



IConAMMA_2017

A Build-Your-Own Three Axis CNC PCB Milling Machine

N. Sathyakumar, Kamal Prasath Balaji, Raja Ganapathi, S.R.Pandian*.

Teaching Learning Centre for Design and Manufacturing Education, Indian Institute of Information Technology, Design and Manufacturing, Chennai 600127, India.

**Department of Electronics Engineering, Indian Institute of Information Technology, Design and Manufacturing, Address, Chennai 600127, India*

Abstract

The need for fabricating a prototype circuit arises frequently in electronics, including education and research laboratories. In resource-poor countries in the developing world, this is hindered by the high cost of commercial Printed Circuit Board (PCB) prototyping machines and long turn-around commercial fabrication process. Practical, hands-on laboratory teaching and experimentation becomes necessary to improve learning in electronics. In this paper, a low-cost build-your-own (BYO) semi-automated three-axis PCB milling machine for double-sided PCB prototyping is developed using commercial components and open source hardware and free open source software to provide students, teachers, and engineers an understandable, affordable source for PCB prototyping. Also, the main problems encountered during fabrication of PCB have been mentioned and the techniques used to solve are discussed in detail.

© 2017 Elsevier Ltd. All rights reserved.

Selection and/or Peer-review under responsibility of International Conference on Advances in Materials and Manufacturing Applications [IConAMMA 2017].

Keywords: Build your own technology; CNC PCB machine; PCB manufacturing; Auto-levelling; Open source hardware and software.

1. Introduction

It is estimated that the global electronics industry output will reach USD 2.4 trillion by 2020. The demand in semiconductor industry in India alone is expected to be USD 400 billion in 2020 and it is expected to create around 27 million job opportunities [1]. By contrast the entire IT sector provides around 3.7 million jobs. Therefore, the Indian government is encouraging initiatives such as Make in India, Digital India, Startup India, and Skills India to facilitate growth of manufacturing and electronics industries in India, creating numerous high-paying jobs in the process.

Thus, semiconductor industry is one of the fastest growing industries in our country and thereby. With this, the production and the standard of printed circuit boards (PCB), which is the heart of every electronic product, are on the rise. PCBs not only provide mechanical support for the electronic components and also provide other services like

electrical impedance matching, electromagnetic shielding and heat conduction [2]. Specialized courses and curriculum in PCB design and electronics assembly are introduced in electronics engineering education. However, due to high cost of commercial and often imported PCB prototyping equipment, there is a severe lack of practical hands-on PCB design teaching and learning in India. This situation can be remedied with the increasing affordability and versatility of open source hardware like microcontrollers and microcomputers, commercial off-the-shelf components like actuators, sensors as well as free, open source software, which can be integrated for design of low-cost PCB prototyping machines for electronics education.

Do-it-yourself (DIY) PCBs can be designed using simple techniques such as using an iron to transfer ink printed on a transparency to a PCB with chemical etching. But these methods lack sufficient consistency for surface mount devices (SMDs) and the drilling of holes is tedious since it has to be done manually. Further, the environmental and health hazards from chemicals used in these processes are significant [3].

Development of safe and high resolution milling and drilling of PCBs is enabled using isolation routing which overcomes many of the above mentioned drawbacks. In Isolation milling, the copper from the board is first removed to recreate pads, which are signal traces and structures based on the pattern generated by a PCB parts layout file.

In this paper, a BYO PCB prototyping machine has been developed and deployed to make both single- and double-sided boards for through-hole technology and Surface Mount Technology (SMT).

The common problems encountered while soldering the components on the PCB and aligning the board layers have been discussed. Modern and innovative approaches used on an industrial level to overcome these problems have been implemented in our machine and studied. Commercial techniques used in easy and comfortable operation of PCB prototyping machines have been incorporated in the machine.

The above features have been implemented using open source software programs so that teachers and students themselves would be able to fabricate high-resolution PCBs in an academic environment matching near-commercial quality. A major advantage of the proposed system is that users can maintain and repair the machine on their own, without expensive annual maintenance contracts or import of costly spare parts. With the understanding and experience gained, the users can also gradually add advanced features like fully automated PCB machines with pick and place assembly, vision feedback, etc.

The paper is organized as follows. Section 2 discusses how a PCB is designed from open source software and fabricated using an inexpensive CNC milling machine. In 3 Softwares and Procedure to make a PCB layout file and how to generate the fabrication data. 4. Issues and techniques prevalent in fabricating PCBs. 5. Discussions of experiments and results. Then, future scope of work and conclusion.

