Dr.T.M.SELVAKUMARI

Assistant Professor

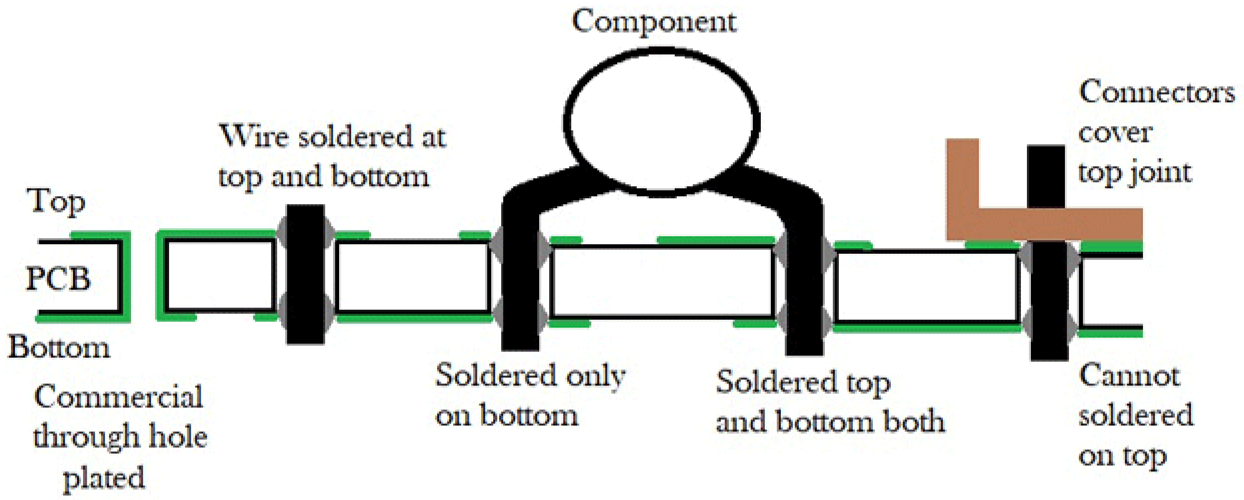
Department of Physics

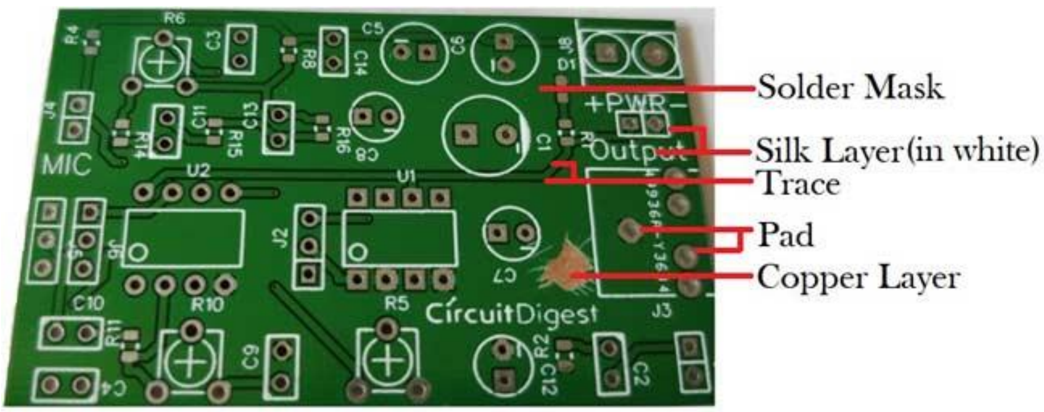
Designing & Fabrication of PCB

PCB- Introduction:

PCB is a copper laminated and non-conductive **Printed Circuit Board**, in which all electrical and electronic components are connected together in one common board with physical support for all components with base of board. When PCB is not developed, at that time all components are connected with a wire which increases complexity and decreases reliability of the circuit, by this way we cannot make a very large circuit like motherboard. In PCB, all components are connected without wires; **all components are connected internally**, so it will reduce the complexity of the overall circuit design. PCB is used to provide electricity and connectivity between the components, by which it functions the way it was designed. PCBs can be customised for any specifications to user requirements. It can be found in many electronics devices like; TV, Mobile, Digital camera, Computers parts like; Graphic cards, Motherboard, etc. It also used in many fields like; medical devices, industrial machinery, automotive industries, lighting, etc.

Different Parts of PCB:





Pad: Pad is nothing but a piece of copper on which lead of components are mounted and on which soldering are done. Pad provides the mechanical support to the components.

Trace: In PCB, components are not connected with the help of wires. All components are connected with a conducting material like copper. This copper part of PCB is used to connect all components that is known as trace. Trace is looks like below figure.

Layers: According to application, cost and available space of circuit, user can choose the layer of PCB. Most simple in construction, easy to design and most useful in routine life is single layer PCB. But for very large and complex circuit, double layer PCB or Multi-layer PCB is most preferred compared to single layer PCB. Now a day, in multi-layer PCB, 10-12 layers can be connected and most critical thing is to communicate between the components in different layer.

Silk layer: Silk layer is used for printing line, text or any art on the surface of PCB. Usually, for screen printing epoxy ink is used. Silk layer can be used in top and/or bottom layer of PCB according to user requirement which is known as silk screen TOP and silk screen BOTTOM.

Top and bottom layer: In Top layer of PCB, all components are mounted in this layer of PCB. Generally, this layer is green coloured. In bottom layer of PCB, all components are soldered through the hole and lead of components is known as bottom layer of PCB. Sometime, in top and/or bottom layer PCB is coated with green colour layer, which is known as solder mask.

Solder Mask: There is one additional layer on the top of copper layer called as Solder Mask. This layer generally has green colour but it can be of any colour. This insulating layer is used for to prevent accidental contact of pads with other conductive material on PCB.

PCB Materials:

The main element is **dielectric substrate** which is rigid or flexible. This dielectric substrate is used with conducting material like copper on it. As dielectric material, the glass epoxy laminates or composite materials are used.

1) FR4:

FR is stand for FIRE RETARDENT. For all type of PCB manufacturing, most common glass laminated material is FR4. Based on woven glass-epoxy compounds, FR4 is a composite material which is most useful because it provides very good mechanical strength.

2) FR-1 and FR-2:

This material is made from paper and phenol compounds and this material is used for only single layer PCB. Both FR1 and FR2 has similar characteristic, the only difference is glass transition temperature. FR1 has higher glass transition temperature compared to FR2. These materials are also sub divided in standard, halogen free and non-hydrophobic.

3) CEM-1:

This material is made from paper and two layer of woven glass epoxy and phenol compounds and this material is used for Single sided PCB only. CEM-1 can used instead of FR4, but price of CEM1 is higher than FR4.

4) CEM-3:

This material is white coloured, glass epoxy compound which is mostly used in double layer PCB. CEM-3 has lower mechanical strength compared to FR4, but it is cheaper than FR4. So, this is a good alternative of FR4.

5) Polyimide:

This material is used in flexible PCB. This material is made from kepton, rogers, dupont. This material has good electrical properties, felicity, wide temperature range and high chemical resistance. Working temperature of this material is -200℃ to 300℃.

PCB material classification:

1, classification according to reinforced materials (the most commonly way)

a. Paper board (FR-1, FR-2, FR-3)

b. Epoxy Glass Cloth (FR-4, FR-5)

c. Composite board (CEM-1, CEM-3)

d. HDI board (RCC-Resin Coated Coppe)

e. Special board (metal board, ceramic board, etc.)

2, classification according to the different types of resin

a. Epoxy resin

b. Polyester resin

c. PI resin (polyimide)

PCB Material Properties

1.Mechanical Properties

- **Bending (Flexural) Strength**

The ability of your board to resist deformation or breakage when under bending stress. Standards for rigid, flex and rigid-flex must be adhered to according to IPC-6013C.

- **Time to Delamination**

A measure of how long a PCB's layers will remain attached when exposed to temperature changes or moisture.

- **Density (g/cm^3)**

The amount of mass per volume for a dielectric.

