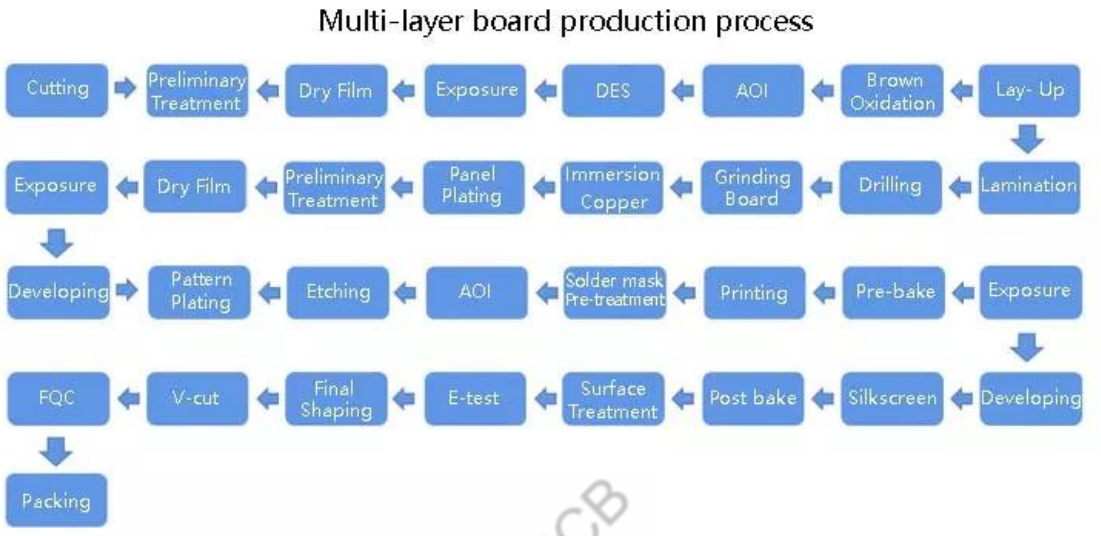
PCB manufacture process and technology

Below we use pictures and texts to show you the processing and manufacturing process of PCB boards:



China Multi-layer PCB Manufacture production process

1. Cut

Cut is the process of cutting the original copper clad laminate into a board that can be made on the production line.
First, let's understand a few concepts:

(1) Unit: refers to the unit graphics designed by PCB design engineers.

(2) Set: refers to the engineer in order to improve production efficiency, convenient production and other reasons, put together multiple “Unit” into a whole pattern. That is, we often call the panel, which includes unit graphics, process edges and so on.

(3) Panel: refers to a board that puts multiple “Set” together.

---

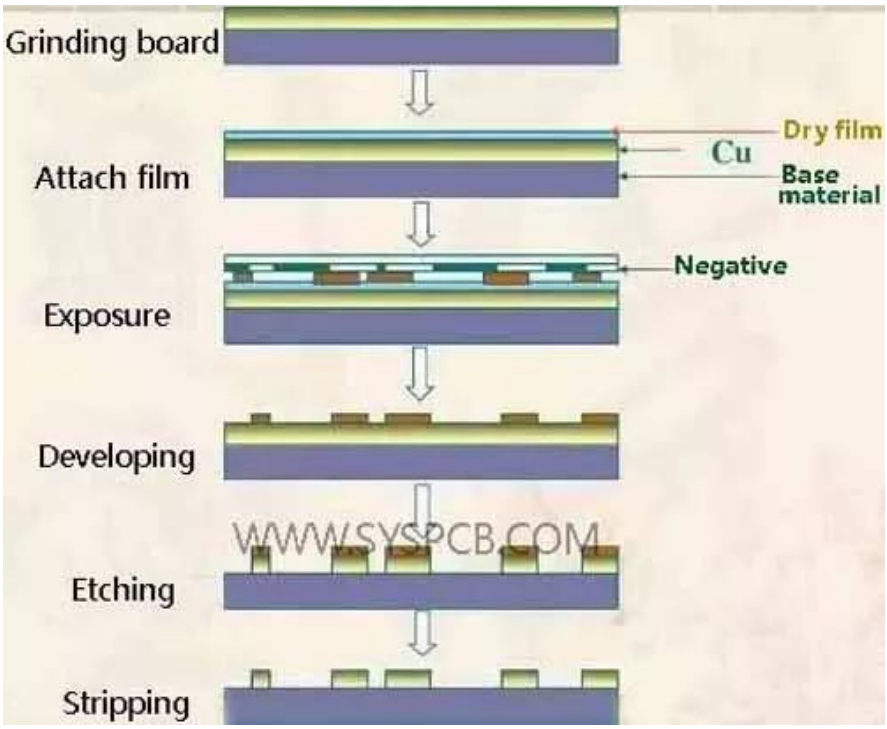
2. Inner dry film

The inner dry film is the process of transferring the inner line image to the PCB board.

In PCB production, we will mention the concept of image transfer, because the production of conductive graphics is the foundation of PCB production. Therefore, the image transfer process is of great significance to PCB production.

