Printed Circuits

Danilo Manstretta

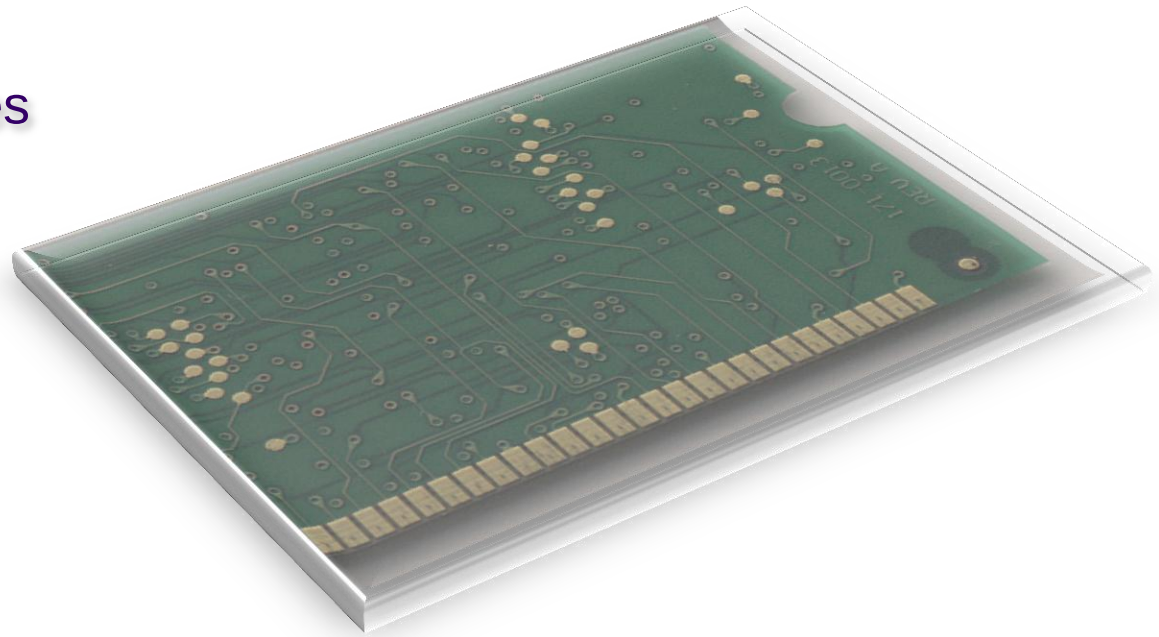
microlab.unipv.it/

danilo.manstretta@unipv.it

Università degli Studi di Pavia

Printed Circuits

- Printed Circuits Materials
- Technological steps
- Production technologies:
  - single layer, double layer and multi-layer boards
- Assembling:
  - Soldering techniques
  - Insertion mounting
  - Surface mounting



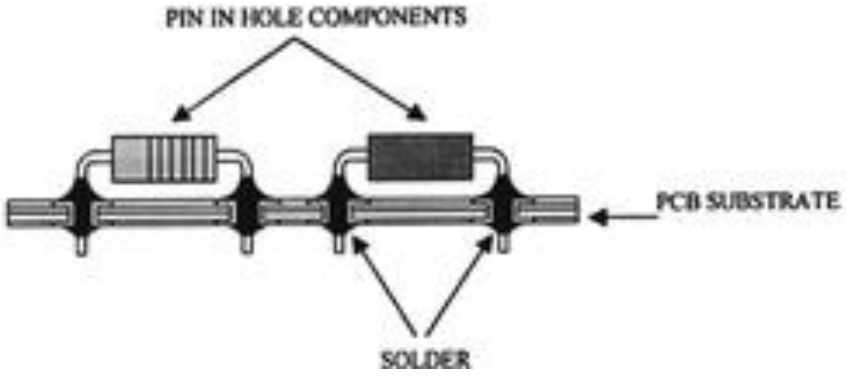
Printed Circuit Boards

- The main functions of a PCB are to provide *mechanical support* and *electrical connectivity* between different devices mounted on it
- An increasingly important issue is *thermal dissipation*
- Interconnects are created using two basic techniques:
  - Subtractive: the desired conductor pattern is defined by etching away the unwanted portion from a copper foil
  - Additive: the desired conductor path is formed on a bare substrate. The conductor can be plated or screen printed

PCB Technologies

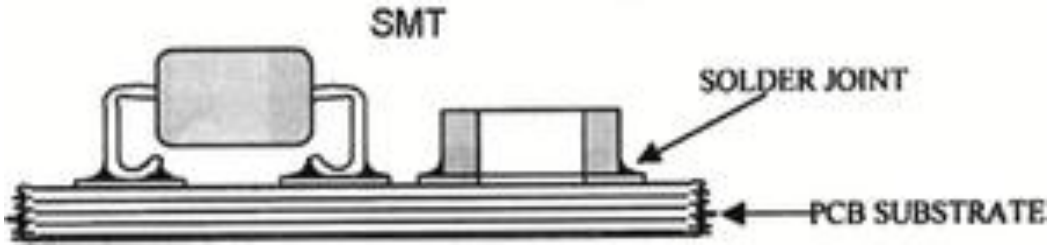
- **Insertion Mount Technology (IMT)**

  components pins are mounted on the board using Plated Through Holes (PTH)



- **Surface Mount Technology (SMT)**

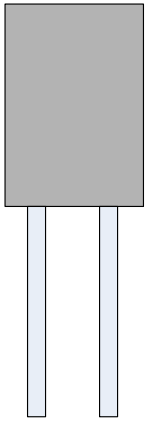
  requires Surface Mount Devices (SMD) that are soldered on metal pads



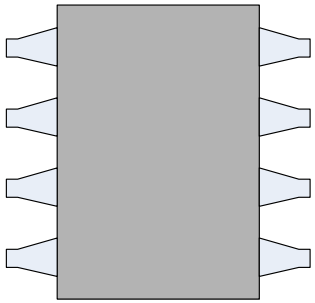
Through-Hole Insertion Devices

Axial

Radial



Dual In-line Package (DIP)



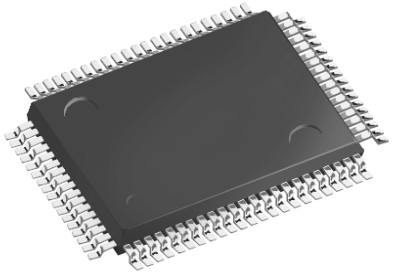
TOP VIEW

SIDE VIEW

Surface Mount Devices

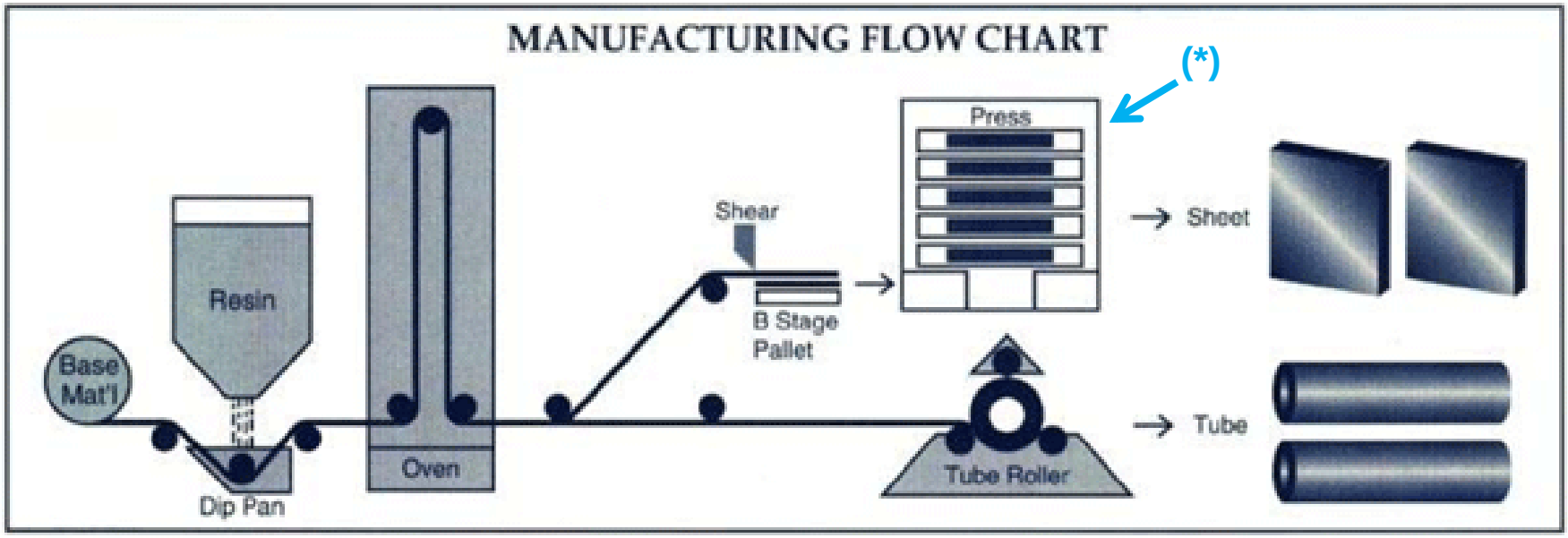
Passives

Quad Flat Package



Chip carrier (Leadless)

PCB Manufacturing Flow



- Batch processing: printed circuits production operations are performed on a standard-sized substrate (e.g. Eurocard 20x28cm^2). Individual boards are cut at the end of the production flow. This allows for improved automation.
- Cu sheet is added through deposition or as a laminated sheet added before substrate polimerization under pressure (*).

Laminates for PCBs

Laminate Material	ε_r	Tgδ @1MHz	Mechanical Resistance	MAX T [° C]	Cost
Fenolic	4	$3 \cdot 10^{-2}$	Low	100	1
Melamminic	8.5	$2 \cdot 10^{-2}$	Interm.	130	1.5
Epoxidic	5	10^{-2}	High	140	2
Siliconic	4.2	$3 \cdot 10^{-3}$	High	160	3.5
Teflon	2.6	10^{-3}	high	130	6

- Standard PCB laminate is Flame Retardant epoxy resin (FR4).
- For high frequency applications Rogers

- Cu layer (35μm to 500μm thick) is deposed by electrochemical deposition, glued or attached before polymerization

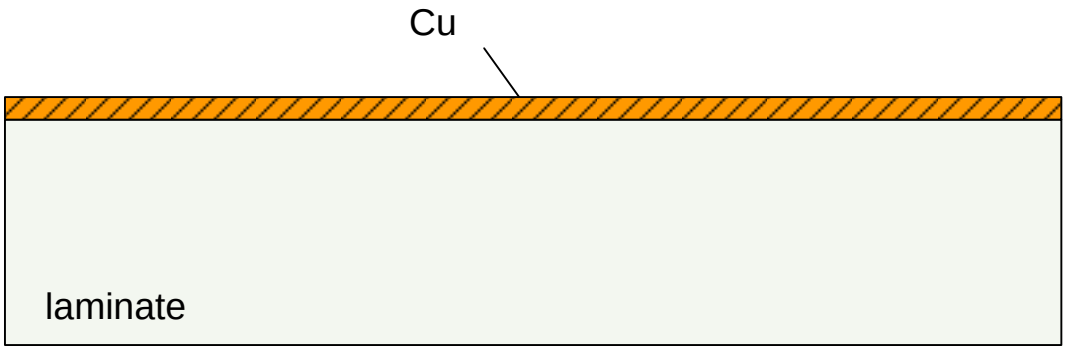
Basic Production Steps

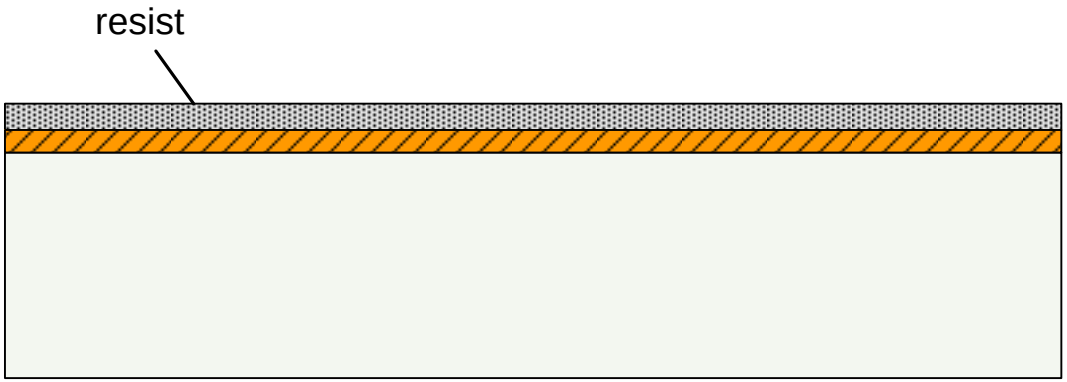
- Cleaning
- Drilling
- Masking operations
  - Photolithographic techniques
    - Etching resist
    - Plating resist
  - Screen printing
- Metal Film Deposition
- Etching
- Finishing