2. Prototype 3-Axis PCB milling machine

2.1. Electromechanical Setup

The design of the CNC milling and drilling machine used in the proposed PCB prototyping system is based on our earlier developed 3-axis CNC mill, as presented by Pandian and Pandian [4]. A detailed explanation about the mechanical design, hardware and the software can be found in [4].

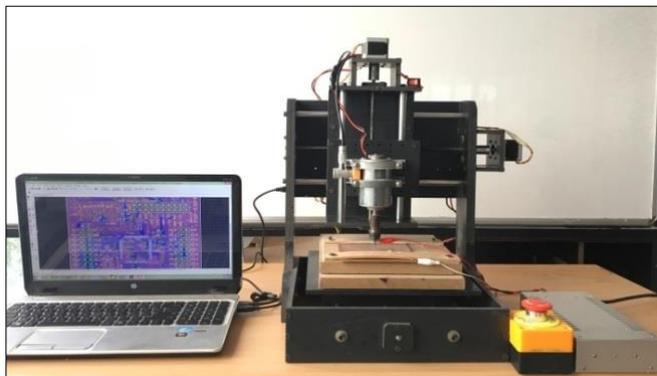


Fig. 1. Mechanical Setup of the Machine

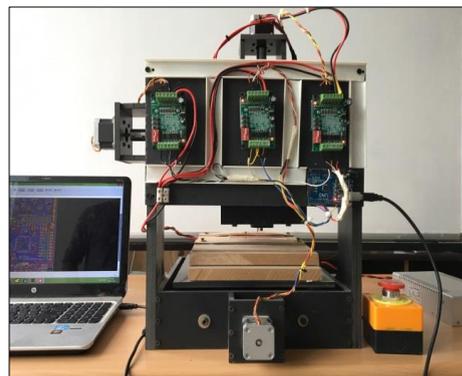


Fig. 2. Electronics hardware assembly.

The mechanical system. Fig.1 consists of three stepper motors for motion along the x, y, and z axes, a PVC-built frame, a high-speed spindle motor, lead screws for power transmission and other related accessories. The main specifications of the milling machine are listed below:

Table 1. PCB milling machine specifications.

X,Y,Z axes travel	180 x 180 x 50 mm
Motors	Steppers: 3x NEMA 17,200 step/rev, 2-phase, 1.3A Spindle motor: 5000rpm @24 V DC, 0.3A no load
Lead screws	Stainless steel, 3xM8x1.25, 20 tpi
Stepper drivers	3 x single axis, rated 3A, up to 1/16 micro stepping
Speed	X, Y axes: 8 mm/sec
Resolution	Electrical: 1.8° (0.0062 mm/step) Mechanical: 0.01mm/step
Weight	14 kg
USB microscope	Resolution: 640x480 pixel
Microcontroller	Arduino Uno

2.2. Tooling

The high speed DC spindle motor is used for isolation routing through milling and for through hole drilling. The milling and drilling tool bits are mounted in turn on the spindle motor with a precision ER-11 chuck which holds bits with 1/8 shank dia. A 30° V-engraving bit with end width of 0.1mm is used for routing. A 0.6mm single-fluted drill bit is used for making holes and a 0.8mm end mill bit is used for copper rubout and for cutting sections of the boards. All bits used are tungsten carbide bits because of their extended tool life.

2.3. Electronics assembly

The electronics unit Fig.2. is placed compactly on the rear of the milling machine. Single stepper motor drivers are used to power the individual stepper motors for moving the bits along the x, y, z directions. The stepper drivers are powered by a switched mode power supply (SMPS) DC power source of 24V.

Stepper drivers are reconfigurable with different settings for maximum permissible running current, stall current, excitation modes of stepper motor actuation, etc. The running current can be varied from 0.3A up to 3A. Excitation modes are available for full step, 1/2, 1/4, 1/8, and 1/16 of stepper motor turns. An emergency kill switch is included from the input side and also a probing setup is used as explained later

2.4. Arduino-based controller

GRBL [5], [6] is a free, open source software (FOSS) used to generate CNC machine G code and M code signals that will drive stepper motors using the Arduino microcontroller. G-code and M-code are programming languages [7] that are interpreted by CNC controllers. G-code provides instruction commands for position, speed and path of the motion, whereas M-code provides auxiliary commands for spindle motor speed, coolant flow, tool change etc. Compiled version of GRBL software code is available in the form of hex file and is burned into the Arduino. Grbl CNC code will receive signals from Arduino's serial buffer and parses it to decode the serial data into G-code. Grbl settings in Arduino will be stored in the EEPROM of Arduino and so configuring once will not be erased during power off. Settings can also be viewed and modified anytime by sending corresponding configuration characters. \$\$ symbol is used as configuration character from which we can view different settings of the machine, such as axis feed rate, steps/mm, software limits, axis acceleration values, and resolution of movement.