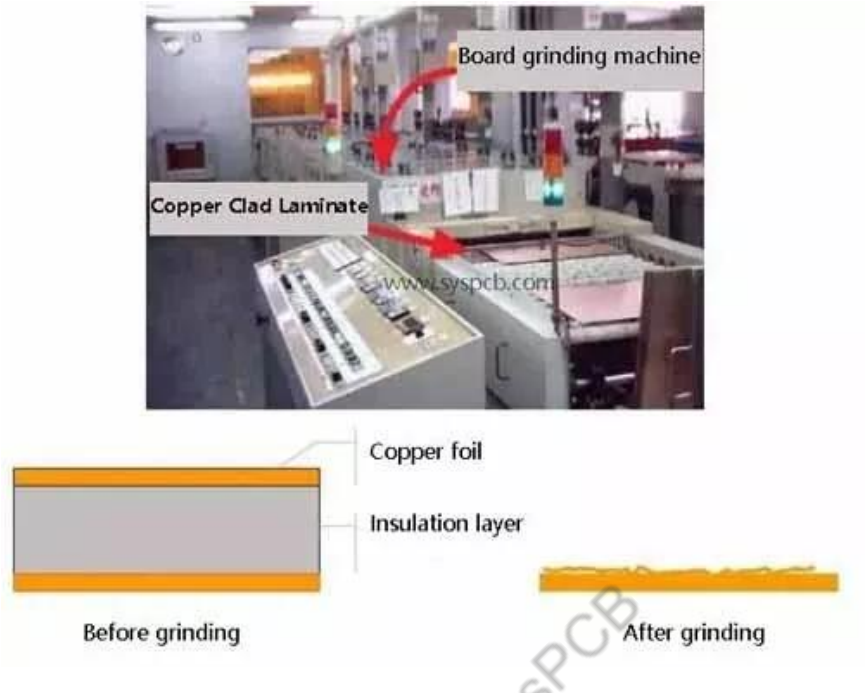
The inner dry film includes multiple processes such as attach inner film, exposure, developing, and inner layer etching. The inner film is to attach a special photosensitive film on the surface of the copper plate, which is what we call dry film. This film solidifies when exposed to light, forming a protective film on the board. Exposure developing is to expose the plate with a film attached, the light-transmitting part is cured, and the non-light-transmitting part is still dry film. Then, after developing, the dry film that has not cured is stripped, and the plate with the cured protective film is etched. After the film stripping, the line image of the inner layer is transferred to the board. The entire process flow is shown in the figure below.

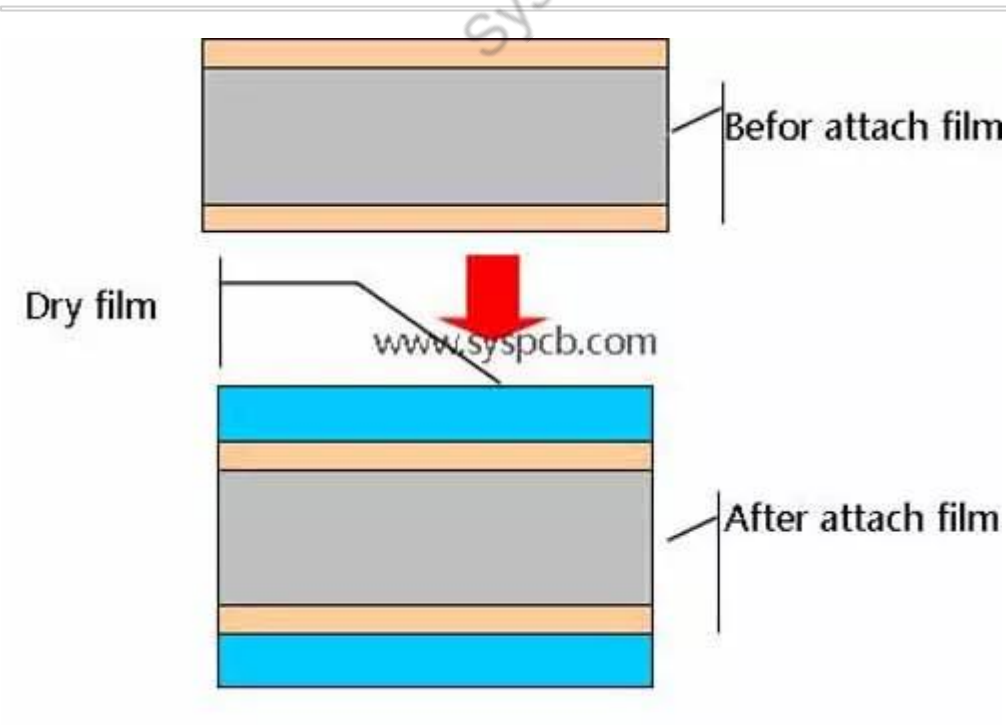
For designers, the main consideration is the **minimum line width of the wiring, the control of the spacing and the uniformity of the wiring**. Because the spacing is too small, the film will be sandwiched, and the film cannot strip clean and will cause a short circuit. The line width is too small, and the adhesion of the film is insufficient, resulting in an open circuit. Therefore, the safety spacing during circuit design (**including line to line, wire to pad, pad to pad, wire to copper surface, etc.**), all must consider the safety spacing during production.