- **Peel Strength**

The ability of copper layers and dielectrics to remain bonded under thermal and chemical stress.

2. Electrical Properties

- **Dielectric Constant,** *dk*

Affects the signal integrity and impedance of a PCB material. Should be constant over a broad range of frequencies for high-speed applications.

- **Relative Permittivity,**

Referred to as relative *dk*.

- **Surface Resistivity, ρS (Ω-m)**

A measure of a dielectric material's surface resistance to the flow of electricity. Susceptible to change with temperature and moisture. Should be high to facilitate good signal integrity.

- **Volume Resistivity, ρ (Ω/m2)**

Measure of a dielectric's volume resistance. Preferably high to maintain isolation between layers.

- **Dissipation Factor, Df**

A measure of signal or power loss of a PCB material. Not a factor for digital signals, but can be significant for analog signals, especially for frequencies above 1GHz as Df increases with increased frequency.

- **Electrical Strength (V/mil)**

The ability of a PCB to resist electrical breakdown in the z-direction.

3. Thermal Properties

- **Glass Transition Temperature, Tg (°C)**

The temperature at which a PCB substrate will reversibly change from hard to soft. Should be lower than solder temperature.

- **Decomposition Temperature, Td (°C)**

The temperature at which a PCB substrate will permanently decompose. Should be lower than solder temperature.

- **Coefficient of Thermal Expansion, CTE (ppm)**

The rate of expansion for a PCB in response to heat. Generally should be kept as low as possible.

- **Thermal Conductivity, k (W/m)**

The rate at which heat is transferred through a material. Higher for conductors as compared to insulators.

4.Chemical Properties

- **Methylene Chloride Resistance, MCR (%)**

Gives the percentage absorption of methylene chloride, a volatile toxic solvent used in PCB production. Typically, ≤ 0.2% for dielectrics.

- **Water Absorption (%)**

This value gives the amount of moisture that will be absorbed by the PCB material when submerged. For most materials, this value is ≤ 0.2%. May effect electrical and thermal properties of a dielectric.

- **Combustibility**

According to the Standard for Safety of Flammability of Plastic Materials for Parts in Devices and Appliances (UL 94), there are time limits on how long a material may burn in response to combustion. PCB materials are limited to 10s per specimen.

As shown above, there are many PCB material properties to consider. Now, let's see how to use this information to optimize your board's manufacturing process. Managing the Impact of PCB Material Properties on Your Board's Manufacture

Once your board is manufactured, the PCB material properties presented above become fixed parameters that cannot be manipulated or altered. These parameters will determine how your PCB will function as well as your board's quality and reliability. Therefore, it is necessary to define them during design so they can be incorporated into the manufacturing process. In most cases, it is difficult and unnecessary to explicitly define your material properties. Instead, ensuring that the properties are within acceptable ranges for your design will suffice. This can be achieved by following these procedures:

1. Select the best materials for your board design

Board material requirements are greatly influenced by the functionality and signal types of your board. Therefore, it is important to know the correlation between board type and material properties in order to choose materials that are capable of withstanding your design requirements.

2. Determine your board's stackup

In addition to selecting the materials that will comprise your PCB's construction, you also need to determine its vertical layout or PCB stackup. Especially important are the thickness values for the layers which will impact PCB material properties.

3. Understand the board manufacturing process

In order to select a contract manufacturer (CM) that is capable of meeting your quality control, turnaround time or critical system capability, you need to understand the PCB manufacturing steps and how your choices impact them.

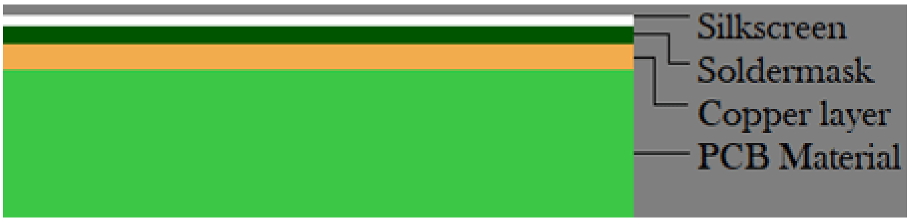
<u>Types of PCB:</u>

There are several types of PCB available for the circuit. Out of these types of PCB, we have to choose the appropriate type of PCB according to our application.

1. Single-layer PCB
2. Double-layer PCB
3. Multi-layer PCB
4. Flexible PCB
5. Aluminium backed PCB
6. Flex-rigid PCB

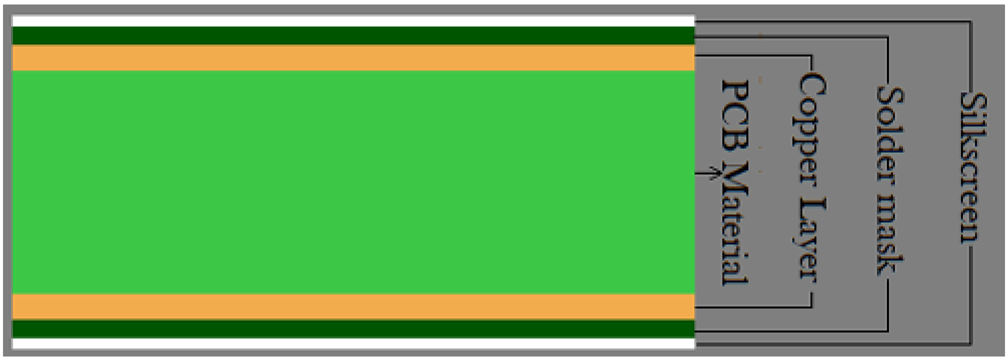
1) Single Layer PCB:

A single layer PCB is also known as **single sided PCB**. This type of PCB is simple and most used PCB because these PCBs are easy to design and manufacture. One side of this PCB is coated with a layer of any conducting material. Generally, copper is used as conducting material for PCB, because copper has very good conducting characteristic. A layer of solder mask is used to protect PCB against oxidation followed by silk screen to mark out all of the components on the PCB. In this type of PCB, only one side of the PCB is used to connect different types of electrical or electronics components like resistor, capacitor, inductor, etc. These components are soldered. These PCBs are used in low cost and bulk manufacturing application like calculators, radio, printers and the solid-state drive.



2) Double Layer PCB:

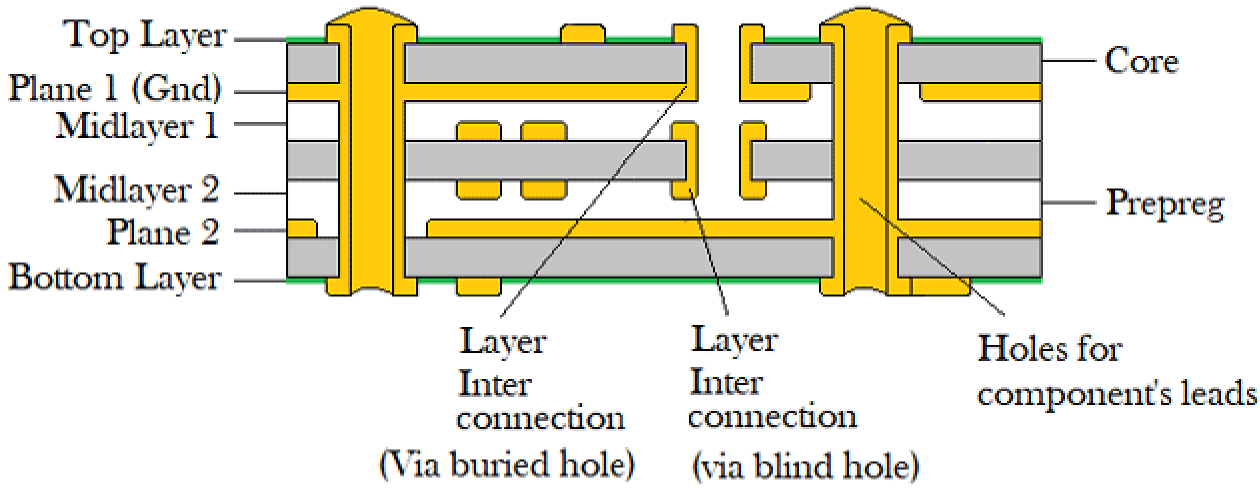
Double layer PCB is also known as **double sided PCB**. As name suggests, in this type of PCB, a thin layer of conducting material, like copper is applied to both top and bottom sides of the board. In PCB, on different layer of board, consist via, which has two pads in corresponding position on different layers. These are electrically connected by a hole through the board, which is shown in figure-2b. More flexible, relatively lower cost, and most important advantage of this type of PCB board is its reduced size which makes circuit compact. This type of PCB is mostly used in industrial controls, converter, UPS system, HVAC application, Phone, Amplifier and Power monitoring systems.



