



DEPARTMENT OF ELECTRONIC SCIENCE

K.T.H.M College, Nashik

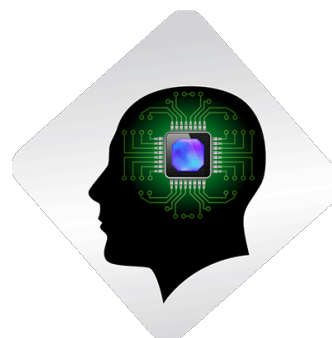
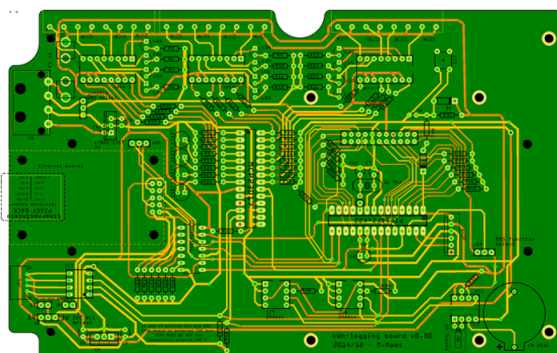
EL 368: Paper VIII: Practical Course II

T.Y.B.Sc. Electronic Science 2021(CBCS Pattern)

Academic Year 2021 -22

PCB MANUFACTURING PROCESS

Group D: ACTIVITY(Project Report)



- NAME: SANJEET KUMAR
- T.Y.B.Sc.(Electronic Science)
- SEMESTER VI
- ROLL NO. 29



Guide Teachers

- 1) Prof. Smt. S.S.Kulkarni
- 2) Prof. Smt. S.K.Jadhav

Abstract :-

A printed circuit board (PCB) is a board made for connecting electronic components together. These are used in almost all computers and electronics today. The "card" is made of a material that does not conduct electricity, usually fiberglass. Usually copper is etched (set in thin lines) inside the board between the layers of fiberglass, or on the surface of the board. This makes the electricity go only where it is wanted. Electronic components are then attached to this board using a metal to conduct electricity. The metal etched into the board allows electricity to travel from one component to another in electrical circuits. Boards can have many different parts which are connected and work together.

The most common circuit boards are made in large numbers for a specific job, for example to run a computer, a mobile/cell phone or a television. Some circuit boards are made plain so a person can build their own for a new electrical task. Most things that use electricity have at least one circuit board inside of them that makes them run.

Flexible circuit boards are those that are made thin enough and of the right material to flex (bend)

INTRODUCTION :-

A printed circuit board (PCB) mechanically supports and electrically connects electrical or electronic components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. Components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it.

Printed circuit boards are used in all but the simplest electronic products. They are also used in some electrical products, such as passive switch boxes.

Alternatives to PCBs include wire wrap and point-to-point construction, both once popular but now rarely used. PCBs require additional design effort to lay out the circuit, but manufacturing and assembly can be automated. Electronic computer-aided design software is available to do much of the work of layout. Mass-producing circuits with PCBs is cheaper and faster than with other wiring methods, as components are mounted and wired in one operation.

Large numbers of PCBs can be fabricated at the same time, and the layout only has to be done once. PCBs can also be made manually in small quantities, with reduced benefits. PCBs can be single-sided (one copper layer), double-sided (two copper layers on both sides of one substrate layer), or multi-layer (outer and inner layers of copper, alternating with layers of substrate). Multi-layer PCBs allow for much higher component density, because circuit traces on the inner layers would otherwise take up surface space between components.

The rise in popularity of multilayer PCBs with more than two, and especially with more than four, copper planes was concurrent with the adoption of surface mount technology. However, multilayer PCBs make repair, analysis, and field modification of circuits much more difficult and usually impractical.

1The world market for bare PCBs exceeded \$60.2 billion in 2014.[1] In 2018, the Global Single Sided Printed Circuit Board Market Analysis Report estimated that the PCB market would reach \$79 billion by 2024.[2][3]

