

PCB Fabrication Facility

Indian Institute of Technology Kanpur



PCB Design Guide Sheet

2009

The Department of Electrical Engineering at IIT Kanpur has a variety of devices and machines to produce single layer, double layer plated through printed circuit boards (PCBs), multi layer (max 8 layers) plated- through PCBs, for through pin and surface mounted devices.

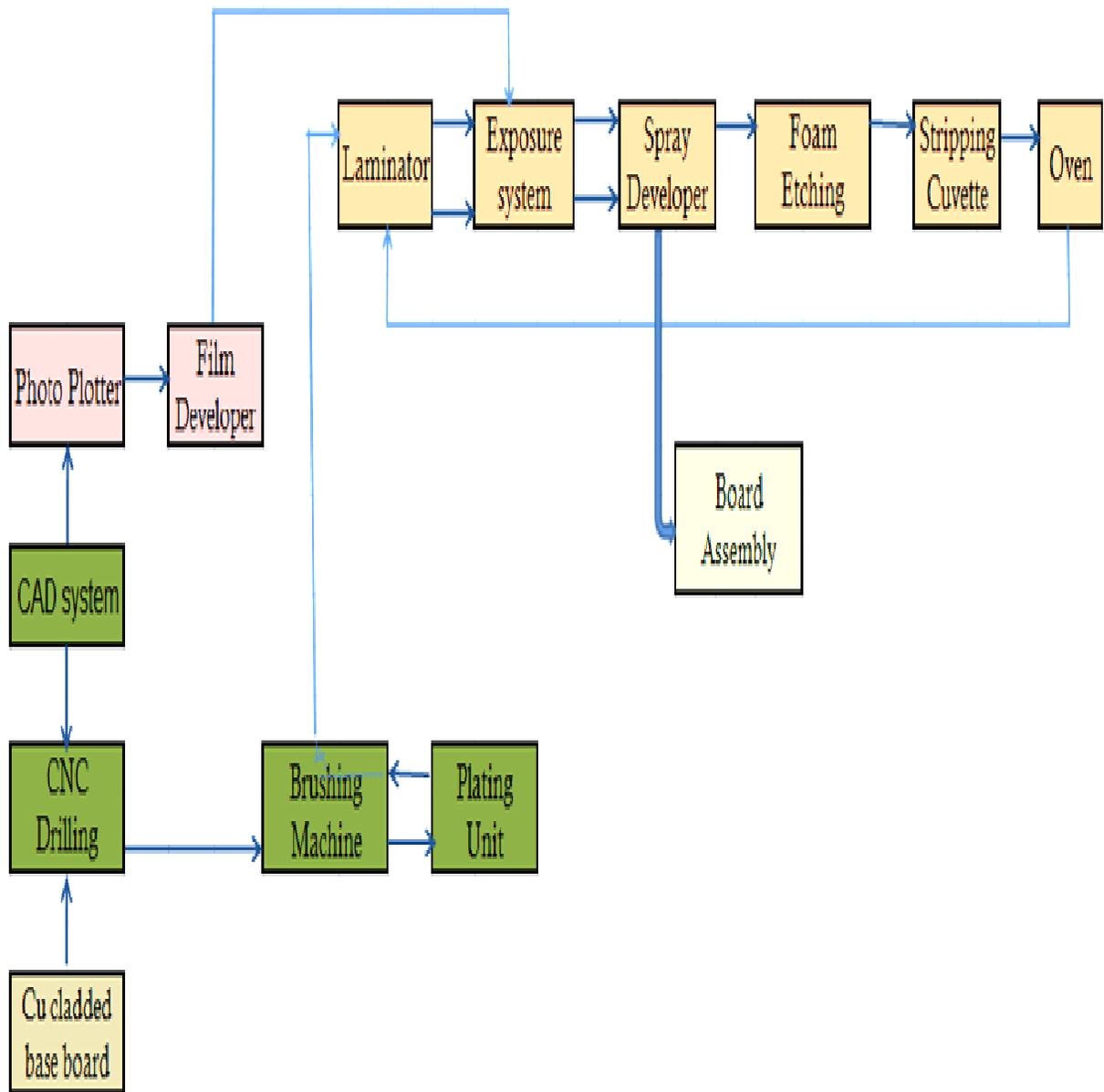
Printed circuit boards are electronic circuits boards created for mounting electronic components on a nonconductive board, and for creating conductive connections between them. The creation of circuit patterns is accomplished using both additive and subtractive methods. The conductive circuit is generally 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.

