ISSN 2229-5518

PREPARATION AND TESTING OF A PRINTED CIRCUIT BOARD AND ITS VARIOUS APPLICATION IN THE WORLD OF ENGINEERING

Anirban Chakraborty,
Gopal Karmakar,
Joyita Chakraborty

Abstract-- This article shows the scope and limits of PCB designing as a tool for electronic circuits. Also, illustrations are given as to how the PCB's are actually made in the industry. The software which goes behind making the PCB is also briefly presented in the article.

Index Terms— PCB, Etching, surface mounting, Schematic, through hole components, soldering, Eagle software.

———————————

INTRODUCTION: A **printed circuit board** (**PCB**) mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate. PCBs can be *single sided* (one copper layer), *double sided* (two copper layers) or *multi-layer* (outer and inner layers). Multi-layer PCBs allow for much higher component density. Conductors on different layers are connected with plated-through holes called vias. Advanced PCBs may contain components - capacitors, resistors or active devices - embedded in the substrate.

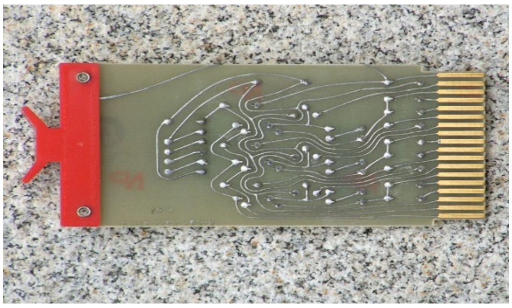
Design: Initially PCBs were designed manually by creating a photomask on a clear mylar sheet, usually at two or four times the true size. Starting from the schematic diagram the component pin pads were laid out on the mylar and then traces were routed to connect the pads. Rub-on dry transfers of common component footprints increased efficiency. Traces were

ISSN 2229-5518

made with self-adhesive tape. Pre-printed non-reproducing grids on the mylar assisted in layout. To fabricate the board, the finished photomask was photolithographically reproduced onto a photoresist coating on the blank copper-clad boards

Modern PCBs are designed with dedicated layout software, generally in the following steps:

1. Schematic capture through an electronic design automation (*EDA*) tool.
2. Card dimensions and template are decided based on required circuitry and case of the PCB.
3. The positions of the components and heat sinks are determined.
4. Layer stack of the PCB is decided, with one to tens of layers depending on complexity. Ground and power planes are decided. A power plane is the counterpart to a ground plane and behaves as an AC signal ground while providing DC power to the circuits mounted on the PCB.



A board designed in 1967; the sweeping curves in the traces are evidence of freehand design using self-adhesive tape.

Manufacturing: there are various steps by which a PCB layout is formed.

PCB CAM:

Manufacturing starts from the PCB fabrication data generated by CAD: Gerber layer images, Gerber or Excellon drill files, IPC-D-356 netlist and component information. The Gerber or Excellon files in the fabrication data are never used directly on the manufacturing equipment but always read into the CAM (Computer Aided Manufacturing) software. CAM performs the following functions:

1. Input of the fabrication data.
2. Verification of the data; optionally DFM
3. Compensation for deviations in the manufacturing processes (e.g. scaling to compensate for distortions during lamination)
4. Panelization

Panelization: Panelization is a procedure whereby a number of PCBs are grouped for manufacturing onto a larger board - the panel. Usually a panel consists of a single design but sometimes multiple designs are mixed on a single panel. There are two types of panels:

assembly panels - often called arrays - and bare board manufacturing panels. The assemblers often mount components on panels rather than single PCBs because this is efficient. The bare board manufactures always uses panels, not only for efficiency, but because of the requirements the plating process.

Copper 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.
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 z axis.

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.

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,

Plating and coating

PCBs are plated with solder, tin, or gold over nickel as a resist for etching away the unneeded underlying copper.

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.

Matte solder is usually fused to provide a better bonding surface or stripped to 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

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" (liquid photoimageable solder mask). 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.

Bare-board test

Unpopulated boards are usually *bare-board tested* for "shorts" and "opens". 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 is used to make contact with copper lands on the board. Building the adapter is a significant fixed cost and 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.

Assembly: In assembly the bare board is populated with electronic components to form a functional *printed circuit assembly* (PCA), sometimes called a "printed circuit board assembly" (PCBA). In through-hole technology component leads are inserted in holes. In surface-mount technology (SMT) the components are glued on *pads* or *lands* on the surfaces of the PCB. In both component leads are then mechanically fixed and electrically connected to the board by soldering.

