University of Glasgow | Department of Physics & Astronomy

UK Advanced Instrumentation Training

PCB Layout

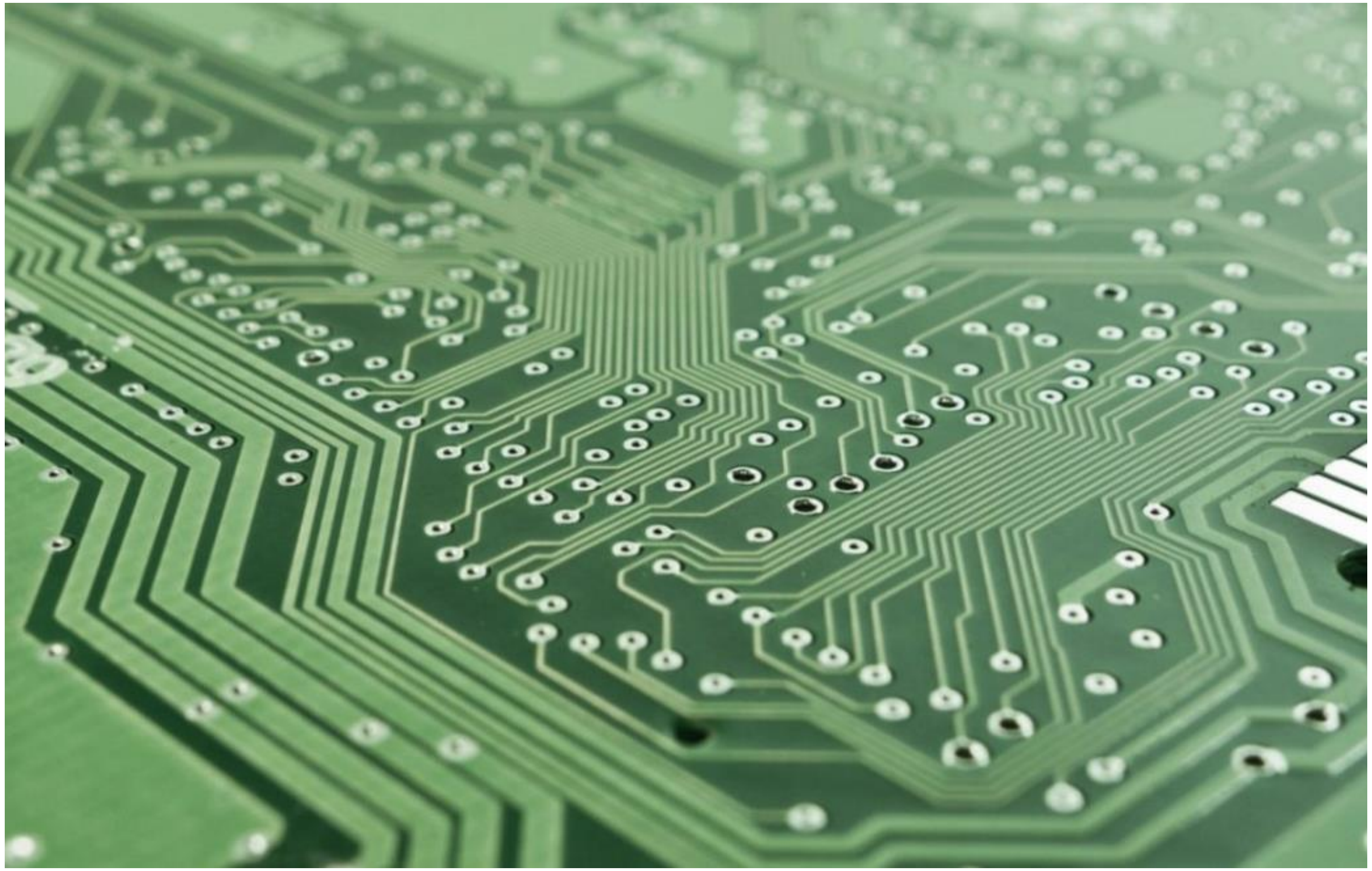
Sneha Naik
sneha.naik@glasgow.ac.uk

What is a Printed Circuit Board (PCB)

- A printed circuit board (PCB) is an electronic assembly that uses copper conductors to create electrical connections between components.

- It provides mechanical support for electronic components so that a device can be mounted in an enclosure.

- It can be rigid, flex, rigid-flex hybrid and the thickness can range from a few millimeter to a few microns based on technology.



PCB design process

1. Schematic design and creating a netlist
2. Setting the PCB design environment
3. PCB layout
4. Generating manufacturing data
5. Submitting files for fabrication
6. Quality control (QC)