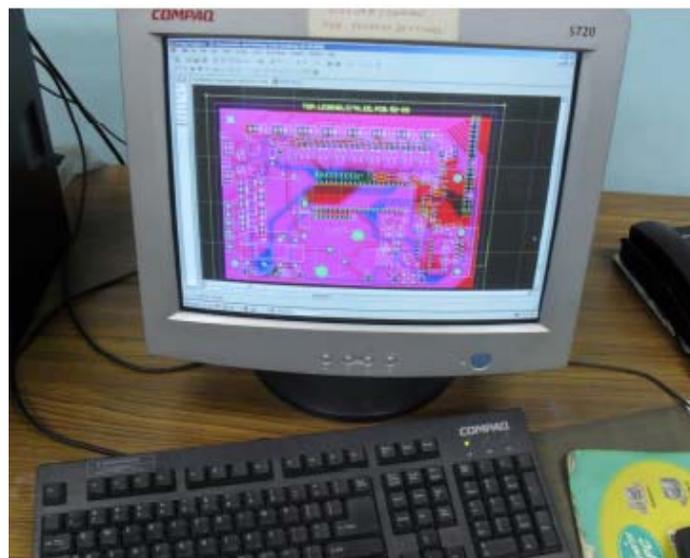
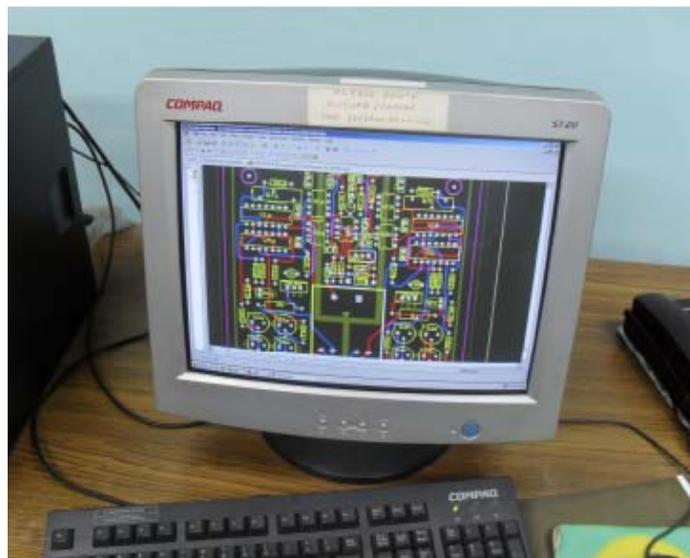
Process steps involved in PCB fabrication are shown in the following block diagram.