Masking Operations

- Patterned metal shapes can be obtained using two alternative masking techniques:
  - Photolithographic techniques
  - Screen printing
- Photolithographic techniques are described in this presentation. The advantage over screen printing is the better resolution (50-100μm)
- Screen printing is similar to the techniques used in thick film technologies and allows for higher productivity

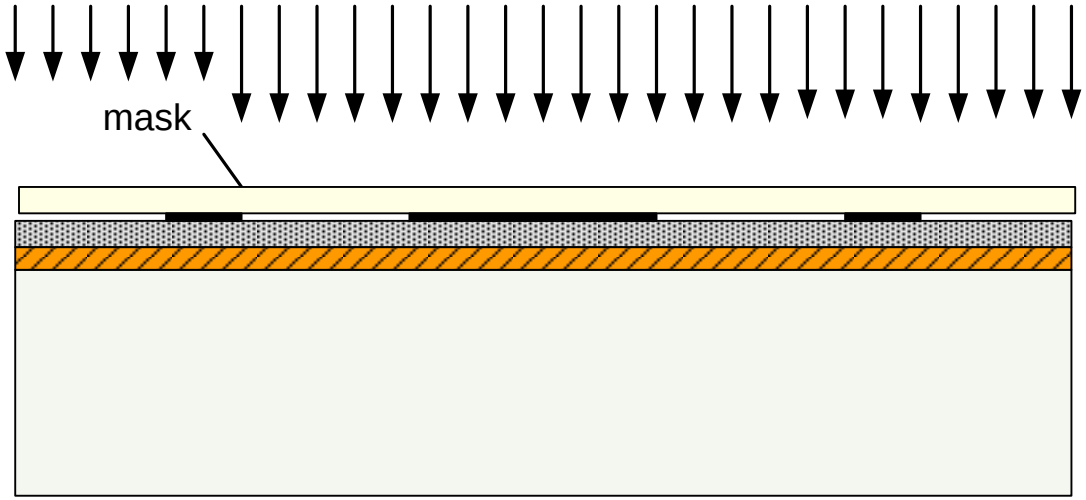
PCB Photolithography



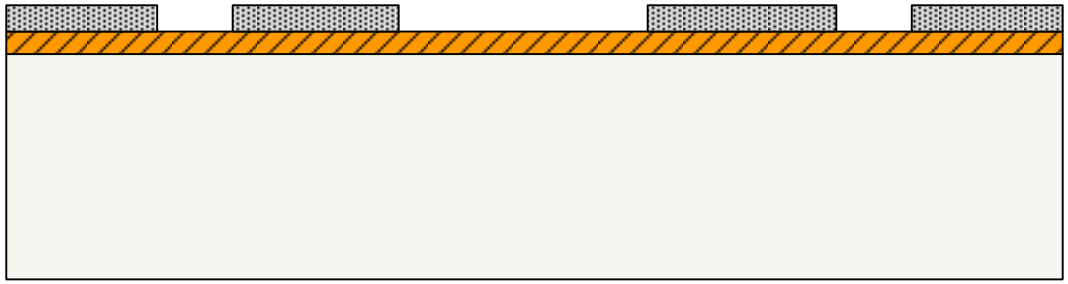


Liquid / dry resist application

PCB Photolithography

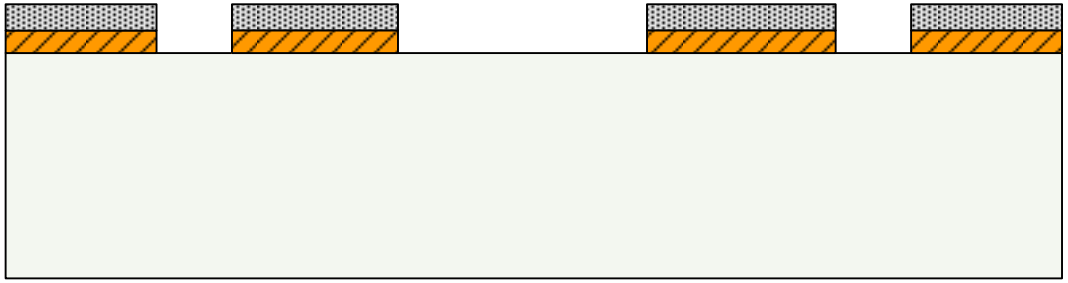


Contact mask exposure

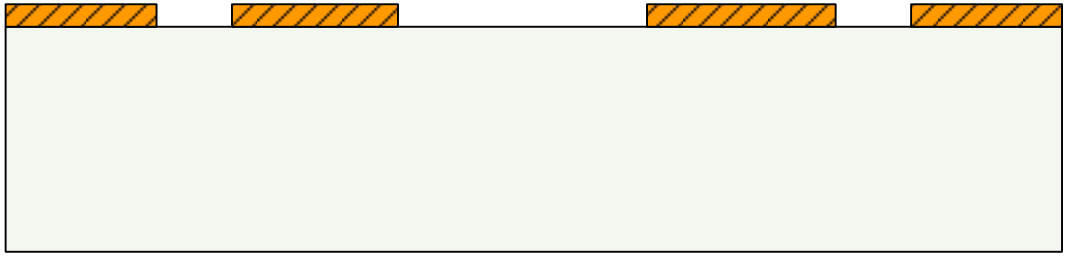


(Negative) photoresist develop

PCB Photolithography

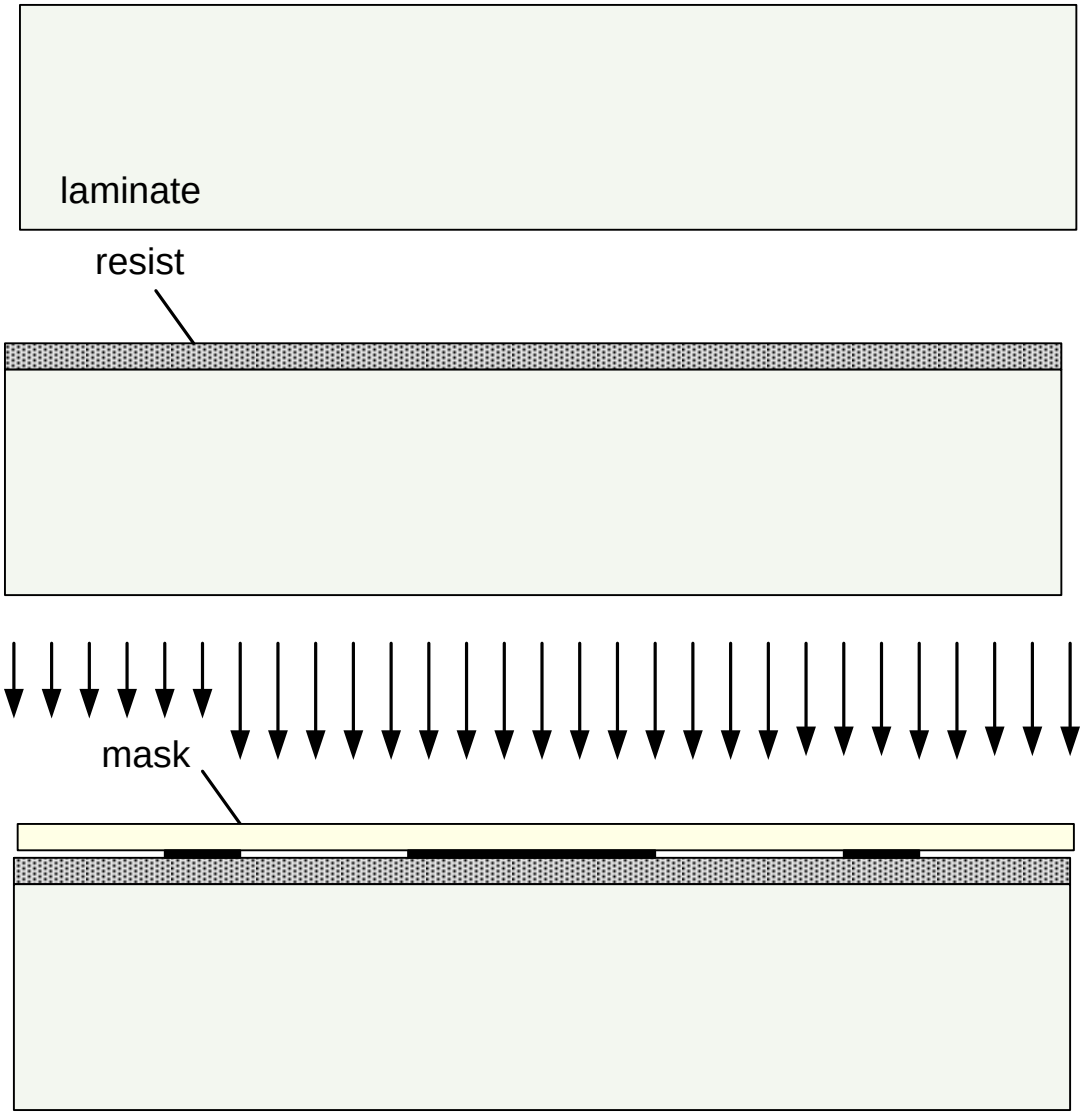


Metal etching



Photoresist stripping

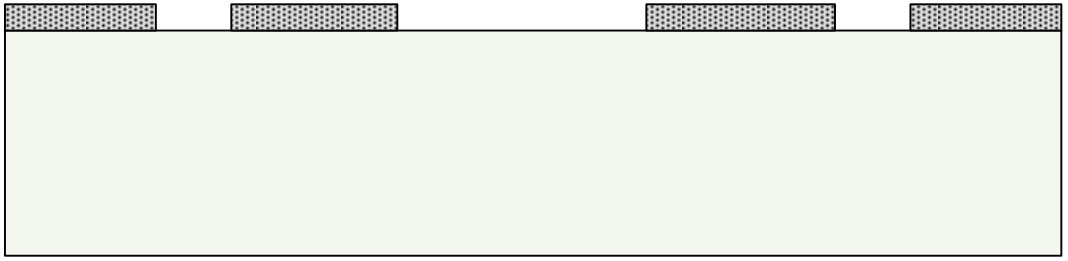
Plating Resist



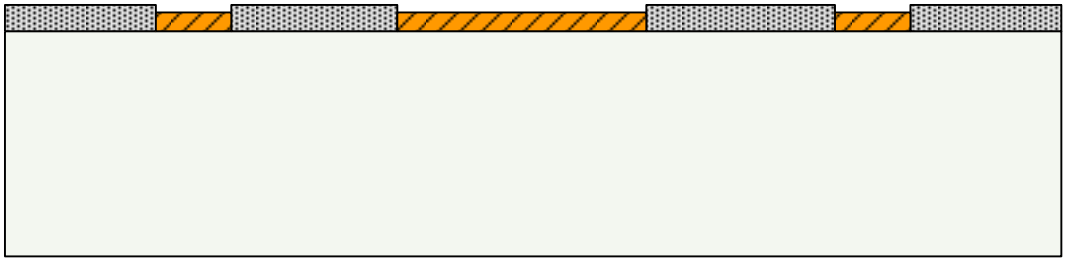
resist application

Contact mask exposure

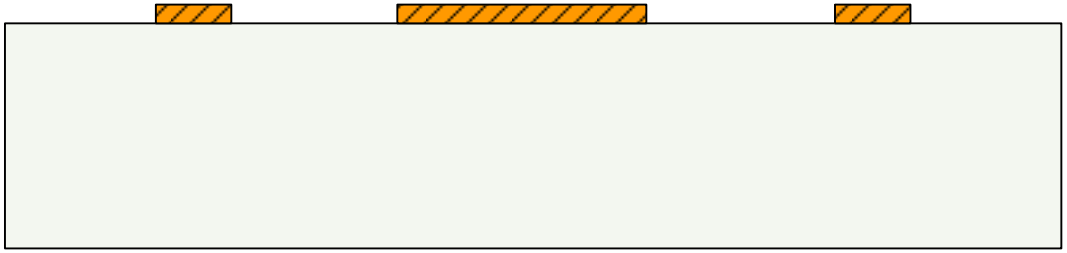
Plating Resist



photoresist develop



Metal deposition

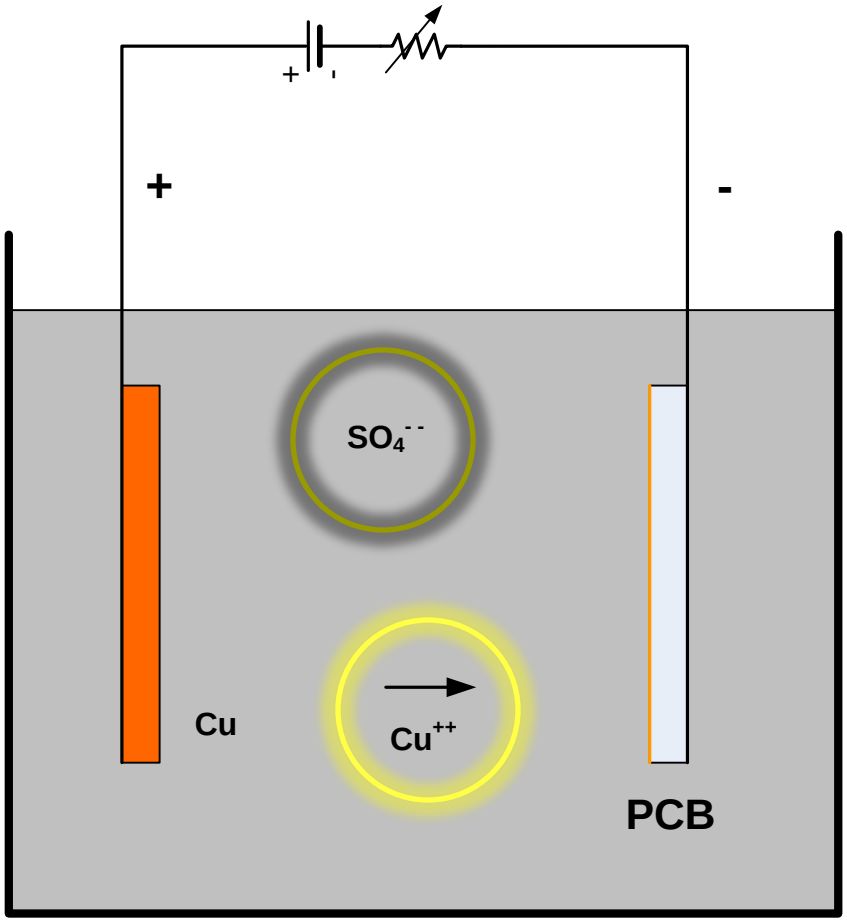


Photoresist stripping

Metal Film Deposition

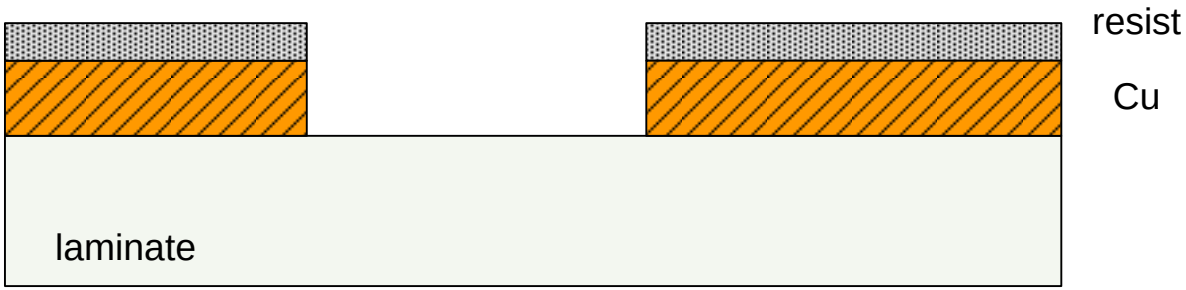
- Electro-less chemical deposition
  - After cleaning, the board is immersed in a $CuSO_4$ solution at moderate temperature (35-70°C)
  - Metals deposed using this technique are Cu, Au, Ni, Pd
  - This technique is expensive due to the low throughput (~μm/h)