Double Layer PCB

3) Multi-Layer PCB:

Multilayer PCB has more than two layers. It means that, this type of PCB has at least three conductive layers of copper. For securing the board glue is sandwiched between the layer of insulation which ensures that the excess heat will not damage any component of circuit. This type PCB designing is very complex and used in very complicated and large electrical task in very low space and compact circuit. This type of PCB is used in large application like GPS technology, satellite system, medical equipment, file server and data storage.

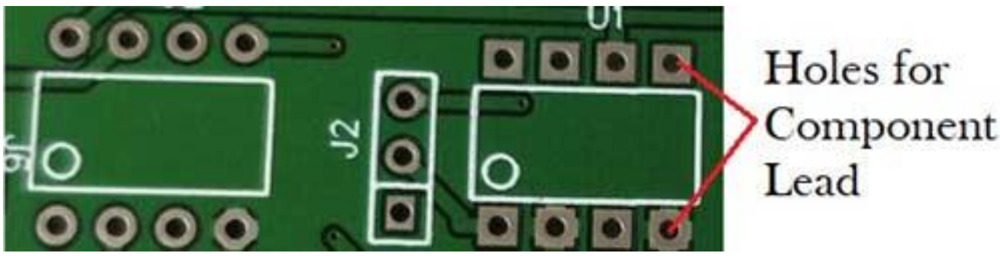


Types of PCBs According to Mounting System

1. Through-hole PCB
2. Surface mounted PCB

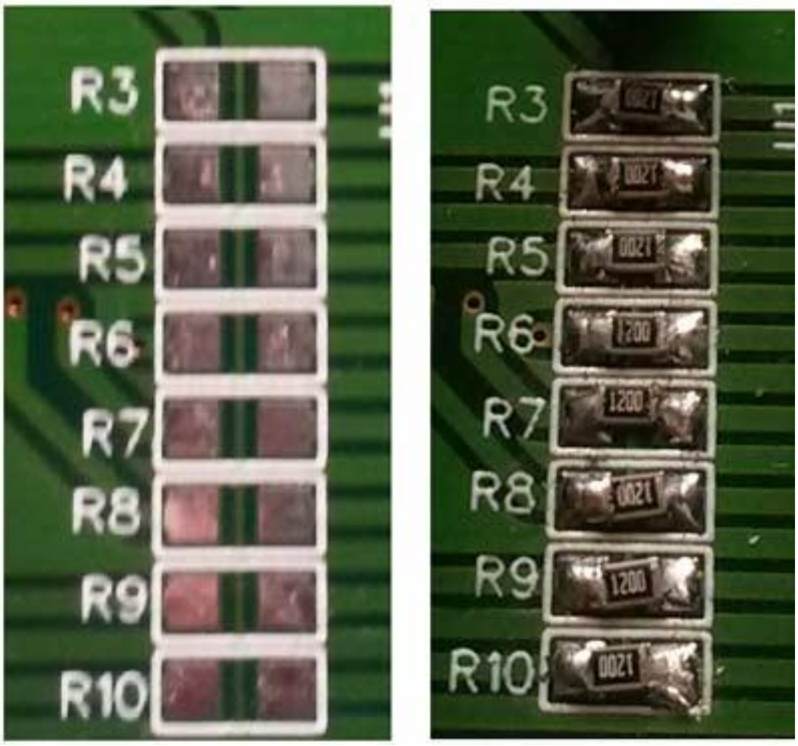
1) Through-hole PCB:

In this type of PCB, we have to make hole using drill on PCB. In these holes, leads of components are mounted and soldered to pads situated on opposite side of PCB. This technology is most useful because it gives more mechanical support to electrical components and very reliable technology for mounting of components but drilling in PCB make it more expensive. In single layer PCB, this mounting technology is easy to implement, but in case of double layer and multi-layer PCB making hole is more difficult.



2) Surface mounted PCB:

In this type of PCB, components are small in size because these components have very small lead or no leads are required for mounting on the board. Here, in this technology, SMD components are directly mounted on the surface of the board and not require to make hole on board.



The **main advantages** of SMT over the older through-hole technique are:

- Smaller components.
- Much higher component density (components per unit area) and many more connections per component.
- Components can be placed on both sides of the circuit board.
- Higher density of connections because holes do not block routing space on inner layers, nor on back-side layers if components are mounted on only one side of the PCB.
- Small errors in component placement are corrected automatically as the surface tension of molten solder pulls components into alignment with solder pads. (On the other hand, through-hole components cannot be slightly misaligned, because once the leads are through the holes, the components are fully aligned and cannot move laterally out of alignment.)
- Better mechanical performance under shock and vibration conditions (partly due to lower mass, and partly due to less cantilevering)
- Lower resistance and inductance at the connection; consequently, fewer unwanted RF signal effects and better and more predictable high-frequency performance.

- Better EMC performance (lower radiated emissions) due to the smaller radiation loop area (because of the smaller package) and the lesser lead inductance.
- Fewer holes need to be drilled. (Drilling PCBs is time-consuming and expensive.)
- Lower initial cost and time of setting up for mass production, using automated equipment.
- Simpler and faster automated assembly. Some placement machines are capable of placing more than 136,000 components per hour.
- Many SMT parts cost less than equivalent through-hole parts.
- A surface mount package is favoured where a low profile package is required or the space available to mount the package is limited. As electronic devices become more complex and available space is reduced, the desirability of a surface mount package increases. Concurrently, as the device complexity increases, the heat generated by operation increases. If the heat is not removed, the temperature of the device rises shortening the operational life. It is therefore highly desirable to develop surface mount packages having high thermal conductivity.

Disadvantages

- SMT may be unsuitable as the sole attachment method for components that are subject to frequent mechanical stress, such as connectors that are used to interface with external devices that are frequently attached and detached.
- SMDs' solder connections may be damaged by potting compounds going through thermal cycling.
- Manual prototype assembly or component-level repair is more difficult and requires skilled operators and more expensive tools, due to the small sizes and lead spacing's of many SMDs. Handling of small SMT components can be difficult, requiring tweezers, unlike nearly all through-hole components. Whereas through-hole components will stay in place (under gravitational force) once inserted and can be mechanically secured prior to soldering by bending out two leads on the solder side of the board, SMDs are easily moved out of place by a touch of a soldering iron. Without developed skill, when manually soldering or desoldering a component, it is easy to accidentally reflow the solder of an adjacent SMT component and unintentionally displace it, something that is almost impossible to do with through-hole components.
- Many types of SMT component packages cannot be installed in sockets, which provide for easy installation or exchange of components to modify a circuit and

easy replacement of failed components. (Virtually all through-hole components can be socketed.)

- SMDs cannot be used directly with plug-in breadboards (a quick snap-and-play prototyping tool), requiring either a custom PCB for every prototype or the mounting of the SMD upon a pin-leaded carrier. For prototyping around a specific SMD component, a less-expensive breakout board may be used. Additionally, stripboard style proto boards can be used, some of which include pads for standard sized SMD components. For prototyping, "dead bug" bread boarding can be used.
- Solder joint dimensions in SMT quickly become much smaller as advances are made toward ultra-fine pitch technology. The reliability of solder joints becomes more of a concern, as less and less solder is allowed for each joint. Voiding is a fault commonly associated with solder joints, especially when reflowing a solder paste in the SMT application. The presence of voids can deteriorate the joint strength and eventually lead to joint failure. SMDs, usually being smaller than equivalent through-hole components, have less surface area for marking, requiring marked part ID codes or component values to be more cryptic and smaller, often requiring magnification to be read, whereas a larger through-hole component could be read and identified by the unaided eye. This is a disadvantage for prototyping, repair, rework, reverse engineering, and possibly for production set-up.

