# Conductive Rapid Inking Prototype


*Group 7 Authors:*

| Ethan Baker | Andrew Eppich | Jeremy Jiang | Giancarlo Luna |
|---|---|---|---|
| *Computer Engineering* | *Computer Engineering* | *Computer Engineering* | *Electrical Engineering* |

*Review Committee:*

| | | |
|---|---|---|
| Dr. Avra Kundu | ECE | Lab Manager |
| Dr. Nazanin Rahnavard | ECE | Professor |
| Michael Haas (tentative) | Tensentric | Vice President of Engineering |

*Advisors:*

| | | |
|---|---|---|
| Dr. Arthurs Weeks | ECE | Senior Lecturer/**Sponsor** |
| Dr. Chung Yong Chan | ECE | Senior Lecturer |

# Table of Content

# Table of Figures



# Table of Tables

# Chapter 2 – Project Description

## 2.1 Motivation and Background

Conductive ink and other additive technologies are relatively new methods of creating circuits. Traditionally, Printed Circuit Boards (PCBs) have been complex to design, requiring multiple stages to make a functional board. The simplest boards that most hobbyists can make are nothing more than milled copper using a traditional CNC machine. By using conductive ink, it is possible to make circuits in an additive method. Beyond creating PCBs on rigid surfaces, it allows for new inventive uses such as the uses that have been observed in the RF industry which allow for creating receivers on curved surfaces. Conductive ink also expands the possibilities of what materials can be used in PCB design, allowing for flexible PCBs or ones made for biosensors. Even the simplest uses of conductive ink expand the possibilities for the average individual to get involved in circuit design through conductive ink pens which can be used to literally draw circuits on paper.

The Conductive Rapid Inking Prototyping (CRIP) is a project for UCF's ECE department senior design course over Spring 2026 – Summer 2026. The CRIP is an additive manufacturing device sponsored by Dr. Arthur Weeks to add rapid PCB prototyping using conductive ink to UCF's manufacturing capabilities. By dispensing and then curing conductive ink on an appropriate substrate, it is possible to create both traditional rigid PCBs and also nontraditional PCBs on flexible and environmentally friendly materials. Additionally, it is possible to fix defective PCBs in a more permanent manner. Before this design, someone may need to jump a wire on the board or send a PCB back to design and manufacturing, which is often 2-week lead time at minimum. This solution will ultimately save time and money spent on production and shipping over boards that are made 'out-of-house'.

The overall goal of this project is to allow UCF ECE department students the ability to design and test circuits more rapidly and with non-traditional technologies, expanding what is possible for their projects. The base of the project will consist of a striped Creality Ender 3 printer, and the electrical and software components will be completely redesigned to implement a modular printing head system. The goal of this system is to allow for printing heads that perform different functions to be swapped out. So not only can there be a conductive ink head that performs printing of traces on the PCB, but also an insulator ink head that would allow for traces to cross over each other, enabling enable more complex circuits.

## 2.2 Existing Product, Past Projects and Related Work

### 2.2.1 Prior Previous Work

#### *2.2.1.1 Solder Paste Dispenser*

The Solder Paste Dispenser (SPD) [1] is a previous Department of ECE senior design project by group 4 of fall 2024/spring 2025. The goal of the Solder Paste Dispenser (SPD) was to take a modified Ender 3 printer and allow it to automate solder paste dispensing on a PCB when given the Gerber files. The paste would be dispensed via a pneumatic pump that would push the paste out. The SPD would manage quality control through a stereolithography system which could detect PCB placement as well as determining the height of paste mounds. The basic goal of the system was to achieve SMD sizes of 0603(imperial)/1608(metric), 0402(imperial)/1005(metric) as an advanced goal, and 0201(imperial)/0603(metric) as a stretch goal. The SPD replaced all the preexisting circuitry with a custom motherboard and custom motor drivers. The SPD utilized a Raspberry Pi 4 to host a webserver for uploading Gerber files, as well as managing the stereoscopy. It also uses an ESP32-WROOM-32D to control the motor drivers and pneumatic pump, interface with limit switches and touchscreen, and to connect wirelessly to the Raspberry Pi. The SPD lacked the ability to cure the solder paste and required the use of an external reflow oven to finish the process. This provided a good time saving method for people who had multiple boards to manufacture or lacked a stencil but required the use of a second device to finish the process and could not place components independently.