## CAD System

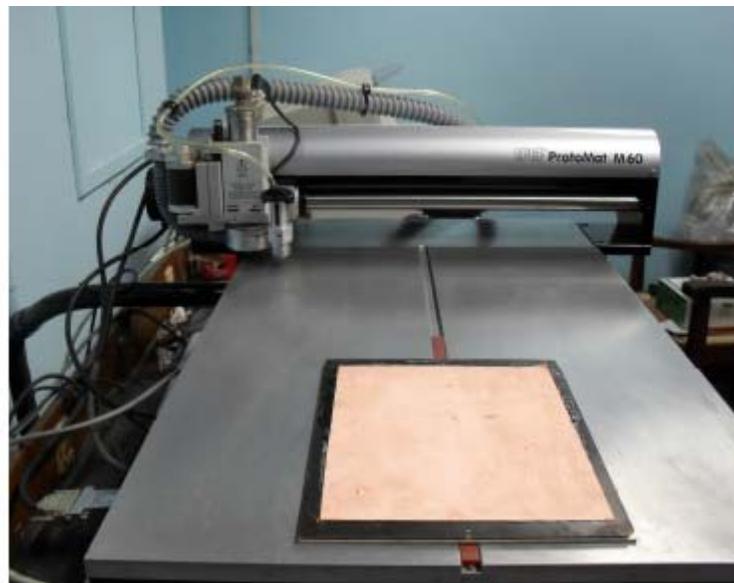
Recommended software for the CAD system is Altium. This software has enhanced features like improved interactive routing, user selectable track width, internal routing loops, PCB routing completion detector, preserving track angles while dragging, designing bottom layer layout (mirror has to done) just by flip and editing the board etc.

Note: Minimum track width requirement is 0.2 mm and minimum spacing between tracks is 0.2 mm



### CNC drilling/milling machine

A computer automated, single spindle CNC machine is used which allows direct processing of Excellon/Sieb & Meyer drill data or HP/GL data for PCB production (drilling, milling, isolation routing) and milling/drilling of holes. Selection of drill bit is also automated and the drill bit moves to the corresponding coordinate to make a drill. Likewise drills of different dia can be made.



## Photo-plotter

With the photoplotter, films are developed for various layers and for the solder masks. Moreover, both positive and negative type films can be developed. This is also a computer automated equipment. It plots images from a CAD database by using Gerber data file on a photographic film.

Here, the film is rolled over the inner drum in the plotter. On running the software in computer, the laser diode moves step by step along the rotating drum by means of stepper motor driven lead screw. The laser diode moving speed, resolution can also be adjusted as per requirement.



## Film Developer

After photoplotting, the individual PCB layer's film is produced by a developer machine. Then, under a light table, the film is manually inspected for any track breaks, line shortages or missing of any track.

Here, the films are developed manually in a tray.

## Brushing Machine

After completion of drilling, the boards are to be brushed thoroughly to ensure that no stray particles remain on the board. Boards are brushed thoroughly, rinsed and dried by this wet brushing machine.



## Laminator

This is used to laminate the entire board area with photoresist material. Here either positive or negative photoresist material can be used for laminating the board. We generally use negative photo resist. Lamination is performed by rolling the photoresist coil under pressure on both sides of the moving board by heatable feed rollers. However rotational speed is adjustable as per requirement.



## Plating unit

This unit is used for making printed through hole (PTH) circuit boards, where connectivity from top to bottom layer can be made through holes. Here top and bottom layers are connected by depositing copper in holes of the board. This unit comprises all the tanks and equipment for degreasing, rinsing, catalyzing, activating, pickling and plating the drilled and cleaned boards. These process of degreasing, rinsing, catalyzing, activating, pickling are used for removing grease, oil, dirt, soot and other contaminants from surfaces. After thorough cleaning only copper is deposited.



## Exposure system

After fixing layout-films exactly on both sides of the laminated board the light sensitive cover is exposed by a ultraviolet light in double-sided drawer exposure unit with a vacuum system. After the exposure that part of the laminate, which is not protected by the layout film becomes etch-resistant (if we use negative film & negative photo resist).



## Spray Developer

After exposing the board to UV light, the PCB is brought into a spray developing unit where the board is treated with aqueous alkaline solution (like TCl alcohol) so that the nonexposed part may get softened. A rotary system present inside the chamber allows the simultaneous treatment of the entire board surface.

## Foam Etching Center

After treating the board with aqueous alkaline solution, the PCB is subjected in foam etching center by hanging it into a foaming fluid (usually acids) to etch out the copper from the part of the PCB not exposed to UV. The copper part of the PCB which is not exposed to UV gets softens and it is etched by acids. Now the board is ready with printed layout tracks, pads which are covered by photo resist.