3. PCB fabrication procedure

The design of PCB is drawn in Eagle computer-aided design (CAD) software and the fabrication data is generated in one of many acceptable formats such as .brd or Gerber files. This file is then imported into computer-aided manufacturing (CAM) software, which outputs a G-Code file which is a machine readable form of instructions. The overall procedure in PCB design, conversion and fabrication is explained below.

3.1. PCB layout design

The process starts with circuit schematic design of PCB in Eagle computer-aided design (CAD) software [8]. The free student version of the software allows us to create up to 999 sheets including all 16 layers. Firstly, all the parts needed for the circuit are added and connected according to the circuit diagram on the Schematic sheet. Eagle has a library navigator which has a collection of parts used in circuitry to be added in the schematic. Thousands of user-created library packages are made available online by the open source PCB community and could be added to the existing libraries. The voltage and the ground terminals are added and the components are connected with each other. A typical circuit diagram is shown in Fig. 3, and its schematic in Eagle is shown in Fig. 4.

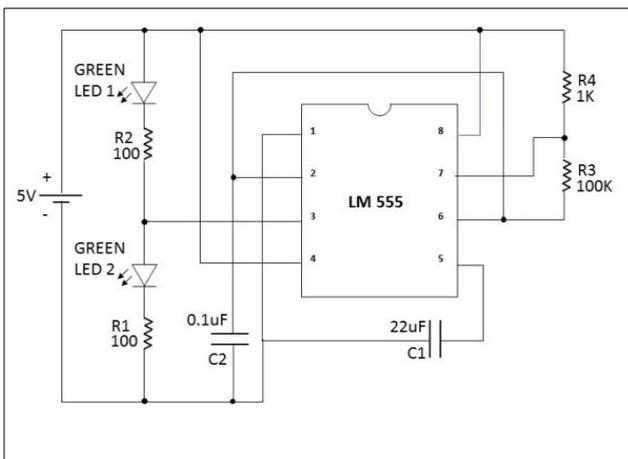


Fig. 3. Circuit diagram of LED flasher using 555 timer.

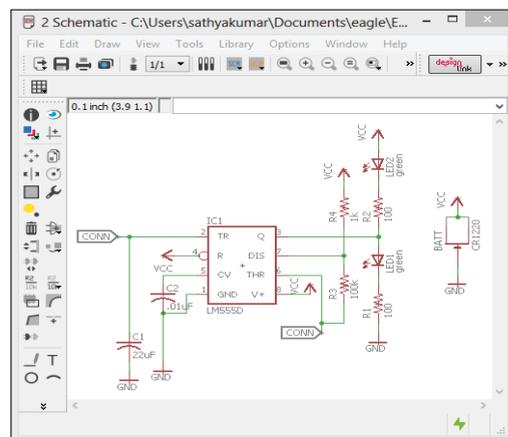


Fig. 4. Circuit drawn in schematic.

The conceptual circuit schematic is next converted into the board Layout, where the actual positioning of components and copper traces are done. Initially, all the parts from the schematic would appear in the Board file stacked upon each other connected by airwires corresponding to the net connections made in the schematic. Once the components are positioned onto the board, they are connected by the copper traces set on a single side or double side. Copper traces can be set in different bent styles, angles, width, distances, etc. The parts are arranged in such a way that the PCB is compact and small. A sample board design layout appears in Fig. 5.

Finally, we use the Design Rule Check (DRC) of Eagle to check our design for errors. Some of the important and common errors DRC finds are minimum clearance distance between traces, inappropriate trace widths or drill sizes and overlap between pads or traces.