Schematic design and creating a netlist

Defines the circuit with symbols and nets

Creating symbols and footprints

Associating symbols and footprints

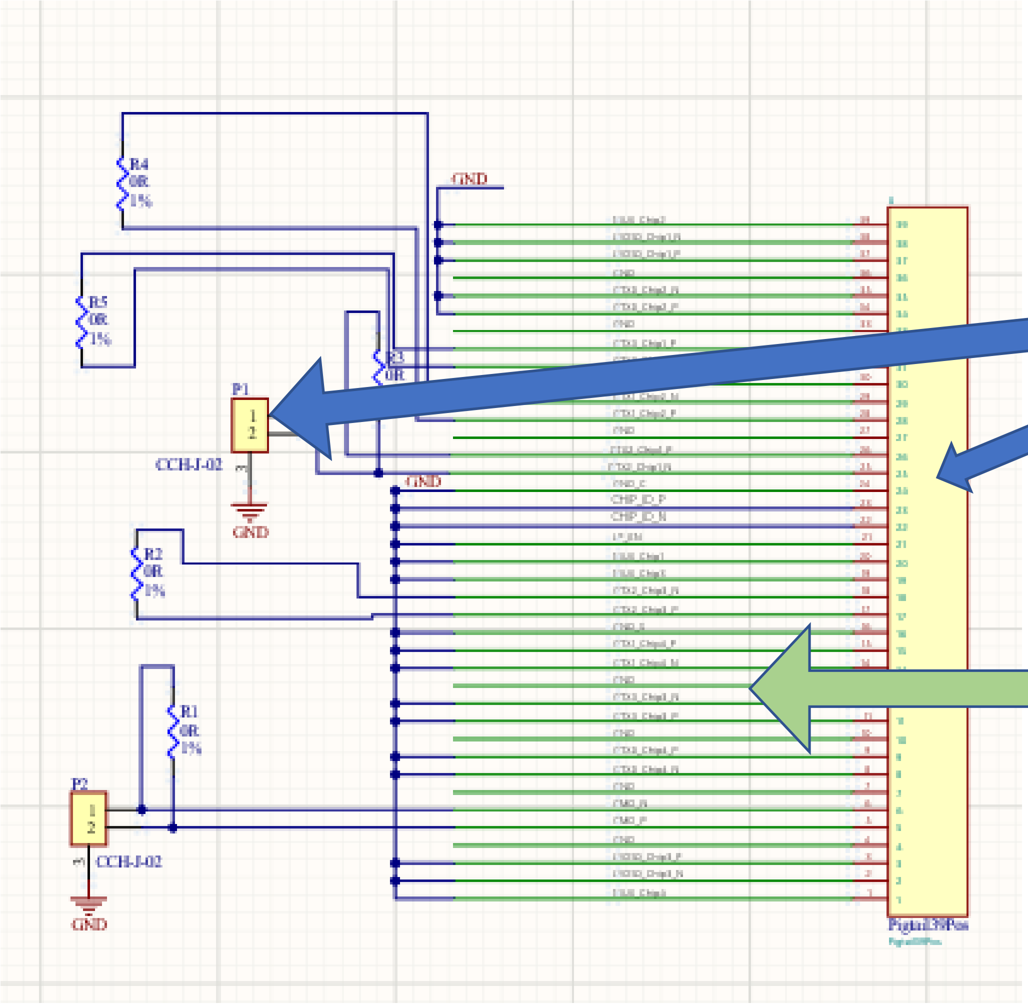
Creating rules for critical nets

Creating a netlist

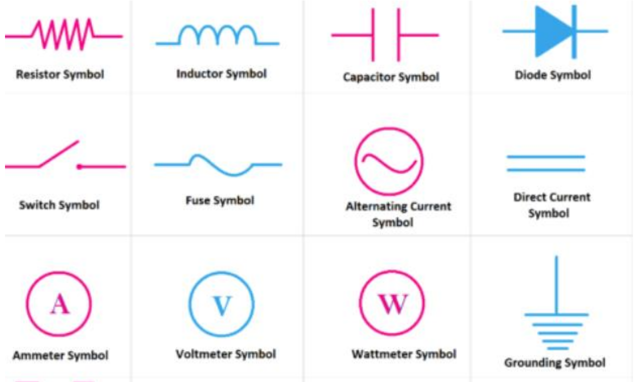
Netlist is a description of the connectivity of the circuit.

Describes each component and its connections

Schematic design



- Various tools are available like Altium, KiCad, Eagle, Mentor Graphics PADS, Cadence Allegro.
- Schematic comprises of –
  - Symbols



  - Nets (connections)

- Design Rules are rules added to the schematic like no duplicate reference designator, floating net labels etc.
- Design Validation highlight the errors /warnings based on the defined rules.

Setting the PCB environment

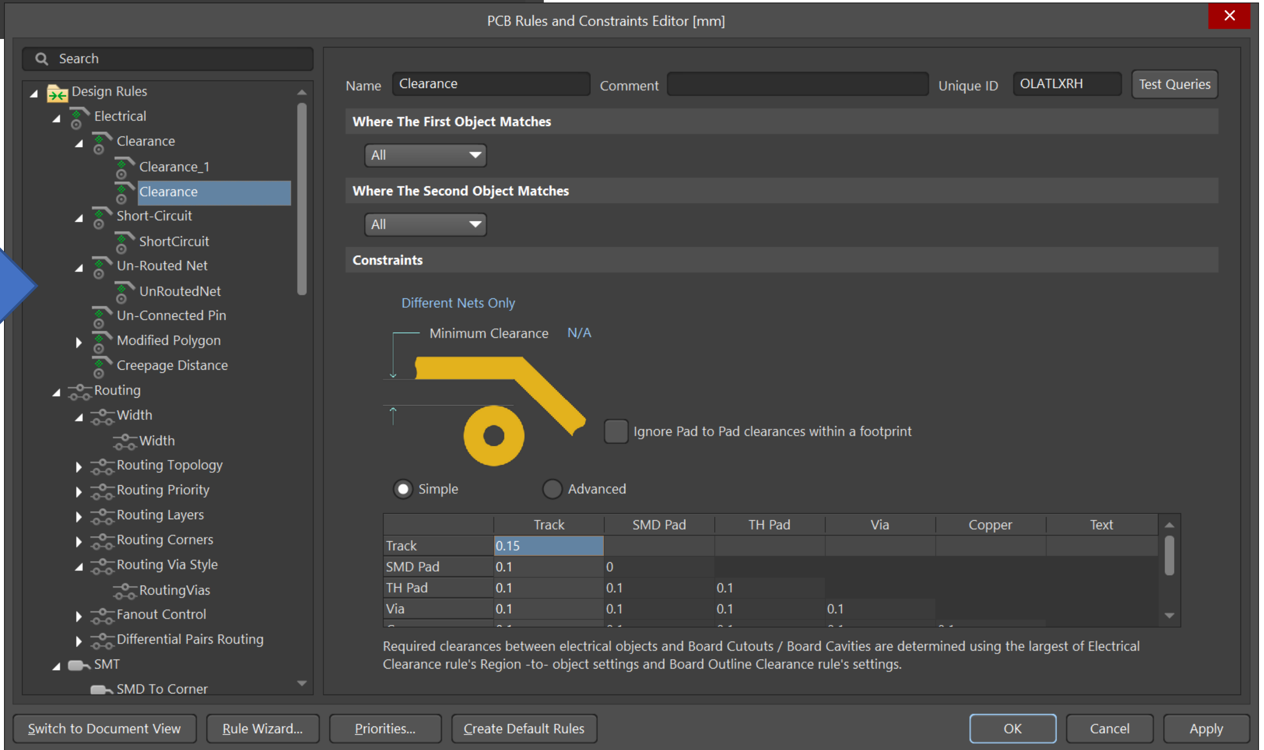
- Layer stackup and design rules
- PCB composition
- PCB configuration

Layer Stackup and design rules

#	Name	Material	Type	Thickness	Dk	Weight	Df
	F.SilkS		Overlay				
	F.Mask	SM-001	Solder Mask	0.03mm	4		0.03
	Top Surface Finish	Nickel, Gold	Surface Finish	0.005mm			
1	F.Cu	CF-003	Signal	0.02mm		1/2oz	
	Dielectric 1	Core-006	Core	0.075mm	3.2		0.002
2	In1.Cu		Signal	0.009mm		1/2oz	
	Dielectric2	PP-001	Prepreg	0.03875mm	3.2		0.002
3	B.Cu	CF-003	Signal	0.02mm		1/2oz	
	Bottom Solder	SM-001	Solder Mask	0.05mm	4		0.03

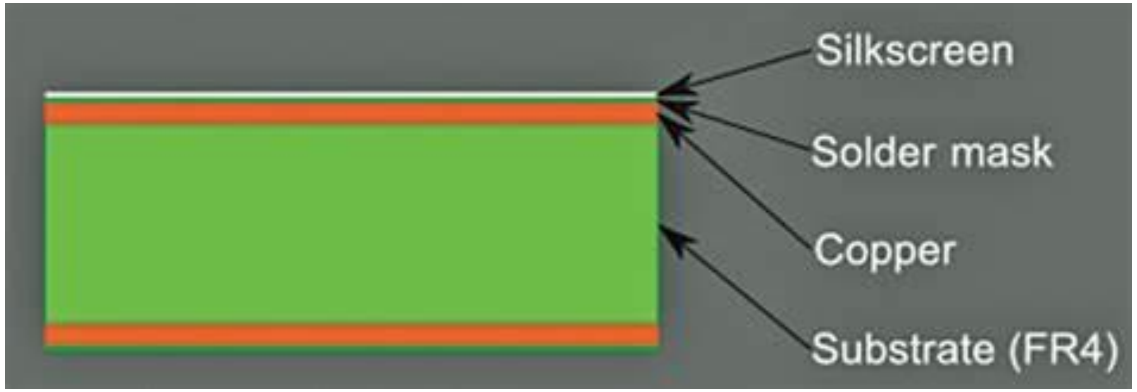
- **Define how many layers the PCB design would include.**
  - Single sided, double sided or multilayer
  - Rigid, flex, rigid-flex

- **Define design rules**
  - Clearance from traces to pads
  - Min/max hole size
  - Clearance of Cu from edge of the PCB
  - Routing layers, routing topology
  - Via geometry, etc.