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

International Journal of Scientific & Engineering Research, Volume 7, Issue 4, April-2016
ISSN 2229-5518

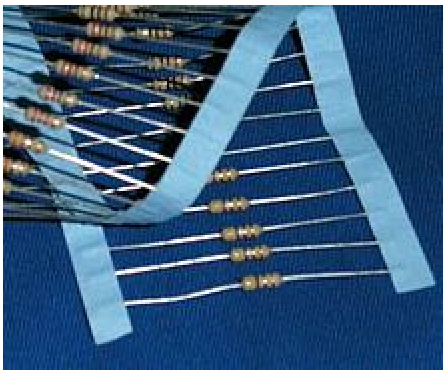
for PCB component placement, soldering, and inspection are commonly used to maintain quality control in this stage of PCB manufacturing.

- 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.

PCB characteristics:

Much of the electronics industry's PCB design, assembly, and quality control follows standards published by the IPC organization.

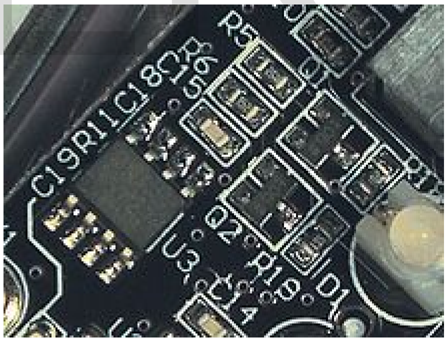
Through-hole technology:



Through-hole (leaded) resistors

The first PCBs used through-hole technology, mounting electronic components by leads inserted through holes on one side of the board and soldered onto copper traces on the other side. Boards may be single-sided, with an unplated component side, or more compact double-sided boards, with components soldered on both sides. Horizontal installation of through-hole parts with two axial leads (such as resistors, capacitors, and diodes) is done by bending the leads 90 degrees in the same direction, inserting the part in the board (often bending leads located on the back of the board in opposite directions to improve the part's mechanical strength), soldering the leads, and trimming off the ends. Leads may be soldered either manually or by a wave soldering machine

Surface-mount technology



Surface mount components, including resistors, transistors and an integrated circuit

Surface-mount technology emerged in the 1960s, gained momentum in the early 1980s and became widely used by the mid-1990s. Components were mechanically redesigned to have small metal tabs or end caps that could be soldered directly onto the

PCB surface, instead of wire leads to pass through holes. Components became much smaller and component placement on both sides of the board became more common than with through-hole mounting, allowing much smaller PCB assemblies with much higher circuit densities.

Advantages &Application:

1) The circuit board fabrication cost (pcb cost) is lower with mass quantity production.
2) Electronic circuit characteristics will be maintained without introducing parasite capacitance with a proper circuit board design.
3) Component wiring and assembly can be mechanized in a circuit board manufacturing facility.
4) PCB's offer uniformity of electrical characteristics from assembly to
assembly.
5) The location of electronic parts is fixed and so it simplifies components identification and maintenance of equipment.

Conclusion: it has already been proved that connections on PCB's are way better than wired connections that we use while testing circuits on bread boards. The wired connections for complex circuits use up a lot of space and error finding and correcting becomes a lot harder. PCB's that we use nowadays are fast taking up the industry with its compact design and easy to handle nature. Also its advantage is that it can be easily replaced, defective components can be put aside and new components can take their place easily, since there are already dedicated paths connecting them. Another fact to be taken into consideration is that once the layout has been etched on the board, placing the components and fixing them on the board is relatively easy and hassle free. It is more like placing the missing parts of a jigsaw puzzle in their right places!

Soldering the components on a Vero board requires a lot of precision job, because if the soldering goes wrong or if we apply more solder accidentally then there is a chance that the circuit connection may get shorted. But this problem does not quite arise while soldering components on a PCB. But that does not mean that we can just solder blindly! It also requires precision, no doubt, but much less than that of a Vero board.

All things may be full of advantages but they are never perfect. They should, and do have limitations. Similarly in that respect, PCB's may have a lot of advantages over Vero boards and bread boards but that is not always enough to mask their limitations. When it comes to space requirement between Vero boards, bread boards and PCB's, surely the latter would emerge with flying colours. But even then, sometimes the problem persists when the circuit becomes complex and a lot more components fight for space. Components like resistors, low power diodes, potentiometers etc. Do not usually occupy a huge amount of space, but high power rectifiers, electrolytic capacitors of the order of 100 microfarads or more tend to occupy more amount of space on board. Capacitors specially protrude out from the board, which can make the availability of space, a challenging task, especially if the board is going to be used in miniature products like processors, smart phones and the like.

Problems of this nature can be dealt with only if the components are 'embedded' on the board. Testing and implementation of this process is already in full swing by the scientists and by the end of another decade or so, we might have another revolution in the electronics technology, which would announce another leap towards the future for mankind.