Comparison of Printed Circuit Board Fabricated by using conventional method and Laser Cutting Method

DALP Karunarathne[1#], WSP Fernando[1], PSH Pallemulla[1] and MKAJ Maldeniya[2]

[1]*Department of Mechanical Engineering, Faculty of Engineering, General Sir John Kotelawala Defence University*

[2]*Department of Mathematics, Faculty of Engineering, General Sir John Kotelawala Defence University*

#lahirukarunarathna@gmail.com

Abstract: *There are multiple methods of manufacturing printed circuit boards (PCB). The approach used by electronic enthusiasts is ironing the circuit from the glossy paper onto the copper plate (the homemade method). Industrial PCB manufacturers use laser cutting or computerized numerical control milling to cut the pathways, thus etching the circuit on the board. The former is time-consuming and the latter carries the risk of damaging the board during the manufacturing process, in addition to several other drawbacks. The approach that is presented in this paper avoids these disadvantages. In the proposed method, a polyvinyl chloride sticker is pasted on the copper board and unwanted sections of the sticker are removed by using laser cutting. Subsequently, the copper board with the sticker is immersed in a ferric chloride solution, washed off and the remaining parts of the sticker are removed. The advantages of this approach are the capability of accurately obtaining fine tracks (of 30 mil in width) and being comparatively less time-consuming than other methods used by electronics enthusiasts. In this paper, the procedure for making a PCB with the proposed method is laid out. Then, the function and design of a low-power laser cutter to accomplish the former is discussed.*

Keywords: *printed circuit board, laser cutting, polyvinyl chloride sticker*

1. Introduction

A printed circuit board (PCB) is used to connect electronic components with one another, forming a compact circuit. These components are fastened to the surface of the PCB, which are then electrically connected via conductive tracks. When developing a circuit on a PCB, components are typically soldered on to the board to hold them in place. High-quality PCBs are fabricated for industrial applications by large-scale manufacturers but many electronic enthusiasts use relatively simple methods to develop PCBs of their own, which they can use for minor projects.

In the typical method that electronic enthusiasts use to make a PCB, the circuit is first printed on photo (glossy) paper and a hot iron is used to apply pressure and heat to transfer the ink on the paper to a copper plate. This method has the disadvantages of being time consuming as the copper plate has to be printed, ironed slowly, washed and dried. Ironing may have to be done multiple times to get an acceptable print of the tracks on the plate. Moreover, it can be difficult to achieve perfect tracks by ironing, thus resulting in improper electrical connections when electronic components are soldered onto the board. In large-scale manufacturing, a variety of processes are used (Mitzner 2007), including laser cutting (Durillo, Falcon & Aranda 2018; Triano & Collins 2013; Sooraj & Vasa 2020), (Plesha & Kycia 2020), milling

(Choudhary et al. 2017; Basniak & Catapan 2012; Madekar et al. 2016) and maskless lithography (Hansotte, Canigan & Meisburger 2011), (Lee 2010). The most prominent of these methods are the use of a laser cutter or a computerized numerical control (CNC) milling machine to etch the circuit onto the conductive plate. Both these processes are subtractive i.e. removing material from the conductive plate in order to obtain the tracks. This results in the possibility of the board being damaged due to the cutting. Another drawback is that fine tracks and gaps between tracks cannot be obtained due to the diameter of the tool in CNC milling (Bogatin 2021), (Shinde & Mutthurajan 2016) and the width of the beam in laser cutting. The drawbacks mentioned above may result in a common problem faced by users of printed circuit boards, which is the intermittent malfunctioning of the circuit. This problem may also be caused by imperfect connections on the copper plate due to loose or damaged connections caused by handling the PCB. The unreliability of the electrical connections of the PCB has resulted in users testing their circuits on a breadboard to check whether the PCB circuit is working as intended. Such redundancy can be avoided if the tracks on the PCB are properly cut out. To avoid such difficulties, the researchers have proposed a novel approach.

In the proposed approach, a polyvinyl chloride (PVC) sticker is pasted on a copper plate instead of using glossy paper and an iron. The parts of the sticker, where the tracks are to be etched, are cut away by a laser cutter. To achieve fineness in the tracks and minimize thermal damage to the PVC sticker, a CNC laser cutter has been designed for this purpose. The design and the functioning of the laser cutter are provided in Section II B of this paper. After this subtractive process, the copper board with the sticker is immersed in a solution of ferric chloride. The board is then washed with water and the remainder of the sticker is removed.

