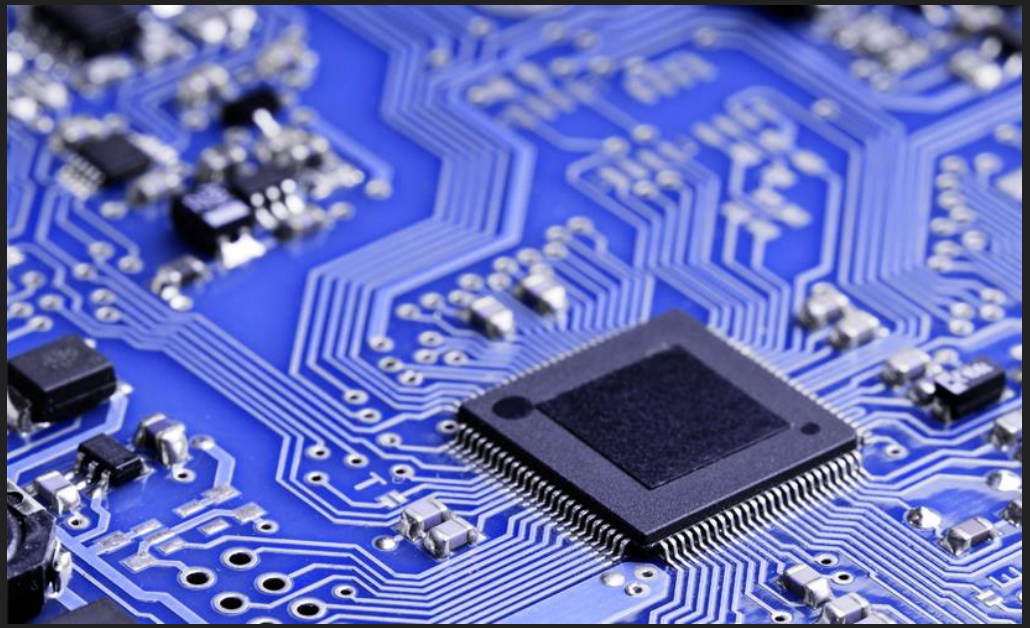
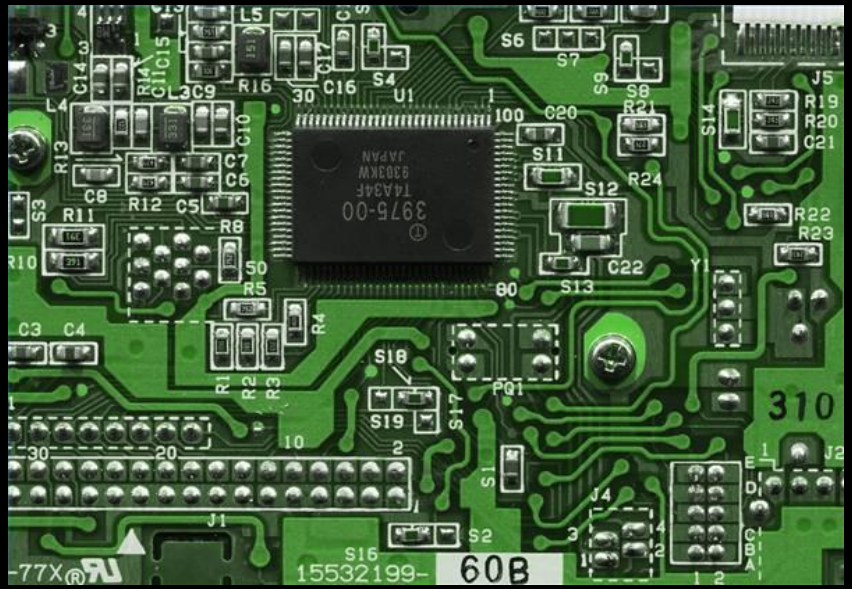
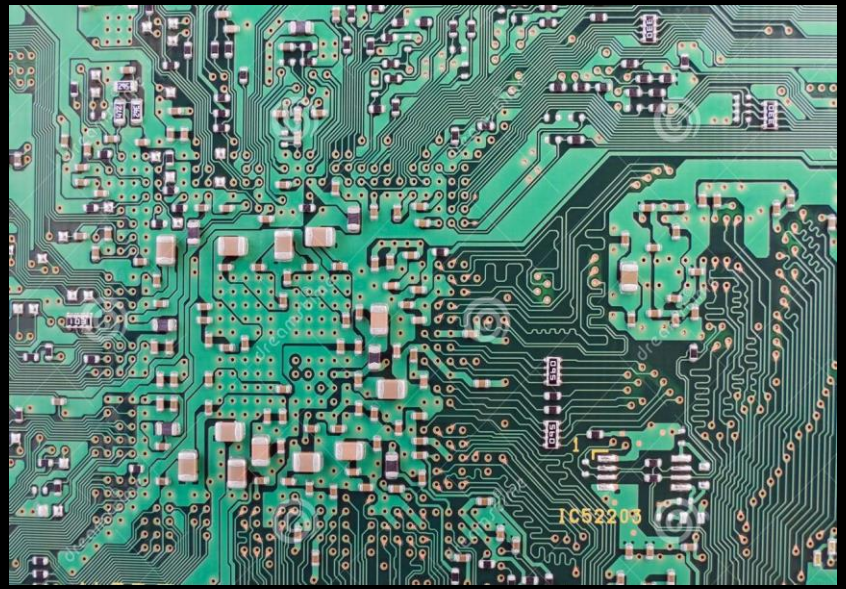