- Electro-chemical deposition
  - After a thin metal film has been deposited using another technique (e.g. electro-less chemical deposition), the board is plated using an electrolytic cell formed by the board, a Cu bar and acid solution (e.g. $CuSO_4$)
  - Deposition can be made selective using a plating resist film
  - This technique is relatively fast (~μm/min)

Electrochemical Deposition



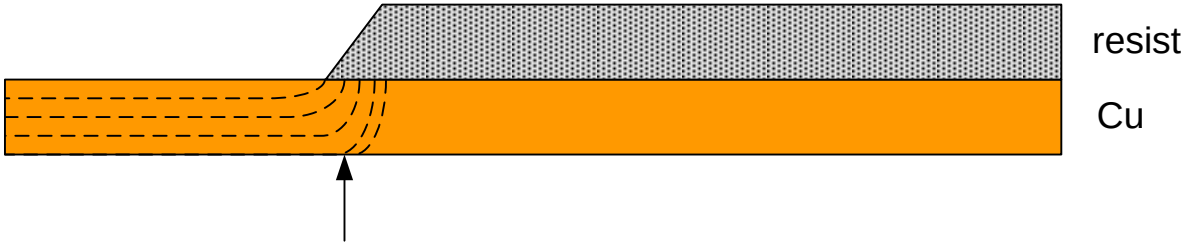
- Deposition can be made selective using a resist film
- Deposition thickness is controlled by current (per unit area) and time
- Typical deposition speed: few μm / min
  - Ex. 100mA/cm^2 results in a deposition speed of 1μm/min
- The same technique can be used for soldering materials deposition (eg. Sn-Pb) and metal plating (e.g Au)

Metal Etching



- Copper etching is usually performed using $FeCl_3$, $CuCl_2$, H_2SO_4+H_2O_2,… (High elimination cost!)

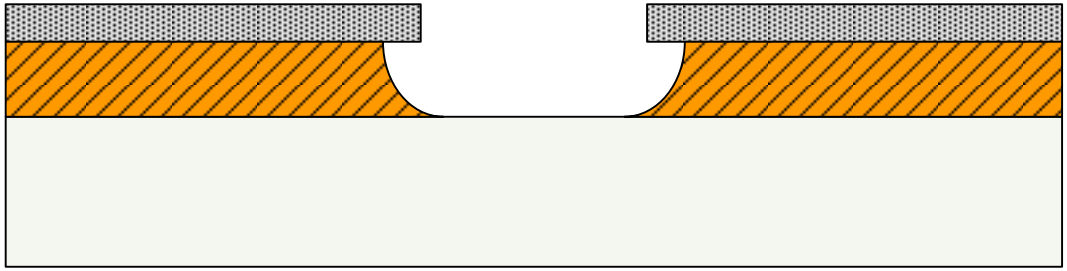
- Lateral etching



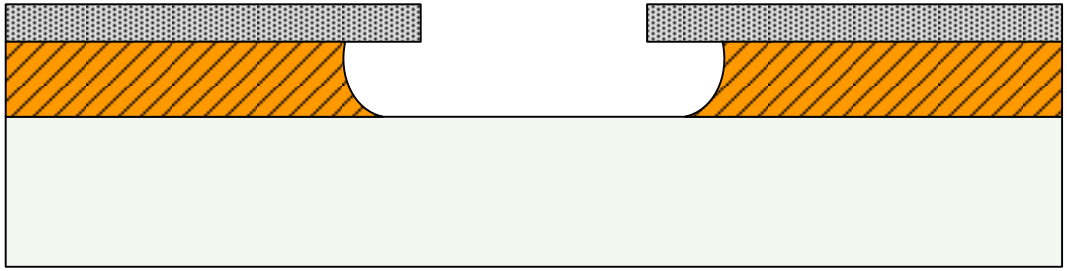
Lateral over-etch: p = 0.6/0.8 s (s = Cu thickness)

Metal Etching

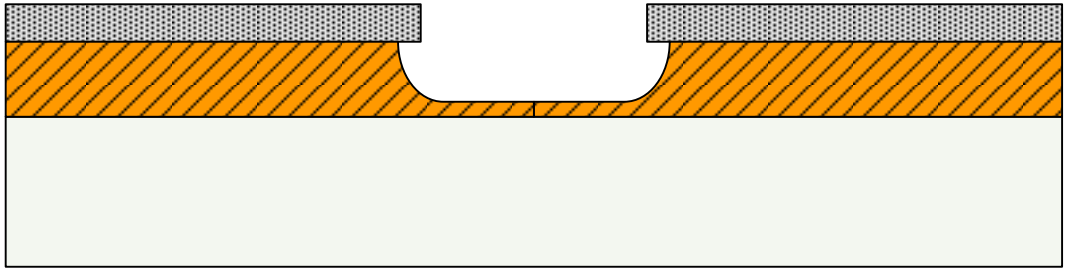
IDEAL



OVERETCH

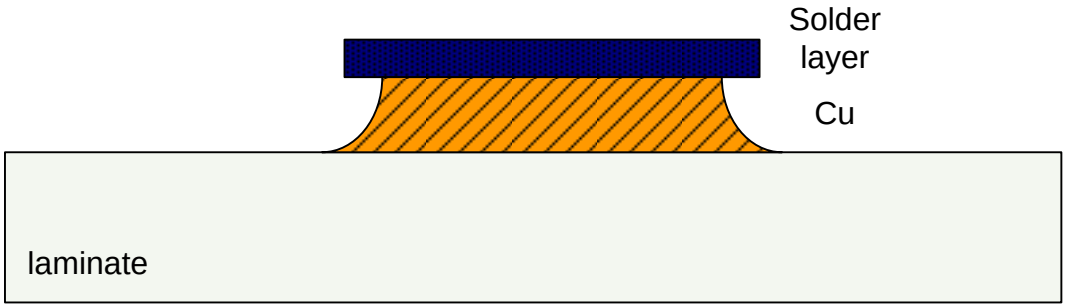


UNDERETCH



Finishing

- Metal plating



Metal plating is usually performed using soldering materials (e.g. Sn-Pb) or other metals (e.g. Au) used to protect copper metal traces.

Plating is carried out before Cu etching.

Heating can help to reduce plating overhang.

