

PCB DESIGNING & IT'S MANUFACTURING

INDUSTRIAL TRAINING REPORT

Submitted by

JAVED KHAN (2016-333-024)

Under the supervision of

VIVEK SHARMA

SAHASRA ELECTRONICS PVT.LTD.



in partial fulfillment for the award of the degree of

BACHELOR OF TECHNOLOGY (BTECE)



Department of Computer Science

School of Science and Technology

JAMIA HAMDARD

New Delhi-110062

(2019)

ABSTRACT

A printed circuit board, or PCB, is a self-contained module of interconnected electronic components found in devices ranging from common **beepers**, or pagers, and radios to sophisticated radar and computer systems. The circuits are formed by a thin layer of conducting material deposited used in printed circuit boards is a glass fiber reinforced (**fiberglass**) epoxy resin with a copper foil bonded on to one or both sides. PCBs made from paper reinforced phenolic resin wi, or "printed," on the surface of an insulating board known as the substrate. The substrate most commonly th a bonded copper foil are less expensive and are often used in household electrical devices.

There are three major types of printed circuit board construction: single-sided, double-sided, and multi-layered. Single-sided boards have the components on one side of the substrate. When the number of components becomes too much for a single-sided board, a double-sided board may be used. The third type, a multi-layered board, has a substrate made up of layers of printed circuits separated by layers of insulation.

Components on a printed circuit board are electrically connected to the circuits by two different methods: the older "through hole technology" and the newer "surface mount technology." With through hole technology, each component has thin wires, or leads, which are pushed through small holes in the substrate and soldered to connection pads in the circuits on the opposite side. With surface mount technology, stubby J-shaped or L-shaped legs on each component contact the printed circuits directly. A solder paste consisting of glue, flux, and solder are applied at the point of contact to hold the components in place until the solder is melted, or "reflowed," in an oven to make the final connection. Although surface mount technology requires greater care in the placement of the components, it eliminates the time-consuming drilling process and the space-consuming connection pads inherent with through hole technology. Both technologies are used today.

The miniaturization of electronic products continues to drive printed circuit board manufacturing towards smaller and more densely packed boards with increased electronic capabilities. Advancements beyond the boards described here include three-dimensional molded plastic boards and the increased use of integrated circuit chips. These and other advancements will keep the manufacture of printed circuit boards a dynamic field for many years.

ACKNOWLEDGEMENT

I am highly indebted to **Mr. Vivek Sharma**, for his supervision as well as for providing necessary information regarding the report and also for his support in completing the project work. I would like to express our deep and sincere gratitude to our coordinator, **Mr. Indira Singh Bora**, for his unflagging support and continuous encouragement throughout the project work. Without his guidance and persistent help this project would not have been possible.

I must acknowledge the faculties and staff of Automation Department of **ICFAI University**, who gave the permission to do six months training and to complete the project work and the report.

It's my great pleasure to acknowledge my classmates for giving me appreciable suggestions for my project topic and helping me in solving problems that occurred while making this project. Their support made it possible for me to complete this project.

JAVED KHAN

B.Tech (ECE) 7th Semester

Jamia Hamdard

CONTENTS

Topics	PAGE NO.
CERTIFICATE	
ABSTRACT	
ii ACKNOWLEDGEMENT iv LIST OF FIGURES	
vii LIST OF TABLES	
viii	
Chapter 1 ABOUT THE INDUSTRY	1
• About Sahasra Electronics Pvt. Ltd.	1
• LED Lighting	2
• Value Added Services	2
• Evolution of Sahasra Group	3
Chapter 2 INTRODUCTIONS TO PCB	4
2.1 History of PCB	5
• Classification of PCBs	6
• Single sided PCBs	6
• Double sided PCBs	7
• Multi layered board	8
2.2.4 Rigid & Flexible PCBs	9
• Base material for Printed Circuit Boards	10
• Laminates	10
• Glass Epoxy	11
• Design Process of PCB	11
2.4.1 PCB Designing	13
Chapter 3 INTRODUCTIONS TO PCB MANUFACTURING	21

3.1 Manufacturing	21
3.2 PCB manufacturing process in detail	25
Chapter 4 PRINTED CIRCUIT ASSEMBLY	36
4.1 Process of Printed Circuit Assembly	36
• Through Hole Technology	41
• Axial v/s Radial leads	42
• Types of soldering	42
4.2.2.1 Do's and Don'ts for solder paste	44
• Surface Mount Technology	45
4.3.1 Assembly Techniques	45
• Advantages & Disadvantages of SMT over THT	49
• Advantages	49
4.4.2 Disadvantages	50
Chapter 5 ELECTROSTATIC DISCHARGE (ESD)	51
Chapter 6 SAFETY AND ENVIRONMENTAL ISSUES IN PCB INDUSTRY	53
• Safety precaution in PCB industry	53
• Health & environment issues	54
6.1.2 Polluting agents	54
6.2 Quality Control	55
6.3 Toxic materials & safety considerations	55
Chapter 7 THE FUTURE	57
7.1 PCBs use to fit changing consumer demands	57
7.2 Design changes to match new technological needs	59

APPENDICES

Appendix 1

Appendix 2

REFERENCES

LIST OF FIGURES

FIGURES	PAGE NO.
Fig. 1.1 Sahasra Electronics Pvt. Ltd., Noida	1
Fig. 2.2.1 Single sided PCB	7
Fig. 2.2.2 Double sided PCB	8
Fig. 2.2.3 Multi-layer PCB stack	8
Fig. 2.4 (i) Designing of PCB on software	12
Fig. 2.4 (ii) Printed circuit layout	12
Fig. 2.4.1 (i) Steps in PCB designing	13
Fig. 2.4.1. (ii) A simple circuit schematic	14
Fig. 2.4.1 (iii) Schematic capture screen of typical PCB CAD software	16
Fig. 2.4.1 (iv) PCB layout using CAD software	16
Fig. 2.4.1 (v) Contents of Gerber File	17
Fig 3.1 Sheet cutting machine	22
Fig. 3.2 (i) Flow chart of single sided PCB manufacturing process	26
Fig. 3.2 (ii) Film plotted	27
Fig. 3.2 (iii) Copper foil laminate	28
Fig. 3.2 (iv) Drill bits	29
Fig. 3.2 (v) Tip of a CNC drill	29
Fig. 3.2 (vi) Holes drilled on laminate	29
Fig. 3.2 (vii) Green solder resist	30
Fig. 3.2 (viii) Laminate after patterning	31
Fig. 3.2 (ix) Laminate after stripping and etching	33
Fig. 4.2 (i) Through Hole PCB	41
Fig. 4.2 (ii) Through Hole resistor	41
Fig. 4.2.2 Wave soldering machine	43
Fig. 4.3 SMT components	45
Fig. 4.3.1 (i) Stencil Printing process	47
Fig. 4.3.1 (ii) SMT Lines	48

CHAPTER 1 – ABOUT THE INDUSTRY

● ABOUT SAHASRA ELECTRONICS PVT LTD



Fig.1.1 Sahasra Electronics Pvt. Ltd., Noida

“Sahasra” was conceived in the year 2000, the new millennium and since then has been one of the most successful and fastest growing electronic companies in India. The group comprises of 6 businesses providing end-to-end electronic solutions from design to manufacturing to distribution.

With 4 manufacturing plants situated in the NCR, India, one manufacturing plant in Rwanda, Africa and 8 sales & marketing offices in India, USA, Canada, Africa & Europe. The group has emerged as the leading exporter of electronic products and services.

