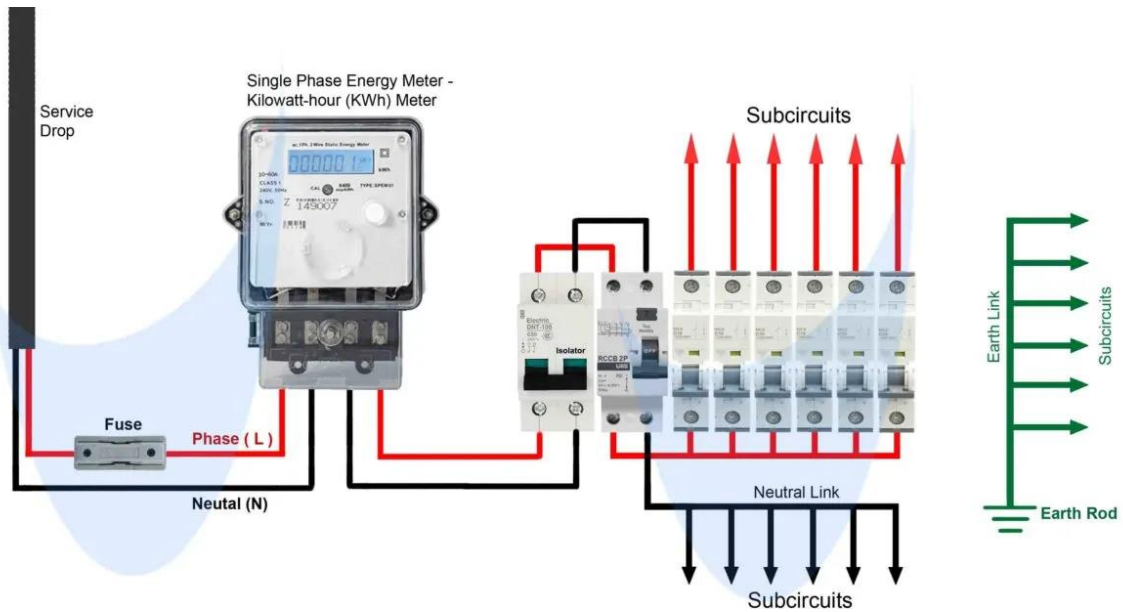


# Basic Skills on Electronic Equipments

## SEC0200903

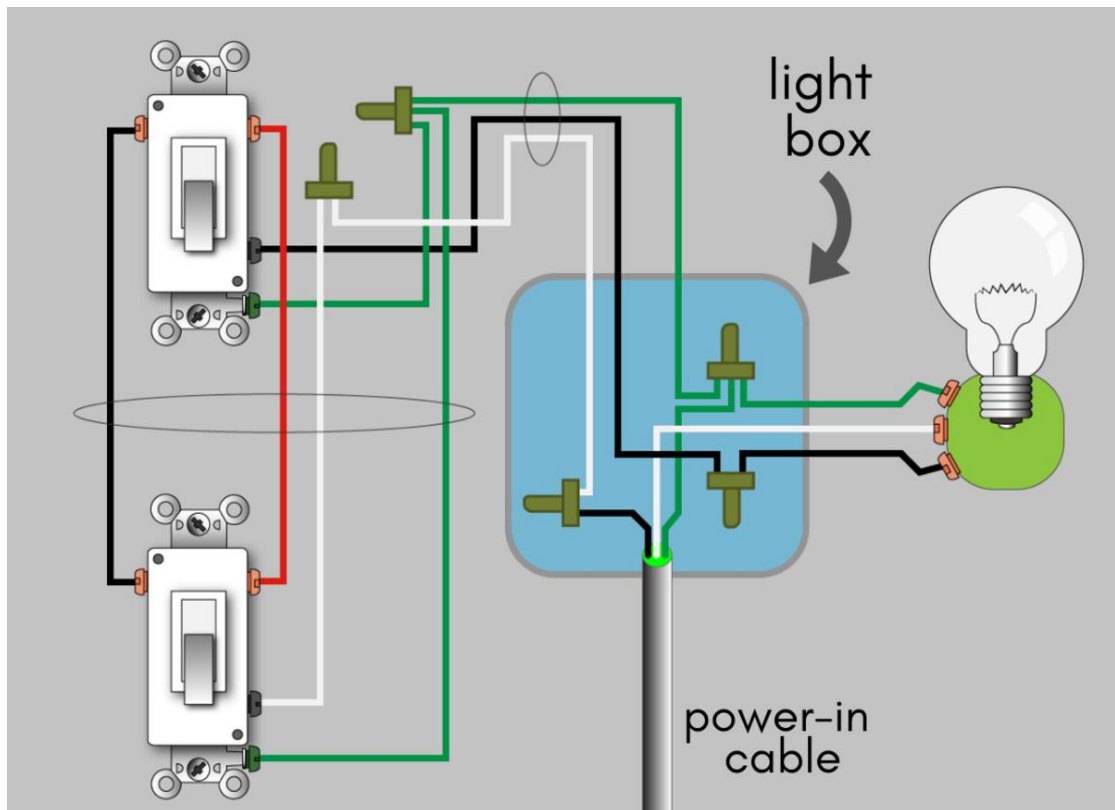
UNIT 5: Electrical switch board, Power supply and PCB

