

PCB Designing with Rai

PCB DESIGN

How to Become a PCB Designer

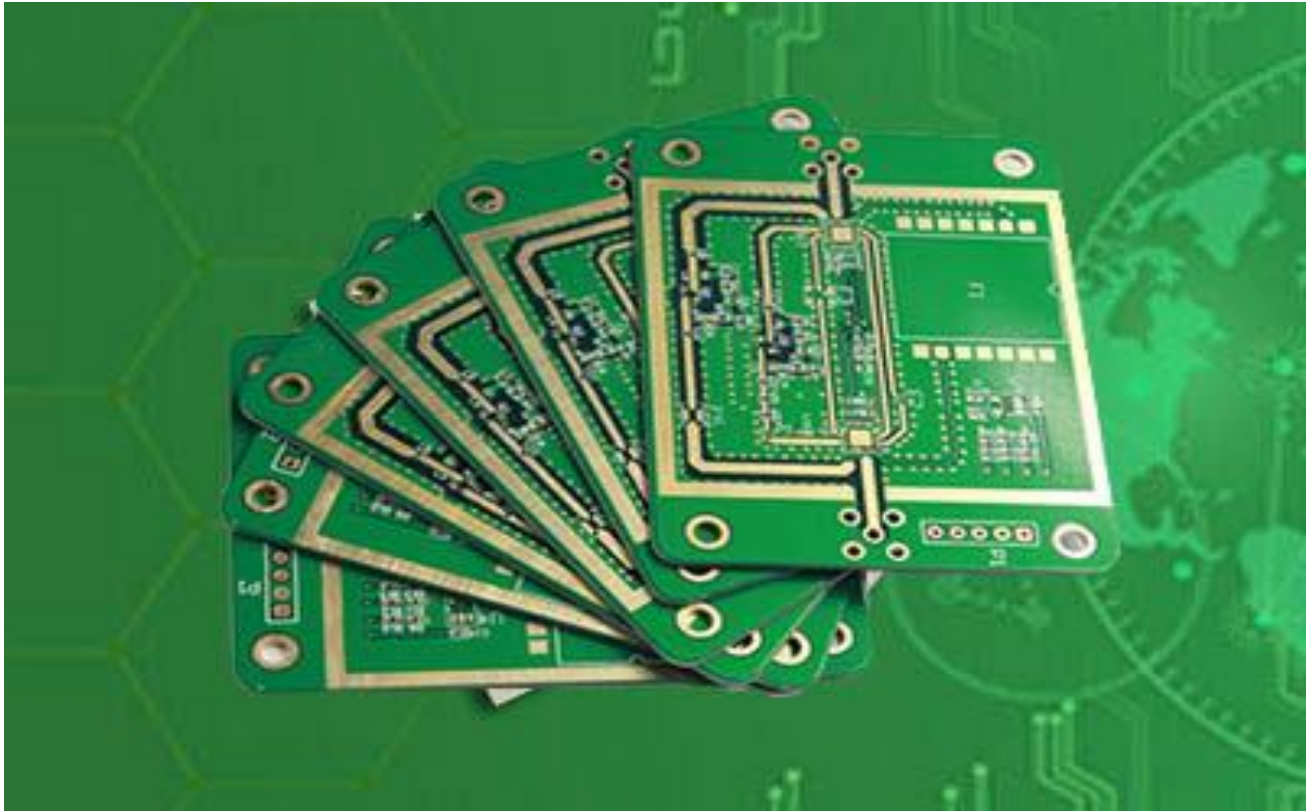


BEGINNER



What is PCB designing

PCB designing, or printed circuit board designing, is the process of creating the layout and design for a printed circuit board (PCB). A PCB is a flat board made of a non-conductive substrate material (usually fiberglass or composite material) with conductive pathways etched or printed on it. These conductive pathways, also known as traces or tracks, provide the electrical connections between various electronic components mounted on the board.

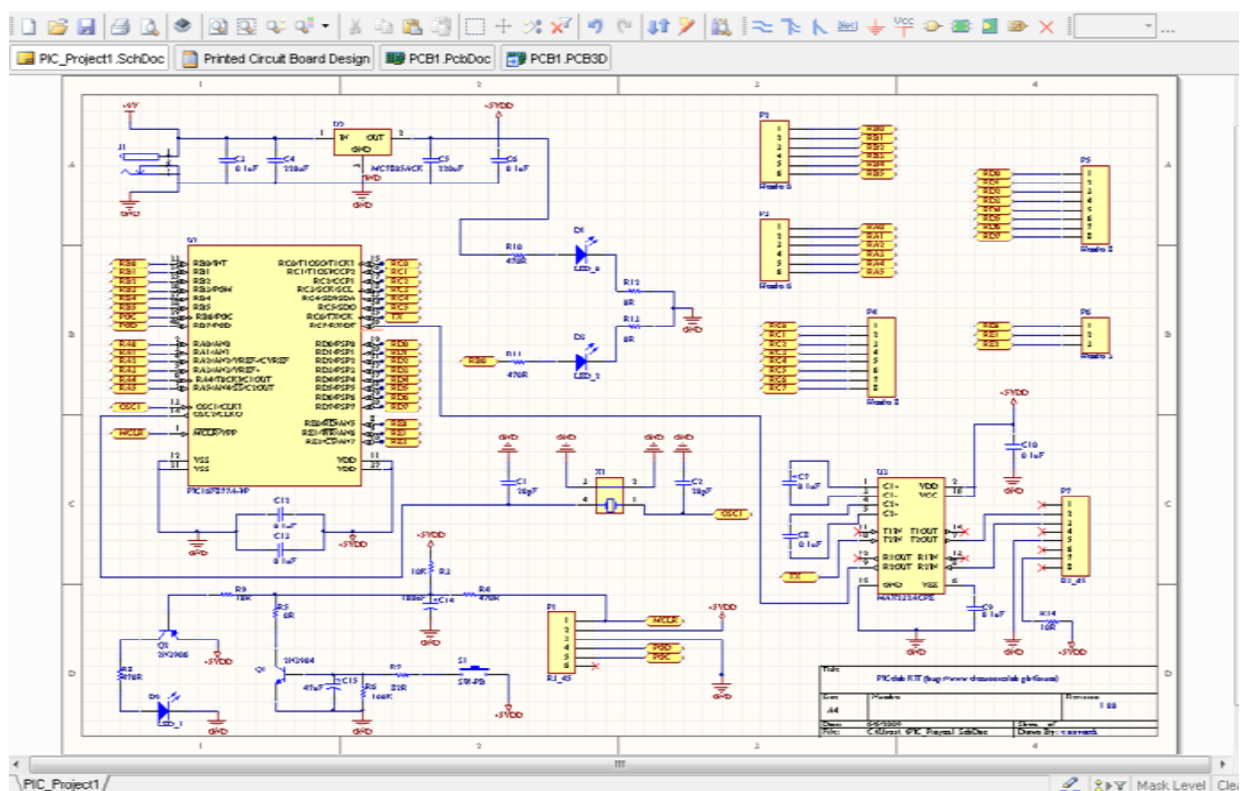


Without PCB



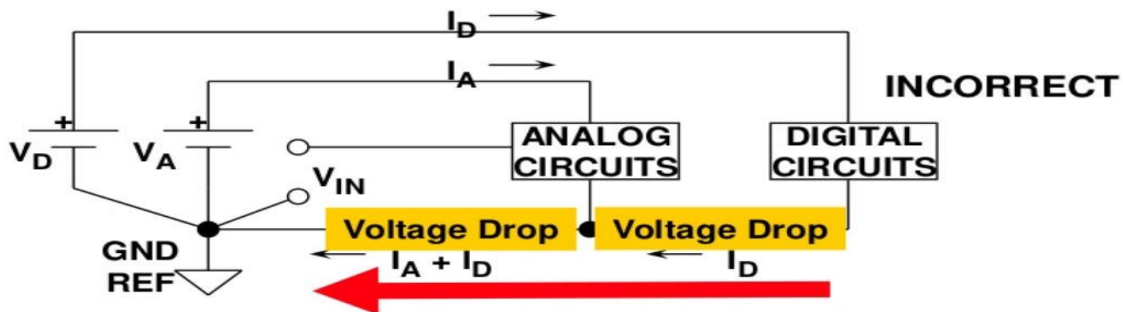
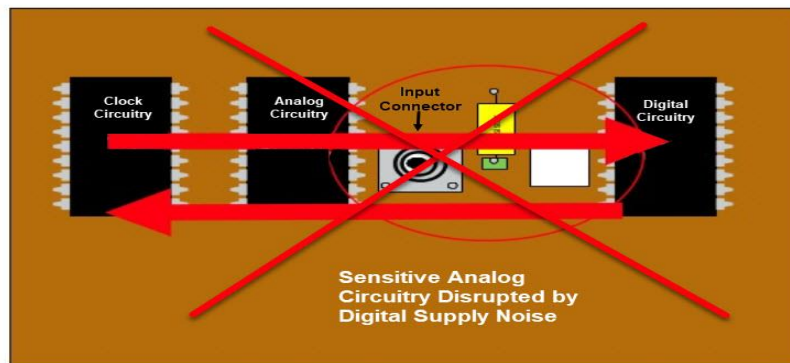
The PCB design process involves several key steps

- **Schematic Design:** This is the initial phase where the circuit diagram is created using schematic capture software. The schematic represents the electronic components and their interconnections.



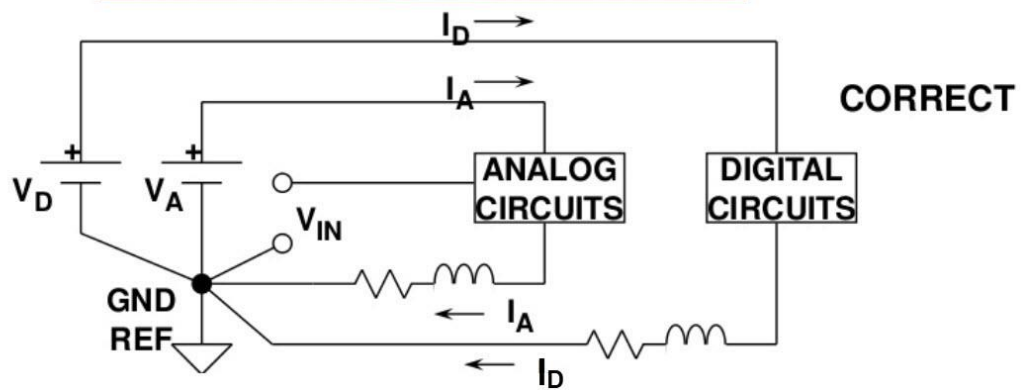
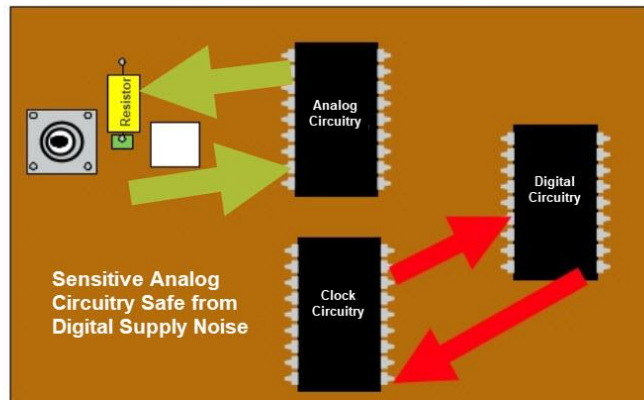
- **Component Placement:** Once the schematic is complete, the next step is to place the electronic components on the PCB layout. The goal is to arrange the components in an optimal way to minimize signal interference and ensure efficient use of space.