### 2.2.2 Existing and Related Products

#### *2.2.2.1 Voltera V-One*

The Voltera V-One [2] is a pre-existing 4 in 1 PCB design platform. The V-One is capable of dispensing silver ink, dispensing solder paste, milling with a drill head, and curing with a reflow heat bed. The Voltera V-One represents an ideal end goal for our project with its manufacturing prowess. It can use silver ink to make PCBs without milling, or it can mill copper boards to design more traditional PCBs. Once the silver ink is set, it can be cured using the built-in reflow hotplate. The V-One is also capable of milling to design two-layer PCBs using rivets to connect their layers. Finally, it can place solder paste to finish the board. Once all components are placed by hand, the V-One can act as a reflow hotplate to cure the solder paste. The V-One is currently listed at $3500 with shipping for the base option and $4200 when adding the milling head. While the V-One provides extensive manufacturing options when making a PCB, its cost keeps it out of the hands of most individuals without significant financial investment.

#### *2.2.2.2 Voltera NOVA*

The Voltera NOVA [3] is another manufacturing device produced by Voltera that is designed for manufacturing that is more like screen printing. The NOVA is more focused on designing circuits on flexible materials. It can design one sided circuits which are ideal for non-traditional boards or wearable electronics. The NOVA uses an insulator ink to allow for crossovers but is incapable of allowing for component placement on both sides of the board. Like the V-One, the NOVA uses EFD 5cc syringe barrels sold by Voltera but also allows for more freedom in material choice by sourcing materials from suppliers. The NOVA does not list a price but instead gives the option to request a quote from Voltera; however, the price of the smart dispenser and smart probe heads are $1599.99 and $1199.99 respectively, which is also prohibitive to many. The NOVA does not have a hotplate or any built-in curing method for materials which provides the need for another device for completing any circuit built by the NOVA.

#### *2.2.2.3 Circuit Jet*

The Circuit Jet by Electronic Inks [4] is a desktop PCB printer that uses inkjet technology like the NOVA to print PCBs on a variety of substrates. The printer uses cartridges similar to traditional ink cartridges, and they can be disposed of in the same way. Not too much is given on the manufacturer's site, but it can be inferred that the Circuit Jet functions in a similar manner to the NOVA and suffers the same drawbacks such as a lack of ink curing capability.

#### *2.2.2.4 Brother MFC-J450DW*

Brother Inkjet printers (or any other type of classic inkjet printer) can be modified to print with conductive ink [5]. Inkjet printers typically use piezoelectric or heat-based deposition methods. When a current is passed through, the nozzle will either heat slightly, creating a gas bubble depositing the ink, or the piezoelectric material will slightly expand pushing ink out. This option, while the least refined, is also the least expensive option. Using an inkjet printer cuts down on cost and allows rapid prototyping of circuits for under $200 depending on ink material and printer choice. However, the downsides of this method are numerous as the circuits are limited to printing only on paper. This means that traditional soldering or curing won't be a viable option as it will destroy the paper. Copper tape makes it possible to connect the printed traces to components, but tape is not mechanically solid for designing a production circuit. Using a traditional inkjet printer also means there is no way of curing the circuit, such as by UV, which means an additional machine will be needed.

#### *2.2.2.5 Aerosol Jet 5x*

The Aerosol Jet 5x designed by Optomec [6] is an advanced 5 axis additive inkjet circuit manufacturing machine. The 5x is capable of printing not just with conductive inks but with a variety of other materials. The 5x also has capabilities for UV curing and laser sintering. Using the inkjet methodology and 5-axis printing, the 5x can also print on a variety of material, which allows for both traditional hard and flexible PCBs. On the manufacturer's site there is no price listed, but the option is given to contact for a quote. The only main drawback besides the assumably exorbitant price is the lack of drilling ability, so it is only possible to create circuits that can connect SMD components on one side (unless the object is pre-milled as shown in one of the example videos).