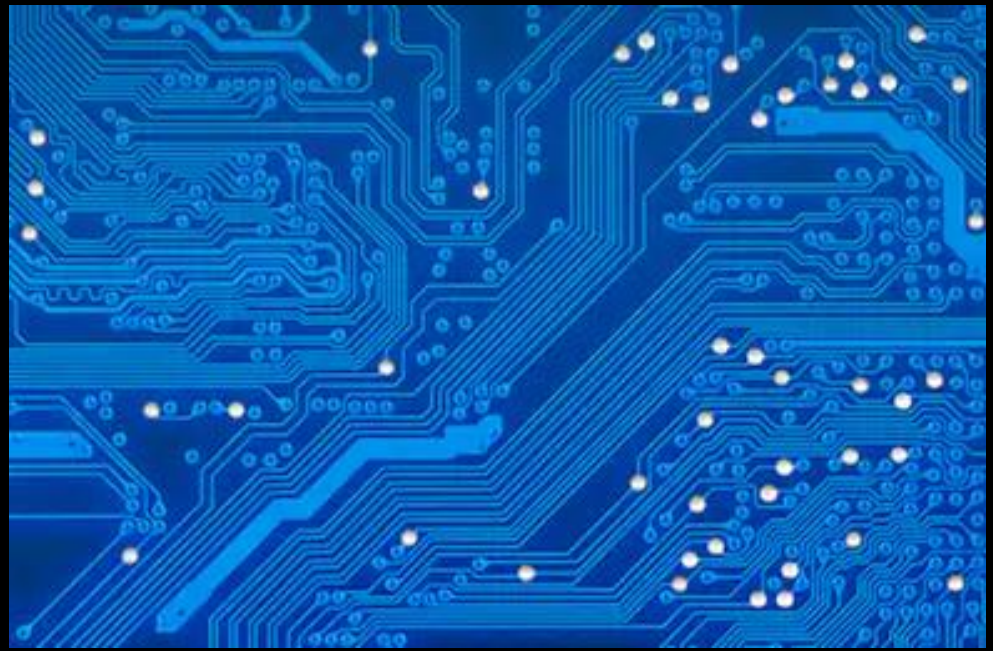
PCB Designing

PCB
Printed Circuit Board

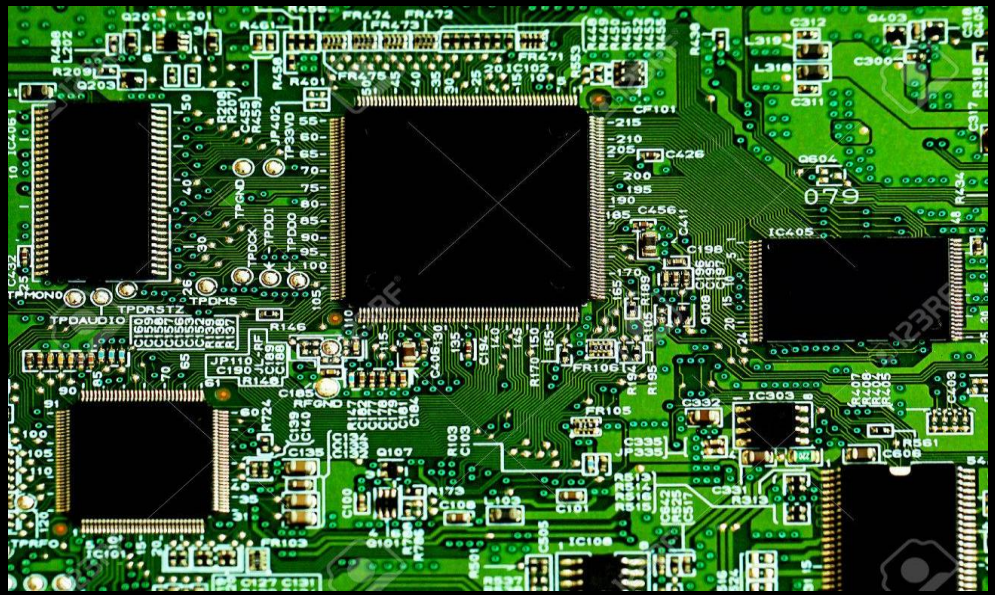




