

## **Team 3 – Thy Mill be Done**

### ***Project Proposal and Feasibility Study***

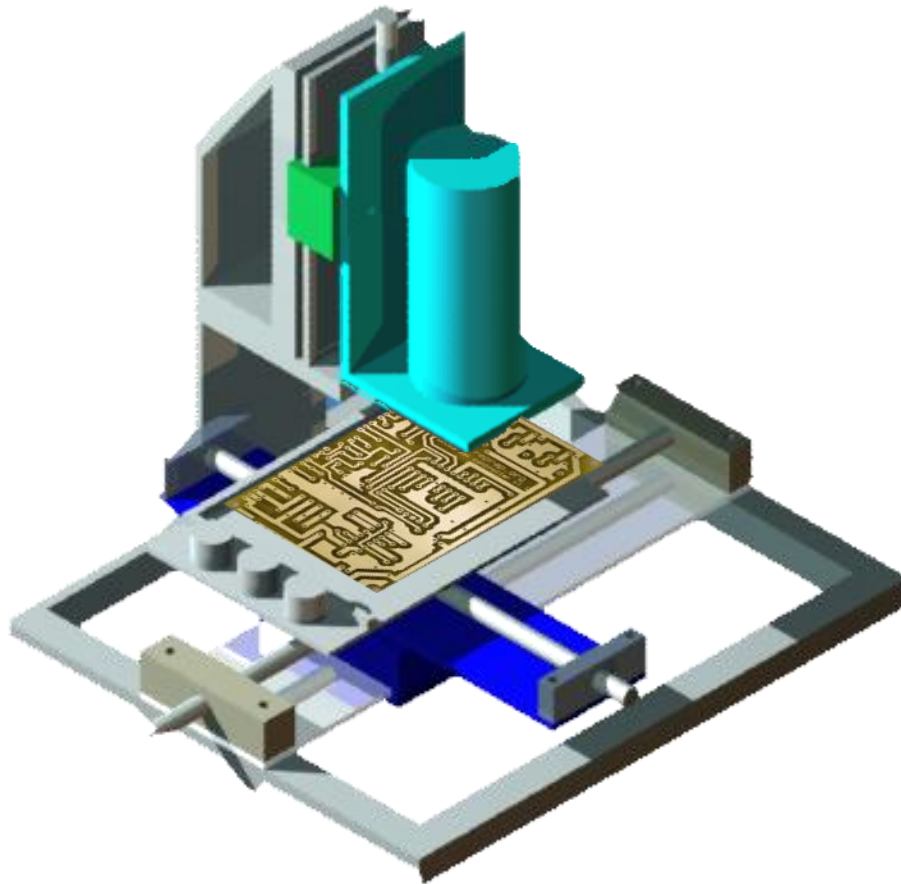
December 10, 2007

Dave Lyzenga

Brian Medema

Dan Michalowski

Aaron Scott



## **Executive Summary**

In today's market, printed circuit boards (PCBs) are often either fabricated at off-site processing plants or chemically etched in a time consuming and equipment heavy process. With these restrictions, many small design based businesses and universities lack a simple, affordable, and quick method to fabricate PCBs on site. Our project aims to remedy this problem by developing a computer numerically controlled (CNC) machine which uses drill and routing bits to fully fabricate a prototype printed circuit board. This feasibility study indicates that the project can be designed to safety and accuracy specifications, and can be done in the projected timeframe.

1	Introduction .....	1
1.1	Project Objectives.....	1
1.2	Market Analysis.....	1
1.3	CNC Machine Basics.....	2
1.4	Current PCB Fabrication Technology .....	2
2	Design Norms.....	3
2.1	Transparency .....	3
2.2	Trust.....	3
2.3	Integrity .....	3
3	Project Management.....	4
3.1	Team Breakdown.....	4
3.2	Management Organization .....	5
3.3	Project Timeline .....	6
3.4	Budget.....	7
4	System Design Process.....	7
4.1	Project Requirements.....	7
4.2	Tolerances.....	9
4.3	Operation .....	9
4.4	System Design .....	10
4.4.1	System Description .....	10
4.4.2	Electrical Design.....	10
4.4.2.1	Software Design .....	10
4.4.2.1.1	Current PCB Output.....	10
4.4.2.1.2	Translation into Router Movement .....	11
4.4.2.1.3	User Interface.....	11
4.4.2.2	Hardware Design .....	11
4.4.2.2.1	Electrical Systems Block Diagram .....	11
4.4.2.2.2	Position Control .....	12
4.4.3	Mechanical Design .....	12
4.4.3.1	System Description.....	12
4.4.3.2	Mechanical Calculations .....	13
4.4.3.2.1	Vibration Analysis .....	15
4.4.3.2.2	Sheer Stress for Bit .....	15
4.4.3.2.3	Safety .....	15
4.4.3.2.4	Router Speed.....	15
4.4.3.2.5	Precision Issues.....	16
4.4.3.3	Physical Design .....	16
4.4.4	Component Selection.....	17
4.4.4.1	Motors .....	17
4.4.4.1.1	Criteria .....	17
4.4.4.1.2	Alternatives .....	18
4.4.4.1.3	Decision .....	18
4.4.4.2	Drill and Routing Bits .....	18
4.4.4.2.1	Criteria .....	18
4.4.4.2.2	Decision .....	19
4.4.4.3	Ball Screws.....	19
4.4.4.3.1	Criteria .....	19
4.4.4.3.2	Alternatives .....	20
4.4.4.3.3	Decision .....	20
4.5	System Testing .....	21

4.6	Alternative Designs .....	22
4.6.1	Radial .....	22
4.6.2	YZ + X .....	23
4.6.3	Laser .....	23
5	Schedule of Development .....	24
6	Design Hurdles .....	25
7	Future Work and Alternatives .....	26
8	Conclusion .....	26

## Appendices

Appendix 1. DIP Part Specifications .....	A-1
Appendix 2. MathCAD Calculations .....	A-2

## List of Figures

Figure 1. Team 3 - Thy Mill be Done. ....	5
Figure 2. First Semester Project Timeline. ....	6
Figure 3. Electrical Systems Block Diagram. ....	12
Figure 4. XY Table. ....	13
Figure 5. Radial Design. ....	22
Figure 6. X Table, YZ Router. ....	23

## List of Tables

Table 1. Fabrication Costs and Lead Times. ....	2
Table 2. Preliminary Budget. ....	7
Table 3. Natural Frequency Calculations. ....	17

## List of Acronyms

CNC – Computer Numerically Controlled  
DIP – Dual Inline Package  
EMI – Electro-Magnetic Interference  
OSHA – Occupational Safety and Health Association  
PCB – Printed Circuit Board  
UV – Ultra-Violet  
USB – Universal Serial Bus

# 1 Introduction

## 1.1 Project Objectives

The primary objective of the project is to create a computer numerically controlled (CNC) machine to streamline the process of fabricating printed circuit boards. The machine will take in a design schematic (in a number of supported formats) and use a routing bit to mill away the copper on a raw PCB, outlining the intended traces and leaving the remaining copper as an insulating shield / ground plane. The machine will support two sided boards (by manually flipping the board after milling), and its primary function will be building preliminary electrical prototypes to find and correct design flaws before generating a more permanent PCB for further prototyping or the final product.

## 1.2 Market Analysis

The target audience of the machine is any organization that is PCB heavy, such as colleges and universities or design focused enterprises. Calvin College, like many colleges and universities, has a lot of the expensive equipment necessary for the long, manned process of creating PCBs (printing transparencies, baking the boards, exposing them, chemically etching, and curing them). This process can become increasingly frustrating given the number of redesigns necessary in the learning stages of PCB fabrication. Our design is aimed to be self-contained to avoid the continuous use of such equipment (or the need for it altogether), as well as alleviate the frustration of small design errors and their subsequent reprints.

An even worse situation often encountered by small hardware design firms occurs when a PCB has been designed and sent out to be spun by a company only to find a critical design flaw upon its return. It is not economically feasible for smaller companies to own and maintain their own PCB etching equipment simply for the designers' convenience, even if it would mean avoiding numerous day delays awaiting off site reprints. This project aims to solve these inefficiencies for design work, and aims to be an inexpensive chemical alternative that is economically viable for small business and colleges alike.

A third, underrepresented market of the device would be the at-home hobbyist who enjoys creating small analog electronics, but may not have access to the large amount of equipment necessary to etch boards. Sending schematics away to companies is often viewed as less "hands-on" for the home enthusiast. In this case, the prototyping tool could be a way for the user to hone her design skills before going off to locate the equipment necessary for a full-length etching of the final product.

The ideal consumer will use the device to prototype any electrical design. By working out design flaws and real-life issues that may not have simulated correctly, the user can then get the board fabricated in a more permanent method, knowing that it is the correct design. A professionally fabricated PCB would be better suited for mechanical prototyping and intense testing, but this device allows for the user to only have to go through the process of waiting for a professional board once (having hammered out their design flaws, previously).

### 1.3 CNC Machine Basics

CNC machine is an abbreviation for Computer Numerical Controlled machine. The idea behind a CNC machine is that a computer will send signals to control the speed and motion of electrical motors guiding a machine tool. The machine usually removes material from a raw piece of material in a subtractive way to create a fine-tuned part. The complexity and size of the parts created varies enormously from one application to the next. Most basic CNC machines allow the drill to move on all x-y-z axes, though they don't have to. More advanced drills will be allowed to rotate in the x-y planes as well as translate through them, allowing them to do more intricate designs. One of the major goals of CNC machines (what spurred their creation) is to allow for unmanned and efficient production of parts.