When the PCB is made with this approach, finer tracks can be obtained compared to other methods. It is also relatively cheap and the fabrication of the board is fast. The results that confirm these facts are presented and discussed in this paper. In order to obtain an objective metric of similarity, an image processing approach is used to compare the results of the homemade method with that of the proposed method.

In this paper, Section II details the steps in the fabrication of the PCB, the function of the laser cutter and calculations related to its input signal. In Section III, the quality of PCBs made using the homemade method and the proposed method are discussed, followed by an assessment of the time taken to cut the PCB tracks using multiple power/cut-rate combinations in the proposed method. A comparison between the time taken to complete the PCB fabrication process in the homemade method and the proposed method is also made in the same section. Lastly, in Section IV, the feasibility of using the proposed method is commented on.

2. Methodology

A. PCB fabrication

The method proposed in this paper uses the following components: a copper board, a PVC sticker (e.g. ORACAL 100G or 100M series), sandpaper, a laser cutter/engraver, Ferric Chloride as the etchant solution and a plastic flat container.

First, the copper board is cut into pieces and the piece to be used as the PCB must be sanded down using the sandpaper. Such a piece is shown in Figure 1. Then, the PVC sticker is pasted on the piece. Subsequently, the piece must be placed in the laser cutter, to which the PCB tracks have been programmed. After the laser cutter has finished cutting the tracks, the remaining parts of the sticker must be removed and the piece must then be placed in the etchant solution so that it is completely immersed. An immersed copper piece is shown in Figure 2. It is well-known that the etchant solution can be produced by mixing hydrochloric acid and hydrogen peroxide or iron (III) chloride. The copper will gradually dissolve in the solution and when the final trace of copper has been removed, the piece must be dried to stop the chemical reaction. Once dried, any remaining stickers must be removed to obtain the proper PCB tracks. The final product is shown in Figure 3.

Figure 1: A copper piece that has been sanded down

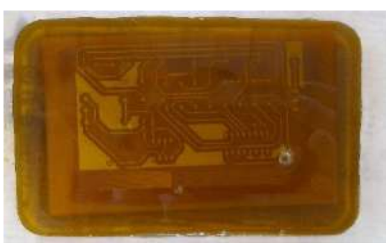

Figure 2: A copper piece with the laser-cut PVC sticker immersed in the Ferric Chloride etchant solution

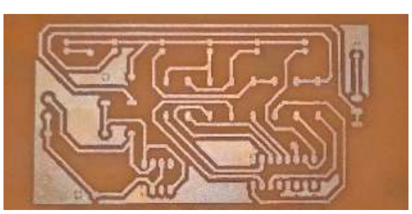

Figure 3: Copper piece after being removed from the etchant solution and dried

B. Function and Design of the Laser Cutter

The laser cutter is an integral part of the proposed approach as it provides a means of generating a low-power laser beam (up to 3.5 W) that is capable of printing the PCB with no damage to the PVC sticker. A wavelength of 450 nm is used for the laser, generating a blue light. The diameter of the laser spot at this wavelength is ≤Φ0.08 with a lens of focal length 20 mm. In contrast to lasers in industrial laser cutters that exhibit fluctuations in power, the proposed design generates a laser of stable power.

Figure 4 shows the block diagram of the CNC laser cutter. In the figure, the computer (A) has the Universal G-code Sender (UGS) platform installed. UGS enables communication between the computer and the laser cutter via universal serial bus (USB) (B). The sketch of the PCB tracks must be drawn by the user and it must be converted to G-code, using appropriate software. Software capable of performing these functions are freely available. The G-codes are then transmitted to the laser cutter by the UGS via the USB.

The G-code stream is sent to directly to the microcontroller unit (C), which is pre-programmed to convert the received G-codes to motor-controlling and laser-controlling commands. The G-code handling in the microcontroller is achieved with the GRBL software, which is installed for the express purpose of generating control commands from G-code input.

