GENERATION OF PRINTED CIRCUIT BOARD LAYOUT AND ARTWORK

Debretsion G. Michael, G. Devarajan
Department of Electrical Engineering
Addis Ababa University

ABSTRACT

Printed circuit board (PCB) design plays an important role on equipment performance and reliability. Constraint imposed on PCB size under severe environmental condition is focused. Guidelines for layout planning, its speedy execution, component placements, and future growth/modifications have been addressed. The importance of PCB documentation to assist in on-line production is dealt. Black taping artwork for multilayer PCBs have been discussed. An overall guideline to produce PCBs which will enhance equipment reliability, serviceability by taking into all relevant factors such as conductor width/length, pads plated through holes, electronics and mechanical components, component density, cost etc. have been discussed in detail.

INTRODUCTION

In any electronic equipment fabrication the printed circuit boards (PCBs) plays an important role in terms of equipment quality and reliability. It is the properly designed PCBs that determine the basic limiting factors of the equipment such as: noise immunity, pulse rise time, high frequency handling low-level signals etc. Cost and reliability of the equipment determine the PCB fabrication process where it is being utilized. This implies the PCBs are aimed for highly reliable, professional quality with low investment cost.

The PCB design is considered as the first step towards production of electronic equipment or a gadget. It is also the first step towards an electronic circuit performance and reliability. The PCBs design also decides equipment size, weight and serviceability. The electronic engineer's job in PCB design is not a simple one, as he/she needs to know all the implications of particular equipment, which one can acquire by way of experience. These experiences form the basis for guidelines in similar circuits as well as equipment's [1,3,4]. In designing a PCB, the first step towards it is preparation of layout. The layout design brings out both engineering creativity and practical skill during artwork preparation that will enhance the equipment performance.

In Section II of this paper layout planning is dealt in detail outlining layout scale, PCB sizes, lay out approaches, and documentation. In section III, a detailed discussion of PCB artwork is being carried out with special emphasis on the general design rules for interconnections for various components and how to minimize the parasitic effects connected with such inter-connection. In section IV, the cooling requirements and package density is dealt in details.

LAYOUT PLANNING

The production engineer has to collect all the relevant information of the electronic circuits as well as end product for which one need to go for artwork preparation. With regard to the circuits the relevant details include (i) input signals (ii) output signals (iii) critical portion of the circuits (iv) component details that includes size, configuration, power dissipation, power supply etc. These details of the circuits are essential in order to reduce the modifications during layout preparation. The layout designer must be well informed about the mechanical constraint of the final product such as mounting details, test point details for trouble shooting, connector types, etc.

Layout preparation:

Layout precedes PCB artwork preparation: First the components layout has to be decided. As the production engineer has already collected the circuit diagram and associated details, the layout preparation may be proceeded segmenting the circuit diagrams into smaller functional blocks such as: Oscillators, logic circuits, amplifiers, modulators, etc. Here care should be taken to place the bulky components such as electrolytic capacitors and transformers keeping in mind the bending movement, resonance frequency and other related possible environmental hazards on the PCBs. The layout scale may be the same as

artwork scale. The scale is a function of the required mounting accuracy for various components on the PCBs. It may vary from 1:1 to 4:1. The production engineer must also be familiar with the production facility available to him/her, that is, the reprographic camera compatibility for film master production. This is likely to restrict the scale for layout and hence artwork preparation.

PCB board types are single sided, double sided and multilayer. The board type is a function of cost, application, and size limitations. Single sided PCBs utilize jumper wires to make connections on either side. Double side and multilayer PCBs utilize plated through hole (PTH) facility to make the connection in more than one side where circuit density and complexity is high. In this case one has to take precaution to keep minimum number of PTHs for reasons of reliability, mechanical strength and economy. The number of solder joints must be kept minimum on the component side. This will ease out the component replacement and trouble shooting in the PCBs at a later stage.