# Circuit design for electrical switch board

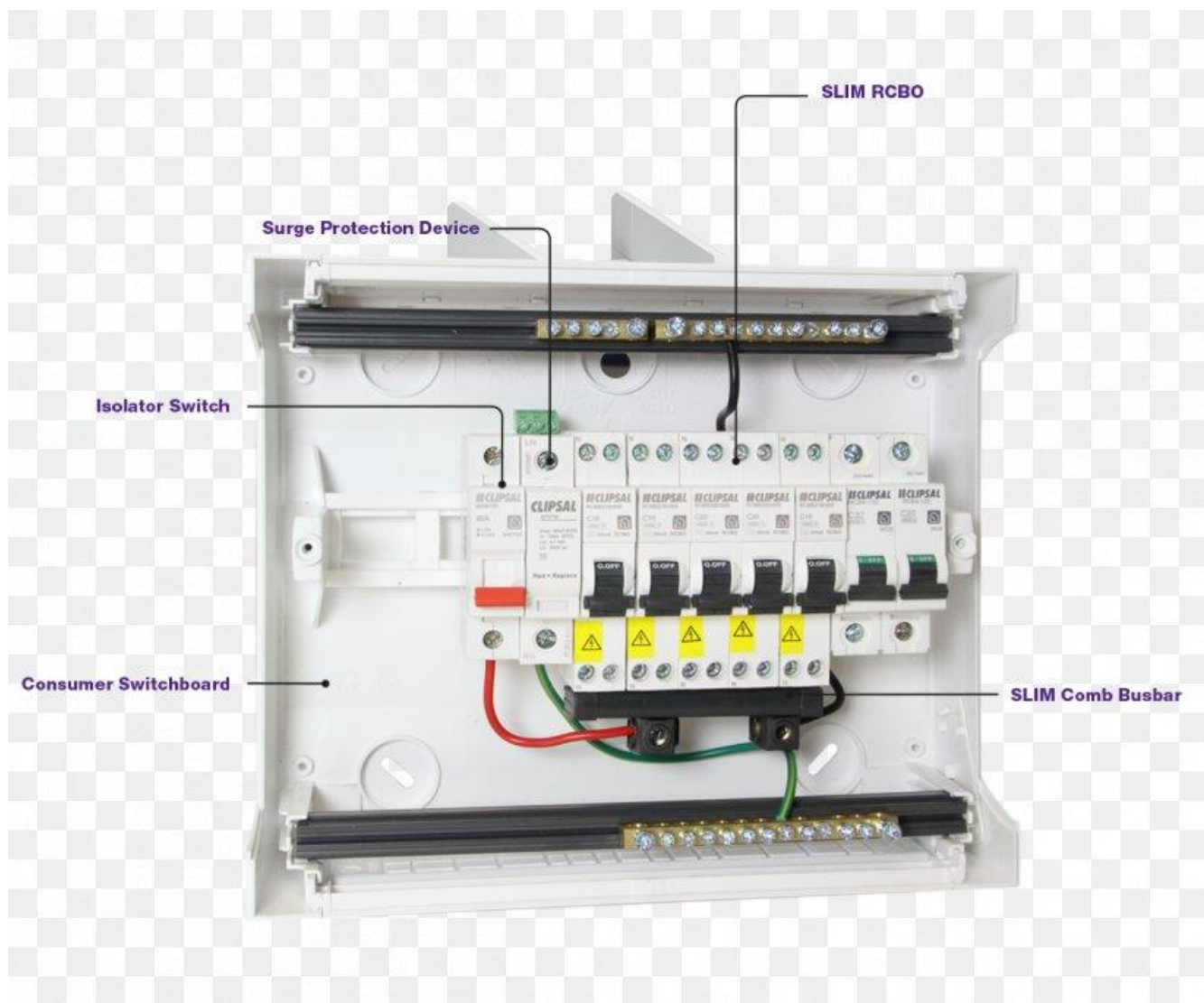


[www.mechatrofice.com](http://www.mechatrofice.com)

