

# Problems of Manual PCB Designing And Solutions

Laxmikant S. Shete<sup>1</sup>, Gaurav V. Navghare<sup>2</sup>, Rupesh G. Waghale<sup>3</sup>,  
D.V.Rojatkar<sup>4</sup>

<sup>1234</sup>Dept. of Electronics and Tele. Govt. College of Engg., Chandrapur, M.S., India

## Abstract

Printed Circuit Board in short also known as PCB is one of the milestone in Electronics industry and Engineering Technology. Before the advent of PCB, the electronic circuit was running with lot of connecting wires. But that arrangement was too much bulky as well as complex. PCB techniques are useful to solve the problems which occurred in old electronic wiring systems. In general the PCB can be described as 'The plastic board which has a particular copper layer pattern and an arrangement for electronic components placing at required position to perform Electronic circuit task'. Before the invention of Digital Computer, PCB designing process was a total manual process. After the development of Computer technology, various software and automatic PCB manufacturing machines were invented by scientists as well as softwares like PROTEUS, EXPRESS PCB, EAGLE, etc. by programmer. These processes are very useful to design a very complex PCB but the set-up of these advanced PCB design is very well costlier and not easily available. There are also some countries which do not have this technology because of some political and economic issues. Maximum PCB Designers design the PCB Manually. In this study, the process of Manual PCB designing, problems arising and their solutions are given on basis of experiments performed.

**Keywords:** Printed Circuit Board, PCB Layout, PCB etching, PCB soldering, etc.

## 1. Introduction

The manual process of PCB Design is very slow as compared to Automatic PCB design with software and machine. The Machine Designing of PCB circuit is very reliable, highly accurate and requires less place as compared to Manual Design technique. While designing PCB manually, the artwork is designed by human so human errors could occur. As we know the human errors cannot be minimized completely but can be reduced using some processes as mentioned in this study. In manual designing, the simple technique is before doing any process think about all the possibilities that can occur which is very important.

## 2. PCB Design Process

The Manual PCB Design process consist of following steps:-

1) Layout Design: - The layout design is the first process in PCB design. The layout is also known as artwork. The artwork is design using the circuit diagram and physical dimension of electronic components. The designer have a clear vision in his mind about the position and task of each component of circuit. The manual layout is not so simple and takes too much time. Then in the next step mirror image of layout is marked on copper board.

2) Layout marking: the mirror image of layout is then marked on a copper board. The copper board is a plastic board which has a thin layer of copper deposited on single or double side. A multilayer copper board is also available in market. The artwork is marked on copper board by using permanent marker pen or UV protection chemical.

3) Etching of Copper Board: After the marking process, the etching process is done with etching solution. The etching solution reacts with copper layer which is not covered by UV protective layer. Generally 'Ferrous Chloride' is most commonly used and easily available etching solution in the market.

4) Cover removing process of artwork: In cover removing process the portion of copper which is covered by a UV protecting layer or permanent ink is removed using chemical solution. The solution used is generally acetone which reacts with ink at a fast rate.

5) Drilling and soldering: finally the PCB is ready for drilling. The drilling is also important process because wrong drilling damages the whole PCB. The drill bits are available in various ranges in market. The normally used drill bit is 0.5 to 1.22 mm diameter. Final stage is soldering of components on the designed PCB. [1, 2]

## 3. Defects Due to Mistakes in PCB and Problems:

The following mistakes are generally done by designer:

- 1) The track width is inappropriate.
- 2) The component position shape is not proper in PCB layout.