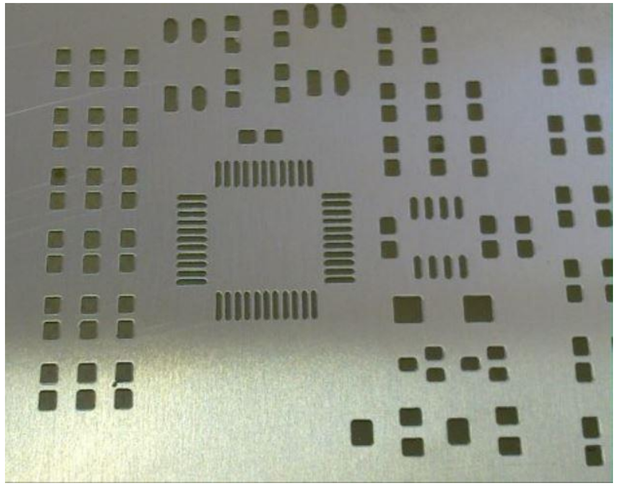


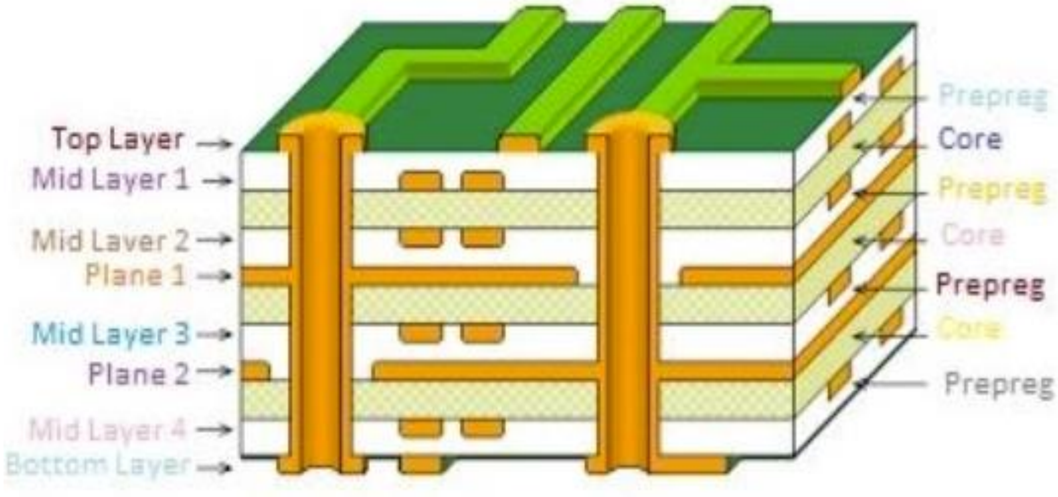
PCB composition

Different layers that make up a PCB



- **Cu foil**
- **Substrate (FR4, Polyamide)**
- **Solder mask**
  - Layer of polymer that protects the copper from oxidation and shorts during operation.
- **Silkscreen**
  - Ink trace used to identify the PCB components, marks, logos, symbols
- **Paste mask**
  - Data for creating stencil for assembly of components



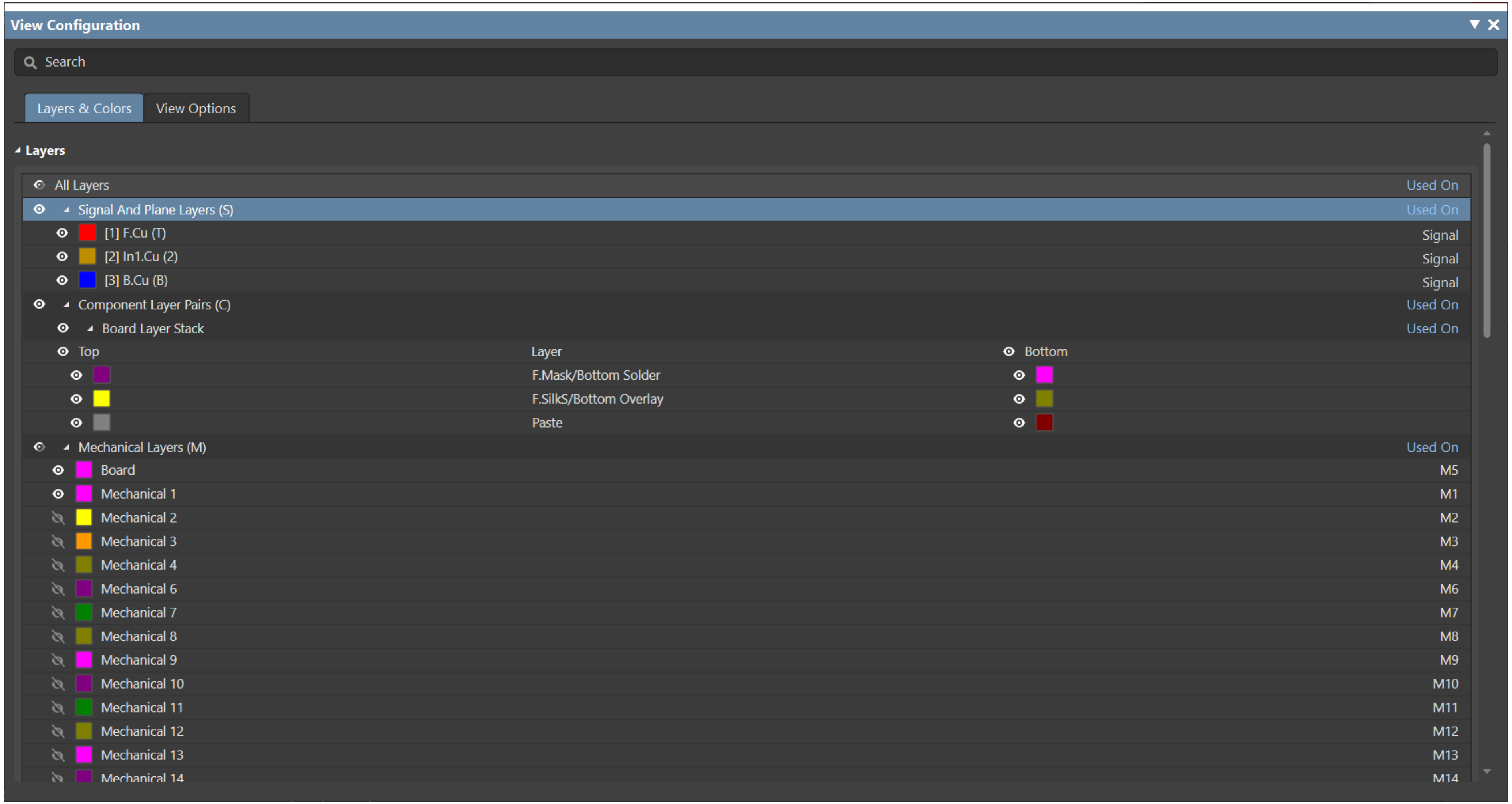


PCB configuration

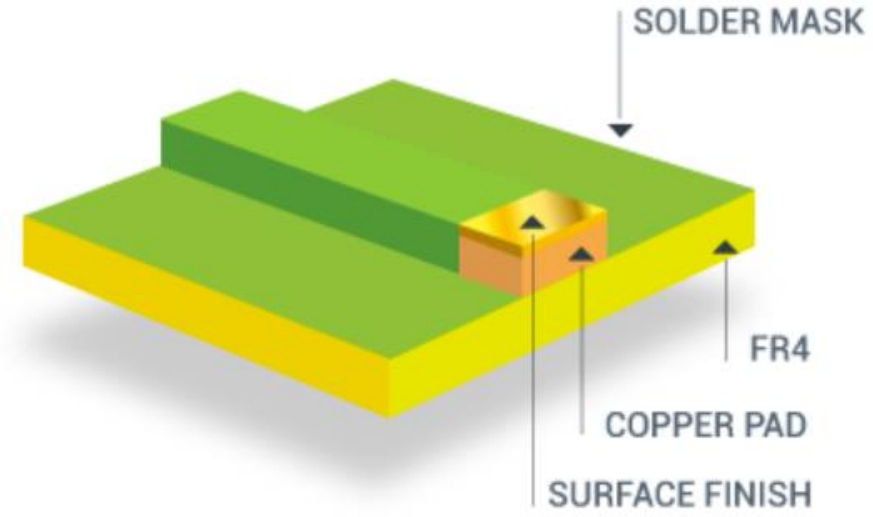
- Layer stack definition reflects in the configuration window.
- Here the colours for the individual layers can be changed

- The layers can be turned on and on as required.