The size of the PCBs and its thickness plays an important role in mechanical strength after assembling with the electronic components: care should be taken to mount the components like transformer and electrolytic capacitors with necessary clamps on the PCBs. Mechanical problem starts when the equipment undergoes vibration, shock, bump and acceleration, etc. This situation may arise in shipborne and airborne equipment's as well as when transported on rough terrain's. During this time, failures may arise in the form of short-circuiting with nearby conductive elements, cracks in the conductors as well as solder joints, deflection in the PCBs leading to loose contact with connectors' etc. To avoid these problems it is advisable to select the PCB size and thickness as per Euro-standard or DIN (German standard organization) which specifies the standard height, length and thickness taking the above mentioned points into considerations.

Components layout precede artwork preparation. Bishop Graphics Inc. is supplying self-adhesive puppets for active and passive electronic components [3, 4]. The most popular method for preparing a layout is to use a transparent tracing sheet and place it over a graph sheet, which facilitates layout-designing time. As the initial layout is likely to change, erasing or modifying is made easy on a transparent sheet. Using the respective puppets for each component can quickly do the component layout. For interconnection, a tracing paper is placed over the component layout and the conductors can be penciled on this tracing sheet. Any change in the component placement is carried out by simply removing that puppet and placing it afresh in some other convenient place on the transparent sheet. The grids in the graph sheet are utilized for lining up of all the pads in a line for components such as ICs, connectors' etc. This makes easy for drilling holes for components on the PCB using programmable, numerically controlled machines at a later stage. The use of transparent tracing sheet with graph sheet and puppets reduce the layout designing time to a greater extend [3]. This way of preparing layout is called " puppet Layout sketching method ".

Guidelines to prepare layout:

1. Start the layout only when the circuit diagram is clear and absolute with all the component details.
2. Prepare the PCB layout as viewed from the component side.
3. Proceed the layout as the signal flows. This will ensure the shortest inter connection.
4. All the larger components' site may be chosen keeping mechanical stress into consideration, which should be fully secured. This will ensure that these components do not pose any problem to the equipment even under severe environmental conditions.
5. Divide the circuit diagram into functional sub-unit and then allocate space for layout. This will be helpful for faster testing, servicing and reliability.
6. PCB designer must ensure that the layout conforms to the requirement of the motherboard. This is an important point, as the designer must know the type of connectors to be used, its location, test points; input, output connections as well as standard power supply termination's etc.
7. In multilayer PCBs, to identify conductor lines on each layer, it is advisable to use different color pencils.

8. While planning PCB layout it is advisable not to use the entire PCB area. Keep some 20% of the space for future growth or modification. This will ensure that the end product (equipment) size does not need any change even at a later stage.

Documentation

Documentation plays a vital role in the production line. The layout designer must be provided with complete circuit diagram and component list as input for layout preparation. Similarly, the output of the layout design must provide all necessary information for the completion of the artwork including mechanical assembly drawings.

The circuit diagram must be functionally separable. All the critical stage such as oscillator, high frequency signal lines etc., may be marked distinctly so that while preparing artwork, conductor length may be minimized, high frequency signal or clock line can be guarded by ground lines, power supply line conductor width etc. may be taken as given in MIL. std. 275 and other manuals [3,4]. The input and output lines must also be clearly marked with appropriate symbol so that these lines will be terminated in the connectors. The power dissipating devices could be high lightened in order to cater for heat sinks. If one takes the above precautions, the components on the PCB is likely to be well distributed

The component list of the circuit diagram may contain the component name, type, specification, tolerance, mechanical dimension, supplier, price etc. While writing the component values on the circuit diagram the following letters are recommended by the American National Standards Institution (ANSI).

Resistor = R (R1, R2,)
Capacitor = C (C1, C2,)
Transistor = Q (Q1, Q2,)
Integrated circuit = U (U1, U2,)
Transformer = T (T1, T2,)

'Layout sketch' document is essential for the artwork preparation. Layout sketch must contain component outline, component holes, interconnection, conductor width, conductor spacing etc. The recommended symbols by ANSI for conductor widths and holes are given in Fig.1.

