

Portable PCB Schemer

¹Sushant Ipte, ²Ravindra Gupta, ³Riddhi Agarwal, ⁴Murtuza Barodawala, ⁵Prof. Shiburaj Pappu

¹Student, ²Student, ³Student, ⁴Student, ⁵Head of Department

¹Computer Department,

¹Rizvi College of Engineering, Mumbai, India

Abstract: In today's world, most of the colleges and industries uses the traditional method of making a PCB. The traditional method consumes a lot amount of time and also much effort is wasted. So the need of a new process to make the PCB prototyping efficient was felt. A printed circuit board (PCB) mechanically electrically and supports as well as connects to electronic and electrical components with the help of conductive tracks, pads and other features defined from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. Soldering of the components are generally done onto the PCB to electrically connect and mechanically fasten them. Initially a photo mask was created manually to design the PCBs on a clear Mylar sheet, and the size of this design was usually at two or four times the true size. The schematic diagram of the component pin pads were put up on the Mylar sheet and then the traces were routed onto the connecting pads. To increase the efficiency of the system they used Rub-on dry transfers of common component footprints. Traces were made with self-adhesive tape. Assisting in the layout of the Mylar sheets was the pre-printed non-reproducing grids which were attached with the Mylar sheets. The finished photo mask was photo lithographically reproduced onto a photo resist coating on the blank copper-clad boards.

To form a functional printed circuit assembly (PCA), the bare board is loaded with electronic components in the assembly, which is also known as "printed circuit board assembly" (PCBA). In through-hole technology, the component of the system are inserted in holes which are surrounded by conductive pads and the holes keep the components in place. While using surface-mount technology (SMT), the components of the PCB are implemented which will help the pins to cast with lands on the surfaces of the PCB or the conductive pads of the PCB. The solder paste, which was previously applied to the pads, holds the components in place temporarily and if surface-mount components are applied to both sides of the board, the bottom-side components are fixed to the board. The components are then soldered on both completed and surface mount and then once it's cooled and solidified, the components are hold in place by the solder for enduring and electrically connecting them to the board.

In our project we will be making a PCB Schemer which will overcome many issues faced by the traditional method and also provide an efficient way of making PCB not only for the student and industries but also to the hobbyist who love to design prototype of a PCB at home. The PCB Schemer will cut down both time as well as effort as to will just prototype the PCB by the use of the computer aided design of the PCB.

Index Terms - PCB prototyping, PCB drilling, PCB plotting methods.

I. INTRODUCTION

Now-a-days, printed circuit boards (PCBs) are used as the immortal in most of the electronics, both as a physical support piece and as the wiring area for the surface-mounted and socketed components. The materials that are used to make the PCBs are fiberglass, composite epoxy, or another composite material.

According to current scenario, PCB designing process include drawing the pattern on the copper plate and then plotting it. Drawing the pattern on the copper plate usually consumes a lot of time. So we have designed the PCB Schemes/plan i.e. it will implement the planned/designed PCB. The PCB schemer will consist of 2 parts conversion and plotting .IN conversion the .Gerber file, which is the general output of the PCB designing software's will be converted into G-code. This will then be given to the schemer for prototyping via cable or Bluetooth. The schemer will then process the G-code and then synchronize with the X and Y axis for prototyping the design on the copper plate.

The Simple electronics which uses PCBs are composed of only one layer. Computer graphics cards, motherboards and other sophisticated hardware's can have multiple layers, sometimes up to twelve.

The PCBs are not only associated with computers but also they can be founded in many other electronic devices such as TVs, Radios, Digital cameras and Cell phones. Different types of PCBs which are used in consumer electronics, computers and it is also used in a variety of other fields, including:

- Medical devices; because of low power consumption and denser electronic components than the previous generations, it is possible to make new and exciting medical technology. Due to the use of high-density PCBs, even the smallest and densest design is possible to be created. Due to these unique constraints which are involved in the developing devices for the medical field which is a necessity of small size and light weight. Everything that is found out from small devices have found their way through PCBs, such as pacemakers and very large devices like equipment's of X-ray or CAT scan machines.

- Industrial machinery; in high-powered industrial machinery the most commonly used device are PCBs. There is the usage of thick copper PCBs where current one-ounce copper PCBs do not fit the requirements. The examples of the usage of thicker copper PCBs would be helpful in the following fields such as motor controllers, high-current battery chargers and industrial load testers.