- Additional mechanical layers are available that can be used to add board outlines, DXFs etc.



PCB surface finishes



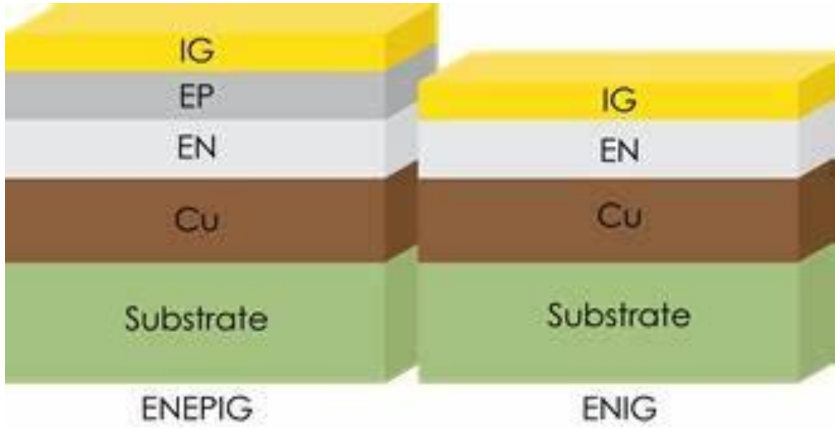
- The purpose of surface finish
  - Prevent the copper from oxidizing
  - Provide a solderable surface.

- **Types of surface finish**
  - Hot Air Solder Leveling (HASL)

  - Organic Solderability Preservative (OSP)

  - Electroless Nickel Immersion Gold (ENIG)
    - This finish provides a thin, gold, solderable layer that protects the copper traces with a nickel barrier between it and the copper. ENIG is a good lead-free option that results in a durable, long-lasting finish.
    - Suitable for Al wedge wirebonding

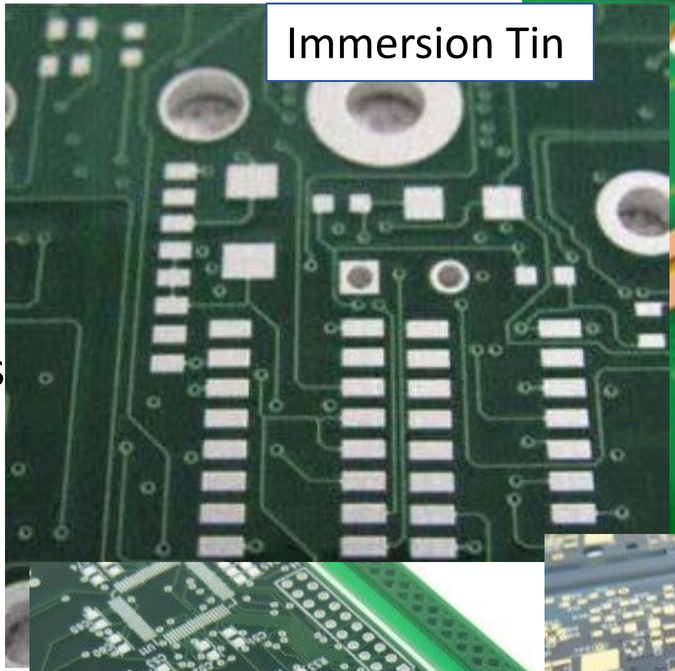
  - Electroless Nickel Immersion Palladium Immersion Gold (ENIPEG)
    - Ideal for gold wirebonding
    - Addresses the Black Pad (the corrosion of underlying nickel) issue.



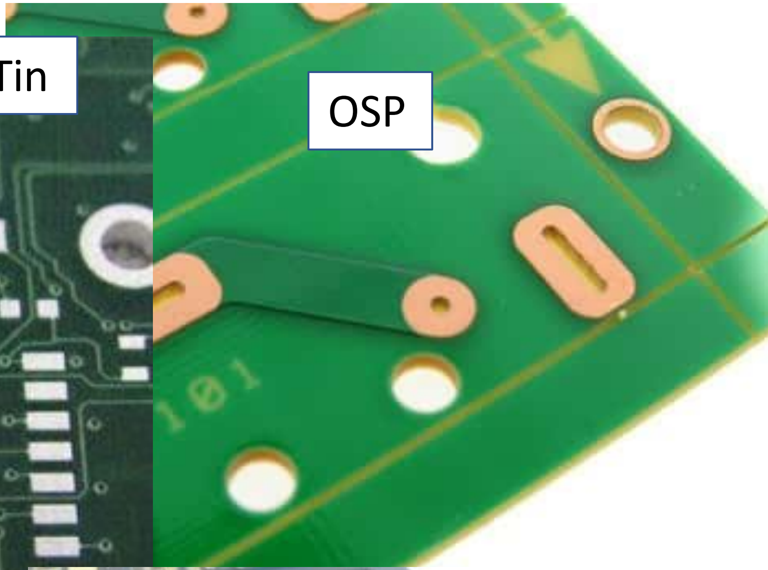
PCB surface finishes

Key Considerations When Choosing Your PCB Finish

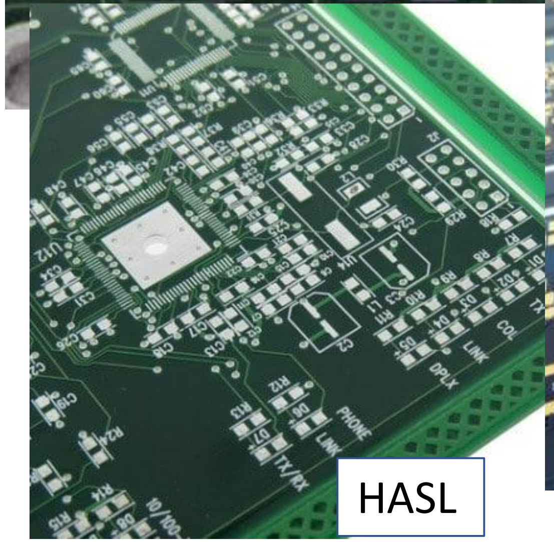
- Price
- Availability
- Shelf life
- Reliability
- Assembly process
- Compliances like Restriction of Hazardous Substances (RoHS)