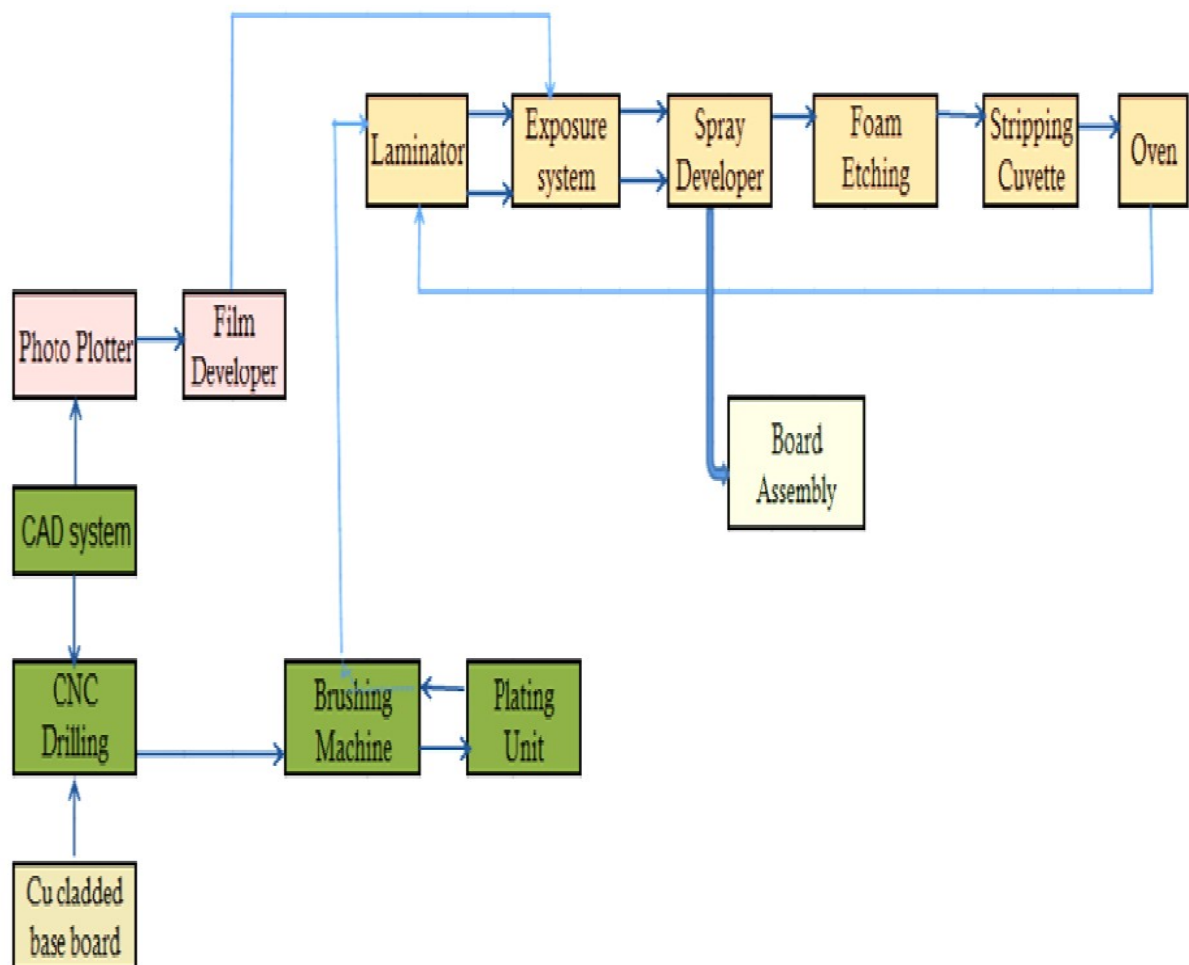
TYPES OF PCB'S :-

Printed circuit boards are electronic circuits boards created for mounting electronic components on a nonconductive board, and for creating conductive connections between them. The creation of circuit patterns is accomplished using both additive and subtractive methods. The conductive circuit is generally copper, although aluminum, nickel, chrome, and other metals are sometimes used. There are three basic varieties of printed circuit boards: single-sided, double-sided, and multi-layered.

1. Single sided PCB: conductors on only one surface of a dielectric base.
2. Double sided PCB: conductors on both sides of a dielectric base, usually the two layers are interconnected by plated-through-holes (PTHs).
3. Multi-layer: conductors on 3 or more layers separated by dielectric material and the layers are interconnected by PTH or pads
4. layer PCB is a sandwich of 2 double layered PCBs. Likewise 6 layer PCB is a sandwich of 3 double layered PCB. This sandwiching is done by placing oxidizing material between double layered PCBs.

The spatial and density requirement, and the circuitry complexity determine the type of board to be produced.