Mechanical drawing is another important document for production units. The mechanical drawing should include PCB board size with tolerance, connectors with tolerance, PCB lamination to be used, plating specification and mounting holes, locations etc. These are some of the essential documentation's necessary before proceeding to artwork preparations [2, 5]

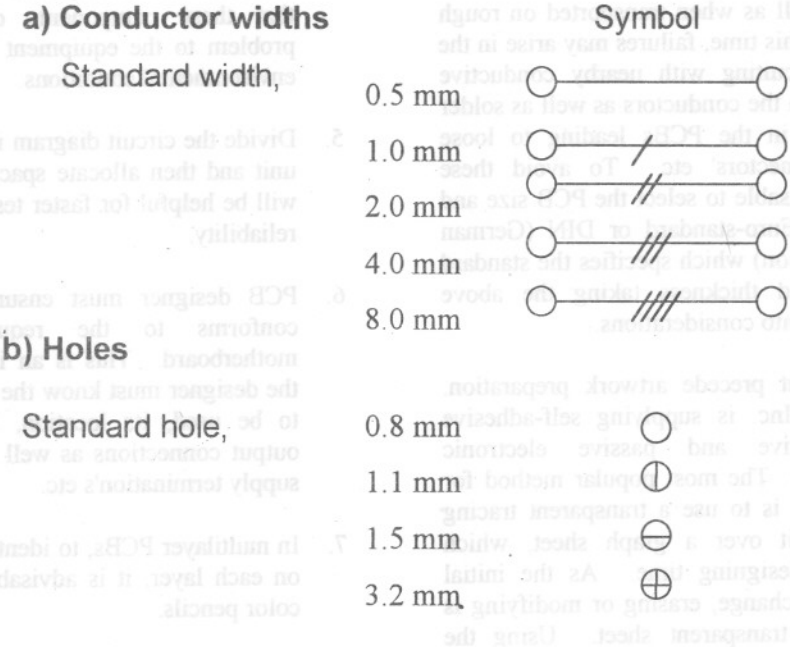

Figure 1 Layout coding.

ARTWORK PREPARATION

The artwork perfection is gaining importance in the production line because it is the first line improvement in any equipment performance. The artwork prerequisites are hardly taught in any training institutions. The artwork designer must have full of patience, skill for self-learning and apply his practical knowledge gathered at the right place. There are about five approaches to prepare artwork. They are (i) Ink drawing (ii) Black-taping (iii) Red/ Blue taping (iv) Black taping on diazo film (v) cut and strip. Among the five approaches the second method viz. Black taping on transparent base foil is considered as the most versatile method and is being widely accepted in industries for various electronic applications. Hence this method is chosen as a model for artwork preparation.

Black taping on transparent base foils

The self-adhesive solder pads, dual in line package (DIP), linear integrated circuits, discrete active components, registration pads, conductor tapes and for various standard connectors are readily available [3] for various dimensions and spacing. These self-adhesive pads and tapes are used as tools to produce artwork at a faster rate with precision. They are shown in fig 2. These materials are black in color and other side covered with adhesive covered by peelable back sheets. As these pads and tapes is one side coated with adhesive, they have a shelf life period of three years. It is advisable to store them in an air-conditioned room.