## 2.3 Goals and Objectives

### 2.3.1 Project Goals

The main goal of this project was to build a conductive ink printer to create PCBs by modifying an Ender 3 3D printer into a modular system. The goals are listed below:

#### *2.3.1.1 Basic Goals:*

- Modify a Creality Ender 3 to function as a conductive ink printer with 8-16 mils trace resolution
- Design and implement a swappable head system to enable rapid switching between conductive ink dispensing and other potential functionalities such as insulator ink
- Develop communication protocols between the printer and swappable heads for automatic head detection and parameter configuration
- Ensure proper trace isolation and spacing (minimum 8 mils) per schematic specifications
- Enable the creation of crossover traces

- Use computer vision to validate printed traces against schematic specifications
- Design and implement a user-friendly interface for printer control and file management

#### *2.3.2.2 Advanced Goals:*

- Create a head that performs solder pasting
- Allow the user to manually intervene in the operation of the conductive ink printer
- Incorporate a hotplate with programmable temperature control for curing and bonding conductive ink deposits

#### *2.3.2.3 Stretch Goals:*

- Design and integrate a milling head as an additional swappable tool option to enable the milling of vias for double-sided PCBs
- Develop a head for component placement
- Create a head to evenly distribute and cure a varnish to create a solder mask on the PCB while avoiding covering pads that should be exposed
- Create a web server for people to remotely request PCBs to be printed, which then can be fulfilled by an operator
- Enable printing on non-planar surfaces

### 2.3.2 Project Objectives

The objectives to achieve the goals are as follows:

#### *2.3.2.1 Objectives for Basic Goals:*

- Research and select appropriate conductive ink formulations compatible with dispensing systems
- Design custom mounting brackets and adapters for the swappable head mechanism
- Develop firmware modifications to enable tool change functionality
- Implement a slicer to convert a copper layer schematic into G-code
- Find what amount of pressure on the ink is necessary to obtain various desired trace widths through the nozzle
- Calibrate X/Y/Z axis movements and dispensing rates to achieve precise trace placement
- Implement computer vision algorithms to automatically inspect and validate printed traces against the original schematic design
- Select appropriate touchscreen hardware, develop GUI software, and integrate with printer firmware for seamless user control

Our basic goals focus on providing advanced PCB prototyping capabilities to existing, functional 3D printing platforms. Using a swappable head design, we can expand the range of fabrication modes and applications beyond standard FDM printing. Additionally, developing multi-layer deposition techniques with dielectric insulating layers would allow the creation of crossover traces, significantly expanding circuit design possibilities beyond simple single-layer circuits. The implementation of intelligent tool detection and configuration allows the system to automatically adapt to slicer parameters based on the installed head type using unique head IDs. The firmware modifications enable seamless transitions between different fabrication processes with minimal manual reconfiguration.

The integration of computer vision for automated quality inspection provides users with real-time validation of printed traces against their original schematics, ensuring accuracy and reducing material waste from failed prints.

#### *2.3.2.2 Objectives for Advanced Goals:*

- Select and integrate appropriate solder paste dispensing hardware
- Incorporate a display directly on the printer for direct interaction
- Source a hotplate with programmable temperature control for reflow profiles

Conductive ink printers have a wide range of applications involving various PCB designs and complexities. Due to the unique requirements of complete PCB assembly, printing conductive traces alone is insufficient for functional prototypes. Many designs require component placement and soldering to create working circuits. To address this limitation while expanding the system's capabilities, our advanced goals provide the ability to create complete PCB assemblies through integrated solder paste dispensing and reflow soldering. These features streamline the entire fabrication process from trace printing to component bonding, minimizing manual intervention between stages. For example, a multi-layer PCB prototype can be fabricated over several hours with semi-automated transitions between conductive ink deposition, solder paste application, and thermal reflow for component attachment.

#### *2.3.2.3 Objectives for Stretch Goals:*

- Research CNC milling requirements and select compatible spindle/motor
- Develop multi-layer deposition techniques for creating crossover traces with insulating barriers
- Develop tool path generation algorithms for PCB milling operations
- Implement surface scanning or mapping capabilities for non-planar detection
- Develop adaptive G-code generation for curved surface printing

As previously mentioned, the versatility of PCB prototyping platforms requires them to accommodate various substrate materials and fabrication techniques. Traditional subtractive methods like PCB milling remain essential for certain applications, particularly when working with rigid substrates or creating isolated copper regions. Complex circuit designs often require traces to cross over one another without electrical connection, a capability typically achieved through multi-layer PCBs or insulating bridge layers. Advanced prototyping scenarios may also involve non-planar surfaces for wearable electronics or conformal applications. If the system is limited to flat, single-layer substrate printing, it cannot accommodate these specialized use cases. Our stretch goals introduce CNC milling capabilities as an additional swappable head option, enabling both additive and subtractive fabrication methods on a single platform. By implementing surface scanning and adaptive path generation, we can accommodate curved or irregular surfaces for conformal electronics applications. This expansion would transform the platform from a single-purpose conductive ink printer into a comprehensive PCB prototyping workstation capable of addressing diverse fabrication requirements that researchers and makers encounter in their projects.