### 1.4 Current PCB Fabrication Technology

One of the goals of this product is to cut down on the time it takes to have a working prototype. Two of the most common methods for obtaining prototyped circuit boards are sending out a design to a third party circuit board manufacturer and doing it "in house," using a chemical-based etching process. Using a third party has some significant drawbacks. The first is that it can be expensive, especially taking into account the iterative process of prototyping. While it may be economical for large batches of circuit boards, there is generally a high fixed cost for only a few boards. Table 1. Fabrication Costs and Lead Times shows the estimated prices and lead times for three third party options. Note that in each case the price was calculated for a lot size of two boards. This was done to provide a fair comparison between options as there is a minimum lot size of two for certain companies.

**Table 1. Fabrication Costs and Lead Times.**

	Price (8 sq in)	Price (16 sq in)	Lead Time	Quantity
PCBexpress	\$60.00+shipping	\$90+shipping	24 hrs	2
ExpressPCB	\$67.40+shipping	\$77.8+shipping	24 hrs	2
PCB123	\$88+shipping	\$97+shipping	24 hrs	2

While this fixed cost may not seem particularly high at first, if a design is reworked and improved multiple times, the project cost can increase rapidly. Also, if even a small error is found and the board must be redone, the fixed cost must be paid again. Another major drawback of the third party approach is the time it takes. It can take days to get a board back (lead time plus shipping time), and if the board is discovered to be flawed, or if it needs to be changed for any other reason, the process must be repeated.

The advantages of doing a board "in house" are that it is faster and, if prototypes are frequently needed, it can be cheaper. However, the process requires a large amount of equipment, which can be expensive. First, a transparency must be developed which only allows light to pass through where the traces are desired. This can be done with any number of freely available circuit board layout editors, but care must be taken to print an adequately opaque transparency. Next, a photo-resist material sensitive to ultraviolet light must be laminated against a blank copper-plated fiberglass circuit board. Care must be taken so that there are no wrinkles in the photo resist.

Next, the board, with the transparency vacuum sealed on top of it, must be exposed to ultraviolet light for approximately five minutes. When the board is finished being exposed to the light, the spare photo resist must be peeled off. This leaves a small layer of the photo resist where the traces will be. The board now must be baked in an oven for another five minutes, hardening the photo resist.

Next, the board must be developed, which again hardens the remaining photo resist. This step requires a special developer, which bathes the board in a chemical developer. Now that the photo resist is adequately hardened, it can be etched. This process, like the developer, requires a machine to continuously spray an etchant such as ferric chloride, which removes any copper not protected by the photo resist.

Finally, the board is treated by one last chemical, which removes the remaining photo resist, leaving only the desired traces. The board is now ready to be drilled, which can be done by either an automated drilling machine or by hand. This whole process can take hours, while requiring constant attention to advance the process to the next step and make sure everything is done correctly. Also, there are both fixed equipment costs and variable costs associated with this process. The fixed costs are the UV exposing box, the oven, the laminator, the developer, and the etcher. The variable costs are the photo resist and chemicals. While this is much faster than sending a design to a third party, it is still very labor intensive and error prone.

## **2 Design Norms**

There are a number of design norms we as a team aim to hold during the design and production of our product. We've concluded that the three most important norms to hold to are transparency, trust, and integrity.

### **2.1 Transparency**

Transparency will come into play as we design the functionality of the device. It aims to be user-friendly, simply designed, and open about the mechanics of its operation. It should be intuitive and well documented, allowing the user full command over its functions without hiding any part of its operation. The reference material will be fully developed and include everything the user needs to know to interact with the product efficiently and safely.

### **2.2 Trust**

Trust is another primary design norm and revolves around the durability, safety, and reliability of the device. We aim to meet all of the stated tolerances, to exceed the usage amounts we promise out of the components, and to maintain a safe and reliable operation.

### **2.3 Integrity**

Integrity is the third design norm we aim to uphold in all areas of the design. We are accountable to ourselves, our customers, and to God in our design. Even when there is a great deal to gain by taking design shortcuts or enhancing test results, we are held to a higher standard through the Christian principle and design norm of integrity.

The overarching themes of honesty and integrity will permeate the design process as we attempt to build a stewardly and trustworthy product in all regards. In this design process, we are looking to use our God-given talents to the best of our abilities in an appropriate and honoring way.

## 3 Project Management

### 3.1 Team Breakdown

From a project management perspective, the primary goal of the project is for each of the team members to learn as much as possible about their specific managed roles along with those of the other members. The philosophy is that we will attempt to share the details of the work evenly across the concentration gap in order for everyone to get a hands-on feel for the entire project. While this is a great theoretical benefit to the team, the need does arise for individuals to ultimately be responsible for the decision making and execution of each part of the project. So while we will all attempt to be usable in all areas, the specifics breakdown as follows:

Dave Lyzenga – Dave will be primarily responsible for the electronic hardware design for the project. The main areas of design include writing the VHDL for the FPGA, prototyping and wiring the physical circuits, and creating the in-depth wiring diagrams for the documentation.

Dave is also responsible for editing and compiling the written documentation of the project, functioning as chief editor for the major publications.

Brian Medema – Brian is principally responsible for the design and implementation of everything relating to movement on the z-axis: from vibrational and finite element analysis to motor speed determination. Brian is also the lead designer responsible for the method to be used to change bits, whether it be manual or automatic.

Dan Michalowski – Dan will be the primary developer of all things related to movement of the XY table and its functionality. He will be looking into calibration techniques, substrate choice, maintaining a level surface, and other design concerns relating to the table.

Aaron Scott – Aaron's principle duty will be to develop the software for the entire design which includes the C code, assembly instructions, and the user interface. He will also be working with Dave to bridge the gap between the hardware and software to develop the fully embedded system.

A team picture can be seen in Figure 1. Team 3 - Thy Mill be Done.



**Figure 1. Team 3 - Thy Mill be Done.**

### **3.2 Management Organization**

The entire team will primarily communicate in the group meetings twice a week, along with any breakout sessions scheduled. On top of this, the electrical and mechanical pairs have individual design meetings about once a week. The primary communication method outside of these times is via e-mail.

All electronic work is stored on the commonly accessible network team space in order to give everyone full access to any of the project files.

Design consultant meetings are planned in advance with everyone's knowledge in order for every member to attend, regardless of individual team role or concentration.

The philosophy of this team's management is open communication, honest and prompt conflict resolution, and an attempt to encourage interdisciplinary growth as we explore the gifts God has given us.

### 3.3 Project Timeline

**Error! Reference source not found.** shows the timeline for the first semester of development.

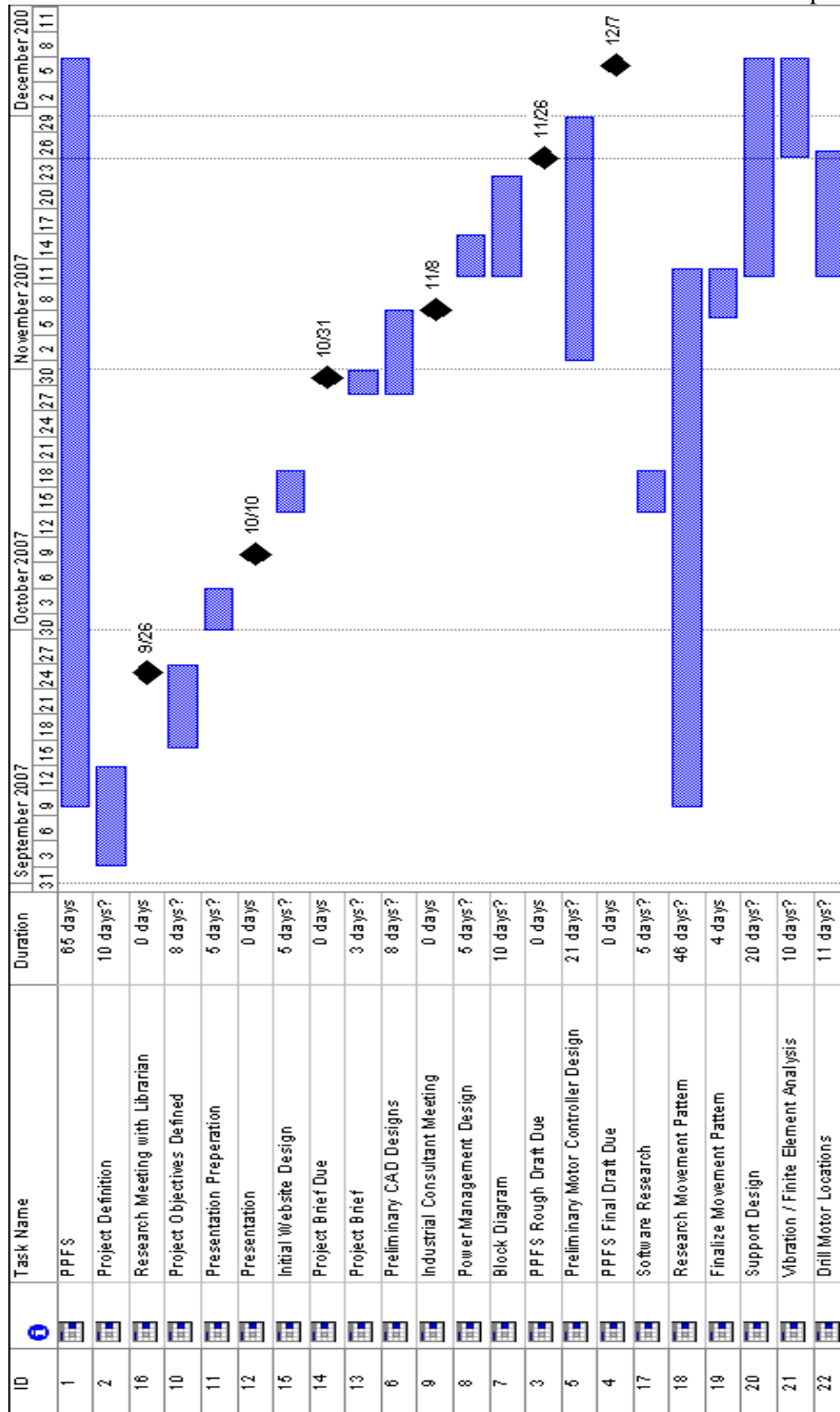


Figure 2. First Semester Project Timeline.