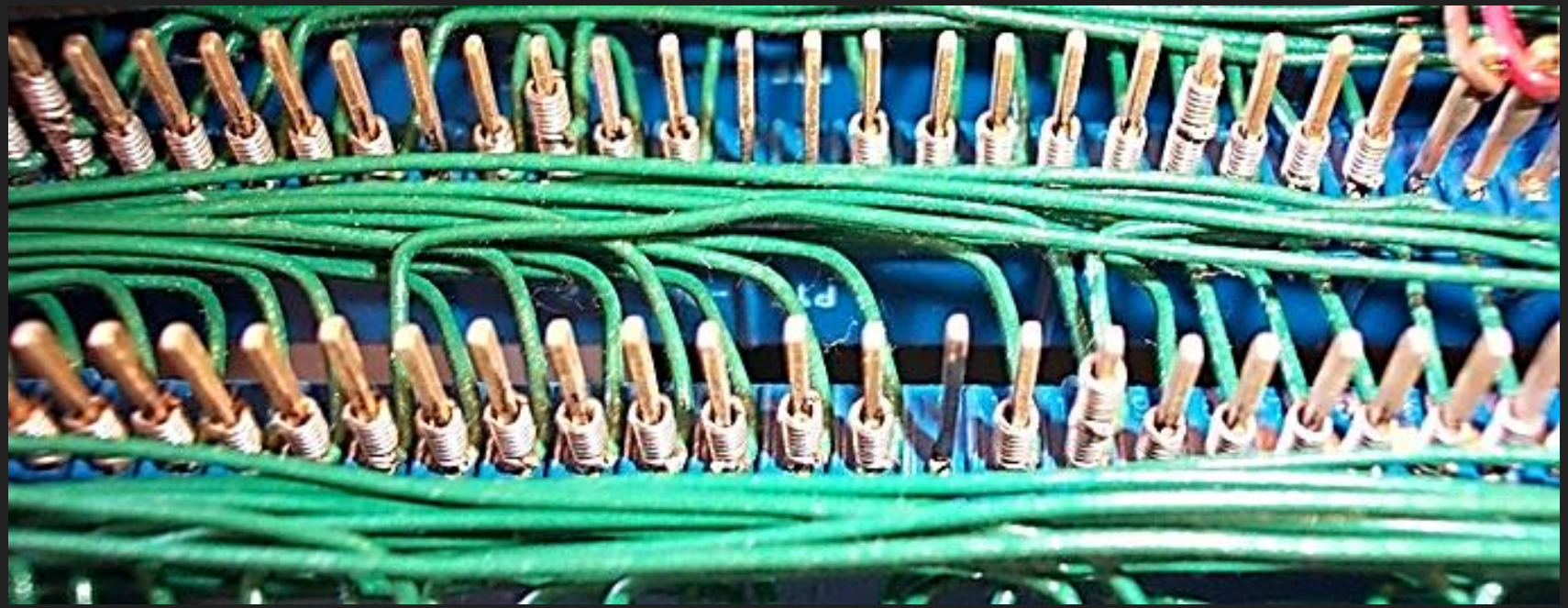
Printed Wiring Board



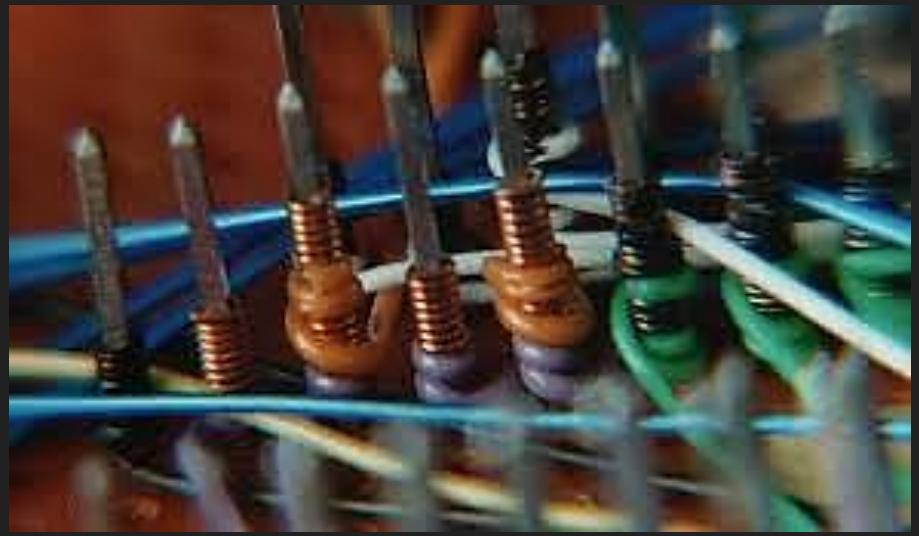
Printed Wiring Cards

