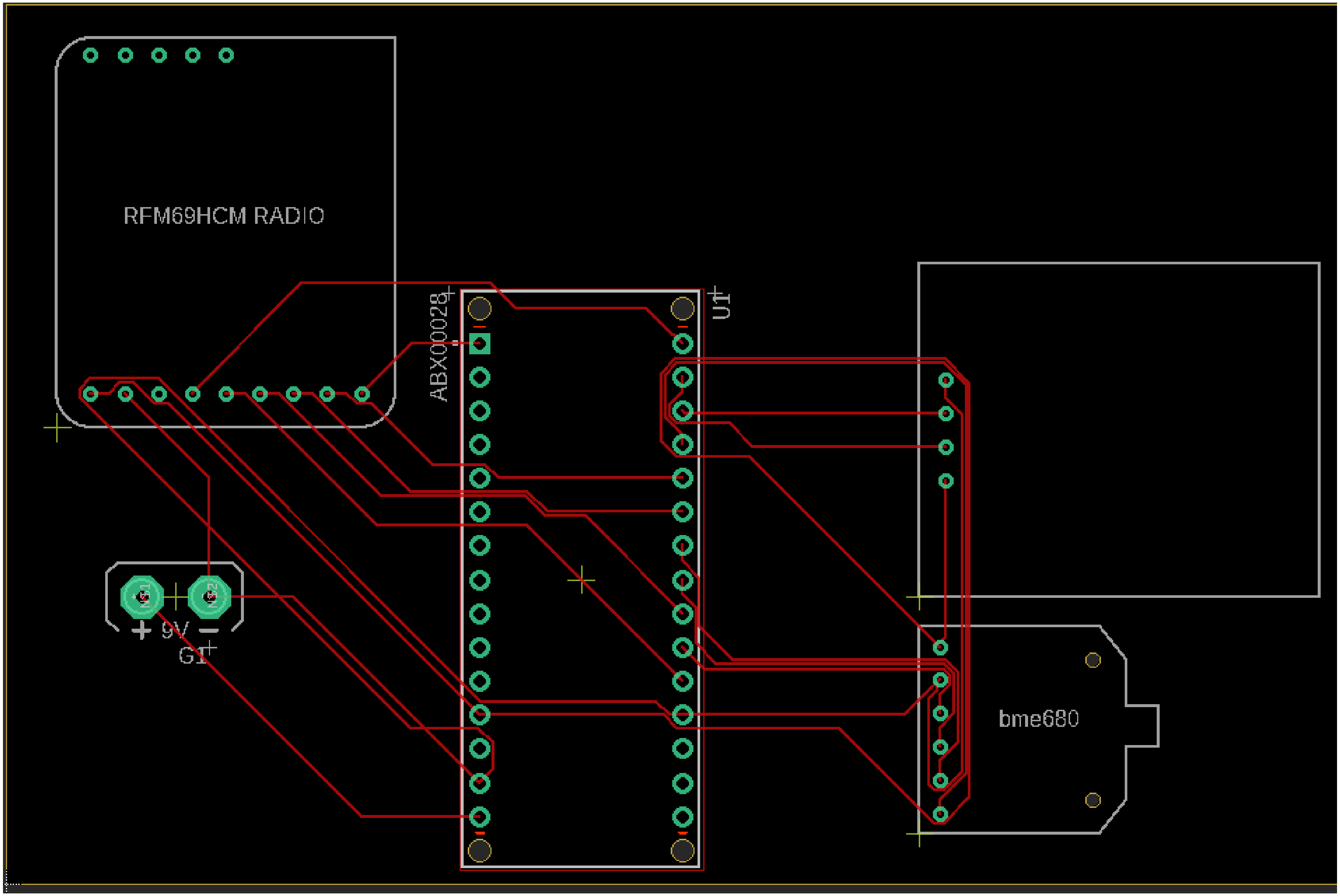


Figure 16: EAGLE Final PCB Layout