(1) Pre-treatment: Grinding board

The main function of grinding board: mainly to solve the problem of surface cleanliness and surface roughness. Oxidation is removed, and the roughness of the copper surface is increased, so that the film can be adhered to the copper surface.





(2) Attach film

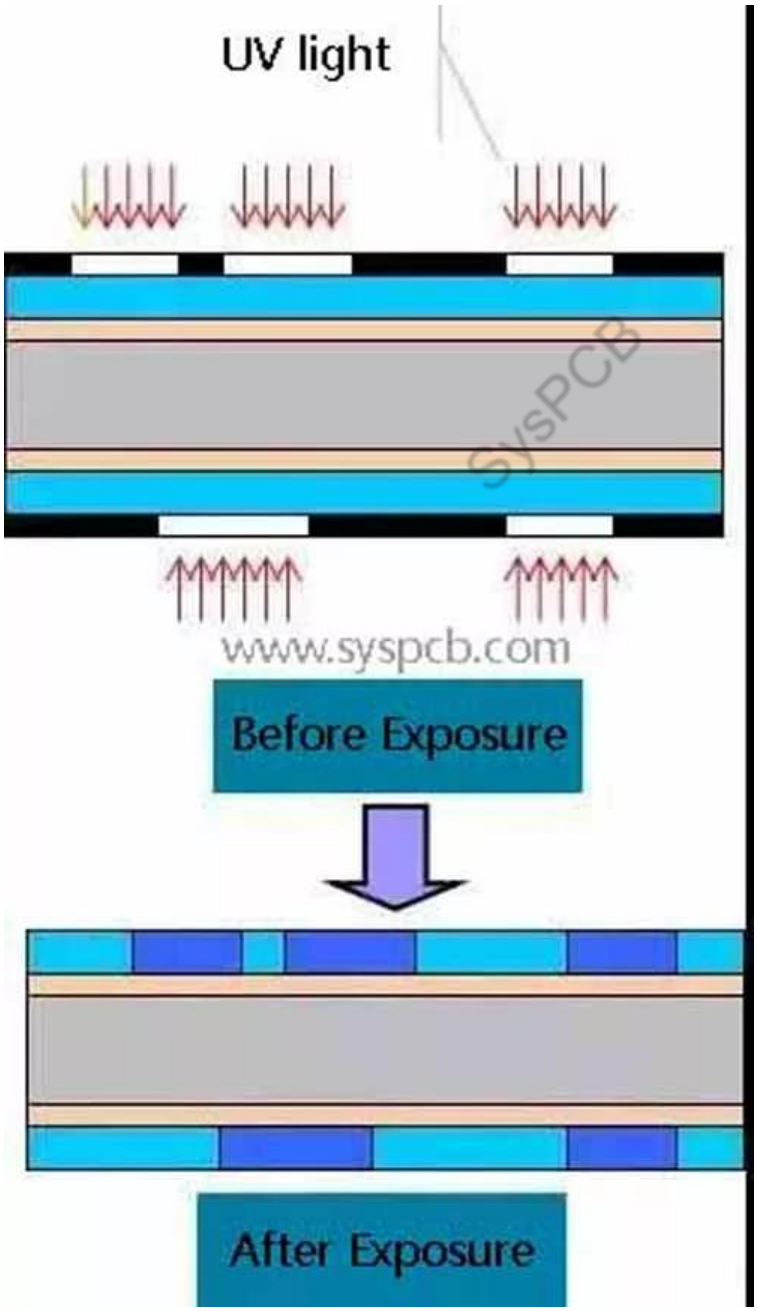
The treated copper clad laminate is hot-pressed or coated with dry or wet film to facilitate subsequent exposure production.

---

(3) Exposure

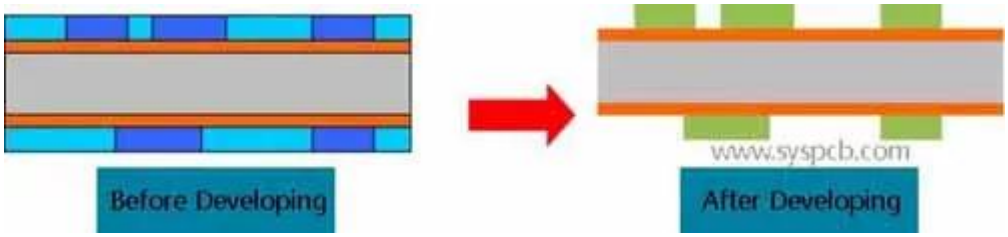
Align the film with the copper clad laminate that has been pressed the dry film, and use ultraviolet light irradiation on the exposure machine to transfer the film image onto the photosensitive dry film.