Surface mount components:

Resistors

For 5% precision SMD resistors usually are marked with their resistance values using three digits: two significant digits and a multiplier digit. These are quite often white lettering on a black background, but other coloured backgrounds and lettering can be used. The black or coloured coating is usually only on one face of the device, the sides and other face simply being the uncoated, usually white ceramic substrate. The coated surface, with the resistive element beneath is normally positioned face up when the device is soldered to the board, although they can be seen in rare cases mounted with the uncoated underside face up, whereby the resistance value code is not visible. For 1% precision SMD resistors, the code is used, as three digits would otherwise not convey enough information. This code consists of two digits and a letter: the digits denote the value's position in the E96 sequence, while the letter indicates the multiplier.

Capacitors

Non-electrolytic capacitors are usually unmarked and the only reliable method of determining their value is removal from the circuit and subsequent measurement with a capacitance meter or impedance bridge. The materials used to fabricate the capacitors, such as nickel tantalate, possess different colours and these can give an approximate idea of the capacitance of the component Generally physical size is proportional to capacitance and (squared) voltage for the same dielectric. For example, a 100 nF, 50 V capacitor may come in the same package as a 10 nF, 150 V device. SMD (non-electrolytic) capacitors, which are usually monolithic ceramic capacitors, exhibit the same body color on all four faces not covered by the end caps. SMD electrolytic capacitors, usually tantalum capacitors, and film capacitors are marked like resistors, with two significant figures and a multiplier in units of picofarads or pF, (10^{-12} farad.)

Inductors

Smaller inductance with moderately high current ratings are usually of the ferrite bead type. They are simply a metal conductor looped through a ferrite bead and almost the same as their through-hole versions but possess SMD end caps rather than leads. They appear dark grey and are magnetic, unlike capacitors with a similar dark grey appearance. These ferrite bead type are limited to small values in the nanohenry (nH) range and are often used as power supply rail decouplers or in high frequency parts of a circuit. Larger inductors and transformers may of course be through-hole mounted on the same board. SMT inductors with larger inductance values often have turns of wire or flat strap around the body or embedded in clear epoxy, allowing the wire or strap to be seen. Sometimes a ferrite core is present also. These higher inductance types are often limited to small current ratings, although some of the flat strap types can handle a few amps. As with capacitors, component values and identifiers for smaller inductors are not usually marked on the component itself; if not documented or printed on the PCB, measurement, usually removed from the circuit, is the only way of determining them. Larger inductors, especially wire-wound types in larger footprints, usually have the value printed on the top. For example, "330", which equates to a value of 33 µH.

Discrete semiconductors

Discrete semiconductors, such as diodes and transistors are often marked with a two- or three-symbol code. The same code marked on different packages or on devices from different manufacturers can translate to different devices. Many of these codes, used because the devices are too small to be marked with more traditional numbers used on larger packages, correlate to more familiar traditional part numbers when a correlation list is consulted.

Integrated circuits

Generally, integrated circuit packages are large enough to be imprinted with the complete part number which includes the manufacturer's specific prefix, or a significant segment of the part number and the manufacturer's name or logo. Examples of manufacturers' specific prefixes include the Philips HEF4066 or Motorola MC14066 (a 4066 Quad Analog Switch) and the Fujitsu Electric FA5502 (a 5502M Boost Architecture power factor correction controller).

Lamination

Lamination is the technique/process of manufacturing a material in multiple layers, so that the composite material achieves improved strength, stability, sound insulation, appearance, or other properties from the use of the differing materials, such as plastic. A laminate is a permanently assembled object created using heat, pressure, welding, or gluing.

Types:

There are different lamination processes, depending on the type of materials to be laminated. The materials used in laminates can be the identical or different, depending on the process and the object to be laminated.

An example of the type of laminate using different materials would be the application of a layer of plastic film—the "laminate"—on either side of a sheet of glass—the *laminated* subject. Vehicle windshields are commonly made composites created by laminating a tough plastic film between two layers of glass. This is to prevent shards of glass detaching from the windshield in case it breaks.

Plywood is a common example of a laminate using the same material in each layer combined with epoxy. Glued and laminated dimensional timber is used in the construction industry to make beams (glued laminated timber, or *Glulam*), in sizes larger and stronger than those that can be obtained from single pieces of wood.

Significance of PCB Cleaning after Surface Mount Soldering

The following aspects are able to fully explain the significance of PCB cleaning after surface mount soldering:

• PCB cleaning after surface mount soldering can stop electrical defects from occurring.

Among all the electrical defects, electrical leakage is the most protruding one, which is an essential element reducing long-term reliability of PCB boards. This type of defects is caused mainly by ionic contaminants, organic residues and other adhesion substances left on the surface of circuit boards.

• PCB cleaning after surface mount soldering can eliminate erosive substances. Erosion will damage circuits, leading components or devices to brittleness. Erosive substances can conduct electricity in humid environment, which will further arouse PCB boards to shorts or even failures. Erosive substances elimination is actually excluding negative elements hindering long-term reliability of PCB boards.

• PCB cleaning after surface mount soldering can make board appearance look clear.

PCB boards cleaned after surface mount soldering look clear in appearance, making some defects exposed, convenient for inspection and troubleshooting such as heat damage and lamination.

Etching is used in micro fabrication to chemically remove layers from the surface of a wafer during manufacturing. Etching is a critically important process module, and every wafer undergoes many etching steps before it is complete.

For many etch steps, part of the wafer is protected from the etchant by a "masking" material which resists etching. In some cases, the masking material is a photoresist which has been patterned using photolithography. Other situations require a more durable mask, such as silicon nitride.

Etching media and technology

The two fundamental types of etchants are liquid-phase ("wet") and plasma-phase ("dry"). Each of these exists in several varieties.

Wet etching

The first etching processes used liquid-phase ("wet") etchants. The wafer can be immersed in a bath of etchant, which must be agitated to achieve good process control. For instance, buffered hydrofluoric acid (BHF) is used commonly to etch silicon dioxide over a silicon substrate.

Different specialised etchants can be used to characterise the surface etched.

Wet etchants are usually isotropic, which leads to large bias when etching thick films. They also require the disposal of large amounts of toxic waste. For these reasons, they are seldom used in state-of-the-art processes. However, the photographic developer used for photoresist resembles wet etching.

As an alternative to immersion, single wafer machines use the Bernoulli principle to employ a gas (usually, pure nitrogen) to cushion and protect one side of the wafer while etchant is applied to the other side. It can be done to either the front side or back side. The etch chemistry is dispensed on the top side when in the machine and the bottom side is not affected. This etch method is particularly effective just before "backend" processing (BEOL), where wafers are normally very much thinner after wafer back grinding, and very sensitive to thermal or mechanical stress. Etching a thin layer of even a few micrometres will remove microcracks produced during back grinding resulting in the wafer having dramatically increased strength and flexibility without breaking.

Plasma etching

Modern VLSI processes avoid wet etching, and use *plasma etching* instead. Plasma etchers can operate in several modes by adjusting the parameters of the plasma. Ordinary plasma etching operates between 0.1 and 5 Torr. (This unit of pressure, commonly used in vacuum engineering, equals approximately 133.3 pascals.) The plasma produces energetic free radicals, neutrally charged, that react at the surface of the wafer. Since neutral particles attack the wafer from all angles, this process is isotropic.

Plasma etching can be isotropic, i.e., exhibiting a lateral undercut rate on a patterned surface approximately the same as its downward etch rate, or can be anisotropic, i.e., exhibiting a smaller lateral undercut rate than its downward etch rate. Such anisotropy is maximized in deep reactive ion etching. The use of the term anisotropy for plasma etching should not be conflated with the use of the same term when referring to orientation-dependent etching.