## 2.4 Description of Features and Functionalities

As this project involves modifying an Ender 3, the features and functions that follow are designed to replace most electrical components and printer firmware as well as change the extrusion technology. The resulting system includes integrated features for operational control, design inspection, and verification.

Users begin by transferring a Gerber file of their PCB design to the slicer, which converts it to G-code for the printer. The file will then be transferred. The system processes the Gerber file and translates the circuit trace patterns into precise movement instructions for the printer. Once the PCB substrate is properly positioned, users initiate the printing process through an integrated interface. The interface provides complete control over printer functions, including process initiation, file management, manual head positioning, and importing data from the connected device.

Accurate PCB positioning is achieved through an L-bracket alignment fixture that establishes a consistent reference origin for all printer movements. A magnetic build surface secures the PCB substrate and prevents any movement during the ink dispensing process, ensuring precise trace deposition and minimizing printing errors. Adding a rubber base to the magnets would increase the friction to prevent slippage as well as preventing the magnet bases from heating up too much.

The printer incorporates an integrated camera mounted to the dispenser assembly for calibration and quality inspection. The imaging system establishes the PCB origin by detecting alignment markers, such as April tags on positioning brackets or board edges. This enables accurate trace placement relative to the substrate. The camera provides magnified views of deposited traces for manual inspection of trace width, spacing, and continuity. Users can verify each layer's quality before proceeding to the next, ensuring proper alignment and identifying defects such as breaks or spacing errors early in the fabrication process.