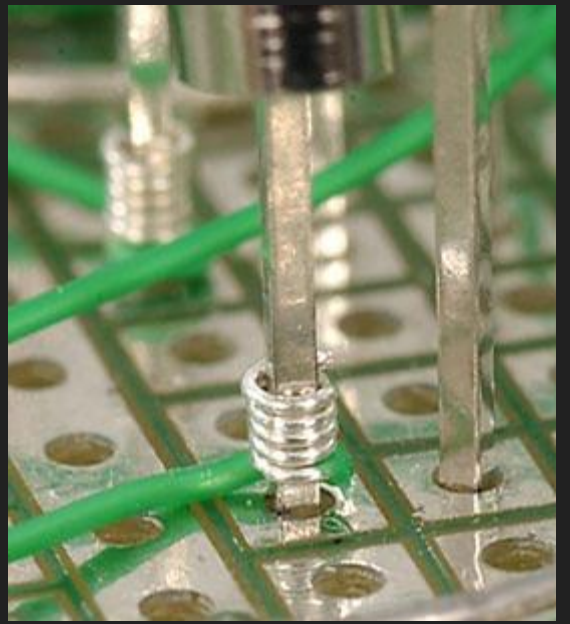


Why PCB designing?

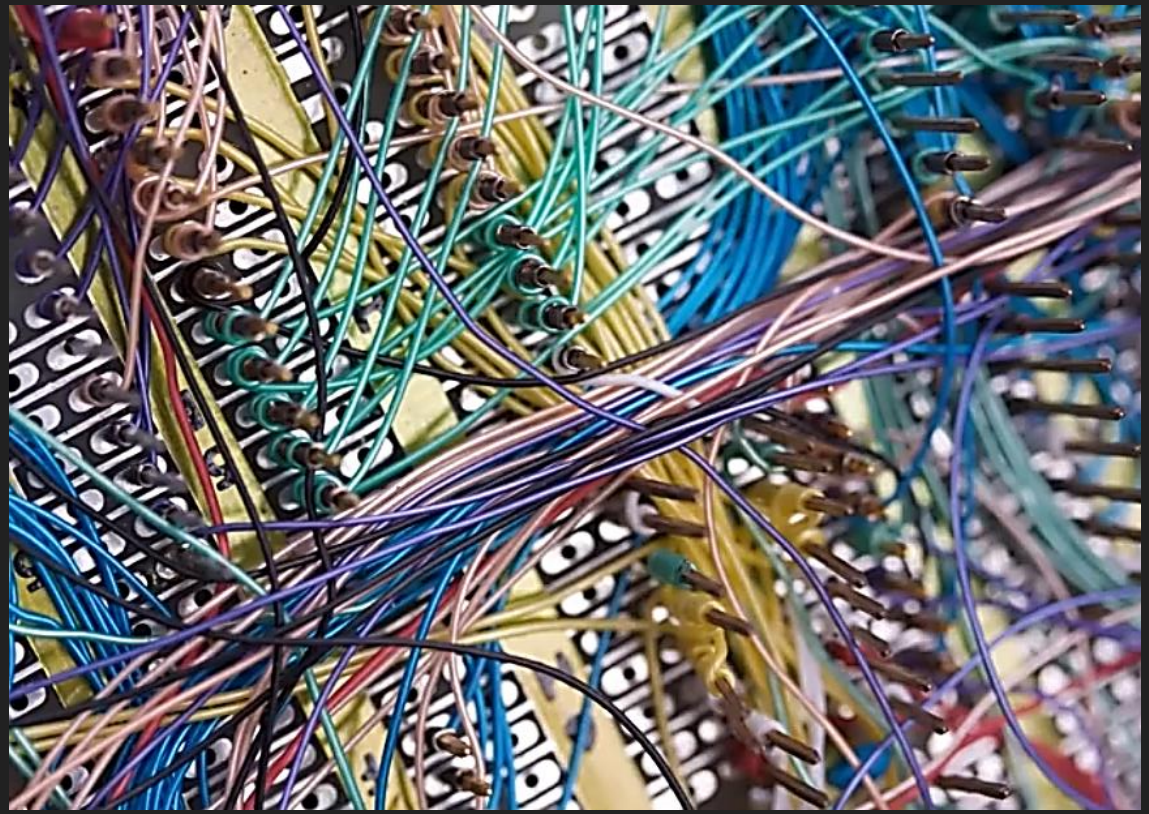


Why PCB designing?