The laser driver (D) receives control commands from the microcontroller and generates the laser accordingly. Transistor-transistor logic (TTL) modulation is used to enable, disable and modulate the laser diode. TTL modulation has been chosen to maintain the overall cost at a

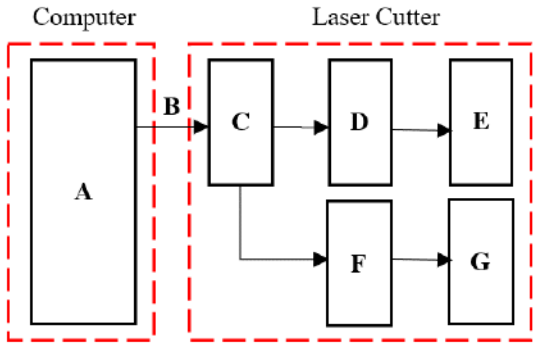

Figure 4: Block diagram of the laser cutter where A – computer, B – USB communication, C – microcontroller, D – laser driver, E – laser module, F – stepper motor driver, G – stepper motor

minimum, in order to facilitate budgeting in minor- and moderate-scale projects. The laser module (E) generates a 450 nm wavelength blue laser in accordance with the commands received from the laser driver.

The stepper motor (F) receives control signals and a pulse-width modulated signal for operation. One pulse of the pulse train, when transmitted to the stepper motor, drives it one microstep. A simple stepper motor model with a STEP input, such as the NEMA 17 can be used here. The NEMA 17 is ideal, since it enables compact design with high torque, when supplied with 12 V.

Figure 5 shows a possible design of the laser cutter that adheres to the block diagram shown in Figure 3.

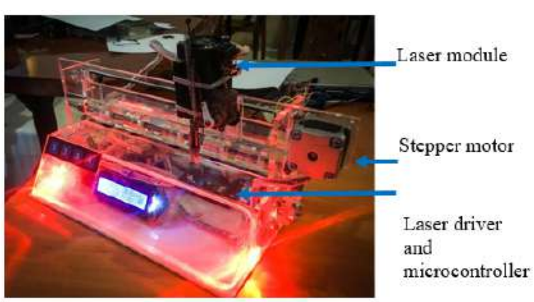

Figure 5: Components of the laser cutter

C. Pulse Width Modulation

Different brands of PVC stickers require varying cutting power and cutting speeds for obtaining a smooth cut. These specifications can be extracted from the sticker's datasheet. The power of the laser module designed above varies between 0 W and 10 W. However, due to a portion of the energy dissipating as heat, the optical output of the laser beam is only 3.5 W. A pulse train, as shown in Figure 6, has to be transmitted to the laser module. The calculation required to compute the pulse width of this signal is presented below.

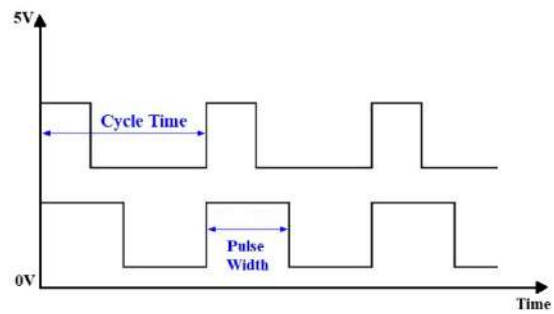

Figure 6: Pulse train to be transmitted to the laser module

The pulse width (PW) of the signal can be calculated from its cycle time (TC) and the duty cycle (DT) as shown in Equation (1).

$$PW = TC * DT \quad (1)$$

TC can be calculated by taking the inverse of the microcontroller output pulse frequency (FC) as in Equation (2).

$$TC = 1 / FC \quad (2)$$

DT can be obtained by dividing the required output power (OP), determined from the datasheet, by the total power (TP) as in Equation (3).

$$DT = OP / TP \quad (3)$$

Equations (1), (2) and (3), can be combined to Equation (4), which shows how PW can be calculated when FC, OP and TP are known.

$$PW = OP / (TP * FC) \quad (4)$$

For example, for a pulse of frequency 490 Hz, if the total power required is 0.5 W and the total power is 3.5 W, the width of the pulse train is.

$$PW = 0.5 / (3.5 * 490) = 0.000291\ s = 0.291\ ms \quad (5)$$

Once the pulse width is known, the microcontroller must be programmed to produce the appropriate pulse train.