## Stripping Cuvette

Here the remains of photoresist material from layout tracks, pads are removed and rinsed in liquid, usually an aqueous alkaline solution.

## Oven

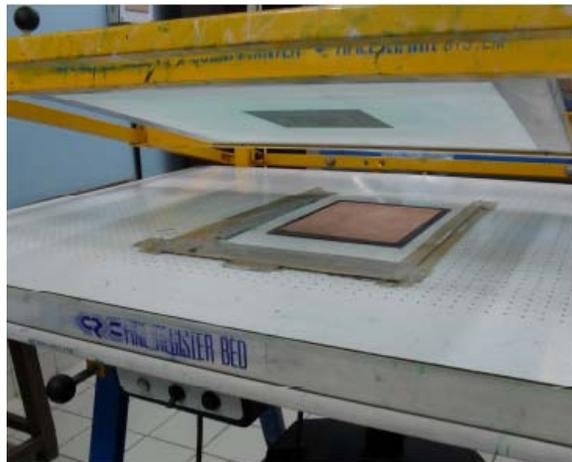
After getting the board from stripping cuvette, the board has to be dried. This is done in the oven, where temperature is adjustable.



## Masking and Legend printing

The bare copper PCB is silkscreened with a solder mask (usually green) (Sometimes the solder mask is applied by photo imaging or dry film) which is designed to insulate and protect the copper tracks and keep them from shorting together during the soldering process. The solder mask covers the whole board except solderable surfaces such as thru-hole and surface mount pads.

The solder mask is then dried or cured. The PCB is tinned or plated, i.e., solder, silver or gold is applied to exposed pads. The PCB is silkscreened with component identification lettering (usually white). The silkscreen legend is dried or cured. Any final drilling of holes that are not to be plated through and any extra routing are now performed, and the laminate is cut into individual printed circuit boards.



## Multilayered PCB

For fabricating multilayer PCB, we have to first fabricate corresponding number of double sided PCBs.

For a 4 layered PCB, we have to first fabricate two double layered PCBs, and for 6 layered, 3 double layered PCBs.

These double layered PCBs are sandwiched one over another exactly with oxidizing agent between them. Then the sandwiched board inserted in the multilayer press machine. It has a control unit for all pressure supply, press plates and heaters. The large loading door that allows quick and easy access to the pressure part is security switch protected. The unit is controlled by two digital and adjustable thermostats, one digital timer as well as a pressure valve with pressure meter. Two strong air ventilators are activated automatically during cooling cycle.

After complete curing and cooling, the sandwiched multilayered PCB is taken out from the press machine is subjected to masking, legend printing as per requirement.



The PCB lab. has a variety of chemicals in the store for PCB fabrication process. The lab. has facility for producing deionized water which is used in plating purpose.



It has got the facility of inspection of photo films and PCBs under a table lamp with a microscope.



## PCB Manufacturing steps

Following are the steps involved in fabrication of PCB.

- Base material cutting
- Drilling
- Through hole plating
- Layer film generation
- Solder resist film generation
- Legend printing film (optional)
- Laminating
- Exposing
- Spray developing
- Etching process
- Solder resist masking
- Legend printing (optional)

### Base material cutting

Base material which is selected based on application, may be epoxy resin (FR4), duroid (of required dielectric constant), is cut into the required size

### Drilling

Holes are drilled through the board using an automated drilling machine. A cnc drilling/milling machine, which is controlled by an Excellonfile with the hole position data and alis of the labels of the required drill bits, is used to drill the holes, to mill certain board contours and to produce multiple printed panel. The holes are used to mount electronic components on the board and to provide a conductive circuit from one layer of the board to another.