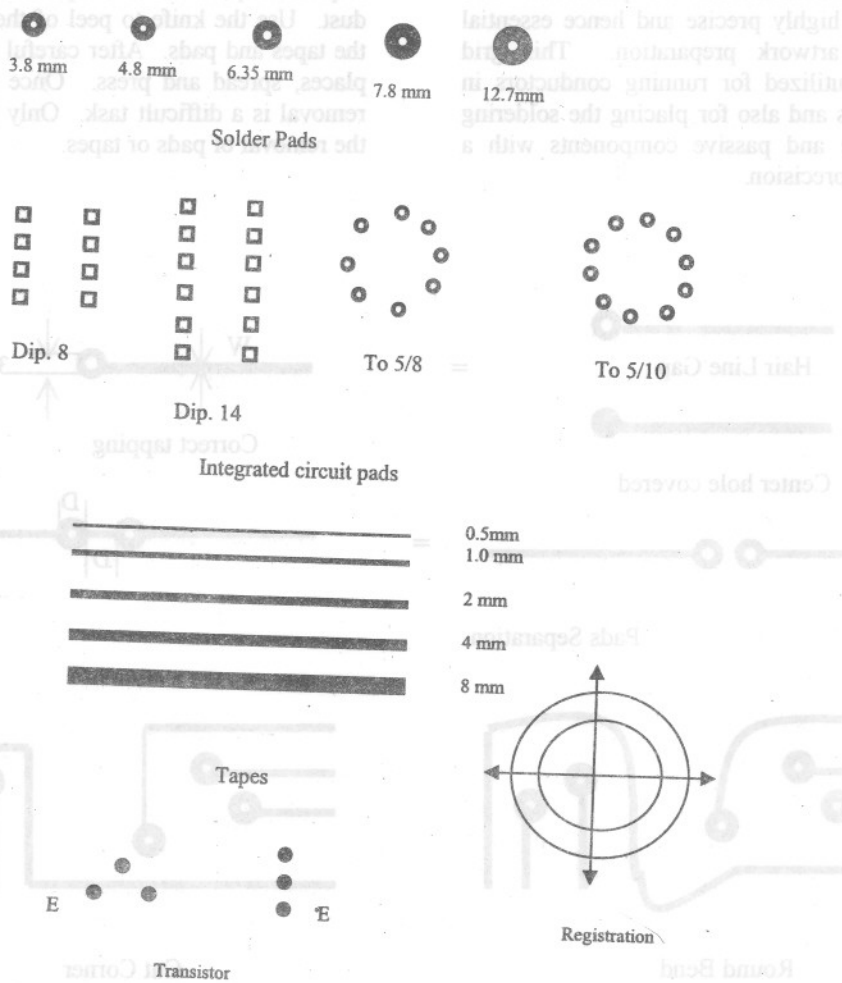

Figure 2 Self-adhesive pads and tapes

Artwork accessories

To produce better result in artwork, transilluminated artwork table is required for more convenience to work. This table must have smooth surface and illuminated evenly to cover the entire surface area. To reduce the designer's fatigue, it is advisable to have a slant table with adjustment facility.

The important hand tool for artwork is the specially designed artwork knife. It is used to cut the conductor tape to the required length, to peel of the back sheet of conductor as well as pads. This knife is available from the artwork aid suppliers and also in surgical instrumentation shops.

It is advisable to use plain artwork base foil with a special grid sheet underneath for precision. The grid sheets are highly precise and hence essential for the good artwork preparation. This grid intersection is utilized for running conductors in X/Y coordinates and also for placing the soldering pads for active and passive components with a good degree of precision.

Procedure for artwork [2, 5]

Fix the artwork base foil on the table. First PCB outlines are marked by using 2-mm tape in the form of "L" in all the four corners. Here one has to provide sufficient margin for the PCB card guides, connectors and PCB fixtures.

For multilayer PCBs, the artwork should have a perfect alignment. Before proceeding to artwork, the designer has to fix the Registration adhesive pads at least in two corners (diagonally opposite) of the base foil for perfect alignment.

The pads are to be placed exactly at the center of the grid intersections, to enable the holes are aligned to this center.

Tapes and pads are to be protected from dirt and dust. Use the knife to peel of the bottom sheet of the tapes and pads. After careful selection of their places, spread and press. Once it is pressed, its removal is a difficult task. Only erasing performs the removal of pads or tapes.