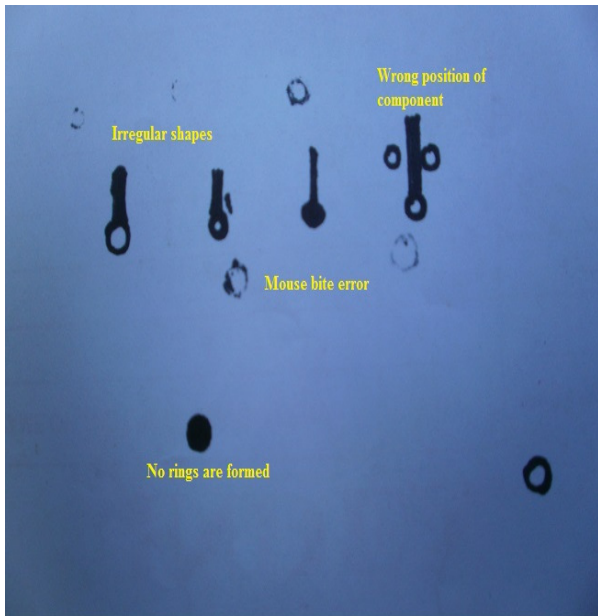


Fig.2:- Component position shapes errors in layout

- 3) The etching solution is stronger than required for etching process.
- 4) Wrong drilling bit range and drilling method.
- 5) Defected soldering method and flux problem.
- 6) Mechanical stability problems in PCB design

Sometimes the track width is major issue in PCB manufacturing. Normally width of track on PCB is greater than component terminal width. If this mistake occurs then the problem with soldering happens. The width of the drilled hole at which a particular component is to be soldered should have slightly greater width as compared to the width of the thread of that particular component. If width of the hole is inappropriate it causes problems in PCB designing. This mistake happens very frequently in manual PCB design. The etching solution is Ferrous Chloride which is very important content in this whole process. The concentration of etching solution is very well responsible for a good PCB designing. When the concentration is high then it proves dangerous to covered portion on copper board. The highly concentrated solution removes not only the covered copper portion but also the marking ink. As ink is removed during etching process the track of PCB layout gets disturbed and non-continuous causing open circuit problems. The drilling bit is another issue in this process as the small width drill bit is not easily available in the market. Drilling process looks very simple

but actually it is not. The width of drilling affects the continuity problem in tracks. Also drilling at wrong place also causes some major problems in the circuit. The soldering method of designer is sometimes wrong. The automatic soldering machine solders with an accurate amount of lead metal. Due to wrong soldering, the electrical contact between the copper track and electronic components becomes weak because of which current leakage and open circuit problems arise. The mechanical stability of electronic components is not good which also affects the circuit operations. [3,4]

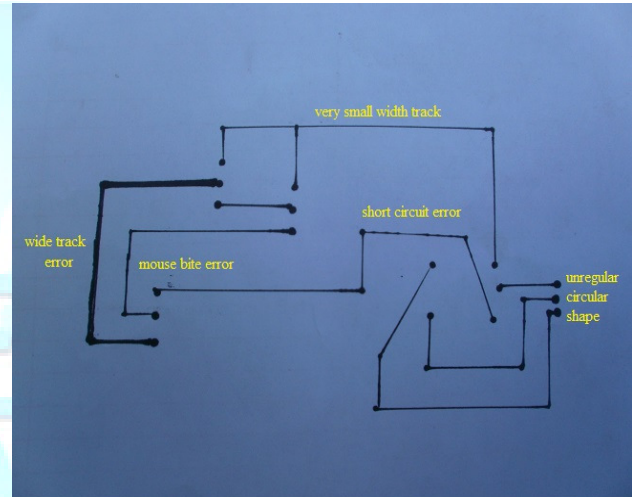


Fig.1:- Manual Layout Designing-with errors

#### 4. Solutions

There are various online automatic software based defect detection systems available. Some of these systems depend on image subtraction method [5, 6] .But we can also detect the defects manually in our PCB layout in the following manner.

- 1) Before the designing of layout, firstly determine the maximum terminal width of all components. For example, a circuit contains a 1.4mm diameter of relay so, design the track with the suitable range for that component needs to be adjusted completely.
- 2) The component position shape is a circular form as shown in fig.3. The shape looks like two concentric small circles . This arrangement gives very good mechanical stability and electrical contact.
- 3) The etching solution Ferrous Chloride is used for removing the uncovered copper on PCB board. Using simple techniques of chemistry we can increase or decrease the concentration of solution. First try the solution on a dummy copper board and note down the reaction time. In the sun light the reaction is very fast. Check the covered portion during the etching process. If the covering material

is slightly removed then firstly dry the board and mark on the removed portion and then reduce the concentration of solution.