## 2.5 Prototype Illustration

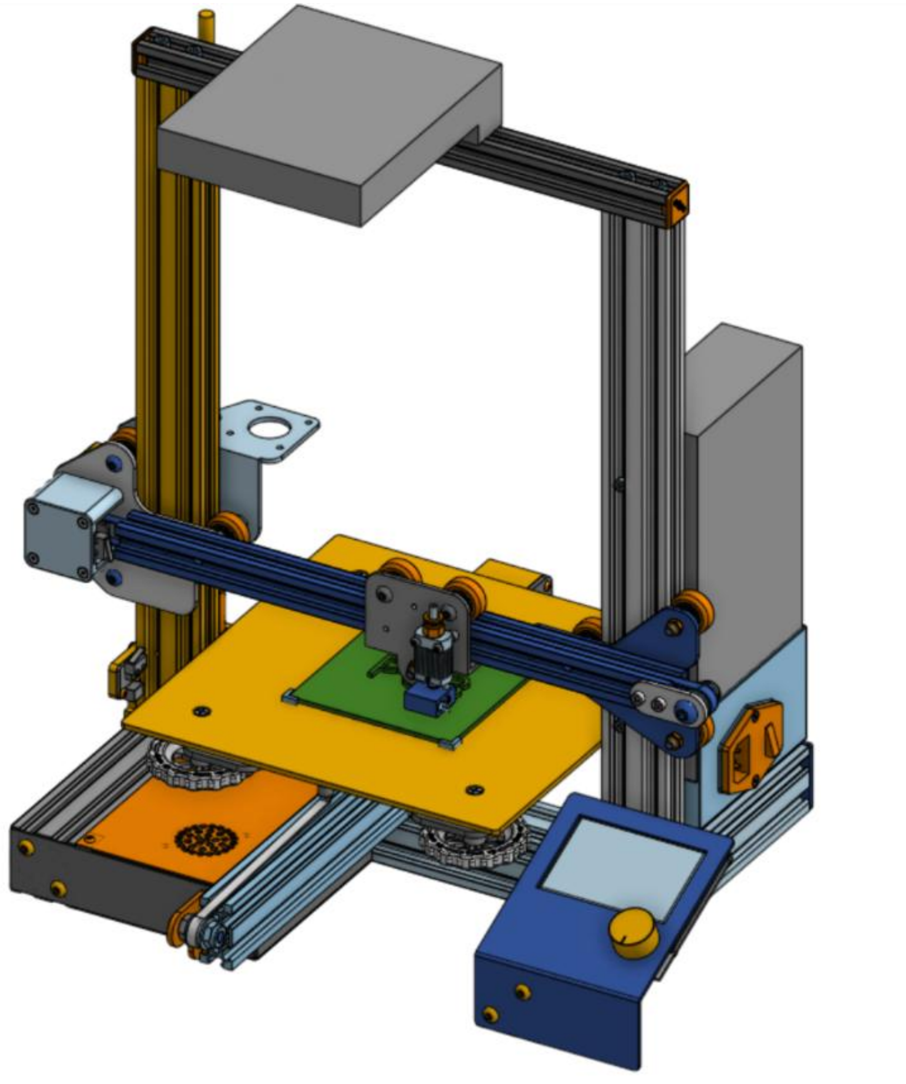


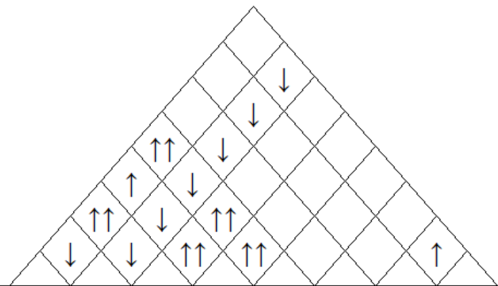
*Figure 1. Prototype Illustration*

This image represents a rough draft of the finalized product. The base of it will be a modified version of an Ender 3. Most of the mechanical body will be kept the same, but the printer's electronics will be replaced. The standard dial and screen will be replaced with updated components to further electrical component replacement. A camera will be added to capture an image of the PCB that will be lined up with and compared to the Gerber reference. Mounts will also be added to hold the PCB in place while the ink printer is active. The base of the printer will be replaced with a new heat bed to reach the higher temperatures needed for the silver ink to cure. The head of the printer will finally be replaced completely with a new head capable of storing an ink cartridge with a linear actuator to push silver ink onto the PCB.

## 2.6 House of Quality

| Relationships | |
|---|---|
| Strong Positive | ↑↑ |
| Weak Positive | ↑ |
| Weak Negative | ↓ |
| Strong Negative | ↓↓ |

| Direction of Improvement | |
|---|---|
| Maximize | + |
| Minimize | - |



| | Column # | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|---|
| | Direction of Improvement | + | - | - | - | - | + | + | + |
| Direction of Improvement | Engineering Requirements / Customer Requirements (Explicit and Implicit) | XYZ resolution | Cost | Minimum Trace Width | Minimum 2-terminal package | Minimum IC pin-to-pin pitch | Print Area | Connectivity | Power Consumption |
| + | Accuracy | ↑↑ | | ↑↑ | ↑↑ | ↑↑ | | | |
| + | Portability | | ↓ | | | | ↓ | ↑ | |
| - | Noise | | ↓ | | | | | | ↑ |
| + | Versatility | | | ↑↑ | ↑↑ | ↑↑ | ↑ | | |
| - | Manual Intervention | | ↑ | | | | ↑ | | |
| + | Responsiveness | | ↑ | | | | | ↑↑ | |
| - | Cost | | ↑↑ | | | | ↑↑ | | ↓ |
| | Target | 0.1 mm +/- | <=$1300 | <=16 mil | <=805 (Imperial) | <=1.27 mm (SOIC/SOP package pitch) | >=200 mm by 200 mm | Link rate is at least 50 Mbps | 120 V @ 60 Hz |

*Figure 2. House of Quality*

We felt the customer would prioritize the requirements listed above for various reasons. The machine should have high accuracy to be effective for PCB prototyping. If the traces are too large or overlap, then it would lead to a failure of a PCB. Increased accuracy also relates to increased versatility, which we feel is a big selling point for this machine. We hope to increase surface area and support as many PCB designs as possible.