- Lighting; aluminum-backed PCBs are used because of their consumption of low power and high levels of efficiency has caused in the popularity in LED-based lighting solutions. In addition to this these PCBs serve as heat sinks and allow for higher levels of heat transfer than a standard PCB. The high-lumen LED applications and basic lighting solutions use aluminum-backed PCBs.

- Automotive and aerospace industries; to withstand the high-vibration environments which are common in both the fields, flexible PCBs are used in the automotive and aerospace industries. Their components can be very light weighted depending on specifications and design, which is a necessity while the manufacturing of the parts of these industries. The tight spaces which might be present in these applications are tight filled inside instrument panels or behind the instrument gauge on a dashboard.

There are many types of PCB boards with each own particular manufacturing specifications, usages and material types: Single-layer PCBs, Double-layer PCBs, Multi-layer PCBs, Rigid PCBs, Flexible PCBs, Rigid-Flex PCBs, High-frequency PCBs, and Aluminum-backed PCBs.

- Single-layer PCBs

A single-sided or single-layered PCB is one that is made out of a single layer of base material, which has its one side coated with a thin layer of metal. Copper is the most common coating as it is a good electrical conductor. First we apply a protective solder mask which is then followed by last silk-screen to mark out all the elements on the board on the copper base plate which is used. The single-layer/single-sided PCBs are easy to design because of various circuits and components which are soldered onto one side. Due to such a popularity, they can be purchased at a low-cost, especially for bulk-volume orders. Because of low-cost, bulk volume model they are commonly used for a variety of applications which include calculators, radios, cameras, and stereo equipment, printers, solid state drives and power supplies.

- Double-layer PCBs

A conductive metal of thin layer is the base material for Double-layer or double-sided PCBs like copper which is fixed to both sides of the board. Drilled holes through the board allow the connectivity of the both sides of the circuit. Industrial controls, Power supplies, Instrumentation, LED lighting, automotive dashboards, HVAC systems, Vending machines and Amplifiers these intermediate level of circuit complexity use the typical double-sided PCBs. are one of two ways: either Utilization of a through-hole or the use of a surface-mount is usually connected through the circuits and components of a double-layered PCB board. A through-hole connection means that small wires, known as leads, are fed through the holes, with each end of the leads then soldered to the right component.

- Multi-layer PCBs

Multi-layer PCBs consist of a series of three or more double-layered PCBs. These boards are then secured together with a specialized glue and sandwiched between pieces of insulation to ensure that excess heat doesn't melt any of the components. Multi-layer PCBs come in a variety of sizes, going as small as four layers or as large as ten or twelve. The largest multi-layer PCB ever built was 50 layers thick.

- Rigid PCBs

Rigid PCBs are printed circuit boards that are made out of a solid substrate material that prevents the board from twisting. Possibly the most common example of a rigid PCB is a computer motherboard. The motherboard is a multilayer PCB designed to allocate electricity from the power supply while simultaneously allowing communication between all of the many parts of the computer, such as CPU, GPU and RAM.

- Flexible PCBs

Unlike rigid PCBs, which use unmovable materials such as fiberglass, flexible PCBs are made of materials that can flex and move, such as plastic. Like rigid PCBs, flexible PCBs come in single, double or multilayer formats. As they need to be printed on a flexible material, they tend to cost more for fabrication. Flex-rigid PCBs combine the best of both worlds when it comes to the two most important overarching types of PCB boards. Flex-rigid boards consist of multiple layers of flexible PCBs attached to a number of rigid PCB layers. Flex-rigid PCBs have many advantages over just using rigid or flexible PCBs for certain applications. For one, rigid-flex boards have a lower parts count than traditional rigid or flexible boards because the wiring options for both can be combined into a single board.

- High-frequency PCBs

High-frequency PCBs refer to a general PCB design element, rather than a type of PCB construction like the previous models. High-frequency PCBs are circuit boards that are designed to transmit signals over one gigahertz. High-frequency PCB materials often include FR4-grade glass-reinforced epoxy laminate, polyphenylene oxide (PPO) resin and Teflon. Teflon is one of the most expensive options available because of its small and stable dielectric constant, small amounts of dielectric loss and overall low water absorption. Many aspects need to be considered when choosing high-frequency PCB board and its corresponding type of PCB connector, including dielectric constant (DK), dissipation, loss and dielectric thickness.

- Aluminum-Backed PCBs

Aluminum-backed PCBs are designed in much the same way as their copper-backed counterparts. However, instead of the usual fiberglass used in most PCB board types, aluminum-backed PCBs make use of aluminum or copper substrate board. The aluminum backing is lined with thermally insulating material that is designed to have a low thermal resistance, meaning less heat is transferred from the insulating material to the backing. Once the insulation is applied, a circuit layer of copper, ranging in thickness from one ounce to ten, is applied.