Physical image of negative film



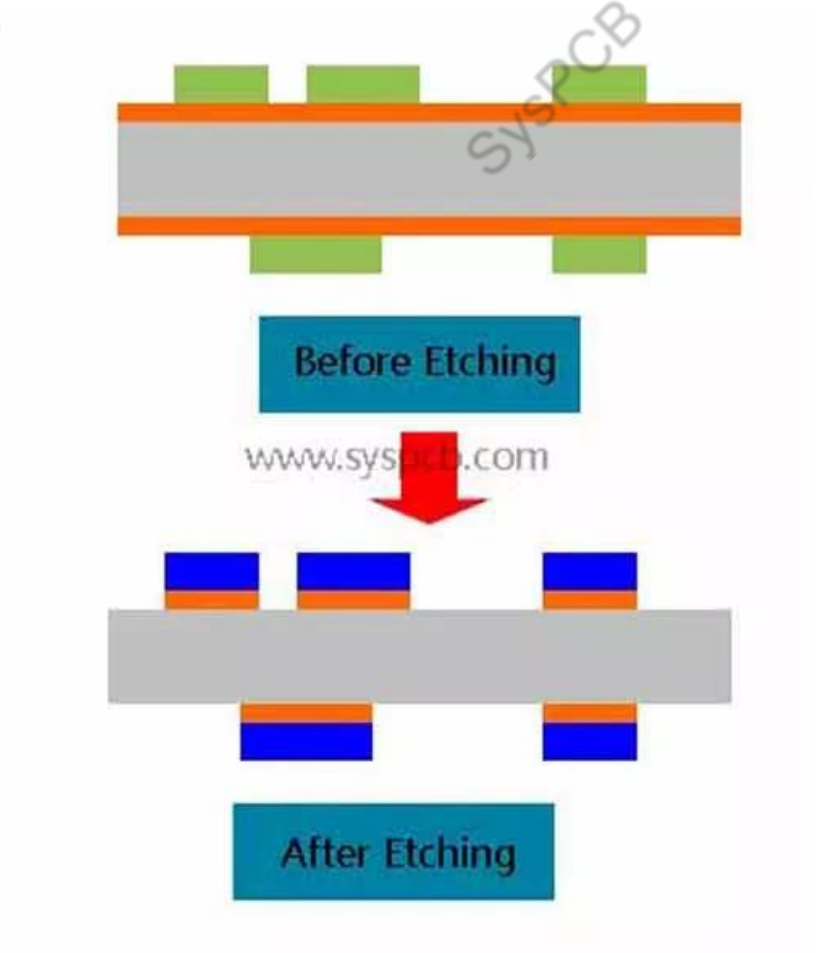
(4) Developing

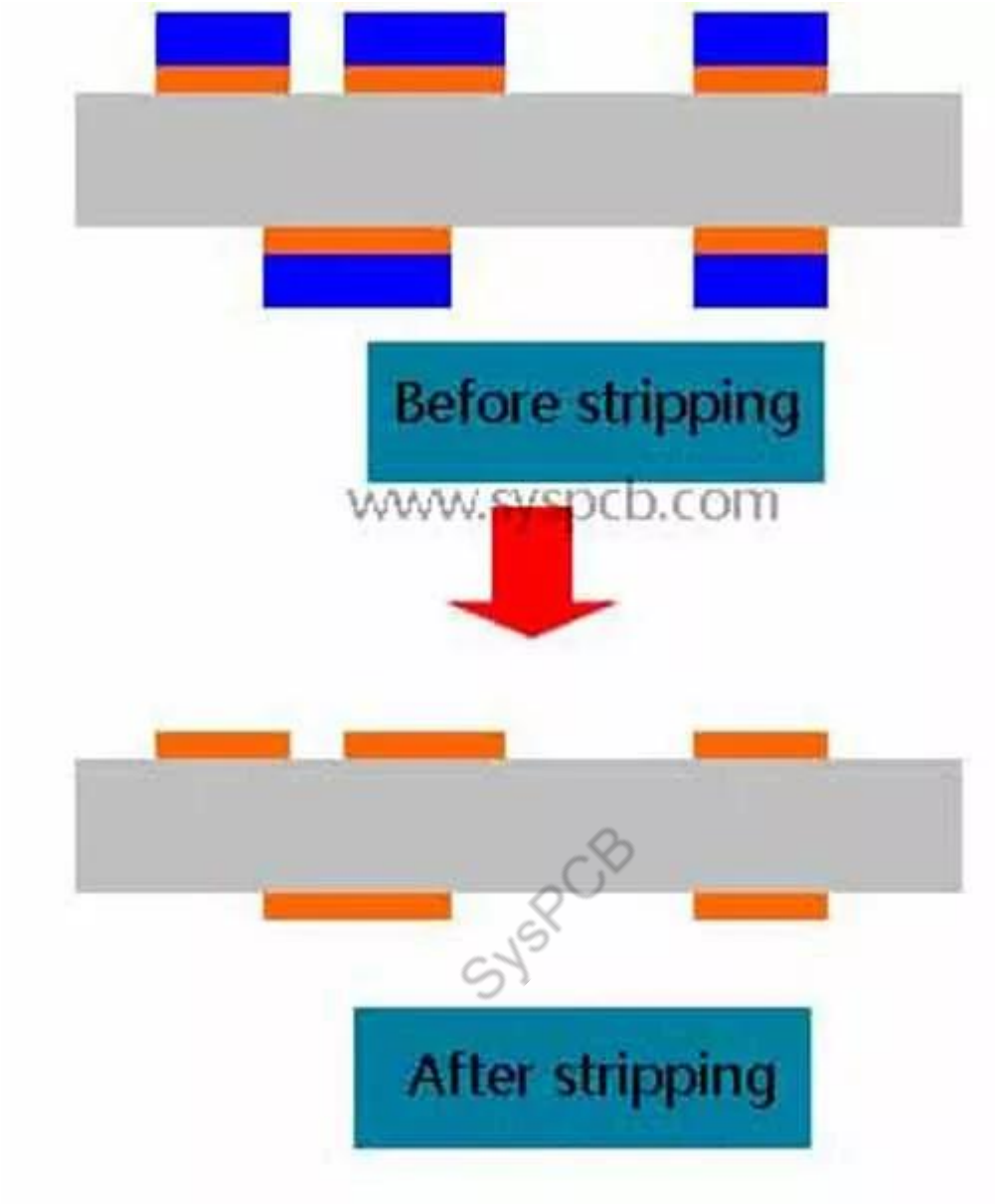
Dissolve and rinse off the unexposed dry/wet film using the weak alkalinity of the developing solution (sodium carbonate), while retaining the exposed part.