The group's mission *"to deliver goods and services of high quality level at competitive prices with business velocity that can provide complete satisfaction to our customers anywhere on the globe"* has been the success mantra over the last decade. With 510+ (1/5 female employees) people across its businesses, a group sales turnover of US\$25MM and a brand new 20-20 vision the group is well poised to become one of the leading international electronic solutions company will continue to invest heavily in PCB manufacturing for the next 2 years as part of its PCB manufacturing expansion plans in the higher layer count and complex PCB's space. The boards are fabricated to IPC standards and meet RoHS and REACH regulations. The PCB facility has the distinction of being the first UL approved Metal Core PCB fabrication facility in India for LED Lighting applications besides UL approval for its FR-4 boards.

1.1.1 LED Lighting:

As part of the new '20-20 vision' the company has charted out, LED Lighting will be one of the key strategic areas fuelling its growth. Today Energy efficient Lighting and solar products, assemblies and services comprise of 20% of the group's total turnover and will be a major contributor to its future growth as well.

The group has focused on providing solutions for the lighting industry by designing ready to use OEM solutions, turnkey assembly of complete luminaries' as well as in-house manufacturing of Metal Core based light engines and driver solutions based on customer designs and requirements. The Sahasra Group is working with companies in Automotive Lighting, General Lighting, Emergency Lighting, Entertainment Lighting and many other verticals for providing LED lighting solutions.

The Group invested \$3M in its state of the art manufacturing facility in Kigali Prime Economic Zone, Rwanda in the East African region of the African Continent. This strategic investment in Rwanda, as called out by Rwanda Development Board(RDB) is helping Rwanda transform its energy landscape and benefitting the people of Rwanda by creating jobs, making the country energy efficient and self-sufficient besides other economic benefits of helping with import substitution, generating exports and giving impetus to locally made and sourced Rwandese products. Sahasra Rwanda is the leading LED Lighting Company in Rwanda with more than 70% market share and will be exporting products to other countries in the East African Community as trade grows between these countries as part of their Free Trade Agreement.

1.1.2 Value Added Services:

As part of its endeavor to provide end to end electronic solutions the group assists its customers in distributing their products in India as well as providing value added solutions and services.

Cable assembly, plastic and metal boxes, PCB design, value engineering, system assembly, warehousing and 3rd party logistics are some of the value added services offered from a manufacturing perspective.

The distribution division of the group focuses on helping customers set up a distribution base in India and opening doors to one of the largest and fastest growing markets in the world. By distributing only those products which are manufactured at its own plants the group is able to provide the right level of after sales and service.

- **EVOLUTION OF SAHASRA GROUP**

- In 2000: Started Electronics distribution business – Sahasra Electronics (SE).
- In 2001: Commenced EMS operations at NSEZ, NOIDA under Flagship Co. – Sahasra Electronics Pvt. Ltd. (SEPL)
- In 2005: Started PCB Fabrication operations at NSEZ, NOIDA under - Nano ElectrotechPvt. Ltd. (NEPL)
- In 2011: Acquired PCB Fabrication Co., catering to the Domestic demand – InfoPower Technologies Ltd. (IPTL). Now offering EMS as well for the domestic market.
- In 2012: Set up LED based lighting products division, new manufacturing facility in Rwanda, Africa and distribution of LED products in India under its own brand name.
- In 2013: Started SahasraSambhav Skill Development Pvt. Ltd. – To adequately train the youth and make them industry ready.
- In 2016: Manufacturing IT Products such as UFD, OTG, SSD, SD &mSD Cards in technical collaboration with Phison and Power Banks in technical collaboration with Getac, Taiwan.

CHAPTER 2 - INTRODUCTION TO PCB

A Printed Circuit Board usually called PCB is the board you see inside common electronic gadgets. It is used to mechanically support and electrically connect electronic components together. Connections between the components are made through copper connections called routes which become passage for electrical signals. An Austrian scientist, Dr. Paul Eisler, is credited with making the first operational printed wiring board in 1943. Printed circuits did not become commonplace in consumer electronics until the mid-1950s. PCBs are found in devices ranging from common radios, smart phone, computer systems or any electronic device you can think of.

There are three major types of printed circuit boards: single-sided, double-sided, and multilayered. Single-sided boards have the components on one side. When the number of components becomes too much for a single-sided board or the circuits become too much complicated to be prepared on a single sided board, double-sided boards are used. The third type, a multi-layered board is made up of layers of printed circuits separated by layers of insulation.

Components on a printed circuit board are electrically connected to the circuits by two different methods: the older "through hole technology" and the newer "surface mount technology." With through hole technology, each component has thin wires, or leads, which are pushed through small holes in the board and soldered (connected with an alloy called solder) to connection pads in the circuits on the opposite side. In multilayered boards, the components on the surface connect through plated holes drilled down to the appropriate circuit layer. With surface mount technology, stubby J-shaped or L-shaped legs on each component contact the printed circuits directly.

Although surface mount technology requires greater care in the placement of the components, it eliminates the time consuming drilling process and the space consuming connection pads inherent with through hole technology. Both technologies are used today.

2.1 HISTORY OF PCBs

Printed circuit boards evolved from electrical connection systems that were developed in the 1850s. Metal strips or rods were originally used to connect large electric components mounted on wooden bases. After that time the metal strips were replaced by wires connected to screw terminals, and wooden bases were replaced by metal chassis. Smaller and more compact designs were needed due to the increased operating needs of the products that used circuit boards. In 1925, Charles Ducas of the United States submitted a patent application for a method of creating an electrical path directly on an insulated surface by printing through a stencil with electrically conductive inks. This method gave birth to the name "printed wiring" or "printed circuit". In 1943, Paul Eisler of the United Kingdom patented a method of etching the conductive pattern, or circuits, on a layer of copper foil bonded to a glass-reinforced, non-conductive base. Widespread use of Eisler's technique did not come until the 1950s when the transistor was introduced for commercial use. Up to that point, the size of vacuum tubes and other components were so large that the traditional mounting and wiring methods were all that was needed. With the advent of transistors, however, the components became very small, and manufacturers turned to printed circuit boards to reduce the overall size of the electronic package.

Originally every electronic component had wire leads, and the PCB had holes drilled for each wire of each component. The component leads were then passed through the holes and soldered to the PCB trace, the method called through hole construction. PCB's history in the 60's and early 70's led to the processes that were developed for plating copper on the walls of the drilled holes in circuit boards, permitting top and bottom circuitry to be electrically interconnected. These double sided boards became the industry standard. As the densities and complexities of electronic components increased, the multilayer circuit board, a process of sandwiching several circuit layers together, was developed. Integrated circuit chips were introduced in the 1970s, and these components were quickly incorporated into printed circuit board design and manufacturing techniques. From the 1980s small surface mount parts have been used increasingly and this has led to smaller boards for a given functionality and lower production costs. Flexible PCBs have also been introduced recently to cater to the needs of sophisticated electronic product market. Today we have reached an era where industry is competing to reduce the size of every electronic gadget which in turn needs smaller and denser PCBs.

2.2 CLASSIFICATION OF PCBs

Printed Circuit Boards may be classified into three based on their applications - consumer, professional and high reliability boards. Professional boards were made of better quality material to achieve tighter electrical environmental specifications using controlled fabrication techniques. Higher reliability boards, normally used in strategic applications like military and space crafts, were meant to provide the best of electrical properties through the use of high quality base material and tightly controlled manufacturing process. A more simple and understandable classification is now used, which is based on the number of planes or layers of wiring, which constitute the total wiring assembly of structures, and to the presence or absence of plated-through holes. This method of classifying boards has the advantage that it can be related directly to the board specifications.