3. Results and Discussion

A. *Comparison of PCBs made using the Homemade Method and the Proposed Method*

In this subsection, the conformance of the output PCB to the PCB template drawn using software are discussed. Additionally, the outputs of the homemade method and the proposed method are compared. In order to do the comparison, binary images of a PCB made using the homemade method and a PCB made using the proposed method were obtained by a simple thresholding process. The original images of the PCBs and the results of their binarization are presented in Figure 7.

In Figure 7, (a) is the template of the PCB designed using software and (d) is the same image with inverted colour. Colours are inverted so that the PCB tracks are displayed in white and the rest of the PCB is displayed in black.

(b) and (c) show the outputs of the homemade method and the proposed method respectively. To obtain the binarized images (e) and (f), (b) and (c) are rotated, resized and then subject to a thresholding process. Otsu's method has been used to automatically calculate the optimal threshold value for (b) and (c). It should be noted that the two aforementioned images have been captured under uniform illumination so as to minimize any image artifacts and noise after the thresholding.

When comparing the homemade PCB (b) and the template (a), one can clearly note gaps in the tracks in the former. This non-continuity of the tracks can lead to loss of electrical connection between any components that may later be soldered on to the PCB. The imperfect tracks are caused by several reasons, the first of which is the improper transfer of ink from the glossy paper to the PCB after ironing. Figure 8 shows how a considerable amount of ink is still left on the glossy paper, which remains a major drawback in the homemade method. Printing the circuit on the glossy paper at a higher resolution will mitigate, but not completely eliminate, this issue.

The second reason for poor tracks in the output, is the quality of the etchant solution. Etchant solutions, especially ferric chloride, stored for a long time may undergo a chemical reaction with oxygen, which consequently lowers the concentration. Overly dilute solutions lead to incomplete dissolution of the copper plate, resulting in sketchy PCB tracks. This remains a problem in both the homemade method and the proposed method.

When comparing the PCB made using the proposed method (c) and the template (a), one can observe that the output conforms with the template to a greater degree than (b). A majority of the PCB tracks are continuous, with the exception of a few tracks on the right edge, which are caused by imperfections in the etchant solution.

To determine the similarity of a PCB (either (b) or (c)) to the template (a), the images, (b) and (c), are first cropped so that only the circuit area is visible. Then, the images are rotated and scaled so that they match the orientation and size of the template image. Afterwards, the images are binarized using a threshold computed by Otsu's method, after which the logical AND of the binary images are obtained. This results in matching pixels appearing in white and non-matching pixels appearing in black. Subsequently, the number of white pixels are counted and expressed as a fraction of the white pixels in the template image. Hence, if the two images match perfectly, a score of 1.0 will be obtained. This calculation provides a method to compute a similarity score between a PCB and the template.

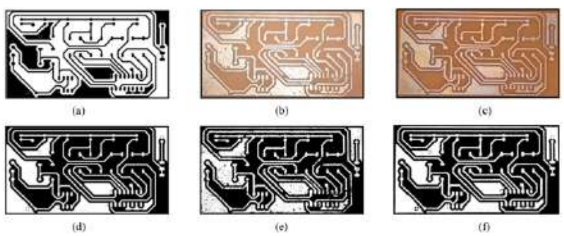

Figure 7: Images of (a) Template, (b) PCB made using the homemade method, (c) PCB made using the proposed method, (d) Template with colours inverted, (e) Binary image obtained from b, (f) Binary image obtained from c.

Comparing the binary image of the homemade PCB (e) to that of the template (a) results in a similarity score of 0.6334. A comparison of the binary image made using the proposed method (f) with that of the template results in a similarity score of 0.9265. The higher similarity score for the latter further reinforces the fact that the proposed method results in PCBs of higher quality that conform with the template, when compared with PCBs manufactured with the homemade method.

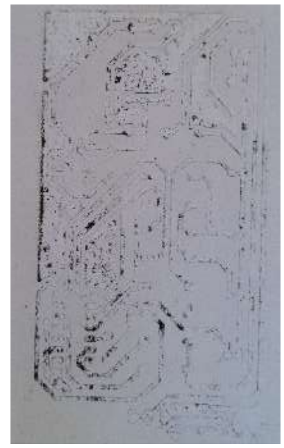

Figure 8: Ink remaining on the glossy paper after ironing, when making a PCB with the homemade method

A. Variation of Laser Power, Cut Rate and Time

In the proposed method, once the vinyl sticker is pasted on the copper plate, the tracks must be cut using the laser cutter. The time required to cut the tracks at different cut rates and power of the laser cutter were measured and are presented in Table 1. The sticker used is a PVC waterproof sticker of brand Oracal 651.