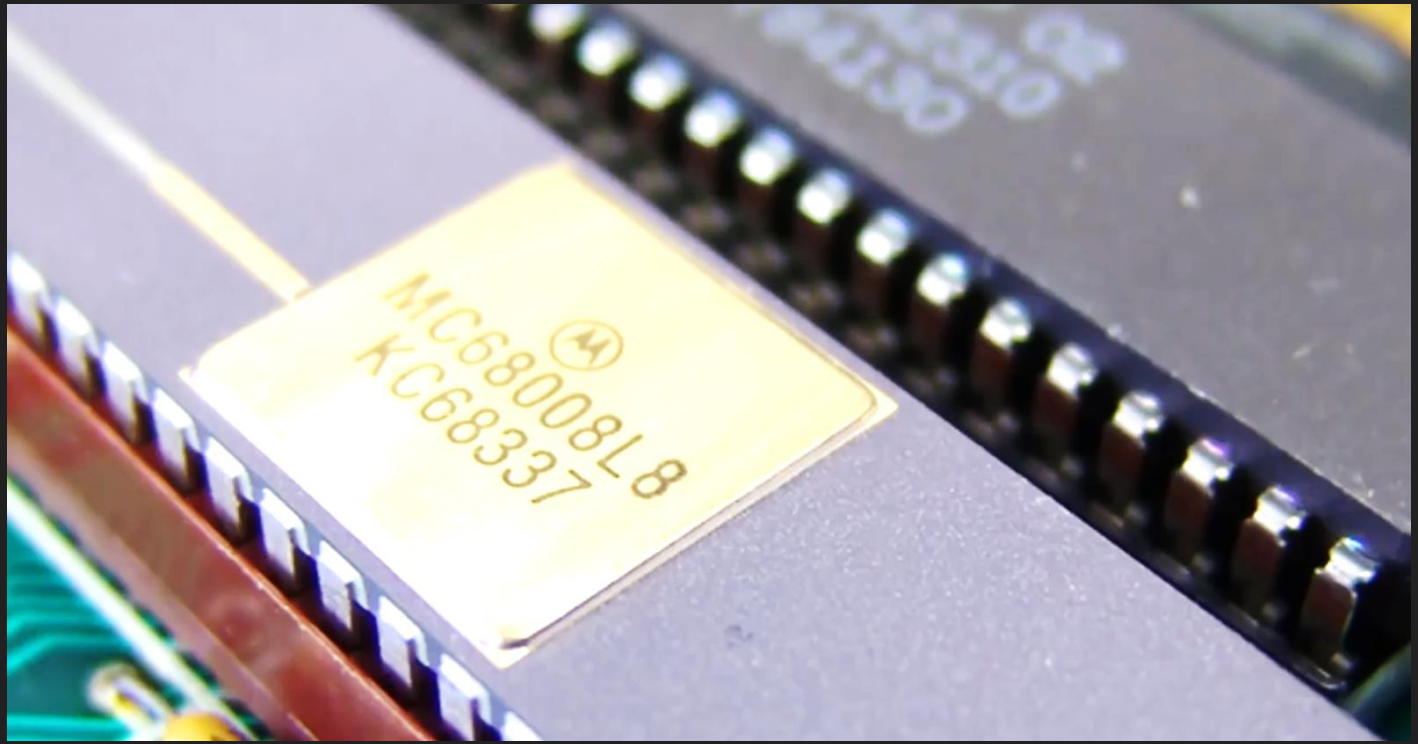




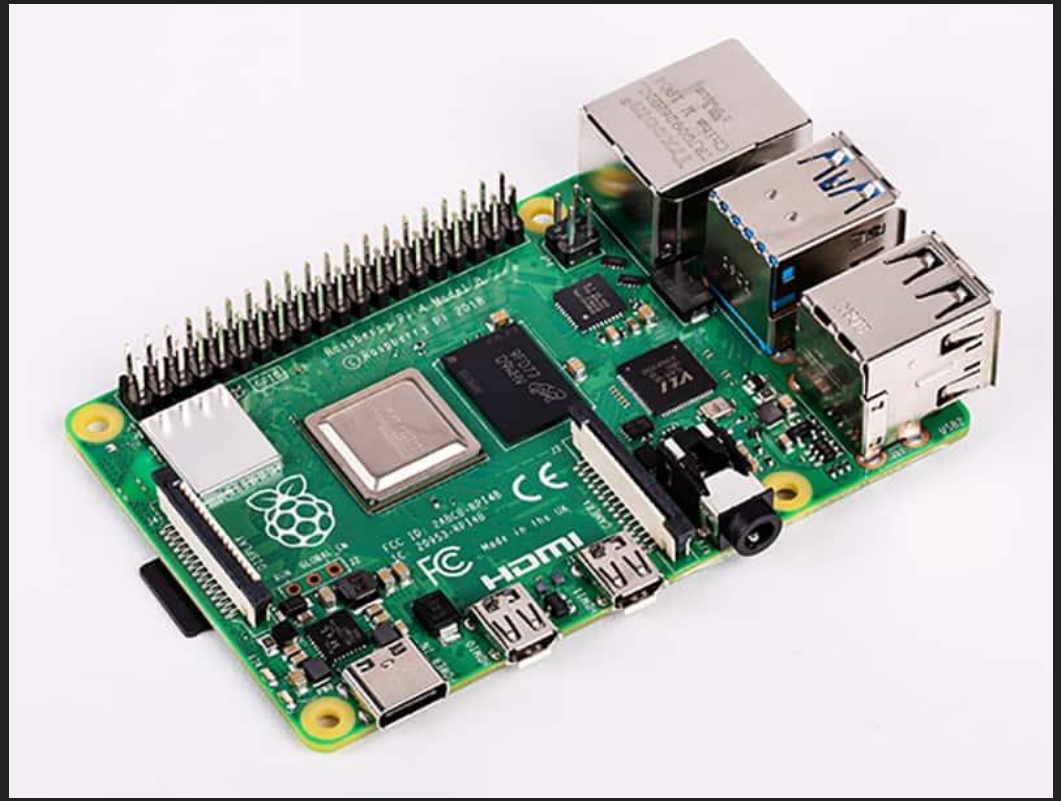
Macintosh Wire Wrap Board

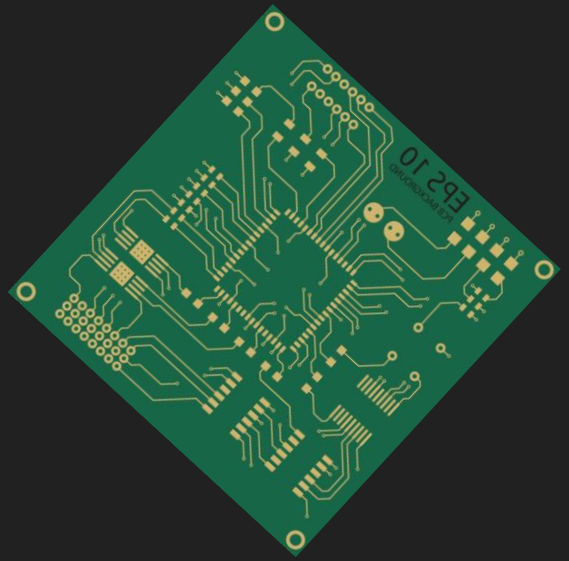


Motorola 68008

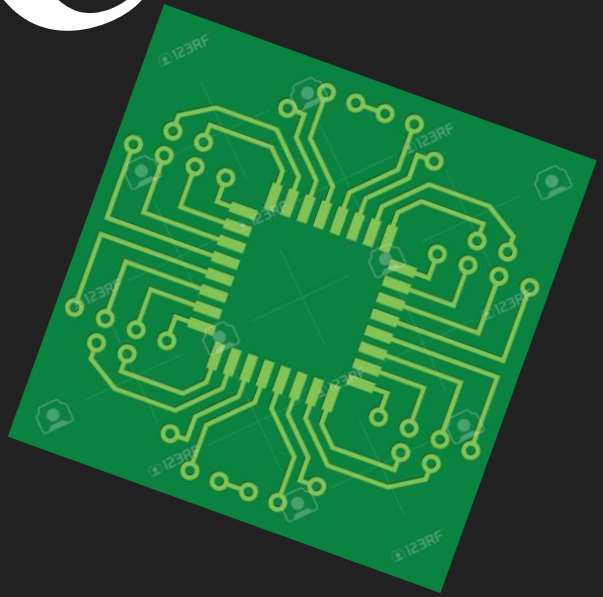


Surface Mount Technology





What is PCB ?



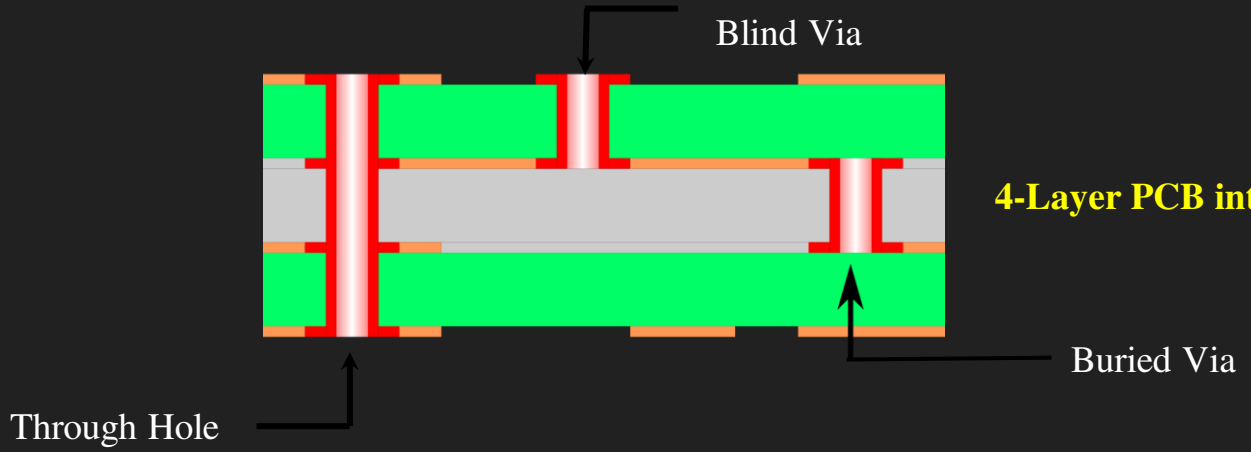
- It is a board that supports mechanically,
- electrically connects electrical or electronic components using **conductive tracks**, **pads** and
- other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate.
- components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it.

- can have **multiple** copper layers like (2-layer, 4 layer, etc)
- two-layer board has copper on both sides
- multi-layer boards sandwich additional copper layers between layers of insulating material
- Conductors on different layers are connected with **VIAS**
- **Through-hole** component leads sometimes also effectively function as vias
- After two-layer PCBs, the next step-up is usually four-layer
- 4-layer: two layers are dedicated as **power supply** and **GROUND PLANES**, and the other two are used for signal wiring between components

What is

VIA ?