2.2.1 Single Sided PCBs-

Single-sided means that wiring is available only on one side of the insulating substrate. The side which contains the circuit patterns is called the 'solder side' whereas the other side is called the 'component side'. Normally, components are fixed on component side and soldered over conductor tracks, if this is not possible, jumper wires are used. The number of jumper wires on a board cannot be accepted beyond a small number due to economic reasons, resulting in the requirement for double-sided boards. A single sided board is made from rigid laminate consisting of a woven glass epoxy base material clad with copper on one side of varying thickness. These types of boards have components on one side and printed circuit layer on the other side. They are used for simple schematics with fewer components. In other words, they are used for lower density circuits. In Single-sided PC, Circuit is only on one side of the insulating substrate. In SS PCB, Side which contains the **circuit patterns is called the 'solder side'** whereas the other side is called the '**component side**'. Single sided boards can be easily fabricated.

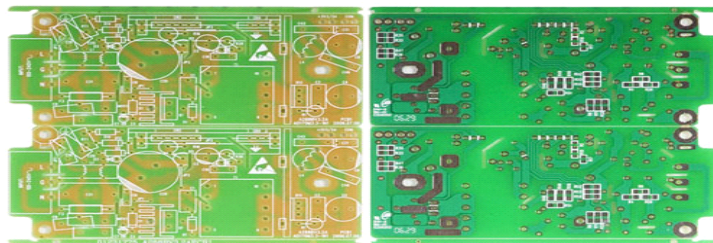


Fig. 2.2.1 Single Sided PCB

2.2.2 Double Sided PCBs-

‘Double-sided’ printed circuit boards have wiring pattern on both sides of the insulating material, i.e.; the circuit pattern is available both on the components side and the solder side. Obviously, the component density and the conductor lines are higher than the single-sided boards. Two types of double-sided boards are commonly used, which are:

- Double-sided boards with plated through-hole connection called PTH or vias; and
- Double-sided boards with surface mount technology called SMT

Double-sided PTH boards have circuitry on both sides of an insulating substrate, which is connected by metalizing the wall of a hole in the substrate that intersects the circuitry on both sides.

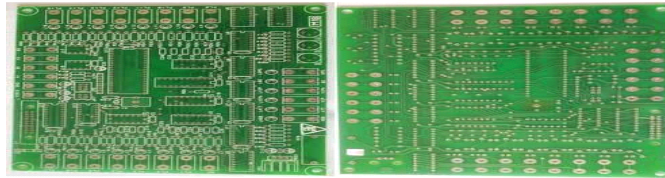


Fig.2.2.2 Double Sided PCB

This technology, which is the basis for the most printed circuits produced, is popular in cases where the circuit complexity and density is high. Double-sided non-PTH board is only an extension of a single-sided board. Its cost is considerably lower because plating can be avoided. In this case, through contacts are made by soldering the component leads on both sides of the board, wherever required. In the layout design of such boards, the number of solder points on the component side should be kept to a minimum to facilitate component removal, if required. Double sided boards are made from the same type of base material clad with copper on two sides of varying thickness. This is because you have a whole extra layer where you can add tracks to

connect the components, freeing space on the other to place the components closer together or add extra connections for IC with high pin counts.

2.2.3 Multi-Layered Board-

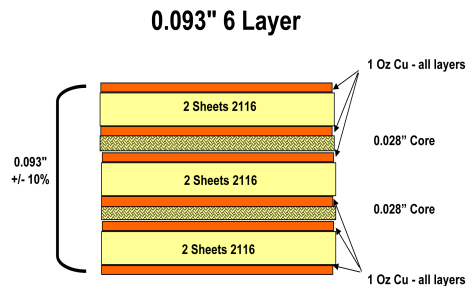


Fig. 2.2.3 Multi-Layer PCB Stack

The multi-layer board makes use of more than two printed circuit boards with a thin layer of ‘pre-preg’ material placed between each layer, thus making a sandwich assembly. The printed circuit on the top board is similar to the conventional printed circuit board assembly except that the components are placed much closer or they may have terminals, which necessitates the use of traditional board layers for the required interconnections. This electrical circuit is completed by interconnecting the different layers by plated through-holes, placed transverse to a board at appropriate places. Multi-layer boards have three or more circuit layers, while some boards have even as many as 50 layers. By virtue of multi-layer conductor structure, multi-layer printed wiring has facilitated a reduction in the weight and volume of the interconnections commensurate with the size and weight of the components it interconnects.

Multi-layer boards on the other hand are made from the same base material with copper foil on the top & bottom and one or more “inner layer” cores. The number of “layers” corresponds to the number of copper foil layers. As mentioned earlier the selection is usually made depending upon the complexity of schematic. Multi-layer boards are used in modern day computers, mobile phones; tablet computers where the gadget size is so small that the circuit cannot be accommodated in double sided circuit boards. As the schematic becomes more complex or denser, the number of layers in the circuit board becomes more.

2.2.4 Rigid & Flexible PCBs-

Printed circuit boards can also be classified on the basis of the type of insulating material used, i.e. rigid and flexible. While rigid boards are made of a variety of materials, flexible boards use flexible substrate material like polyester or polyamide. The base material, which is usually very thin, is in the range of 0.1mm thickness. Laminates used in flexible boards are available with copper on one or both sides in rolls. Rigid-flex boards, which constitute a combination of rigid and flexible boards usually bonded together, are three-dimensional structures that have flexible parts connecting the rigid boards, which usually support components. This arrangement gives volumetrically efficient packaging and is therefore gaining widespread use in electronic equipment. Flexible PCBs may be single sided, double-sided (PTH or non-PTH) or multi-layer.

2.3 BASE MATERIALS FOR PRINTED CIRCUIT BOARDS

The essential components of a printed circuit board are-

- The base, which is a thin board of insulating material, rigid or flexible, supports all conductors and components; and
- The conductors, normally of high purity copper in the form of thin strips of appropriate shapes are firmly attached to the base material.

The base provides mechanical support to all copper areas and all components attached to the copper. The electrical properties of the completed circuit depend upon the dielectric properties of the base material and copper. Copper conductors provide not only the electrical connections between components but also solderable attachment points for the same.

2.3.1 Laminates-

The basic function of the laminate is to provide mechanical support for electronic components. Laminates are manufactured by curing under pressure and temperature, layers of cloth or paper with thermoset resin to form an integral final piece of uniform thickness. The size can be up to 4 by 8 feet (1.2 by 2.4 m) in width and length. Varying cloth weaves (threads per inch or cm), cloth thickness, and resin percentage are used to achieve the desired final thickness and dielectric characteristics. The cloth or fiber material used, resin material, and the cloth to resin ratio determine the laminate's type designation (FR-4, CEM-1, G-10, etc.) and therefore the characteristics of the laminate produced. Important characteristics are the level to which the

laminate is fire retardant, the dielectric constant (ϵ_r), the loss factor ($\tan \delta$), the tensile strength, the shear strength, the glass transition temperature (T_g), and the Z-axis expansion coefficient (how much the thickness changes with temperature). There are quite a few different dielectrics that can be chosen to provide different insulating values depending on the requirements of the circuit. Well known pre-preg materials as they are called, used in the PCB industry are FR-2 (Phenolic cotton paper), FR-3 (Cotton paper and epoxy), FR-4 (Woven glass and epoxy), FR-5 (Woven glass and epoxy), FR-6 (Matte glass and polyester), G-10 (Woven glass and epoxy), CEM-1 (Cotton paper and epoxy), CEM-2 (Cotton paper and epoxy), CEM-3 (Non-woven glass and epoxy), CEM-4 (Woven glass and epoxy) and CEM-5 (Woven glass and polyester). Thermal expansion is an important consideration especially with ball grid array (BGA) and naked die technologies. Glass fiber offers the best dimensional stability. FR-4 is by far the most common material used today. The board with copper on it is called "copperclad laminate". Copper foil thickness can be specified in ounces per square foot or micrometers. One ounce per square foot is 1.344 mils or 34 micrometers.