Table 1. Time taken to cut PCB tracks at different cut rates and power of the laser

Power (W)	Cut rate (mm/s)	Time to Cut (s)
1.65	5	480
1.95	20	448
2.25	35	416
2.4	50	385

Since the laser is low-power (between 1.65 W – 2.4 W), the risk to the operator's safety is minimal. However, laser safety goggles are recommended when working near a laser beam of any power. Apart from the lower risk to the user, the potential for the PVC sticker to be damaged by excess heat is also low (Owen 2001).

The results of the laser-cut PVC sticker pasted on the copper plate at each of the power/cut-rate combinations given in Table 1 are shown in Figure 9. It can be observed that cutting at the lowest power and cut rate causes broken lines to be cut on the sticker (a). These are caused by low power in the laser due to heat dissipation, coupled with less time spent on the cutting area. On the other hand, using higher power/cut-rate combinations has caused all lines to be cut continuously, with no gaps in the lines as seen from a), (b) and (c). No significant difference can be seen among these three images.

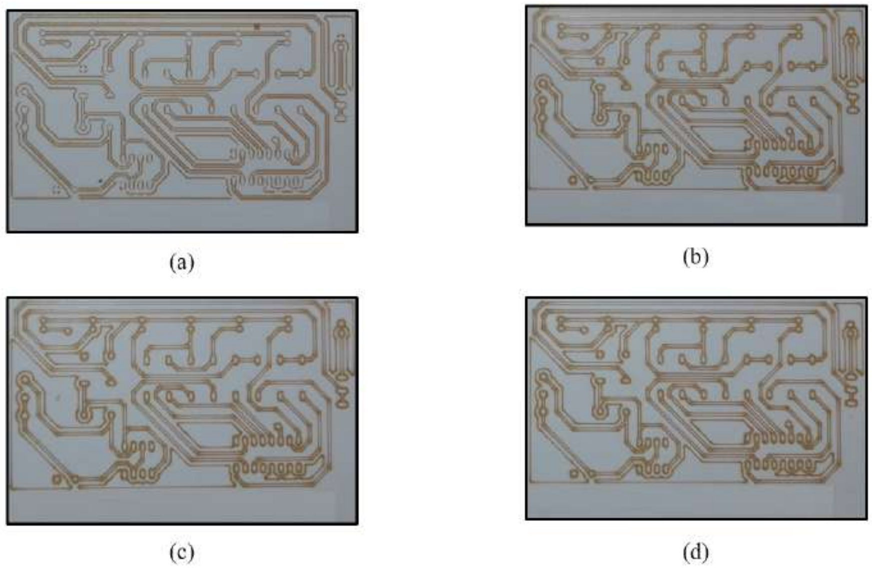

Figure 9: PVC sticker pasted on the copper plate after cutting by laser (a) at 1.65 W and 5 mm/s, (b) at 1.95 W and 20 mm/s, (c) at 2.25 W and 35 mm/s, and (d) at 2.4 W and 50 mm/s

The proposed approach is less time-consuming when compared to the typical method that electronic enthusiasts use, even when the laser is operated at its lowest power and lowest cut rate i.e., 1.65 W and 5 mm/s. One major reason for this is that the tracks don't have to be printed on the paper and pasted on the copper plate. Printing on glossy paper has another drawback, which is the possibility of the printed lines being erased dues to frictional forces. This is entirely eliminated in the proposed method. When the two processes were timed, the method used by electronic enthusiasts took an average of 44.0 minutes to complete, whereas the proposed approach took an average of 16.0 minutes. 5 PCBs each were fabricated by each approach in order to get these results. The maximum speed and power of the laser has been used when timing the proposed approach i.e., 50 mm/s and 2.4 W, respectively. It should be noted that the copper plate has to be immersed in etchant solution in both methods, which took an average of 10.5 minutes for each method.

Assuming the optimum laser power as 2W, obtained similarity scores for different cutting rates.