Wrong way of components placement

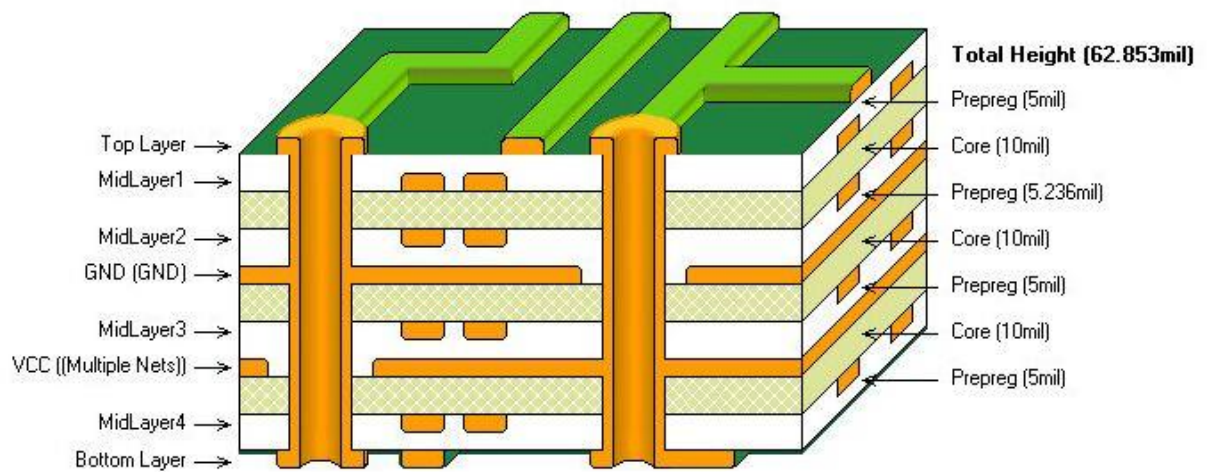


Right way of components placement

Right Way



- **Layers Stack up:** PCBs can have multiple layers, and the designer must decide how to stack these layers. This is crucial for accommodating complex circuits and optimizing the performance of the board.



6 Layer PCB Stackup

#	Name	Material	Type	Thickness	Weight	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask	0.4mil		3.5	
1	SIG TOP		Signal	1.4mil	1oz		
	Dielectric 1	PP-006	Prepreg	8mil		4.1	0.02
2	GND1	CF-004	Signal	1.378mil	1oz		
	Dielectric 2	PP-006	Core	15mil		4.8	0.02
3	PWR	CF-004	Signal	1.378mil	1oz		
	Dielectric 3	FR-4	Prepreg	8mil		4.1	0.02
4	SIG	CF-004	Signal	1.378mil	1oz		
	Dielectric 4	PP-006	Core	15mil		4.8	0.02
5	GND2	CF-004	Signal	1.378mil	1oz		
	Dielectric 5	PP-006	Prepreg	8mil		4.1	0.02
6	SIG BOT		Signal	1.4mil	1oz		
	Bottom Solder	Solder Resist	Solder Mask	0.4mil		3.5	
	Bottom Overlay		Overlay				

Controlled
impedance/analog

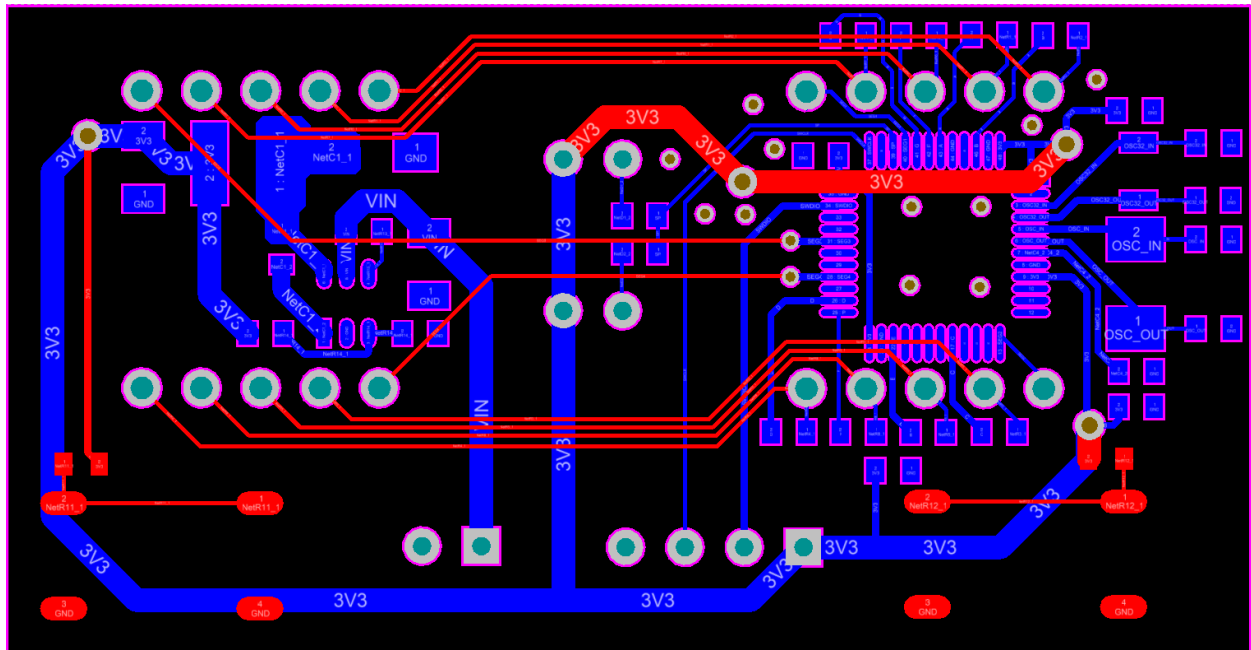
Plane pair for higher
power and decoupling

Other digital or control
signals

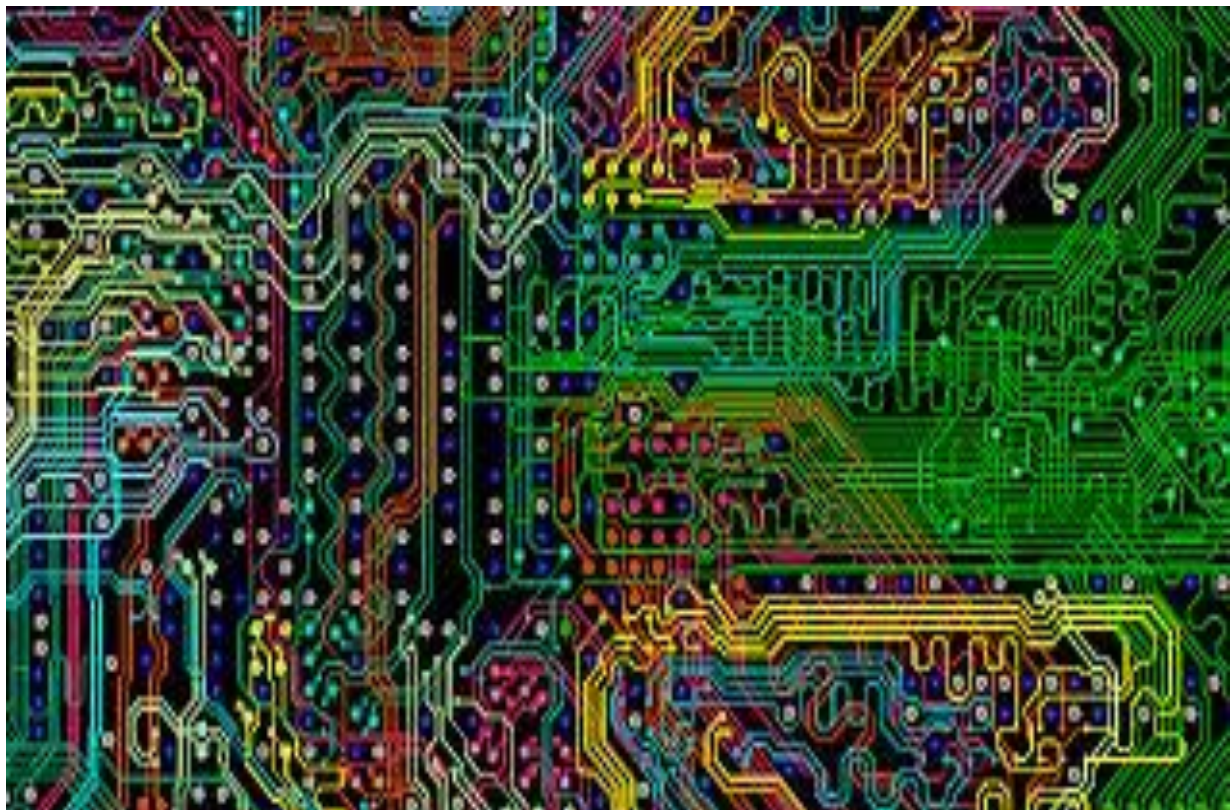
Controlled
impedance/analog

- **Routing:** After placing the components, the designer needs to route the electrical connections between them by creating traces on the PCB. The routing must consider factors like signal integrity, impedance matching, and avoiding interference.

