Simplified Procedure for Drawing of Artwork of Prototype PCB

Pravin Bhadane[1*,] Aparna Lal[2]

[1,2]Department of Electronics, Nowrosjee Wadia College, Pune, India-411001

Corresponding Author: pravin16163@gmail.com, Tel.:9823855341

Abstract: Work on simplifying the drawing of Artwork required for making of prototype PCB has been carried out and reported in the present article. The students of electronics find PCB Artwork making procedure tedious and time consuming. To overcome this problem, we have presented here the simplified as well as motivating procedure. Work started with a selection of circuit diagram that contains various types of components and connectors. After that the rough layout is planned with consideration of dimensions of components. Various steps in drawing of Artwork like placement of footprint of components, making netlist as per the circuit diagram, manual routing, etc. are discussed with example. Utilisation of various facilities provided by computer aided PCB layout making softwares are also discussed in the present work. Procedure of making of postscript file and its printout using postscript editor like corel draw is also discussed. Present work is helpful to the hobbyist and students of electronics for making of their project work.

Keywords: PCB, Artwork, footprint, layout.

I.   INTRODUCTION

Printed circuit board (PCB) is vital part of any electronic instrument. It is used for connecting the discrete components to each other with the help of copper tracks. The electronic components are soldered to the copper pads of PCB. PCB is a copper clad of some insulating material on which a thin layer of copper is deposited on one side or both sides. Layout of PCB is also called an Artwork which is the exact copy of connection diagram which is to be traced on the copper clad. In old days when there were no computers, the artwork was prepared manually with the help of tools of draftsman. It was done by PCB professionals on a special drawing paper in 2X dimensions [1]. Nowadays, Artwork is prepared on computer using PCB layout software. The comparison between manual and computerised artwork is given in table 1.

In the present work, we have simplified the PCB Artwork making procedure and discussed in this article. Section II contains discussion on Manual procedure, section III contains computerised procedure of layout, section IV contains the experimental work and section V contains the important conclusions and future scope.

II.   MANUAL PROCEDURE

If computer and PCB making software are not available then the artwork can be prepared manually by use of following steps.

Table 1: Comparison between manual and computerised artwork.

Manual artwork	Computerised artwork
Manual procedure	Computerised procedure
Time consuming	Relatively needs less time
Correction is difficult	Correction is easy
Expensive	Inexpensive
Double size	Same size
Need of mirror image	No need of mirror image
Prepared from bottom side	Prepared from top side
Double sided needs more skill	Double sided is easier

1) Read dimensions of components from datasheet.
2) Prepare rough layout.
3) Keep minimum jumpers or cross over.
4) Take mirror image of transistors and ICs.
5) Draw artwork of above layout on graph paper.
6) Trace the artwork on copper side of clad with the help of carbon paper.
7) Paint the copied artwork by some enamel or oil paint and dry it for about six hours.
8) Remove the non coated copper from the clad by etching method.
9) Etching is done by placing the copper clad into the 20% dilute solution of ferric chloride.
10) Wash the copper clad and remove the remaining part of paint with the help of acetone.

11) Make the holes of required size using drilling machine.
12) Place the components from the top side and solder them with the help of 25W solder gun.

III.   COMPUTERISED PROCEDURE

After the availability of computers many manual processes are automated [2]. Manual PCB design is also automated by use of computer softwares. If computer layout software is available then artwork can be prepared on computer. There are many PCB layout softwares which are either freely available or can be purchased online. Some of the popular softwares in the students community are Express PCB, Eagle, Protease, Deeptrace, Orcad Light [3], etc. Evaluation version of some of them can be obtained from the developer's website as well as from website of hobbyist. Evaluation versions are necessary for the benchmarking of performance of softwares [4]. In the present work we have used Orcad Light and the simplified procedure [5] of using it is given below.