### 3.4 Budget

An estimation of the project prototype cost can be seen below in Table 2. Preliminary Budget.. The aluminum for the frame will hopefully be gleaned from Calvin's metal shop, and there is a high probability that many of the electric components will be available as well.

**Table 2. Preliminary Budget.**

Expected budget		
Dremel Tool		\$49.99
Drill Bits	1/32"	\$11.25
	1/16"	\$9.45
	1/8"	\$9.45
	shipping	\$7.00
		\$37.15
Aluminum for frame		\$0.00
Ball Screws		\$75.00
Stepper motors	X-axis	\$30.00
	Y-axis	\$30.00
	Z-axis	\$30.00
		\$90.00
Electronic components		\$150.00
Total		\$402.14

## 4 System Design Process

### 4.1 Project Requirements

#### I. Physical and Overall Design

- a. Aim to be semi-portable, desktop sized, and movable by a single person (aim for 60lbs or less)
- b. Be large enough to operate on copper boards 6" by 8" in footprint size
- c. Support dual inline pin sized components
- d. Should maintain a significant price and time advantage over chemical etching (aiming for around \$500 production cost)
- e. Should be ergonomic, efficient, and aesthetically appealing

#### II. Power

- a. Will use wall-power and not require any battery power
- b. Use standard 120V, 60Hz input

### III. Reliability and Maintenance

- a. The bit should be easily replaceable and changeable to provide quick alternation between drilling and routing functionality
- b. Appropriate safety measures should be in place with accordance with OSHA standards
- c. Include an emergency stop button
- d. The design should eliminate vibration error, following the overall tolerance design of  $5/1000^{\text{th}}$  of an inch
- e. The drill and routing bits should be reliable (aim for resilience through 100 boards)
- f. The bit will need to have a high capacity for managing sheer-stress
- g. There will be a protective substrate layer under the board to prevent the drill bit from breaking should it miss or puncture through the board
- h. The waste copper will be collected automatically, be it through a vacuum, gravity, or some other method

### IV. Operation

- a. From the moment the Start button is pressed, the process should be completely automated through completion of its current process (drilling or routing)
- b. The process should be significantly faster than the chemical etching process, aiming to be completed within 15 minutes
- c. The drill should be accurately placed to within  $5/1000^{\text{th}}$  of an inch
- d. The PCB will move on the X and Y plane and the router will move on the Z axis
- e. There will be a clamping system to secure the board in one of the corners
- f. The process will be user friendly, with simple push button operation and feedback LEDs to provide the machine status
- g. Motor speeds will be produced through testing to ensure efficiency and lack of slipping
- h. The PCB should be able to support components as small as a dual inline package (DIP). See appendix 1 for specifications.

### V. Interface

- a. A software package will be included to convert a number of supported schematic files (from current PCB fabricating software) into drill movements readable by the machine
- b. The interface with the machine should be a commonly available connection type such as USB

### VI. Additional and Expansion Objectives

- a. The device could be self-calibrating

- b. Two routing bits could be employed for either fine or mass milling
- c. Two modes of operation could be provided, allowing for the removal of the large amount of copper left after tracing to provide a more accurate, non-shielded response
- d. The PCB could be movable in x and y directions simultaneously to allow for arced and curved motion
- e. Potential support for surface mount parts could be added

## 4.2 Tolerances

### X+Y tolerances

We selected a value of .005 inches for the maximum tolerance for both the x plane and the y plane. Both planes encounter similar tolerance problems. They will both be using ball screws with a lead variance of .001in/foot. The lead variance is over the entire length of the screw, so it is only included for the most conservative estimates. The variance from backlash and lubrication will be about .001 inch in total, since preload is used for the ballscrews. The variation from the stepper motors is almost negligible, at the very most being .0004 inches. This leaves about 3 thousandths of an inch tolerance for component deflection from axial deflection in the screws and mostly bending deflection in the support structure.

### Z tolerances

The main factor in determining the z-tolerance comes from the maximum depth we want to cut into the board. We measured a typical board to be 63 thousandths of an inch thick, with the copper itself only about 2 thousandths of an inch thick. At the very most we wanted to be cutting through half of the distance of the board, if it were single sided, but this is the absolute maximum. Our more common implementation strategy (allowing double-sided boards) gives a maximum tolerance of 15 thousandths of an inch for the z-axis. In this tolerance 1 thousandth is taken up from lubrication and backlash in the screws (although lubrication is an uncertainty, it will probably assist our spring loaded ballscrews by eliminating excess lead contact space). Lead variance is negligible because of the small distances moved in the Z direction. Motor tolerance is also considered to be negligible.

A tolerance for 1 thousandths of an inch is left for height variance across the surface on which the PCB is placed. The natural bow in the PCB is a large contributor to the tolerance at about 5 thousandths, as we haven't yet developed an in-depth system for reducing it. There will also be play due to the sliding system for the X and Y axis, which will be around 1 thousandth. This leaves a generous 7 thousandths for component deflection. We doubt it will be this large, but this is a conservative estimate.

## 4.3 Operation

The plan is that the device will operate on the concept of simplicity. After creating a design file and using the developed tool to convert it into a drill movement file readable by the machine, the user need only transfer the file to the machine (planned through USB), await the ready signal, and press the "Go" button to initiate movement.

The device will automatically calibrate to the board, which will be placed at one of the corners of the XY table, and begin routing based on the converted schematic file. When the traces have been completely routed around, the device will indicate completion of the first phase. The user then has the option of

drilling through-holes for component placement (also indicated in the schematic file). The user will be prompted to switch to the drill bit and press the “Go” button a second time to initiate phase 2.

The device will again signal completion when the holes have been drilled. At this point, one side of the board will be completely routed, and all of the through-holes will be drilled. The alternate side of the board could also be routed as an optional stage 3 (using a second schematic file), or the design can be completed.

At any point during operation, the user can cancel the board manufacturing by hitting the “Stop” button. Depending on the difficulty of the design, the goal is to allow the user to either cancel the process altogether or pickup the operation at the stopped point.

When completed, the user will potentially have a board that is routed on two sides, drilled through each of the part holes, and ready for a quick bur brush-off and primary prototype testing.

## **4.4 System Design**

### 4.4.1 System Description

The commands to the machine will come from the controller. The controller will consist of a microprocessor, some external memory, and interfaces to the motor and host computer. The controller will be responsible for calibration of the X, Y, and Z axis, receiving the program from the host computer, responding the user interface, controlling the motors during operation, and monitoring position data.

The controller will operate in a hybrid open-loop configuration with a closed-loop feedback system that will tell the controller when the error, or actual position relative to the commanded position. The motors will be commanded using micro-stepping techniques. The size of the step will be at least one half step and is a function of both desired precision and mechanical tolerances. To operate in an open-loop fashion, the controller will micro-step the motors to the commanded position and a fixed rate of speed. The main control algorithm will not require any type of feedback to do this. However, there will also be an optical encoder generating feedback position data, if deemed necessary in the final design. This will give the controller a means of calculating actual verses commanded position. If for some reason the actual and commanded position differ by a preset amount, an exception will be thrown and the controller will enter closed-loop mode, where another algorithm will reposition the table to a point back on the path, and the controller will again resume open-loop mode.

### 4.4.2 Electrical Design

#### *4.4.2.1 Software Design*

##### 4.4.2.1.1 Current PCB Output

There will need to be a software program that will translate a circuit board drawn from a standard layout program into a program that the controller understands. This will be done by taking a standard file such as Gerber or Excellon and converting it into a text file containing position points as well as raise and lower commands. A Gerber file is a standardized format that contains information for a CNC machine to draw exact patterns on printed circuit boards. An Excellon file contains information about the holes that need to be drilled, including location and diameter. These points will essentially be a connect-the-dots illustration of the printed circuit board outlining the desired traces. This program will also need to know both the tool type and size so that it can check from any type of clearance issues.

#### 4.4.2.1.2 Translation into Router Movement

The Gerber and Excellon files will be translated into G-code, which is the RS-274 machine tool programming language, using freely available conversion scripts. The program that the controller sees will be a list of vertices with instructions to raise and lower the bit interspersed. Therefore, a program to convert the G-code into a list of vertices will need to be developed. The controller will interpolate between two points in a straight line, micro-stepping the motors from point to point. Curves will be approximated by a series of points close enough together to adequately resemble a smooth curve.