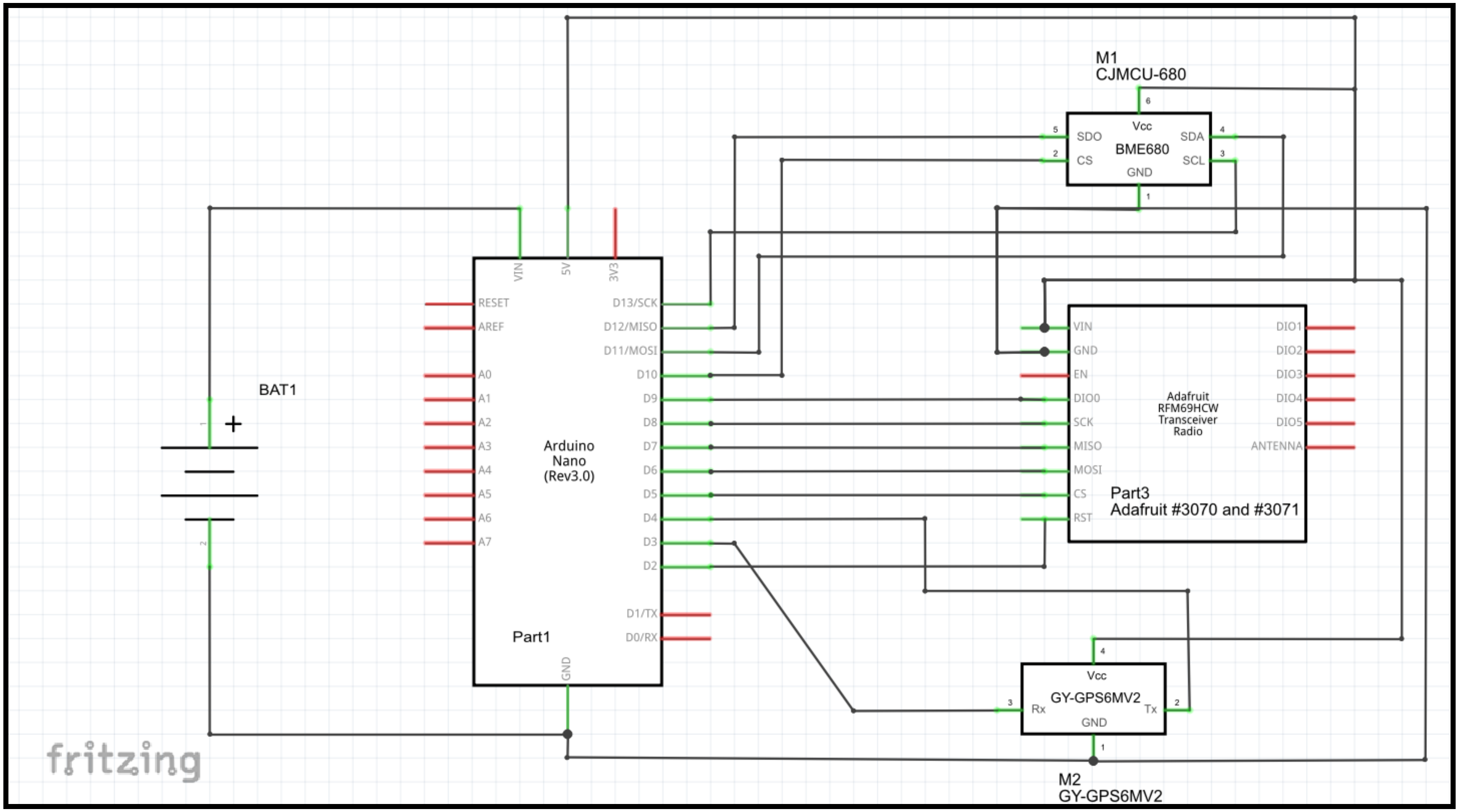


Figure 20: Organized Fritzing Schematic