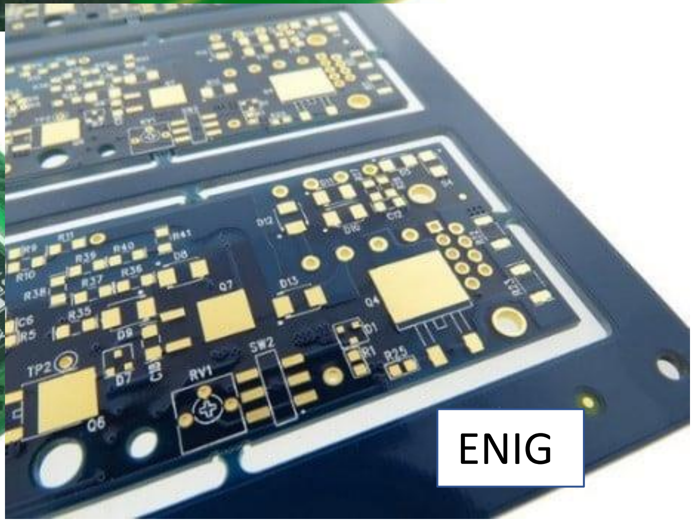
Immersion Tin



OSP



HASL



ENIG

Examples:

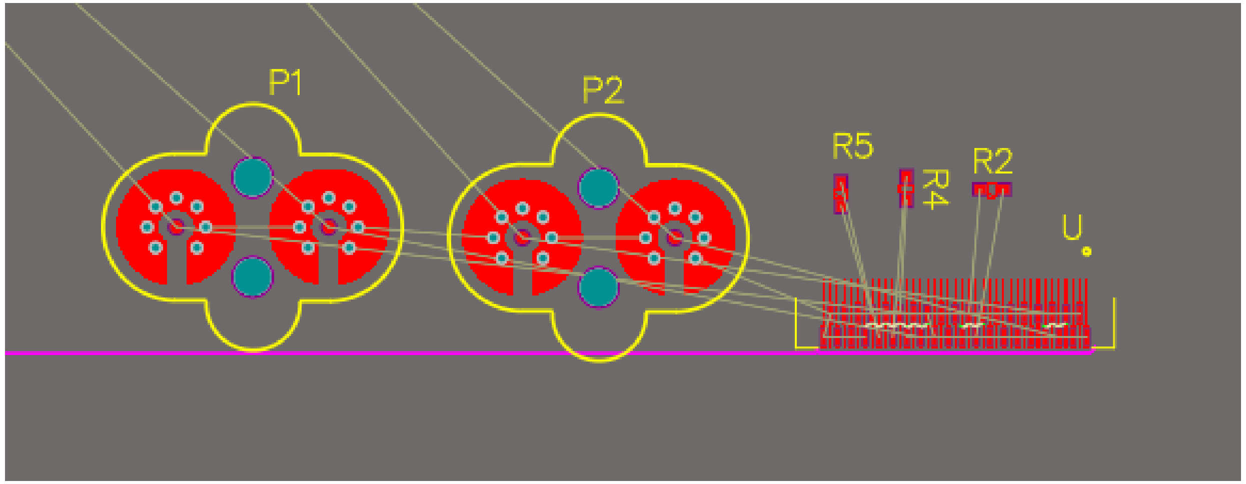
- If you do not need to be RoHS, Sn/Pb HASL may be your best option. It is low in cost and widely available.

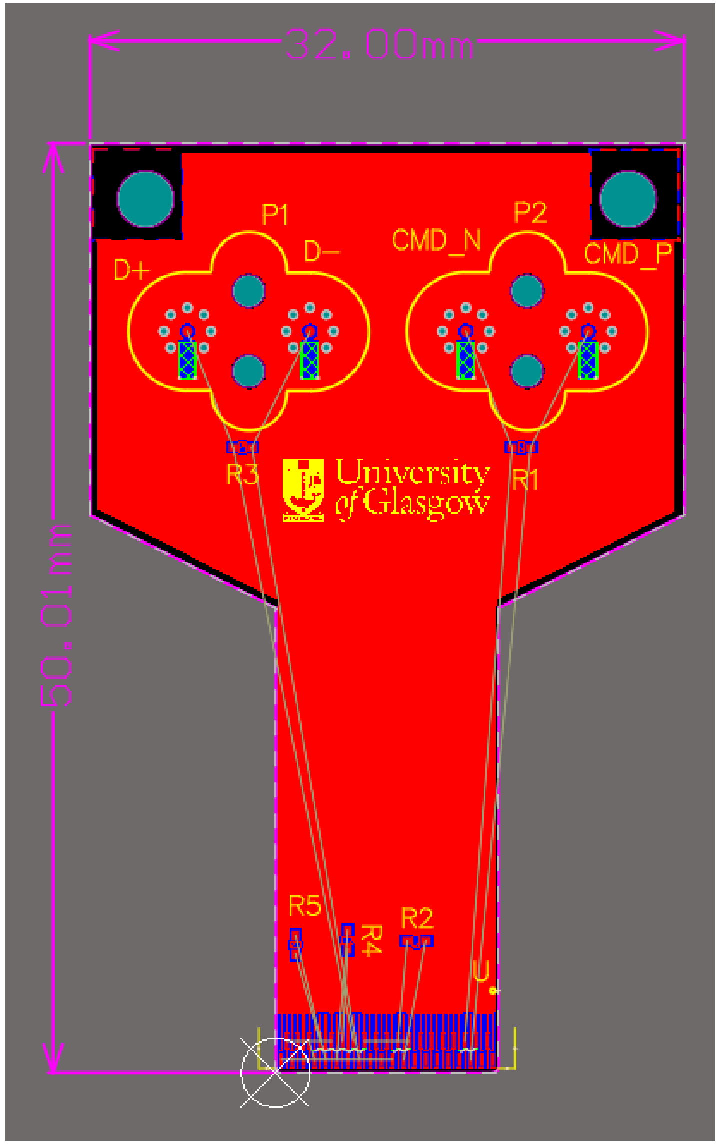
- If your boards need to be RoHS and have fine pitch components including BGA’s we recommend ENIG or immersion silver.

PCB layout

- Placement of components
- Define routing strategy, layout guideline, signal integrity etc.
- Routing and design rule check
- Mechanical considerations

