**DESIGNING OF PCB: PRINTED CIRCUIT BOARD (PCB):**

In electronic system it would be virtually impossible for package without in corpora tin printed circuit in there design. Printed circuit is metal foil conducting pattern serves as the connection medium for the electronic medium that are assumable on the opposite side of the board.

Conducting materials available are silver, brass, aluminum, & copper. Copper is most widely used. The thickness of conducting material depends upon the current carrying capacity of circuit thus a thicker copper layer will have more current carrying capacity,

1. It provides mechanical support for the components mounted on it.

2. It provides necessary electrical interconnection.

3. It acts as the heat sink i.e. it provides a conduction path leading to removal of most of the heat generated in the ckt.

ADVANTAGES OF PCB:

* When a number of identical assemblies are required PCBs provide cost saving because once a layout is approved there is no need to check the circuit every time.
* For large equipments such as computer the saving on checking connections or wires is substantial.
* PCBs have controllable and predictable electrical and mechanical properties.
* A more uniform product is produced because writing errors are eliminated.
* The distributed capacitance is constant from one production to another.
* Soldering is done in one operation instead of one connection discrete component by wires.
* The PCB construction land itself for automatic assembly.
* Spiral type of inductors may be printed.
* Weight is less.
* It has miniaturization potential.
* All the signals are accessible for testing it at any point along the conductor track.

# current carrying capacity depends upon the track with. due to high currents there is temperature raise caused in conductors.

Ideally, under normal working conditions there should not temperature raise in conductors. When two conductors are running parallel to each other, depending upon the dielectric constant eliminate their exit certain capacitors. Similar in double sided boards. When track are either side of boards there is capacitance as two tracks are considered to metal conductor with laminate as dielectric .Mechanical considerations have to taken on account like heavy components like transformer may be given separate mechanical support. The number of jumper should be minimum. Heat sensitive components must be kept away from the heat producing ones Sufficient test points must be given and components must easy accessibility for replacement. When two signal lines are running close to each other is possibly of cross take. To reduce this, an electromagnetic interference all unused copper surfaces are connected to ground line is made sufficiently broad. Normally low power and high power level wire are twisted out side PCB to protect the circuit from the electromagnetic coupling.

**7.1 LAYOUT:**

Layout designing is the pencil sketch of component and conductors drawing which contain all relevant information for preparation of artwork. Layout is designed is the pencil sketch of component and conductor drawing which contain all relevant information for preparations of artwork. Layout is designed on tracing paper for better accurate.

**7.2 Etching**

Removal of unwanted copper, to give final copper pattern as ETCHING solutions, which are using on Etching, is known as enchant.

1. Ferric chloride
2. Cupric chloride
3. Chronic acid
4. Alkaline ammonia

Out of these ferric acids is widely used because it has short etching time and it can be stored for longer time rising follow etching,

**7.3 SOLDERING:**

It is the process in which the components and the connecting tracks get in join firmly. We used 25-watt soldering iron and flux coated soldering wire. The solder which is to be applied to the till the gap of joints is placed close to the iron bit. It immediately melts and become bright and fluid. Enough soldering was applied to fill up the gap of the joints to give good strength and conductivity to the connections. When the joint happen to be sufficiently filled with the soldering irons removed and the joints are allowed to cool down. After soldering PCB is cleaned and made free from dust. All the connection is checked with the help of the multi-meter (AVO).

The dry soldering or short connections are done. Now wiring flexible multicore wire of require gauge, the wiring is completed. First the wires are cut down into the sufficient length. The inside core and connections are cleared and soldered connections are checked with multimeter (AVO).

Fluxes are auxiliary materials used while soldering is done.

1. They dissolve and remove oxide and contain elements from the surface of metal to solder.
2. They protect the metal surface and molten soldered from oxidation.
3. They reduce the surface tension of motion solder.
4. They improve the ability of solder to wet the metal.