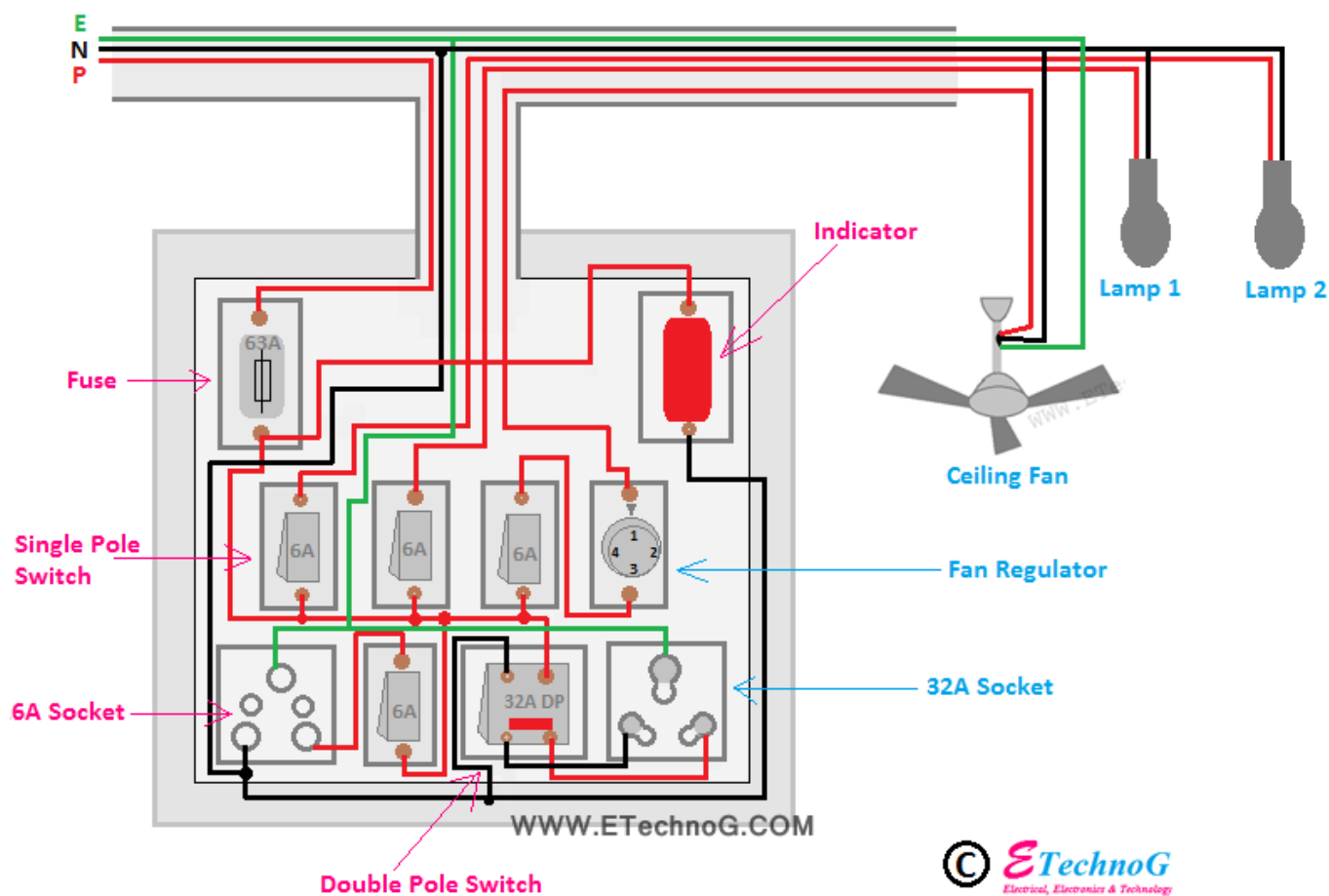
Distribution Board Wiring



3 Way Switch Wiring Diagram



Electric Switchboard Circuit Breaker Wiring Diagram



# Circuit design and principle of regulated power supply (AC to DC)

We have several electrical and electronic circuits that require a DC power supply to operate. But, due to some technical and economical limitations, we cannot use conventional sources of DC power such as batteries and DC generators. In this situation, we use an electronic circuit that can convert AC power into DC power, such a circuit is known as **rectifier**.

However, the DC power output of a rectifier is a fluctuating DC voltage and current which may damage the electronic circuit. Thus, to overcome the problem of variation in voltage, some voltage regulating devices are used so that we can obtain a steady DC power supply.