(5) Etching

After the unexposed dry/wet film is removed by the developer, the copper surface will be exposed. The exposed copper surface will be dissolved and corroded by acidic copper chloride to obtain the required circuit.





(6) Stripping

Stripping off the exposed dry film protecting the copper surface with sodium hydroxide solution, expose the circuit pattern.

---

3. Brown oxidation

Purpose: To form a micro rough and organic metal layer on the inner copper surface, enhancing the adhesion between layers.

Process principle:
By chemical treatment, a uniform and well adhesive organic metal layer structure is produced, which controls the coarsening of the copper layer surface before the inner layer is bonded, and is used to enhance the bonding strength between the inner copper layer and the semi cured sheet after pressing.

4. Lamination

Lamination is the process of bonding each layer of circuit into a whole using the adhesive properties of PP sheets. This bonding is achieved through the mutual diffusion and infiltration of macromolecules at the interface, resulting in interweaving. The discrete several plates and PP sheets are pressed together to form the required number of layers and thickness of multilayer board. During actual operation, materials such as copper foil, Polypropylene sheet (pp sheet), inner layer CCL, stainless steel, isolation plate (Kraft paper), outer layer steel plate, etc. shall be stacked according to the process requirements.

For designers, the first consideration for lamination is **symmetry**. Because the board is affected by pressure and temperature during the lamination process, there is still stress in the board after the lamination is completed. Therefore, **if the two sides of the laminated board are uneven, the stress on both sides will be different, causing the board to bend towards one side, greatly affecting the performance of the PCB.**

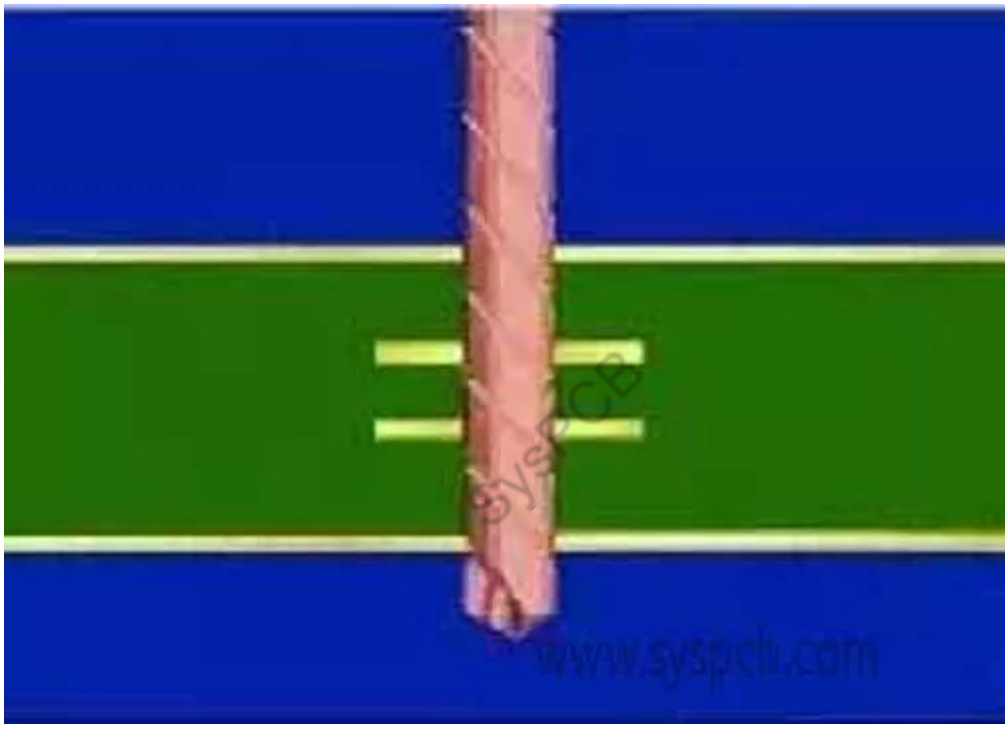
In addition, even on the same plane, if the distribution of copper is uneven, it will cause the resin flow rate to be different at each point. This way, the thickness of the area with less copper will be slightly thinner, while the thickness of the area with more copper will be slightly thicker.