- Vertical Interconnect Access
- an electrical connection between layers in a physical electronic circuit that goes through the plane of one or more adjacent layers
- are copper-plated holes that function as electrical tunnels through the insulating substrate



4-Layer PCB internal layered view

What is

GROUND PLANES ?

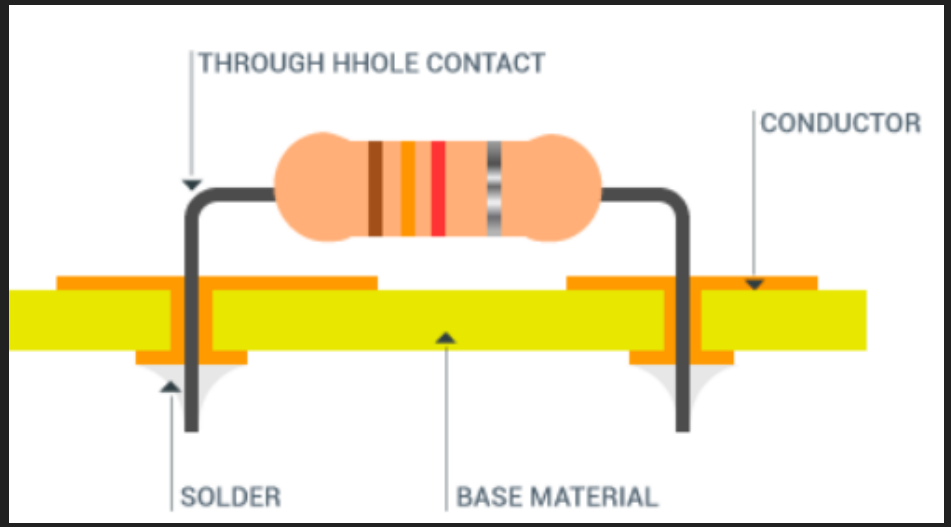
- an electrically conductive surface, usually connected to **electrical ground**
- ground plane is a large area of copper foil on the board which is connected to the power supply ground terminal and serves as a return path for current from different components on the board
- made as large as possible, covering most area of the PCB which is not occupied by circuit traces
- In multilayer PCBs, it is often a separate layer covering the entire board
- The large area of copper also conducts the large return currents from many components without significant voltage drops, ensuring that the ground connection of all the components are at the same reference potential

PTH

&

NPTH

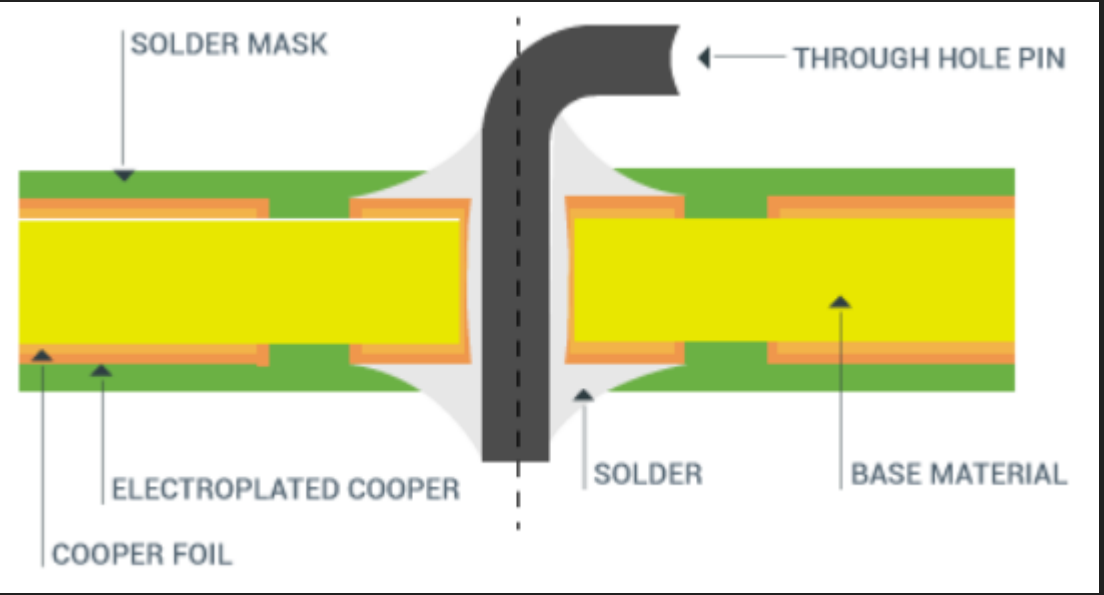
- Through-hole technology, also called as "**thru-hole**"
- This technology holes go completely through the boards
- This technology was famous in 1970s and 1980s, but it is still in use nowadays
- It can be plated (PTH) or non-plated (NPTH)



What is

Plated Through Holes (PTH) ?