2.3.2 Glass Epoxy-

Glass epoxy PCB is often called **FR4** laminate. It is an epoxy filled glass sheet with 1/2, 1 or 2 ounces of copper per square foot. FR-4 (or FR4) is a grade designation assigned to glass reinforced epoxy laminate sheets, tubes, rods and printed circuit boards (PCB). FR-4 is a composite material composed of woven fiberglass cloth with an epoxy resin binder that is flame resistant (self-extinguishing). FR-4 glass epoxy is a popular and versatile high pressure thermoset plastic laminate grade with good strength to weight ratios. With near zero water absorption, FR-4 is most commonly used as an electrical insulator possessing considerable mechanical strength. These attributes, along with good fabrication characteristics, lend utility to this grade for a wide variety of electrical and mechanical applications. G-10, the predecessor to FR-4, lacks FR-4's self-extinguishing flammability characteristics. Hence, FR-4 has since replaced G-10 in most applications. FR-4 epoxy resin systems typically employ bromine, a halogen, to facilitate flame-resistant properties in FR-4 glass epoxy laminates. Some applications

where thermal destruction of the material is a desirable trait still use G-10 non flame resistant PCBs.

2.4 DESIGN PROCESS OF PCB

Depend on printed circuit board manufacturer; there are numerous ways available for designing PCBs. This circuit board design can be manufactured as bulk using several machines in PCB fabrication industries including drilling, punching, plating and final fabrication processes that are performed through highly automated machines. Laser drilling with CNC machines, automatic plating machines, strip etching machines, and use of optical inspection equipment, flying probe testers for electrical testing of printed circuit board process result in high-quality PCBs (with a greater production yield). Draw the schematic circuit diagram with the PCB layout software such as CAD software, **Eagle** and **Multisim** software. This type of [PCB design software](#) contains a library of components that can be used to build the circuit. It is also possible to change the circuit design's position and then to modify according to your convenience and requirement. Here we have selected Eagle software to design the circuit and its procedure is as follows:

- Open the Eagle circuit board design software.
- A window with a menu bar appears.
- Click on the file menu.
- Select 'new design' from the drop-down menu.
- Click on the library menu.
- Select 'pick devices/symbol' from the drop-down menu.
- Select a relevant component by double clicking on it, so that the component appears on the window.
- Add all the components and draw the circuit with proper connections as shown in the figure.
- Enter the rating of each component according to the requirement.

- Go to Command Toolbar and click Text editor varriages, click on the Varriages, and then close the window.
- Next, a black screen appears which is of the layout or the film diagram of the circuit as shown in the below figure, and save this as an image format.

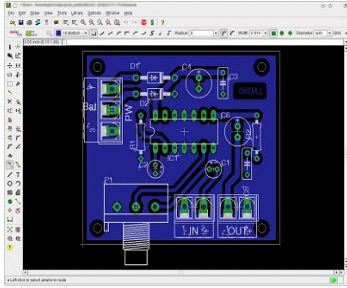


Fig. 2.4(i) Designing of PCB on software

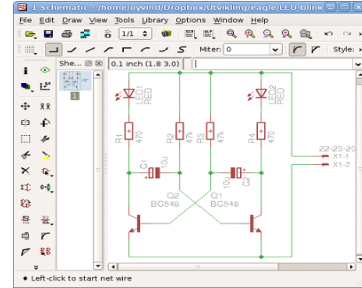


Fig. 2.4 (ii) Printed Circuit Layout

2.4.1 PCB Designing-

The basic input to the manufacturing process is software file called Gerber file. We will now look into what Gerber files are, how Gerber files are made and how the printed circuit boards are designed using these. Gerber is a standard file format in electronics industry. It is used to communicate design information for the manufacturing of printed circuit boards. In many ways, Gerber is the electronic world's equivalent of PDF. This odd little format, a hybrid machine control language, is a core component of the electronics manufacturing supply chain. A hybrid machine control language is something which machines can only understand – all with numbers, digits and special characters. You will be hereafter dealing with Gerber files very often. Now we will look into how Gerber files are generated. There are several basic steps involved in designing printed circuit boards. Most designs begin with a hand drawn schematic (circuit diagram) and design plan. Then, using software, an electronic version of the schematic is created. A file called *net list* file is created from the electronic schematic and used in layout software to create the physical layout of the PCB. Next, the components are placed and routed in the physical layout software and Gerber files are created. These Gerber files are used in a prototyping system to mill, drill, and cut the PCB substrate. The components are then placed and soldered to the substrate.

Finally, the board is tested to verify that it works as expected. The major steps in the PCB design and fabrication process are as follows:

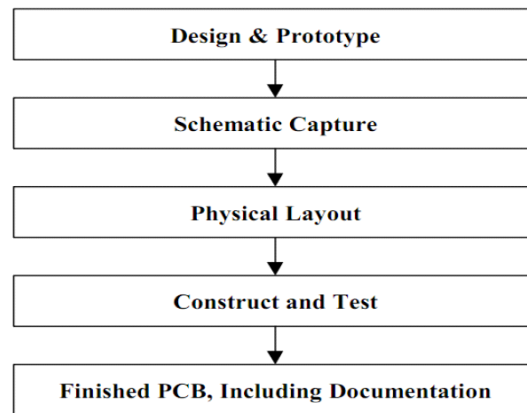


Fig. 2.4.1 (i) Steps in PCB designing

With this general procedural framework in mind, read on to learn more about each step in this process.

Prototyping

With a basic idea in mind, a circuit schematic is developed and analyzed to ensure the desired functionality and performance. When creating a circuit for PCB production, a designer would select specific components at this time. Commonly, the next step is to prototype and to test the circuit. It is also possible to use the schematic capture software along with related software to simulate the circuit without building it on a prototyping board. As shown in figure, a rough sketched schematic is initially created, along with some design specifications.

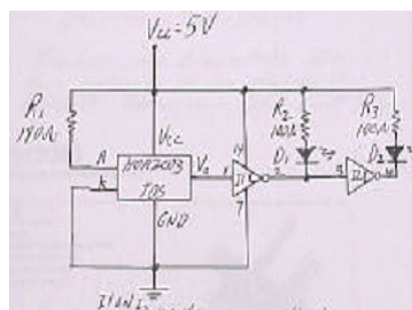


Fig. 2.4.1 (ii) A sample circuit schematic

The necessary components are chosen and the circuit is built on a prototyping board to verify that it meets the design criteria.

Schematic Capture

Schematic capture software comes in several forms. There are software like OrCAD Capture, Protel, Proteus, Eagle software, KiCad etc. Schematic capture allows the PCB designer to create an electronic schematic. This electronic schematic contains more information than its paper relatives. For example, every part symbol in Capture contains information telling what footprint the symbol is associated with. (Footprints are the symbols used in layout software to define the physical design of each component.) Capture parts symbols are used in a symbolic manner. Thus, the part symbol on Capture's screen does not show what the actual physical component looks like. It does allow the designer to connect all the components in a circuit and to test the workings of the circuit by exporting files to other software. For our purpose, Capture provides the starting point for creating a physical layout in layout software.