In addition, we decided customers would prefer high portability, low noise, minimized manual intervention, and high responsiveness as quality-of-life improvements. The printer should be relatively portable to compete with other printers like the Voltera and beat large and immobile mills. Using an existing 3D printer somewhat restrains this portability in terms of dimensions, but increasing wireless connectivity and decreasing print area could help. For customers to willingly use this machine on their desktop or in a workshop, it would be preferable to keep noise low so as not to bother the user. Minimizing power consumption and spending more on quieter and more power efficient parts would help reduce noise. As for manual intervention, we would like to keep the need for it as minimal as possible, only requiring the user to upload the Gerber file, swap a head if necessary, and start the printing process. However, we intend to support complete control over the machine by the user if desired.

Additionally, we would like to minimize the cost of this project to make it a less expensive product compared to alternatives.

## 2.7 Budget and Financing

This project is being sponsored by Dr. Arthur Weeks from the Department of Electrical and Computer Engineering at UCF. The budget table is provided below:

| Subsystem/Category | Estimated Cost |
|---|---|
| Printer Hardware* | $280 |
| Dispensing System | $305 |
| Control System | $90 |
| Swappable Head System | $70 |
| Vision System | $70 |
| Mechanical Components | $80 |
| PCB Manufacturing | $140 |
| Shipping & Handling | $50 |
| Contingency (20%) | $215 |
| Total | $1300 |

*Table 1. Project Budget Estimates*

**The 3D printer was donated to the project and excluded from the total cost*

## 2.8 Milestones

| Milestone Number | Milestone Description | Start Date | Estimated End Date | Advisors and requirements |
|---|---|---|---|---|

| | | | | |
|---|---|---|---|---|
| 1 | Select Reviewers and sponsors | 1/13/26 | 1/26/25 | Get email agreement from all reviewers and secure sponsorship |
| 2 | Finish D&C Report | 1/13/26 | 1/25/26 | All team members finish writing respective sections of D&C report and we have 1 week to review |
| 3 | D&C Review Meeting | 1/26/26 | 1/29/26 | Review D&C report with Dr. Chan and finish any edits before due date on 1/30/26 |
| 4 | D&C Group Meeting | 2/2/26 | 2/4/26 | Meet with group advisor for approval of D&C and overall project |
| 5 | Upload D&C to website | 2/4/26 | 2/6/26 | Upload D&C to website and ensure all reviewers have access |
| 6 | Start Midterm Report | 2/9/26 | 3/26/28 | Work on midterm report and finish a week early to give more time to meet with advisor/reviewers |
| 7 | Start working on demo | 2/9/26 | 4/14/26 | Work on demo while finishing paper |
| 8 | Work on PCB Design | 3/10/26 | 4/28/26 | Work on PCB design for motherboard and order over break |
| 9 | Midterm Report Meeting | 3/30/26 | 4/1/26 | Meet with advisor to discuss midterm report |
| 10 | Finish Final Report | 4/6/26 | 4/21/26 | Finish writing final report once midterm report is finished |
| 11 | Record Demo Video | 4/14/26 | 4/21/26 | Record demo video |

*Table 2. Project Milestones*

## 2.9 Engineering Specifications

| Engineering spec | value | unit |
|---|---|---|
| Max flow rate | 0.4 | ml.s-1 (1.5 l.hr-1) |
| Ink laying precision* | .3 | mm +/- |
| Minimum trace width | 16 | mil |
| Print area | 200 X-Y, 5 Z | mm |
| XYZ resolution | .1 | mm +/- |
| Syringe capacity | 10 | ml |
| Substrate thickness | 5 | mm |
| Connectivity | 50 - 150 | Mbps |
| Curing temp* | 175 | C |
| Cure time* | 5 | minutes |
| Solubility* | water | n/a |