Finishing

- Solder resist
  - Melamminic or epoxydic resin used to protect metal traces from soldering material deposition.
  - Solder resist is usually deposed by screen printing and then cured. The typical green color of PCBs is given by the solder resist.
- Solder coating
- Legend layer
  - Typically white varnish deposed by screen printing on top of the solder resist layer. This layer is used to print information on components, on the board, etc.

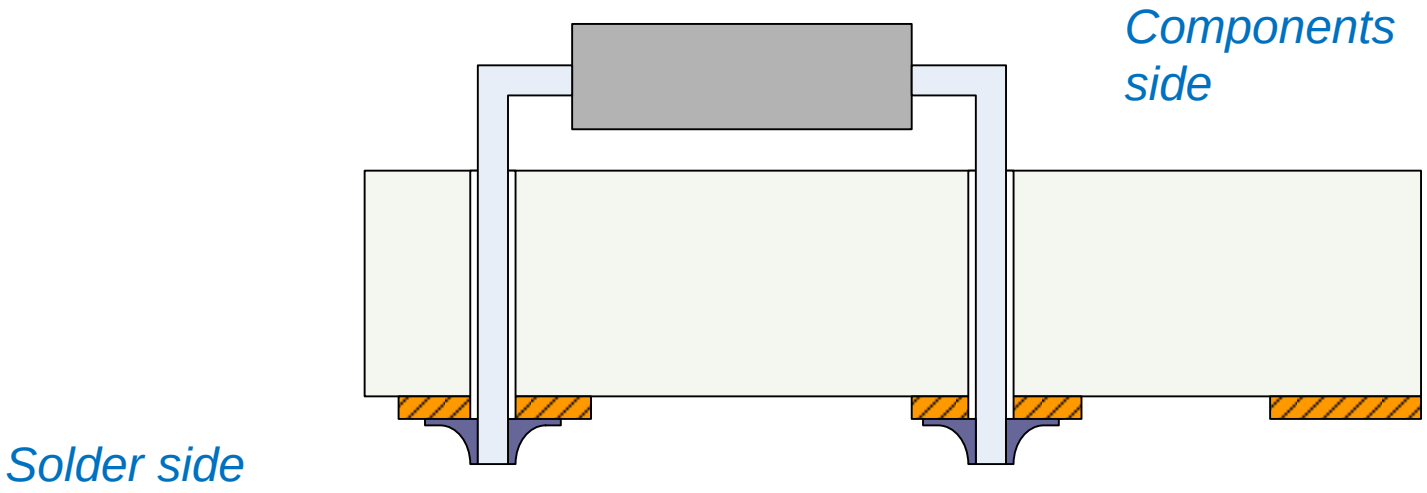
PCB Process Flows

- **Subtractive Flows**
  - Single layer
  - Dual Layer
    - Panel plating
    - Pattern plating
    - Tenting *
- **Additive flow**
- **Semi-additive flow**
- **Multi-layer circuits**

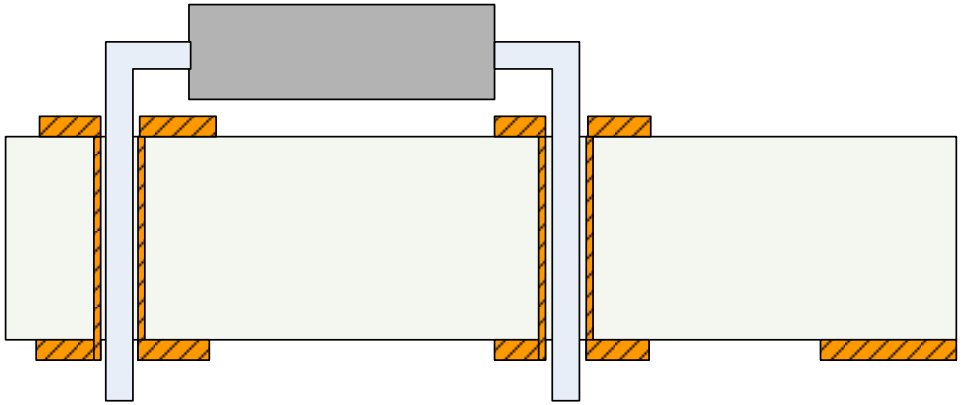
*not discussed in this presentation

Single Layer Circuits

- Simplest and cheapest process. All connections are carried out on the same side. Holes are not plated. Typically metal traces are defined by screen printing ("print and etch" or "P&E" process). A second (patterned) metal layer can be used on the components side as a ground plane.

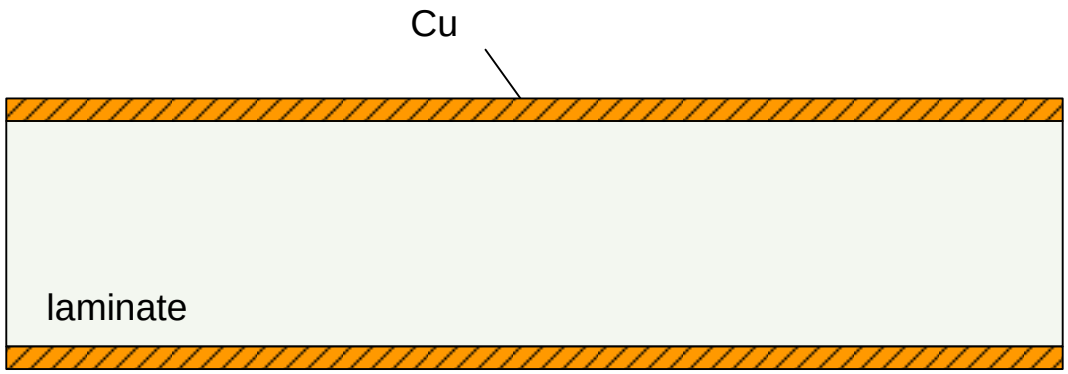


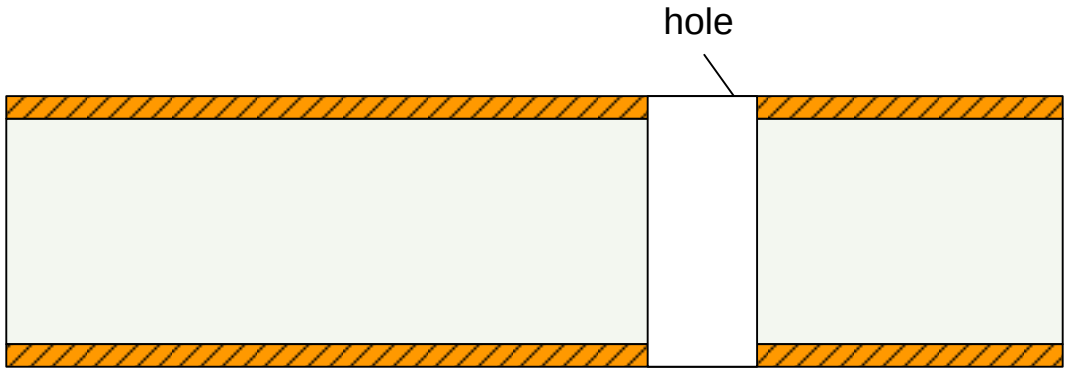
Dual Layer Flows



- The dual layer process with plated through holes is the most common. Connections are carried out on both sides. Plated holes improve device mounting and simplify connections. The two main (subtractive) process categories are:
  - Panel plating
  - Pattern plating

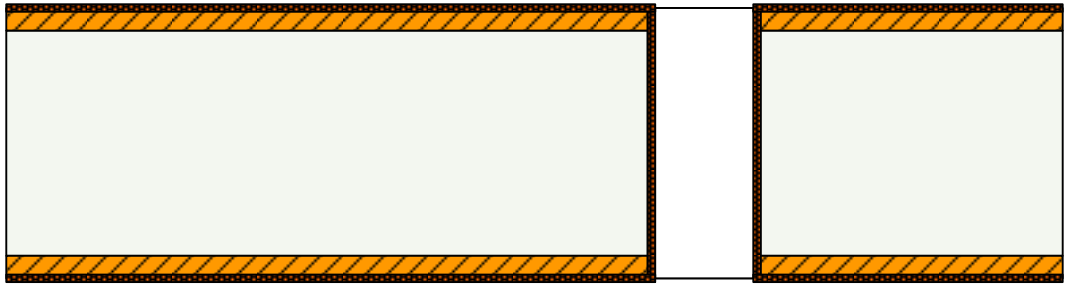
Panel Plating



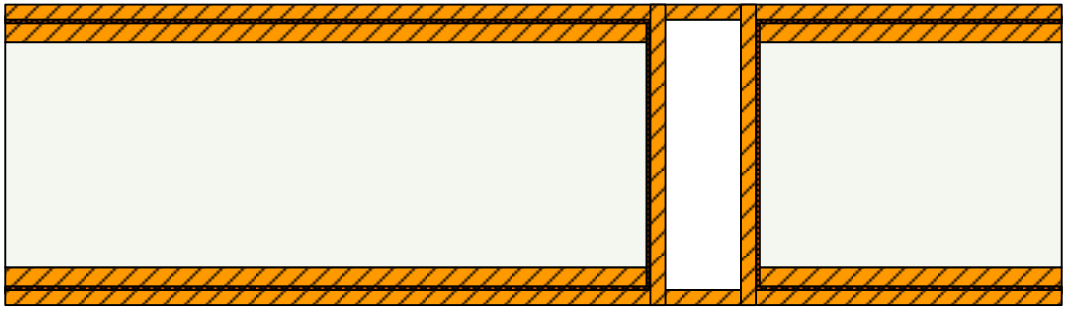


Holes drilling.

Panel Plating (2)

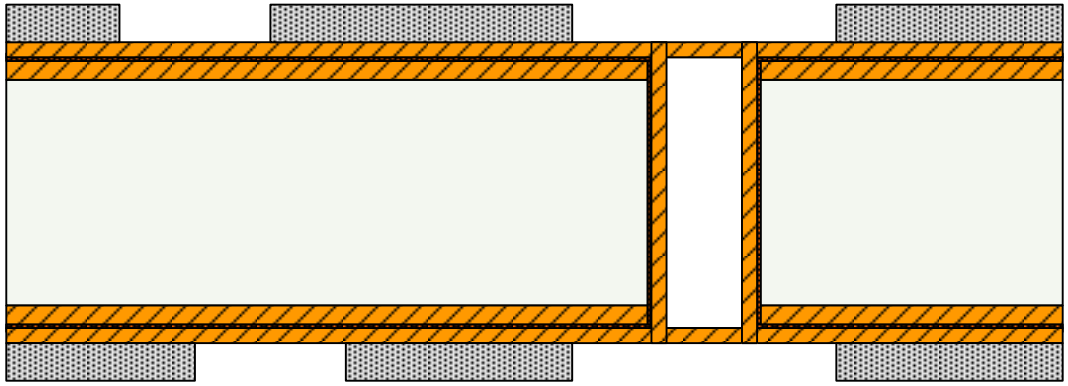


Electroless copper deposition (whole panel)

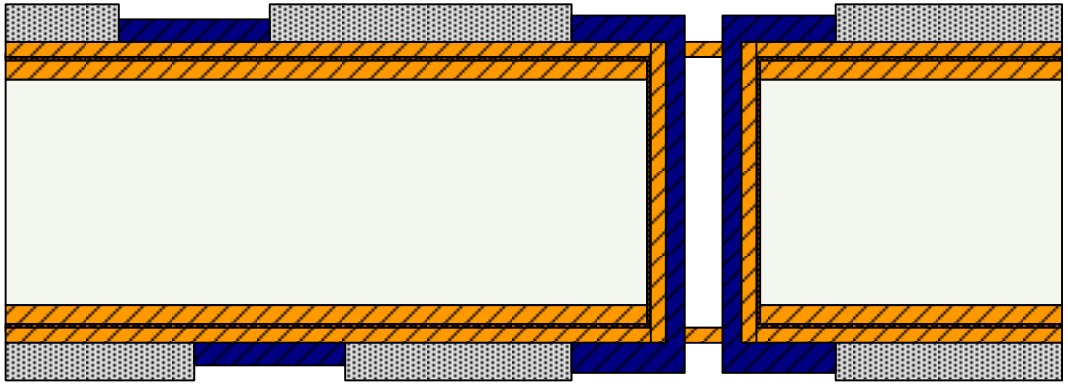


Electrochemical copper deposition (plating) on the whole panel

Panel Plating (3)