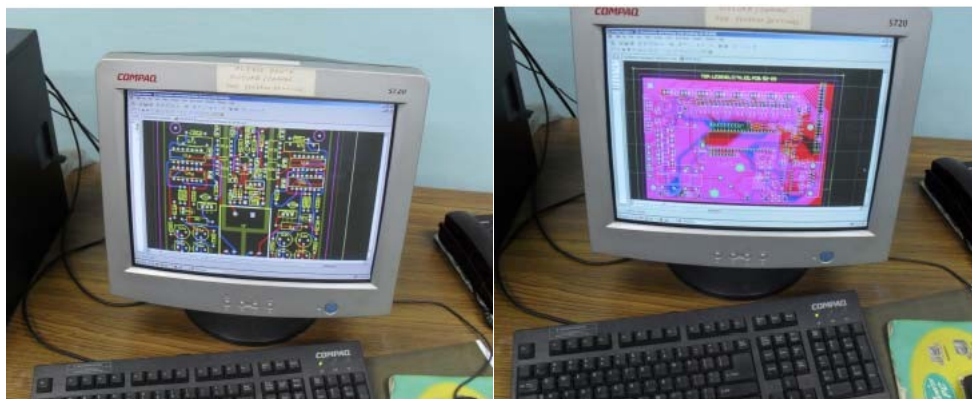
PCB MANUFACTURING PROCESS BLOCK DAIGRAM :-



PCB CAM /CAD

Manufacturing starts from the fabrication data generated by computer aided design, and component information. The fabrication data is read into the CAM (Computer Aided Manufacturing) software. CAM performs the following functions:

1. Input of the fabrication data.
2. Verification of the data
3. Compensation for deviations in the manufacturing processes (e.g. scaling to compensate for distortions during lamination)
4. Panelization
5. Output of the digital tools (copper patterns, drill files, inspection, and others)



Panelization :-

Several small printed circuit boards can be grouped together for processing as a panel. A panel consisting of a design duplicated n-times is also called an n-panel, whereas a multi-panel combines several different designs onto a single panel. The outer tooling strip often includes tooling holes, a set of panel fiducials, a test coupon, and may include hatched copper pour or similar patterns for even copper distribution over the whole panel in order to avoid bending.

The assemblers often mount components on panels rather than single PCBs because this is efficient. Panelization may also be necessary for boards with components placed near an edge of the board because otherwise the board could not be mounted during assembly. Most assembly shops require a free area of at least 10 mm around the board.

The panel is eventually broken into individual PCBs along perforations or grooves in the panel [19] through milling or cutting. For milled panels a common distance between the individual boards is 2 to 3 mm. Today depaneling is often done by lasers which cut the board with no contact. Laser depaneling reduces stress on the fragile circuits, improving the yield of defect-free units. 2Copper patterning The first step is to replicate the pattern in the fabricator's CAM system on a protective mask on the copper foil PCB layers. Subsequent etching removes the unwanted copper. (Alternatively, a conductive ink can be ink-jetted on a blank (non-conductive) board. This technique is also used in the manufacture of hybrid circuits.)

1. Silk screen printing uses etch-resistant inks to create the protective mask.
2. Photoengraving uses a photomask and developer to selectively remove a UV-sensitive photoresist coating and thus create a photoresist mask. Direct imaging techniques are

sometimes used for high-resolution requirements. Experiments were made with thermal resist.[20] A laser may be used instead of a photomask. This is known as maskless lithography or direct imaging. [21]

3. PCB milling uses a two or three-axis mechanical milling system to mill away the copper foil from the substrate. A PCB milling machine (referred to as a 'PCB Prototyper') operates in a similar way to a plotter, receiving commands from the host software that control the position of the milling head in the x, y, and (if relevant) z axis.

4. Laser resist ablation Spray black paint onto copper clad laminate, place into CNC laser plotter. The laser raster-scans the PCB and ablates (vaporizes) the paint where no resist is wanted. (Note: laser copper ablation is rarely used and is considered experimental.

5. Laser etching The copper may be removed directly by a CNC laser. Like PCB milling above this is used mainly for prototyping.

The method chosen depends on the number of boards to be produced and the required resolution.

