

Manufacturing of Printed Circuit Board

An Industrial Training Report

Submitted to

Jagdish Namera

By

Munjal Nayak

University roll no 19BEE051

Electrical Engineering

Pandit Deendayal Energy University

Gandhinagar, Gujarat



UGC RECOGNIZED

PDEU PANDIT
DEENDAYAL
ENERGY
UNIVERSITY

Formerly Pandit Deendayal Petroleum University (PDPU)

Technical Specification

The PCB comes in various size and type of materials. The common type of material used in PCB is FR4 (Flame Resistance 4) it is made of glass epoxy resin

The PCB comes in various type Single side copper layer, Double side copper layer, Metal core double and single side copper layer

It also comes in different types of thickness like,0.2, 0.4, 0.8, 1.0, 1.2, 1.6, 2.0, 2.5 & 3.2 all measurement is in mm.

The thickness of copper layer can also vary like 1 Oz, 2 Oz, 3 Oz

The masking layer of PCB also comes in different colour like Green, White, Black, Blue ,Red

The type of material varies from FR1,FR4, CEM1,CEM3, PAPER PHENOLIC, Metal Clad, FR4 TG

1) SINGLE SIDE 2) DOUBLE SIDE 3) METAL CORE

Specifications	Standard
Minimum conductor width	4 mils(0.1 mm)
Minimum conductor Spacing/Air gap	4 mils(0.1 mm)
Minimum Plated hole size	12 mils(0.3mm)
Maximum Plated hole size	240 mils(6mm)
Maximum board size	350 mm x 500 mm
Maximum number of layers	2
Minimum copper thickness	1 Oz (35 microns)
Maximum copper thickness	3 Oz (105 microns)
Dedicated Electrical Testing (BBT / FPT)	Available
PCB Cutting options	Routing , V groove , Punching

PRODUCT FINISH:	
Single Sided	HAL, Lacquer, ROLLER TIN FINISH
Double Sided	HAL, TIN , Electroplated Gold & Nickel

Solder Mask Finish	PISM, Liquid Solder Mask
Solder Mask colour	Green, White, Black, Blue ,Red
Legend/I dentcolour	White, Black, yellow, Green
Special Requirement	Carbon Printing on Push Button , Selective Gold & Nickel plating on Connecting Tabs

Product Raw Material	
Single Side	FR1,FR4, CEM1,CEM3, PAPER PHENOLIC, Metal Clad
Double Side	FR4 TG
Board Thickness (in mm)	0.2 , 0.4 , 0.8 ,1.0 , 1.2, 1.6 ,2.4 , 3.2

Manufacturing Process

1.) Cad Designing

Here the designing team create and send the Gerber file to the company, Where the engineering team inspect the files sent by clients, and set the tolerances according to machine specifications, they create the job card and sends it for further processing.

2.) Shearing

Here the selection of material type, size, width and copper thickness is done.

Then the operator calculates total panel needed and total surface area after the calculation the operator cuts the panel into given size from the job card .