II. RESEARCH METHODOLOGY

2.1 Laser etching method

Laser cutting is the process of cutting a material using a laser beam. This can be done to either cutting down a material or to carve it into more difficult shapes that more conventional drills will have problem with. This process may also have many similarities to the laser drilling and laser engraving processes. The former involves the creation of through-holes in a material or denting them, like an engraving is done in the later process. These dents and holes are essentially and accurately cut, and you'll often see a laser cutting machine being used for laser drilling and laser engraving only. Various kind of materials vast range of thick sizes can be cut using laser cutting, making it a handy, easy and adaptable process. The laser cutting process works by having a focused and precise laser beam run through the material that someone look forward to cut, delivering an accurate and smooth finish. Initially, the laser is used to pierce the material with a hole at the edge, and then the beam is continued to cut from there through out as per the shape you want

The laser being used essentially melts the material away that it is run over, so is more like melting than cutting. This means that it can easily cut light materials such as cloth up to tougher metals and gemstones such as diamonds.

You can use either a pulsed laser beam or a continuous wave laser beam, with the earlier is being delivered in short slits while the other works continuously. You can control the beam intensity, length and heat output depending on the material you are working with, and can also use a mirror or special lens can also be used for further focus on the laser beam.

Laser cutting is a highly accurate process, which is because of the accurate control we have over it and slits can be done with a width as small as 0.1mm can be achieved when the laser cutting process is used.

There are 3 main type cutting 1.co2 cutting technique where in the gas in is a carbon dioxide mixture which has been electrically stimulated. CO2 laser cutting is most often used on non-metal materials as they have a wavelength of 10.6 micrometers.2.crystal laser cutting in which neodymium-doped yttrium aluminum garnet) and nd:YVO (neodymium-doped yttrium ortho-vanadate) crystal are use,3.Fiber laser cutting which includes the type of lasers that we have here at SPI Lasers. A beam is created using a 'seed laser', which is then amplified using glass fibers and pump diodes. They have the same wavelength as above, 1.064 micrometers. This means that the laser beam for a fiber laser is roughly 100 times greater than that of a CO2 laser when it comes to intensity.

Advantages of Laser Cutting Process;

Laser cutting is fast as in comparison with other traditional techniques of cutting metal as most of the work is automatic. Holding the piece of metal is much more adaptor in laser cutting than in mechanical cutting. The slits obtained by laser are more precise that can be done by any other method of cutting metals. It's for the very reason laser cutting is preferred in industries that require absolute exact size of metals as an outcome. In the process, the metal doesn't come in direct contact with other metals to facilitate the cut. Hence the metal is more finely cut without any contaminations or irregularities. In traditional processes where heat becomes the main reason to cut metals, the edge of the metal usually deforms to some extent but laser cutting gives you a fine edge metal cutting. The versatility of the process lets you etched delicate designs on different metals. Laser cutting process requires less energy less effort for cutting the metals than other traditional methods for cutting.

Disadvantages of Laser Cutting Process

Laser cutting demands high power consumption as compared to other techniques. The laser beam is very delicate to handle and can be dangerous. A slight mistake while adjusting the distance and temperature can lead to burning or tarnishing of the metal. The laser beam is harmful if it comes in contact with human workers and can also be threat to worker's life. It can cause severe burns if it happens accidentally. Laser cutting is still not capable of cutting thick metals more accurately. The production is not consistent in laser cutting as the time for cutting various metals may vary. It does not compliment all types of metals.

2.2 Plotter etching method

A PCB consists of a flat sheet of insulating material and a layer of copper foil, laminated to the substrate the heat. Chemical etching divides the copper into separate conducting lines called tracks or circuit traces, pads for connections,are to pass connections between several layers of copper, and features such as solid conductive areas for EM shielding or some other purposes of interset. The tracks function as wires and are intacted in place, and are insulated from each other by surrounding air or made air tight and the board substrate material. The surface of a PCB have a coating that protects the copper from corrosion and reduces the chances of solder shorts between traces or unwanted electrical contact with straight bare wires. This functions in helping to not to alter and prevent solder shorts, the coating is called solder resist. A PCB can have multiple copper layers. A two-layer board has copper on both sides; multi-layer boards sandwich additional copper layers between layers of insulating material or just single layer depending on their thickness. Conductors on different layers are connected via, which are copper-plated holes that function as electrical tunnels through which insulating substrate. Through-hole component leads sometimes also effectively function as vias. After two-layer PCBs, the next step is usually four-layer step. Often two layers are dedicated as power supply and ground planes, and the other two are used for signal wiring between components. Through hole etched components are mounted by their wire leads passing through the board and soldered to traces on the other side. Surface mount components are attached by their leads to copper trades on the same side of the board. A board may use both methods for mounting components or it not use both methods. PCBs with only through-hole mounted components are now trite and not a common use this days. Surface mounting is used for transistors, IC chips, resistors and capacitors.