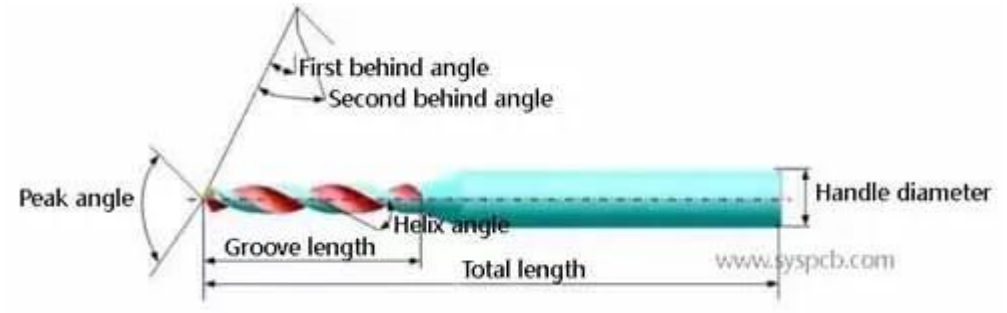
In order to avoid these issues, detailed consideration must be given to various factors such as the uniformity of the laying copper, **the symmetry of the stacking layer, and the design layout of blind buried holes during the design**.

---

5. Drill

Create through holes between the layers of the circuit board to achieve the purpose of connecting the layers.





---

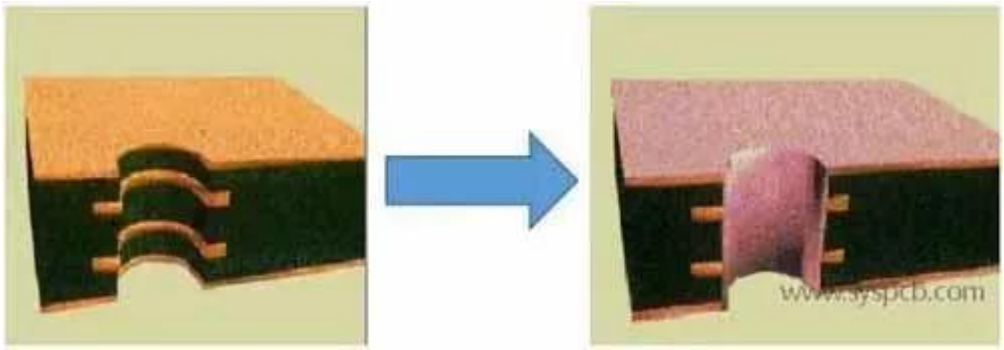
6. Plated Through Hole & Panel Plating

(1) Plated through hole

Also known as chemical copper, the drilled PCB board undergoes an oxidation-reduction reaction in a copper vat, forming a copper layer that metallizes the holes, depositing copper on the surface of the previously insulated substrate, achieving interlayer electrical connectivity.

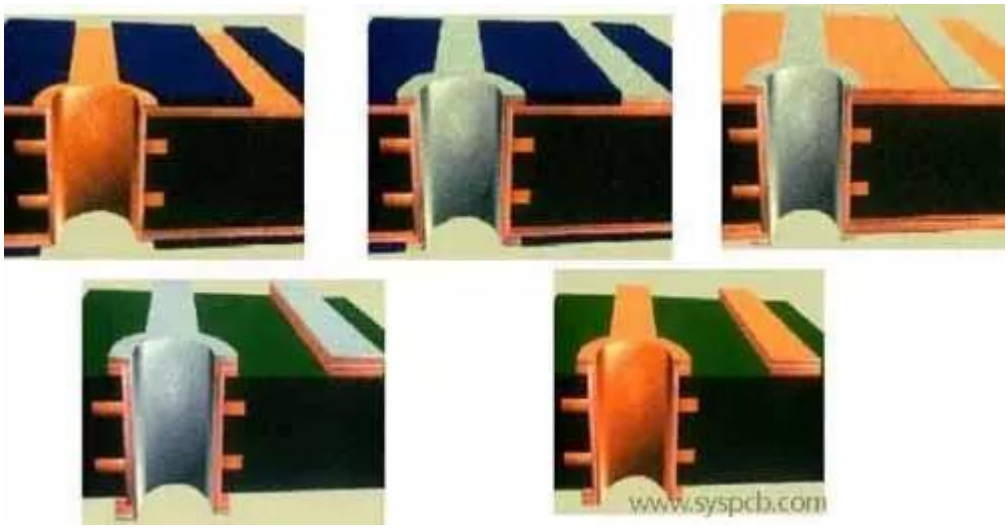
(2) Panel Plating

Thicken the copper on the surface and holes of the newly coppered PCB board to 5-8um, to prevent the thin copper in the holes from being oxidized or slightly corroded before the pattern electroplating, and exposing the substrate.