Table 2. Similarity score of PCB tracks at different cut rates while keeping the power at a constant value

Power (W)	Cut rate (mm/s)	Similarity Score
2	5	0.6884
2	20	0.9265
2	35	0.8638
2	50	0.7965

According to the data obtained, similarity score behaviour shows in Figure 10. It can be observed that low cutting rates causes broken lines to be cut due to heat dissipation and higher cutting rates also causes broken lines to be cut due to Inadequate heat supply to the board.

Based on Table 2, a function of similarity score is interpolated as

$$f(x) = 0.00001462716049x^3 - 0.001546074074x^2 + 0.04684592593x + 0.4909938272 \quad (6)$$

where the variable x is cutting rate. There it uses Lagrange interpolation and graph of f(x) is given in Figure 10. The function f(x) has a local maximum and minimum respectively at x = 22.04943411 and x = 48.41646733, on the range $0 \leq x \leq 60$. Equation (6) can be used to obtain the highest similarity score.

Although industrial manufacturing methods such as CNC milling are capable of mass-producing PCBs at a fast rate, there is a risk of damaging the copper plate since it is a subtractive process. Another drawback of CNC milling is that fine tracks cannot be obtained due to the diameter of the milling tool (Bogatin 2021), (Shinde & Mutthurajan 2016). Laser cutting by directly applying heat to the copper plate also has the same drawback due to the width of the beam. One past study (Durillo, Falcón & Aranda 2018) suggests painting the copper plate with black synthetic spray paint and then drying before applying the laser beam to the plate to minimize any damage. However, this overcomplicates the process and slows it down.

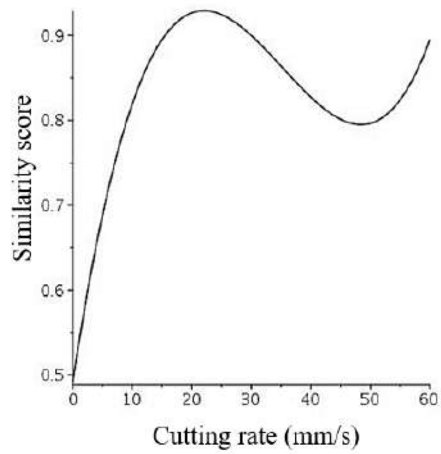

Figure 10: Similarity score and Cutting Rate

One research on convenient PCB fabrication (Alex & Sawn 2013) proposes the use of an ultraviolet (UV) laser on UV-sensitive copper clad boards. The PCB printer developed in the aforementioned study is very similar to the approach proposed in this research, except that the copper board still has to be immersed in, not one, but two solutions (developer solution and etchant solution) in order to get the finished PCB. Another research (Sooraj & Vasa 2020) proposes a cutting method with nanosecond laser ablation, but makes use of Laser Induced Breakdown Spectroscopy. This trades off speed and convenience for extreme accuracy, which is not needed for most small- to moderate- scale PCB-based projects, in general.

The drawbacks of the proposed approach is that it has the possibility of inflicting thermal damage on the sticker pasted on the copper plate, if the laser has unstable power. It is noteworthy that this may also happen in industrial laser cutting methods. Burnt stickers lead to smaller tracks in the PCB. Although finer tracks are preferable in most circuits, those transmitting large or high-frequency current may be disadvantaged by wider tracks. The disadvantage occurs in the form of energy loss in the form of heat, which may warp the copper plate, if it is considerably thin (Yang et al. 2019), (Chung et al. 2013). High heat may even melt the solder attaching electronic components to the PCB if the temperature rises above a critical value (Anand, Singh & Ladwal 2019). Secondly, since a pulse train is used to signal the microcontroller to generate the laser, step-skipping may occur at high pulse frequencies.

4. Conclusion

Existing methods of fabricating PCBs take up a significant amount of time, especially for short-term projects. Additionally, they carry other drawbacks such as improper electrical connections in the finished PCB or the risk of damaging the copper plate while it is being processed.

The approach proposed in this paper provides a better trade-off between speed of fabrication and accuracy of the finished PCB than existing techniques. This method is expected to contribute to accelerating prototyping in projects of minor or moderate scale. In this research, we succeeded in making a PCB of much higher quality than a PCB made using the homemade method in considerably less time.

References

Triano, A & Collins, S (2013): 'Development of a PCB printer using an ultraviolet laser diode', 2013 IEEE Long Island Systems, Applications and Technology Conference (LISAT), p. 1-6.