1.   Execute the layout program by clicking on its icon.
2.   Create new template file by selecting File ➡ New ➡ cancel option
3.   Change the workspace setting as per requirement using CTRl+G. (unit : inch or mm)
4.   Move the datum at suitable position in workspace using Tool ➡Dimension ➡Move Datum.
5.   Draw the board outline with the help of Obstacle tool. E.g. lengthXwidth = 129X88 mm.
6.   Select the required layers from the spread sheet e.g. top and bottom layer.
7.   Save the file with .max extension.
8.   Place the footprints of components from library.
9.   If footprint is not available then create it
10. Make the connections as per diagram (netlist)
11. Rout the tracks with appropriate track width (artwork)
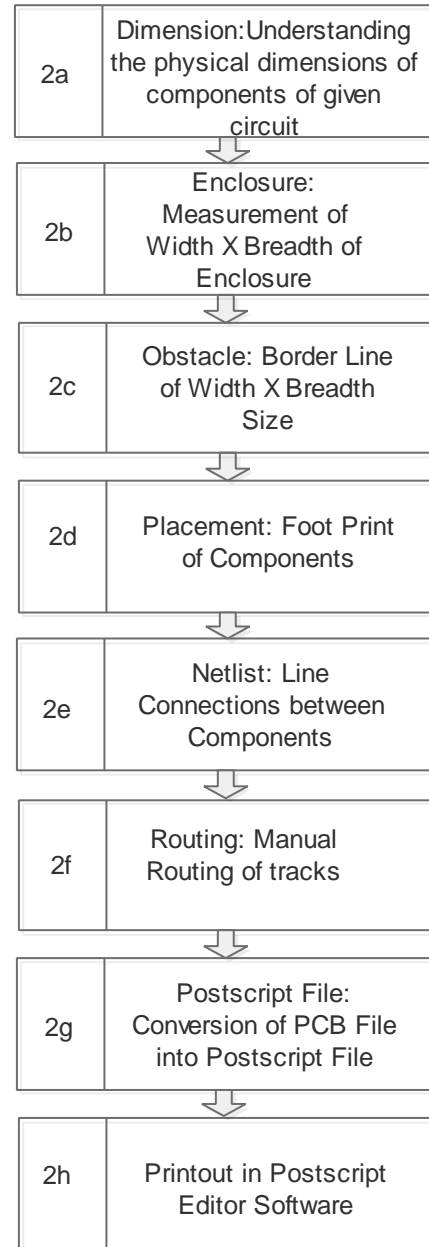12. Take the printout or create the gerber file.

Figure 1 : Flow diagram of sequence of simplified steps for drawing of Artwork

IV.   EXPERIMENTAL WORK

Experimental work is carried out by applying above computational procedure for a given prototype circuit diagram. Figure 1 shows the self explanatory flow diagram for making of an artwork. The numbers shown in it are indicating the successive steps which are also given to the actual outcome of computational PCB drawing software. Figure 2a to 2h shows the result of sequence of operations carried out during making of artwork. After making of Artwork, PCB can be prepared either in the laboratory [6]

or by screen printing technology. Some steps of PCB manufacturing process are shown in figure 3.
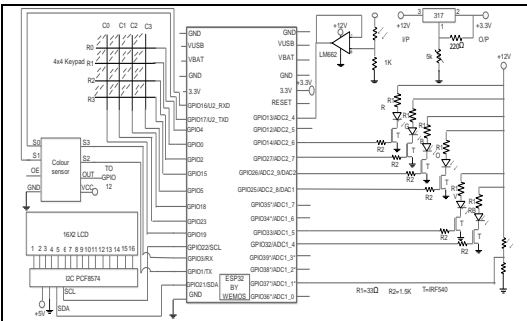

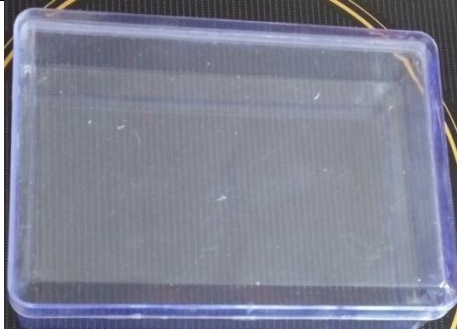Figure 2a: Dimensions of components

Figure 2b: Physical dimension of enclosure

Figure 2c: Obstacle layer
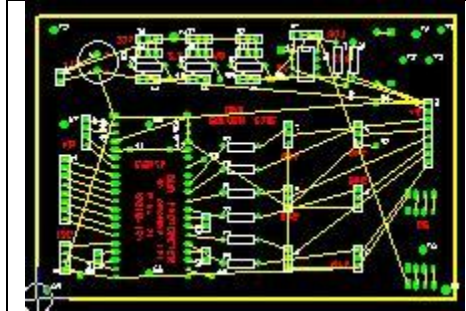

Figure 2d: Placement of footprint
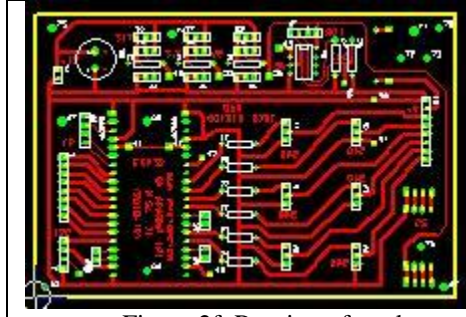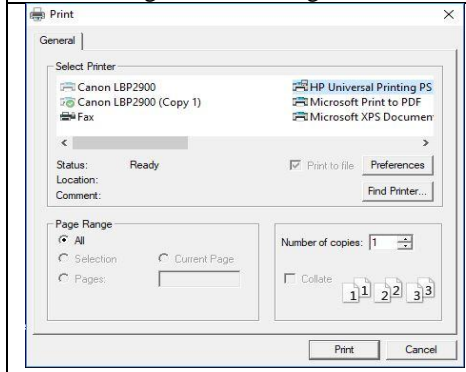

Figure 2e: Netlist
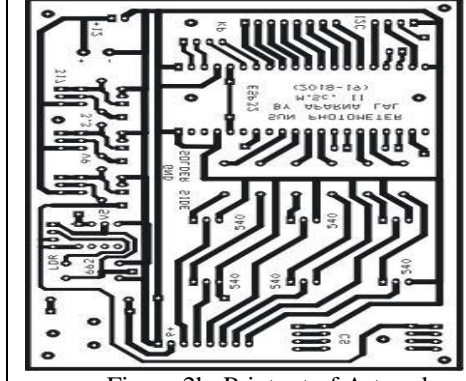

Figure 2f: Routing of tracks

Figure 2g:Creation of postscript file

Figure 2h: Printout of Artwork

Figure 2: Outcome of PCB design software corresponding to the steps given in figure 1.