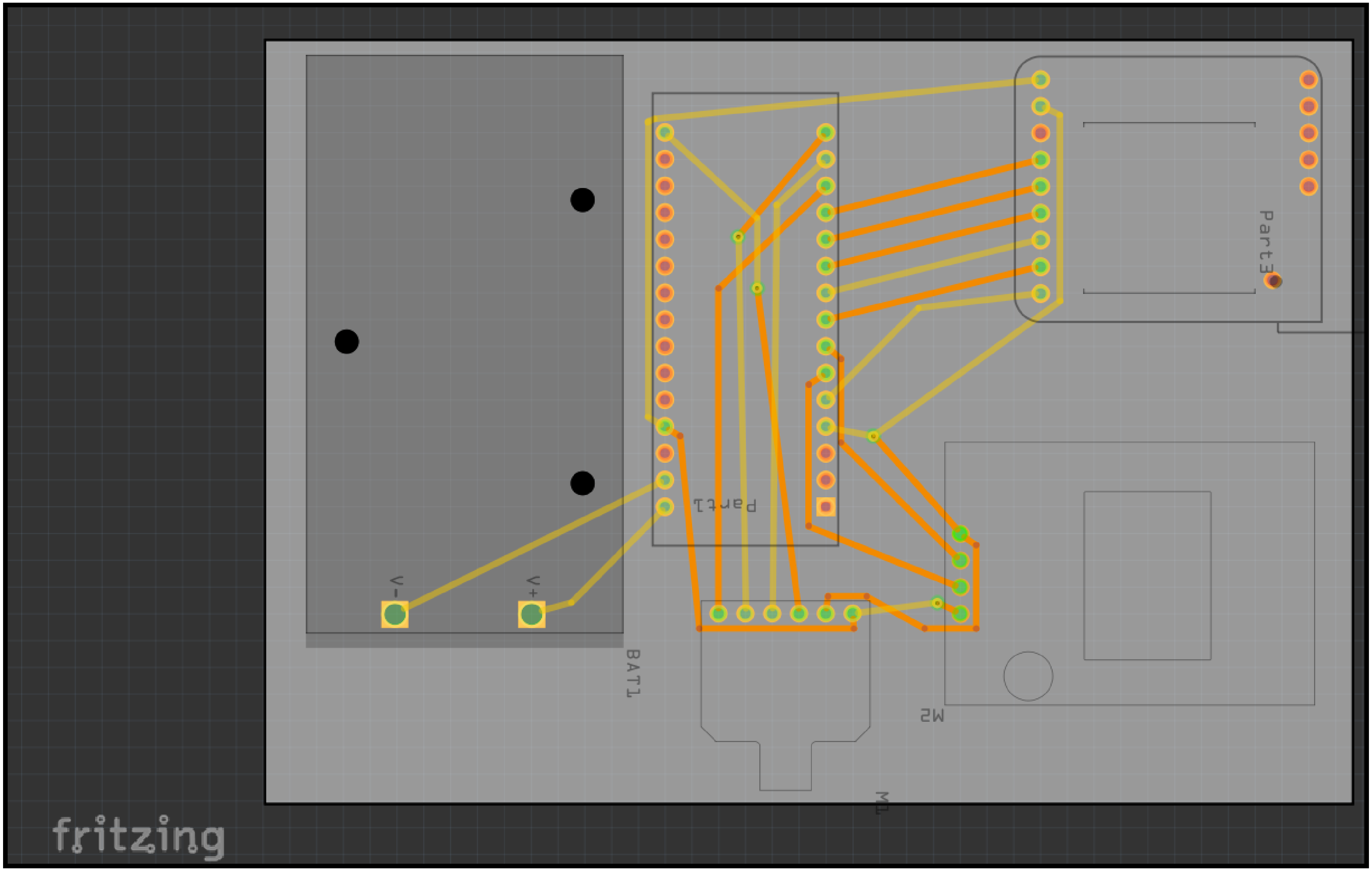


Figure 22: Final Fritzing PCB