### Through hole plating

Following drilling, the board is scrubbed to remove fine copper particles left by the drill. After being scrubbed, the board is cleaned and etched to promote good adhesion and then is plated with an additional layer of copper. Since the holes are not conductive, electrolysis copper plating is employed to provide a thin continuous conductive layer over the surface of the board and through the holes.

### Layer film

For multilayer boards films must be produced according to the number of layers. Powerful software tools generate a plotfile, whose data control a photoplotter. After developing the films, they have to be inspected under table lamp for any track breaks, line shortages or missing of any track.

### Solder resist film

In order to ensure no shortings in the PCB due to formation of moisture, or due to spreading of solder leads, the complete board except the pads are to be soldered masked. This is done in CAD design tools and the photofilm is generated in the photoplotter. Then the photofilm is developed.

### Legend printing film (optional)

This is developed in order to ensure ease and fastness during soldering of components. This is also designed in CAD system and the film is generated in the photoplotter.

### Laminating

By this process the drilled and plated-through boards are laminated by rolling under pressure a photo-resist laminate on both sides of the board.

### Exposing

After properly aligning films on the photoresist laminated board, the board is exposed by a UV light on both sides in the UV exposure unit.

### Spray developing

The exposed board is developed in a spray developer unit. The developer solution is sprayed on both sides of the board by a spray rotating system. Afterwards the rest of the non-exposed laminate is removed by a rinsing process, whereas the exposed laminate is etch resistant and thus protects the circuit pattern against etching

### Etching process

In that process the UV nonexposed copper is etched. Afterwards the etch resistant material over the tracks, pads of PCB is rinsed by means of a solvent attack.

### Solder resist masking/Legend printing (optional)

In our lab this is done manually by screenprinting process.

## A. OVERALL GUIDELINES (SUMMARY)

Maximum PCB size which can be fabricated: 10 inches x 12 inches

Minimum PCB size which can be fabricated: 3 inches x 2 inches

Maximum track width possible: 100 mil

Minimum track width possible: 8 mil

Minimum clearance required for perfect plot: 8 mil

## B. DETAILED GUIDELINES FOR PCB JOB ORDER

### B.1 Pad dimension requirements

Inner diameter is 33 mil minimum. Higher values will automatically be mapped to nearest hole drill bit available. Must be at least 4 mil greater than component lead diameter to allow considerations for minimum copper plating thickness in the hole.

Outer diameter is 55 mil minimum and an integer values thereafter. Difference between inner diameter and outer diameter must be 20 mil minimum to ensure good copper area for soldering. This also applies to those footprints which have been specially created for the purpose by the designer. For SMDs, the standard pad size specified by the component manufacturer may be used.

### B.2 Via requirements

Inner diameter is 24 mil minimum to ensure correct feedthrough. Outer diameter is 40 mil minimum.

### B.3 Fill requirements

Copper fills for power planes need to be created with basic blocks of squares of the following dimensions: blocks of sizes 10 mil to 100 mil in steps of 5 mil; blocks of sizes 100 mil to 200 mil in steps of 10 mil. Considerable overlapping between these building blocks is necessary to ensure no visible gaps in the photo plot.

#### B.4 Maximum primitive dimension

The photoplotter at PCB fab facility, IIT Kanpur can accept and plot aperture sizes up to 100 mil. So the designer might take this into consideration before designing the PCB. If any dimension higher than this is needed, he/she might have to use some basic primitives. For example, if a track of 150 mil is required, a 100 mil track and a 55 mil track can be placed side by side with a 5 mil overlap to get a 150 mil track.