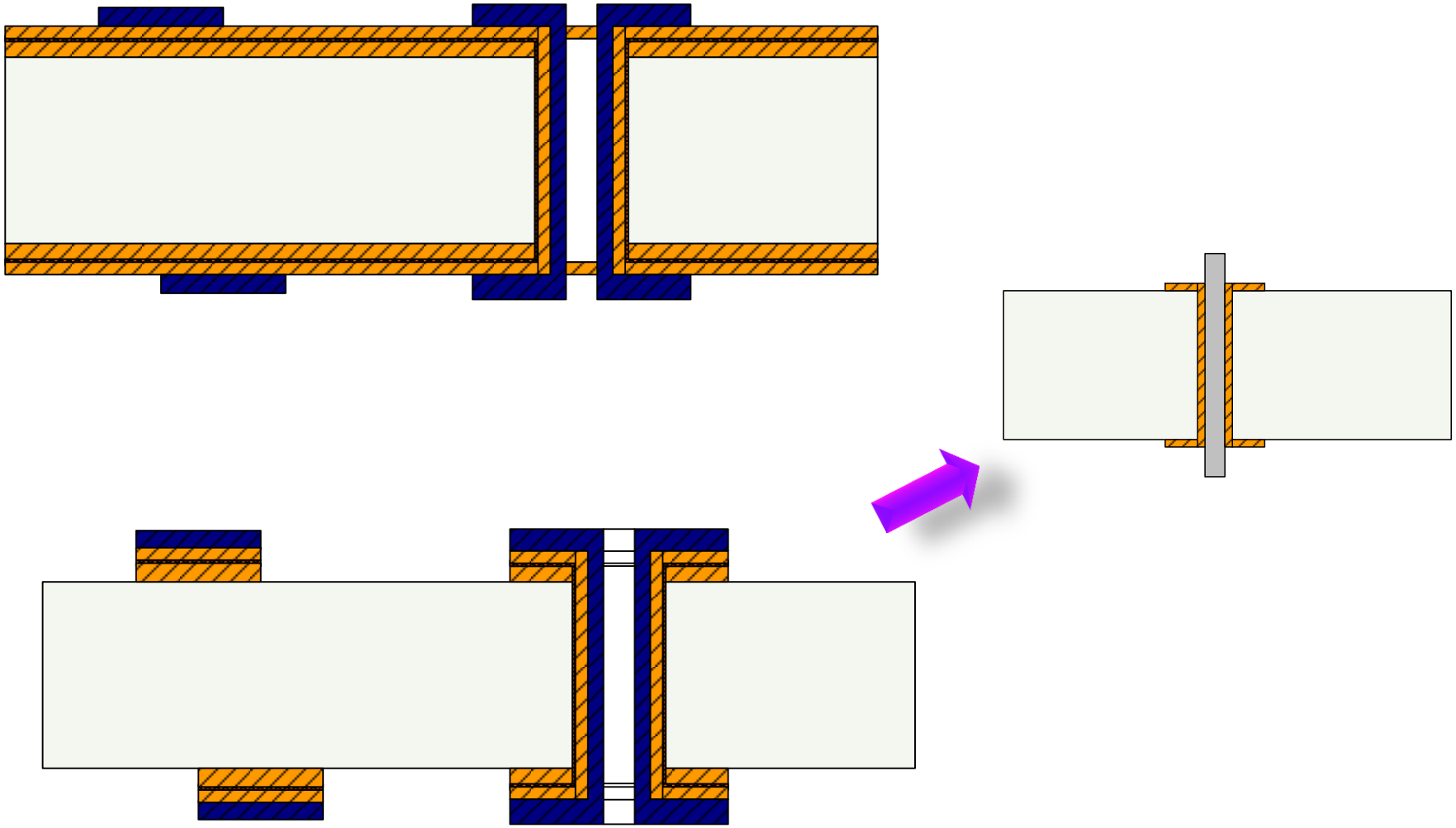


Plating resist deposition and development.

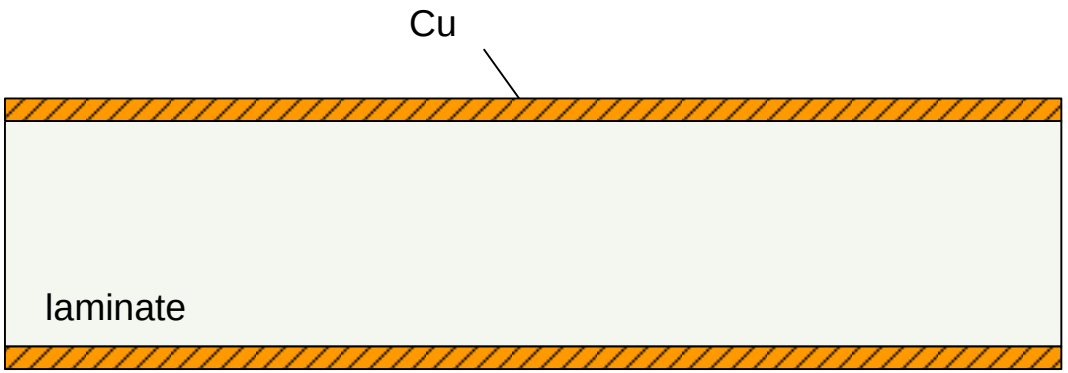


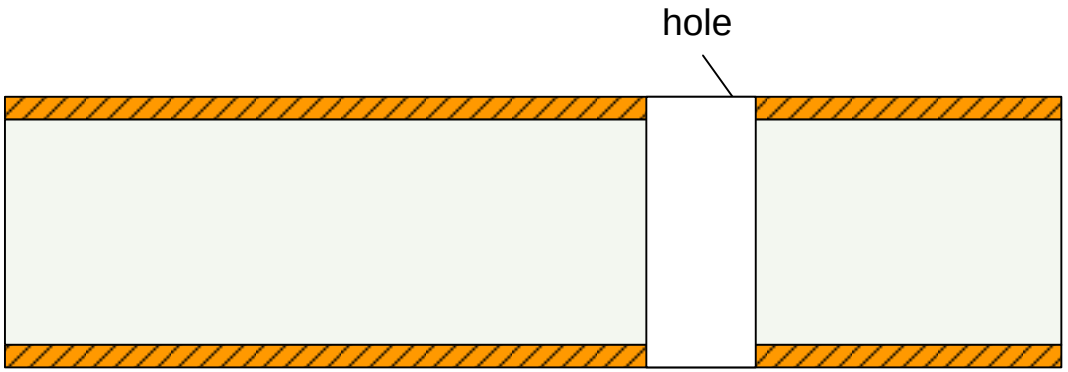
Selective metal / alloy (e.g. Sn-Pb) deposition.

Panel Plating (4)



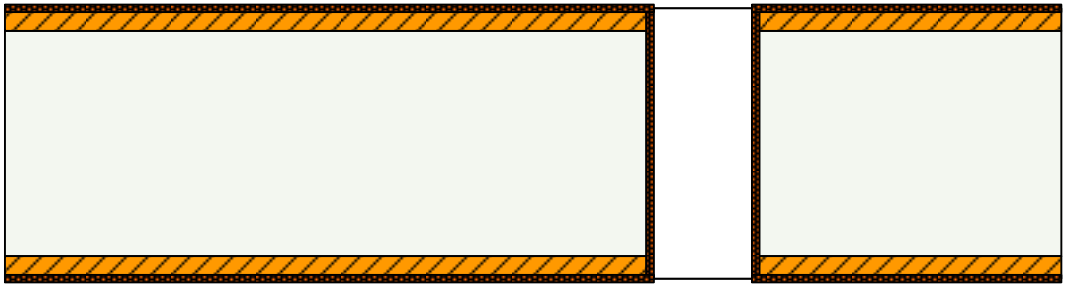
Pattern Plating



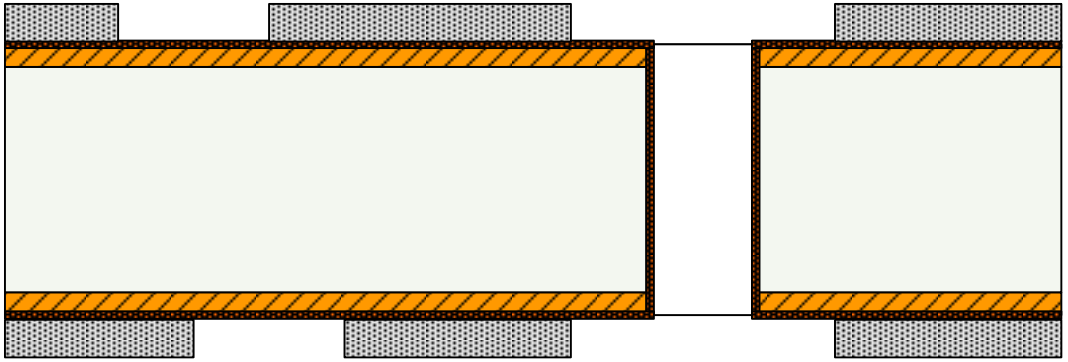


Holes drilling.

Pattern Plating (2)

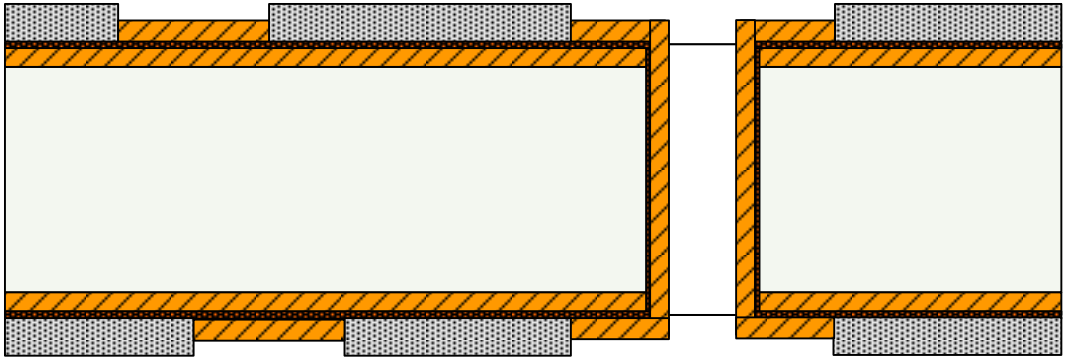


Electroless copper deposition (whole panel)

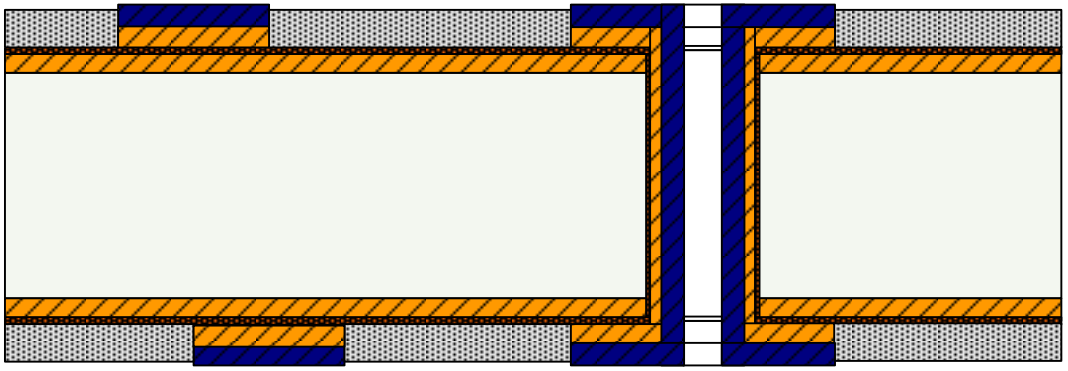


Plating resist deposition and develop.