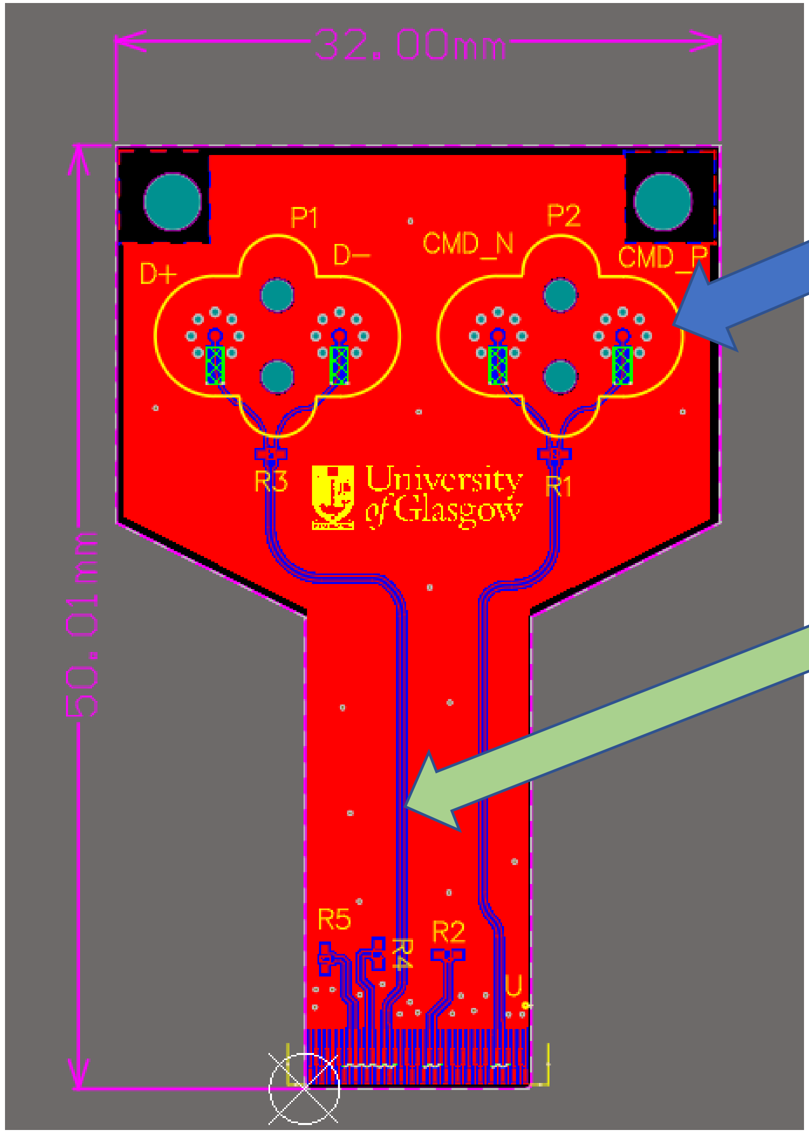
Placement of components





- Import from schematic to PCB file to bring the associated footprints to the symbols in schematic along with the connections.
- There are libraries for symbols (schematic library) and for footprints (PCB library)
- Mechanical requirements need should be considered to define the shape and size of the PCB. This includes mechanical hole positions, sizes as well as cutouts.
- The components can then be placed on the PCB considering routing techniques.

Routing and design rule check (DRC)

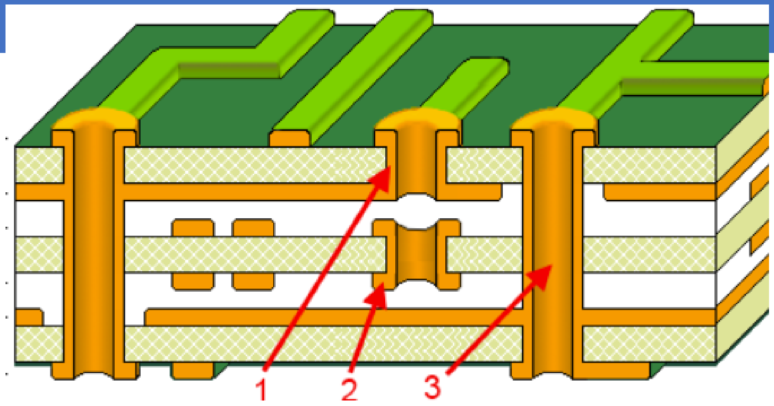


- In the PCB layout –
  - The Symbols translate to footprints
    - The footprint is drawn to specifications as specified in the datasheet of the component.

  - Nets translate to traces
  - Rules and constraints
    - These are added to the design rules as per requirement of the design.
  - Design rule check
    - This will highlight errors based on the set design rules.

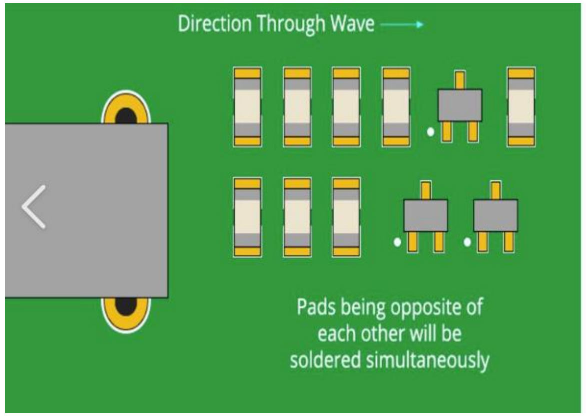
PCB Layout considerations

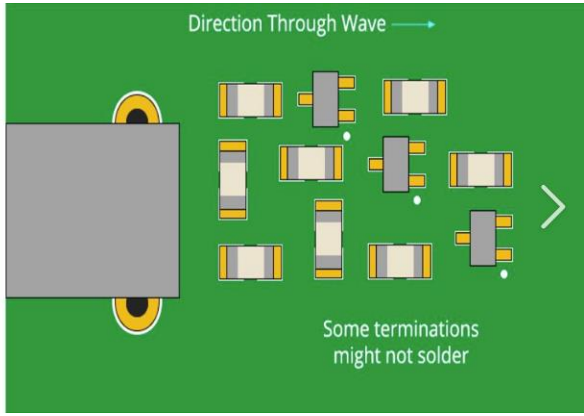
- **Board Constraints**
  - Estimate the size and shape of the board
  - Work around the mechanical requirements to fit the circuit.
  - Reflects on number of layers required
- **Manufacturing process/technology**
  - Surface mounted/through hole blind and buried vias (high density interconnect (HDI))
- **Placement and routing priorities**
  - Placement sequence: Interconnects ->Power circuit ->High speed/critical circuits ->non-critical parts.
  - Orientation of components to ease assembly
    - Place passives in one direction to the extent possible.
  - Single sided/Double sided/multilayer PCB



1- Blind via; 2- Buried via; 3 –Through via

Via - consists of two pads in corresponding positions on different layers of the board, that are electrically connected by a hole through the board