#### B.5 Hole drill bits available at the facility

##### Manual Drill

78.70 mil (2.0 mm), 62.99 mil (1.6 mm), 51.18 mil (1.3 mm), 39.37 mil (1.0 mm), 35.43 mil (0.9 mm), 31.49 mil (0.8 mm), 27.56 mil (0.7 mm), 23.62 mil (0.6 mm), 19.69 mil (0.5 mm)

##### CNC Mill/Drill

118.11 mil (3.0 mm), 59.06 mil (1.5 mm), 51.18 mil (1.3 mm), 43.31 mil (1.1 mm), 39.37 mil (1.0 mm), 35.43 mil (0.9 mm), 31.49 mil (0.8 mm), 27.56 mil (0.7 mm), 23.62 mil (0.6 mm)

##### RF Cutters

78.70 mil (2.0 mm), 39.37 mil (1.0 mm), 15.75 mil (0.4 mm), 7.87 mil (0.2 mm), 5.91 mil (0.15 mm)

#### B.6 Solder Masking/Legend Printing Facility: Available on request

#### B.7 Copper thickness on board

Single layer: Options of 1.38 mil (35 microns) with a board thickness of 62.99 mil (1.6 mm) and 2.76 mil (70 microns) with a board thickness of 94.49 mil (2.4 mm) are available. Board material is Glass Epoxy or Paper Phenolic.

Double layer: Options of 1.38 mil (35 microns) and 0.67 mil (17 microns) both with a board thickness of 62.99 mil (1.6 mm) are available. Board material is Glass Epoxy (FR4).

### C. Basic PCB Design Issues

The designer can use auto-route tools of the PCB design software he/she is using. But care must be taken that all dimensions are of integer values. Usually the designer can restrict the auto-route tool to use only integer dimensions by setting design rules for the PCB. It must also be ensured to keep outer most layer, a little larger in dimension than the actual board. A mechanical layer called 'Mechanical Layer 4' showing the actual dimensions of the board must be incorporated. This helps the fabricator to correctly use the CNC machine for drilling holes into the PCB. The origin of the PCB dimensions must also be set a little far, diagonally from the left bottom corner of the board. Please refer to the documentation of the PCB design software used for further information.

The PCB design name must also be mentioned either on the top layer or the bottom layer for a dual layered PCB. The text placement tool of the concerned PCB design software can be used. It is a good practice to make sure that the name of the file which contains the PCB design represents the PCB design very closely.

#### D. Recommended CAD Software

##### [Altium Protel DXP Suite 2004](#)

Note: If DXP 2004 is being used, please ensure that the .PCBDOC file is exported to PCB Version 4 Binary format (.PCB file) before ordering the PCB. The file which results due to the export must be directly sent to PCB fab for processing. Opening this file in DXP again could result in problems with the existing procedure.

Note: Please send only the final PCB file while ordering.

#### E. Ordering, Fabrication and Delivery

The final file required for fabrication must be submitted in the PCB Lab after filling up the PCB job order form. The process might result in delivery periods of 2-3 weeks for normal PCBs and 3-4 weeks for PCBs with green masking and legend printing.

These times may vary a bit depending on the queue pending at the facility and the process conditions of the PCB. Single layer PCBs will be delivered in less than 2 weeks time. PCB fabrication is a time and resource consuming process and hence is done with utmost care. Any complaints regarding the PCB fabrication and quality may directly be addressed to the PCB fabrication facility in charge.

## F. Glossary of terms

Footprint: The pattern and space on a board taken up by a component.

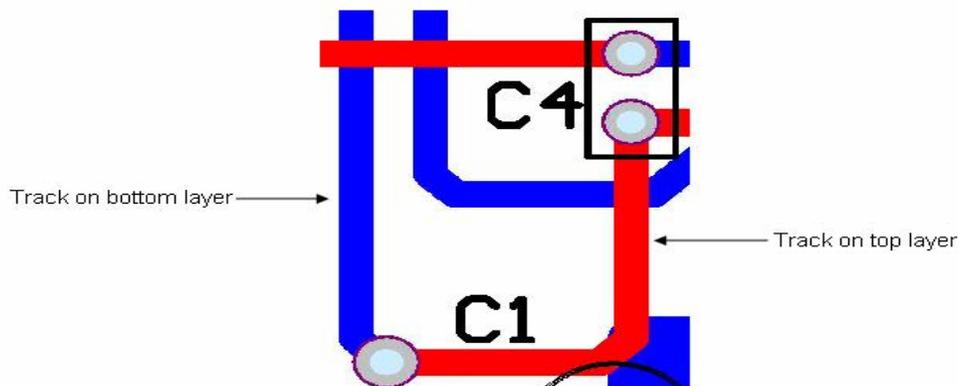
Clearance: The spacing required between two entities on a PCB to ensure safe operation.