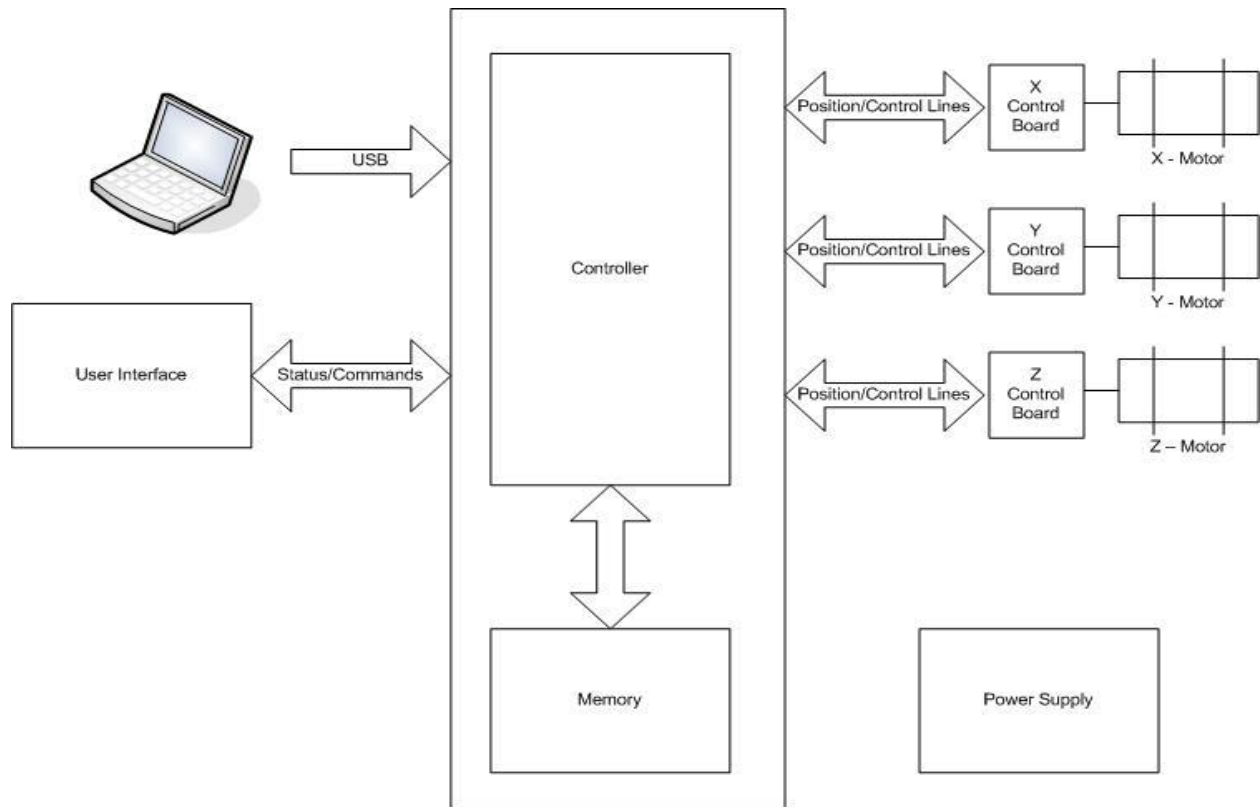
#### 4.4.2.1.3 User Interface

The program will be loaded from the host PC via a USB port. The program will then be stored into a memory device (eg SRAM, SDRAM, FLASH, etc). During the printing process, the controller will read each data point from memory and tell the motors how to get there.

### 4.4.2.2 *Hardware Design*

#### 4.4.2.2.1 Electrical Systems Block Diagram

Figure 3. Electrical Systems Block Diagram illustrates the basic block diagram of the electrical hardware components. The major blocks are the host PC, the controller, the power supply, the motors, the drivers, and the feedback devices. The host PC will communicate with the controller via USB. The power supply will supply voltage to the router, controller, motors, and drivers.



**Figure 3. Electrical Systems Block Diagram.**

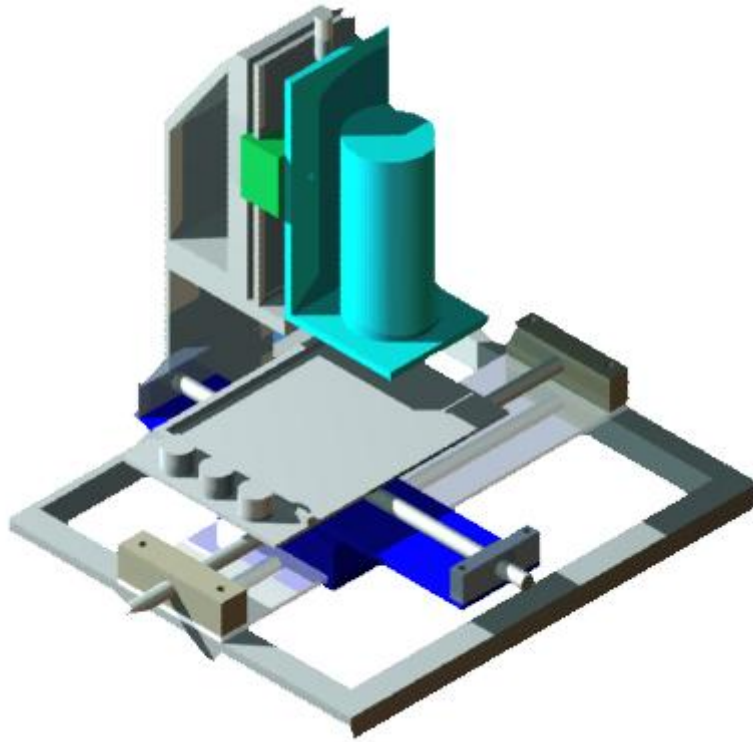
#### 4.4.2.2.2 Position Control

The inclusion of a linear encoded feedback loop could potentially reduce the final error between the commanded and actual position on the X and Y axes. From calculations discussed in the tolerances section, it can be seen that this feedback method will only eliminate potential error in the motor and ball-screw. Considering it is difficult to find extremely precise linear encoding, this method is not currently being pursued. If it turns out that we have significant error in our final positioning, we will attempt to employ highly accurate encoding (among other control systems) in order to eliminate it, although this could potentially introduce a much higher cost than is feasible for such a prototype.

#### 4.4.3 Mechanical Design

##### 4.4.3.1 System Description

After looking at a variety of different configurations we decided to with the x-y table, which moves horizontally in two dimensions (X and Y). The cutter is mounted to the base separately from the platform, and moves vertically in one dimension (Z). See Figure 4. XY Table for reference. The base of this configuration has a depth of 14 inches, a width of 16 inches, and a height of 15 inches.



**Figure 4. XY Table.**

The movement in the various dimensions is driven by balls screws. Ball screws were selected for their accuracy and smoothness. They are expensive, but are necessary for our intended accuracy. In order to minimize inaccuracies due to backlash, we will use two ball nuts loaded in opposite directions for each screw.

For our cutting instrument, we decided to use a Dremel 200 series Two Speed Rotary Tool based on its good reviews and reputation of quality. This tool has two rotational speeds, allowing us to choose a cutting rotational speed based on the cutter's translational speed.

The platform on which the circuit board rests will include a replaceable piece of cork wood. This wood acts as a safety buffer between the cutting bit and the metal of the table. When cutting through-holes, the bit will cut away pieces of wood, but not damage the steel beneath. This wood insert can be replaced as necessary with any sufficiently smooth scrap wood that has been cut to size.

#### *4.4.3.2 Mechanical Calculations*

The overarching mechanical objective for this project is to keep the unit as small and light as possible while still meeting specifications for board size, strength and accuracy.

##### **Arm strength**

The primary strength/weight concern is the arm that supports the routing tool. This arm has to be rigidly attached to the base of the unit (or nearly so), and is cantilevered over the x-y platform to support the routing tool. The routing tool operates at 1500 and 3000 rpm, which needs to be taken into account as well.

The arm also needs to support the up/down motion of the routing tool. Instant acceleration would cause jerk, bouncing the arm. The mechanical solution to this is to strengthen the arm. This can also be remedied electrically using software to control the acceleration (implementing a motor control device to eliminate overshoot, reduce settling time, and reduce steady state error, for example).

Finally, when the routing tool is turned on, it exerts a torque until achieving a constant rotational velocity. The arm has to be strong enough to counter this torque.

#### Screw Strength

The ball screws need to be strong enough to move the x-y platform. See Section 4.4.4.3

#### Rail strength

The rails need to be strong enough to support the weight of the x-y platform, the PCB, and the force of the routing tool while flexing a maximum of 1/32 in. We also need to investigate vibrations transferred from cutting bit. Rail design will be adjusted during construction to allow for the easiest fabrication techniques, however, using Algor and Fempro on our model, a rail thickness of 0.5 inches resulted in a deflection of 0.002 inches, which is acceptable to our design.

#### Base

While we want this unit to be as light weight as possible, we also want it to be stable. This means a wide support base and a fair amount of weight. It is currently planned that we will determine these numbers from rough calculations and experiments.

## Vibration Analysis

The arm must be designed so that the rotating cutting tool will not send it into resonance. We also need to account for possible speed reductions due to friction while cutting. Due to the complexity of the arm, finding its natural frequency will require using a program such as Algor. Preliminary calculations based on a length of 6 inches at a one square inch cross section yielded the results in Table 1.

$$\omega_n = \sqrt{\frac{EI}{mL^4}} \quad \text{Equation 1}$$

$\omega_n$  = Natural frequency

$E$  = Modulus of Elasticity

$I$  = Area moment of inertia

$m$  = Mass

$L$  = Length

$$I = \frac{bh^3}{12} \quad \text{Equation 2}$$

$b$  = length of base

$h$  = height

Equation 1 is taken from <http://www.eng-tips.com/viewthread.cfm?qid=203139&page=6>, while Equation 2 comes from Appendix A of Mechanics of Materials, 6<sup>th</sup> edition by William Riley et al.