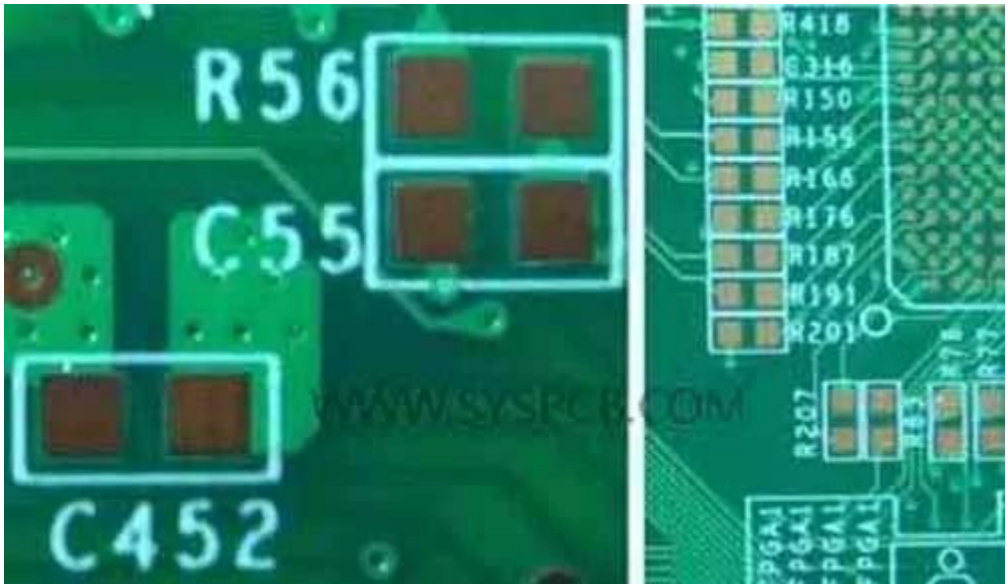
7. Outer dry film

The process is the same as for the inner dry film.



8. Pattern Plate

Add a certain thickness (20-25um) of copper plating to the hole and circuit copper layer to meet the requirements for the final PCB board copper thickness. Etch away the no use copper on the board surface， to reveal useful circuit patterns.



9. PCB Manufacture-Solder Mask

Solder mask, is one of the most critical processes in the production of printed boards, mainly through screen printing or coating of solder resistance ink, exposure and development, expose the pads and holes that will need to be soldered, other areas are covered the solder mask ink, in order to prevent short circuit during soldering.

10. PCB Manufacture-Silkscreen

Silk screen print the required text, trademark, or part symbol on the surface of the circuit board.

---

11. Surface Finishing

Bare copper itself has good solderability, but it is prone to moisture oxidation when exposed to air for a long time, and tends to exist in the form of oxides. It is unlikely to remain the original copper for a long time, so surface finishing of the copper surface is necessary. The basic purpose of surface finishing is to ensure good solderability or electrical performance.

Common surface finishing include **OSP, HASL (Hot Air Solder Leveling), Immersion Gold, Immersion Tin, Immersion Silver, Gold-finger etc**.

12. Final Shaping

Cut the PCB into the required external dimensions using a CNC molding machine.

13. Electrical Test

Simulate the state of the board, check the electrical performance when powered on, and check whether there is an open or short circuit.

14. Final inspection

Inspect the appearance, board size, hole size, thickness, and markings of the board to meet customer requirements.

15. Packaging

Packaging qualified boards for easy storage and transportation.

Use HDI technology to produce smaller PCB

High-density interconnect (HDI) technology allows printed circuit board(PCB) designers to increase routing density, reduce layer count, and improve the thermal and electrical characteristics of their board designs.

PCB designers can create a product that is much smaller using blind and buried vias. PCB vias come in different configurations.

- Through-hole (TH) vias originate and terminate at the outer layers of the PCB, while blind vias originate on an outer layer and terminate on an inner layer.

- Buried vias are completely buried within the board, connecting only inner layers and not reaching the outer layers at all. Microvias are a form of blind vias that penetrate only one or two layers, and they have an extremely small drill hole **(from 0.006 in./0.15 mm all the way down to 0.004 in./0.102 mm)**.

- In addition to using microvias, blind vias, and buried vias, this technique requires careful electromagnetic interference (EMI) and thermal analysis, PCB layer stack up, and power/signal/ground distributions.

You'll also need to be sure that when you place the ball-grid array (BGA) components that require decoupling capacitors that there is space for the capacitors on the locations underneath BGA packages.

With HDI Technology, electric products can achieve more function, weigh less and be physically smaller. Specialty equipment, mini-components and thinner materials have allowed for electronics to shrink in size while expanding technology, quality and speed. The miniaturization of electronic products will bring a prosperous future to HDI manufacturing enterprise.

Vias treatment in PCB manufacture

1. Vias Not Covered

If vias are not covered, they are exposed and the surface finish is applied to via barrel. This is standard process in printed circuit board fabrication.

(1) Advantage:

The via barrels are covered with surface finish metal. This allows circuit board testing access available from both sides of the board.

(2) Disadvantage:

It is possible for wicking of solder paste to get into the via. In the case of BGA rework, paste loss due to wicking into the via is a result of the localized thermal energy causing the LPI solder mask to lift on the short distance between the ball and via capture pads. This is not a concern at first pass assembly.

---

2. Solder Mask Tenting Vias

In tenting vias, no apertures are supplied on the mask data layer. The solder mask covers the via pads and tents hole. No surface finish is applied to the via barrel. For this process it is possible for entrapment to occur.

In the past, tenting of vias with Dry Film Soldermask was a standard offering. Due to the limitations on feature size resolution and high thickness for SMT applications of the Dry Film Masks, this process is not readily available. Assemblers may require plugged vias due to vacuum draw or to prevent paste wicking into vias.