Through-hole mounting may be used for some huge components such as electrolytic capacitors and connectors. The pattern to be carved into every copper layer of a PCB is called the "artwork". The etching is usually done using done by photo resisting which is coated onto the PCB, then exposed to light projecting pattern of the etched design pattern craved. The resist material safeguards the copper from dissolution or discoloring into the etching solution. The etched board is then given finishing touch. A PCB design can be mass-reproduced in a way similar to the way photographs can be mass-duplicated from film negatives using a photographic printer. In multi-layer boards, the layers of material are laminated together in an alternating layers like sandwich: copper, substrate, copper, substrate, copper, etc.; each plane of copper is etched, and any internal via are plated-through, before the layers are laminated together. Only the outer layers need be coated; the inner copper layers are protected by the adjacent substrate layers. A minimum PCB for a single component, used for prototyping, is called a breakout board. The purpose of a breakout board is to "break out" the leads of a component on separate terminals so that manual connections to them can be made easily. Breakout boards are especially used for surface-mount components or any components with fine lead pitch.

Advantages of Plotter Etching Method

The main advantage one gains by using a plotter is that it allows you to work on huge sheets of paper, while still providing exceptional quality resolution accurately. The plotter can also print vast range of materials, and therefore provides you with more options in printing the circuit design. Plotters can easily draw on cardboard, plastic, aluminum, plywood, sheet steel, and other flat sheet materials. The biggest attributes of the plotter include speed, accuracy, reproducibility, and efficiency, while in some plotters you have the option of saving different templates and patterns to get rid of the hassle of loading templates and patterns repeatedly. It can easily draw a single pattern over a 100 times with no degradation.

Disadvantages of Plotter Etching Method

The plotter also has quite a lot of disadvantages, but in the long run, the positives outshine the negatives of this printing machine. If you don't have the necessary space to work, and you don't have the time to find a good location for your plotter, then you will have some issues.

This is because these machines will take more space than regular printers, and you also must consider that plotters are more expensive than printers on the market.

2.3 Traditional method for plotting

A printed circuit board (PCB) mechanically supports and electrically connects electronic components using conductive tracks, pads, and other features are etched from copper sheets laminated onto a non-conductive substrate. A printed circuit board has pre-designed copper track lines on a conducting sheet. The pre-defined tracks reduce clustering of wiring, thereby reducing the faults which may be arising due to loose connections. One simply needs to place the components on the PCB and solder them through the various steps that include traditional PCB plotting are: 1. Take printout of circuit board layout. Mirror print out should be taken. Select the output in black both from the PCB design software and the printer driver settings. Making sure that the printout is made on the glossy side of the paper and cut copper plate for circuit board as per the requirement.

Transfer the PCB print on the copper plate by transferring the printed image (taken from a laser printer or which is drawn) using photo paper to the board. It's important to flip top layer horizontally. Put the copper surface of the board on the printed layout. Ensuring that the board is aligned correctly along the borders of the printed layout and use tape to hold the board and the printed paper in the correct position. Iron the circuit from paper onto the PCB plate impressing the design well on it. After printing on glossy paper, iron the image side down to the copper side, then heat up the electric iron to the max temperature. Put the board and photo paper arrangement on a clean wooden table with the back of the photo paper facing upwards. Using pliers or a spatula, hold one end and keep it steady. Etching the plate and final step is to give final touches to the circuit board. The method has advantages and disadvantages— Some parts cannot handle the thermal shock experienced by being passed through molten solder – When wave soldering is used, it is necessary to hold components in place: Wave soldering is FAST – Reflow soldering does not really work with through-hole parts. Regardless of the method used, particular temperature profiles must be attached to.