Physical Layout

Let us now consider a house plan as an example. A blueprint of a house plan tells the size of timber to use, as well as the dimensions of the living room wall and the dimensions of the window. It gives all the physical information necessary to build the house. Physical layout software can be thought of as a "blueprint" for a PCB. There are several programs available for doing physical layout, same as the ones mentioned in schematic capture. Most cases these software come as bundled. Of course you have to pay for each. There are in fact open source software like KiCad, Express PCB etc that you can freely download. The basic building blocks used in Layout are footprints. A footprint contains all the physical dimensions related to a particular part. For example, a 14-pin DIP footprint defines where each of the 14 drill holes are to be located, as well as associated information, such as text defining the part number of the component. In Layout the footprints of the various parts are placed and then routed. Routing refers to defining where the copper interconnects in the circuit will be located. Interconnects are copper paths on the surface of the PCB that connect one pin to another.

Interconnects are also known as routes or traces. In a house blueprint one have a rough frame blueprint and exterior view blueprint that contain different information about the same part of the house. In much the same way, Layout uses different layers to contain all of the information about the PCB. Some of the layers that will be used in doing two sided PCBs are Top, Bottom, and Silk. The top layer contains information pertaining to the top of your PCB, such as traces (routes) that are drawn on the top. The bottom layer contains information pertaining to the bottom of your PCB, such as traces that are drawn on the bottom. The silk layer is the ‘Silk Screen Top’ layer, and contains the text that will appear on the top of the PCB. There are many other layers as well, some of which will never be used in two sided PCBs.

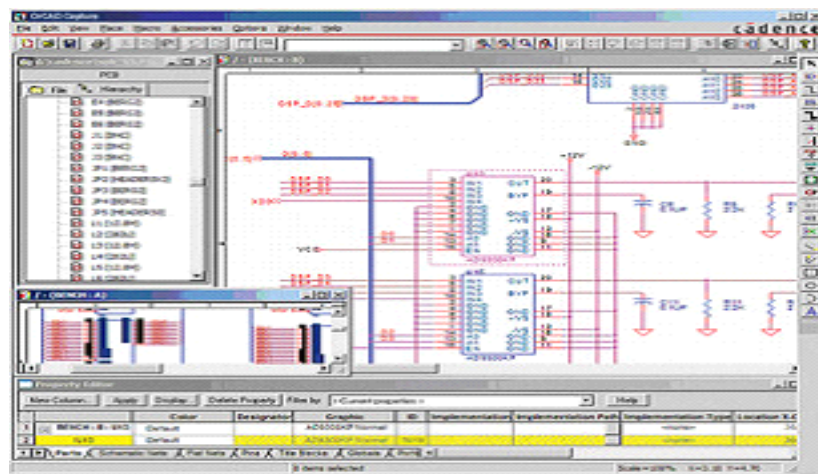


Fig. 2.4.1 (iii) Schematic capture screen of typical PCB CAD software

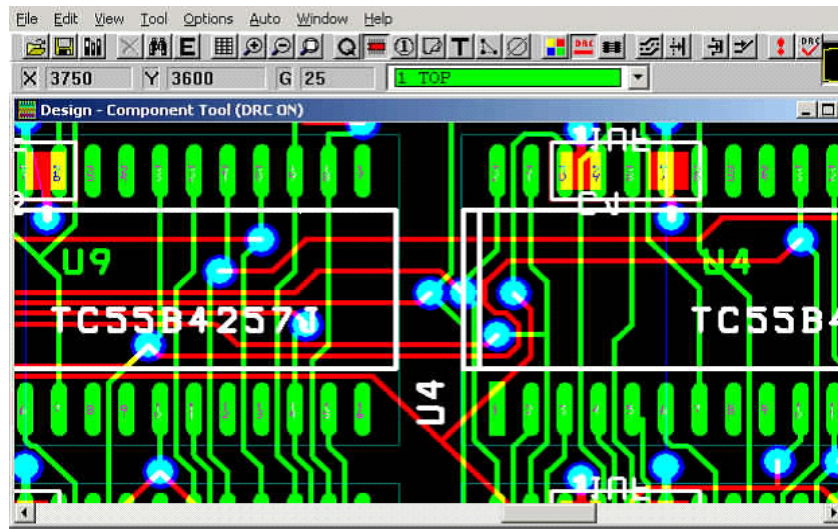


Fig. 2.4.1 (iv) PCB layout using CAD software.

Once the physical layout is finished, all that remains is to double check whether text overlaps routes. It is good to make sure that there are no loose ends anywhere on the board. Once the board has been reviewed and is completed, the final step is to create the Gerber files. A Gerber file is a file that contains the information from Layout that is necessary for the machines to drill, mill, and cut the PCB. A Gerber file is created for every layer of interest. For example, a top Gerber file defines how to drill, mill, and cut the top of the board. To create the Gerber files, it is as simple as clicking the menu item of the lay out software. By default the software layout is set up to make Gerber files for many layers. For a process of double sided PCB, the created Gerber files are of-

- Top layer
- Bottom layer
- Silk Screen Top layer
- Drill layer
- Board outline layer.

For the board outline layer, we need a Gerber file of a layer containing the board outline. These Gerber files are the physical inputs to the manufacturing process.

Contents of a Gerber File

The content of a Gerber file is ASCII text (i.e. English letters, digits and a few specialcharacters) and looks like what a human could *almost* read and understand. Here's an example:

```
G75*  
G70*  
%OFA0B0*%  
%FSLAX24Y24*%  
%IPPOS*%  
%LPD*%  
%AMOC8*5,1,8,0,0,1.08239X$1,22.5*%  
%ADD10C,0.0080*%  
D10*  
X000281Y000835D02*  
X002472Y006196D01*  
M02*
```

Fig. 2.4.1 (v) Contents of Gerber File

When the format was first created, these commands called out instructions for photo plotter for creating photographic film. The other machines also use Gerber because CAD/EDA/CAM software creates Gerber. Alternative formats like ODB++ exist which require the use of more expensive hardware and software.

RS-274D and RS-274X

There are two "standards" for Gerber files: the "old" and the "new". The old standard (D) spreads information for a single layer across at least two files. The new standard (X) allows all the information for a layer to be contained in a single file. The biggest benefit of this comes down to data management, which means it is much easier to keep track of one file than two files.

The Gerber file instructions are-

- Configuration Parameters ("%...*%" blocks)
- Macro and Aperture Definitions (AM and AD parameters)
- Drawing Commands
- X/Y Coordinate information for the locations of features

Configuration Parameters

Gerber files containing blocks of text starting and ending with percentage symbols ("%...*%") are probably the RS-274X variety called configuration parameters. Configuration parameters control various things in the rendering process, including whether the geometry should be considered with white drawn on black, or black drawn on white.

Macro and Aperture Definitions

Apertures define the thickness of traces, size and shape of pads. Macros (AM parameters) define complex shapes for registration marks, logo, lettering and other special geometry that appears in a design. Macros are a "less often" used feature of Gerber, though many CAD systems employ them for filling the gaps in the original specifications for features such as pad teardrops, rotated rectangles and rectangles with rounded corners. Macros support complex composition rules, and even some equation driven parameters but aren't often used in CAM output. It's worth noting

that one reason manufacturers shy away from the “D” variant Gerber’s (the one with apertures defined in an external file) is that customers often forget to include them with their order. If it happens, the best thing that can happen is that the order will be delayed. At worst, it won’t be delayed and it will be discovered only after assembling components only to find it non-functional.

Drawing Commands

Gerber has three drawing primitives: flashes, lines and arcs with thickness, and polygonfills. It can be a challenge to represent complex geometry using only these shapes. Luckily, RS-274X also adds "Step and Repeat" which allows us to create patterns of features. CAMsystems are clever in the way they create Gerber files. For example, the system may “pour” a large area of black (copper) then draw over it with a "scratch" layer of wide geometry, then finally draw a copper trace down inside the clear. Only three commands are issued, but the result seems more complex.