In this article, we will discuss the definition, circuit diagram, working, and applications of the **regulated power supply**.

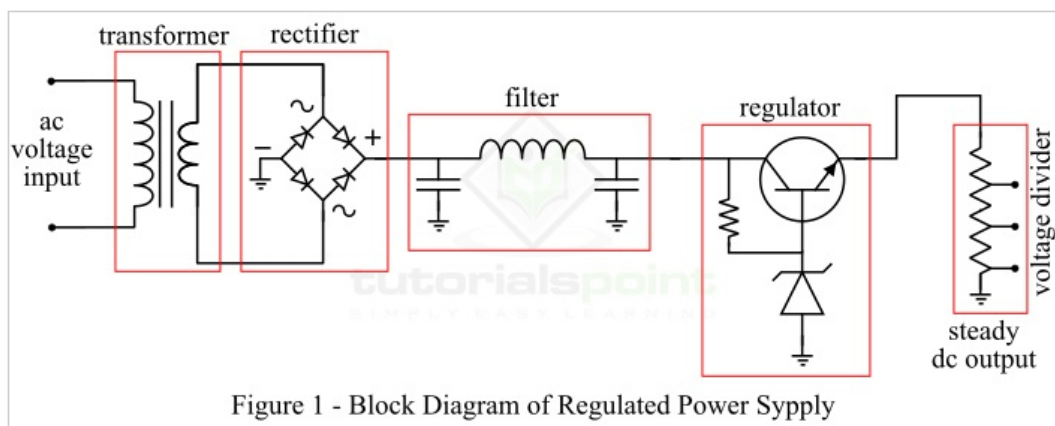
## What is Regulated Power Supply?

An electronic circuit that produces a stable DC voltage of fixed value across the load terminals irrespective of changes in the load is known as **regulated power supply**. Thus, the primary function of a regulated power supply is to convert an AC power into a steady DC power. The regulated power supply is sometimes also called as a **linear power supply**.

The regulated power supply ensures that the output power at the load terminals should remain constant even if the input power varies. The regulated power supply receives an AC power as input and generates a constant DC power as output. A regulated power supply is basically an embedded circuit consisting of various blocks.

## Block Diagram and Operation of Regulated Power Supply

The block diagram of a typical regulated power supply is shown in Figure-1.



The regulated power supply consists of the following four major parts –

- Step-down transformer
- Rectifier circuit
- DC filter circuit
- Voltage regulator

Now, let us discuss the operation of each block of the regulated power supply one by one.

## Function of Step-Down Transformer

The step-down transformer used in the circuit of the regulated power supply changes the input AC voltage to the desired lower voltage value. Also, this transformer provides an electrical isolation between two circuits. The reduced output AC voltage of the step-down transformer is used as the input to the rectifier circuit.

## Function of Rectifier Circuit

The rectifier circuit is used to convert the input AC voltage into a DC voltage. It consists of diodes that perform the rectification process, i.e. conversion of the AC voltage into the DC voltage. However, the output of the rectifier is a pulsating direct voltage. In practice, a full wave rectifier is used for the rectification due to its technical advantages. This full wave rectifier can be a center-tapped full-wave rectifier or a bridge rectifier. The full wave rectifier converts both positive and negative cycles of AC voltage into DC voltage.

## Function of Filter Circuit

Since the output of the rectifier is a pulsating direct voltage which has very high ripple content. Hence, the raw output of the rectifier is undesirable. In order to get a pure ripple free direct voltage, a DC filter circuit is used. We have different types of filter circuits such as capacitor filter choke input filter,  $\pi$ -filter, and LC filter. Therefore, the filter circuit converts the pulsating direct voltage into the constant direct voltage having almost zero ripple content.

## Voltage Regulator

The voltage regulator constitutes the last block of the regulated power supply. It monitors and corrects the fluctuations in the output voltage of the power supply. The output voltage may change or fluctuate due to any change in the input AC voltage or the change in the load or change in any physical parameters such as temperature of the circuit. Thus, the voltage regulator takes care of this problem. The voltage regulator maintains the DC voltage constant at the output terminals.

A zener diode operated in zener region, a transistor series regulator, fixed and variable IC regulators are commonly used in different regulated power supplies as the voltage regulator.

## Features of Regulated Power Supply

Following are the main features of the regulated power supply –

- The regulated power supplies have the efficiency ranging from 20% to 25%.
- Regulated power supplies are relatively more reliable.
- Regulated power supplies have less complex circuit and less weight.
- Regulated power supplies give faster response.
- The cost and noise level of the regulated power supplies is low.