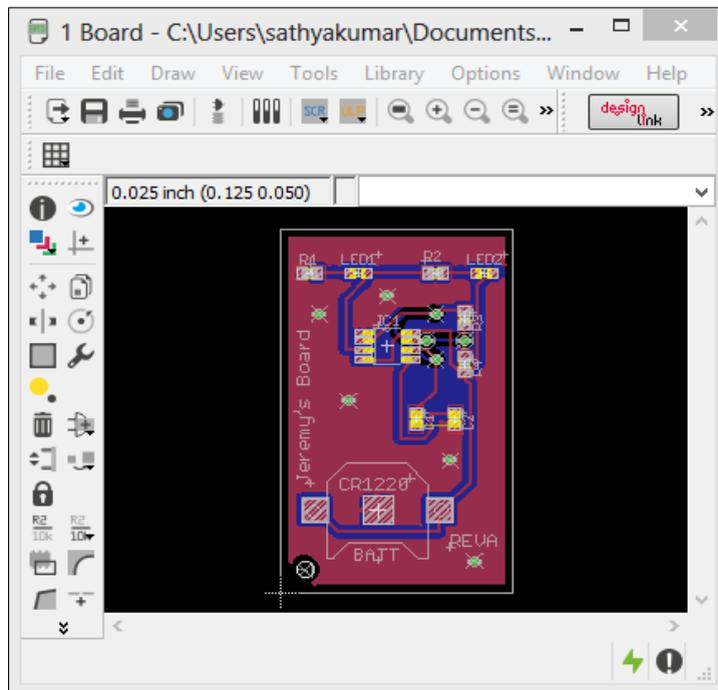


Fig. 5. Board layout corresponding to figure 4.

3.2. G-code generation

The completed file has to be fed into computer aided manufacturing (CAM) software PCB-GCode converter [9] which converts the .brd file into a G code file with the input tool and machine parameters specified. PCB-GCode is the User Language Program (ULP) for Eagle produced by Cadsoft that we use as CAM software for producing G code. ULP loads in the Eagle program and accesses its data structures and creates a variety of output files of desired type and specifications. Various parameters for the toolpaths, design type, feedrates and other machining parameters for PCB are inputted into the settings of the PCB-Gcode software.

Some of the important machining settings used are

1. Isolation - Increasing helps in eliminating small slivers of copper left in-between tracks after cutting process.

$$\text{Isolation} = \text{Etching tool size} + \text{Minimum Isolation} + (\text{Pass No.} * \text{Step size})$$
2. Generation - The settings under this allows you to specify if you are fabricating a single or double-sided board, which side you want the drilling process to happen, mirroring the pattern, etc.
3. Drill - Changes the depth of drills and mounting holes and drill dwell time.
4. Feed rates - These settings control how fast the CNC mill will cut and drill the board. As the feed rate increases, the stress on the tool increases. Also, slower feed rates will ensure a smooth and clean cut on the board.
5. G-Code Profiles - Altering the files under this helps you change the formats used by the program to generate the G-codes to suit your machine.

Once all the input parameters are entered correctly, the software shows the preview and generates G-Code files that cut out tracks individually for top and bottom layer, drill holes, engrave texts, and mill out sections of the board.

3.3. CNC machine control

The G-Code program is sent to Arduino from control computer (typically, a PC) by serial communication. Few standard GUI open source software packages available for this are bCNC, Universal G Code Sender [10] or CNC Grbl controller. These programs are very versatile and control the machine in manual mode or file mode. Options similar to the human-machine interface (HMI) of expensive commercial CNC machines are included which help in moving each axis individually, setting workspace origin, and homing cycles. Also, current position of world coordinates and job coordinates are displayed based on signals sent from the software.

The software has options to set the current position to origin, return the machine to origin positions, as well as write and save complex commands under the macros tab used for tool and work offsets. The files generated by the CAM software are loaded one by one and the tool is moved to the origin position and the job is run.

3.4. Isolation routing

In prototyping single-sided PCBs, the tool is moved and set to an arbitrary origin position (usually on the lower left corner of the board). Now the Etching G-code file is run which cuts out the tracks and the pads. Now the tool is moved again to its previously set origin position and the drilling G-code file is run. Usually the option in the PCB G-code software to create spot drills in the etching stage is used and then the actual drilling G-code file is run. Spot drills create ‘dimples’ or dents in the drill points so that the actual drill tool tip does not deflect and walk-off center and the hole is drilled correctly. In case of double-sided boards, the same procedure is followed and the top side is engraved. Next, the board is flipped and aligned to the axis of the top layer and the bottom layer is engraved and drilled.

4. Techniques used in fabrication

In the following section, various technical problems encountered while fabricating quality PCBs are discussed and the methods employed to solve them have been addressed.

4.1. Zeroing the CNC and probing

Moving the CNC tool tip to its exact origin position is known as Zeroing the CNC. Only then, we can ensure the preciseness of the CNC and it has been a challenge in CNC operation. Several methods and devices have been proposed for this but the best and convenient solution is to use a microscope/camera mounted parallelly next to the spindle. The camera points downwards towards the raw material and shows a magnified image on the computer screen including crosshairs. A hole is first drilled on the board surface using a drill tool and the spindle is moved until the crosshairs of the image line up with the center of the hole. The offset distance between the camera and the tooltip is now measured from the position displayed in Universal G-Code Sender (UGS) and then input the offsets under the macro tab of UGS. Thus, in order to set any point we want as origin, we move the camera to that point and use the macro to set the current position as offset point with respect to origin.