X/Y Coordinate Information

X Y coordinate information gives information about the position of each and every information to be re-entered. You can probably sympathize with the poor manufacturer. For a "simple" two layer PCB, they can receive up to nine files:

- An image of the copper conductor for the top side of the circuit board.
- An image of the copper conductor for the bottom side of the circuit board.
- An image of where solder mask is to be applied on the top layer.
- An image of where solder mask is to be applied on the bottom layer.
- An image of where silkscreen is to be applied on the top layer
- An image of where silkscreen is to be applied on the bottom layer.
- The locations of the drilled holes, and descriptions of the properties of the holes. Often, PTH and NPTH are separated into different files.

- The outline of the board, including internal cutouts and other machining operations (scoring, slots, etc.)
- A README file with the name and intended use of each of the above files.
- **Viewing and editing Gerber files**

Gerber files sent by the customer for manufacturing need to be viewed and sometimes edited to match with machine parameters. There are software available for viewing and editing. The zip file sent by the client can be opened by this software. The software shows all layers pads, screens etc. and can be edited. You can also use free online Gerber viewer for this purpose.

CHAPTER 3 - INTRODUCTION TO PCB MANUFACTURING

3.1 MANUFACTURING

A variety of processes are currently used for manufacturing PCBs. However most of the processes have identical or similar basic steps. Variations in the basic manufacturing steps are usually made by the manufacturers to improve quality or to make a specific yield. There are four specific phases in the PCB manufacturing process. They are design, fabrication, assembly and test. A PCB is used to connect electronic components electrically. The very first thing required for manufacturing a PCB is a non-conducting material usually called the substrate over which a conducting material like copper is plated. There are different methods in which the non-conducting material is fabricated. The creation of circuit patterns is accomplished using either additive or subtractive method. In additive method, on a purely insulating substrate, the conducting circuit tracks are made. In subtractive process, there will already be a conducting layer over the insulating substrate. Then the copper in the shape of the circuit pattern is retained, while the remaining portion is removed (etched away). The conductive circuit is generally copper, although aluminum, nickel, chrome, and other metals are sometimes used. The conventional process is 'print and etch' method. The term 'printed wiring' or 'printed circuit' refers to the conductive pattern that is formed on the laminate to provide point-to-point connection. Single sided boards have conducting tracks on one side only. Double sided boards connect electronic components on both sides. In multilayer boards, there are several inner layers with conducting tracks but the components will be connected to the outer side only. For making the PCB of a desired circuit, initially the layout of the circuit is to be made. 'Layout' means the picture of the circuit which is converted to a form suitable for making the PCB. The layout can be drawn manually for very simple hobby circuits, but for others, there are software which will do the job. This software is called CAD (Computer Aided Design) software. After the layout is designed using a computer, the processes which follow differs for a fully automated industry environment and a small scale manufacturing unit with lesser automation involved. In both cases, although using different techniques the aim is to transfer the layout on to the board. This involves different processes such as silk screen printing, masking, exposing and developing, etching etc. which results in the required conducting tracks over the base substrate.

Sheet Cutting-

The first thing to do is to cut sheets of desired size. Copper sheets, aluminum foil, and mica sheets are cut of appropriate size. Aluminum foil is used to protect copper sheets from cracks which may occur during drilling. Mica sheet is used to provide base in drilling process. Both aluminum foil and mica sheet are destroyed after drilling is done. After cutting the copper foil laminate sheets to the desired sizes, the first thing to do is to drill holes.



Fig. 3.1 Sheet cutting machine

PCB Drilling-

Drilling is an important process where holes of the required sizes are made. There are holes which need to be conducting as in the case of double sided boards and others which need not be. Hence there are processes which would take care of these. Holes of various sizes are drilled through a stack of panels (usually 2 to 3 high). The locations are determined by the board's designer to fit specific components. Drilled hole size are usually 5 mils larger than finished plated through hole sizes to allow for the copper plating process. These drilling process is done by a CNC (computer numerical controlled) drilling machine. Deburr is an abrasive mechanical process that removes the raised edges of the metal or burrs surrounding the holes that occur during the drilling process. Any debris that may be left in the holes is also removed at this time.

Photo Printing-

An Ultra Violet sensitive material known as photo resist is placed on the copper clad sheet. The photo resist is applied on the copper clad laminate using heat and pressure. The layout prepared by the software is printed on to a film by a process called photo plotting. This film is called a photo mask. This mask is placed above the photo resist which is then exposed to UV Light. The

portion of the photo resist which is exposed to UV light is cured (hardened). Then it is rinsed in the developer bath. Those portions where the photo resist was not exposed to UV light (dark areas) are removed. Now the copper clad laminate is covered with tin-lead where the circuit is required. Using appropriate chemicals the hardened photo resist is removed.

Copper Layer Etching-

Etching is one of the major steps in the chemical processing of subtractive PCB process. By this process, the final copper pattern is achieved by selective removal of all unwanted copper to retain the desired circuit patterns. The copper which is not protected by an etch resist is removed by the etching process. Here the tin lead which was laid onto the copper tracks will act as the etch resist to protect the copper circuitry from being etched. The following are the commonly used etching chemicals.

- Ferric chloride
- Ammonium persulphate
- Chromic acid
- Cupric chloride and
- Alkaline ammonia

Here there are two different methods. Either the photo mask could be such that the dark lines (unexposed areas) define the conducting tracks. This is positive mask. If the conducting tracks are white and the other areas dark like a photo film negative, such masks are called negative masks. Usually positive masks are used for subtractive processes and negative masks for additive processes.

Solder Resist-

The next step is to apply solder to the points where the components are to be connected called solder pads. Other points should be masked from solder. So initially areas which are not to be solderable are covered with a solder resist material. It is basically a polymer coating that prevents the solder from bringing traces and possibly creating shortcuts to nearby component leads. There

are several techniques by which solder can be applied to the solder pads of which HASL (Hot Air Solder Leveling) is a popular technique.

Legend-

Finally, the component names are marked on the board using white color. This is achieved using a printing process called screen printing.

PCB Testing-

Finally the manufactured PCBs are tested. In industrial applications, PCBs are tested by different methods such as Bed of Nails Test, Rigid Needle adaptor, CT scanning test, and so on. The basic of all tests include a computer program which will instruct the electrical test unit to apply a small voltage to each contact point, and verify that a certain voltage appears at the appropriate contact points.

PCB Assembling-

After the testing is complete the PCB is ready for assembly of components which is nothing but the process of connecting the components to the PCB. This can be done either by through-hole construction or surface-mount construction. The common aspect in both the methods is that the component leads are electrically and mechanically fixed to the board with a molten metal solder.