Large volume

- Silk screen printing – Used for PCBs with bigger features
- Photoengraving – Used when finer features are required

Small volume

- Print onto transparent film and use as photo mask along with photo-sensitized boards, then etch. (Alternatively, use a film photoplotter)
- Laser resist ablation
- PCB milling
- Laser etching

Hobbyist

- Laser-printed resist: Laser-print onto toner transfer paper, heat-transfer with an iron or modified laminator onto bare laminate, soak in water bath, touch up with a marker, then etch.
- Vinyl film and resist, non-washable marker, some other methods. Labor-intensive, only suitable for single boards

Subtractive, additive and semi-additive processes :-

Subtractive methods remove copper from an entirely copper-coated board to leave only the desired copper pattern. In additive methods the pattern is electroplated onto a bare substrate using a complex process. The advantage of the additive method is that less material is needed and less waste is produced. 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 which removes 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. When the mask is stripped, the PCB is finished. Semi-additive is the most common process: The unpatterned board has a thin layer of copper already on it. A reverse mask is then applied. (Unlike a subtractive process mask, this mask exposes those parts of the substrate that will eventually become the traces.) 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 platings are 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. Some single-sided boards which have plated-through holes are made in this way. General Electric made consumer radio sets in the late 1960s using additive

boards. The (semi-)additive process is commonly used for multi-layer boards as it facilitates the plating-through of the holes to produce conductive vias in the circuit board.

CHEMICAL ETCHING :-

Chemical etching is usually done with ammonium persulfate or ferric chloride. For PTH (plated-through holes), additional steps of electroless deposition are done after the holes are drilled, then copper is electroplated to build up the thickness, the boards are screened, and plated with tin/lead. The tin/lead becomes the resist leaving the bare copper to be etched away.[22]

The simplest method, used for small-scale production and often by hobbyists, is immersion etching, in which the board is submerged in etching solution such as ferric chloride. Compared with methods used for mass production, the etching time is long. Heat and agitation can be applied to the bath to speed the etching rate. In bubble etching, air is passed through the etchant bath to agitate the solution and speed up etching. Splash etching uses a motor-driven paddle to splash boards with etchant; the process has become commercially obsolete since it is not as fast as spray etching. In spray etching, the etchant solution is distributed over the boards by nozzles, and recirculated by pumps. Adjustment of the nozzle pattern, flow rate, temperature, and etchant composition gives predictable control of etching rates and high production rates.[23]

As more copper is consumed from the boards, the etchant becomes saturated and less effective; different etchants have different capacities for copper, with some as high as 150 grams of copper per litre of solution. In commercial use, etchants can be regenerated to restore their activity, and the dissolved copper recovered and sold. Small-scale etching requires attention to disposal of used etchant, which is corrosive and toxic due to its metal content.[24]

The etchant removes copper on all surfaces not protected by the resist. "Undercut" occurs when etchant attacks the thin edge of copper under the resist; this can reduce conductor widths and cause open-circuits. Careful control of etch time is required to prevent undercut. Where metallic plating is used as a resist, it can "overhang" which can cause short-circuits between adjacent traces when closely spaced. Overhang can be removed by wire-brushing the board after etching.



LAMINATION :-

Multi-layer printed circuit boards have trace layers inside the board. This is achieved by laminating a stack of materials in a press by applying pressure and heat for a period of time. This results in an inseparable one piece product. For example, a four-layer PCB can be fabricated by starting from a two-sided copper-clad laminate, etch the circuitry on both sides, then laminate to the top and bottom pre-preg and copper foil. It is then drilled, plated, and etched again to get traces on top and bottom layers.[25]

The inner layers are given a complete machine inspection before lamination because mistakes cannot be corrected afterwards. Automatic optical inspection (AOI) machines compare an image of the board with the digital image generated from the original design data. Automated Optical Shaping (AOS) machines can then add missing copper or remove excess copper using a laser, reducing the number of PCBs that have to be discarded.[26] PCB tracks can have a width of just 10 micrometers.