The source gas for the plasma usually contains small molecules rich in chlorine or fluorine. For instance, carbon tetrachloride (CCl_4) etches silicon and aluminium, and trifluoromethane etches silicon dioxide and silicon nitride. A plasma containing oxygen is used to oxidize ("ash") photoresist and facilitate its removal.

Ion milling, or *sputter etching*, uses lower pressures, often as low as 10^{-4} Torr (10 mPa). It bombards the wafer with energetic ions of noble gases, often Ar^{+}, which knock atoms from the substrate by transferring momentum. Because the etching is performed by ions, which approach the wafer approximately from one direction, this process is highly anisotropic. On the other hand, it tends to display poor selectivity. *Reactive-ion etching* (RIE) operates under conditions intermediate between sputter and plasma etching (between 10^{-3} and 10^{-1} Torr). *Deep reactive-ion etching* (DRIE) modifies the RIE technique to produce deep, narrow features.

A typical printed circuit board, or PCB, contains a large number of electronic components. These components are held on the board by solder flux that creates a strong bond between the pins of a component and their corresponding pads on the board. However, the main purpose of this solder is to provide electrical connectivity. Soldering and desoldering is performed to install a component on a PCB or to remove it from the board.

Soldering with Soldering Iron

A soldering iron is the most commonly used tool to solder components on PCBs. Generally, the iron is heated to a temperature of about 420 degrees Celsius, which is enough to quickly melt the solder flux. The component is then positioned on the PCB such that its pins are aligned with their corresponding pads on the board. In the next step, the solder wire is brought into contact with the interface between the first pin and its pad. Briefly touching this wire at the interface with the heated soldering iron tip melts the solder. The molten solder flows on the pad and covers the component pin. After solidifying, it creates a strong bond between the pin and the pad. Since the solidification of the solder happens fairly quickly, within two to three seconds, one can move to the next pin immediately after soldering one.

Reflow Soldering

Reflow soldering is generally used in PCB production environments in which large numbers of SMD components need to be soldered at the same time. SMD stands for surface mount device and refers to electronic components that are much smaller in size than their through-hole counterparts. These components are soldered on the component side of the board and do not require drilling. The heat-oven method of soldering requires a specially designed oven. The SMD components are first placed on the board with a solder flux paste spread over all of its terminals. The paste is sticky enough to keep the components in place until placing the board in the oven. Most reflow ovens operate in four stages. In the first stage, the temperature of the oven is raised slowly, at a rate of about 2 degrees Celsius per second to about 200 degrees Celsius. In the next stage, which lasts for about one to two minutes, the temperature increment rate is significantly lowered. During this stage, the flux starts to react with the lead and the pad to form bonds. The temperature is further raised in the next stage to about 220 degrees Celsius to complete the melting and bonding process. This stage generally takes less than a minute to complete, after which the cooling stage begins. During cooling, the temperature is rapidly decreased to a little above room temperature, which helps in quick solidification of the solder flux.

Desoldering with Copper Braid

Copper braid is commonly used to desolder electronic components. This technique involves melting the solder flux and then allowing the copper braid to absorb it. The braid is placed on the solid solder and gently pressed with a heated soldering iron tip. The tip melts the solder, which is quickly absorbed by the braid. This is an efficient but slow method of desoldering components since each soldered joint must be worked on individually.

Desoldering with Solder Sucker

Solder sucker is basically a small tube connected to a vacuum pump. Its purpose is to suck the molten flux off of pads. A heated soldering iron tip is first placed on the solid solder until it melts. The solder sucker is then placed directly on the molten flux and a button on its side is pushed that quickly sucks the flux.

Desoldering with Heat Gun

Desoldering with a heat gun is generally used to desolder SMD components, though it can also be employed for through-hole components. In this method, the board is placed on a perfectly flat place and a heat gun is pointed directly at the components to be desoldered for a few seconds. This quickly melts the solder and on the pads, loosening the components. They are then immediately lifted with the help of tweezers. The downside of this method is that it is very difficult to use for small, individual components since the heat can melt the solder on nearby pads, which can dislodge components that are not be desoldered. Also, the molten flux can flow to nearby traces and pads, causing electrical shorts. It is therefore very important to keep the board as flat as possible during this process.

What is Solder?

Solder is used to make electrical connections. A soldering iron is used to heat the metal (base material) of the part to be soldered. Solder is then melted onto the metal (due to wetting and capillary action), to create an alloy of the metal and solder at the connection surface.

What are components of solder?

There are many kinds of solder according to the applications and the ingredients. Lead-containg solder (eutectic solder) was commonly used. However lead-free solder (Sn97C) has become general due to environmental consciousness rising. The most well-used lead-free solder is SnAgCu (Sn97C) type, copmosed of 96.5% tin (Sn), 3% silver (Ag), and 0.5% copper (Cu). The melting point of lead-containing solder is approximately 183 degrees. On the other hand, it is higher, from 217 to 219 degees, in lead-free solder.

Wire solder is popular for soldering. It has a structure whose outside is coverd with solder alloy and flux in the center.

What is Flux?

The flux used in wire solder is made by adding chemicals to natural plant resin (such as pine tar). Flux is an important item in soldering work. The flux primarily fulfills the following three roles.

1. It melts before the solder (at about 90°C), and removes any oxides and dirt that are on the surface of the metal (base material) and molten solder.
2. It reduces the surface tension (stickiness) and improves the wetting of the solder.
3. It covers the surface of molten solder to prevent reoxidation.

What is Base Material?

Base material is the metal that is soldered.

Condition of the surface of the base material

The surface of the base material is covered with many obstacles to soldering. A metal surface that looks clean at first glance is actually covered with a lot of fine dust and dirt, as well as a surprisingly large amount of fats and oils. In addition, if the base material is left exposed to the air, it combines with oxygen to form an oxide film.

Shape of the base material

Since the shape of the base material greatly affects solderability and bonding reliability, the following conditions must be satisfied.

1. The base material must be mechanically and securely fixed. Otherwise, it could move, resulting in cold solder joints.
2. The base material must have appropriate clearance.
3. The temperature of the entire joint area must rise the same amount at the same time.
4. The structure of the base material must prevent the solder from flowing to unnecessary locations.
5. The structure of the base material must prevent the flux from splattering to dangerous locations.
6. Parts that are sensitive to heat must be protected.
7. There must be no stress applied to joint areas.

Key Points for Good Soldering

1. The solder should have a natural glossy shine.
2. There should be clearly visible lines.
3. There should be correctly shaped fillets.
4. Contact angle θ should be small.
5. There should be no cracks or pinholes.

Examples of Poor Soldering

1. Too much flux

Symptom

- There is a film of flux that comes off when pulled.

Measures

- Increase the temperature of the base material to a sufficient level.
- Occurs easily when the surface of the base material is dirty.

2. Tunnel solder

Symptom

- There is an interior gap and only partial bonding.

Measure

- Do not allow any movement until the solder has solidified.

3. Solder blob (cold solder joint)

Symptom

- The surface of the solder is rough with no glossy shine.

Measures

- Lower the temperature of the tip of the soldering iron.
- Do not try to fix it with only the soldering iron without first removing the old solder.

4. Excess solder

Symptoms

- No clearly visible line.
- Solder is flowing to unnecessary locations.

Measure

- Reduce the amount of solder supplied.

5. Solder projection

Symptom

- There is a projection.

Measure

- Pull the soldering iron away quickly and sooner after heating.

6. Burned covering, projections

Symptom

- The wire has been melted.

Measure

- Handle the soldering iron with more care.

What are the Methods of Soldering Electronic Components?

Soldering is the process of fixing one or more components as one by one by dissolving and running a solder in the joint is called soldering. The solder metal has a lower melting temperature than the working piece. The soldering process can be applied in electrical and electronic projects, plumbing, etc. The soldering process is done in various electrical and electronics projects to combine the components with the roots of the printed circuit board. The circuit performance and working depend on the perfect soldering, It needs talent and working on the good soldering techniques will help you to make an excellent working circuit. Here this article explains the **methods of soldering** which require Soldering Lead, Soldering Iron, and Flux along with a printed circuit board and layout diagram of the circuit.