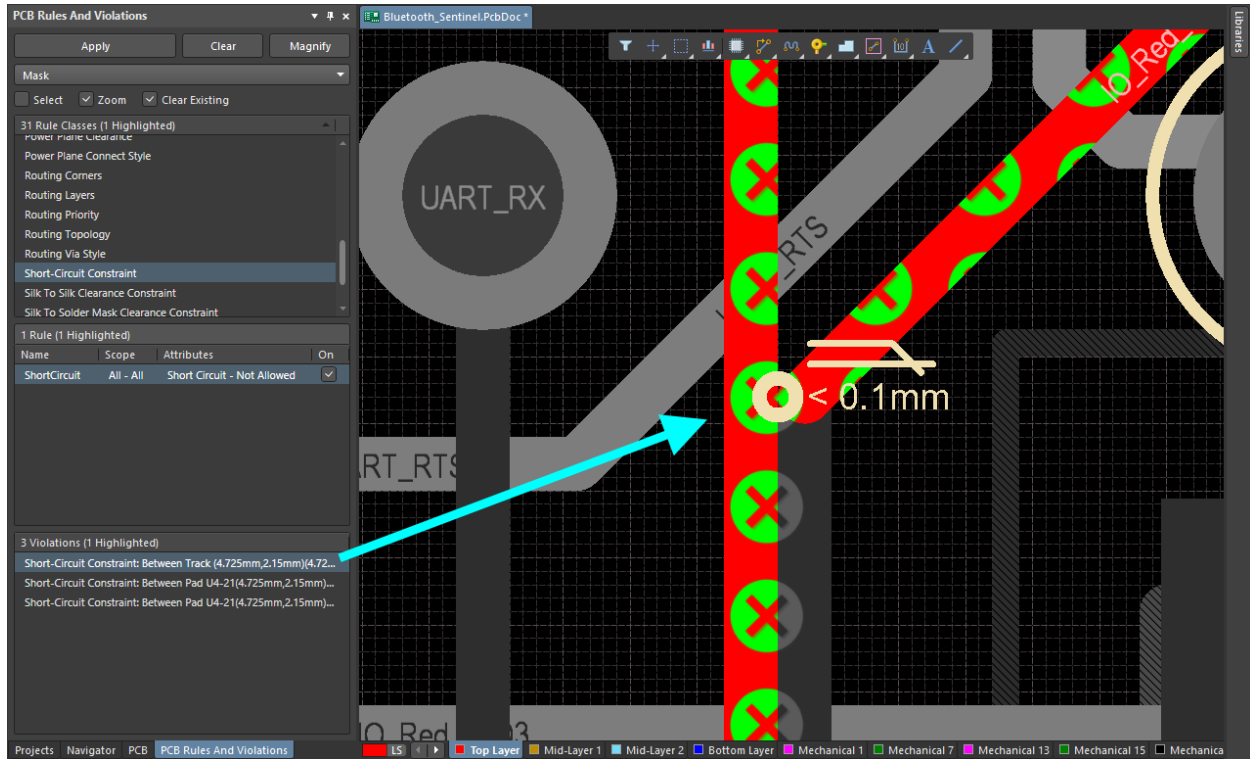
Simple Routing



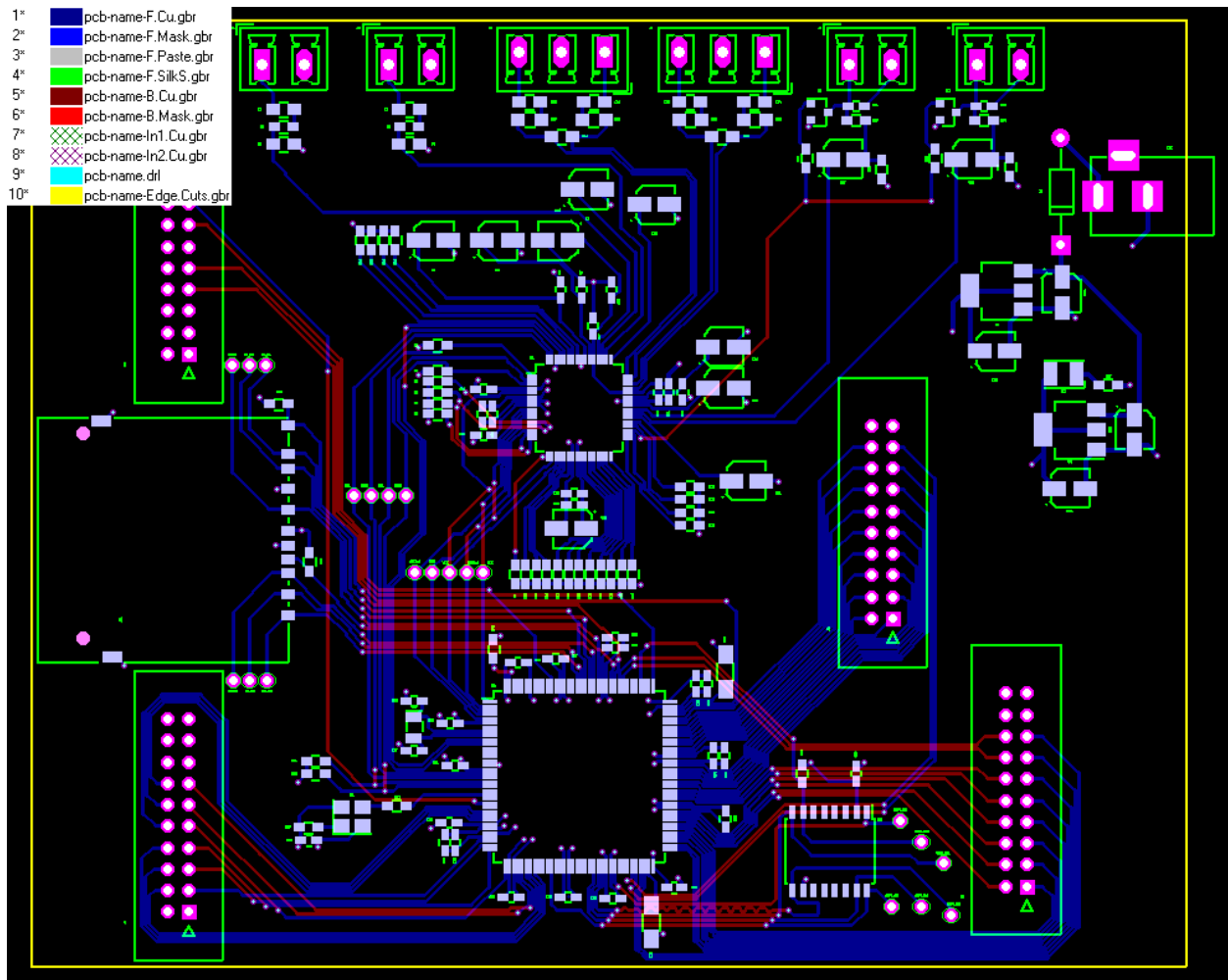
Complex Routing



- **Design Rule Check (DRC):** The design is then checked against a set of rules to ensure that it complies with manufacturing constraints and standards. This includes checking for issues like short circuits, clearance violations, and proper trace widths.



- **Gerber File Generation:** Once the design is finalized, Gerber files are generated. Gerber files contain the information necessary for manufacturing the PCB, including layer details, component placements, and trace information.



- **Prototype and Testing:** After generating the Gerber files, a prototype PCB is manufactured. This prototype is then tested to ensure that it functions as intended and meets the design specifications.
- **Final Production:** If the prototype is successful, the final PCB can be mass-produced using the same design.



Software for PCB designing

- Altium Designer
- Circuit maker
- KiCad
- Eagle (Autodesk Eagle)
- OrCAD
- PADS (PowerPCB)
- KiCad
- DipTrace
- PCB123 (Sunstone Circuits)
- Proteus
- EasyEDA

How to achieve proper grounding

The term ground refers to earth and usually we use third wire to connect to the ground for safety purpose

- Ground on PCB is often considered a region of zero volts potential with zero resistance and impedance etc. but this is not true in reality this is near to zero but not zero
- This is more close to zero for DC only
- Light and Electrical are same type of energy but in light has million times high frequency
- Energy travel in space not in traces
- The energy in a circuit travel in the plastic and fiberglass material of PCB
- the copper elements act as a wave guide
- Energy always follows low impedance path and when we have traces it become a path to guide a signal of low impedance
- Current and voltage travel in the traces but energy travel in dielectric

Why Low frequency current spread out across a board area of the plane?

At low frequencies, currents tend to spread out across a board area or a plane due to the behavior of electromagnetic fields and the skin effect. The phenomena that contribute to the spreading of low-frequency currents include:

1. Capacitive Coupling:

- At low frequencies, the electric field lines generated by a current-carrying trace or conductor extend into the surrounding space. This results in capacitive coupling between adjacent conductors and the ground plane. The capacitance between traces and the ground plane allows electric charges to redistribute, causing current to spread over a larger surface area.

Achieving proper grounding in PCB (Printed Circuit Board) design is crucial for the overall performance and reliability of electronic circuits. Here are some general guidelines to help achieve proper grounding:

1. **Single Point Ground:** Design your PCB with a single point ground, where all ground connections meet at a single location. This helps to minimize ground loops and reduces the risk of noise and interference.
2. **Separate Analog and Digital Grounds:** If your circuit has both analog and digital components, consider separating their ground planes. Connect them at a single point,

but keep the ground planes physically separated to prevent digital noise from affecting analog signals.

3. **Ground Plane:** Use a solid ground plane on one or both sides of the PCB. This provides a low-impedance path for return currents, minimizes loop area, and helps in reducing electromagnetic interference (EMI).
4. **Partitioning:** Divide your PCB into functional blocks or sections and allocate specific ground areas for each section. This helps to contain noise within specific regions and prevents it from spreading to other parts of the circuit.
5. **Star Topology:** Use a star topology for grounding sensitive components. Connect all critical ground points directly to a central ground point, forming a star-like pattern. This minimizes the length of ground traces and reduces the risk of voltage differences across the circuit.
6. **Ground Trace Width and Via Placement:** Ensure that ground traces and vias have sufficient width and are placed strategically to minimize impedance. Low-impedance paths help to carry return currents efficiently.
7. **Ground Isolation:** Isolate sensitive analog components from noisy digital components by using separate ground planes or islands. This prevents high-frequency noise generated by digital components from coupling into sensitive analog circuits.
8. **Decoupling Capacitors:** Place decoupling capacitors close to the power pins of integrated circuits to provide a low-impedance path for high-frequency currents. This helps to suppress noise and maintain a stable ground reference.
9. **Avoid Ground Loops:** Ground loops can introduce unwanted noise into the circuit. Be mindful of the physical layout to minimize loop areas and ensure that ground paths follow the most direct route.
10. **Signal Return Paths:** Ensure that the return paths for signals closely follow their respective trace paths. This reduces the loop area and minimizes the inductance of the return current path.

How do we control electromagnetic interference (EMI) and signal integrity (SI)

Main ideas

- Contain fields
- Choose ICs that can do the jobs with slowest rise/fall time possible
- Keep high energy/frequency traces as short as possible
- Improve power delivery (minimize inductance, decoupling)

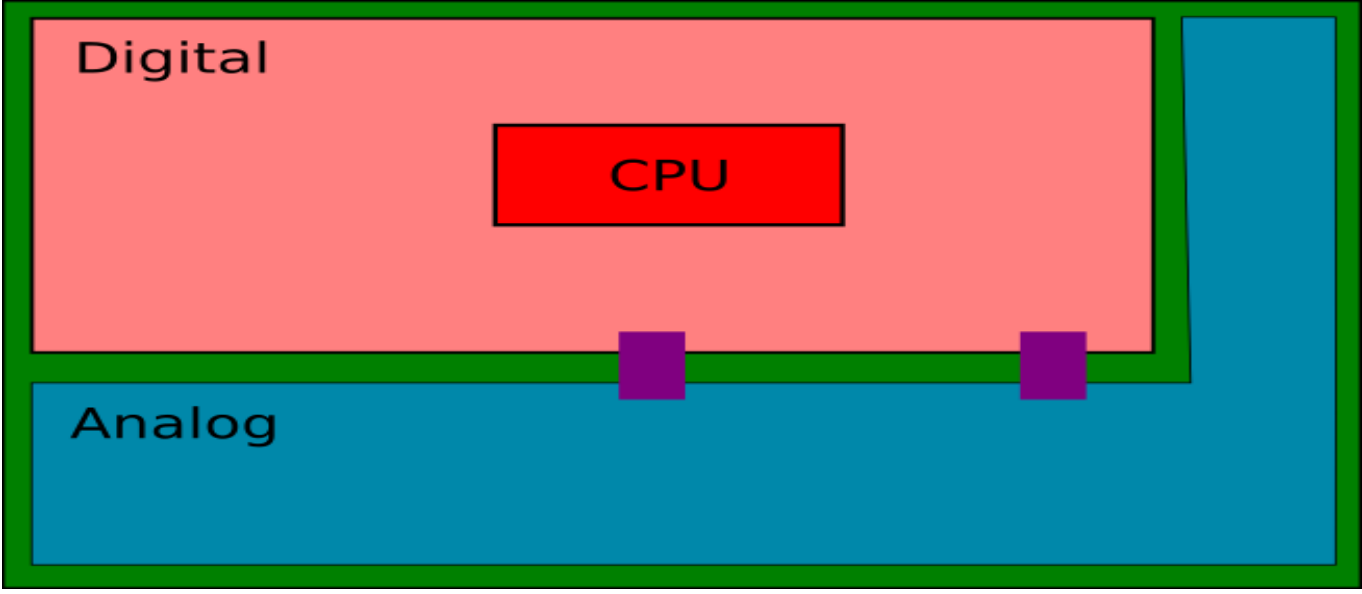
Stack up

- Use thin dielectrics
- Improve coupling, reduce inductance, increases interplanar capacitance etc.
- At least one ground layer adjacent to every signal and power plane
- Contains field, reduces inductance, good return, improved power distribution
- Stripline for high speed signals
- Avoid placing signal layers next to each other
- We will not keep two signal layers from under each other because of cross talk we always keep ground, power plane between two signal planes.
- Vias comes in pair
- Keep vias close to minimize the inductance
- Power, digital, RF and analogs in their own respective section
- Connector placement is an incredibly important part of EMI
- Keep all section away from each other especially analogue far from digital and power

Vias in pair



Rooms



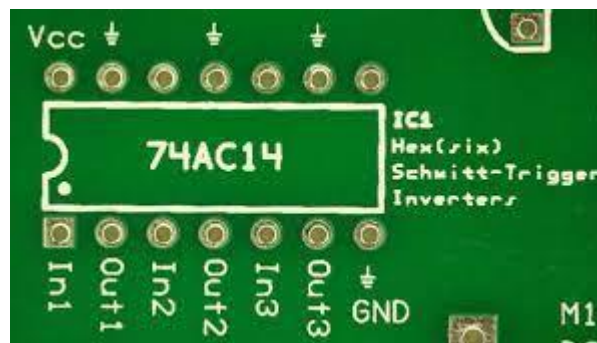
PCB layer stack up

The layer stack up of a PCB (Printed Circuit Board) refers to the arrangement and configuration of different layers that make up the board. A PCB typically consists of multiple layers of copper and insulating material, and the layer stack up defines the order and properties of these layers. The layer stack up is a critical aspect of PCB design, influencing factors such as signal integrity, power distribution, and overall performance. Here's a breakdown of key components in a typical layer stack up:

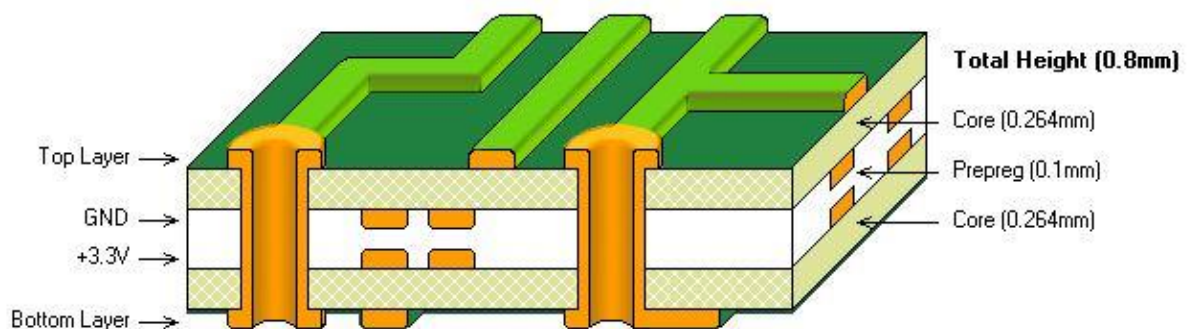
1. **Top Layer (Signal Layer):** This is the topmost layer of the PCB and is often used for routing signal traces. Components are placed on this layer, and it is where most of the signal paths are implemented.
2. **Prepreg:** Prepreg, short for pre-impregnated, is a layer of insulating material (usually fiberglass-reinforced epoxy) that is partially cured. It provides mechanical strength and separation between copper layers. The prepreg layer is sandwiched between copper layers.
3. **Core Layer:** The core layer is made of a solid sheet of insulating material, typically FR-4 (Flame Retardant 4), a type of fiberglass-reinforced epoxy. It provides the main structural support for the PCB and contributes to its overall thickness.
4. **Inner Copper Layers:** Inside the PCB, there can be multiple layers of copper, separated by prepreg layers and core layers. These inner copper layers are used for routing additional signals and power planes. The number of inner layers depends on the complexity of the design.
5. **Ground Planes and Power Planes:** Ground and power planes are dedicated copper layers used for providing a low-impedance return path for signals and distributing power throughout the board. Ground planes are often placed adjacent to signal layers to reduce electromagnetic interference (EMI) and maintain signal integrity.
6. **Inner Core Layers:** These are additional core layers that provide more structural support and insulation. The number of inner core layers depends on the overall thickness and rigidity required for the PCB.
7. **Bottom Layer (Signal Layer):** Similar to the top layer, the bottom layer is used for routing signal traces. It is the bottommost layer of the PCB.
8. **Solder Mask:** Solder mask is a protective layer applied over the top and bottom layers. It covers the areas where components are not soldered and prevents solder bridges during the assembly process. Solder mask also provides insulation and protects the copper traces from environmental factors.