DRILLING :-

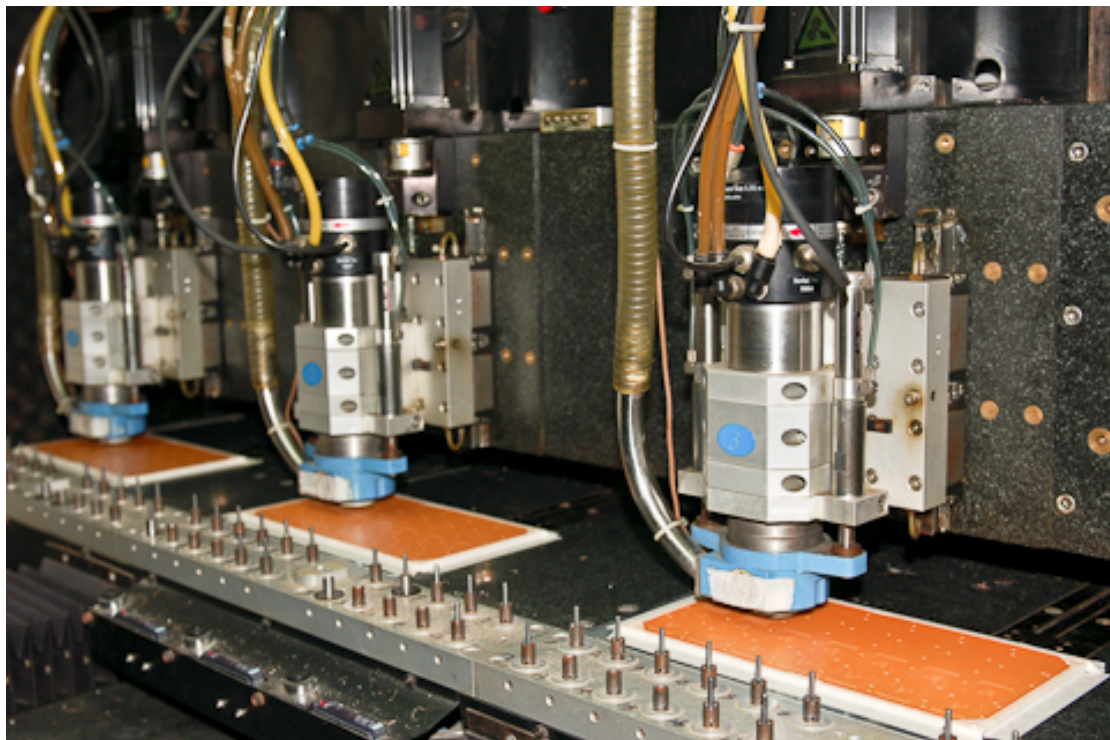
Holes through a PCB are typically drilled with drill bits made of solid coated tungsten carbide. Coated tungsten carbide is used because board materials are abrasive. High-speed-steel bits would dull quickly, tearing the copper and ruining the board. Drilling is done by computer-controlled drilling machines, using a drill file or Excellon file that describes the location and size of each drilled hole.

Holes may be made conductive, by electroplating or inserting hollow metal eyelets, to connect board layers. Some conductive holes are intended for the insertion of through-hole-component leads. Others used to connect board layers, are called vias.

When vias with a diameter smaller than 76.2 micrometers[27][28] are required, drilling with mechanical bits is impossible because of high rates of wear and breakage. In this case, the vias may be laser drilled—evaporated by lasers. Laser-drilled vias typically have an inferior surface finish inside the hole. These holes are called micro vias and can have diameters as small as 10 micrometers.[29][30] It is also possible with controlled-depth drilling, laser drilling, or by pre drilling the individual sheets of the PCB before lamination, to produce holes that connect only some of the copper layers, rather than passing through the entire

board. These holes are called blind vias when they connect an internal copper layer to an outer layer, or buried vias when they connect two or more internal copper layers and no outer layers. Laser drilling machines can drill thousands of holes per second and can use either UV or CO₂ lasers.

The hole walls for boards with two or more layers can be made conductive and then electroplated with copper to form plated-through holes. These holes electrically connect the conducting layers of the PCB. For multi-layer boards, those with three layers or more, drilling typically produces a smear of the high temperature decomposition products of bonding agent in the laminate system. Before the holes can be plated through, this smear must be removed by a chemical de-smear process, or by plasma-etch. The de-smear process ensures that a good connection is made to the copper layers when the hole is plated through. On high reliability boards a process called etch-back is performed chemically with a potassium permanganate based etchant or plasma. The etch-back removes resin and the glass fibers so that the copper layers extend into the hole and as the hole is plated become integral with the deposited copper.



PLATING AND COATING :-

Proper plating or surface finish selection can be critical to process yield, the amount of rework, field failure rate, and reliability.

PCBs are plated with solder, tin, or gold over nickel and a resist for etching away the unneeded underlying copper.[36][37]

After PCBs are etched and then rinsed with water, the solder mask is applied, and then any exposed copper is coated with solder, nickel/gold, or some other anti-corrosion coating.[38] Matte solder is usually fused to provide a better bonding surface for bare copper. Treatments, such as benzimidazolethiol, prevent surface oxidation of bare copper. The places to which components will be mounted are typically plated, because untreated bare copper oxidizes quickly, and therefore is not readily solderable. Traditionally, any exposed copper was coated with solder by hot air solder levelling (HASL).