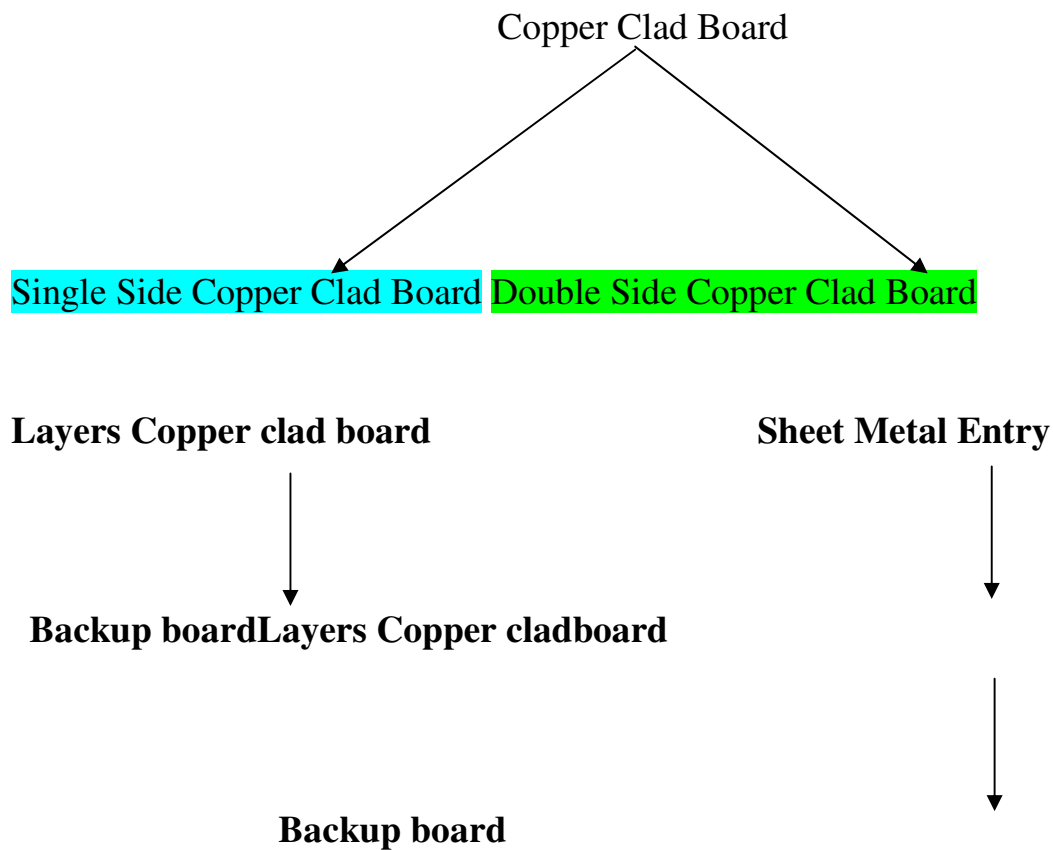
3.) CNC Drilling

CNC (Computer Numeric Control) Drilling is the process of drilling holes into copper clad board so that the required through hole component fits inside properly.

To perform this operation, we need analysis the given hole size and the number of holes. We need stack up the copper clad board according to following steps

0.3mm or less than it and 500 holes	Stack up single layer
0.3mm < 0.4mm and 500 holes	Stack up two layers
More than 0.4mm and 500 holes	Stack up to three layers

We need stack up the copper clad board according to following steps



In single side copper clad board the backup board is added for protection from accidental touch to metal workbench, In Double side Copper Clad Board the sheet metal entry is used for extra protection from fibre resin strands which can damage the copper clad boards and the backup board is also used for above same reason

4.) Deburring

After the CNC drilling is done the copper clad boards go through the process of deburring in which the remains the copper dust and fibres of resin is remove using a random axis orbiter sander machine.

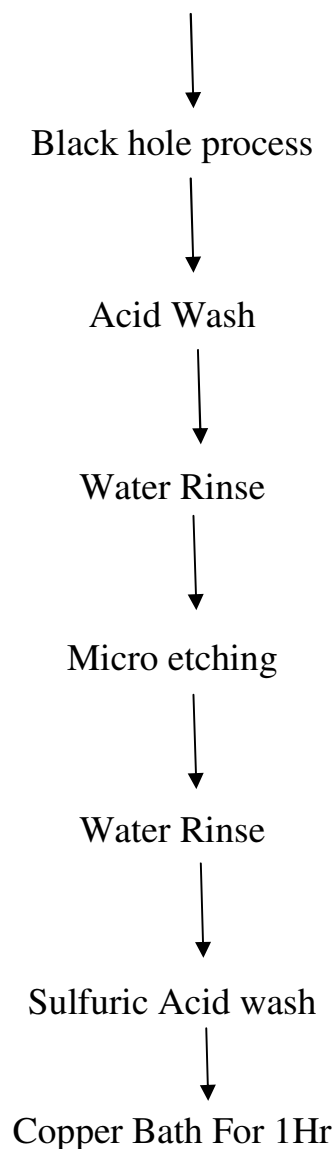
5.) PTH (Plating through hole)

This process is only for double side copper clad board with via holes (Conducting holes which connect both layers of copper).