9. **Silkscreen:** Silkscreen is a layer containing markings, symbols, and labels that help identify components, reference designators, and other important information on the PCB.



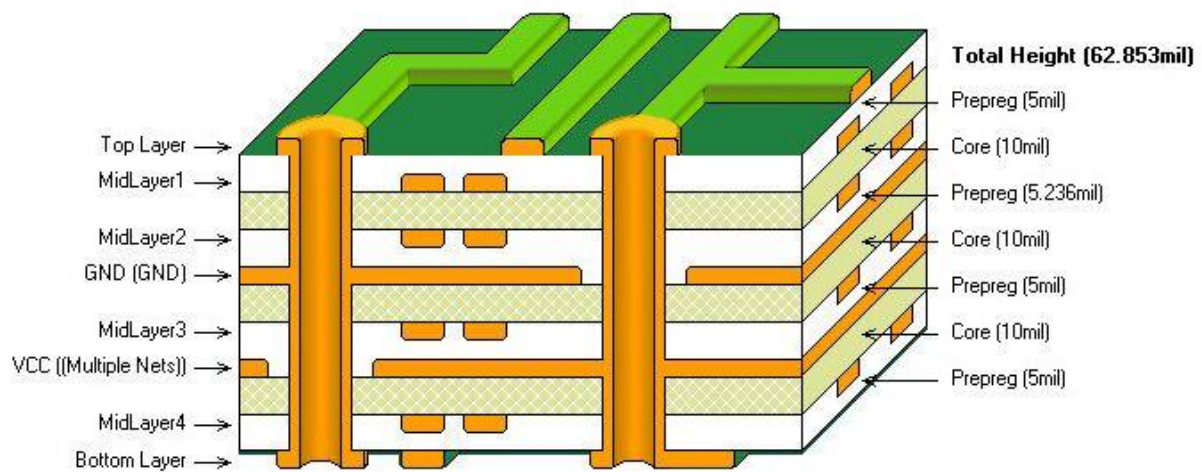
2 layers stack up



4 layers stack up

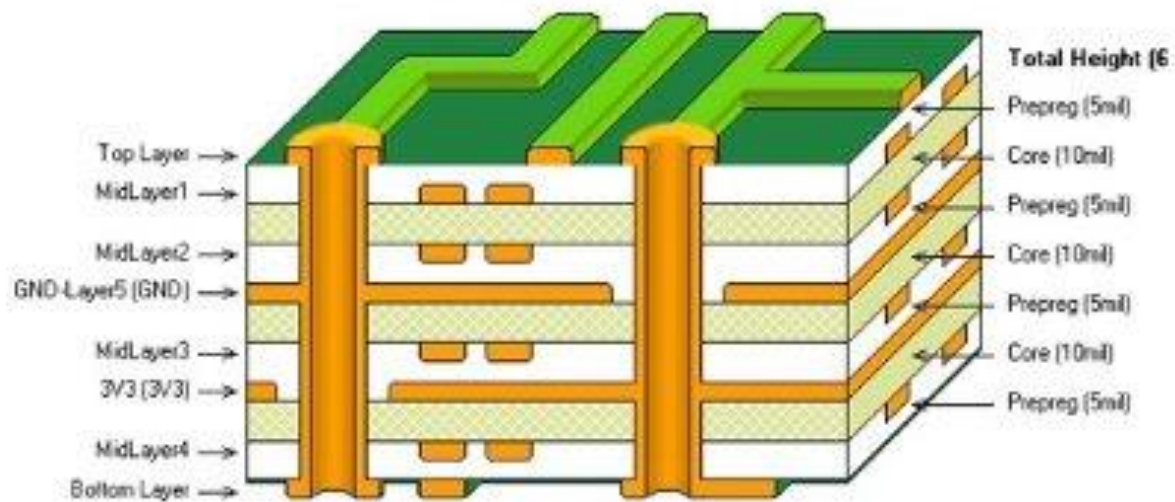


6 layers stack up

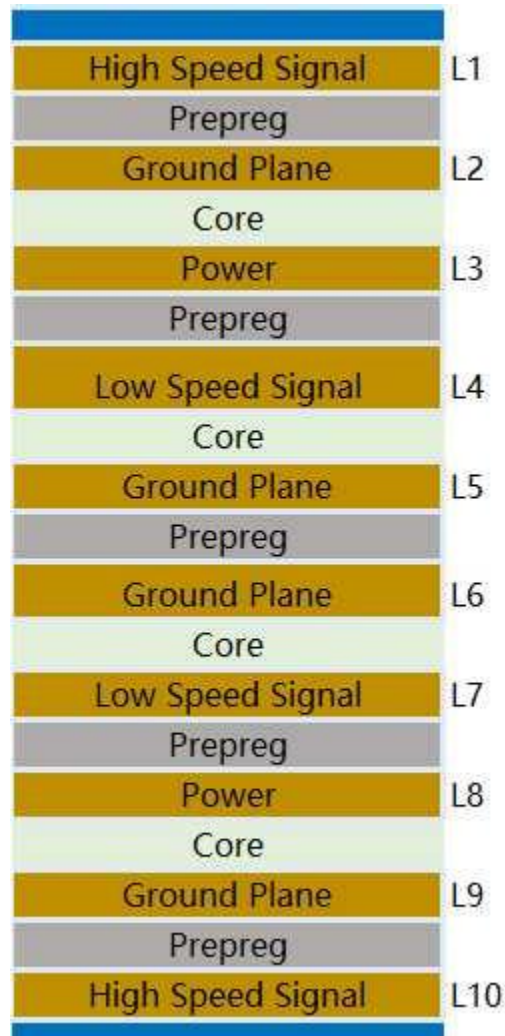


6 Layer PCB Stackup

8 layers stack up



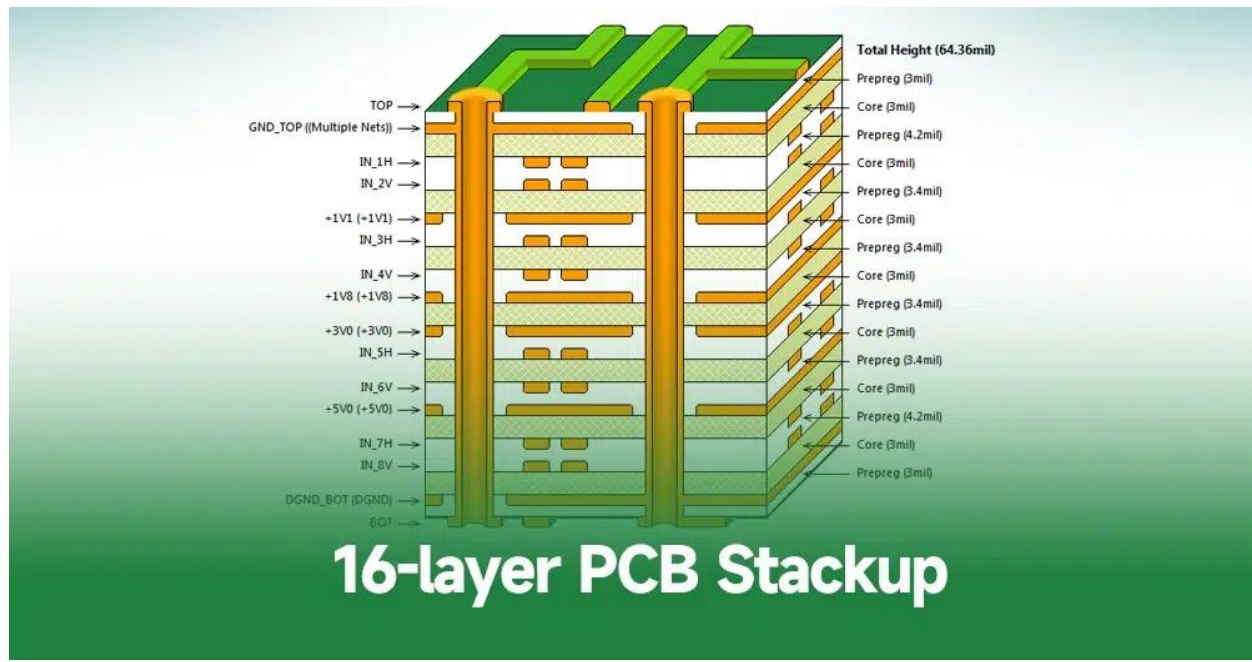
10 layers stack up



12 layers stack up

								Features
#	Name	Material	Type	Thickness	Weight	Dk	Df	
	Top Overlay		Overlay					
	Top Solder	SM-001	Solder Mask	1mil		4	0.03	
	Top Surface Fini...	PbSn	Surface Finish	0.787mil				
1	Top Layer	CF-004	Signal	1.378mil	1oz			
	Dielectric 1	PP-013	Prepreg	3.8mil		4.3	0.02	
2	Int1 (GND)	CF-004	Plane	1.378mil	1oz			
	Dielectric 2	Core-011	Core	4mil		4.4	0.02	
3	Int2 (PWR)	CF-004	Signal	1.378mil	1oz			
	Dielectric 3	PP-013	Prepreg	3.8mil		4.3	0.02	
4	Int3 (Sign)	CF-004	Plane	1.378mil	1oz			
	Dielectric 4	Core-011	Core	4mil		4.4	0.02	
5	Int4 (Sign)	CF-004	Signal	1.378mil	1oz			
	Dielectric 5	PP-014	Prepreg	4.2mil		4.2	0.02	
6	Int5 (GND)	CF-004	Plane	1.378mil	1oz			
	Dielectric 6	Core-011	Core	4mil		4.4	0.02	
7	Int6 (PWR)	CF-004	Plane	1.378mil	1oz			
	Dielectric 7	PP-014	Prepreg	4.2mil		4.2	0.02	
8	Int7 (Sign)	CF-004	Signal	1.378mil	1oz			
	Dielectric 8	Core-011	Core	4mil		4.4	0.02	
9	Int8 (Sign)	CF-004	Plane	1.378mil	1oz			
	Dielectric 9	PP-013	Prepreg	3.8mil		4.3	0.02	
10	Int9 (GND)	CF-004	Signal	1.378mil	1oz			
	Dielectric 10	Core-011	Core	4mil		4.4	0.02	
11	Int10 (PWR)	CF-004	Plane	1.378mil	1oz			
	Dielectric 11	PP-013	Prepreg	3.8mil		4.3	0.02	
12	Bottom Layer	CF-004	Signal	1.378mil	1oz			
	Bottom Surface...	PbSn	Surface Finish	0.787mil				
	Bottom Solder	SM-001	Solder Mask	1mil		4	0.03	
	Bottom Overlay		Overlay					

16 layers stack up



Why 50 ohms impedance

The choice of a 50-ohm impedance in PCB (Printed Circuit Board) traces is often associated with high-frequency RF (Radio Frequency) applications. Here are some reasons why a 50-ohm impedance is commonly used in such scenarios:

1. **Signal Integrity and Transmission Line Matching:** A 50-ohm impedance helps to match the characteristic impedance of the transmission line to the source and load impedances. This matching minimizes signal reflections and ensures efficient power transfer along the transmission line, contributing to better signal integrity.
2. **Reduced Signal Losses:** At higher frequencies, signal losses due to impedance mismatches and reflections become more significant. Using a 50-ohm transmission line helps to minimize these losses, allowing for more reliable signal transmission over longer distances.
3. **Standardization and Compatibility:** A 50-ohm impedance has become a de facto standard in the RF industry. Many RF components, such as connectors, cables, and antennas, are designed to operate optimally within a 50-ohm system. Standardization

simplifies component selection, system integration, and ensures compatibility with other RF devices.