### 4.4.3.2.1 Sheer Stress for Bit

The cutting speed has to be kept low enough to not snap the routing bit. This will be the limiting factor in terms of translational speed. In order to determine this number, we need to purchase a routing bit and determine the amount of allowable sheer stress by snapping the bit.

### 4.4.3.2.2 Safety

This will be a small machine, however, basic safety requirements remain in place. Anyone operating the machine will be required to wear eye protection. Also, we are including a kill switch on the top of the machine (because there will not be a good place on the front of it) that will immediately stop the cutting tool and cause the controlling software to raise the cutting tool off the board.

### 4.4.3.2.3 Router Speed

The translational speed of the cutting tool is a function of how much lateral force can be exerted on the bit without damaging it. This includes fatigue strain diminishing accuracy. One of the objectives for our team

is to prototype a board in approximately fifteen (15) minutes. One method we plan to use to help us achieve this number uses multiple bits. Rather than use one tiny bit (necessary for the fine detail work) to remove large areas of copper, we'll have the machine swap out the current bit for one with a larger cutting area. (See Section 4.4.3.3)

#### 4.4.3.2.4 Precision Issues

Calibration will be based off matching up a corner of the XY platform to a location either on the cutting bit or a different location on the cutting tool bracket.

Appendix 2 walks through the calculations that determine the precision of the position of the router relative to a given origin. Currently we are considering a motor with 200 steps per rotation and a ball screw with a precision of .2 inches per rotation. The motor control boards we are considering also at an extra 8 microsteps per step. This gives us the ability to position a motor at any one of 1600 positions per revolution. This translates to a theoretical linear precision of .125 thousandths of an inch.

Although we will theoretically be able to command the motor at this level of precision, we also must take into account the error associated with each component. The error comes from a number of places, both electrical and mechanical. The error associated with the electrical components comes from the motor controller board. If we use the Allegro Microsystems A3967 stepper controller IC that we are currently considering, we can expect accuracy to within 20% of each microstep. This comes from the two coils on each motor that generate an electric field when current runs through them. The accuracy of the current supplied by the IC is 10%, so if they are both 10% off, we can expect a 20% error. The coils themselves on the motor also have an associated error. If we use the Anaheim Automation D series standard stepper motors, this error is 5% per step, noncumulative. The error from the mechanical components will come from the ball screws, which have both a cumulative error over the length of them and an error from the space between the ball nut and the threads. These errors were given approximate values similar to those of different ball screws that we looked at. The error came out to be approximately 2.5 thousandths of an inch.

It should be noted at this point that our precision far exceeds our accuracy. In fact, even without microstepping, we would still be able to achieve a precision of 1 thousandth of an inch, which far exceeds the precision necessary for DIP parts. The reason for the inclusion of the microstepping is the improved torque response. Microstepping cuts the distance of each step, and the current through the motor coils is optimized for each microstep rather than each full step. Adequate torque is a critical consideration because if we start to skip steps because of not enough torque, we cannot rely on the controller knowing where it is.

#### 4.4.3.3 Physical Design

##### Switching bits

The platform for the circuit board extends beyond the PCB to support 3-4 locations for routing bits. At least one bit will be specialized for drilling through-holes. One bit will be for fine-detail work. 1-2 bits will be for rapid copper removal.

Each bit will be attached to a grooved gear mounted on a permanent magnet. The cutting tool has a mating grooved gear mounted on an electromagnet. In order to pick up a bit, the cutting tool moves above the bit and switches on the electromagnet in order to attract the permanent magnet. To eject a bit, the

cutting tool moves above an empty bit receptacle, lowers into position and switches the polarity on the electromagnet.

Considering, however, that this functionality in large-scale CNC machines ranges in the \$10,000 area (as told to us by Rex Schieler), this step is currently perceived to be out of range of our project goals. Instead, we currently plan to simply have manual bit switching occur after each phase is completed.

## Frame Material

Basic options for the frame material are wood, aluminum or steel. The factoring parameters of the choice are light weight, ease of machining, and strength. Wood could give us the necessary strength, but would be bulky. Also, there are concerns about flexing. Steel definitely has the required strength and stiffness, but the resulting frame would quite heavy. Aluminum makes the most sense: it can be welded, or barring that, bolted. It is lightweight and has good stiffness.

The calculations leading up to natural frequency for a 6 inch beam with a cross section of 1 inch square for each material can be seen in Table 3. Natural Frequency Calculations.

**Table 3. Natural Frequency Calculations.**

	Wood (oak)	Aluminum	Carburized Steel
density [lb/in <sup>3</sup> ]	0.025	0.1	0.283
young's modulus [psi]	1600000	10000000	30000000
I [in <sup>4</sup> ]	0.083	0.057	0.057
volume [in <sup>3</sup> ]	6	2.625	2.625
Mass	0.15	0.263	0.743
natural freq [rad/s]	26	41	42
natural freq [rpm]	9900	15000	16000

## Alternatives: lasers

Lasers sounded like a very attractive cutting tool in our initial design brainstorming. There are a number of attractive advantages to laser implementation: there is no bit to dull, the operation is potentially quick, and the method is much more novel than simple routing. The most obvious disadvantage is an extremely high cost. We were unable to find a laser for less than \$600, with no guarantee that the laser was actually capable of cutting through copper.

Another major concern with lasers is the dual component nature of PCBs. Without testing (which would inherently include purchasing one), we do not have a reliable way of knowing if a laser cutting through copper would simply cut through the fiberglass substrate as well.

### 4.4.4 Component Selection

#### 4.4.4.1 Motors

##### 4.4.4.1.1 Criteria

The motors used will need the ability to respond to a command to go to a specific position. The precision needs to be two thousandths of an inch (contributing to the five thousandths of an inch for total error).

The motors also need to be lightweight, yet powerful enough to move both the X-Y table and the control arm, especially at low speeds. The motors also need to cost less than \$30 each.

#### 4.4.4.1.2 Alternatives

There are two possible motor choices that we have looked into. The first is the servo motor. The basic idea behind a servo motor is to input a voltage corresponding to a position and feed back an error signal that corresponds to the actual position. This error signal is then compared to the input signal, and the motor is allowed to rotate and until the signals equal one another. The main problem with servo motors is the need for a feedback loop, which requires additional lines and inputs and outputs to different subsystems. There is also a physical constraint on how much the motor is allowed to rotate, as it will only be able to rotate generally from 180 to 210 degrees.

The second motor considered was the stepper motor. The idea behind the stepper motor is to divide the angular motor position into a finite number of steps, and each time the motor is stepped, we know where it was supposed to go without any type of feedback system. We are constrained in accuracy to the number of steps, which is a physical parameter that we cannot change. However, we are able, with control logic, to divide the physical number of steps into a discrete number of microsteps (these will be discussed later).

#### 4.4.4.1.3 Decision

The motors we will be using are stepper motors. The reasons for this decision are the available low-speed torque, size, price, and ability to be programmed to an arbitrary amount of precision.

### 4.4.4.2 *Drill and Routing Bits*

#### 4.4.4.2.1 Criteria

##### Size

The size of the bit is of utmost importance in both our design calculations and design performance. The smaller the bit size, the tighter and more complex we can make the traces on the PCB. But with a smaller bit comes many disadvantages. The first is that the relative bit strength goes down. If the bit size is halved, its strength is cut to a fourth, but the shear force is only halved. This offers a large problem, as a finer bit will take longer to complete the milling process. As a temporary solution to this tradeoff, we are thinking of using multiple bit sizes for varying diameter work.

Also, smaller bits are more prone to gum up with material. The bit's ability to fling off material goes down with size, which will most likely be compensated by increasing the rotation speed. A third problem with reducing bit size is that heat rejection can become a great problem. Most of the heat is rejected through the material cut from the part, which is reduced in many ways with a smaller bit. The bit both moves slower and takes a smaller cross section of material per unit length. Drilling bits will not have these same problems as the stress put on them is very low. Their sizes will simply be what we need in the specification of the PCB being made.

##### Strength/Material

The strength of the bit is tied directly to the size of it, but there are a few factors independent of strength. The material of the bit greatly determines the characteristics of its performance, as well as that of the rest of the machine. Having a stronger bit material allows for great improvements in bit speed. The downside is that stronger material is also more expensive. Fortunately, bits are much less expensive than many of

the other design components. Some of the most expensive bits will cost \$20 each, even for very high quality. The relative performance gained per dollar used on bits is large compared to other components.

Also, since we are using very small diameter bits, they require tapering from the base that goes into the chuck to the tip of the bit. One of the problems with tapering is that it requires the bit tip to be further from the drill, which can both amplify vibration and increase the moment on the bit. The upside of a tapered bit is that the standardized chuck size of the bits helps make the bit switching process easier.

## Type

There are a few routing bit types to choose from. One of the variation areas is the cut of the bit: whether a straight cut or helical cut groove. The straight cut may reduce the frequency of burrs and whiskers left over by the bit, since the bit cuts completely perpendicular to the surface of the copper. The downside of a straight cut is that the bit tends to gum up with the routed out copper. Unlike helical edges, straight edges are not made to remove the waste copper naturally, and tend to pack the copper into the bit. The downside to helical edges is that when they wear or get used frequently they generate a lot of burrs on the surface of the copper. A new and sharp bit generates significantly fewer burrs.