Copying of negative film of artwork on the copper side of copper clad

↓

Painting of copied artwork by some etch resistive adhesive paint

↓

Drying of paint for about six hours

↓

Removing the not required copper by etching it with dilute $FeCl_3$ and some catalyst like concentrated HCl

↓

Removing residual paint by acetone

↓

Drilling of holes for passing the leads of components

↓

Tinning by electroplating process
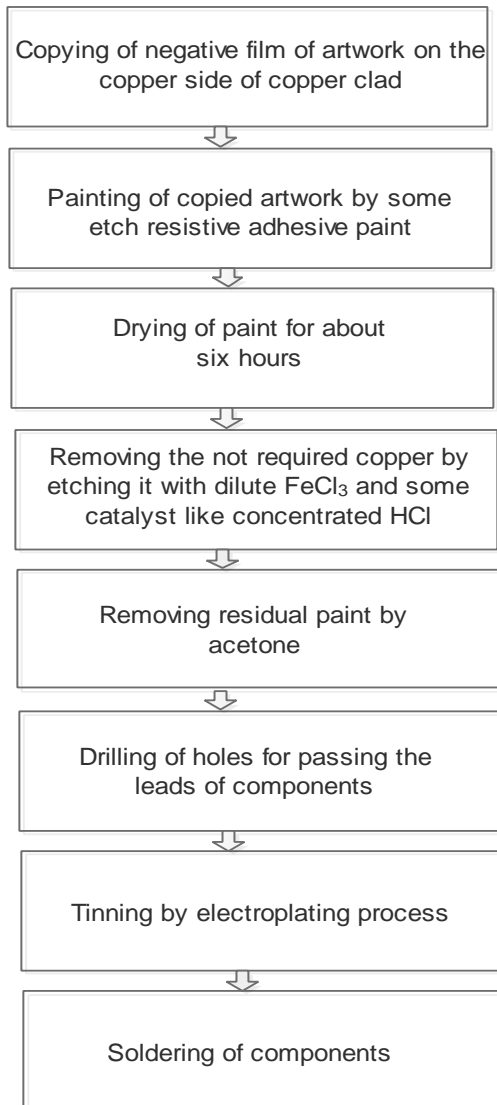
↓

Soldering of components

Figure 3: Steps of PCB manufacturing process

V. CONCLUSIONS AND FUTURE SCOPE

PCB making is a laborious mechanical procedure which involves photography for negative film, etch resistive adhesive coating, chemical etching, electroplating, drilling etc. Such work is generally done in PCB manufacturing

unit which is located in some specific chemical industrial zone. PCB manufacturers require the Artwork of circuit which is prepared by use of PCB design software. Drawing of Artwork is a tedious process and beginners find it difficult to learn. In the present article, simplified procedure of drawing of Artwork of circuit required for the prototype work is presented with suitable example. The work is helpful to the students of electronics for making of their final year project.

Future scope: The work can be extended by incorporating the tools like schematic editor, custom design of footprint and making of gerber file.

REFERENCES

[1]. Walter C. Bosshart, "Printed Circuit Board Design and Technology",1st edn., McGraw Hill Education, (1984).
[2]. R. Sharma, S. Singh, **"**Swarm Intelligence Based Automated Testing for MTAAS", International Journal of Computer Sciences and Engineering, Vol.6, Issue.8, pp.475-477, 2018
[3]. Kraig Mitzner, "Complete PCB Design Using OrCAD Capture and Layout", Elsevier, Armsterdam. (2007).
[4]. Mehzabeen Kaur, Surender Jangra," Execution Time of Quick Sort on Different C Compilers: A Benchmark", International Journal of Computer Sciences and Engineering, Vol.6, Issue.9, pp.786-788, 2018
[5]. Orcad layout user's guide, Orcad Inc. Oregon, (1998).
[6]. Pravin Bhadane, Suchita Bhangale and Aparna Lal, 'Understanding the parts of PCB Layout', International Journal of Electrical and Electronics Research, Vol.6, Issue 3, pp.1-3, (2018).

Authors Profile

Pravin K. Bhadane is Head of the Electronics Department at Nowrosjee Wadia College, Pune (India). His area of research is in the field of electronics.

Aparna Lal is post graduate student of Electronics at Nowrosjee Wadia College, Pune (India).