4. **Matching Network Design:** RF circuits often include matching networks to optimize the impedance matching between different components. A 50-ohm system simplifies the design of these matching networks, making it easier to achieve the desired impedance matching and improve overall system performance.
5. **Antenna Matching:** In RF systems, antennas are designed to work efficiently with a specific impedance, often around 50 ohms. Using a 50-ohm transmission line helps to match the antenna impedance, maximizing power transfer and signal efficiency.
6. **Coaxial Cable Compatibility:** Coaxial cables, commonly used in RF applications, often have a characteristic impedance of 50 ohms. Matching the PCB traces to the impedance of the coaxial cable simplifies the overall system design and integration.
7. **Controlled Impedance PCB Design:** High-frequency PCB designs often involve controlled impedance traces to maintain signal integrity. Using a 50-ohm characteristic impedance allows for controlled and predictable signal behavior on the PCB.

Signal Integrity and Transmission Line Matching:

- Example: In a high-speed data transmission system, such as a Gigabit Ethernet application, a 50-ohm impedance-controlled transmission line is used to match the characteristic impedance of the communication channel. This helps minimize signal reflections and ensures that data signals are transmitted with high integrity, reducing the risk of data errors.

Reduced Signal Losses:

- Example: In a radar system operating at high frequencies, a 50-ohm transmission line is employed to connect the RF components. This choice minimizes impedance mismatches and signal reflections, reducing signal losses and allowing the radar system to detect and process weak signals with improved sensitivity.

Standardization and Compatibility:

- Example: In a wireless communication device, such as a Wi-Fi router, the RF connectors and antennas are designed for a 50-ohm system. This standardization allows users to easily replace antennas or use cables from different manufacturers, ensuring compatibility and simplifying the overall setup of the wireless network.

Matching Network Design:

- Example: In a cellular base station, a 50-ohm matching network is designed to optimize the impedance matching between the power amplifier and the antenna. This ensures maximum power transfer, minimizing signal losses, and improving the overall efficiency of the base station.

Antenna Matching:

- Example: In a radio transmitter for a mobile communication device, the antenna is designed with a characteristic impedance of 50 ohms. Using a 50-ohm transmission line helps match the impedance of the antenna, ensuring efficient power transfer and maximizing the range and performance of the wireless communication.

Coaxial Cable Compatibility:

- Example: In a cable television (CATV) system, the coaxial cables used to transmit RF signals have a characteristic impedance of 50 ohms. Designing the PCB traces and connectors in CATV equipment to match this impedance ensures seamless integration with the coaxial cables, reducing signal losses and maintaining signal quality.

Controlled Impedance PCB Design:

- Example: In a high-frequency radar system, the PCB traces that carry RF signals are designed with controlled impedance, typically set to 50 ohms. This controlled impedance helps maintain signal integrity, reduce losses, and ensure predictable signal behavior, critical for accurate radar signal processing.

IPC standard

The IPC (Association Connecting Electronics Industries) is a global industry association that establishes standards for the design, manufacture, and testing of electronic assemblies, including PCBs (Printed Circuit Boards). The IPC standards are widely recognized and used in the electronics industry to ensure consistency and quality in the production of electronic products. Several IPC standards specifically relate to PCBs. Some notable ones include:

1. **IPC-A-600: Acceptability of Printed Boards:** This standard provides visual acceptance criteria for the quality and workmanship of printed boards. It covers a wide range of conditions, such as surface conditions, hole quality, conductor width and spacing, annular ring, and solder mask coverage.
2. **IPC-2221: Generic Standard on Printed Board Design:** This standard provides guidelines for the design of PCBs, covering aspects such as conductor spacing, thermal considerations, and electrical requirements. It helps ensure that the design is manufacturable and meets industry standards.
3. **IPC-2222: Sectional Design Standard for Rigid Organic Printed Boards:** This standard expands on IPC-2221 and provides specific requirements for the sectional design of rigid PCBs. It includes information on materials, mechanical considerations, and electrical performance.
4. **IPC-4101: Specification for Base Materials for Rigid and Multilayer Printed Boards:** This standard outlines the specifications for base materials used in the construction of rigid and multilayer printed boards. It includes information on materials like laminates, prepregs, and bonding sheets.
5. **IPC-7351: Generic Requirements for Surface Mount Design and Land Pattern Standard:** This standard provides guidelines for the design of surface mount land patterns, ensuring compatibility with component footprints. It covers a variety of components, including resistors, capacitors, and integrated circuits.
6. **IPC-6012: Qualification and Performance Specification for Rigid Printed Boards:** This standard establishes the qualification and performance requirements for rigid printed boards. It includes criteria for materials, fabrication methods, electrical performance, and testing.
7. **IPC-6013: Qualification and Performance Specification for Flexible Printed Boards:** Similar to IPC-6012, this standard focuses on flexible printed boards, providing qualification and performance requirements specific to flexible circuits.
8. **IPC-7711/7721: Rework, Modification, and Repair of Electronic Assemblies:** These standards cover the procedures and requirements for reworking, modifying, and repairing electronic assemblies, including PCBs. They provide guidelines for various repair and rework processes.

It's important for PCB designers, manufacturers, and assemblers to be familiar with relevant IPC standards to ensure that their products meet industry-accepted quality and reliability standards. These standards are periodically updated to reflect advancements in technology and changes in industry practices.

Vias in PCB and their types

Vias are small holes in a printed circuit board (PCB) that connect different layers of the board, allowing electrical signals and power to pass through. There are different types of vias, each serving specific purposes in PCB design. The primary types of vias include:

1. Through-Hole Vias (PTH - Plated Through-Hole):

- **Purpose:** Connects traces on different layers through the entire thickness of the PCB.
- **Construction:** Drill holes through the entire PCB, and then copper is plated on the walls of the hole to create electrical continuity.
- **Size:** Typically ranges from 0.2 mm to 0.8 mm in diameter for standard applications.

2. Blind Vias:

- **Purpose:** Connects an outer layer to one or more inner layers but does not go through the entire thickness of the PCB.
- **Construction:** Drilled from the surface of the board to a specific depth, and then copper is plated to create electrical continuity within the drilled portion.
- **Size:** Diameter and depth can vary based on design requirements.

3. Buried Vias:

- **Purpose:** Connects inner layers of a multilayer PCB without extending to the outer layers.
- **Construction:** Drilled between inner layers and plated to provide electrical continuity within the drilled portion.
- **Size:** Diameter and depth can vary based on design requirements.

4. Microvias:

- **Purpose:** Miniaturized versions of through-hole, blind, or buried vias, used in high-density interconnect (HDI) PCBs to save space.
- **Construction:** Much smaller in diameter compared to traditional vias.
- **Size:** Typically less than 0.15 mm in diameter.

5. Transfer vias:

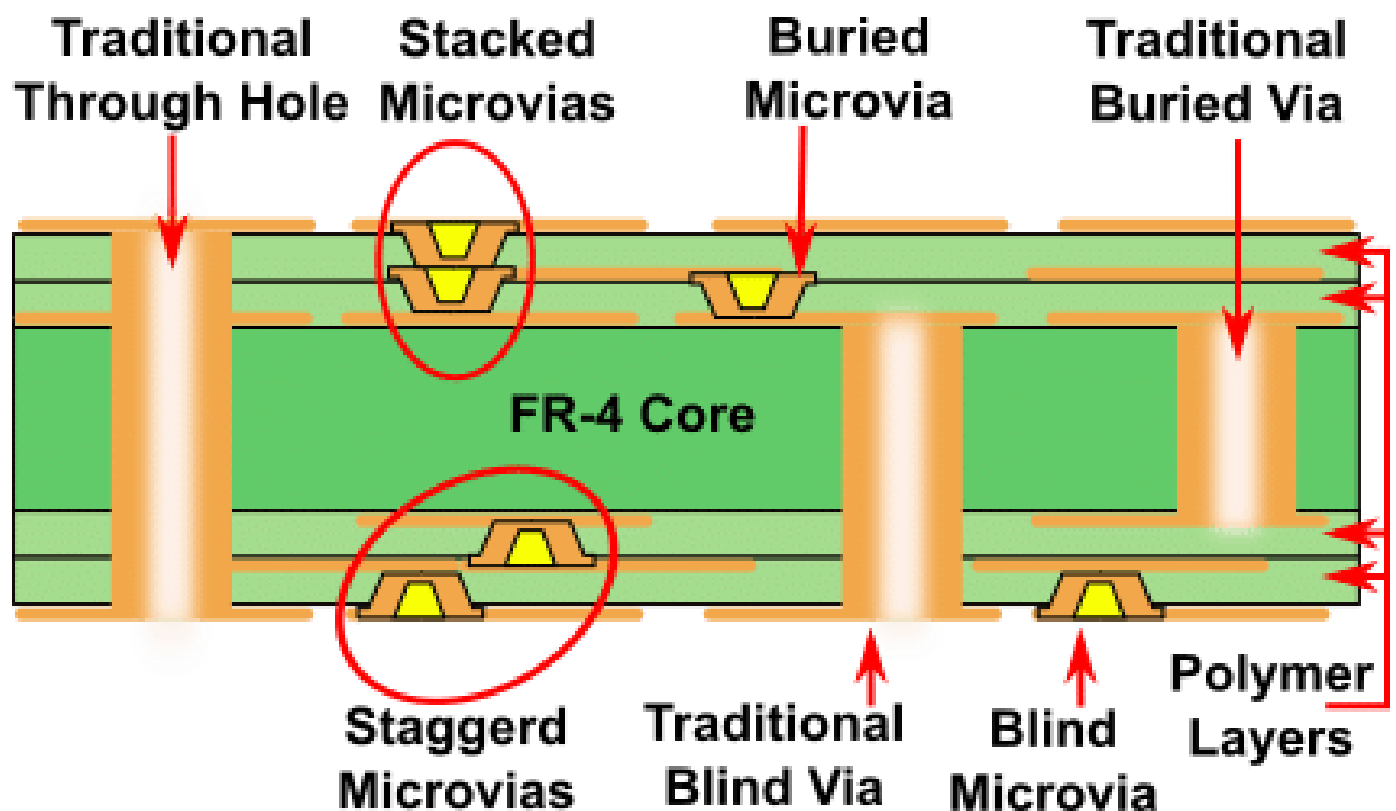
- **Purpose:** we will place a ground via near to signal via because of proper return path.

6. Stitching vias:

Purpose: we use these via incase we have multiple ground and power planes so we use stitching vias to connect properly with each other.

- Mostly we use theses vias to reduce inductance.
- Stitching vias can be used for shielding to suppress energy of electromagnetic waves up to certain frequency entering/leaving a section of pcb.

The size of vias, especially their diameter, is a crucial consideration in PCB design and depends on various factors, including the technology used, the number of layers, and the intended application. Smaller vias are often desirable for high-density designs, but their fabrication may require more advanced processes and may result in increased manufacturing costs.



Differential pairs

It seems like there might be a typo in your question. I assume you meant "differential pairs." If that's the case, a differential pair is a set of two traces that carry equal and opposite signals, commonly used in high-speed digital and analog applications to transmit data while minimizing electromagnetic interference (EMI). Differential signaling helps improve signal integrity by reducing the impact of common-mode noise and enhancing noise immunity.

Here's how you typically route differential pairs on a printed circuit board (PCB):

1. Identify Differential Pair Signals:

- Determine which signals in your design require differential signaling. These are often pairs of signals critical for maintaining signal integrity, such as high-speed data lines.

2. Specify Differential Pair Routing Rules:

- Most PCB design tools provide options to define rules for differential pair routing. Specify parameters such as the separation between the two traces, the width of each trace, and any length-matching requirements.

3. Controlled Impedance Design:

- Ensure that the traces have controlled impedance to maintain signal integrity. The impedance of the traces should match the characteristic impedance of the transmission line. This is crucial for preventing signal reflections and maintaining a consistent signal quality.

4. Use Differential Pair Routing Tools:

- Many PCB design tools have features specifically designed for routing differential pairs. These tools help ensure that the traces are routed in a way that meets the specified rules and requirements.

5. Maintain Symmetry:

- Keep the two traces of the differential pair as symmetric as possible. Symmetry helps ensure that the signals experience similar environmental conditions, reducing the impact of common-mode noise.

6. Avoid Cross Talk:

- Route differential pairs away from other signals, especially high-speed or noisy signals, to minimize the potential for cross talk and interference.