Pattern Plating (3)

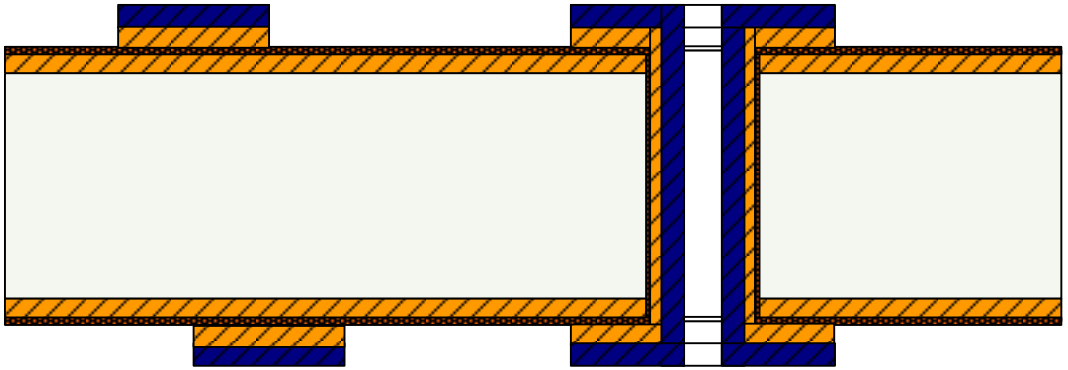


Selective electro-chemical copper deposition (plating)

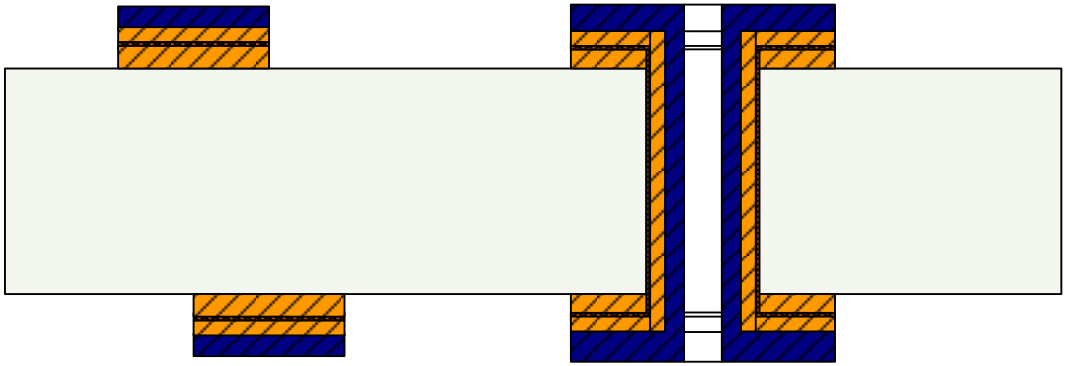


Selective metal (e.g. Sn-Pb) finishing.

Pattern Plating (4)

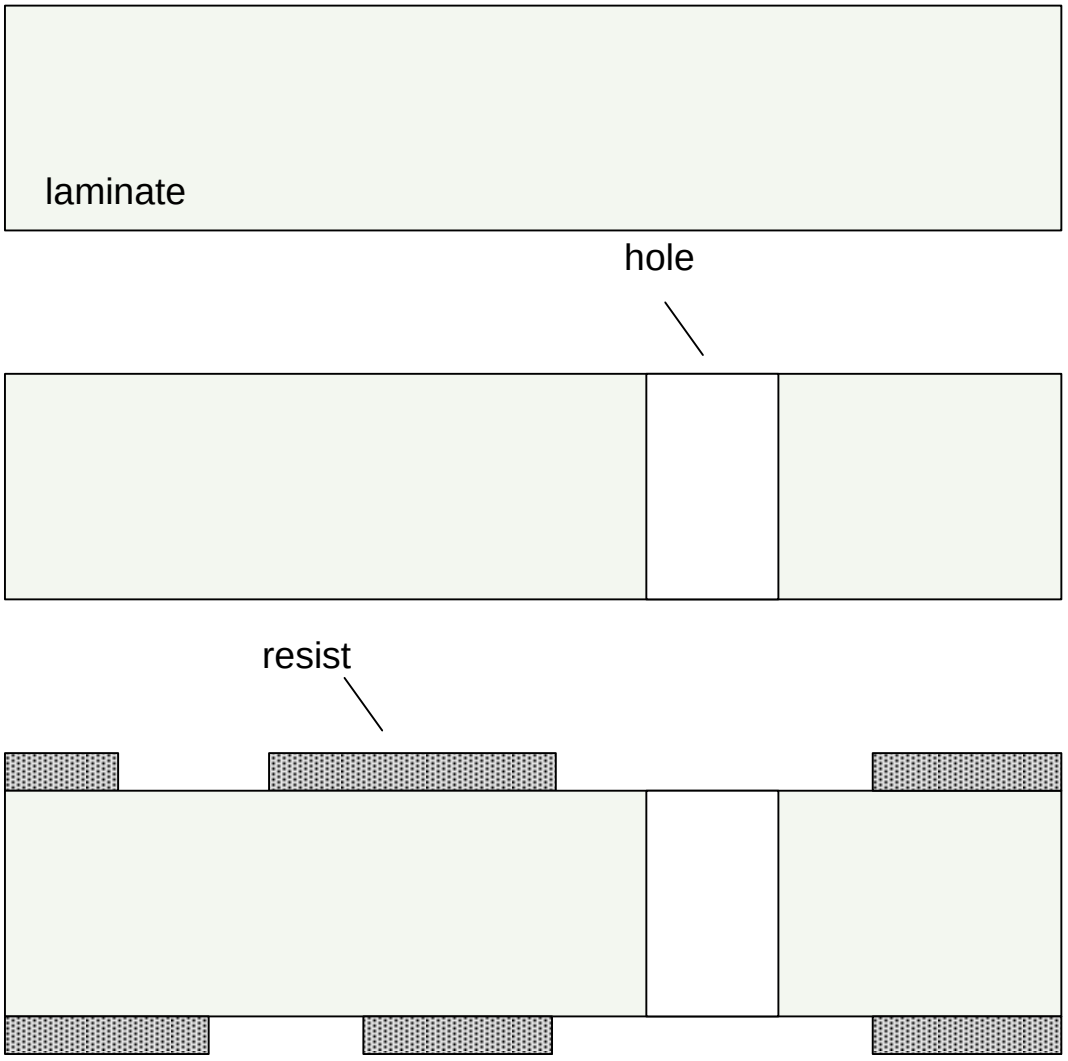


Resist removal



Copper etching.

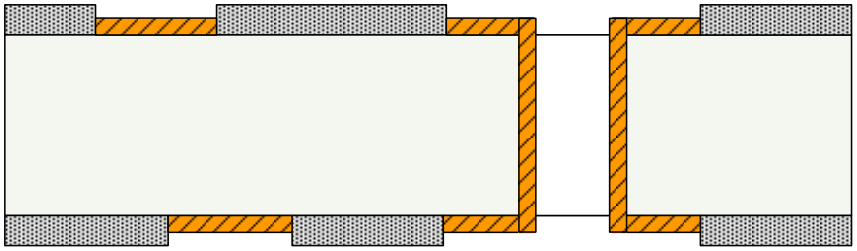
Additive Flow



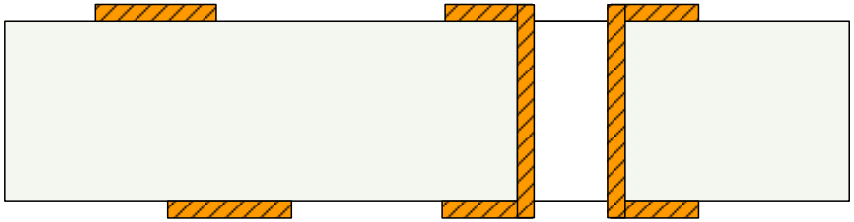
Hole drilling and surface activation.

Resist deposition, exposure and develop.

Additive Flow (2)



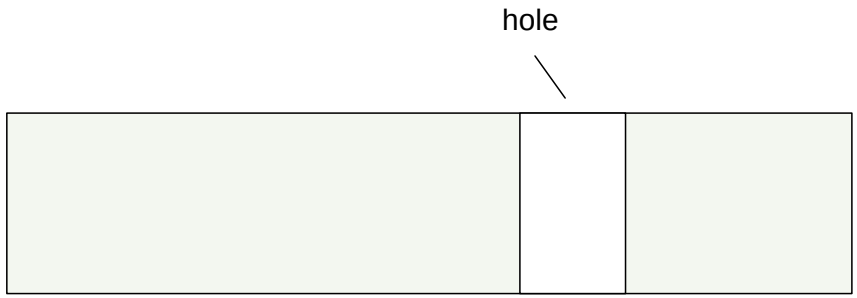
Selective electroless copper deposition.



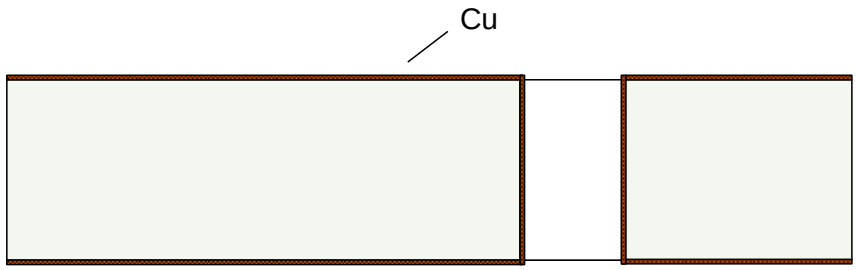
Resist removal.

In additive processes copper etching is not necessary. However copper deposition must be performed using electroless technique, increasing processing time and costs.

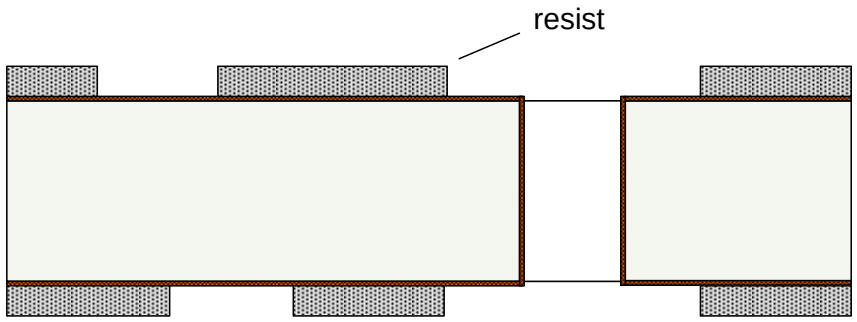
Semi-additive Flow



Hole drilling and surface activation (catalyst).

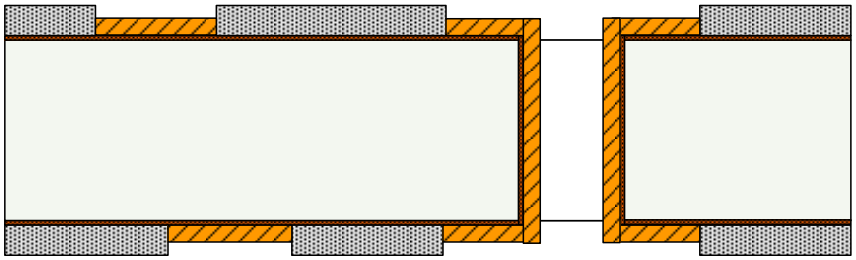


Non selective chemical copper deposition.