### 4.4.4.2.2 Decision

Overall we gauge the final bit sizes will be from 1/16 to 1/32 in diameter, though we haven't tested them ourselves yet. We had Rex Schieler test a 1/16 inch helical bit on his large scale CNC machine to find that the width of the cut was not as large as we thought it would be. Though it seemed a bit coarse, it might work for the larger-scale cutting. Based on what we saw, 1/32 in diameter would be overly fine for our application, though it might be needed in some very tight places. The helical blade left a lot of burrs behind, but Mr. Schieler said he hadn't used the bit in a few years and that they were dull, meaning we would need to gather more evidence. We still need to test and research the decision between helical and straight blades. The main difference between our use of routing bits and other uses is that the depth of material we are removing is incredibly small, close to .001 inches. With this in mind the probability of the bit gumming up seems small, though we can't know for sure yet. The bit material is going to be carbide tipped, which is a widespread and strong material type.

### 4.4.4.3 Ball Screws

#### 4.4.4.3.1 Criteria

##### Precision

Precision is incredibly important in our application of CNC technology. The minimum width of the bit we will be using is 1/32 of an inch. The traces we will be outlining will be a minimum of .05 inches wide. Both of these constraints combined create a very tight tolerance for our x-y positioning. The design hurdle to monitor is that almost all of the variances from the components will compound with each other, so all variances are minimized as much as possible. Some of these include variation from bit vibration/movement and variance from moving/sliding part connections. A variance of 5 thousands of an inch is already a 10% variation in the width of the trace, but this is also assigned to both sides of the trace, making the variance even more pronounced. The minimum precision we are looking for in our screw system is a variance of .005 inches. Most basic ball screw systems have this tolerance, and ballscrews exist with tolerances of .001 inches. Since this is a rather easy way to reduce variations in the traces we will most likely go with the more precise systems.

To further reduce random error, preload will be introduced on the ballscrews. Without preload, ballscrews can have anywhere between .003 in and .015 in of backlash. Preload essentially gets rid of all significant backlash in the system. Also, adjustable preload is favored since we want to be able to tailor it specifically to our system, and it is difficult to foresee it without testing it directly.

#### Strength

Strength is not a very problematic aspect of the screw selection process. One of the main goals of our project is to keep the weight of the entire system down as much as possible, simply because almost all the mechanical design factors benefit from a lighter construction. As a result, the screws are far from running into feasibility restrictions when it comes to cost or size. The approximate minimum diameter of the screws is going to be .5 inches. A .5 inch diameter screw comes very close to maxing out its strength in the most conservative estimate of the system, so in actuality the screws will most likely be slightly larger, as the additional cost and design requirements are rather insignificant.

#### Friction/Drag

Friction and drag is a rather important aspect in the screw selection process. More friction on a system means more wear, more maintenance, effort needed for lubrication, possible heat management, and bigger motor sizes. Since simplicity is of utmost importance on senior design projects, especially ones as complex as ours, a reduction in friction is incredibly important. Reduction in friction lowers the cost of maintenance and initial capital invested in the components. Also, since friction increases with a larger screw (the drag has more torque, and a larger surface area to act on) having less friction will allow us to be more conservative in our screw size selection when it comes to strength. Another negative aspect of friction specific to our application is the static friction from non-roller screws will reduce the accuracy of our routing. Having to constantly overcome static friction upon start and stop could cause small slingshot effects in the movement of the drill on the board. Ball screws reduce or even completely remove this possibility for error. Ball screws have significantly less friction, which greatly improves their usefulness in our system.

#### 4.4.4.3.2 Alternatives

The other most common type of screw for this kind of application would be a machine screw. Machine screws have the precision required for our application, but have many other downsides. They sacrifice increased friction for increased accuracy. This would be a good alternative if we were not already well within our tolerance range with ball screws. Machine screws were ruled out because of this.

#### 4.4.4.3.3 Decision

Our decision was heavily influenced by Chuck Holwerda's CNC machine. His machine was used for drilling holes for components in PCBs, and used a Y-Z router / X table system. The design incorporated the use of ball screws on all three axes. The lack of maintenance and reliability of the design was very attractive to us, as the machine has been around for 20 years. The only blemish in using his design choices as an example to follow is that precision is slightly more important to our design, so logically it is not wise to follow it blindly. This spurred us on to research more about the tolerances and other specs of available ball screws.

The PCB drilling machine gave us quite a huge head start on figuring out the feasibility of a ball screw system. One of the biggest drawbacks of a ball screw system is the possibility it has for a large amount of error. Holwerda's design solved this problem, as he had two ballnuts rigged in opposition to each other to create tension between them. This design gets rid of the fundamental flaw of precision in using a ballnut

system. Many ball screws are sold in this fashion, with an adjustable amount of preload between the two nuts.

## **4.5 System Testing**

### Drill Force

One of the most basic tests we will have to do is to find out the sheer force the bit exerts on the copper as it cuts. This will mostly be a test of force vs. routing speed, though other factors like drill RPM, cut depth, and bit size will also be tested. The basic idea is to rig a test support out of wood to hold the drill. The board will be pulled at a constant speed as a force gauge is measured. If possible, the data will be computerized via DAQ for better estimation and documentation.

### Support arm strength

This is a rather far off test, as it requires the support arm to be built completely. The arm needs to be tested for defection in two ways. The first manner in which the arm is put under stress is via moment forces from the drill/drill carrier weight and the drag forces from the routing bit. The magnitude of the drag forces may turn out to be negligible, but to be conservative it is included in preliminary testing plans. The arm can be simply tested for a correlation of force and deflection on the z, x, and y planes via a force meter and a precise measuring device such as a laser, as the deflection expected is on a very small scale. Secondly, the resonant frequency and frequency response of the cantilever arm will be tested by some sort of vibration motor that can vary its frequency through the ranges we desire. The resultant magnitude of vibration can also be measured by a laser.

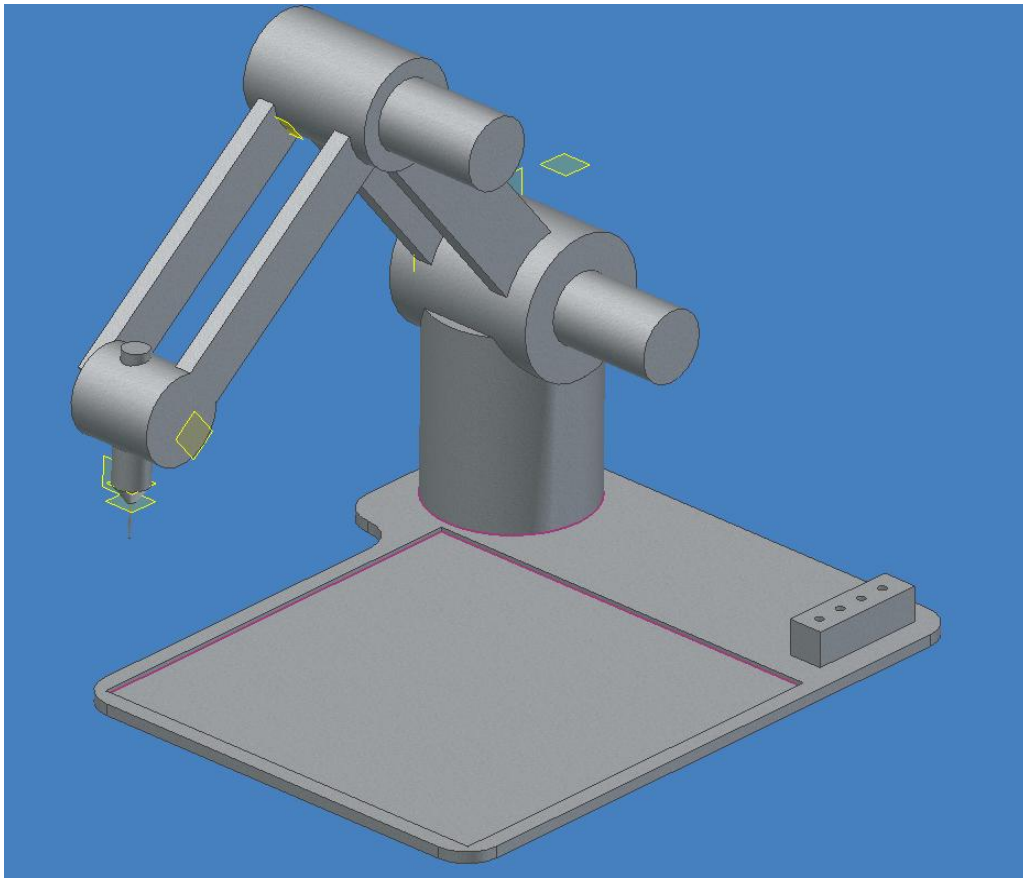
### X-Y platform

The main concern for the x-y platform is small amounts of deflection from initiating the routing cuts, and static forces in the z direction as it cuts. Normally, the forces being applied by the bit on the board would be of very little concern, but the tolerances in the z direction are very, very small. The width of the copper being cut is on the order of magnitude of .001 inches. A simple test to find the deflection of the platform under varying weights can be done. This would also be measured by a laser. The platform would be tested in many different points across its face to find the most severe deflection spots, which will most likely be in the middle of the board.

## 4.6 Alternative Designs

### 4.6.1 Radial

A radial system incorporates the use of an arm fashioned at one end so that it can rotate on the x-y plane. There were two designs we had in mind for this. The first was made so that the radial length of the arm could be adjusted by a motor and that the z axis could be controlled by a motor mounted vertically at the end of the arm. The second was made so that there were two horizontally mounted motors at the shoulder and elbow of a robotic-esque arm to simultaneously control the radial length of the arm and the Z positioning of the drill. Both of these designs have major drawbacks, all of which are related to the cantilever nature of the arm used. Both of the arms would have suffered from displacement issues when extended completely. Since the precision of our routing trace is very important, it would have been difficult to make the arm strong enough not to go outside the desired tolerances, especially when being accelerated. The motors could have been taken off the end of the arm and attached to the bit via a rotating chord, but the weight of the arm is enough of a problem itself. The design can be seen in Figure 5. Radial Design.