## Applications of Regulated Power Supplies

As discussed earlier, the regulated power supplies are the embedded circuits that convert an unregulated AC power supply into a steady DC power supply which is the basic requirement of several electronic circuits. Hence, the regulated power supplies are extensively used in several applications such as –

- Mobile charging circuits
- Testing circuits
- Bench power supplies
- Oscillators and amplifiers
- Electronic computers
- Automatic control systems, etc.



# Printed Circuit Board (PCB)



Electronic devices saturate the modern world. Whether it is a device that silently monitors vitals or a smartphone with an endless stream of notifications, all contain a PCB circuit board at the heart of their design. Over the years, printed circuit board manufacturing has continued to grow to keep up with the increasing demands of newer, faster, and more complex electronic circuitry. Discussions on the processes involved in devising and designing a PCB circuit board could fill a library, but here we will provide a surface-level introduction to the basics.

## What Is a Printed Circuit Board?

A printed circuit board is a rigid structure that contains electrical circuitry made up of embedded metal surfaces called traces and larger areas of metal called planes. Components are soldered to the board onto metal pads, which are connected to the board circuitry. This allows components to be interconnected. A board can be composed of one, two, or multiple layers of circuitry.

Circuit boards are built with a dielectric core material with poor electrical conducting properties to ensure pure circuitry transmission and interspaced with extra layers of metal and dielectric as needed. The standard dielectric material used for circuit boards is a flame-resistant composite of woven fiberglass cloth and epoxy resin, known as FR-4, while the metal traces and planes for the circuitry are usually composed of copper.

A printed circuit board carries electrical circuits and allows signals and power to be routed between devices. PCBs consist of several layers of material:

- **Substrate:** At the core of the board is a substrate or base material. Traditional boards use a fiberglass material called FR4, though other boards may use aluminum, ceramic or polymer substrates.
- **Copper:** The substrate of the PCB is covered with a thin copper foil and laminated with heat and adhesive. The number of "layers" in a board is determined by the number of copper layers – for example, double-sided PCBs will have copper applied to both sides of a substrate, and a four-layer board will have four layers of copper total. Most PCBs contain one ounce of copper per square foot, though high-power PCBs may use two or three ounces.

## WHAT IS PCB FABRICATION?

PCB fabrication is the process by which the aforementioned materials of a board are combined into functional layers to form the bare printed circuit board. The **fabrication of bare printed circuit boards** works as follows:

**Design review:** The fabrication process does not start until the engineering team reviews the design. This is called a Design for Manufacturability check, which ensures the design is valid and fulfills the tolerances and specifications can be met during manufacturing.

**Film application:** The PCB design is printed onto films using laser direct imaging, which provide photo negatives of the board layout. These films are applied to the material layers and used as a guide for each manufacturing process.

**Copper etching:** Using various methods, the unwanted or excess copper is etched or removed to create traces and pads. This process usually involves multiple rounds of cleaning to remove excess material and unwanted chemicals.

**Layer stack-up and lamination:** If the board is a multi-layer board, the inner and outer layers are aligned with one another after the copper etching process. This is usually accomplished using alignment holes in the board itself. After inspecting the layers, an epoxy-based prepreg is placed between each layer, and the board is laminated together by applying heat – the heat melts the epoxy and adheres the layers together.

**Drilling:** After the board layers have been laminated together, a drill is used to create holes through the board. These holes are used for mounting holes, through-holes and vias.

**Plating:** Once drilling is complete, the board is plated with a micron-thick layer of copper, which coats the inside of the drilled holes, creating plated through-holes. This is usually followed up with more etching and copper removal on the outer layer of the board.

**Solder mask application:** After the copper layers are complete, the panels are cleaned and covered with solder mask. Solder mask give the PCB its characteristic green color. The solder mask insulates copper traces from accidental contact with other conductive materials that could create functional issues and helps during assembly so that solder is only applied to the correct locations. Once applied, unwanted areas of the solder mask are removed, and the PCB is placed in an oven to cure the solder mask layer.

**Silkscreen:** Whether or not this is needed, the next step is the application of the silkscreen, after which point the board undergoes the final curing stage. On top of the solder mask layer, is where the silkscreen layer is applied. Silkscreen is usually white in color and adds letters, numbers and symbols to the PCB. This silkscreen layer helps assist with component type and location during the assembly of the PCB.

**Surface finishing:** After the solder mask and silkscreen are cured, the PCB may be plated with gold or silver or other surface finish to ensure protection and functionality.

After the boards complete these steps, they typically undergo electrical reliability testing to ensure they are functional. After that, if the boards were produced in a panel, they are cut apart, inspected and repaired as needed.