The HASL finish prevents oxidation from the underlying copper, thereby guaranteeing a solderable surface. This solder was a tin-lead alloy, however new solder compounds are now used to achieve compliance with the RoHS directive in the EU, which restricts the use of lead. One of these lead-free compounds is SN100CL, made up of 99.3% tin, 0.7% copper, 0.05% nickel, and a nominal of 60 ppm germanium.[citation needed]

It is important to use solder compatible with both the PCB and the parts used. An example is ball grid array (BGA) using tin-lead solder balls for connections losing their balls on bare copper traces or using lead-free solder paste.

Other platings used are OSP (organic surface protectant), immersion silver (IAG), immersion tin, electroless nickel with immersion gold coating (ENIG), electroless nickel electroless palladium immersion gold (ENEPIG) and direct gold plating (over nickel). Edge connectors, placed along one edge of some boards, are often nickel-plated then gold-plated. Another coating consideration is rapid diffusion of coating metal into tin solder. Tin forms intermetallics such as Cu_6Sn_5 and Ag_3Cu that dissolve into the Tin liquidus or solidus (at 50°C), stripping surface coating or leaving voids.

Electrochemical migration (ECM) is the growth of conductive metal filaments on or in a printed circuit board (PCB) under the influence of a DC voltage bias.[39][40] Silver, zinc, and aluminum are known to grow whiskers under the influence of an electric field. Silver also grows conducting surface paths in the presence of halide and other ions, making it a poor choice for electronics use. Tin will grow "whiskers" due to tension in the plated surface. Tin lead or solder plating also grows whiskers, only reduced by reducing the percentage of tin. Reflow to melt solder or tin plate to relieve surface stress lowers whisker incidence. Another coating issue is tin pest, the transformation of tin to a powdery allotrope at low temperature.[41]



Solder resist application :-

Areas that should not be soldered may be covered with solder resist (solder mask). One of the most common solder resists used today is called "LPI"

2(liquid photoimageable solder mask).[42] A photo-sensitive coating is applied to the surface of the PWB, then exposed to light through the solder mask image film, and finally developed where the unexposed areas are washed away. Dry film solder mask is similar to the dry film used to image the PWB for plating or etching. After being laminated to the PWB surface it is imaged and developed as LPI. Once but no longer commonly used, because of its low accuracy and resolution, is to screen print epoxy ink.

In addition to repelling solder, solder resist also provides protection from the environment to the copper that would otherwise be exposed.



SILK-SCREEN PAINTING :-

A legend is often printed on one or both sides of the PCB. It contains the component designators, switch settings, test points and other indications helpful in assembling, testing, servicing, and sometimes using the circuit board.

There are three methods to print the legend.

1. Silk screen printing epoxy ink was the established method. It was so common that legend is often misnamed silk or silkscreen.
2. Liquid photo imaging is a more accurate method than screen printing.
3. Ink jet printing is new but increasingly used. Ink jet can print variable data, unique to each PWB unit, such as text or a bar code with a serial number.

BARE-BOARD TEST :-

Boards with no components installed are usually bare-board tested for "shorts" and "opens". This is called electrical test or PCB e-test. A short is a connection between two points that should not be connected. An open is a missing connection between points that should be connected. For high-volume production, a fixture or a rigid needle adapter makes contact with copper lands on the board. The fixture or adapter is a significant fixed cost and this method is only economical for high-volume or high-value production. For small or medium volume production flying probe testers are used where test probes are moved over the board by an XY drive to make contact with the copper lands. There is no need for a fixture and hence the fixed costs are much lower. The CAM system instructs the electrical tester to apply a voltage to each contact point as required and to check that this voltage appears on the appropriate contact points and only on these.

ASSEMBLY :-

In assembly the bare board is populated (or "stuffed") with electronic components to form a functional printed circuit assembly (PCA), sometimes called a "printed circuit board assembly" (PCBA).

In through-hole technology, the component leads are inserted in holes surrounded by conductive pads; the holes keep the components in place. In surface-mount technology (SMT), the component is placed on the PCB so that the pins line up with the conductive pads or lands on the surfaces of the PCB; solder paste, which was previously applied to the pads, holds the components in place temporarily; if surface-mount components are applied to both sides of the board, the bottom-side components are glued to the board. In both through hole and surface mount, the components are then soldered; once cooled and solidified, the solder holds the components in place permanently and electrically connects them to the board.