4) The drilling process is done by selecting appropriate drill. While drilling, the drill machine is placed exactly perpendicular on the PCB so that no extra portion is removed. After drilling the circular copper shape must be present around hole. Due to this we get the better electrical contact and mechanical stability.

5) If the soldering is not proper then open circuit and close circuit problems are occurred. To avoid this problem, choose a good quality flux and sharp nip soldering gun.

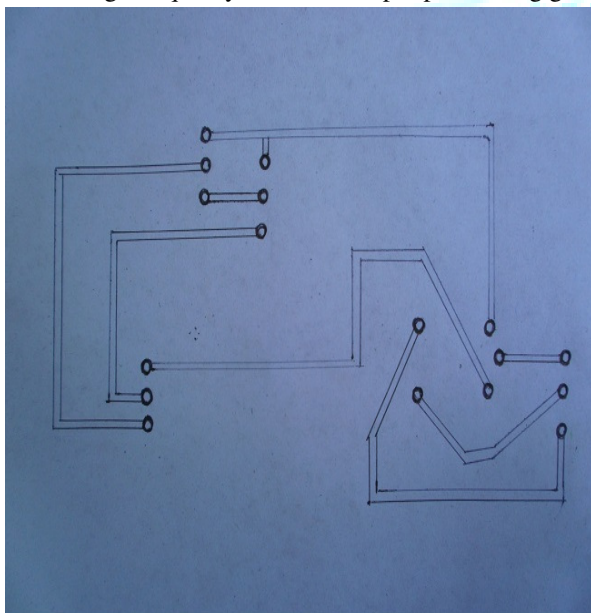


Fig.3: Improved version of PCB layout-with maximum possible minimised errors

## 5. Discussion and Conclusion

Now the question arises that why anyone will be using the manual manufactured PCBs instead of the advanced machine made PCBs which are readily available in the market. Actually machine made PCBs are manufactured by insulating material and so we cannot change any components if we want. The soldering on machine PCB after complete manufacturing is so hard that it is almost impossible to make any changes. So the manual PCB is required. Also every user does not have the machine set up and knowledge of advanced PCB design. Hence manual PCB has much more importance. As mentioned above, the process of PCB designing and simple techniques of avoiding the defects in PCB manufacturing are proposed. After the experiment it was observed that the error on manual PCB board was reduced as we required. The extra

care is needed to design the manual PCB and it is too much efficient as per our want.

## References

- [1] "Building a Printed Circuit Board", Advanced Circuit Inc.
- [2] Indrani Bose,"HOW TO MAKE GOOD HOMEMADE PCBs?" ELECTRONICS FOR YOU ,NOVEMBER 2009.
- [3] "Guide To Designing And Fabricating Printed Circuit Boards",UNIVERSITY OF TORONTO, JANUARY 2006.
- [4] Research International, Solder Reflow Technology Handbook.
- [5] Amit H.Choksi,"Printed Circuit Board Defect Detection Using Wavelet Transform", International Journal of Current Engineering and Technology, Vol -4, No-4(Aug-2014).
- [6] Syamsiah Mashohor," Automatic Hybrid Genetic Algorithm Based Printed Circuit Board Inspection", Proceedings of the First NASA/ESA Conference on Adaptive Hardware and Systems (AHS'06), IEEE COPUTER SOCIETY.
- [7] Ismail Ibrahim," AN IMPROVED DEFECT CLASSIFICATION ALGORITHM FOR SIX PRINTING DEFECTS AND ITS IMPLEMENTATION ON REAL PRINTED CIRCUIT BOARD IMAGES", International Journal of Innovative Computing, Information and Control, Volume 8, Number 5(A), May 2012.
- [8] S.Sridevi," Online Inspection of Printed Circuit Board Using Machine Vision", International Journal of Innovative Research in Science, Engineering and Technology Volume 3, Special Issue 3, March 2014.
- [9] Ms. Prachi P. Londe,"A REVIEW ON AUTOMATIC PCB DEFECTS DETECTION AND CLASSIFICATION", International Journal of Emerging Trends in Engineering and Development, Issue 4, Vol.1 (January 2014).
- [10] Jeremy Dion," Fast Printed Circuit Board Routing", DIGITAL Western Research Laboratory 250 University Avenue Palo Alto, California 94301 USA.