This process is not advised for long term reliability. Many printed circuit board designs are seen with vias tented via primary mask. This could be a result of a lack of reliability data.

(1) Advantage:

The pros to primary LPI solder mask tenting vias is it is only a one step application.

(2) Disadvantage:

Via tenting cannot be guaranteed with LPI mask. There are three common methods of applying LPI mask. Curtain, Spray and Screen coating. Curtain and Spray coating cannot ensure that the via is tented both sides. Screen coatings ability to tent is limited by the hole size, surface tension of the liquid mask, and board thickness.

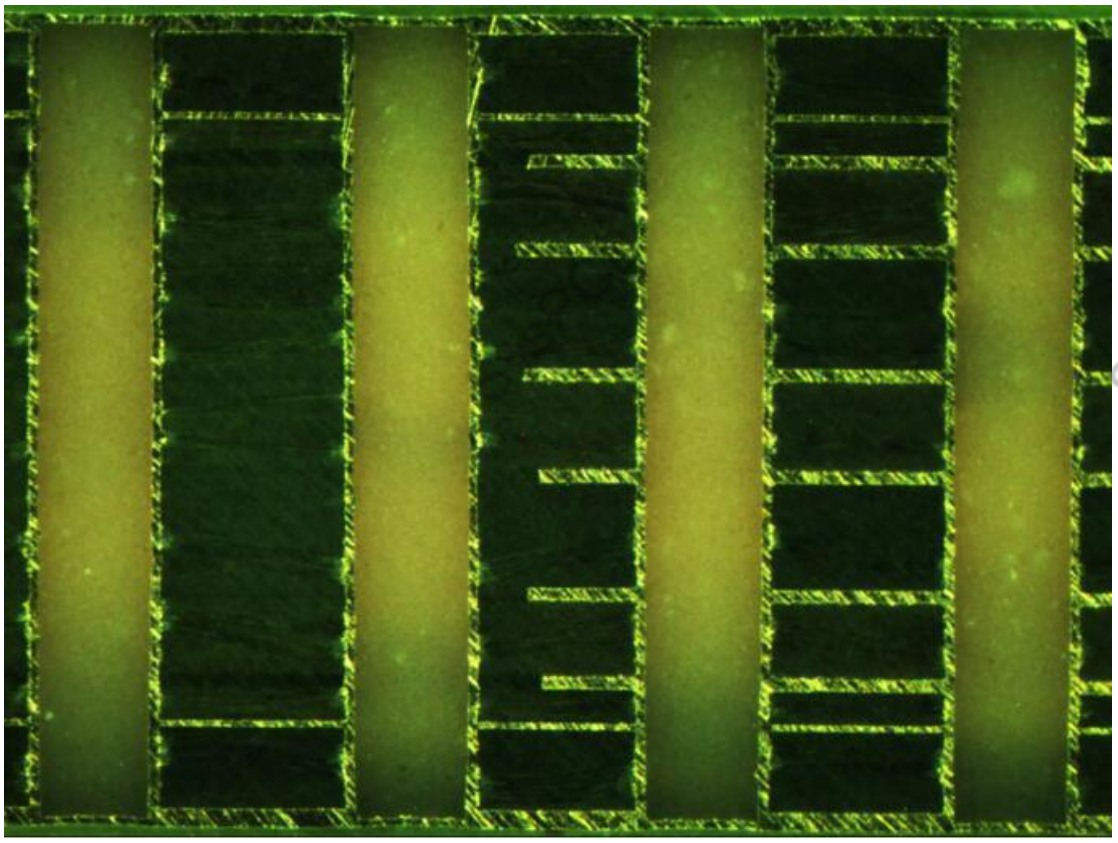
If a via is not tented on both sides, chemical entrapment from surface finish preclean lines is probable. All the finishes will be subject to a micro-etch process. The micro-etchant that gets trapped in the capped via will crystallize rapidly causing copper sulfate crystals. Over time, these crystals can cause long term reliability issues. In the case on ENIG finish, the gold and small area of exposed copper near the cap could form a galvanic cell, accelerating the etch process.

3. Plugged Vias

In the plugged vias process, vias are plugged with solder mask or some other non conductive media. LPI mask is then applied over plug. During the plugged vias process, no surface finish is applied to via barrel.
This process was developed as a modification from the LPI tent to guarantee that 100% of the vias are fully tented.

(1) Advantage:

In plugged vias, 100% of the required vias are tented.



(2) Disadvantage:

For plugged vias, additional process steps are required during fabrication. No surface finish is applied to the via and the via size is restrictive. The control of rate of rise during curing is critical to ensure 100% of volatiles are evacuated. Failure to control this can lead to soldermask smearing the surface during the assembly reflow process.

There are other treatment of Vias in PCB, we only introduce 3 of them this time. Normally, PCB manufacturers will choose Solder Mask Tenting Vias if not specified in Gerber file or manufacturing requirements, but few PCB manufacturers do just as the design of PCB, it means strict follow the design, so if your PCB design with Vias not tented but your requirements is tented, better tell PCB manufacturer when inquiry. For Plugged Via, it need more process then other 2 Options, price is slightly high than normal requirement of Vias.