Advantages of Traditional Method of Plotting

One must be knowing more about the process of chemical PCB etching rather than PCB milling – so the difference, as method should be capable of producing high quality printed circuit boards. PCB milling is a subtractive process which means it involves the removal of areas of copper from a sheet of PCB material in order to recreate sheets, signal traces lines and structures based on supplied patterns on a digital circuit board plan or 'layout file'. When the copper is removed it facilitates electric isolation and creates the necessary ground planes. With the plotting method, the circuit board's quality will be determined by the accuracy of the PCB plotting system. It also includes the control and the sharpness and overall condition of the bits and also the system's feed and speed of rotation. With the chemical PCB etching process, the quality is dependent on the quality and accuracy of the photo masking and also the quality of the chemicals used in the etching.

III. WORKING MODEL

The output of most of the PCB designing software is in the form of .Gerber file which will be first converted into the G-code file. The G-code file will then be given as input to the Arduino. According to input the Arduino will control both the stepper motor and the servo motor will help the system as a backup provider to perform the movement of the both x and y axis. User can draw on a comprehensive parameter library for numerous common and exotic material for their projects. And there's more—when the user enters information on the lab equipment available, it will optimize the work steps for the entire manufacturing process.

IV. SIMULATION TOOLS

First the PCB diagram is plotted on the CAD tool, then we input that file which has been created on the Cad tool is then be converted into .Gerber file. After this we input it as a python file and then we would convert this file into gcode and then we will input this gcode file into Arduino. This Arduino will be then be the controller for the x y and z axis. These axis will control functionalities, x axis will move the x axis of the plank, similar for y axis and z axis will move the pen up and down into to plot.

V. TERMINOLOGIES

G-code, which has many variants, is the common name for the most widely used numerical control (NC) programming language. It is used mainly in computer-aided manufacturing to control automated machine tools. G-code is sometimes called G programming Language.

G-code is a language in which people tell computerized machine tools how to make something. The "how" is defined by g-code instructions provided to a machine controller (industrial computer) that tells the motors where to move, how fast to move, and what path to follow. The most common situation is that, within a machine tool, a cutting tool is moved according to these instructions through a tool path and cuts away material to leave only the finished work piece. The same concept also extends to non-cutting tools such as forming or burnishing tools, photo plotting, additive methods such as 3D printing, and measuring instruments.

VI. METHODOLOGIES

Gerber is an open ASCII vector format for printed circuit board design that shows parts of a board like copper layers, solder mask design, etc. Many applications can read this format because it simple, has detailed specifications, and has few bugs because of its wide use and subsequent debugging. The simple format of a Gerber means it is easy to identify and fix bugs as well. There are three Gerber generations: Gerber X2 (current format), Extended Gerber, or RS-274X, and Standard Gerber, or RS-274-D, which is now obsolete. The Gerber X2 format contains layer stack up information, which was not in previous versions. Software that supports the X2 format will read and identify each item's function in the file.

A RS-274X file contains the complete description of a PCB layer image without requiring any external files. It has all the imaging operators needed for a PCB image. Any aperture shape can be defined as well as positive and negative objects. Planes can be specified without the need to "paint" or "vector-fill" as in RS-274-D.

RS-274X is a complete, powerful and unambiguous standard to describe a PCB layer. It can be input and processed fully automatically. This makes it well suited for fast and secure data transfer and for reliable and automated workflows.

G-code is created as the output from sophisticated CAD/CAM (computer aided design/computer aided manufacturing) software. Since there are many different brands of design software available the type of code they generate will also be different types. However, most major brands have translation ability that allows them to be compatible with the vast variety of commercially available CNC machines.

Each line of code tells the machine to perform one discrete action, including position, speed, rotation, etc. Shapes are made by stringing together point-by-point sets of instructions. Even simple parts can require hundreds or thousands of lines of code, and ultimately they must all work flawlessly together to achieve the desired result.

Because G-code works as a series of points and line-by-line instructions, it's therefore possible to make the same part using a variety of paths and instructions, and the resulting parts are may not always the same. Technically a machine operator doesn't need to know G-code and if the CAM software has already created a cutting program, it feeds that information to the CNC machining center. The software has already determines the speed and feed, the tool path and all the other variables needed to make the part accurately. The operator can simply use the start button and see etching the part being made, but there are many problems with this approach.