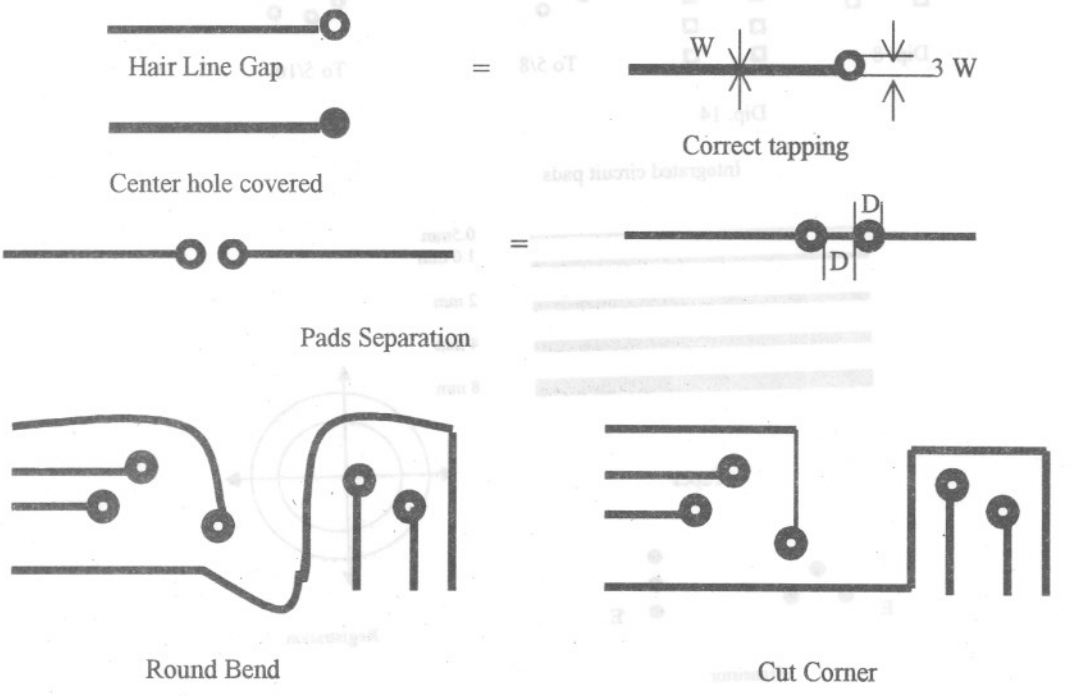

Figure 3(a) Tapping

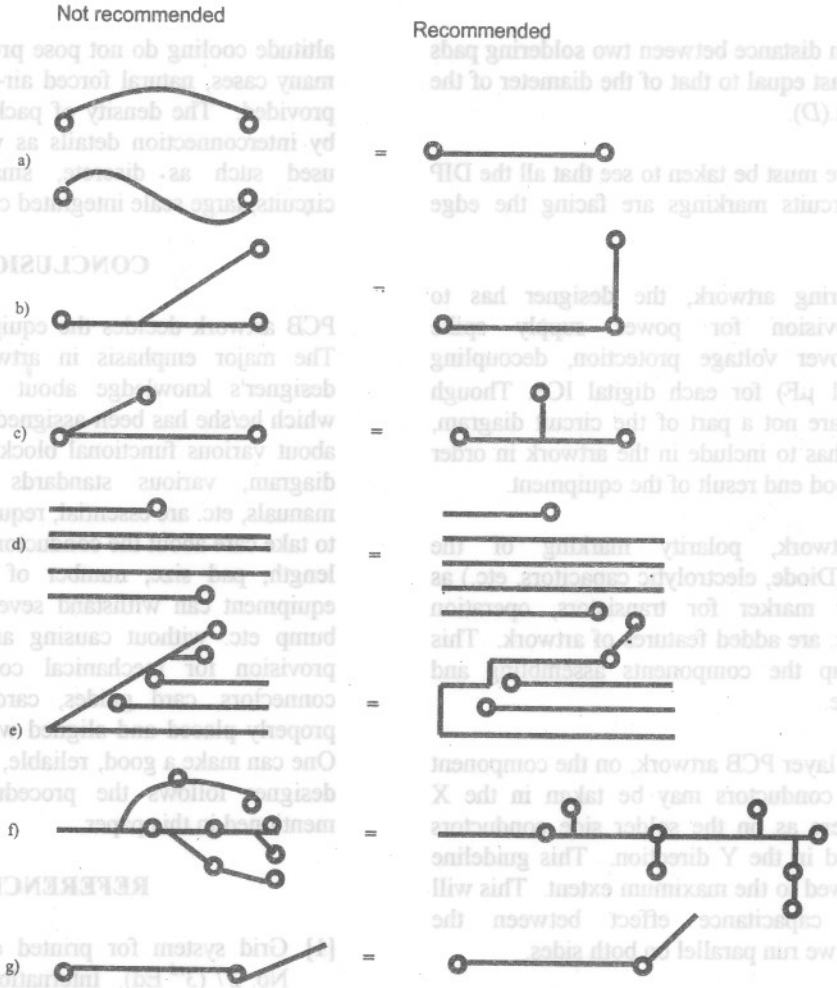

Figure 3(b) Conductor Routing

Some of the tips for artwork preparation are illustrated in Figs. 3 (a) and (b). Tape joining with solder pad is to be carried out very carefully. There should not be any hairline gap or the tape should not cover the center hole of the solder pad.