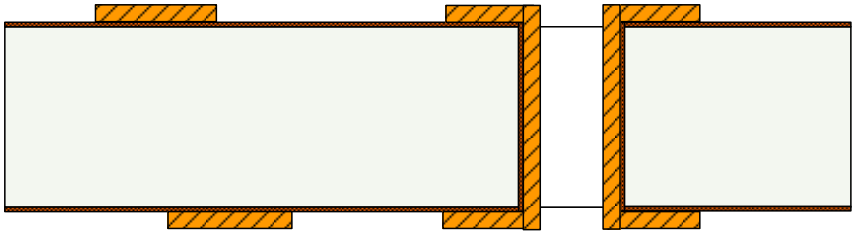


Resist deposition, exposure and develop.

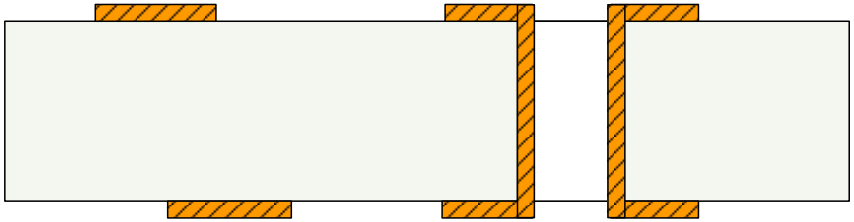
Semi-Additive Flow (2)



Selective electro-chemical copper deposition

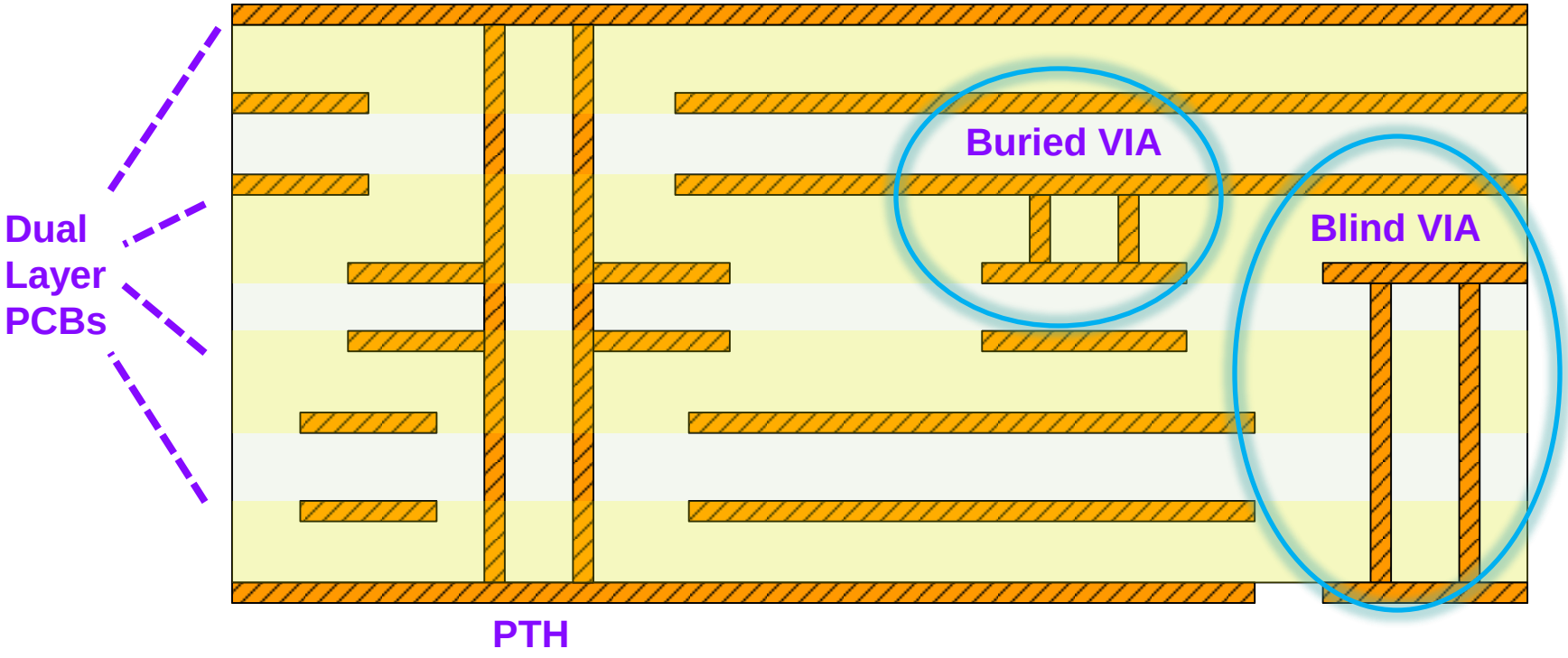


Resist deposition, exposure and develop.



(Electroless) copper etching

Multilayer Circuits



Several (thin) dual-layer PTH boards are stacked together with prepeg layers. PTH in each dual layer board act as VIA holes. Through the board and blind plated holes are also possible. Only external layers need plating.

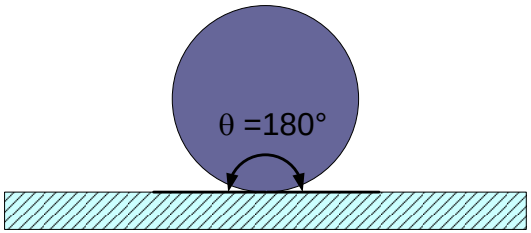
Printed Circuits

- Printed Circuits Materials.
- Technological steps
- Production technologies:
  - single layer, double layer and multi-layer boards
- **Assembly:**
  - Soldering techniques
  - Insertion mounting
  - Surface mounting

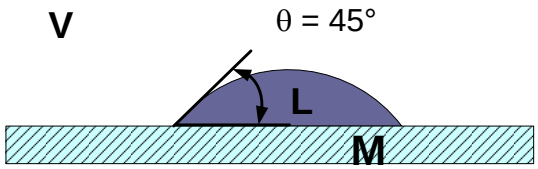
Soldering

- Soldering is defined as a metallurgical joining technique involving a molten filler metal which wets the surface of both metals to be joined and, upon solidification, forms the bond.
- The bonding occurs at the interface of the two metals and is strongly dependant on the wettability (or solderability) of the base metal by the molten alloy.
- Wettability is measured by the dihedral (contact) angle: a good angle is 75° or less. Wettability is improved using flux to make the surface clean and active.

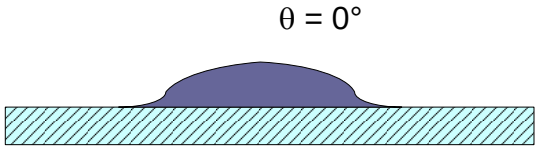
Non-wetting (θ=180°)



Partial wetting (0<θ<180°)



Total wetting (θ=0°)



Soldering Process Steps

1. Application of heat to solder:
   - Brings the solder alloy up to the melting point
   - This involves the formation of oxides that represent a barrier preventing alloy formation
2. Chemical preparation of surfaces
   - Flux attacks metal surfaces removing oxides and other dirt
3. Solder wetting and solder joint formation
   - Temperature and time control are key for a good soldering joint formation.
4. Solder quenching
   - Solder joint must be in solid state before PCB handling.

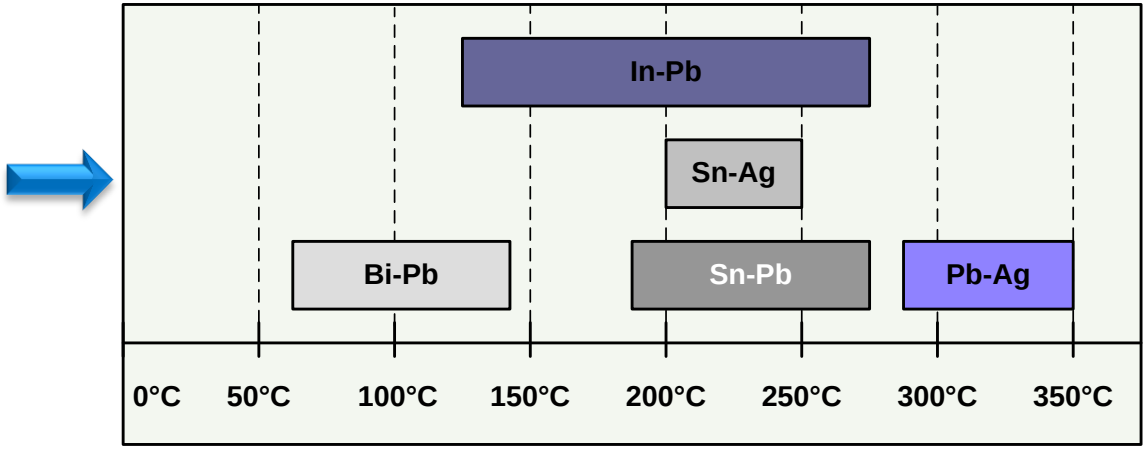
Soldering Materials

- Most common soldering alloy is tin-lead (Sn-Pb).
- Eutectic composition (37% Pb 63% Sn) offers the lowest melting temperature (183°C) and best wettability. Molten phase is avoided, minimizing the probability of "cold solder".
- Minimum solder temperature depends on the minimum cavity width to be wet by the solder alloy. For eutectic SnPb:

$$T_{min} = 183 + 8.6 / d$$

  where d is in mm and T in °C
- 70% Pb composition may be used for high temperature applications