There are a variety of soldering techniques used to attach components to a PCB. High volume production is usually done with a pick-and-place machine and bulk wave soldering or reflow ovens, but skilled technicians are able to hand-solder very tiny parts (for instance 0201 packages which are 0.02 in. by 0.01 in.)[45] under a microscope, using tweezers and a fine-tip soldering iron, for small volume prototypes. Selective soldering may be used for delicate parts. Some SMT parts cannot be soldered by hand, such as BGA packages. All through-hole components can be hand soldered, making them favored for prototyping where size, weight, and the use of the exact components that would be used in high volume production are not concerns

Often, through-hole and surface-mount construction must be combined in a single assembly because some required components are available only in surface-mount packages, while others are available only in through-hole packages. Or, even if all components are available in through-hole packages, it might be desired to take advantage of the size, weight, and cost reductions obtainable by using some available surface-mount devices. Another reason to use both methods is that through-hole mounting can provide needed strength for components likely to endure physical stress (such as connectors that are frequently mated and demated or that connect to cables expected to impart substantial stress to the PCB-and-connector interface), while components that are expected to go untouched will take up less space using surface mount techniques. For further comparison, see the SMT page.

After the board has been populated it may be tested in a variety of ways:

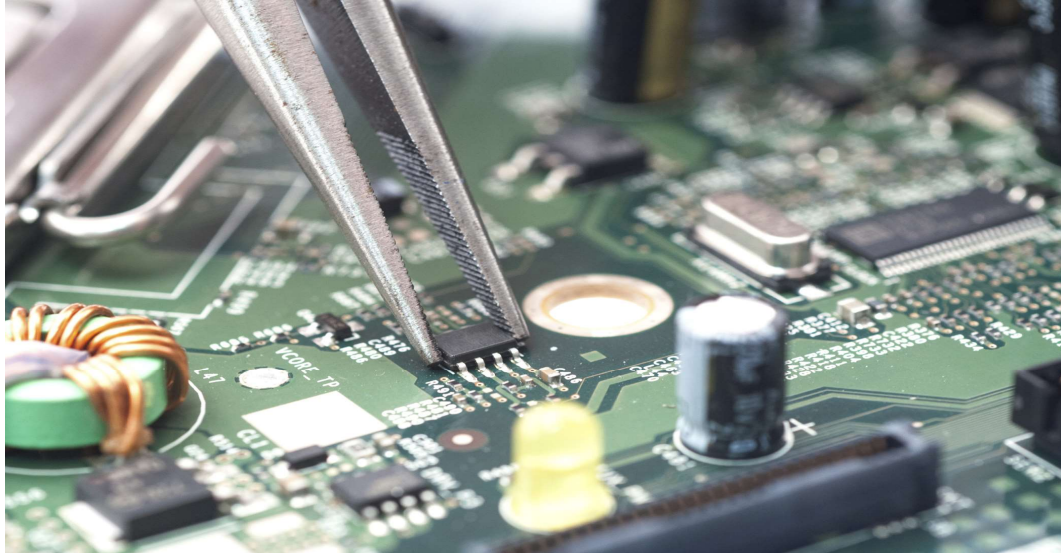
- While the power is off, visual inspection, automated optical inspection. JEDEC guidelines for PCB component placement, soldering, and inspection are commonly used to maintain quality control in this stage of PCB manufacturing.
- While the power is off, analog signature analysis, power-off testing.
- While the power is on, in-circuit test, where physical measurements (for example, voltage) can be done.
- While the power is on, functional test, just checking if the PCB does what it had been designed to do.

To facilitate these tests, PCBs may be designed with extra pads to make temporary connections. Sometimes these pads must be isolated with resistors. The in-circuit test may also exercise boundary scan test features of some components. In-circuit test systems may also be used to program nonvolatile memory components on the board.

In boundary scan testing, test circuits integrated into various ICs on the board form temporary connections between the PCB traces to test that the ICs are mounted correctly. Boundary scan testing requires that all the ICs to be tested use a standard test configuration procedure, the most common one being the Joint Test Action Group (JTAG) standard.

The JTAG test architecture provides a means to test interconnects between integrated circuits on a board without using physical test probes, by using circuitry in the ICs to employ the IC pins themselves as test probes. JTAG tool vendors provide various types of stimuli and sophisticated algorithms, not only to detect the failing nets, but also to isolate the faults to specific nets, devices, and pins.

When boards fail the test, technicians may desolder and replace failed components, a task known as rework.