The above procedure will accurately set desired X, Y = 0. But, the vertical position of the spindle, Z axis is also an important axis to zero out properly. For this, one wire is connected to the tool tip, i.e. the probe and the other end is pulled high with a resistor and connected to an input pin of Arduino (Fig.6). Another wire connected to the ground pin is attached to the surface of the PCB when we mount it. We then use probing command “G38.2” in UGS which moves the tool towards the board with a given feed rate until the tip touches the board and makes an electrical contact through the wire. Now, this way the CNC easily determines the exact top of the surface and sets the origin position.

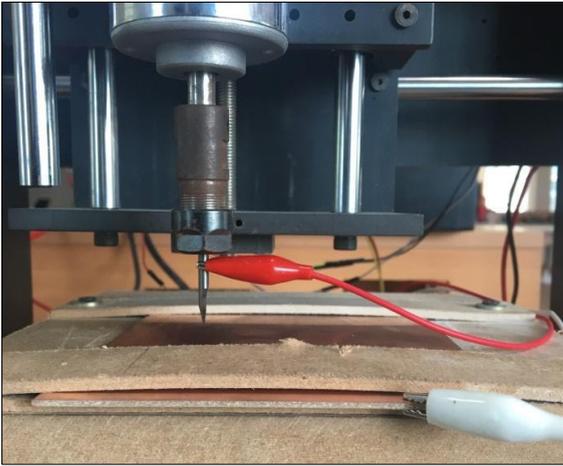


Fig. 6. Probes attached to tool tip and board

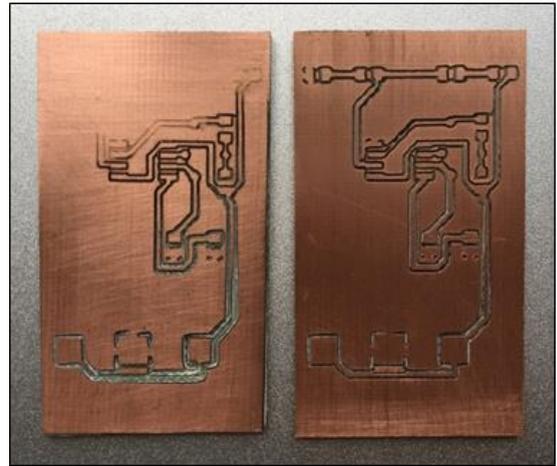


Fig. 7. Effect of autoleveller in eliminating height variations.

4.2. Autolevelling

One problem that exists while trying to create finely etched traces is the inconsistency in the height across the surface of the board. Reasons for this are that the CNC bed may not be flat or that the boards could be warped or bent, which is usually the case if the boards are larger in size. The ideal depth of cut is found to be 0.01” i.e. 0.254 mm. A 30° V-bit with this depth of cut gives a groove of width of 0.136 mm. Even minute height variations such as 1mm would increase the groove width to 0.672mm. This can in turn cause the copper tracks between grooves to become too narrow, or create shallow “aircuts” forming incomplete traces when the height reduces; also increase in the depth of cut in the dielectric layer causes impedance variations in high-frequency PCBs [11]. The technique used here to solve this is to probe the PCB surface in a grid pattern for height variations and modify the G-code so that there is uniform depth while milling. The software used here to do this is Autoleveller [12] which interacts with the CNC controller and modifies and outputs a G-Code file for us to run with UGS eliminating the height variations. PCB board milled without and with the use of Autoleveller respectively when the bed is not flat are shown below (Fig. 7).

4.3. Alignment for double sided PCB

One of the biggest challenges in making double-sided boards is aligning the board after it is flipped for milling the bottom layer. Many methods for alignment have been proposed. One example is using alignment holes along the axis of flipping and driving pins to secure the board in place. But this method has been found to give some errors. So, the best technique found is to drill reference holes along two corners of the PCB and locate them using the camera and measure by how much angle the board is skewed. The points in the etch file and drill file are then rotated according to this angle and run.[13],[14]. The matrix for rotation of coordinates around the origin by an angle θ from reference axis (X,Y) to rotated axis(X',Y') is given as (Fig. 8.a,b).

$$\begin{bmatrix} X' \\ Y' \end{bmatrix} = \begin{bmatrix} \cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{bmatrix} \begin{bmatrix} X \\ Y \end{bmatrix}$$