To achieve the conductance in the holes, we need to perform the process called black hole. In this process a layer of conducting carbon is made around the holes and the process also helps in the plating of copper inside the wall of holes.

The copper clad boards go through process of plating according to following steps

Deburred copper clad board



This process also benefits in increasing the copper layer of board by some micron.

6.) Scrubbing

In this process the residue of the chemicals used in previous steps is removed by using the solution of soap and water then it is pass through a air heated convection machine which dry the copper clad board

7.) Track Printing

There are two ways to print tracks on copper clad board and both the methods are used according to requirement.

1.)Dry film Method

This method is used in double side copper clad board or if we need to print very complex tracks

In dry film method the photopolymer film resist is used, this film is sensitive to white or UV light, therefore the whole process happens in a yellow room (yellow light has the wavelength which doesn't interfere with photopolymer film resist)

With the help laminator the film is laminated onto copper clad board.

To move further in this process we need the track mask layer which is designed by the engineering team and printed in a transparent plastic sheet

The layer mask is pined on the surface on UV exposure machine, then the copper clad board is place on the layer mask then the lid is closed and a vacuum pump pumps out the air between the gap of the glass and lid which give enough pressure on copper clad board so that it can print properly then the UV light is turned on for 3 seconds which prints a negative image of layer mask on copper clad board

By exposing the laminated copper clad board to UV light cures the photosensitive ink which is present in photopolymer film resist which gets print on copper clad board and the uncured ink gets

removed in the developer machine after peeling the protective plastic layer.

The developer machine uses a solution of sodium carbonate which removes the uncured ink and develop the copper clad board

Hence after this process the tracks are successfully printed on copper clad board

Then the board passes through a heater which dry the board

2.)Normal Method

In this method the copper clad board is placed beside the t20 net and a layer of solder mask (DH203) is put by applying pressure Then the copper clad board goes in oven for about 35 minutes@72 degree Celsius which makes the solder mask little stiff

To move further in this process we need the track mask layer which is designed by the engineering team and printed in a transparent plastic sheet

The layer mask is pined on the surface on UV exposure machine, then the copper clad board is place on the layer mask then the lid is closed and a vacuum pump pumps out the air between the gap of the glass and lid which give enough pressure on copper clad board so that it can print properly then the UV light is turned on for 3 seconds which prints a negative image of layer mask on copper clad board

UV light cures the solder mask which gets print on copper clad board and the uncured solder mask gets removed in the developer machine

The developer machine uses a solution of sodium carbonate which removes the uncured solder mask and develop the copper clad board