3.2PCB MANUFACTURING PROCESS IN DETAIL

Following is the process of PCB manufacturing-

- Sheet cutting
- Inner layer drilling
- Inner layer PHP
- Inner layer PHP inspection
- Acid etching
- Inner layer dry film stripping

- Inner layer post etch inspection
- Inner layer bond film
- Multi-layer pressing
- Drill FA
- CNC drilling
- PTH plating
- Photo print FA
- Photo printing
- Photo printing inspection
- Pattern plating
- Dry film stripping
- Alkaline etching FA
- Alkaline etching
- Tin stripping
- Post etch inspection
- Solder masking
- Solder mask exposing
- Solder mask inspection
- Gold immersion
- Lead free HAL
- HASL inspection

- Non PTH FA
- Non PTH drilling
- Legend print FA
- Legend printing
- CNC routing FA
- CNC routing
- Final fabrication
- Electrical testing dedicated

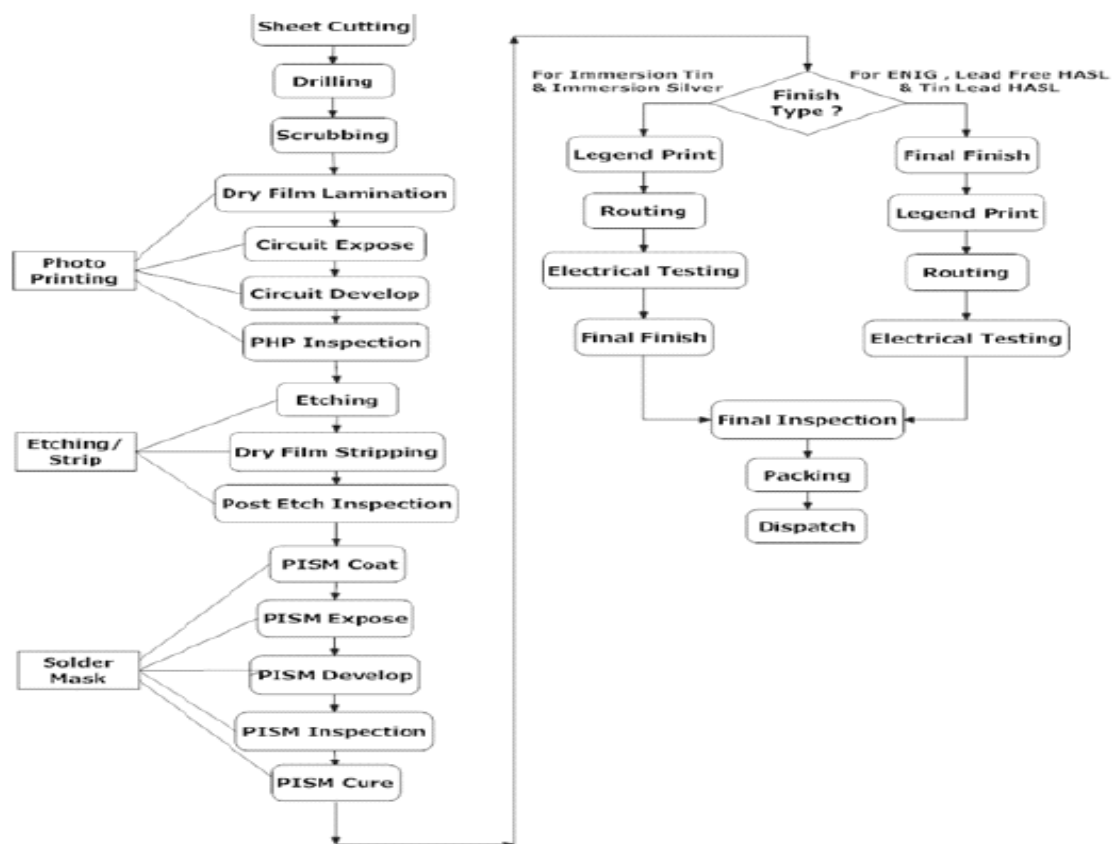


Fig 1.14 Flow chart of a single sided PCB manufacturing process.

Fig. 3.2(i) Flow chart of a single sided PCB manufacturing process

Step 1- From Gerber files to PCB production data

The board designer prepares his layout on a Computer Aided Design or CAD system. Each CAD system uses its own internal data format, so the PCB industry has developed a standard output format to transfer the layout data to the manufacturer. This is Extended Gerber or RS274X. The Gerber files define the copper tracking layers as well as the solder masks and component notations. First step is to check whether the data meets our manufacturing requirements. These checks are mostly done automatically. We have to check the track widths, the space between tracks, the pads around the holes, the smallest hole size etc. The engineer can also check and measure individual areas where he wishes. Once the data is verified as good he will output all the tool files needed to drive the machines that will make and test the PCB.

Step 2- Film Generation

We have to create an exact film representation of the design, from the Gerber files, one film per layer. A laser photo plotter in a temperature and humidity-controlled darkroom is used to make the films which will be used later to image the PCBs. The photo plotter takes the board data and converts it into a pixel image. A laser writes this onto the film. The exposed film is automatically developed and unloaded. The films are ready now for the PCB fabrication process. Films are also developed for various layers and for the solder masks. Now the films are registered (aligned) with each other so that the different layers of the PCB will be perfectly aligned. This is done by punching precise registration holes in each sheet of the film.

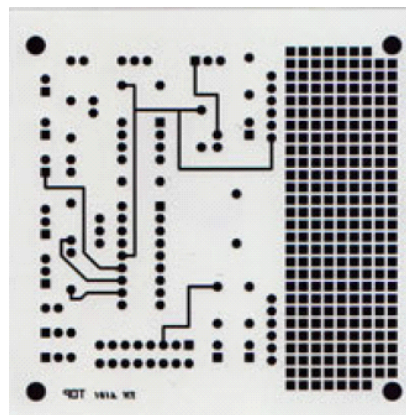


Fig. 3.2 (ii) Film Plotted

The operator puts the film on the table of the punch and then micro-adjusts the table until the targets on the film are exactly lined up with the targets on the film punch. He then punches each sheet of film with the registration holes which will fit onto the registration pins in the imaging equipment.

Step 3- Shear Raw Materials

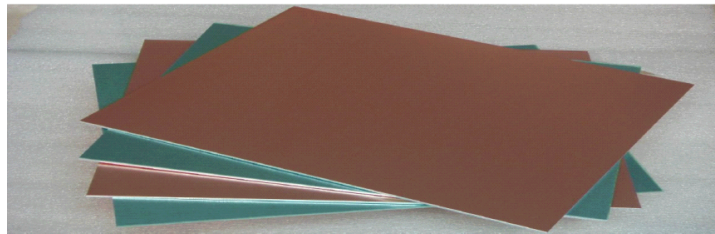


Fig. 3.2 (iii) Copper Foil Laminate

A PCB is used to connect electronic components electrically. The very first thing required formanufacturing a PCB is a non-conducting material usually called the substrate over which aconducting material like copper is plated. There are different methods in which the non-conducting material is fabricated. The creation of circuit patterns is accomplished usingeither additive or subtractive method. In additive method, on a purely insulating substrate,the conducting circuit tracks are made. In subtractive process, there will already be a conducting layer over the insulating substrate as shown in figure above. Then the copper inthe shape of the circuit pattern is retained, while the remaining portion is removed (etchedaway). The conductive circuit is generally copper, although aluminum, nickel, chrome, andother metals are sometimes used. The conventional process is ‘print and etch’ method.The term ‘printed wiring’ or ‘printed circuit’ refers to the conductive pattern that is formed onthe laminate to provide point-to-point connection. Single sided boards have conductingtracks on one side only. Double sided boards connect electronic components on both sides.In multilayer boards, there are several inner layers with conducting tracks but thecomponents will be connected to the outer side only.Industry standard 0.059" thick, double-sided copper clad panels will be sheared (cut) to accommodate many boards. Material used is FR4. The sheared board is cleaned thoroughly and dried in an oven at 110 C for 4 hours. Printed circuit board processing and assembly are done in an extremely clean environment where the air and components can be kept free of contamination. Most electronic manufacturers have their own proprietary processes.

Step 4- Drill Holes

The drill hole information is extracted from the Gerber file. Drilling is an important process where holes of the required sizes are made. There are holes which need to be conducting as in the case of double sided boards and others which need not be. Hence there are processes which would take care of these.