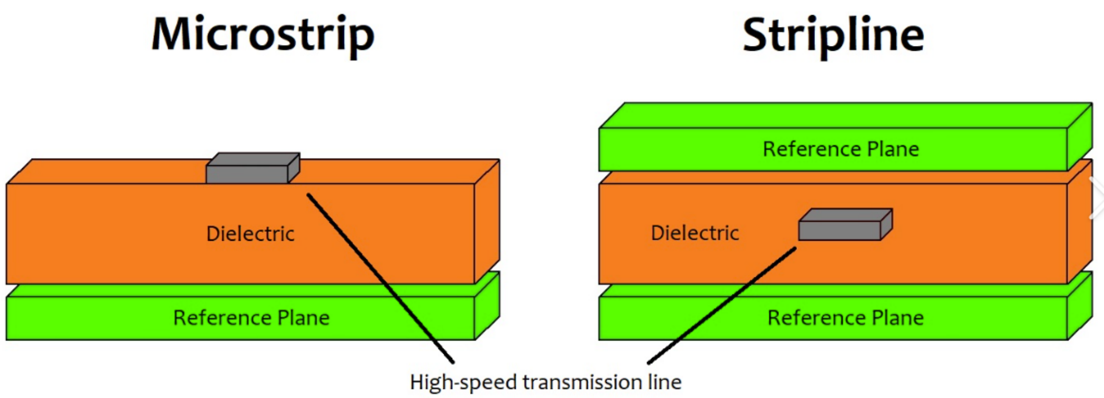


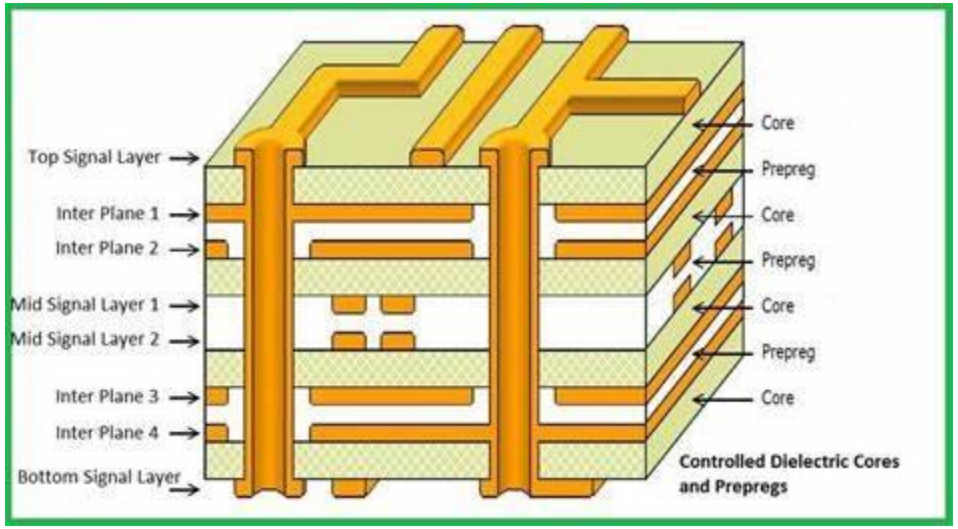


PCB Layout considerations

- **Routing signals on PCB**
  - Signals can be routed as Microstrip or Stripline
    - Microstrip or Stripline are transmission line structures on PCB
    - Signals in Microstrip move faster but are more prone to noise
    - Signals in Stripline have a more influence of the dielectric surrounding it but are more shielded.
  - Defining layer stackup is important to ensure the signal routing technique is defined
    - Signal and plane layers based on density of the components and routing.
  - Critical signals should be routed with a continuous ground reference to ensure controlled impedance of the trace.

- **Power and ground planes**
  - Keep the power and ground planes internally in the PCB stackup.
  - Ideally planes should be centered and symmetrical to prevent bowing and twisting of the PCB
  - Keep the analog and digital grounds, power grounds separate.

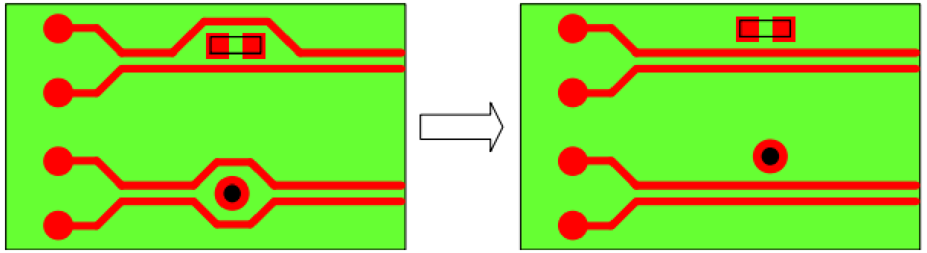




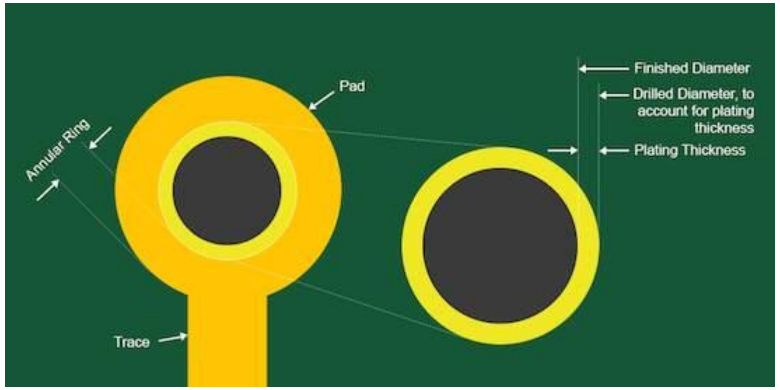
PCB Layout considerations

- **Trace widths/trace spacings/ Via geometry**
  - Ensure that the specifications are such that the PCBs can be readily manufactured in industry.
  - Design to meet the impedance requirement for high-speed signals (transmission lines at high speed).
  - Keep critical signals well spaced from fast switching signals.
  - Check via geometry meets thermal requirement

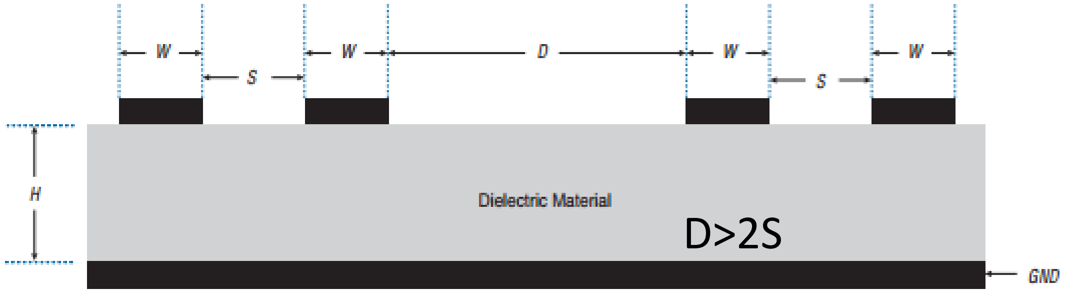
- **Signal Integrity**
  - Avoid large loops of signal and ground-return lines that carry high frequencies.
  - Ensure continuous ground reference for critical signals
  - Try using differential signaling scheme that is less prone to crosstalk
  - Eliminate antennas, which can radiate electromagnetic energy
  - Reduce trace stubs, reduce vias, terminate traces with termination resistors.
  - Choose material suitable for high-speed signal transmission



Routing technique



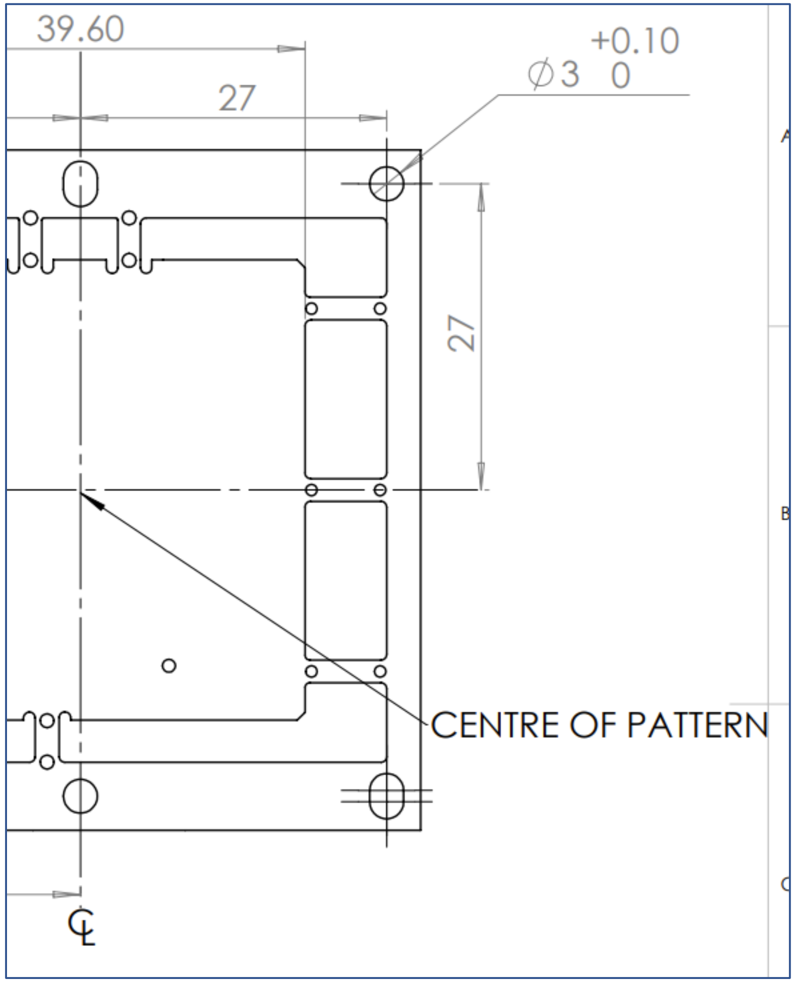
Via definition



Clearance rules to avoid crosstalk

Mechanical Considerations

- Every PCB layout would have some mechanical requirements on allowed dimensions, thickness, allowed components heights, envelop etc.
- It is a critical step to liaise with mechanical team to understand these requirement and add these as constraints to the PCB layout.
- DXF/DWG files are a good way to share the design details between mechanical and electronics design.
- After the layout is complete, a final DXF is shared with the mechanical team to ensure the design is meeting the specifications.