7. Follow Design Guidelines:

- Adhere to the guidelines provided by the IC manufacturer or industry standards for routing differential pairs. These guidelines often specify trace separation, maximum length mismatch, and other critical parameters.

8. Consider Serpentine Routing:

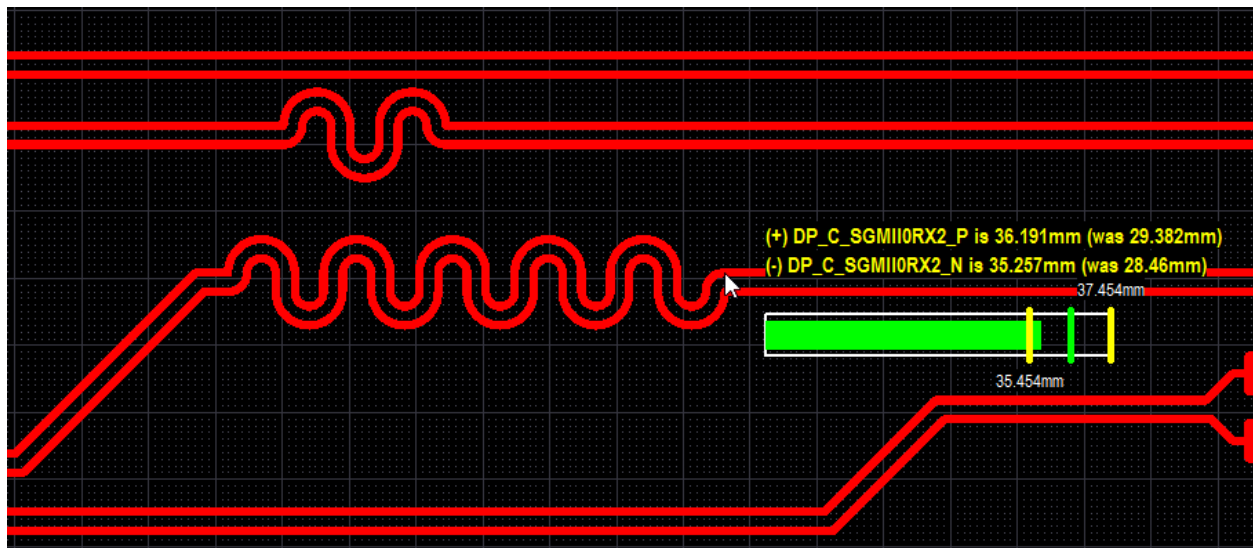
- In some cases, using a serpentine routing pattern (a series of alternating curves) for the differential pair can help balance the lengths of the positive and negative traces.

9. Verify Signal Integrity:

- After routing, perform signal integrity analysis, such as checking for impedance mismatches, signal reflections, and crosstalk. This ensures that the routed differential pairs meet the performance requirements of the system.

10. Document the Design:

- Clearly document the routing details, including the specifications for each differential pair. This documentation is valuable for future reference and for communication with other team members or manufacturers



- Do not place any component and via between differential pair

Width of traces

Temp Rise	10°C			20°C			30°C		
Copper	1/2oz.	1oz.	2oz.	1/2oz.	1oz.	2oz.	1/2oz.	1oz.	2oz.
Trace Width (:inch)	Maximum Current Amps								
0.01	0.5	1	1.4	0.6	1.2	1.6	0.7	1.5	2.2
0.015	0.7	1.2	1.6	0.8	1.3	2.4	1	1.6	3
0.02	0.7	1.3	2.1	1	1.7	3	1.2	2.4	3.6
0.025	0.9	1.7	2.5	1.2	2.2	3.3	1.5	2.8	4
0.03	1.1	1.9	3	1.4	2.5	4	1.7	3.2	5
0.05	1.5	2.6	4	2	3.6	6	2.6	4.4	7.3
0.075	2	3.5	5.7	2.8	4.5	7.8	3.5	6	10
0.1	2.6	4.2	6.9	3.5	6	9.9	4.3	7.5	12.5
0.2	4.2	7	11.5	6	10	11	7.5	13	20.5
0.25	5	8.3	12.3	7.2	12.3	20	9	15	24

High speed design in PCB

Designing a high-speed PCB (Printed Circuit Board) involves careful consideration of various factors to ensure signal integrity, minimize electromagnetic interference (EMI), and meet the performance requirements of high-frequency applications. Here are key guidelines for high-speed PCB design:

1. Controlled Impedance:

- Maintain controlled impedance for critical transmission lines, such as high-speed data lines and clock signals. Use impedance matching techniques to minimize signal reflections and ensure signal integrity.

2. Signal Integrity Analysis:

- Perform signal integrity analysis using simulation tools to identify and address potential issues, including signal reflections, crosstalk, and impedance mismatches.

3. Grounding:

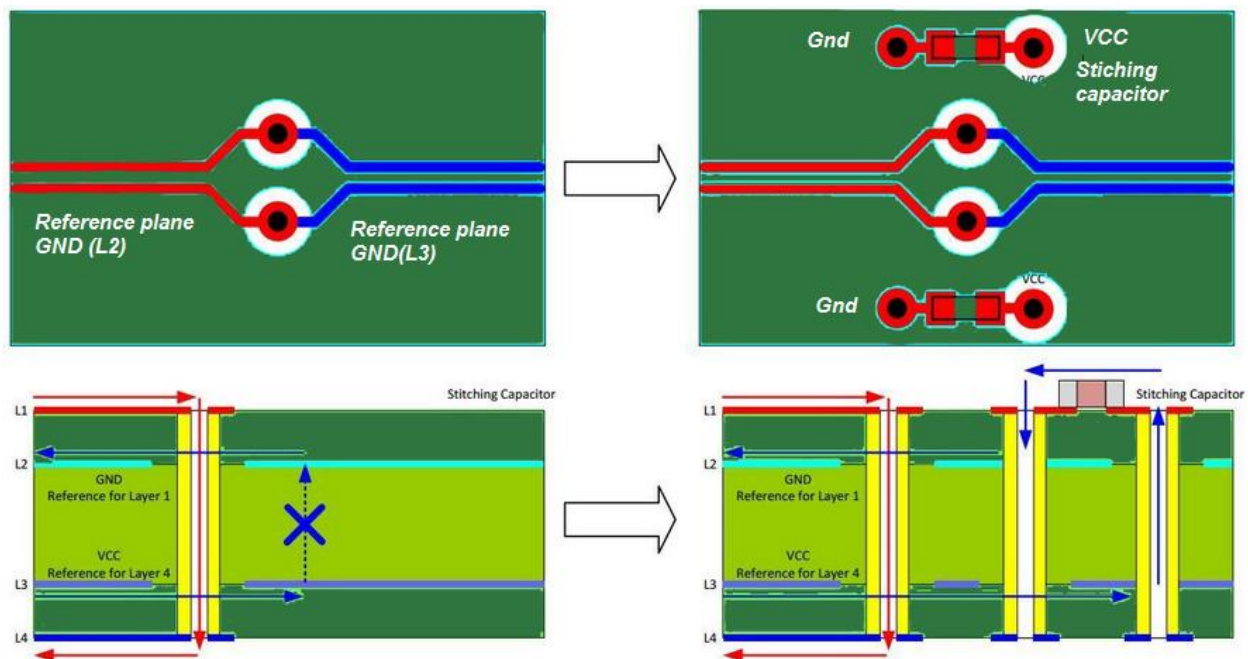
- Implement a solid and low-impedance ground plane. Proper grounding is crucial for minimizing ground loops and reducing electromagnetic interference. Use stitching vias to connect ground planes across different layers.

4. Power Integrity:

- Design a clean and stable power distribution network. Use decoupling capacitors strategically placed near power pins to provide a low-impedance path for high-frequency components.

5. Bypass Capacitors:

- Place bypass capacitors close to high-speed ICs to filter out high-frequency noise from the power supply. Choose capacitor values based on the frequency of operation.



6. Trace Length Matching:

- Match trace lengths for critical differential pairs to maintain signal timing and reduce skew. Tools and features in PCB design software can assist in achieving precise length matching.

7. Differential Pair Routing:

- Route critical signals, such as differential pairs, using controlled impedance traces. Maintain symmetry between positive and negative traces, and follow recommended routing guidelines provided by IC manufacturers.

8. Cross-Hatching:

- Use cross-hatching in ground and power planes to improve thermal dissipation and reduce conductor losses.

9. Via Placement:

- Strategically place vias to minimize signal reflections. Via stubs can introduce impedance mismatches, so consider using blind or buried vias for cleaner signal paths.

10. Shielding:

- Consider using shields or ground planes between sensitive analog and digital sections to reduce interference. Shielding can be particularly important in mixed-signal designs.

11. Connector Placement:

- Carefully place connectors to minimize signal degradation and crosstalk. Avoid routing high-speed traces near the edges of the board, where signal quality can be compromised.

12. Thermal Management:

- Address thermal considerations, especially in designs with high-power components. Use thermal vias to dissipate heat effectively and ensure that the board operates within acceptable temperature limits.

13. Design for Manufacturability:

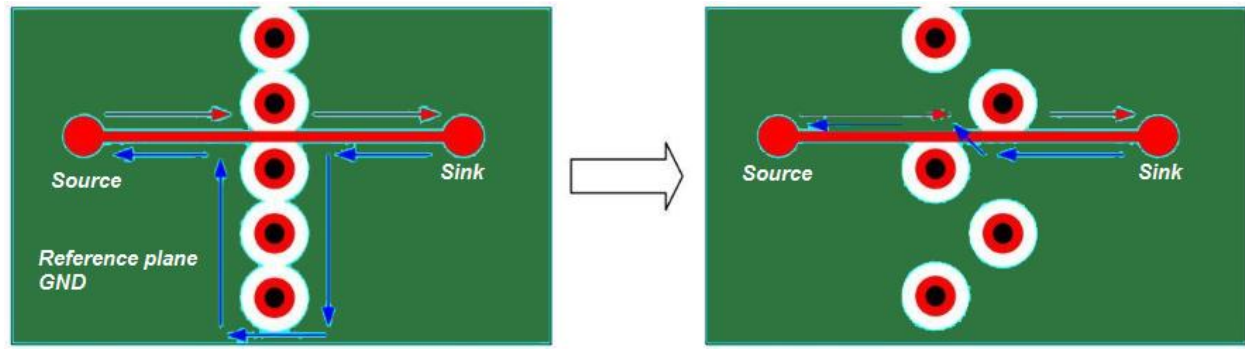
- Consider the manufacturing process, such as the capabilities of your chosen PCB fabrication technology. Ensure that the design can be manufactured reliably and cost-effectively.

14. EMI/EMC Compliance:

- Follow EMI/EMC (Electromagnetic Interference/Electromagnetic Compatibility) guidelines to minimize the emission of unwanted electromagnetic radiation and ensure compliance with regulatory standards.

15. Documentation:

- Clearly document the design, including specifications, constraints, and rationale for specific design choices. This documentation is valuable for collaboration with other team members and for future reference.