MELTING TEMPERATURE RANGES OF THE MOST COMMON SOLDERING ALLOYS



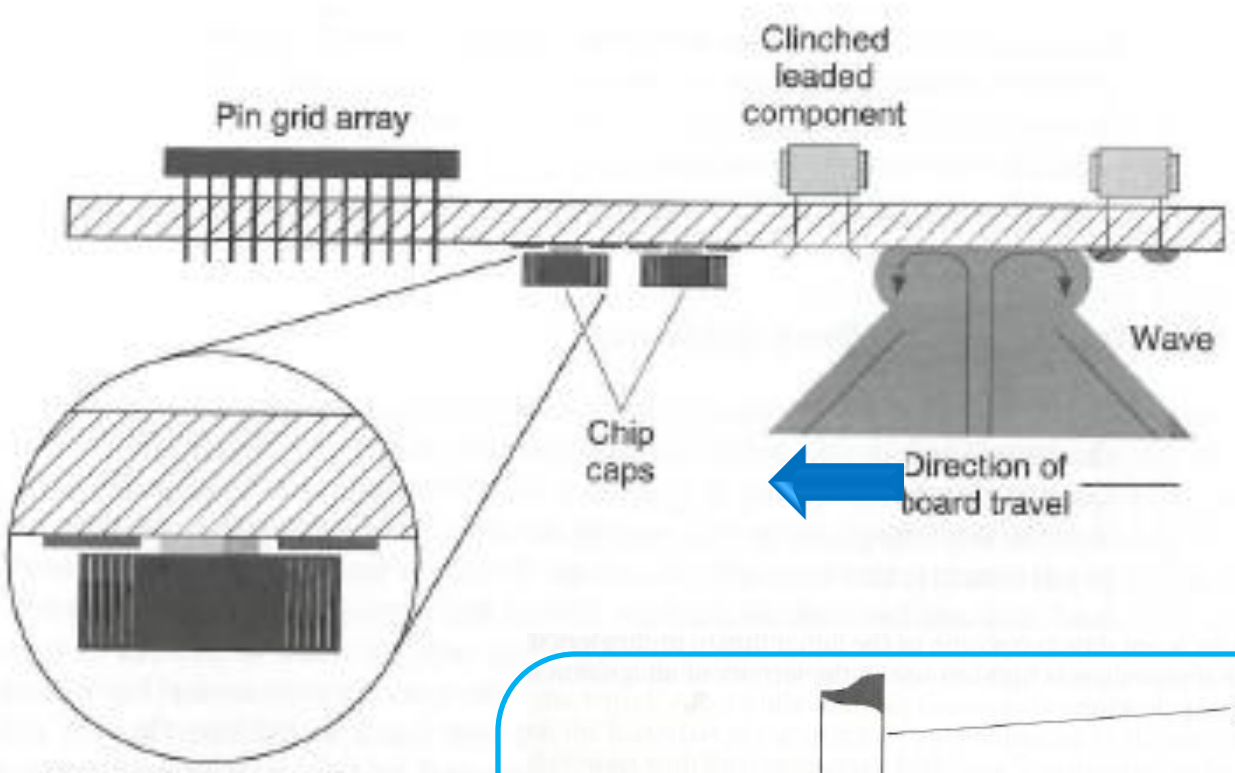
Fluxes

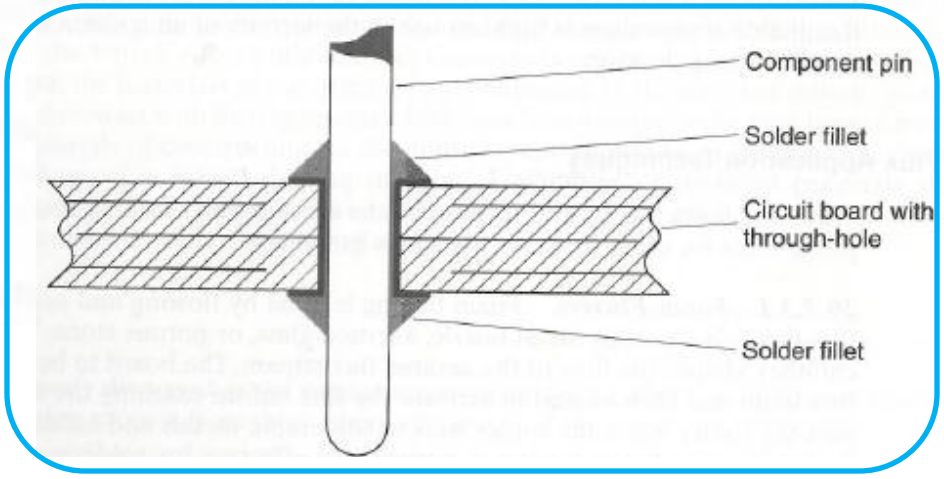
- The flux is a chemically active compound that, when heated, removes minor surface oxidation, minimizes oxidation of the basis metal, improves wetting and promotes the formation of an alloy layer between solder and basis metal.

Soldering Techniques

- Soldering techniques can be divided in:
  - Mass soldering: all components are soldered at the same time. In this category fall:
    - Wave soldering: Once predominant, still used for through-hole components.
    - Oven reflow
    - Vapor phase reflow
  - Direct energy methods: only one or a small number of components are soldered at the same time. In this category fall:
    - Hot-gas soldering
    - Hot-bar soldering
    - Laser soldering
- Each method has its advantages and disadvantages. Mass soldering techniques are by far the most common for high volume manufacturing. In some instances direct energy methods are being increasingly used.

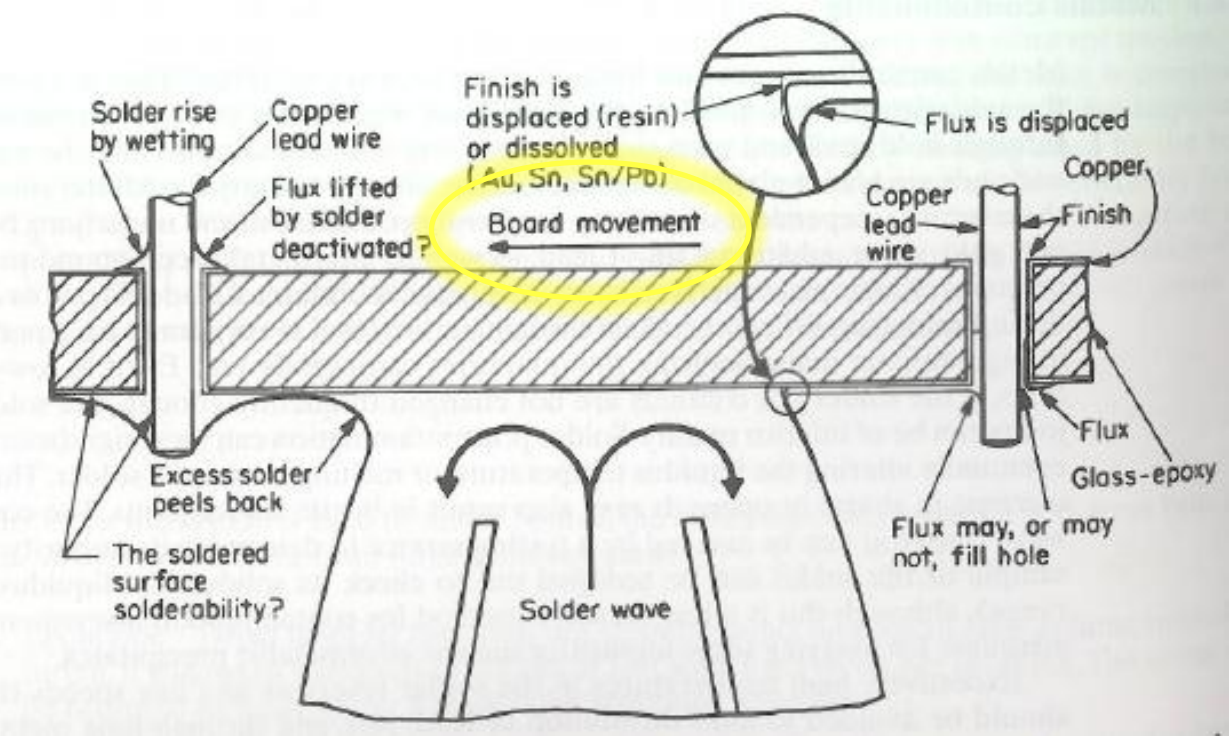
Wave Soldering





- Wave soldering utilizes a reservoir of molten solder pumped and circulated to form a standing wave.
- The components stay in the solder wave for a very short time. For this reason pre-heating is critical to ensure proper flux activation

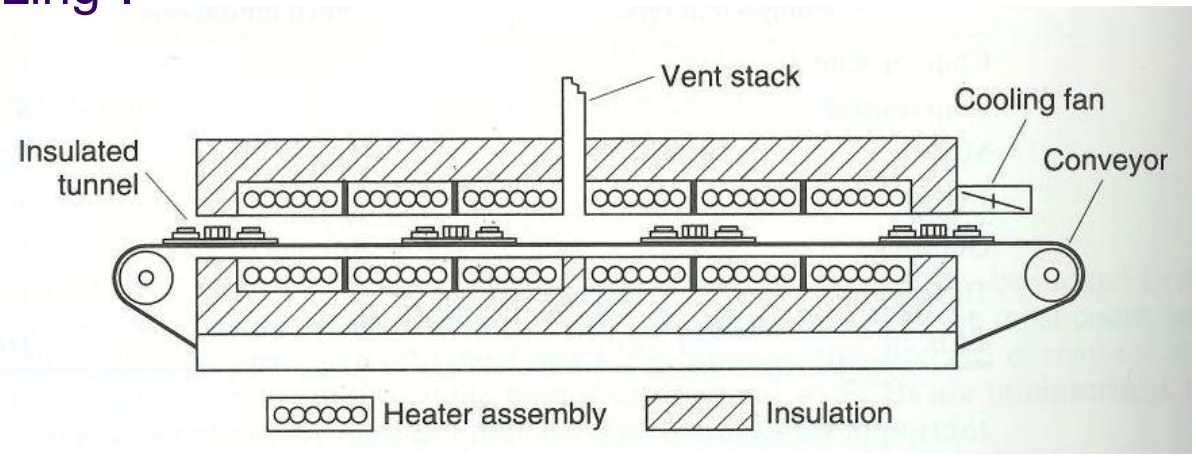
Wave Soldering: Detail



- Solder is prone to rapid oxidation the air interface. A stationary film composed mainly of tin oxide and other contaminants (dross) forms at the surface.
- The board meets the crest of the wave, disrupting the oxide film. The components are then immersed in the flowing solder.
- Solder resist prevents wetting of metal traces.
- As the board exits the wave and cools down, solder joints are formed.

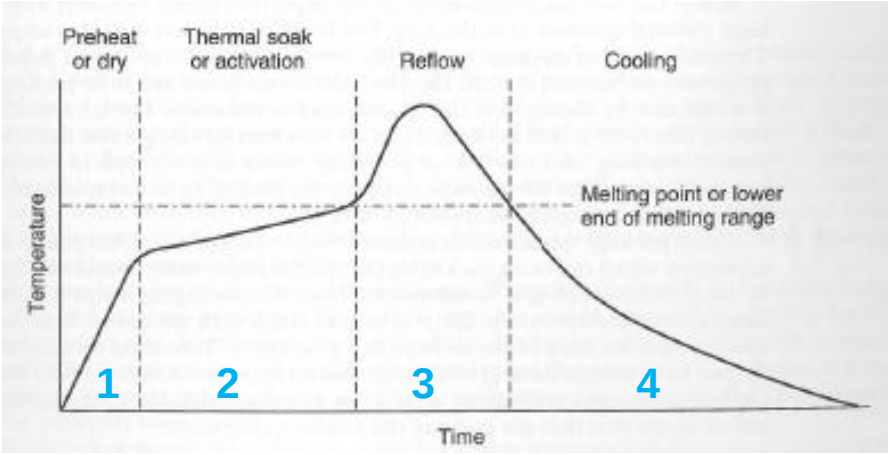
Oven Reflow Soldering

- In reflow soldering, primarily used for SMD, solder paste is printed on the board and on the device terminals. The paste contains both the solder and the flux
- Components are placed on the solder paste and then go through a tunnel oven, made of several sections at different temperatures
- Through the tunnel oven the temperature gradually raises, activates the flux in the solder paste and finally liquefy the solder
- The temperature cycle must be carefully controlled. Rapid heating of the solder paste is known to result in solder ball formation and flux "caramelizing".



Reflow Profile

- The temperature profile is dependent on the solder and flux in use.
- Rapid temperature changes my induce thermal shocks, damaging some of the components.
- The thermal cycle is usually divided in 4 phases:
  1. **Preheat/drying**: vehicle volatiles are evacuated from the solder paste.
  2. **Activation**: slow and prolonged heating that brings the flux to the activation temperature.
  3. **Reflow spike**: this phase is characterized by a rapid rise in temperature beyond the melting point (by about 20-50°C)
  4. **Cooling**: board temperature is brought well below the melting point.



Conductive Adhesives

- Increasingly used for integrated circuits using exposed pad packages or using chip-on-board techniques
- Adhesive material (typically resin) is made conductive adding a fine conductive dust (typically Ag)
- The conductive resin is dispensed on the board pads with the same tools used for SMD attach in mixed technologies. Parallel deposition is possible for high volume manufacturing.
- The main advantages are:
  - Absence of thermal shocks (Epoxy resins are used with polymerization temperatures of 80-100°C)
  - Compatibility with Au
  - No-clean process
- The main disadvantages are:
  - High cost
  - Long polymerization time

SMT vs IMT: Conclusions

IMT		SMT
☹	Miniaturization	☺
☹	Automation	☺
☹	Electrical performances	☺
☹	Production (variable) costs	☺
☺	Production tools (fixed) costs	☹
☺	Thermal excursion	☹
☺	Testing	☹

References

- References:
  - C.F. Coombs, “Printed Circuits Handbook”, Fourth Ed., McGraw Hill

- Reading Material:
  - Dispense di G. Torelli e S. Donati, *“Tecnologie e Materiali per l’Elettronica”*, Ed. CUSL