Anand, V; Singh, V & Ladwal, V K (2019): 'Study on PCB Designing Problems and their Solutions', 2019 International Conference on Power Electronics, Control and Automation (ICPECA), pp. 1-5.

Basniak, R & Catapan, M F (2012): 'Design of a PCB milling machine', ACBM Symposium Series in Mechatronics, pp. 1339-1348.

Bogtin, E (2021): Bogatin's Practical Guide to PCB Design for New Product Development, Artech.

Choudhary, R; Sambhav; Titus, S D; Akshaya, P; Mathew, J A & Balaji, N (2017): 'CNC PCB milling and wood engraving machine', 2017 International Conference On Smart Technologies For Smart Nation (SmartTechCon), pp. 1301-1306.

Chung, S, Heo, G, Kwak, J, Oh, S, Lee, Y & Kang, C (2013): 'Development of PCB design guide and PCB deformation simulation tool for slim PCB quality and reliability', 2013 IEEE 63rd Electronic Components and Technology Conference, pp. 2157-2162.

Durillo, J C M; Falcón, P M & Aranda, A R (2018): 'Low-cost Laser Cutter and PCB exposure', 2018 XIII Technologies Applied to Electronics Teaching Conference (TAEE), pp. 484-493.

Hansotte, E J; Canigan, E C & Meisburger, D (2011): 'High speed maskless lithography of printed circuit boards using digital micromirrors', Proceedings of SPIE - The International Society for Optical Engineering.

Lee, D -H (2010): Optical System with 4 μm Resolution for Maskless Lithography Using Digital Micromirror Device, Journal of the Optical Society of Korea, 14(3) pp. 266-276.

Madekar, K J; Nanaware, K R; Phadtare, P R & Mane, V S (2016): Automatic mini CNC machine for PCB drawing and drilling, International Research Journal of Engineering and Technology (IRJET), 03, pp. 1106-1110.

Mitzner, K (2007): Complete PCB Design Using OrCAD® Capture and Layout, Newnes Publications.

Owen, M D (2001): Electronic Packaging: Electrical Interconnects, LIA Handbook of Laser Materials Processing, First Edition, Magnolia Publishing Inc.

Plesha, C E & Kycia, J B (2020): Resonant Frequency Adjusting of an MSA via Laser Ablation of the Input Coil, IEEE Transactions on Applied Superconductivity, 30(8), pp. 1-4.

Shinde, S & Muthurajan, H (2016): 'PCB fabrication unit for electronics circuit prototyping', 2016 IEEE International Conference on Advances in Electronics, Communication and Computer Technology (ICAECCT), pp. 1-5.

Sooraj, S & Vasa, N J (2020): 'Pulsed laser assisted micro-scribing of PCB combined with LIBS based depth monitoring', 2020 Conference on Lasers and Electro-Optics Pacific Rim (CLEO-PR), pp. 1-2.

Yang, D; Cheng, Z; Li, H; Won, S; Zhou, B & Tian, J (2019); PCB Layout Optimization of High-Frequency Inverter for Magnetic Coupled Resonance Wireless Power Transfer System, IEEE Access, 7, pp. 171395-171404.

Author Biography

DALP Karunarathne BSc. (Hons) in Mechatronics Engineering graduated from the Department of Mechanical Engineering of General Sir John Kotelawala Defence University. Sri Lanka. He is currently the Managing Director of his own company, which specializes in industrial automation. He is also an independent researcher.

WSP Fernando BSc. (Hons) graduated from the Department of Computer Science and Engineering of University of Moratuwa, Sri Lanka and specialized in the field of control systems. He has also completed an MPhil from the University of Moratuwa. He is currently a Senior Lecturer at General Sir John Kotelawala Defence University.

PSH Pallemulla BSc. (Hons) in Mechatronics Engineering graduated from the Department of Mechanical Engineering of General Sir John Kotelawala Defence University in Sri Lanka in 2016. He is currently a Lecturer (Probationary) at the same university and is reading an MPhil at the University of Moratuwa.

MKAJ Maldeniya BSc. in Mathematics, graduated from the University of Jayawardenepura, Sri Lanka and completed an MS in Mathematics at the New Mexico State University. He is currently a Senior Lecturer at the Department of Mathematics at General Sir John Kotelawala Defence University, Sri Lanka.