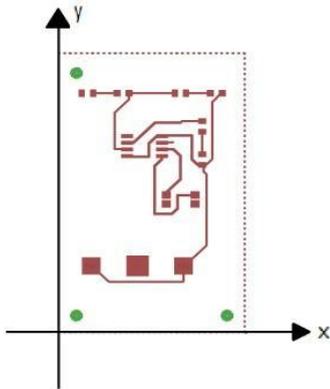


Fig.8.a. Top side of PCB

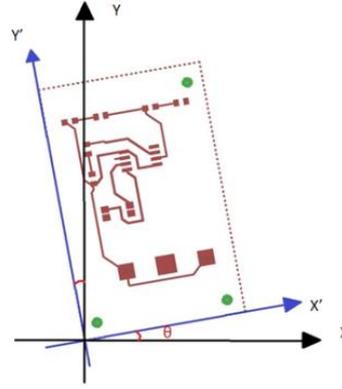


Fig.8.b. PCB flipped and rotated through an angle

The angle is measured by the work position of the reference points displayed in the UGS. Now instead of finding the transformation matrix and multiplying it to all the points in the G-Code file, we simply rotate the machine's coordinate axis system by the required angle using the macro command (G68) and all the matrix transformation of the points occur automatically inside the CNC controller. This step-by-step method ensures surety in rotating the matrix than using separate software for matrix transformation. Fig.9 shows the hole drilled being identified with camera and the corresponding work position.

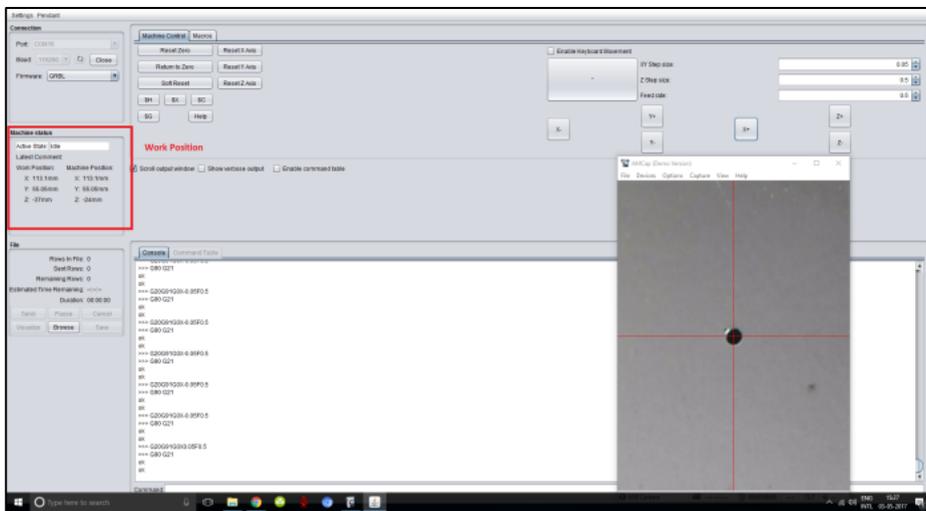


Fig.9. Capture of reference hole in camera image and measuring current work position in Universal G-code Sender.

4.4. Soldering

Soldering of SMDs has been found to be difficult and is believed to be a job of the expert. Microwave heating as described below would enable users to perform the soldering conveniently. Formation of solder bridges are prevented by routing tracks with high isolation distance and this has found to provide good soldering without the need for solder mask in procedure. Lead-free solder paste is first applied on the fluxed tracks and components are mounted on top of them. This assembled board is then preheated for a while and then baked at moderate temperature in a microwave oven. The solder joints are formed using reflow process and the components align themselves precisely with the flux. Use of microwave heating facilitates both simultaneous and selective heating process which does not cause damage to the components. The quality and reliability of the solder joints formed are found to be reliable. The other properties and advantages of soldering using microwave heating are explained in [15], [16].

5. Results and discussions

Different sets of experiments have been done with the designed machine similar to [17] to verify its practical utility and presented below. In the former part, the line resolution of the PCB is tested and a series of SMD footprints has been fabricated. The fabrication of RF circuits is discussed in the later part.

5.1. Wire width and resolution

A series of copper tracks with diminishing width has been tried and the thinnest track line that could be safely fabricated with the machine is found to 0.2mm. Table 2 shows the calculation of the resolution of the line width measured using microscope. (ΔW) is the deviation in the width from the theoretical input width (W_{IN}) to the actual routed width on PCB (W_{PCB}). Fig.10.a.shows the lines routed in the order of diminishing thickness from top to bottom and a 0.1mm irregular track line at the top.