# HOW FABRICATION FALLS INTO THE OVERALL MANUFACTURING PROCESS

Fabrication is only one step in the PCB manufacturing process. To fully understand the importance of printed circuit board fabrication, it's important to know where it lies in the cycle.

The PCB manufacturing process falls within three general stages, described in more detail below:

**PCB development:** PCB development is the initial stage of the manufacturing process. Essentially, this is where the PCB design is conceptualized, modified and finalized. PCB development usually includes multiple rounds of design, prototyping and testing, especially for more complex designs.

**PCB manufacturing:** After the PCB design is finalized in the development stage, the PCB moves on to the manufacturing process. Manufacturing involves two steps — fabrication and assembly. Fabrication produces the design of the board, while assembly applies components to the board's surface. At the end of this process, the boards are complete.

**PCB evaluation:** The final stage of the manufacturing process is an evaluation. This includes a final round of testing after the manufacturing process is complete. This testing stage evaluates the assembled board's ability to perform as intended and eliminates boards that fail the tests. In some cases, testing may reveal that the board requires a design adjustment, and the cycle starts over. If boards pass the tests, they undergo a final inspection for any defects before being sent out.

## WHY PCB FABRICATION IS IMPORTANT

PCB fabrication is an outsourced activity performed by a third-party manufacturer using the designs provided by their customers. Fabrication is important to understand because manufacturers only see the design, not the performance intended for your design. If designers do not keep the fabrication process and limitations in mind during the design process, fabricated boards may be impacted in the following areas:

- **Manufacturability:** Design choices impact the manufacturability of PCBs. Board designs need to take into account the material properties of the board and allow adequate clearance between surface elements. If the board needs to be produced in panels, this should also impact the final design. Designs that neglect these factors may be impossible to fabricate without redesigns.
- **Yield rate:** In some circumstances, boards can be successfully fabricated using the existing design. However, certain design choices may result in fabrication issues that render multiple boards unusable — for example, stretching the tolerance boundaries of your manufacturing equipment.
- **Reliability:** Most boards are regulated under industry specifications, which dictate a required level of performance reliability. Failing to follow industry requirements in design may result in unreliable or nonfunctional boards.

The best way to avoid these issues is to consider the PCB fabrication process during the design cycle. You can do this by utilizing design for manufacturability (DFM) rules and checking the capabilities of your **PCB manufacturer** during the design cycle. When you partner with a high-quality PCB manufacturer, you can work closely with them to help optimize your design for the PCB fabrication process.

# Types of Printed Circuit Board (PCB)

There are several types of PCB, each with their own particular manufacturing specifications, material types and usages 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. Rigid PCB
5. Flexible PCB
6. Flex-rigid PCB

## 1. Single-layer PCB

Single layer PCBs also known as single sided PCBs are the simple and most used PCBs. These PCBs are easy to design and manufacture.

It is made out of a single layer of base material or substrate.

One side of this PCB is coated with a thin layer of any conducting material.

Generally, copper is used as conducting material for PCB, because of its very good conducting characteristic.

Once the copper base plating is applied, a layer of solder mask is used to protect the 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, which are soldered onto one side.

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

But despite this low cost, they're used infrequently because of their inherent design limitations.



## 2. Double-layer PCB

Double-layer PCB also known as double-sided PCB has a thin layer of conducting material, like copper applied to both top and bottom sides of the board.

Holes drilled through the board allow circuits on one side of the board to connect to circuits on the other.

The circuits and components of a double-layer PCB board are usually connected in one of two ways: either utilizing a **through-hole** or with the use of a **surface-mount**.

A **through-hole** connection means that small wires, known as leads, are fed through the holes, with each end of the leads then soldered to the right component.

**Surface mount** PCBs don't utilize wires as connectors. Instead, many small leads are soldered directly to the board, which means the board itself is used as a wiring surface for the different components.

This allows circuits to be completed using less space, freeing up space to allow the board to complete more functions, usually at higher speeds and a lighter weight than a through-hole board would allow.

Double-sided PCBs are typically used in applications which require an intermediate level of circuit complexity, such as Industrial controls, Power supplies, Instrumentation, HVAC systems, LED lighting, Automotive dashboards, Amplifiers and Vending machines.



Fig.2: Double-layer PCB

## 3. Multi-layer PCB

Multilayer PCBs consist of more than two layers. It means that, this type of PCB has at least three conductive layers of copper.

These boards are then secured together with a specialized glue and sandwiched between the layers of insulation to ensure that excess heat doesn't melt and thereby damage any component of the circuit.

Multi-layer PCBs come in a variety of sizes, going as small as four layers or as large as ten or twelve. The largest multi-layer PCB ever built was 50 layers thick.