PCB Design for manufacturability

Bare board fabrication, critical parameters

- Size of board and number that fit in a panel
- Material of construction
- Number of layers
- Layer total vs Board thickness
- Number of holes per panel
- Smallest hole diameter
- Pad to hole size ratio
- Hole to board size ratio
- Hole to board thickness aspect ratio
- Power/Ground plane clearance from hole wall

When you put 16 layers into a 62mil thick board the dielectrics it is painfully thin and the job for fabricators become much harder

For the 12 layers board the thickness of a board is around 75mil and it is become easy to manage target impedance

Let suppose you have 62mil thick board and it is standing on 4 holes it is become more challenging for them to manage

Most of the fabricator follow the IPC standard as the guideline for producing circuit

We must need to know IPC standards

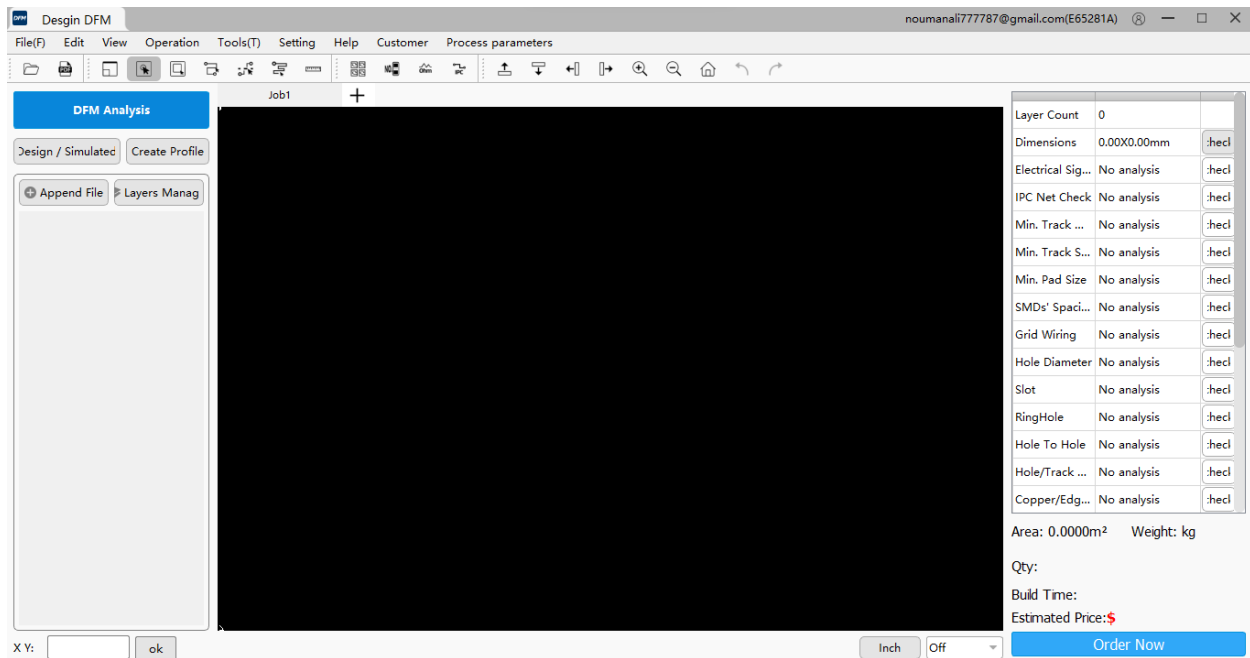
2200, 2221, 2222, 6000, 6011, 6012, 6013, 610, 611

If we are PCB design Engineer we have to buy these standard the cost is around 20 to 50 dollars

Software to manage Gerber file

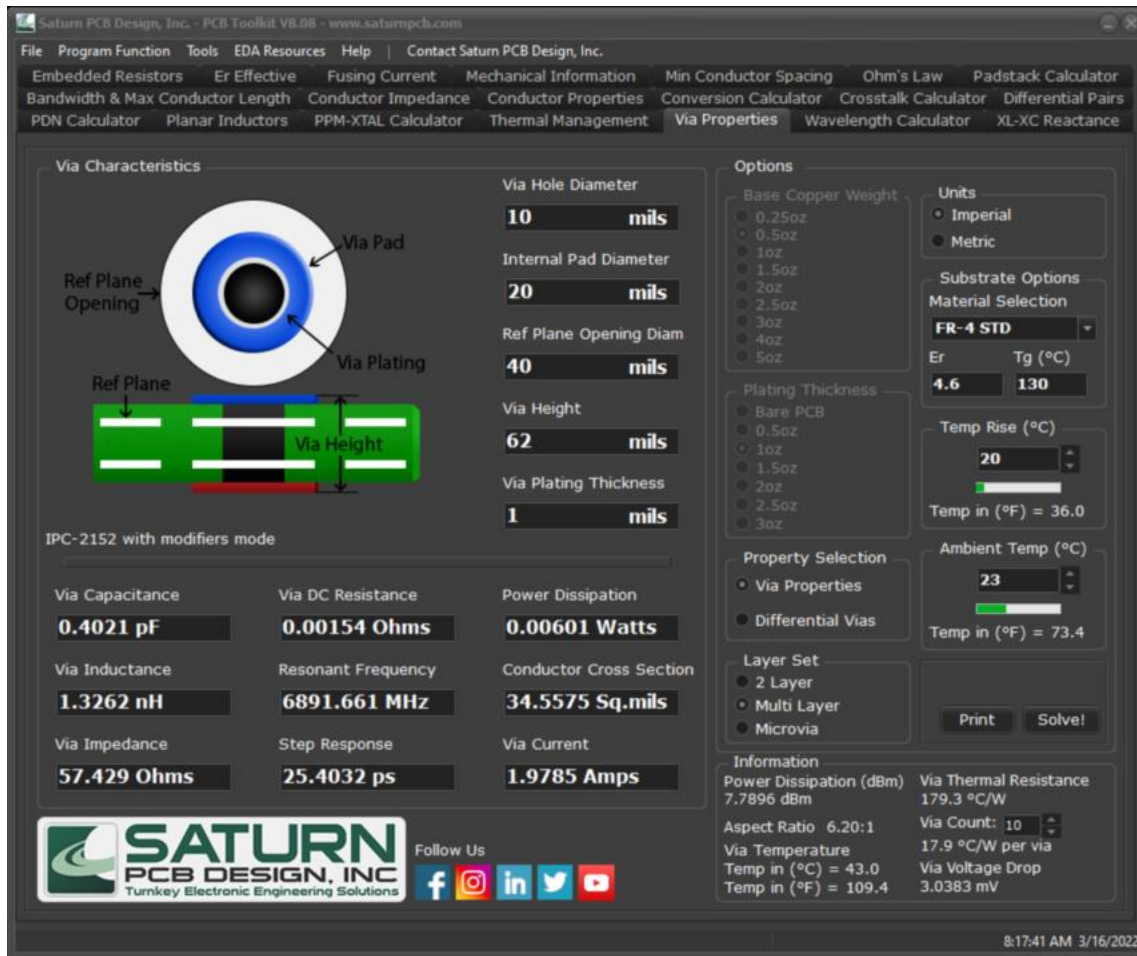
Next DFM

Link for download: <https://www.nextpcb.com/dfm/>



Software to calculate width of traces and via size

Link for download: <https://saturnpcb.com/saturn-pcb-toolkit/>



Decoupling capacitor

Designing a high-speed PCB (Printed Circuit Board) involves careful consideration of various factors to ensure signal integrity, minimize electromagnetic interference (EMI), and meet the performance requirements of high-frequency applications. Here are key guidelines for high-speed PCB design:

1. Controlled Impedance:

- Maintain controlled impedance for critical transmission lines, such as high-speed data lines and clock signals. Use impedance matching techniques to minimize signal reflections and ensure signal integrity.

2. Signal Integrity Analysis:

- Perform signal integrity analysis using simulation tools to identify and address potential issues, including signal reflections, crosstalk, and impedance mismatches.

3. Grounding:

- Implement a solid and low-impedance ground plane. Proper grounding is crucial for minimizing ground loops and reducing electromagnetic interference. Use stitching vias to connect ground planes across different layers.

4. Power Integrity:

- Design a clean and stable power distribution network. Use decoupling capacitors strategically placed near power pins to provide a low-impedance path for high-frequency components.

5. Bypass Capacitors:

- Place bypass capacitors close to high-speed ICs to filter out high-frequency noise from the power supply. Choose capacitor values based on the frequency of operation.

6. Trace Length Matching:

- Match trace lengths for critical differential pairs to maintain signal timing and reduce skew. Tools and features in PCB design software can assist in achieving precise length matching.

7. Differential Pair Routing:

- Route critical signals, such as differential pairs, using controlled impedance traces. Maintain symmetry between positive and negative traces, and follow recommended routing guidelines provided by IC manufacturers.

8. Cross-Hatching:

- Use cross-hatching in ground and power planes to improve thermal dissipation and reduce conductor losses.

9. Via Placement:

- Strategically place vias to minimize signal reflections. Via stubs can introduce impedance mismatches, so consider using blind or buried vias for cleaner signal paths.

10. Shielding:

- Consider using shields or ground planes between sensitive analog and digital sections to reduce interference. Shielding can be particularly important in mixed-signal designs.

11. Connector Placement:

- Carefully place connectors to minimize signal degradation and crosstalk. Avoid routing high-speed traces near the edges of the board, where signal quality can be compromised.

12. Thermal Management:

- Address thermal considerations, especially in designs with high-power components. Use thermal vias to dissipate heat effectively and ensure that the board operates within acceptable temperature limits.

13. Design for Manufacturability:

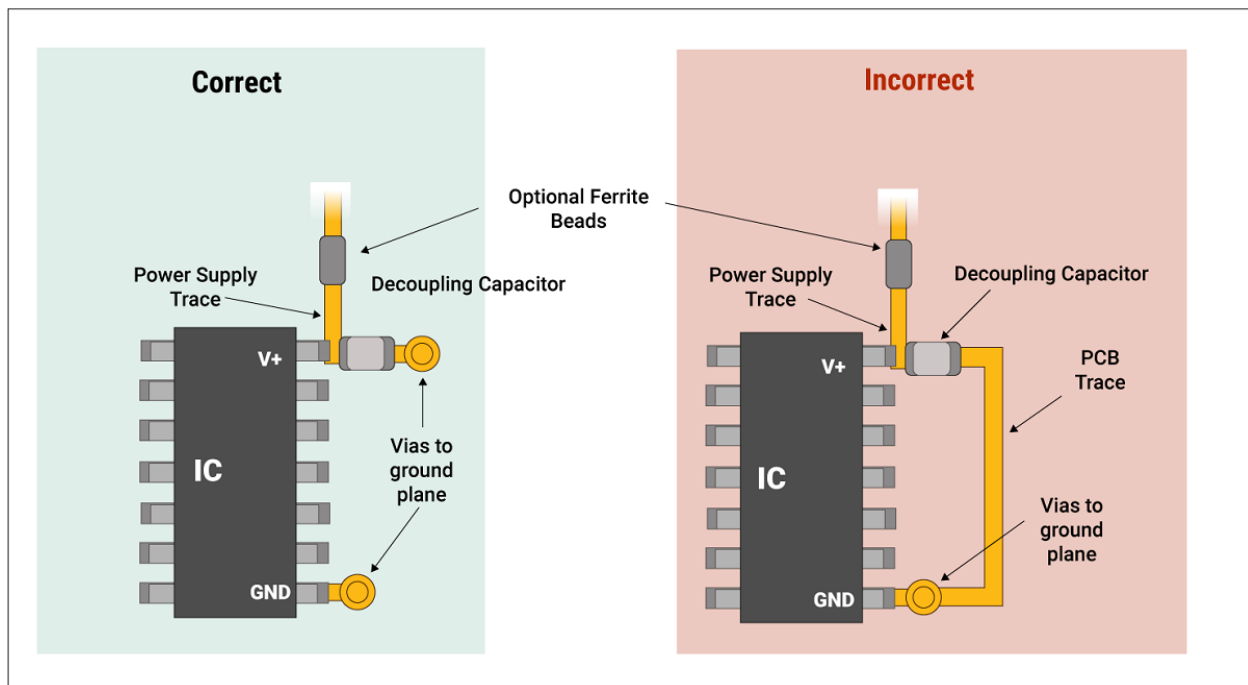
- Consider the manufacturing process, such as the capabilities of your chosen PCB fabrication technology. Ensure that the design can be manufactured reliably and cost-effectively.

14. EMI/EMC Compliance:

- Follow EMI/EMC (Electromagnetic Interference/Electromagnetic Compatibility) guidelines to minimize the emission of unwanted electromagnetic radiation and ensure compliance with regulatory standards.

15. Documentation:

- Clearly document the design, including specifications, constraints, and rationale for specific design choices. This documentation is valuable for collaboration with other team members and for future reference.



Ferrite bead

A ferrite bead, also known as a ferrite choke or ferrite ring, is a passive electronic component used to suppress high-frequency noise in electronic circuits. It is commonly employed in printed circuit board (PCB) designs to reduce electromagnetic interference (EMI) and radio frequency interference (RFI) in both power and signal lines. Ferrite beads are particularly effective at attenuating high-frequency noise while allowing lower-frequency signals or power to pass through with minimal impedance.

Here's why ferrite beads are used in PCBs:

1. **High-Frequency Noise Suppression:**

- Ferrite beads are most effective at suppressing high-frequency noise, typically in the range of tens of megahertz to several gigahertz. They provide a high impedance at these frequencies, acting as a filter to attenuate unwanted noise.

2. **Impedance at High Frequencies:**

- At high frequencies, a ferrite bead exhibits significant impedance due to its magnetic properties. This impedance effectively blocks or attenuates high-frequency noise, preventing it from propagating along the conductor.

3. **Easy Integration into Circuits:**

- Ferrite beads are available in various form factors, including through-hole and surface-mount devices, making them easy to integrate into PCB designs. They can

be placed in series with signal lines or power lines, providing a simple and effective means of noise suppression.

4. Dual Functionality - Choke and Filter:

- Ferrite beads act as both chokes (inductive components) and filters. They provide inductive impedance to attenuate high-frequency noise, and their inherent parasitic capacitance adds a capacitive filtering effect, further contributing to noise suppression.