Different Methods of Soldering

The methods of the soldering process can be classified into two, namely soft soldering and hard soldering.

Soft Soldering

Soft soldering is a process for fitting very minute compound parts possessing low liquefying temperature, which have been broken during the procedure of soldering is performed at high temperature. In this process, a tin-lead alloy is used as space filler metal. The liquefying temperature of the space filler alloy must not be less than 400oC / 752oF. A gas torch is used as a heat source, for the procedure. Some of the examples of this kind of soldering metals include tin-zinc for bonding aluminum, tin-lead for general usage; zinc-aluminum for aluminum, cadmium-silver for power at high temperature; lead-silver for strength higher than room temperature, weakening confrontation, tin-silver & tin-bismuth for electrical products.

Hard Soldering

In this type of soldering a solid solder unites two elements of metals by spreading out into the holes of the component that are unlocked due to high temperature. The space filler metal grips a higher temperature of more than 450oC/840oF. It comprises of two elements: Silver soldering and Brazing.

Silver Soldering

It is an unsoiled method supportive to fabricate small components, carrying out abnormal maintenance and built-up tools. It makes use of an alloy containing silver as a space-filler metal. Though silver provides a free-running individuality, yet silver soldering is not suggested for space-filling, and thus, different flux is recommended for accurate silver soldering.

Braze Soldering

This type of soldering is a procedure for connecting two terminals of the base metals by forming liquid metallic space filler, which runs by the attraction of a vessel through the joints and cools down to give a solid union through diffusion and atomic magnetism. It produces a very strong joint. It makes use of a brass metal as a space-filler agent.

Required Tools for Soldering

The required tools for soldering include soldering iron, solder flux, soldering paste, etc.

Soldering Iron

Here, soldering iron is the required primary thing, which is used as a heat source for liquefying solder. And 15W to 30W soldering guns are good for the majority of electronics or PCB (printed circuit board) jobs. For soldering heavy components and cable, you require to spend on the iron of advanced wattage approx 40W or a larger solder gun. The major difference between a gun and an iron is that iron seems like a pencil and comprises of a pin-point heat supply for the precise job, whereas a gun is like a gun in shape with a high wattage point excited by running electrical current simple through it.

A soldering iron device is used for soldering electronic components by hands. It sends heat to make softer the solder so that it can sprint into the breaks among two work terminals. Soldering irons are often brought into engaging in recreation for setting up, protect, and incomplete fabrication work in assembling the components.

Solder Flux

Flux is a chemical purifying agent. In soldering metals, flux provides three functions: it eliminates rust from the components to be soldered; it closes air out as a result ending extra rust, and by making easy mix improves dripping individuality of the fluid solder.

Soldering Paste

Soldering cream is employed to connect the leads of included chip packages to connection ends in the circuit blueprint on a PCB.

Step by Step Soldering Process

The fundamental step by step procedure of soldering is executed by the following steps

Step by Step Soldering Process

- Start with the small components to the taller components and connecting wires
- Place the element into the PCB, making sure it goes in the correct way around
- Twist the leads a little to secure the part.
- Make sure the soldering iron has warmed up and if required, use the moist sponge to clean the tip.
- Place the soldering iron on the component of the pad and feed the solder's end onto the board
- Take away the solder and the soldering iron from the board.
- Leave the terminal to cool for a few seconds.
- Using a couple of cutters neat the excess component terminal
- If you make a mistake while heat up the joint with the iron, place the solder tip of your solder extractor and push the button.

Soldering Tips

Use Heat Sinks: Heat sinks are essential for the connecting wires of sensitive apparatus namely transistors and integrated circuits. If you do not have a clip-on this, then a pair of pliers is a superb choice.
Clean the Iron Tip Neat: A clean iron tip indicates the conductivity of improved heat and also a better joint. Make use of a wet piece of sponge to clean the tip among joints. Keep the tip of solder well tinned.
Check the Joints: When complex circuits are being collected it is an excellent practice to confirm joints after soldering them.
Solder Tiny Parts Initially: Solder jumper terminals, diodes, resistors and all other small parts previous to moving ahead to connect bigger parts such as capacitors and transistors. This makes assembling much easier.
Connect Sensitive Components at the End: Put in CMOS, MOSFETs, ICs and other inactive sensitive parts at the end to avoid damaging them while connecting the other components.

Use Sufficient Ventilation: Avoid breathing the smoke formed and make sure that the region you are operating in has plenty of ventilation to put a stop to increase of toxic smoke.

Thus, this is all about types of soldering, required tools and tricks and tips.

Tips on how to properly design/layout a Printed Circuit Board (PCB)

- General tips
  - Make important nodes accessible
  - Give space between components
  - Place components with the same orientation
  - Print the layout to see if components' sizes match
  - Exchange wiring directions between layers
  - Select the width of lines depending on current
  - Know the specifications of the manufacturer
  - Avoid 90º angles with traces
  - Use the silk layer
  - Use the schematic vs. layout comparison
  - Create a ground plane
  - Place bypass capacitors
  - Route the differential signal traces in parallel
  - Consider spots of heat
  - Make parallel footprints for hard to find components
- Tips for power circuits
  - Keep power and control grounds separate
  - Use an inner layer for the control ground
  - Make power traces wider to withstand higher currents
- Tips for mixed-signal circuits
  - Keep digital and analog grounds separate
  - Protect analog grounds from noise
- Tips for PCB mounting
  - Solder from small to large components
  - Be aware of cold solder joints
  - Use flux for easy soldering
  - Do not trust the multimeter continuity test

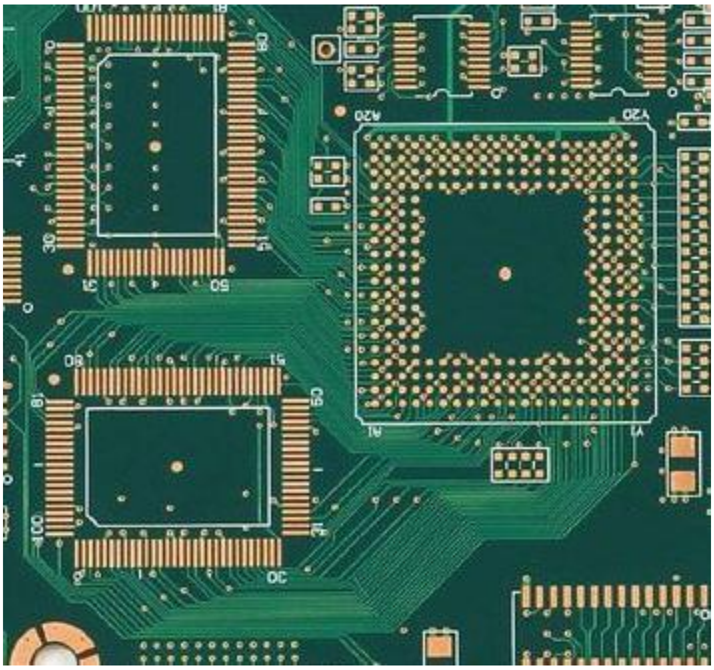
General tips

Make important nodes accessible

It will eventually happen that you are trying to figure out why something is not working and you want to measure a signal inside your PCB. Before designing the PCB, you should think which points will be important to troubleshoot your circuit and, in case they are not easily accessible, add a test point somewhere connected to them. There are various forms of test points, but the ones that form a loop are great for test probes with hooks.

Give space between components

It is tempting to pack the components as close as possible, only to realize that there is no room for the routing of wires. Give some space between components so that wires can spread. The more pins the component has, the more space it will need. Spacing will not only facilitate auto-routing as it will make soldering easier.



Place components with the same orientation

Components generally have a standard pin numbering, with pin #1 in the upper-left corner. If all components are oriented equally, you will not make mistakes when soldering or when inspecting a component.