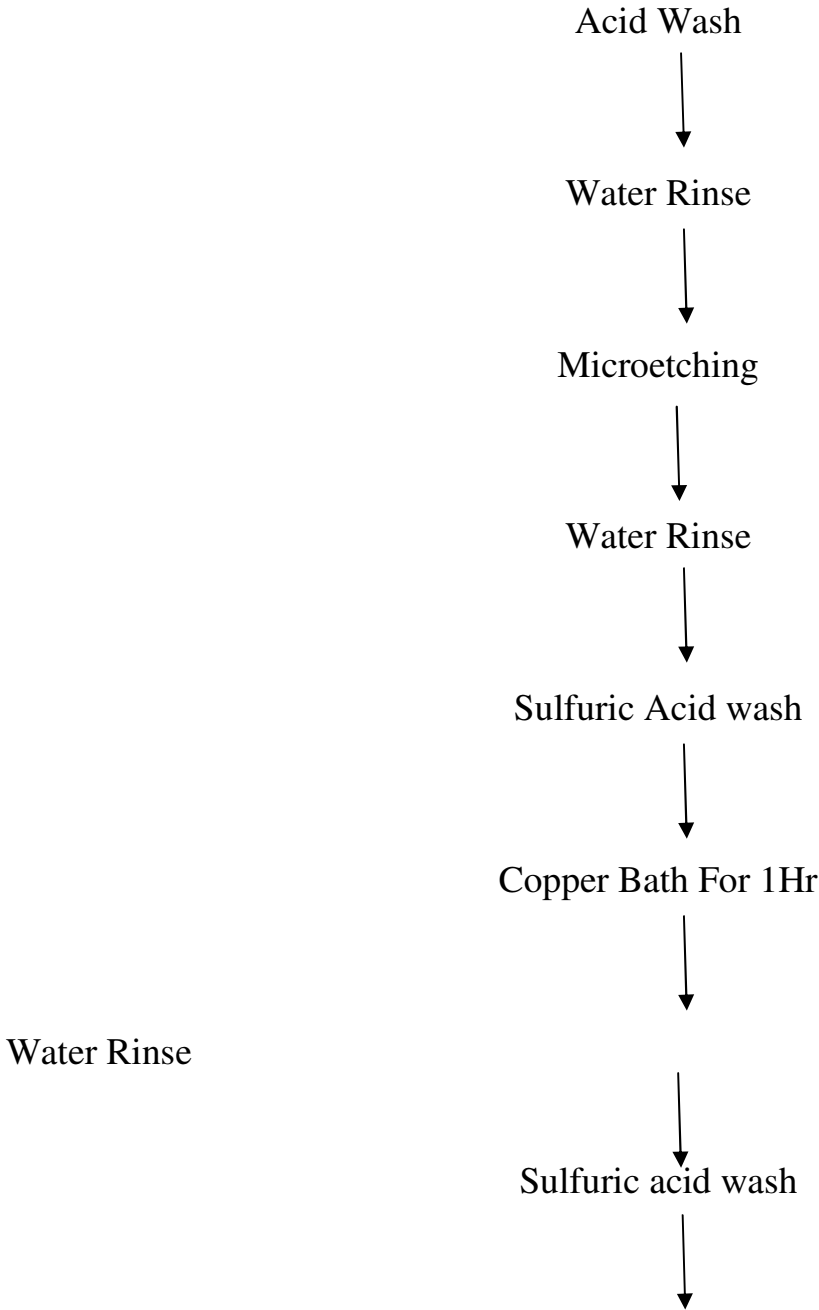
Then the board passes through a heater which dry the board

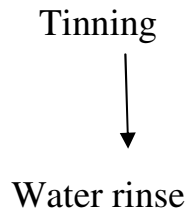
8.) Quality Checking 1

In this process the quality inspector check for any under cut size copper clad board, under developed boards, cuts on boards, short circuit on boards, Proper plating of copper in via holes is done or not and any other damages of deformation

9.) Plating and Tinning

Here the process of plating and tinning is done in this process the copper board goes through the following steps





This process is done by the following reason

1. Acid wash

The process of acid wash is done to remove unwanted chemical, dust, other disturbing materials

2. Water rinse

This process is done to dilute the acid on the copper clad board or water can completely remove acid.

3. Micro etching

This process is done to remove small scratches and micro strands of copper which were left during deburring.

4. Water rinse

This process is done to the copper clad board to remove micro etchant completely.

5. Sulfuric Acid wash

The process of acid wash is done to remove water which stops the oxidation on copper.

6. Copper Bath

The process electro plating is used here, when the copper clad board is dip into copper sulphate containing copper balls and a huge amount of current us passed through the system (solution of copper sulphate and copper clad board) according to the board size and quantity (on an average 7 amp for medium size board is kept)

7. Water Rinse

This process is done to the copper clad board to remove copper sulphate completely.