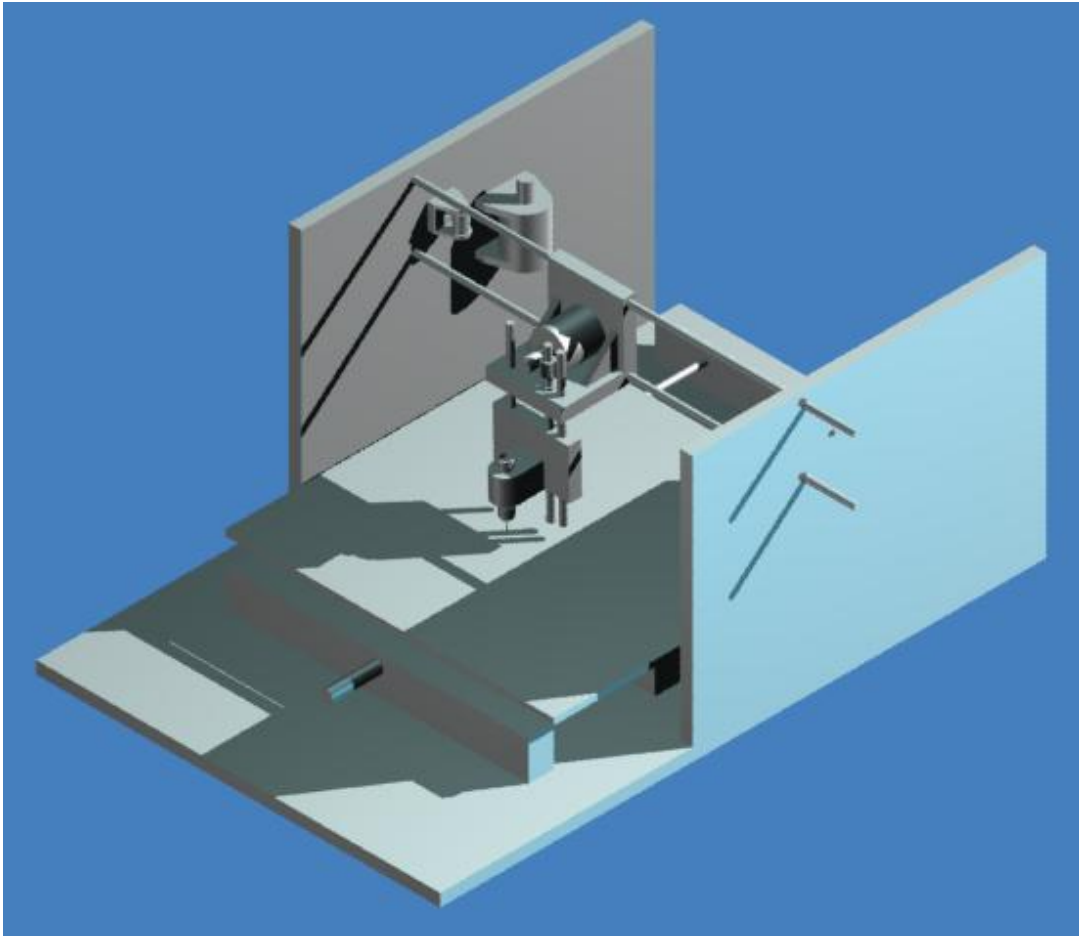
**Figure 5. Radial Design.**

Along with the tolerance issue due to the arm length, there was a tolerance issue pertaining to the accuracy of the motors. The arms would have been required to operate under a very small range of motion, and thus a very small amount of rotation. The motors would have required a large gearing system to allow for the motor movements to be precise enough. The gearing system would have introduced further variance, as error sources such as backlash would have to be accounted for. Overall the system is

just too complex and requires too precise of equipment. Our design is already flooded with complexity, so this was ruled out.

#### 4.6.2 YZ + X

Many designs were considered such as x-y-z router and y-z router / x table along with our final choice of x-y table / z router. These designs did not necessarily have obvious drawbacks like radial or laser did. The tradeoffs given with each design are largely cosmetic and convenience oriented. The traditional x-y-z router design can be rather boxy, though its table space is somewhat less than the others. We came up with a more appealing design for the x-y-z router (seen in CAD model) but this also sacrificed table space. Figure 6. X Table, YZ Router. shows a preliminary design in this configuration.



**Figure 6. X Table, YZ Router.**

#### 4.6.3 Laser

Our industrial consultant brought up the idea of a laser system to vaporize away the copper. This system is very precise, can cut very small, and eliminates the problem of metal shavings produced by routing with a bit. On the downside, this system is very expensive, quite large, and has the possibility of creating a hazardous breathing environment from all the vaporized metal. We did some research and found that the laser required for this would cost on the order of thousands of dollars, something way out of the range of our budget. The size and weight were also ridiculously out of the scope of our project, as a laser powerful

enough to vaporize copper would weight about 20 pounds and be about two feet in length. The mechanical systems needed to guide the laser would have been enormous. The initial idea was to simply rotate the laser above the PCB on two axes. Though this would only require supports, it has many other obvious complications. With our research it is obvious that this design is not feasible for the scope of our project.

## **5 Schedule of Development**

In order to address the foreseeable problems and design around them, the spring semester will be a continuous cycle of prototyping, testing, and revising. This will allow us to test our design ideas and hopefully improve upon them. However, to make efficient use of the cyclical design strategy, there needs to be some sequential, rigid goals set to guide our design efforts. We have come up with some major testing milestones and goals that address specific foreseeable problems in our design.

### *Milestone 1: X-Y table movement*

Date: February 15, 2008

Inputs: A functional X-Y table prototype complete with motors attached will be constructed. This table should be able to move freely from any point to any other point in a 6" x 8" workspace. The table will be moved with stepper motors controlled by the second input, a controller. The controller will be implemented on the Altera DE2 evaluation board. The controller will be able to receive a list of vertices and move the X-Y table from one vertex to the next. Also, the DE2 Evaluation Board will be programmed to allow keys KEY(0) through KEY(3) to allow for manual adjustment of X-Y table position, with each key moving the table up, down, left, or right. In addition, poison commands will be able to be entered via a JTAG UART terminal in the Altera Debug Client, and commanded and actual position will be able to be read from the terminal.

Test Method: The controller will be given a list of points in the addressable 6" x 8" workspace. The controller will then be allowed to take in a point and move the X-Y table. Software breakpoints will be inserted into the controller code such that upon reaching a point, measurements will be able to be made to assess accuracy and precision. We will use the Altera Debug Client to manage the breakpoints and debugging procedure.

Outputs: The results of the precision and accuracy measurements will allow us to see if we have designed a usable control algorithm and whether the mechanical parts chosen are precise enough. We will also be able to determine whether the electrical control parts will provide enough functionality for both operation and debugging.

### *Milestone 2: Z-axis movement and initial routing*

Date: March 7, 2008

Inputs: Routing arm and motor added to X-Y table for total X-Y-Z movement in a 6" x 8" x 2" workspace. Functionality added to controller to interpret 3-dimensional coordinates. DE2 board programmed to add 3-axis manual adjustment for zero calibration.

Test Method: The controller will be given a series of programs each testing different issues. Basic shapes such as a line, square, and circle will be routed and the routed boards will be measured for another gauge of precision and accuracy. At this point, if precision and accuracy are deemed acceptable, a basic circuit

board featuring a DIP part and supporting components will be routed. It should be noted that these programs will be written by hand, as PCB translation software will not have been programmed at this time. This board will then be drilled and populated by hand and tested to measure robustness of the actual circuit. Various combinations of bits and speed will also be tested.

Outputs: Another measure of precision and accuracy will allow for further refinements of the control algorithm and motor driving issues such as current, speed, and microstepping size. The combinations of speed and bit size will also allow us to see how fast we can run the machine at different sizes of bits. The physical full-sized prototype will allow us to examine issues such as clearance, weight, aesthetics.

### *Milestone 3: Initial drill testing and full program run-through*

Date: March 28, 2007

Inputs: Fully functional mechanical prototype and control software. The controller will be able to receive via USB a program in the final standardized format that includes the tool paths for both routing and drilling.

Test Method: First, the machine will be programmed to drill an array of holes of different sizes. Once finished, the board will be removed and measured for accuracy and precision. If deemed acceptable, The machine will be programmed to both route and drill another test circuit board, again featuring a DIP part and supporting components.

Outputs: We will be able to experience functionally what a user would experience. This will give us an idea of what things don't seem intuitive, and we will determine what kinds of these we can change from an interactive standpoint.

### *Milestone 4: Final run-through*

Date: April 7, 2008

Inputs: Fully functional final prototype, with control hardware implemented on custom printed circuit board, fully independent of any evaluation board or debug client. PCB translation software with user-friendly interface.

Test Method: Using a circuit board designed in commercial layout software, sequentially go through entire process including PCB layout to tool-path translation, downloading program via USB to controller, actual routing and drilling, circuiting board populating, and finally circuit board testing.

Outputs: A complete user experience and precision and accuracy data for an entire process. This will allow us to make final tweaks in operating parameters.

## **6 Design Hurdles**

One of the main issues expected will be electromagnetic interference (EMI). Because our three axis control stepper motors will be running at relatively low speeds, we don't expect EMI to be a problem with them. However, the Dremel tool will be running much faster and therefore throwing off much more electrical noise. Because it will be running in the near proximity of digital control logic, there is a likelihood that our controller will be affected by it. The digital control logic can be sensitive to EMI, and it remains to be seen specifically how much this issue will be a concern.