Print the layout to see if components' sizes match

After laying out all the components, print out the layout. Place each component on top of the layout paper to see if they match. Sometimes datasheets may have errors.

Exchange wiring directions between layers

Draw vertical traces on one side and horizontal traces on the other. This facilitates wiring of lines that have to cross over the others. For multiple layers, alternate between directions.

Select the width of lines depending on current

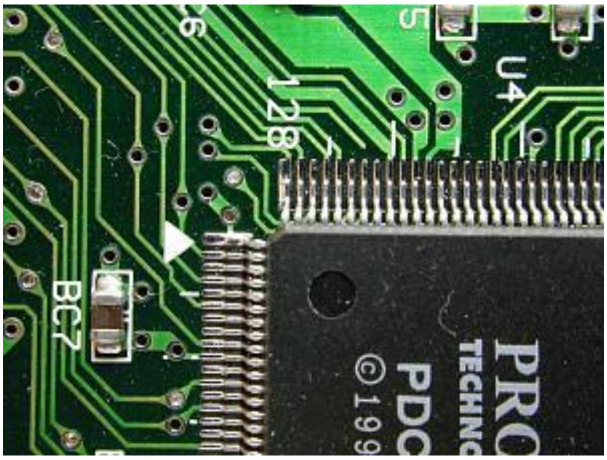
Larger width reduces resistance, which in turn reduces the heat caused by dissipation. The width of the lines should be sized according to the estimated current that flows through them. You can use this online calculator to calculate their width. Therefore, power lines should be wider because all the current is supplied by these wires.

Know the specifications of the manufacturer

Each manufacturer has its own specifications, such as minimum trace width, spacing, number of layers, etc. Before starting design, you should consider what you need and find a manufacturer that meets your requirements. Your requirements also include the grade of materials of the PCB. There are grades ranging from FR-1 (paper-phenolic mixture) up to FR-5 (glass cloth and epoxy). Most PCB prototyping manufacturers use the FR-4, but FR–2 is used in high-volume consumer applications. The type of material affects the circuit board's strength, durability, moisture absorption and Flame Resistance (FR).

Avoid 90º angles with traces

Sharp right angle turns are difficult to keep the trace width constant. This is a reason of concern for narrow traces, where a small difference makes a significant fraction of the trace. A better approach is to do two 45º bends.



Use the silk layer

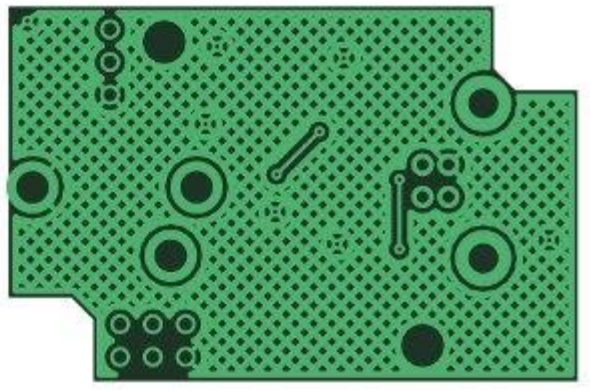
This layer is pretty standard in professional PCB manufacturers and it is extremely useful for labeling. Label your components (the PCB layout software usually does this) and add some information regarding what the board is about, a revision number, and the author/owner.

Use the schematic vs. layout comparison

Many PCB layout softwares have a comparison tool between schematic and layout. Use it to guarantee that your layout is matching the schematic.

Create a ground plane

Especially in analog circuits, it is important that "ground" means the same voltage throughout the PCB. If you use traces to route the ground signal, their resistance will create voltage drops that will make different "grounds" in the PCB have different potentials. To avoid that, you should create a ground plane, i.e., a large area of copper (or even better, reserve a layer for the plane) where the components connecting to ground can do it directly through vias. The ground plane can be completely filled with copper (better for heat dissipation) or in a square grid like the picture below.

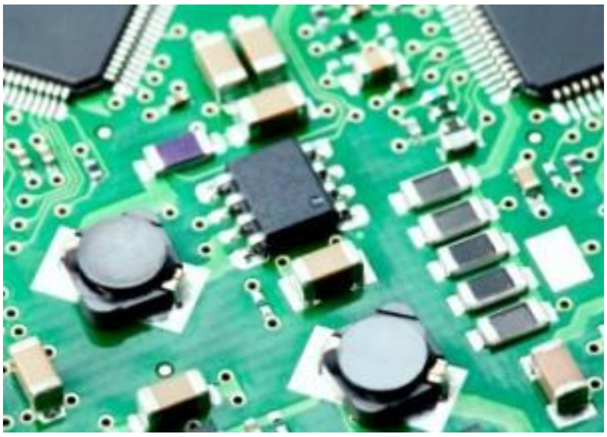


One of the downsides of a plane is the difficulty to solder a component, since the heat gets dissipated quickly through the plane. To avoid this, the contacts to planes can be made through thin traces, like the picture below.

Place bypass capacitors

Bypass capacitors are used to filter AC components from your constant power supply. They reduce noise, ripples and other unwanted AC signals. They do so by bypassing these AC fluctuations to ground, which gives them the name. Therefore, they are usually connected between wherever voltage we want to filter (supply voltage, reference signals, etc.) and ground.

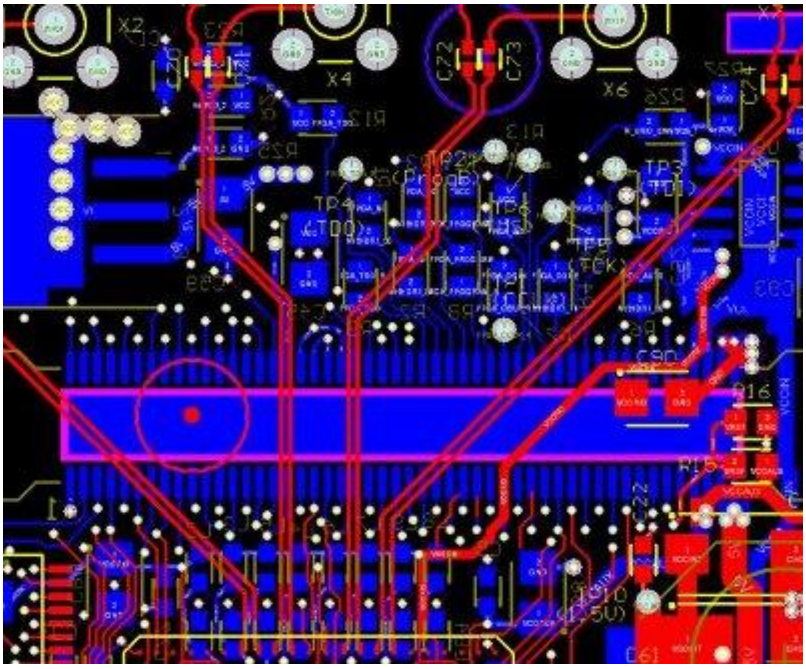
A good place to choose for these capacitors is at the power inlet to your PCB: the wires connecting the power supply to your PCB are usually long and act as antennas, collecting lots of RF signals. Another effective place is close to the ICs (as close as possible to the power and ground pins), to reduce any noise added inside your PCB. The same holds true for reference pins, or any other pin where you need a very stable voltage.



The values of the capacitors depend on the frequencies of the AC components. Each capacitor has its own frequency response determined by its resistance and Equivalent Series Inductance (ESL) that is tuned to a range of frequencies. For example, to filter low frequencies you need a larger capacitor. As a rule, a capacitor of 0.1-1µF suffices for the mid-range frequencies, if you have slow fluctuations, you may choose around 1-10µF and for high-frequency noise you can use 0.001-0.1µF capacitors. You can also use any combination of bypass capacitors to remove a wider range of frequencies. For chips that drive a lot of current, you may put 10 µF - 100 µF capacitors to work as buffers. If the value of the capacitor allows, use monolithic ceramic capacitors because they are small and cheap.