PROTECTION AND PACKAGING :-

PCBs intended for extreme environments often have a conformal coating, which is applied by dipping or spraying after the components have been soldered. The coat prevents corrosion and leakage currents or shorting due to condensation. The earliest conformal coats were wax; modern conformal coats are usually dips of dilute solutions of silicone rubber, polyurethane, acrylic, or epoxy. Another technique for applying a conformal coating is for plastic to be sputtered onto the PCB in a vacuum chamber. The chief disadvantage of conformal coatings is that servicing of the board is rendered extremely difficult.[46]

Many assembled PCBs are static sensitive, and therefore they must be placed in antistatic bags during transport. When handling these boards, the user must be grounded (earthed). Improper handling techniques might transmit an accumulated static charge through the board, damaging or destroying components. The damage might not immediately affect function but might lead to early failure later on, cause intermittent operating faults, or cause a narrowing of the range of environmental and electrical conditions under which the board functions properly. Even bare boards are sometimes static sensitive: traces have become so fine that it is possible to blow a trace (or change its characteristics) with a static discharge. This is especially true on non-traditional PCBs such as MCMs and microwave PCBs.



RESULT OF REPORT :-

A basic PCB consists of a flat sheet of insulating material and a layer of copper foil, laminated to the substrate. Chemical etching divides the copper into separate conducting lines called tracks or circuit traces, pads for connections, vias to pass connections between layers of copper, and features such as solid conductive areas for electromagnetic shielding or other purposes. The tracks function as wires fixed in place, and are insulated from each other by air and the board substrate material. The surface of a PCB may have a coating that protects the copper from corrosion and reduces the chances of solder shorts between traces or undesired electrical contact with stray bare wires. For its function in helping to prevent solder shorts, the coating is called solder resist or solder mask.

A printed circuit board can have multiple copper layers. A two-layer board has copper on both sides; multi layer boards sandwich additional copper layers between layers of insulating material. Conductors on different layers are connected with vias, which are copper-plated holes that function as electrical tunnels through the insulating substrate. Through-hole component leads sometimes also effectively function as vias. After two-layer PCBs, the next step up is usually four-layer. Often two layers are dedicated as power supply and ground planes, and the other two are used for signal wiring between components. "Through hole" components are mounted by their wire leads passing through the board and soldered to traces on the other side. "Surface mount" components are attached by their leads to copper traces on the same side of the board. A board may use both methods for mounting components. PCBs with only through-hole mounted components are now uncommon. Surface mounting is used for transistors, diodes, IC chips, resistors and capacitors. Through-hole mounting may be used for some large components such as electrolytic capacitors and connectors.

The pattern to be etched into each copper layer of a PCB is called the "artwork". The etching is usually done using photoresist which is coated onto the PCB, then exposed to light projected in the pattern of the artwork. The resist material protects the copper from dissolution into the etching solution. The etched board is then cleaned. A PCB design can be mass-reproduced in a way similar to the way photographs can be mass-duplicated from film negatives using a photographic printer.

In multi-layer boards, the layers of material are laminated together in an alternating sandwich: copper, substrate, copper, substrate, copper, etc.; each plane of copper is etched, and any internal vias (that will not extend to both outer surfaces of the finished multilayer board) are plated-through, before the layers are laminated together. Only the outer layers need be coated; the inner copper layers are protected by the adjacent substrate layers.

A minimal PCB for a single component, used for prototyping, is called a breakout board. The purpose of a breakout board is to "break out" the leads of a component on separate terminals so that manual connections to them can be made easily. Breakout boards are especially used for surface-mount components or any components with fine lead pitch. Advanced PCBs may contain components embedded in the substrate.

References :-

- I. "iconnect007 :: Article". www.iconnect007.com. Retrieved 2016-04-12.
- II. Research, Energias Market. "Global Printed Circuit Board (PCB) Market to Witness a CAGR of 3.1% during 2018-2024". GlobeNewswire News Room. Retrieved 2018-08-26.
- III. "Global Single Sided Printed Circuit Board Market - Growth, Future Prospects and Competitive Analysis and Forecast 2018 - 2023 - The Industry Herald". The Industry Herald. 2018-08-21. Retrieved 2018-08-26.

IV. Electronic Packaging:Solder Mounting Technologies in K.H. Buschow et al (ed),
Encyclopedia of Materials: Science and Technology, Elsevier, 2001 ISBN 0-08-043152-6,
pages 2708–2709

V. <https://www.youtube.com/watch?v=gqBTGupyquo>

(Science to Technology)

VI. <https://www.youtube.com/watch?v=gMxVK3h5K1g>

(Vishal Rajor Electrical)

VII. <https://www.youtube.com/watch?v=df16r2JRiJg>

(Learn EEE)