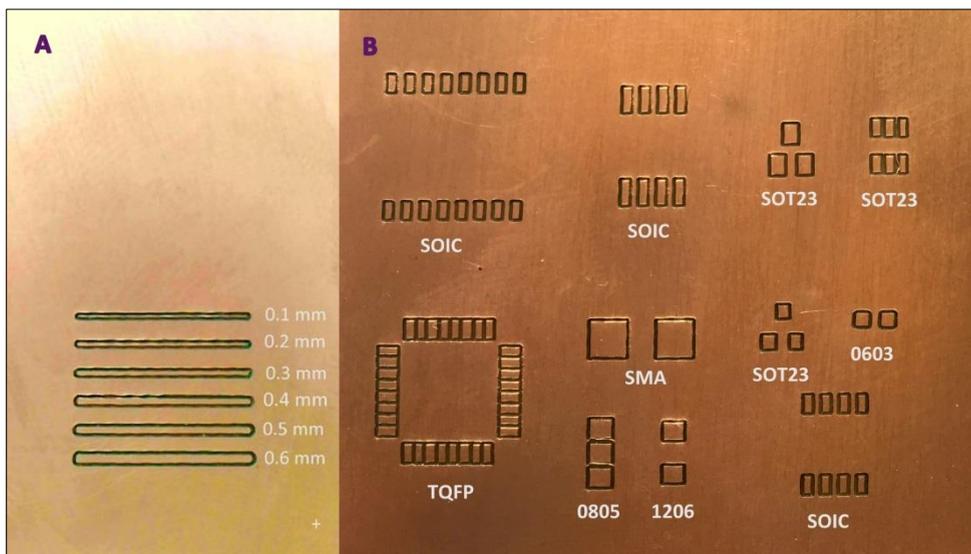


Fig. 10. PCB fabricated with Lines and SMT footprints

Table 2. Calculation of line width resolution.

Input unit W_{IN} (mm)	W_{PCB} (mm)	ΔW (mm)
0.6	0.563	0.037
0.5	0.492	0.008
0.4	0.422	-0.022
0.3	0.352	-0.052
0.2	0.212	-0.012
0.1	NIL	NIL

5.2. SMT board

Fig 10.b. shows a fabricated board with some of the conventionally used SMT. It consists of SMT IC packages such as SOIC, TQFP, SOT23 and resistor and capacitor packages such 0603, 0805, 1206, SMA etc. It has been inferred that a SMT footprint with a pitch size of as low as 0.3mm can be fabricated safely with the machine.

5.3. Radio-Frequency circuits

A conventional branch line coupler is constructed employing four $\lambda/4$ transmission line in a ring. By following the synthesis approach in [18], [19], we have designed and fabricated a branch line coupler (Fig.11) using microstrip technology working at 2.45 GHz on a low cost 1.65 mm thick FR4 epoxy material with dielectric constant of 4.4 and a loss tangent of 0.02. The physical dimensions were calculated using microstrip line calculator and the board was drawn in eagle. Fig.12 illustrates the full-wave simulated S-parameters of the designed branch line coupler. From the graph, the return loss (S_{11}), throughput (S_{21}), coupled (S_{31}) and isolation (S_{41}) are calculated as -3.7 dB, -3.7 dB, -27.4 dB, and -31.63 dB, respectively. The experimental measurements of the fabricated prototype and further calibration are under progress.

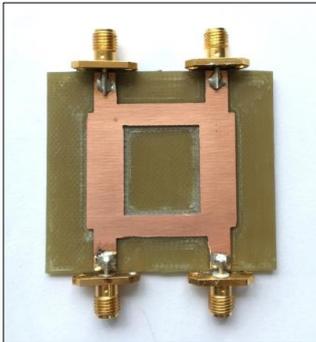


Fig.11. Fabricated PCB of a Conventional BLC.

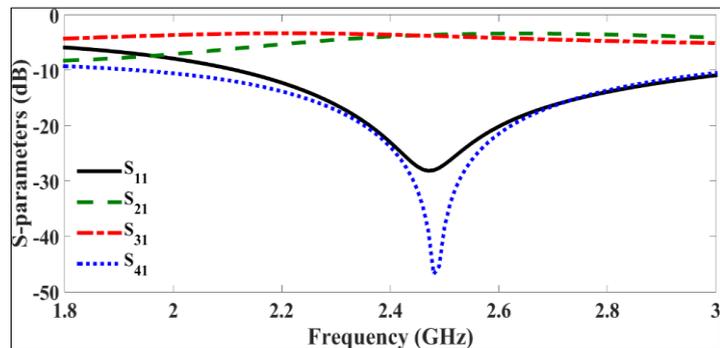


Fig.12. Full wave simulated S-parameters of BLC.