Photoplotter: (also "vector plotter", or "Gerber photoplotter" after Gerber Scientific Co., which built the first vector photoplotters for commercial use). It plots images from a CAD database on photographic film in a darkroom by drawing each line with a continuous lamp shined through an annular-ring aperture, and creating each shape (or pad) by flashing the lamp through a specially sized and shaped aperture.

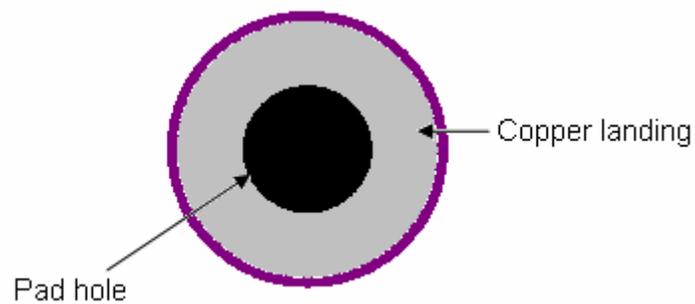
Gerber File: Data file used to control a photoplotter. Named after Gerber Scientific Co., who made the original vector photoplotter.

Aperture: An indexed shape with a specified x and y dimension, or line-type with a specified width, used as a basic element or object by a photoplotter in plotting geometric patterns on film. The "apertures" are thin trapezoidal pieces of plastic which are mostly opaque, but with a transparent portion that controls the size and shape of the light pattern passing through it.

Track: Besides its usual definition of a strand of conductor, wire on a printed board also means a route or track.

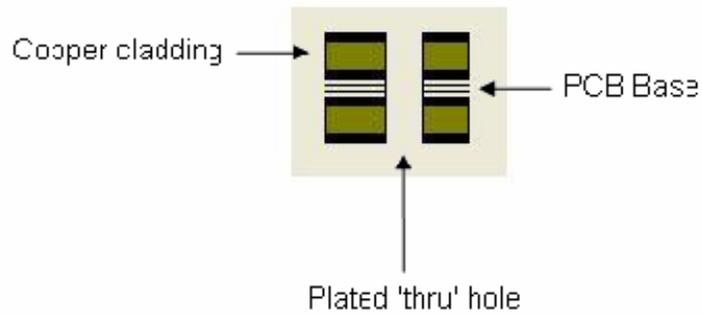


Pad: The PCB Component footprints have copper landing areas where the components are fixed by means of solder.

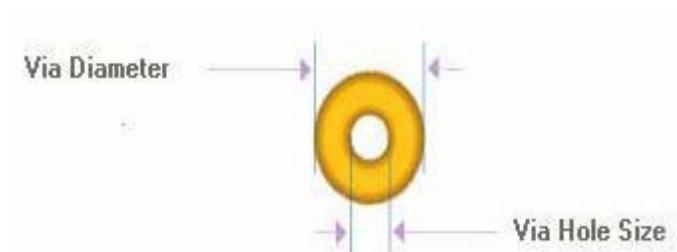


Through Hole: (Of a component, also spelled "thru-hole"). Having pins designed to be inserted into holes and soldered to pads on a printed board.

Plated-Through Hole: A hole in a PCB with metal plating added after it is drilled. Its purpose is to serve either as a contact point for a through-hole component or a via.



**Via:** A plated-through hole in a PCB used to route a trace vertically in the board, that is, from one layer to another.



**SMD:** Surface Mount Device, where the components' leads are not fixed into PTHs but are soldered to pads directly.