Route the differential signal traces in parallel

Differential signals are often used to improve immunity to noise and amplify the dynamic range. This is only effective if the traces of both signals follow similar paths, so that the noise disturbs both paths equally. To that end, the two lines of a differential signal should be made parallel to each other and as close as possible.



Consider spots of heat

Heat can degrade performance of circuits and even damage them, if not well dissipated. Consider which components consume more power and how the heat produced is being diverted by the package. The datasheet has a parameter called "Thermal Resistance" that states how much temperature increases per Watt of power given certain conditions. The conditions can be for example with a copper area of x by y mm underneath the IC. For stronger heat dissipation, you should add heat sinks or even a fan to cool down the IC. Furthermore, keep critical parts of the circuit board isolated from these heat sources.



Do not trust the multimeter continuity test

A normal multimeter has a continuity test mode that beeps when the resistance between the two probes is small. This is commonly used to test continuities, because you don't need to jump between looking at where you put the probes and at the value on the multimeter. However, a resistance of a few ohms still beeps, although it is a sign of bad soldering. Assuming low resistance lines and a few solder points, the resistance should be below 1 ohm. Use the resistance measurement mode to guarantee that the resistance between two points is as low as it should be, which also means good solder joints.

1.PCB is---------- a laminated and non-conductive **Printed Circuit Board**

a.Zinc

b.Aluminum

c.**Copper**

d.Iron

2. Which is used for printing line, text or any art on the surface of PCB?

a.**Silk layer**

b. Top bottom layer

c. Bottom layer

d. none of the above

3**.** In FR4_FR stand for

FUSE RESISTANCE

FUSE RETARDENT

FIRE RETARDENT

FIRE RESISTANCE

4. Which material is used in flexible PCB?

Cem-1

Polyimide

Cem-1

FR-4

5. Classification of PCB according to the different types of resin

a. Epoxy

b. Composite

b. Polyester

d. **a & c**

6. A measure of signal or power loss of a PCB material

a) **Dissipation Factor**

b) Combustibility
c) Peel Strength
d) none of the above

7. CEM-1 material is made from paper and two layer of woven glass epoxy and phenol compounds and this material is used for----------

Single sided PCB only

8. Pad is nothing but a piece of --------on which lead of components are mounted and on which soldering are done
Copper

9. The ability of copper layers and dielectrics to remain bonded under thermal and
Peel Strength

10. Which material is made from paper and phenol compounds?

FR1 and FR2

11. PCBs used in low cost and bulk manufacturing application like calculators, radio, printers and the solid-state drive are----

Single layer PCB

12. Which type of PCB board is reduced size which makes circuit compact
Double layer PCB

13. PCB is used in large application like GPS technology, satellite system is------
Multilayer PCB

14. In single layer PCB, this mounting technology is----
easy to implement

15. SMDs' solder connections may be -----------by potting compounds going through thermal cycling
Damaged

16. Smaller inductance with moderately high current ratings are usually of the-------- type

ferrite bead

17. SMD electrolytic capacitors, usually---------- capacitors

Tantalum

18. In inductors ferrite bead type are limited to small values in

nanohenry (nH)

19. In case of double layer and multi-layer PCB making hole is more

Difficult

20. Which of the following is not the advantage of SMT?

Components can be placed on both sides of the circuit board

21. Silk Screens

a. Are not required for every component

b. Are highly recommended for prototype boards

c. Are not necessary for production boards

d. All of the above

22. The solder mask impacts the characteristic impedance of a trace.

a. True

b. False

23. The grid used in a PCB layout tool should be

a. In metric (mm)

b. In imperial (mils)

c. Both A and B interchangeably

d. Either A or B

24. The solder paste files are generally

a. Produced in the assembly file

b. Produced in the fabrication files

c. Manually drawn

d. Part of the bill of materials

Reason: The assembly file has nothing to do with the solder paste. It is a product of the fabrication file generation.

25. Though schematic tools do have to use a grid for pin connectivity, in theory, the grid used in a schematic is

a. In metric (mm)

b. In imperial (mils)

c. Both A and B interchangeably

d. Dimensionless

Reason: Schematics, by their nature are dimensionless.

26. Etching efficiency increases with agitation and decreased temperature

True

False

27. Can PCBs be ordered with top and bottom silk screen?
Yes

No

28. Circuit designers need ________ circuits
a) tighter
b) smaller layout
c) decreased silicon area
d) **all of the above mentioned**

29. Design rules does not specify
a) line widths
b) separations
c) extensions
d) **colours**

What is the use of a decoupling capacitor?
A decoupling capacitor is used to smoothen the power supply noise. It should be placed as close to the ICs for which it is intended as possible.

30. For small quantity orders, can you produce prototype PCBs?
Yes. Bittele has the capability to produce PCBs in any amount. But, the greater the quantity, the greater the cost savings.

31. PCB Artwork has its roots in the advanced technological fields to simple ------------

DIY techniques

32. Upon completing the PCB electrical Artwork design, you need to convert your design into--
----------to have a digitalized version of your creation

Gerber files

33. The most common software involved in PCB design manufacturing are

KiCad, OrCAD, and Altium Designer

34**.** Moving on photo films of the PCB are created using **Laser** scanners to deliver an advanced
and detailed film of the PCB artwork design

35. What is PCB printing using screen printing?

Screen printing techniques actually the process that patterns the metal conductor to
form the circuit.

36. What is lamination?

Ans. Lamination: Some PCBs have trace layers inside the PCB and are called *multi-layer* PCBs. These are formed by bonding together separately etched thin boards.

37. 7What is Photo engraving?

Photoengraving uses a photo mask and developer to selectively

remove a photoresist coating. The remaining photoresist protects the copper foil. Subsequent etching removes the unwanted copper. The photomask is usually prepared with aphotoplotter from data produced by a technician using CAM, or computer-aided manufacturing software.

38. What is Silk screen?

Silk screen printing uses etch-resistant inks to protect the copper foil. Subsequent etching removes the unwanted copper. Alternatively, the ink may be conductive, printed on a blank (non- conductive) board. The latter technique is also used in the manufacture of hybrid circuits.

39. What do you mean by Etching of the PCB?

The final copper pattern is formed by selective removal of the unwanted copper which is not protected by an electric resist. $FeCl_3$ solution is popularly used etching solution. $FeCl_3$ powder will remove the copper from the unprotected part of the PCB. After removing the PCB it is dried for some time.

40. What do you mean by Drilling of PCB?

After etching of the PCB the next step is to drill the PCB for the interconnection of the various components on the PCB. The drill hole is having a diameter of generally one mm but the resistance sometimes require 1.5mm diameter. The drilling of the PCB is very important in terms of the working of the PCB hence the drilling is done by drilling machine of large precision and accuracy.

41. ------------is a process for fitting very minute compound parts possessing low liquefying temperature, which have been broken during the procedure of soldering is performed at high temperature

Soft soldering

42. What is the type of soldering in which a solid solder unites two elements of metals by spreading out into the holes of the component that are unlocked due to high temperature

Hard soldering

43**. ------------** produces a very strong joint.

Braze Soldering

44. **------------**soldering guns are good for the majority of electronics or PCB (printed circuit board) jobs

15W to 30W

45. What is Base Material?

Base material is the metal that is soldered.

46. Which type of solderability testing is carried out for the generation of solder sample due to immersion of wire or sheet metal specimen in a bath of molten solder?

a. Solder Bath Testing
b. Meniscus Rise Testing
c. Solder Iron Testing
d. None of the above

47. Which among the below stated soldering methods is also renowned as 'High Frequency Resistance Soldering'?

a. Iron Soldering
b. Furnace Soldering
c. Torch Soldering
d. Electrical Soldering

48. Which among the below mentioned approaches belongs to the category of In-circuit Testing?
a. Impedance Testing
b. Component Testing
c. Apply Signal and check output
d. All of the above

49. What material is used for Chemical etching?

Chemical etching is done with ferric chloride, ammonium persulfate, or sometimes hydrochloric acid

50. What is Solder resist?

Areas that should not be soldered may be covered with a polymer *solder resist*.