Due to design complexity, these type PCBs are used in very complicated and large electrical task in very low space and compact circuit.

Applications where multi-layer PCBs would be beneficial include File servers, Data storage, GPS technology, Satellite systems, Weather analysis and Medical equipment.

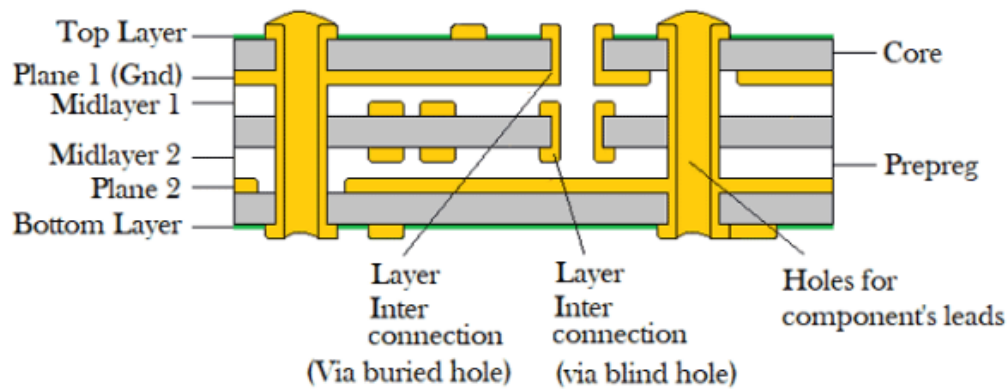


Fig.3: Multi-layer PCB

## 4. Rigid PCB

Rigid PCBs are made out of solid substrate material which does not allow PCBs from twisting. The most common example of a rigid PCB is a computer motherboard. The motherboard is a multilayer PCB designed to allocate electricity from the power supply while simultaneously allowing communication between all of the many parts of the computer, such as CPU, GPU, and RAM.

Rigid PCBs can have different layer configuration like single layer, double layer, and multi-layer Rigid PCBs.

This PCB cannot be bent according to the shape of the base that's why this PCB is known as RIGID PCB.

The lifespan of this type of PCB is very high, so this is used in many parts of the computer like **RAM, GPU, and CPU**.

Due to simplicity in design, these are the most used and most manufacture PCB.

## 5. Flexible PCB

Flexible PCBs are also known as **Flex circuits**.

Unlike rigid PCBs, which use unmoving materials such as fiberglass, flexible PCBs are made of materials that can flex and move, such as plastic.

This type of PCBs use flexible plastic material like polyimide, PEEK (Polyether ether ketone) or transparent conductive polyester film.

The circuit board is generally placed in folded or twisted.

Flexible PCBs offer many advantages over rigid PCBs. The most prominent of these advantages is the fact that they are flexible. This means they can be folded over edges and wrapped around corners.

Their flexibility can lead to cost and weight savings since a single flexible PCB can be used to cover areas that might take multiple rigid PCBs.

Like rigid PCBs, flexible PCBs come in single, double or multilayer formats. As they need to be printed on a flexible material, they tend to cost more for fabrication.

Flex circuits are used in organic light emitting diode, LCD fabrication, flex solar cell, automotive industries, cellular telephones, camera and complex electronics devices like laptops.

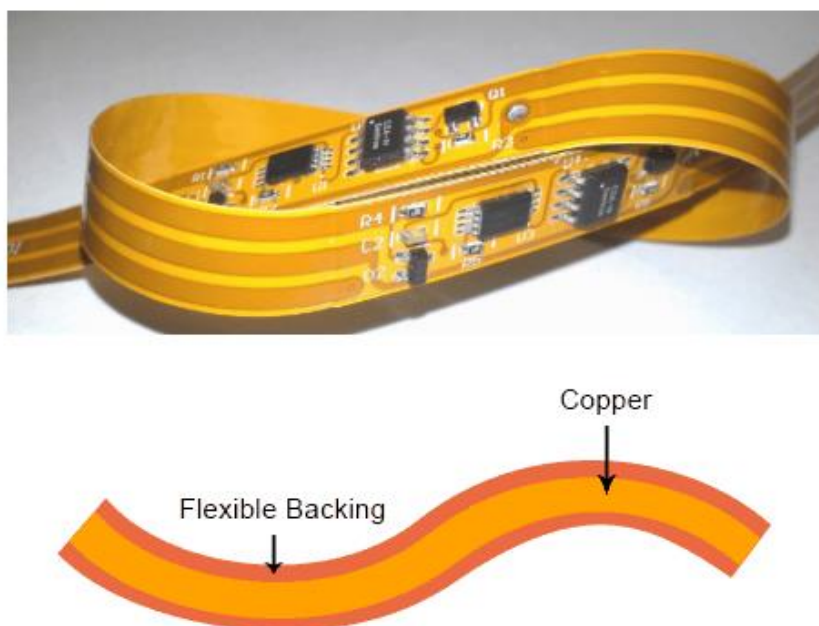


Fig.4: Flexible PCB



## 6. Flex-rigid PCB

Flex-rigid PCBs combine the best of both Flexible PCBs and Rigid PCBs.