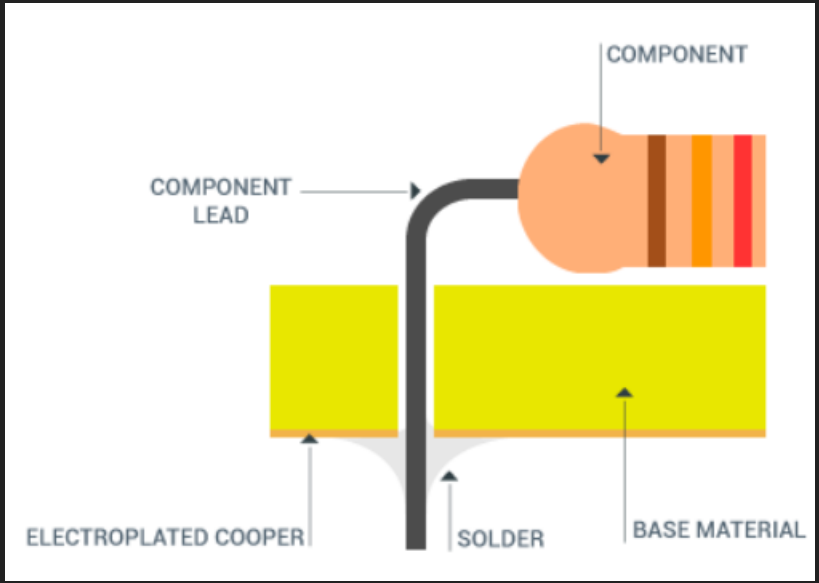
- during the manufacturing process, after drilling the boards a thin copper layer is plated onto the walls of the holes, providing them with electrical conductivity
- after the PCB Assembly is finished, the link between the component's leads and the copper tracks has a lower resistance and better mechanical stability



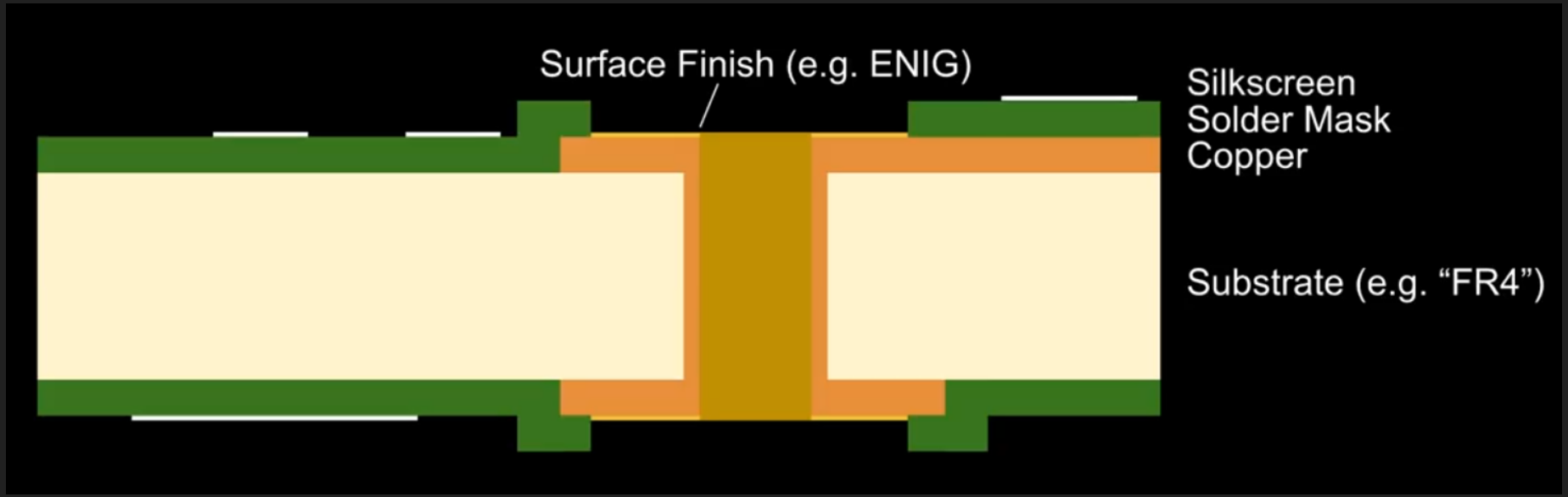
What is

Non-Plated Through Holes (NPTH) ?

- no copper plated onto the walls of the holes,
- so the barrel of the hole has not electrical properties
- popular when Printed Circuits only had copper tracks printed on one side,
- but their use decreased as the number of layers are increased
- The main advantages of NPTH are that the manufacturing process for these is simpler, and obviously quicker



Microscopic Crossection of PCB



Steps to design PCB from software to in hand

- ❑ Design Circuit
- ❑ Create Symbols
- ❑ Schematic Capture
- ❑ Create footprints
- ❑ Generate netlist
- ❑ Board Layout
- ❑ Production
- ❑ Generate BOM