8. Sulfuric Acid wash

The process of acid wash is done to remove water which stops the oxidation on copper

9. Tinning

The process electro plating is used here, when the copper clad board is dip into Tin sulphate containing Tin alloy balls and a huge amount of current us passed through the system (solution of Tin sulphate and copper clad board) according to the board size and quantity (on an average 7 amp for medium size board is kept)

10. Water Rinse

This process is done to the copper clad board to remove tin sulphate completely

10.) Etching

This process is carried out by the etching machine, the machine uses the solution of liquid ammonia and remains of copper particles

In this process first we have to remove ink from copper clad board, the ink is removed by the solution of caustic potash

The machine only removes layer of copper which was hidden under the ink layer but does not remove layer of tin

After the extra copper is removed then the PCB (Now we can call it PCB cause it now Printed Circuit Board) is passed through the Tin striping machine

The Tin striping machine uses mild nitric acid which only remove the layer of tin. Hence the copper tracks are printed on the copper clad board

11.) Quality Checking 2

In this process the quality inspector check for any under cut size copper clad board, under developed boards, cuts on boards, short circuit on boards, Proper plating of copper in via holes is done or not and any other damages of deformation

12.) MASKING

In this process a layer of desired ink is masked on the panel on both side. Solder mask, also known as solder resist is a strong, permanent layer that protects copper traces and the interfaces between them on printed circuit boards (PCBs). The main function of a solder mask is to prevent conductive solder bridging between different electronic components and causing short circuits. The ink of the masking can vary according to the clients demand but usually the colour for masking is green. When the masking of ink is completed, the panels are sent again to the yellow room for the PSIM (Photo Imaginable Solder Mask) process. This process exposes inverted part of the part of copper on which solder is to be applied. The mask layer of film are placed on the PCB panels and exposed through UV lights. Once the panels are sent to developer the copper is exposed on the panels where the solder is to be applied. Before the panels are sent to the yellow room it is kept inside the oven and baked for 35 minutes @ 150 degree C

13.) HASL (Hot Air Solder Levelling)

In this process the soldering is applied on the PCB panels To do this process first a soldering flux is applied on the panels of PCBs and then one by one the panels are put inside HASL machines where the soldering is applied aroundholes. The PCB is typically dipped into a bath of molten solder so that all exposed copper surfaces are covered by solder. When the HAL process is done the panels are sent for the legend printing process.

14.) LEGEND PRINTING ON PCB

In this process the legends (letters) of a PCB are printed on top of the solder mask. Silkscreen or component identification layer are terms used. This process is done to properly understand whether which component are to be placed and where are they going to be placed.

15.) CUTTING OF PCB BOARDS

The border of panel are cut into size according to their desired final panel size and then it is sent to the V groove machine for making V shape groove on both side, where the panels need to break or separate

16.) TESTING OF PCB BOARD WITH FPT AND BBT

The final steps for manufacturing process PCB testing

There are two ways of testing a PCB

A Flying Probe Machine

In a flying probe testing machine a panel is set up in the machine and then the program is set up in the machine's software, there are total 4 probes in the machine which check for isolation and continuity in the panel. The software indicates if there is error for the continuity error and the isolation error. The panel are to be checked manually too as some times the FTP shows the error due to the positioning gap.

Bare board Testing

A Bareboard is a solderless device for Testing through hole PCB. In this process we need to setup a hole mask PCB in which we need to insert the pins after that we need to connect the wires to the all the pins and the other side to the BBT machine via connector To move Further in the process, we need to teach the machine by giving different PCB of same design. According to that the machine calculates which pin is open circuited and which circuit is closed circuited after the teaching the machine remembers the pins and test other PCB according to that model.

17.) FINAL QC

At final QC Inspector will verify that manufacture PCBs are qualified in following criteria, Masking, legend printing, continuity and isolation. They also verifies the number of PCBs and check whether the requirement by the client is full field or not.

18.) Packing

After the panels are verified, the PCBs are sent for packaging and are ready for dispatch.