Fig. 3.2 (iv) Drill bits

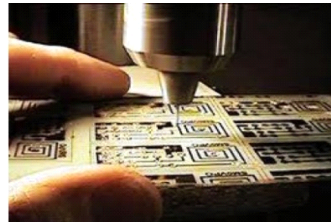


Fig. 3.2 (v) Tip of a CNC drill

Holes of various sizes are drilled through a stack of panels (usually 2 to 3 high). The locations are determined by the board's designer to fit specific components. Drilled hole sizes are usually 5 mils larger than finished plated through hole sizes to allow for the copper plating process. This drilling process is done by a CNC (computer numerical controlled) drilling machine and carbide drills. Deburr is an abrasive mechanical process that removes the raised edges of the metal or burr surrounding the holes that occur during the drilling process. Any debris that may be left in the holes is also removed at this time.

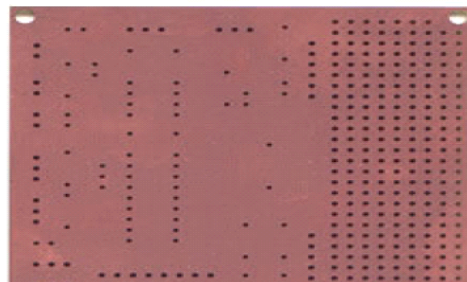


Fig. 3.2 (vi) Holes drilled on laminate

Step 5- Photo-printing

An Ultra Violet sensitive material known as photo resist is placed on the copper clad sheet. The photo resist is applied on the copper clad laminate using heat and pressure. The layout prepared by the software is printed on to a film by a process called photo plotting. This film is called a photo mask. This mask is placed above the photo resist which is then exposed to UV Light. The portion of the photo resist which is exposed to UV light is cured (hardened). Then it is rinsed in the developer bath. Those portions where the photo resist was not exposed to UV light (dark areas) are removed. Now the copper clad laminate is covered with tin-lead where the circuit is required. Using appropriate chemicals the hardened photo resist is removed. The unwanted copper is removed by chemical etching using suitable chemicals as mentioned in the next section.

Step 6- Copper layer etching

Etching is one of the major steps in the chemical processing of subtractive PCB process. By this process, the final copper pattern is achieved by selective removal of all unwanted copper to retain the desired circuit patterns. The copper which is not protected by an etch resist is removed by the etching process. Here the tin-lead which was laid onto the copper tracks will act as the etch resist to protect the copper circuitry from being etched.

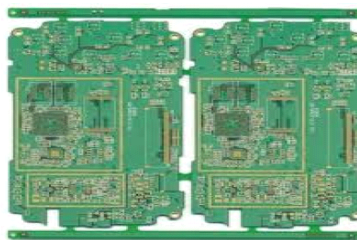


Fig. 3.2 (vii) Green solder resist

The following are the commonly used etching chemicals.

- Ferric chloride
- Ammonium persulphate
- Chromic acid
- Cupric chloride and
- Alkaline ammonia

Here there are two different methods. Either the photo mask could be such that the dark lines (unexposed areas) define the conducting tracks. This is positive mask. If the conducting tracks

are white and the other areas dark like a photo film negative, such masks are called negative masks. Usually positive masks are used for subtractive processes and negative masks for additive processes.

Step 7- Pattern Plating

Pattern plating is an electrochemical process to build copper in the holes and on the trace area followed by application of tin to the surface. There are two fundamental manufacturing processes to make the conductive layers for PCBs. Patterning (etching) - The unwanted copper removed by various methods leaving only the desired copper traces, from the laminate is called subtractive process. You have to remember that a laminate had layer of copper from which the unwanted portion (the portion other than the actual conductive part which is needed called traces) is to be removed. An additive process adds desired copper traces to an insulating substrate. In the full additive process the bare laminate is covered with a photosensitive film which is imaged (exposed to light through a mask and then developed to remove the unexposed film). The exposed areas are sensitized in a chemical bath, usually containing palladium and similar to that used for through hole plating which makes the exposed area capable of bonding metal ions. The laminate is then plated with copper in the sensitized areas. The advantage of the additive method is less pollution of the environment. The method chosen for PCB manufacture depends on the desired number of boards to be produced. Double-sided boards or multi-layer boards use plated-through holes, called via, to connect traces on different layers. The most common is the "semi-additive" process: the unpattern board has a thin layer of copper already on it. A reverse mask is then applied.

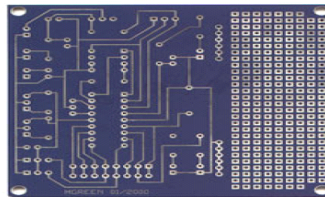


Fig. 3.2 (viii) Laminate after patterning

Additional copper is then plated onto the board in the unmasked areas; copper may be plated to any desired weight. Tin-lead or other surface plating is then applied. The mask is stripped away

and a brief etching step removes the now-exposed bare original copper laminate from the board, isolating the individual traces. The additive process is commonly used for multi-layer boards as it facilitates the plating-through of the holes to produce conductive via in the circuit board.

Step8- Strip & Etch

In this step, dry film is removed, and then exposed copper is etched. The tin protects the copper circuitry from being etched.

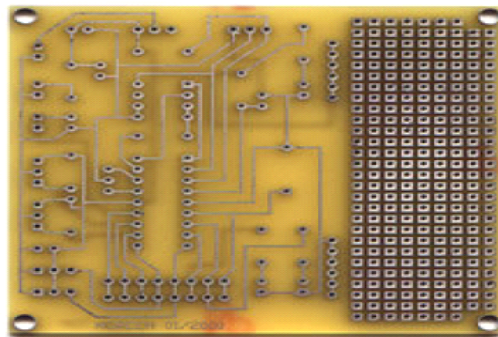


Fig. 3.2 (ix) Laminate after stripping and etching

List of chemicals used for PTH Plating-

- Proprietary Chemicals:
 - Cleaner Securiganth 902
 - Etch Cleaner HS
 - Neoganth B Predip
 - Neoganth V8 Activator
 - Reducer Neoganth WA
 - Makeup Solution Priutoganth 820
 - Additive Solution Priutoganth 820
 - Reduction Solution Cu
 - StabilizerPriutoganth 820

Base Chemicals:

- Sulphuric Acid LR
- Sodium Hydroxide LR
- Boric Acid LR & Hydrogen Peroxide 30-50%

Step 9- Solder mask

In this step, solder mask is applied to entire board with the exception of solder pads.

Step 10- Solder coat Hot Air Solder Leveling (HASL)

The panels are coated with flux –a viscous compound that promotes even coating of the solder. Then the panels are dipped completely into a bath of molten solder. The solder covers all exposed metal surfaces. As the panel is removed from the solder, high pressure hot air is directed at both sides of the panel. The “blast” of air removes excess solder from the holes and smoothes the surface of the pads.

Step11- Legend

In this step, white letter marking is made using screen printing process. The process is same as that for multi-layer.

Step12- Rout, Fabrication, Score, and Bevel

After HASL the boards are cut to size on a CNC machine or router. Most panels have the individual parts routed out into single pieces or arrays of varying sizes. Boards or arrays can also be scored so that they can be easily broken apart after assembly. Then the boards are generally checked for cleanliness, sharp edges, burrs and other fabrication requirements. Chamfers, slots, countersinks and bevels are added during the rout & fabrication processes.

Step13- Bare board electrical test

Boards are tested for opens and shorts in the circuitry, in one of the last steps of production. Test Programs can be loaded directly onto various types of test machines or used to create specific fixtures and test programs. Shorts are repaired when possible and retested for verification. 100% of the networks on the board are tested for continuity and isolation (opens and shorts) using test programs generated from the Gerber data.

Final Inspection- Boards are visually inspected to assure they meet customers' requirements, industry specifications as well as having the physical dimensions and hole sizes verified.