5. Compatibility with Power Supplies:

- Ferrite beads are commonly used on power supply lines to suppress conducted EMI. Placing ferrite beads on the power lines of sensitive components helps prevent high-frequency noise from entering the power supply network.

6. Preventing EMI from Radiating:

- By suppressing high-frequency noise, ferrite beads help prevent electromagnetic interference from radiating into the environment. This is particularly important in designs where compliance with electromagnetic compatibility (EMC) standards is required.

7. Signal Integrity Improvement:

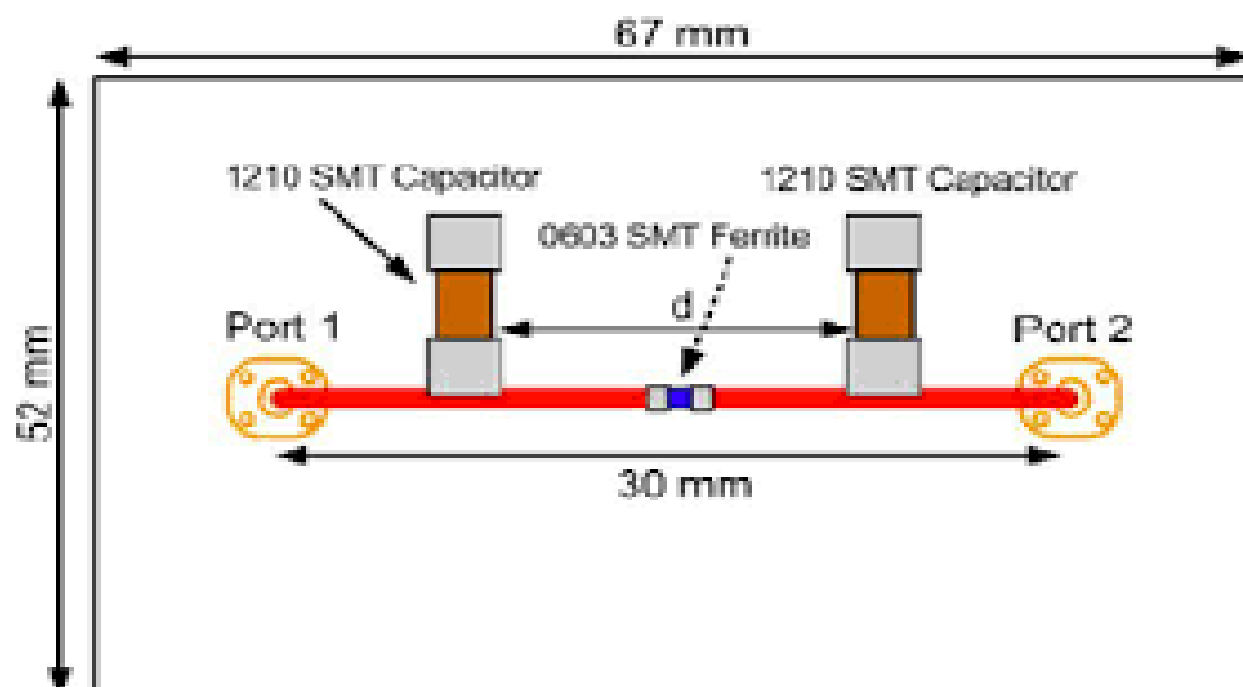
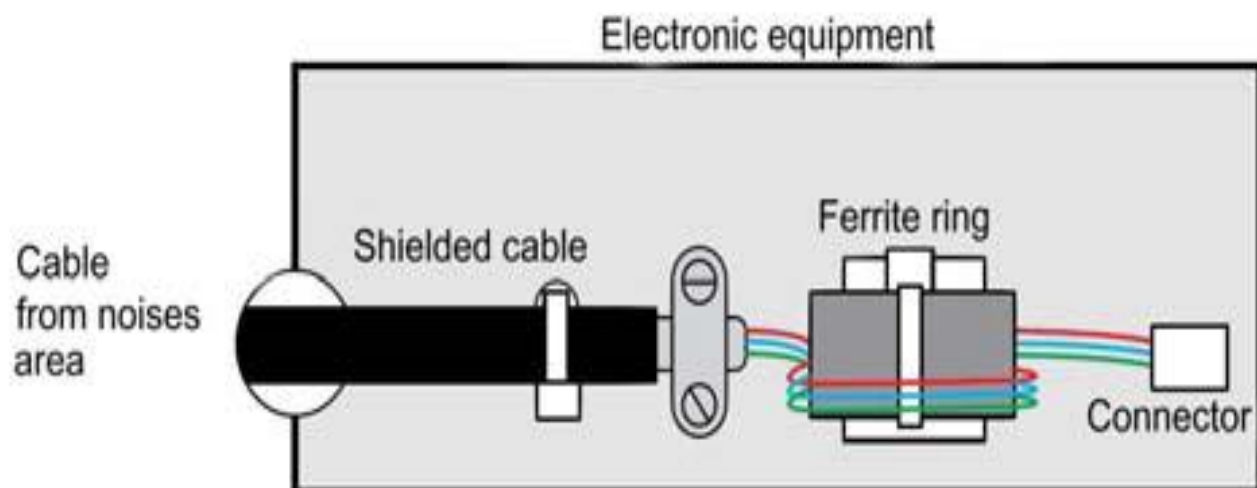
- In some cases, ferrite beads can improve signal integrity by reducing reflections and attenuating noise that may affect the performance of high-speed digital or analog signals.

8. Frequency Selectivity:

- Ferrite beads can be chosen based on their impedance characteristics at specific frequencies. This allows designers to tailor the filtering effect to address the frequency ranges where noise is most problematic.

9. Reducing Common-Mode Noise:

- Ferrite beads are effective in reducing common-mode noise, which is noise that appears in-phase on both conductors of a signal pair. Placing ferrite beads on signal lines can help break the common-mode current path and reduce noise.



SMD components

SMD stands for "Surface Mount Device" or "Surface Mount Technology." It refers to the method of assembling and mounting electronic components directly onto the surface of a printed circuit board (PCB). In contrast to through-hole technology, where components have leads that pass through holes in the PCB, surface mount components have no leads and are soldered directly to the surface of the board.

Surface mount components come in various sizes, and their dimensions are typically specified using a standardized system that includes package codes. The size of an SMD component is often expressed in terms of its dimensions, such as length, width, and height. Additionally, the component's package code provides information about its physical size and characteristics.

Common SMD component package codes include:

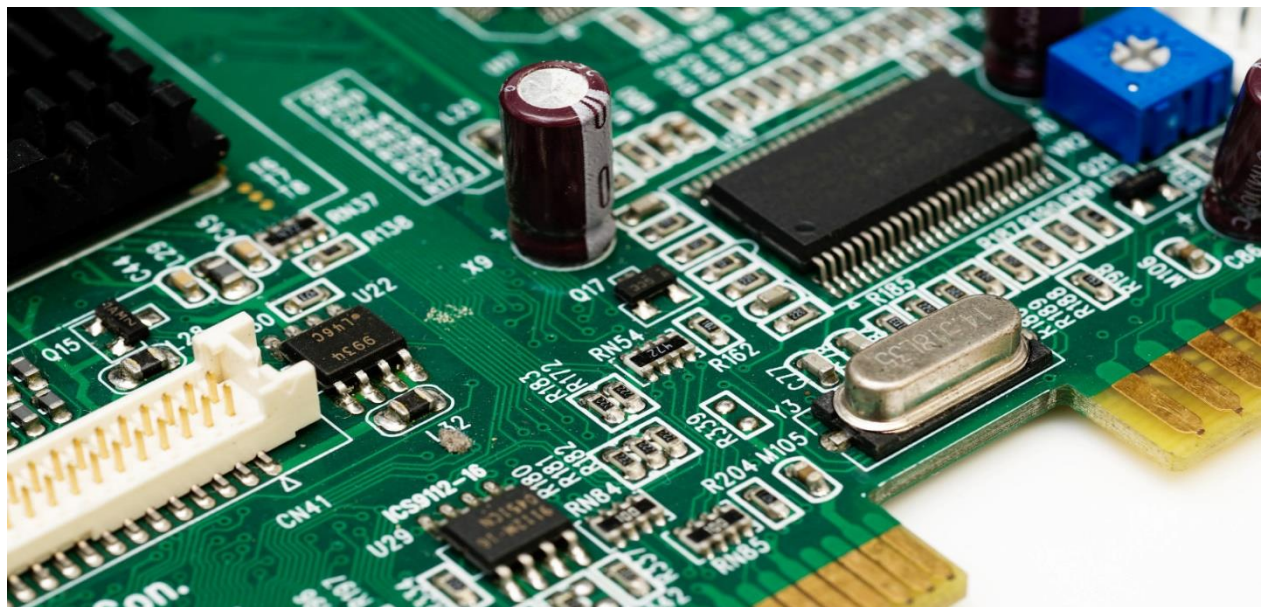
1. Basic Outline Codes:

- **Size Codes:** These codes often use a combination of numbers and letters to represent the size of the component. For example, "0603" indicates a component with dimensions of 0.06 inches by 0.03 inches.
- **Metric Codes:** Some SMD components use metric codes, such as "1206," where the numbers represent the dimensions in metric units (in this case, 12.0 mm by 6.0 mm).

2. Specific Component Package Codes:

- **SMD Resistors and Capacitors:** Common package codes include 0402, 0603, 0805, 1206, 1210, 1812, and 2010. The numbers in these codes often correspond to the length and width dimensions in hundredths of an inch or millimeters.
- **SOT (Small Outline Transistor) Packages:** Examples include SOT-23, SOT-223, SOT-323, etc. These are used for small transistors and other semiconductor devices.
- **QFP (Quad Flat Package):** Example codes include QFP-44, QFP-64, QFP-100, etc. These are used for integrated circuits (ICs) with a square or rectangular shape.
- **BGA (Ball Grid Array):** Examples include BGA-256, BGA-676, etc. BGA packages have solder balls on the bottom of the component and are often used for high-pin-count ICs.
- **CSP (Chip Scale Package):** CSP-10, CSP-20, etc. CSP packages are designed to be similar in size to the actual silicon chip they encapsulate.
- **DFN (Dual Flat No-Lead):** Example codes include DFN-8, DFN-10, etc. DFN packages have a small, flat body with no leads extending from the sides.

Resistor Case Code	Imperial			Size	Metric			
	Approx. Length (in)	Approx. Width (in)	Power (W)		Resistor Case Code	Approx. Length (mm)	Approx. Width (mm)	Power (W)
01005	0.016	0.008	0.031	-	0402	0.4	0.2	0.031
0201	0.02	0.01	1 / 20 (0.05)	-	0603	0.6	0.3	1 / 20 (0.05)
0402	0.04	0.02	1 / 16 (0.062)	-	1005	1.0	0.5	1 / 16 (0.062)
0603	0.06	0.03	1 / 10 (0.10)	-	1608	1.6	0.8	1 / 10 (0.10)
0805	0.08	0.05	1 / 8 (0.125)	-	2012	2.0	1.25	1 / 8 (0.125)
1206	0.125	0.06	1 / 4 (0.25)	-	3216	3.2	1.6	1 / 4 (0.25)
1210	0.125	0.10	1 / 2 (0.5)	-	3225	3.2	2.5	1 / 2 (0.5)
1812	0.18	0.125	3 / 4 (0.75)	-	4532	4.5	3.2	3 / 4 (0.75)
2010	0.20	0.10	3 / 4 (0.75)	-	5025	5.0	2.5	3 / 4 (0.75)
2512	0.25	0.125	1	-	6332	6.3	3.2	1



Through hole components

Through-hole components are electronic components that have leads (metallic wires or pins) designed to be inserted through holes in a printed circuit board (PCB). These components are then soldered to the opposite side of the board to establish electrical connections. Through-hole technology (THT) has been widely used in electronic manufacturing for many years, but it has become less prevalent in recent times with the increased adoption of surface mount technology (SMT).

Key characteristics of through-hole components include:

1. Leads:

- Through-hole components typically have long, straight leads that are designed to be inserted into holes on a PCB. The leads are then bent or clinched on the reverse side of the board to hold the component in place.

2. Connection Method:

- Through-hole components are soldered to the PCB by applying solder to the joint where the leads emerge on the reverse side. The solder forms a mechanical and electrical connection between the component and the board.

3. Component Types:

- Various types of electronic components are available in through-hole packages, including resistors, capacitors, diodes, transistors, integrated circuits, and more. Many older and larger components are commonly found in through-hole packages.

4. Ease of Assembly and Repair:

- Through-hole technology is known for being relatively easy to assemble and repair manually. It is often more forgiving of manual soldering techniques, making it accessible for hobbyists and small-scale production.