Another main issue will be the vibration of the control arm. It is difficult to foresee the effects of adverse vibrations, but we wish to design the arm to minimize them. This will be done by designing an arm with a resonant frequency that does not correspond to the frequency of the spinning dremel.

The ability to keep the circuit board exactly level will also be a design hurdle. Initial testing at RouteOne showed that if the board bends or bows at all, the Y-axis position data becomes unreliable. We plan to minimize uneven boards by keeping them down in size (6" x 8") and by fixing double-sided tape to the board.

Maintaining stepper motor accuracy will also present challenges. By the nature of open-loop systems, we will not have much feedback as to whether the motors are exactly where we program them to go. Designing reliable control algorithms and hardware will be our method of tackling this hurdle.

## **7 Future Work and Alternatives**

Improvements over prototype

Possible Improvements on the mechanical prototype consist of using a better grade of aluminum for the frame, making it lighter and stiffer. It might also be possible to switch out ball screws for linear motors, which would reduce part wear and the need for stepper motors. It might be possible to come up with some way to use a laser to remove the copper, rather than a mechanical system.

## **8 Conclusion**

From the research done this semester, we have concluded that this project is feasible, both mechanically and electrically. Using the stated development schedule, we will be able to produce a prototype by early April in order to make final revisions and to finalize documentation.

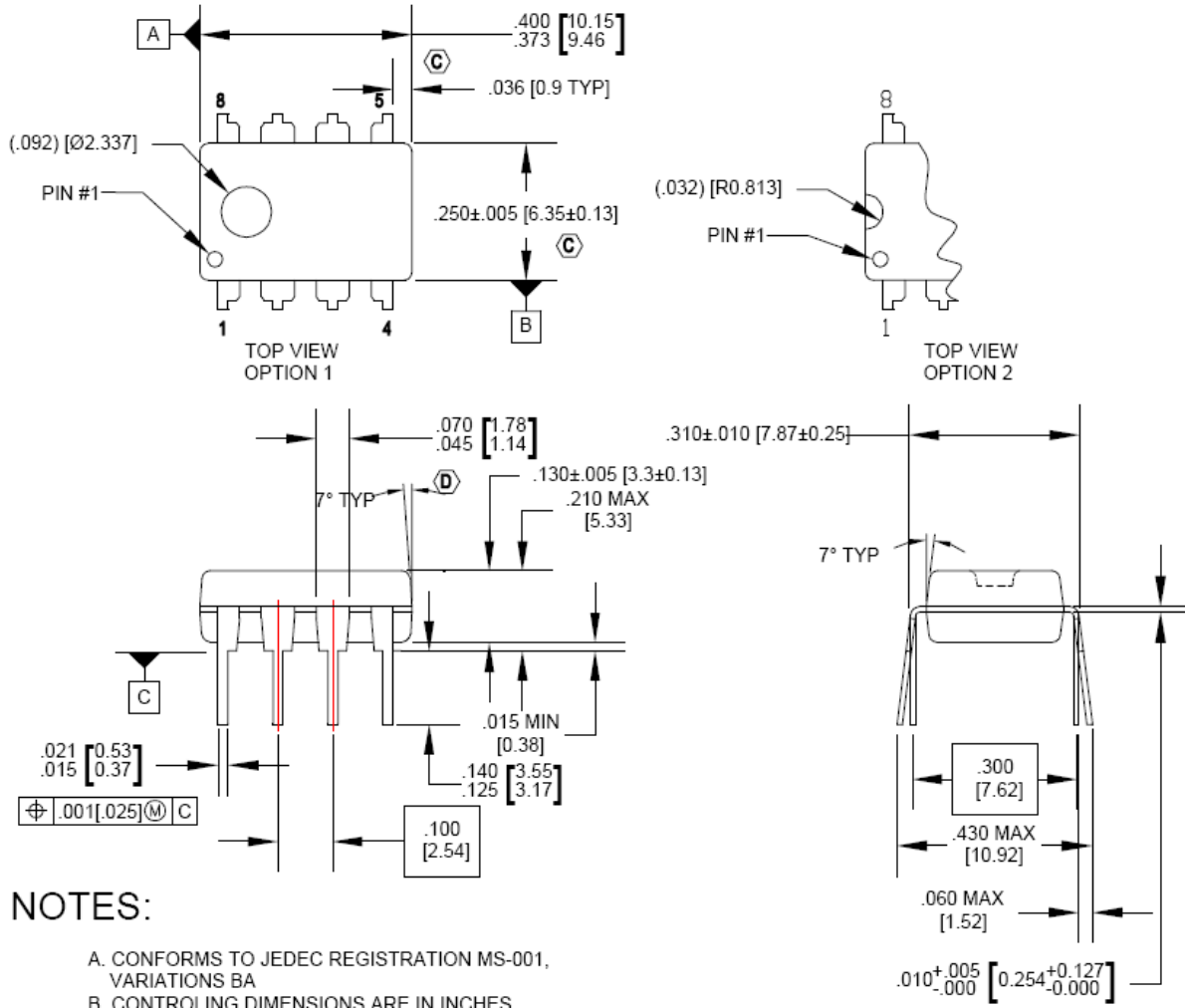
From an electrical standpoint, we have a clearly defined system with each component appearing to be feasible, both in software and hardware. The process of taking a PCB file, converting it to position data, and controlling a three-axis CNC machine has been laid out and adequately specified.

The fabrication of a CNC milling machine appears to be feasible. Using a system of stepper motors and ball screws, we can accurately position the routing bit at any point on a 6"x8" board (within a projected 5/1000<sup>th</sup> of an inch). The doubts about feasibility arise from whether a routing bit can be small enough to cut fine traces for PCBs without snapping regularly, if the overall error will exceed the inherent accuracy requirements, and if the machine will be able to work quickly and reliably enough to be competent in today's market. However, even if our accuracy is not great enough to produce a circuit board, we still have a viable project in the creation of a CNC milling machine capable of milling out name plates (for example) in wood or soft metals.

## Appendices

Appendix 1. DIP Part Specifications.....	A-1
Appendix 2. MathCAD Calculations.....	A-2

## Appendix 1. DIP Part Specifications



Taken from <http://www.fairchildsemi.com>

The smallest component package that our machine will support is the dual inline package, or DIP. The DIP is a standardized package that is big enough for us to fit a bit between any two leads. This will allow us to mill traces small enough to support it. This package was selected because many components and ICs are available in it. Although the length and number of pins can vary, the space between each pin remains standardized. Although most final products are using surface mount components, we will not be able to support them because the bit would need to be too small. However, for purely electrical prototyping, many of these parts can be ordered in the DIP package and boards can be tested for electrical functionality.

## Appendix 2. MathCAD Calculations

### Precision Calculations

Step Angle:

$$S_{\text{size}} := 1.8\text{deg}$$

Steps per rotation:

$$S_{\text{rot}} := \frac{360\text{deg}}{S_{\text{size}}}$$

$$S_{\text{rot}} = 200$$

Microsteps per step:

$$M := 8$$

Travel per rotation:

$$T := 0.2\text{in}$$

Precision:

$$P := \frac{T}{S_{\text{rot}} \cdot M}$$

$$P = 1.25 \times 10^{-4} \text{ in}$$

note: information taken from

<http://www.anaheimautomation.com/standard.aspx>  
General Specifications

### Accuracy Calculations

Step Accuracy (of one step):

$$S_{\text{Acc}} := 5\%$$

Step Error:

$$E_{\text{Sdeg}} := S_{\text{size}} \cdot S_{\text{Acc}}$$

$$E_{\text{Sdeg}} = 0.09 \text{ deg}$$

$$E_{\text{Sin}} := E_{\text{Sdeg}} \cdot 2 \frac{\text{in}}{360\text{deg}}$$

$$E_{\text{Sin}} = 5 \times 10^{-5} \text{ in}$$

Microstep Gain Error:

$$E_{\text{G1}} := 10\% \quad \text{from phase current 1}$$

$$E_{\text{G2}} := 10\% \quad \text{from phase current 2}$$

$$E_{\text{G}} := E_{\text{G1}} + E_{\text{G2}} \quad \text{note: taken from A3967SLB data sheet at 8}$$

$$E_{\text{G}} = 20\% \quad \text{microsteps per step}$$

$$E_{\text{MSdeg}} := \frac{S_{\text{size}}}{M} \cdot E_{\text{G}}$$

$$E_{\text{MSdeg}} = 0.045 \text{ deg}$$

$$E_{\text{MSin}} := E_{\text{MSdeg}} \frac{.2\text{in}}{360\text{deg}}$$

$$E_{\text{MSin}} = 2.5 \times 10^{-5} \text{ in}$$

Maximum Travel:

$$T_{\text{max}} := 1.5\text{ft}$$

Ball Screw Error:

$$E_{\text{sys}} := 1 \cdot 10^{-3} \frac{\text{in}}{\text{ft}}$$

$$E_{\text{sysin}} := E_{\text{sys}} \cdot T_{\text{max}}$$

$$E_{\text{sysin}} = 1.5 \times 10^{-3} \text{ in}$$

$$E_{\text{rand}} := 1 \cdot 10^{-3} \text{ in}$$

$$E_{\text{BSin}} := E_{\text{sysin}} + E_{\text{rand}}$$

$$E_{\text{BSin}} = 2.5 \times 10^{-3} \text{ in}$$

Total Error:

$$E_{\text{total}} := E_{\text{Sin}} + E_{\text{MSin}} + E_{\text{BSin}}$$

$$E_{\text{total}} = 2.575 \times 10^{-3} \text{ in}$$