6. Scope for future work

A software program to find the rotation angle for the bottom layer during fabricating two sided boards and to perform G-code transformation is being developed. It will identify the reference holes on the corners of the board using computer vision (OpenCV) and send signals to the CNC to move the camera to the centroid of the rectangle and measure the angle of the axis of bottom layer. It will then create a transformation matrix and modify the G-Code and run the etch files automatically. This program will also have the ability to interact with the CNC controller and automatically send macros, find the camera offsets and perform zeroing of CNC and run the whole operation cycle on its own.

Pick and place mechanisms for changing the tool and placing the SMD components can be added in future. In order to implement more precise and fast operation of the machine and as well as for laboratory teaching, CNC machines with stepper or servomotors having optical encoder feedback can be developed.

7. Conclusion

A BYO PCB prototyping machine has been designed and developed for fabrication of both single and double-sided boards with through-hole and surface mount technology. The resolution of the machine was studied and fabrication of RF circuits has been presented. An industrial method for soldering the components has been mentioned. Problems usually encountered while fabricating high precision and double-layered PCBs have been discussed. Commercial techniques used for easy and comfortable operation of PCB prototyping machines have been incorporated in the procedure. The machine is fabricated with commercial and inexpensive open source hardware components and software and so can be readily disseminated, adapted and improved for widespread use in electronics education.

References

- [1] Shweta Sengar, 2015, The rising semiconductor industry in India, <http://www.electronicshub2b.com/industry-buzz/rising-semiconductor-industry-india/>

- [2] Dr. Ron, 2010, <http://www.indium.com/blog/the-printed-circuit-board-more-important-now-than-ever.php>
- [3] P. H. Wald and J. R. Jones, (1987), Semiconductor manufacturing: An introduction to processes and hazards. *Am. J. Ind. Med.*, 11: 203–221.
- [4] S. Pandian and S. R. Pandian, 2014, "A low-cost build-your-own three axis CNC mill prototype", *Int. J. Mechanical Engineering and Robotics*, 2(1), pp. 6-11.
- [5] http://dank.bengler.no/-/page/show/5470_grbl?ref=checkpoint
- [6] <http://onehosshay.wordpress.com/2011/08/21/grbl-how-it-works-and-other-thoughts/>
- [7] Peter Smid, 2008, *CNC Programming Handbook*, Industrial Press Inc., U.S.
- [8] M. Scarpino, 2014, *Designing Circuit Boards with EAGLE*, Pearson.
- [9] <http://www.richa1.com/RichardAlbritton/create-g-code-from-an-eagle-file/>
- [10] <https://github.com/winder/Universal-G-Code-Sender>
- [11] M. Gatchev, S. Kamenopolsky, V. Bojanov and P. Dankov, "Influence of the milling depth on the microstrip parameters in milled PCB-plates for microwave applications," *Int. Conf. Microwaves, Radar and Wireless Communications*. 2002, pp. 476-479, Poland.
- [12] <https://www.autoleveller.co.uk/autoleveller-guide/>
- [13] <https://planet-cnc.com/how-to-mill-and-drill-double-sided-pcb-using-transformations-and-warp/>
- [14] <http://www.alessiovaleri.it/using-transform-matrix-for-pcb-drilling-part-1/>
- [15] M. Nowotnick and R. Diehm, 2007, "Soldering Technology for 3D PCB Assemblies with Microwave Heating," 2007 IEEE Int. Symp. Industrial Electronics, pp. 3273-327, Spain.
- [16] R. Diehm, U. Pape, M. Nowotnick and W. Kempe, 2006, "Low Temperature Processing of Highly Integrated Assemblies by Selective Heating," *Proc. Electronic System Integration Technology Conf.*, Dresden, Germany, pp. 608-611.
- [17] J. Ou, A. Maldonado, C. Saephan, F. Farahmand and M. Caggiano, "A low-cost PCB fabrication process," 2014 IEEE 64th Electronic Components and Technology Conference (ECTC), Orlando, FL, 2014, pp. 2159-2162.
- [18] R. K. Barik, K. V. P. Kumar and S. S. Karthikeyan, "A compact wideband harmonic suppressed 10 dB branch line coupler using cascaded symmetric PI sections," *Microwave and Optical Technology Letters*, vol. 58, no. 7, pp. 1610-1613, July 2016.
- [19] I. S. Krishna, R. K. Barik, S.S. Karthikeyan, and P. Kokil, "A compact wideband harmonic suppressed 10 dB branch line coupler using cascaded symmetric PI sections," *Microwave and Optical Technology Letters*, vol. 59, no. 4, pp. 913-918, April 2017.