Manufacturing data

Gerber files (artwork) and drill files

Documentation for fabrication and assembly (mounting components)

Creating specific documented highlighting the requirements for fabrication/assembly

Submitting files for fabrication

Data checked by vendor

Solve engineering queries from vendor on the Gerber files.

Close issues to start fabrication

Quality control (QC)

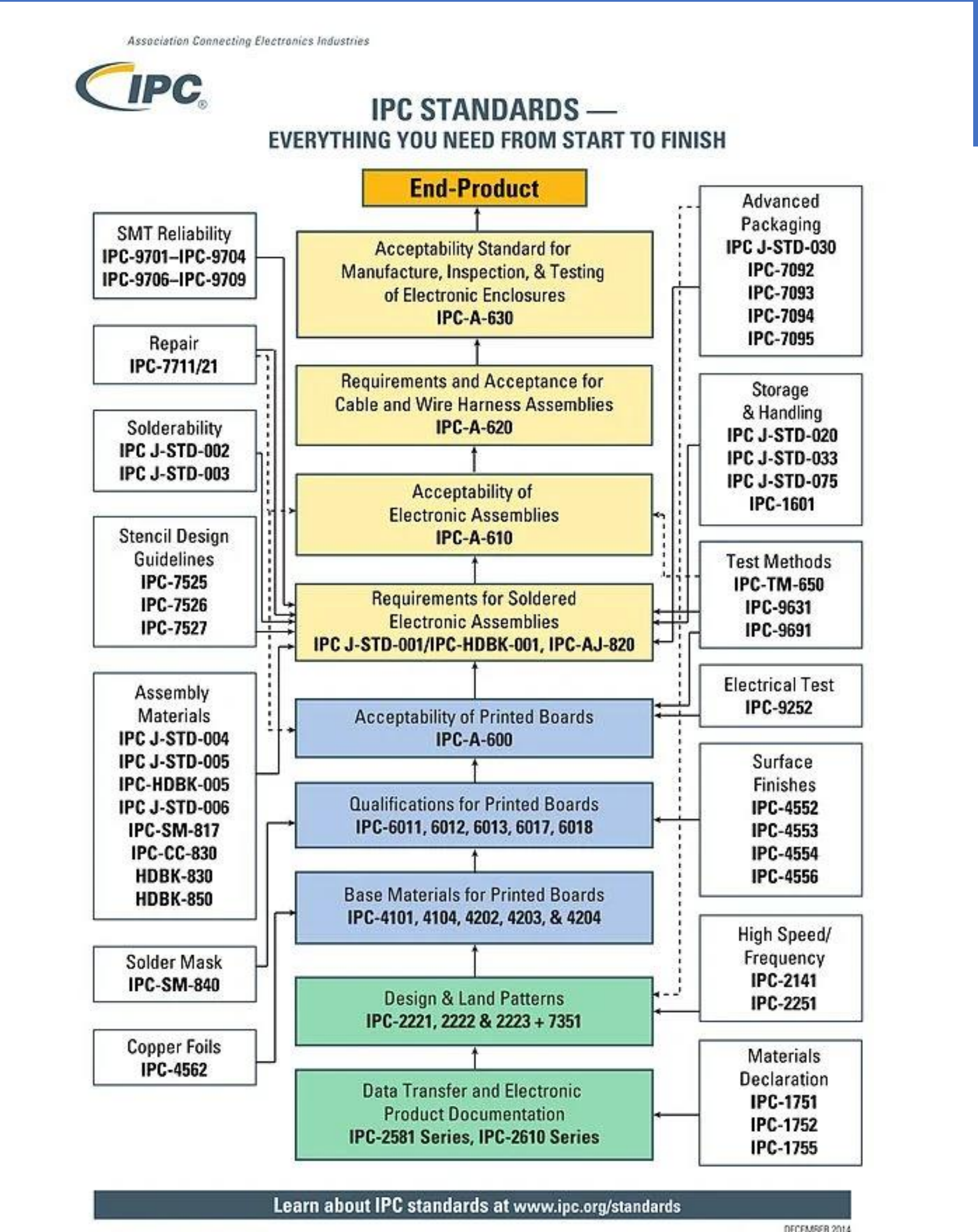
Check to ensure PCBs are fabricated as per specifications

IPC standard are used for PCB manufacturing and assembly

In-house QC tests like visual inspection, layer thickness measurements, etc. based on the application.

IPC Standards

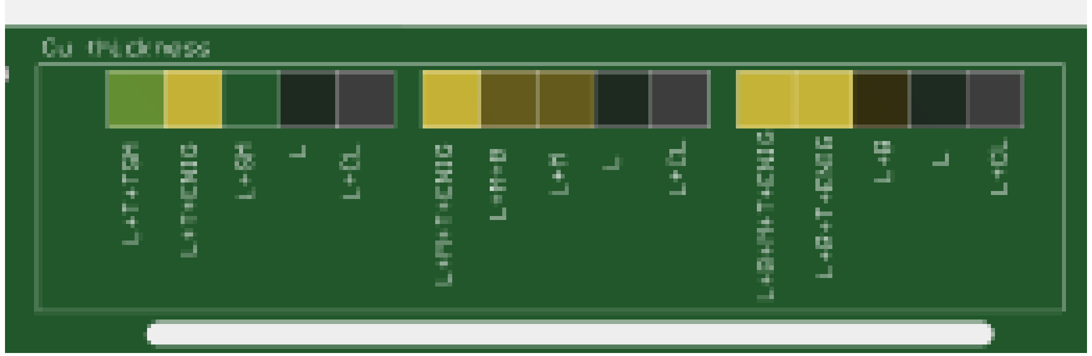
- IPC formerly called the Institute for Printed Circuits, presently called the Association Connecting Electronics Industries standards.
- It is a trade association whose aim is to standardize the assembly and production requirements of electronic equipment and assemblies.
- There is an extensive set of IPC documents that ensure PCB are produced the right way.
- Every step for fabrication and assembly is associated with an IPC standard to carry out the process.
- Similar IPC tests are linked to the assembly process of mounting PCBs with components.
- IPC also defines various testing procedures for testing PCB as a part of QC process .



In-house QC tests

The QC tests will vary based on application and specifications. Some example tests are-

- **Optical inspection**
  - Check PCBs for contaminations and defects
- **Metrology**
  - Measure size of the PCB, holes etc.
- **Layer stackup check for individual layers**



- **Signal transmission tests**

  (TDR measurements for impedance)

- **Electrical tests to test functionality**

Thank you for listening
Any questions?