| | | |
|---|---|---|
| Cost | 1300 | $ |
| Max print speed | 200 | mm/s |

*Table 3. Engineering Specifications*

* - dependent on ink properties

## 2.10 Software Flowchart

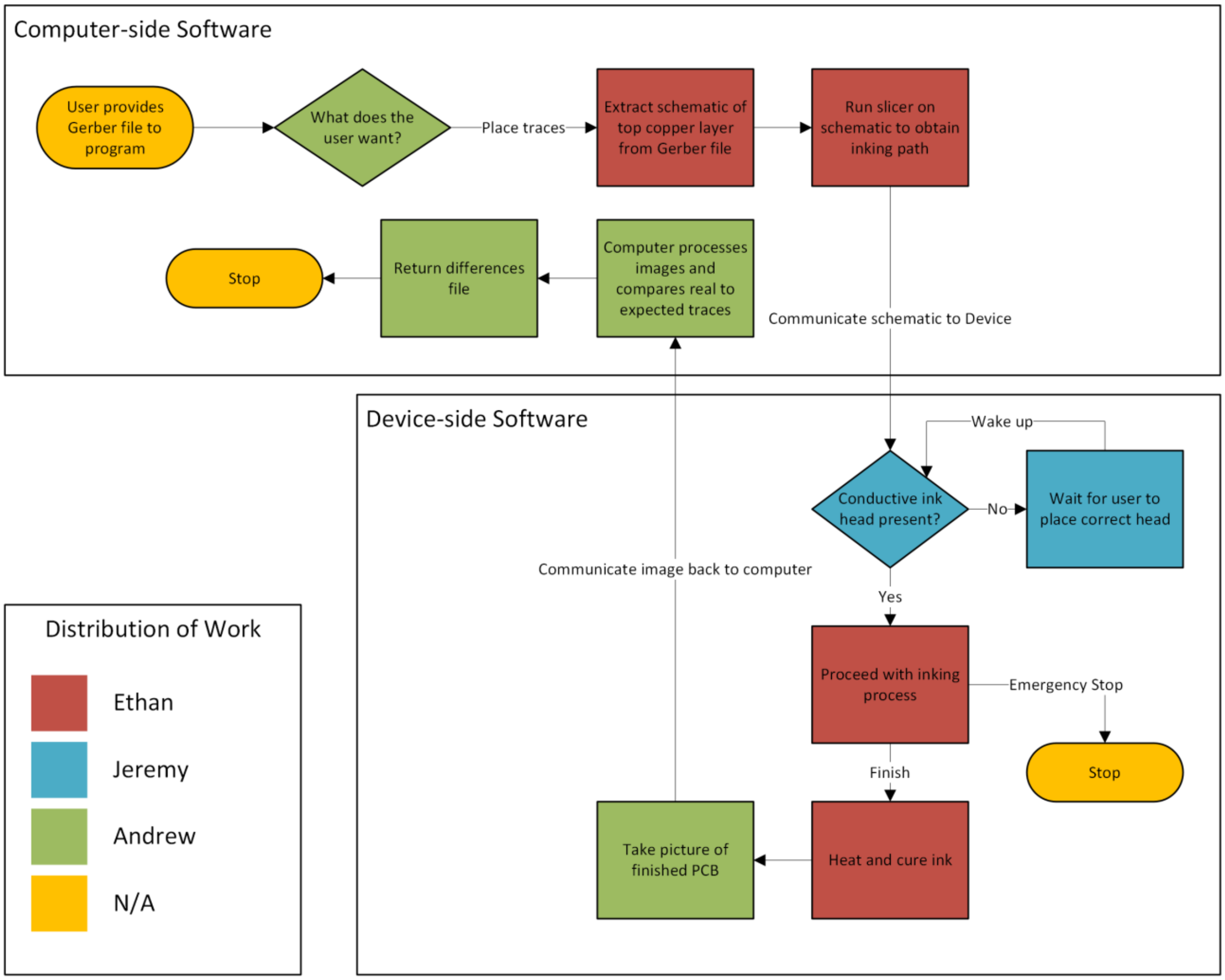



*Figure 3. Flow and interaction of computer- and device-side software*

## 2.11 Hardware Flowchart

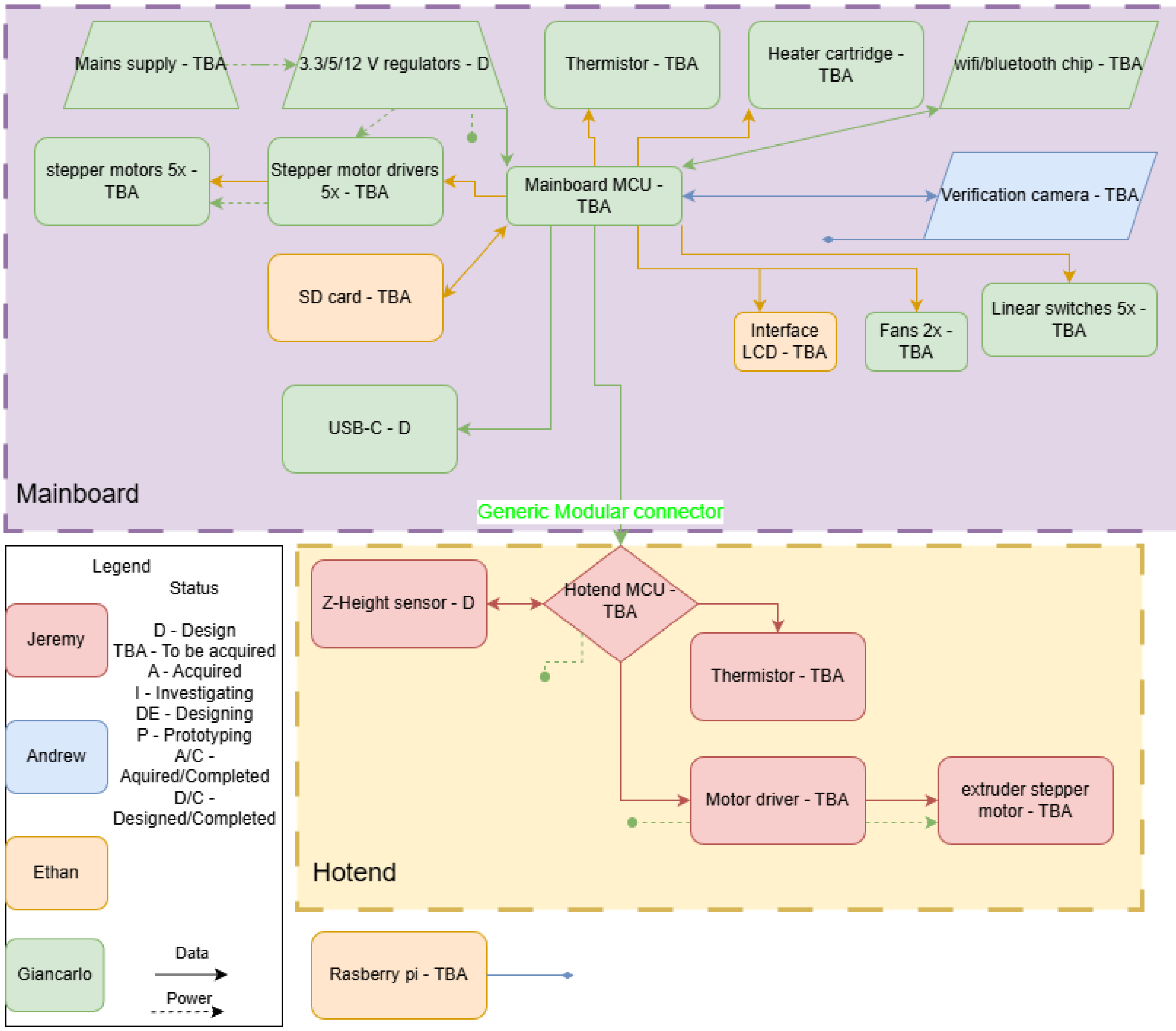



*Figure 4. Hardware diagram depicting all components and direction of data*

This hardware flowchart makes a couple of assumptions. That all data lines will use some kind of protocol and for legibility we chose to not denote the exact protocol. We will be documenting the exact protocols in use in another diagram better suited for this info.

# Appendices

## Appendix A – References

[1] "Project showcases," Spring 2025 Showcase – CREOL, The College of Optics and Photonics, https://creol.ucf.edu/academics/senior-design/spring-2025/ (accessed Jan. 25, 2026).

[2] "Voltera V-one PCB printer: Drill, dispense, and reflow," Voltera, https://store.voltera.io/products/v-one (accessed Jan. 21, 2026).

[3] "Flexible material dispensing - nova printed electronics platform," Voltera Inc., https://www.voltera.io/products/nova (accessed Jan. 25, 2026).

[4] "Circuit jet," Electroninks, https://electroninks.com/circuit-jet/ (accessed Jan. 25, 2026).

[5] Maximous, "Print conductive circuits with an inkjet printer," Instructables, https://www.instructables.com/Print-Conductive-Circuits-With-An-Inkjet-Printer/ (accessed Jan. 25, 2026).

[6] "AJ 5X," Optomec, https://optomec.com/aj-5x/ (accessed Jan. 25, 2026).