CAM programs produce the optimal tool path for the fastest and most efficient cutting of a part, especially for complex geometries figures. This is because, as reason, it is plotting point-by-point and step-by-step, not thinking globally. Only a human machinist with real-world experience is capable of determining the ideal function of the machine tool to meet the expectations of the customer's design intent. CAM is also optimized for maximum safety and machine tool life, which translates as slow

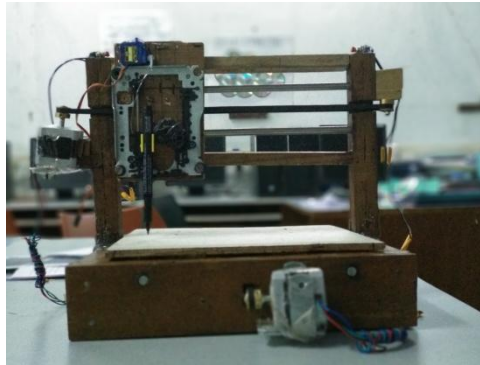
Also, CAM software can sometimes make errors and alteration, or create a etching program that may need to be hitching. If an operator doesn't know how to modify individual lines of code they would then re-programming needs to be done from the very start which would waste time and efforts. But the experienced operator can fine tune the program, one line of G-code at a time, to create the most efficient program to create the highest quality part with the lowest cycle time. Now we will explains the actual framework of the system i.e. the working of the wooden frame. There is base plank which is frame with a square shape of your desirable cm side which is hollow from inside. There are two rods which are fitted in between them on which we have two shafts on individual rods which will support for the paper on which we would draw the PCB diagram. There are two vertical rods which will support the movement of the pen controlling it. All these movements will be conducted by two motors and one servo motor on dc adapter. The plank would support x and y axis and the pen movement will manage the z axis. The Arduino will be attached to the motor which will precise its speed and its movement.

TABLE 1 VARIOUS EXTENSIONS

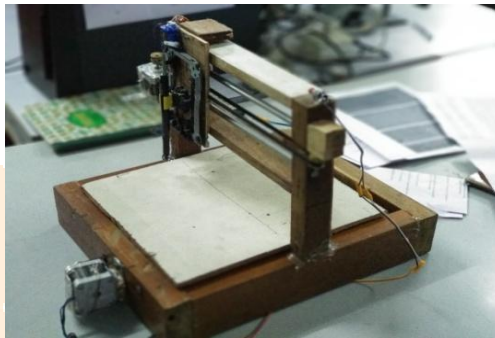
| Extensions | Descriptions |
|---------------------|--------------------------|
| Cmp | Top copper |
| Sol | Bottom copper |
| Stc | Top solermask |
| Plc | Bottom sildermask |
| Pls | Top silkscreen |
| Drd | Nc drill |
| Top | Top copper |
| Bot | Bottom copper |
| Smt | Top silkscreen |
| Smb | Bottom silkscreen |
| Sst | Nc drill |
| Ssb | Bootom silkscreen |
| Thruhole tap | Nc drill |
| gtl | Top copper |
| Gbl | Bottom copper |
| Gts | Top solermask |
| Gbs | Bottom solermask |
| Gto | Top silkscreen |
| Gbo | Bottom silkscreen |
| Drl | Nc drill |

IV. RESULTS AND DISCUSSIONS

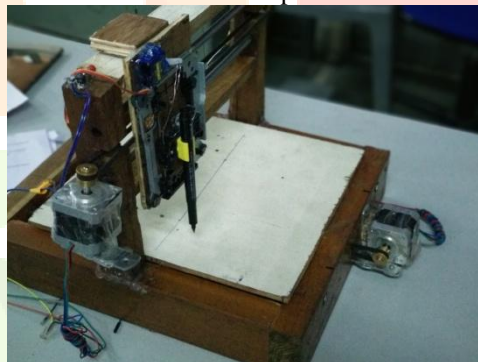
The printing time for etching of circuit takes a bit longer or more time but the accuracy level obtained from the portable PCB schemer is 0.2mm. Below are the images of the working model.



Portable Pcb schemer



Side view of plotter



Plotting circuit diagram

VII. ACKNOWLEDGMENT

We would like to express our special thanks of gratitude to our project guide Professor Shiburaj Pappu and Principal Dr. Varsha Shah who gave us golden opportunity to do this wonderful project. We came to know about so many new things. Secondly we would also like to thank our parents and friends who helped us a lot in finalizing this project in limited time of frame.

REFERENCES

- [1] Improved DDA Line drawing anti-Aliasing based on embedded graphics system. INSPEC Accession Number 1153781
- [2] Parallelization of Bresenham's line and circle algorithm. INSPEC Accession Number: 12657144. DOI:10.1109/ISdea.20
- [3] Bhatti, U. and Hanif, M. 2010. Various methods of etching. Evidence from KSE-India. European Journal of Economics, Finance and Administrative Science, 3 (20).