While routing the conductors, it is advisable to have cut corner rather than round bend edges. Both these routing have their own merit and demerit.

The width of the conductors for ground supply and signal [6] are related as given below.

$$W_{gnd} \geq 2\, W_{supply};\quad W_{supply} \geq 2\, W_{sig}$$

Where, $Wgnd$ = width of the ground conductor
W_{supply} = width of the power supply conductor
W_{sig} = width of the signal conductor

The width of the supply line conductor should be able to withstand in case of short circuit in the power supply (MIL. Std. 275)

The ground lines for analog circuits and digital circuits should be maintained separately, other wise circuit malfunction is likely to take place. In order to reduce the ground noise level it is advisable to run ground lines to guard the high frequency signals. This will minimize the pickup from adjacent conductors.

The conductor width of the signal line is related to the soldering pad in this form,
(i.e.) $D = 2\ W_{sig}$

Where D is the diameter of the soldering pad.

The minimum distance between two soldering pads must be at least equal to that of the diameter of the soldering pad (D).

Sufficient care must be taken to see that all the DIP integrated circuits markings are facing the edge connector.

While preparing artwork, the designer has to provide provision for power supply spike suppresser, over voltage protection, decoupling capacitor (0.1 μF) for each digital ICS. Though these things are not a part of the circuit diagram, the designer has to include in the artwork in order to produce good end result of the equipment.

On the artwork, polarity marking of the components (Diode, electrolytic capacitors, etc.) as well as key marker for transistors, operation amplifiers etc. are added features of artwork. This will speed up the components assembling and soldering time.

In the double layer PCB artwork, on the component side, all the conductors may be taken in the X direction where as on the solder side conductors may be routed in the Y direction. This guideline may be followed to the maximum extent. This will reduce the capacitance effect between the conductors if we run parallel on both sides.

COOLING AND PACKAGE

Artwork designer should have some knowledge about the heat dissipation of the PCB, mounting details and the environment in which it is going to be put in operation. Some equipment may utilize forced air-cooling. In that case, the air should not be obstructed by larger components mounted on the PCB. In some other cases, an exhaust fan may be used to suck the hot air; proper cooling of the equipment increases the performance of the equipment and reduced the mean time between failure (MTBF) of the equipment.

Coming to the packaging details, one needs to take sufficient care so that the PCB looks elegant and attractive. The density of packaging is directly related to cost. Reliability and serviceability of the board reduces as the density increases. The density is normally decided where it will be used and for what purpose. For air borne equipments normally one prefers to go for high-density packing as the size and weight are at high premium. Here at high altitude cooling do not pose problem. However in many cases, natural forced air-cooling is normally provided. The density of packing is also decided by interconnection details as well as components used such as discrete, small-scale integrated circuits, large scale integrated circuits etc.

CONCLUSION

PCB artwork decides the equipment performance. The major emphasis in artwork design is the designer's knowledge about the equipment for which he/she has been assigned, his/her knowledge about various functional blocks in a given circuit diagram, various standards specified in the manuals, etc. are essential, requirements. One need to take care about the conductor width as well as its length, pad size, number of PTHs so that the equipment can withstand severe shock, vibration bump etc. without causing any failure. Further, provision for mechanical components such as connectors, card guides, card lifters are to be properly placed and aligned within the card cage. One can make a good, reliable, elegant PCBs if the designer follows the procedure and guidelines mentioned in this paper.

REFERENCES

[1] Grid system for printed circuits, publication No. 27 (3rd Ed). International Electrotechnical commission, Geneva, Switzerland 1970.

[2] Dattals .D. P., Improve your PC artwork techniques, Electronic design, Philadelphia. P.A. Dec 6, 1976, pp 72-76.

[3] Bishop Graphics, Technical manual and catalogue 106, Bishop Graphics Inc. chats worth, California, 1976.

[4] Lyman T. Fine line printed circuits catch on – Electronics, McGraw Hill, New York, Jan 19, 1978, pp 84-86.

[5] Lund. P., Generation of Precision artwork for printed circuit boards, John Wiley & sons, Chichester, 1978.

[6] Walter, C. Bosshort, Printed circuit boards-Design and technology, Tata McGraw Hill, nineteenth reprint 1998.