

What is PCB? Printed Circuit Board

- PCB is an integral part of any Electrical and Electronics device.
- PCB is made up of Copper, Dielectric material, Solder mask and Silkscreen.

PAD: It is a conductive area on the board surface where electronic components are soldered.

- (1) Through-hole pads: These pads are designed for components that have leads or pins passing through the PCB.
- (2) Surface Mount Pads: SMD pads are designed for components that are mounted directly onto the surface of the board.

Solder mask: It is a process of covering the PCB to protect against corrosion, rust, prevent solder bridges from forming between closely spaced solder pads.

- Otherwise it may lead to short circuit.
- These masking should be in many colours but green colour is cost effective.

Silkscreen: It is used as a reference indicator for placing components on a PCB Board, marks, logos, symbols and so on.

- For soldering purpose it is very useful to identify the polarity of the component which is going to be soldered on.

Types of PCB:

- Single sided
- Double sided
- Multi-layer
- Rigid
- Flexible
- Rigid-Flex

Layer stack up:

Top Silkscreen

Top soldermask

L1

Pre-preg

L2

core

L3

Pre-preg

L4

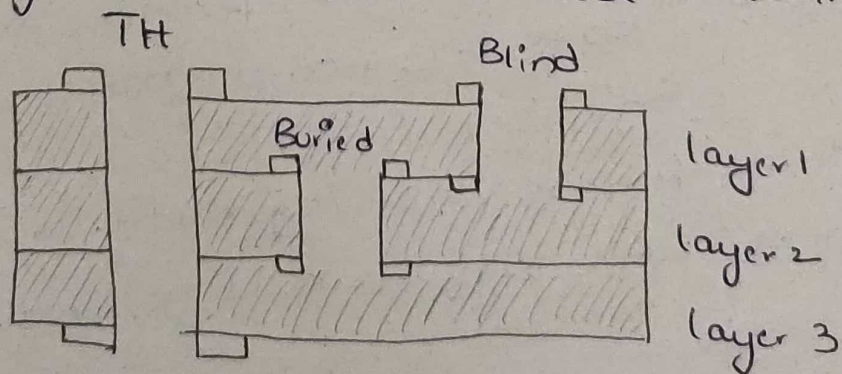
Bottom Soldermask

Bottom Silkscreen

Via: It allows a conductive connection between different layers.

Types of Via:

- Through-hole via: It extends through the entire thickness of the PCB, connecting the top and bottom layers.
- Blind via: It connects an outer layer of the PCB to one or more inner layers but does not go through the entire board.
- Buried via: A buried via connects two or more inner layers of the PCB without reaching the outer layers.

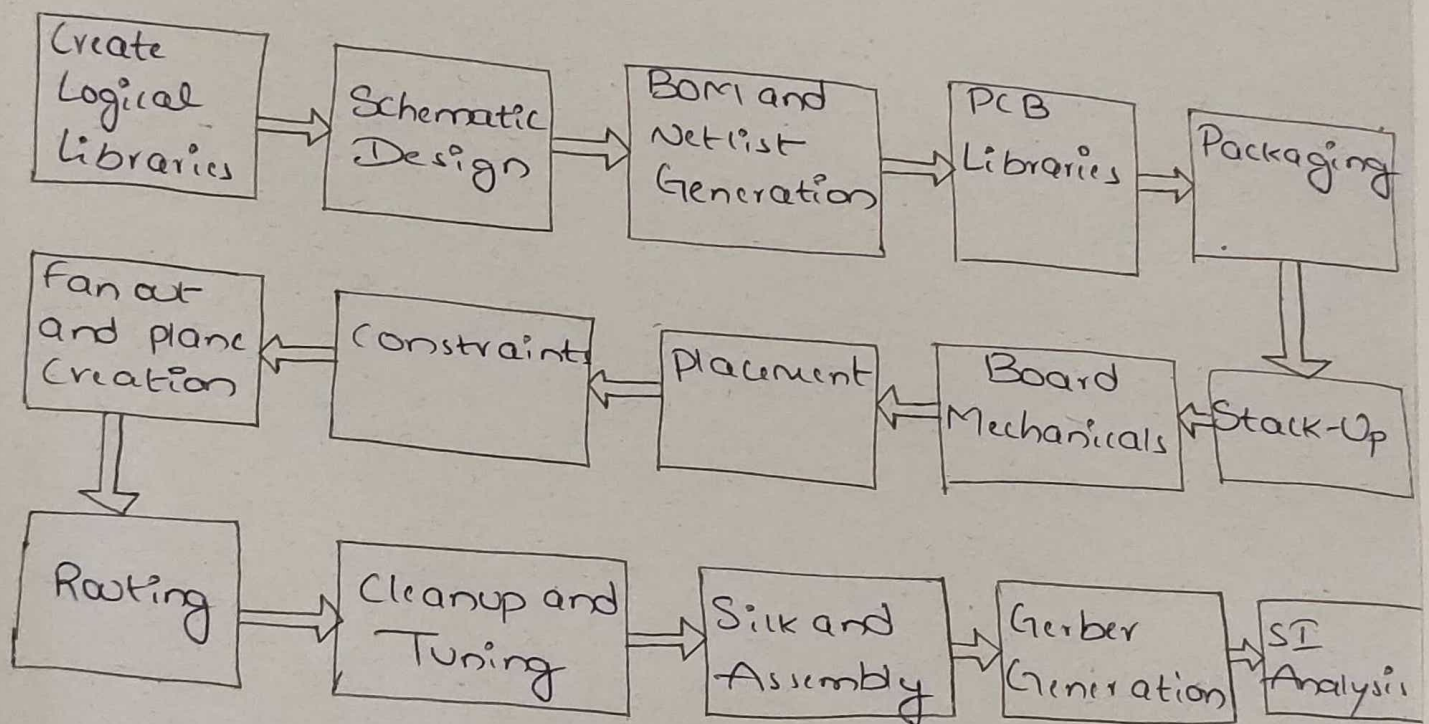




## PCB Performance classes.

- class 1 : General Electronic Products.  
Consumer electronics, Toys etc.,
- class 2 : Dedicated Service Products  
Professional Systems, Computers, Communication system
- class 3 : High Reliability Products  
Radars, Weapon systems, Satellite Communication equipment, Aerospace.

## PCB Design flow Basic:



DFA: "Design for Assembly" have set of principles and their goal is to create a PCB layout that is not only functional, electrically sound, efficient and cost-effective to assemble.

DFT: "Design for Testability" have set of Principles and their goal is to enhance the efficiency and accuracy of testing procedures, identifying and isolating faults or defects in PCB.