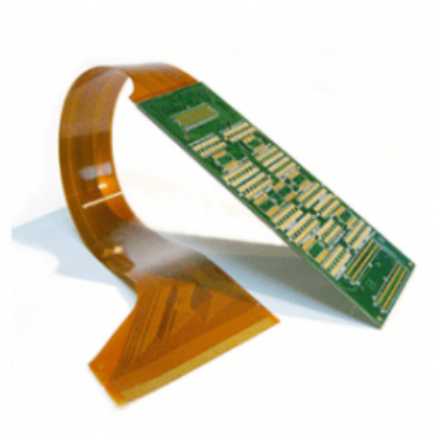
Flex-rigid boards consist of multiple layers of flexible PCBs attached to a number of rigid PCB layers.

Flex-rigid PCBs have many advantages over just using rigid or flexible PCBs for certain applications.

For example, rigid-flex boards have a lower parts count than traditional rigid or flexible boards because the wiring options for both can be combined into a single board.

The combination of rigid and flexible boards into a single rigid-flex board also allows for a more streamlined design, reducing the overall board size and package weight.

Flex-rigid PCBs are most often found in applications where space or weight are prime concerns, including Cell phones, Digital cameras, Pacemakers, and Automobiles.



# Steps involved in development of PCB using FeCl<sub>3</sub> solution.

There are in all three basic methods to make a PCB:

1. Iron on Glossy paper method.
2. Circuit by hand on PCB.
3. Laser cutting edge etching.

Since laser method is the industrial method to make PCBs, we will go into detail for the first two methods.

## PCB Design:

**PCB design** is usually done by converting your circuit's schematic diagram into a PCB layout using PCB layout software. There are many cool open source software packages for PCB layout creation and design.

Some are listed here to give you a head-start:

1. [Autodesk Eagle](#)
2. [PCBWizard](#)

## Designing a Circuit Schematic in Autodesk Eagle:

In Eagle: File> Export>ImageBe sure to set DPIG to 1200 for better quality

## What Materials are Required to Make a Circuit Board?

Required materials	
	<b>Magazines or advertising brochures</b> (More on this later).
	<b>Laser printer</b> Alternately, a photocopier should work
	<b>Household clothes iron</b>
	<b>Copper clad laminate</b>
	<b>Etching solution</b>
	<b>Kitchen scrubs</b>
	<b>Thinner (e.g. acetone)</b>
	<b>Plastic coated wire</b>

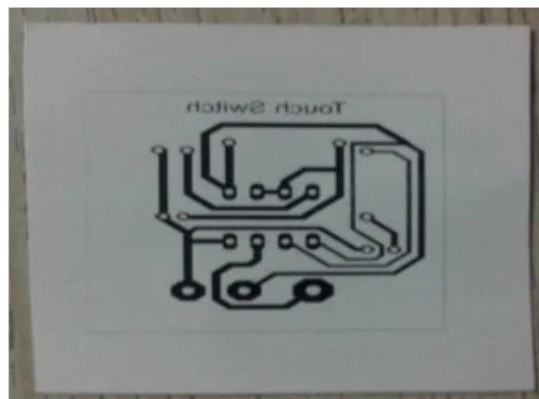
You also need:  $\text{FeCl}_3$  powder/solution (same as etching solution), photo/glossy paper, a permanent black marker, a blade cutter, sandpaper, kitchen paper, and cotton wool.

For this tutorial, let's consider making a PCB for a simple project- a Touch Switch using IC555.

### **STEP 1: Take a printout of circuit board layout**

Take a print out of your PCB layout using a laser printer and the A4 photo paper/glossy paper. Keep in mind the following points:

- You should take the mirror print out.
- Select the output in black both from the PCB design software and the printer driver settings.
- Make sure that the printout is made on the glossy side of the paper.



### **STEP 2: Cut the Copper Plate for the Circuit Board**

Cut the copper board according to the size of the layout using a hacksaw or a cutter.



Copper clad plate



Cutting the plate

Next, rub the copper side of the PCB using steel wool or abrasive sponge scrubs. This removes the top oxide layer of copper as well as the photoresist layer. Sanded surfaces also allow the image from the paper to stick better.



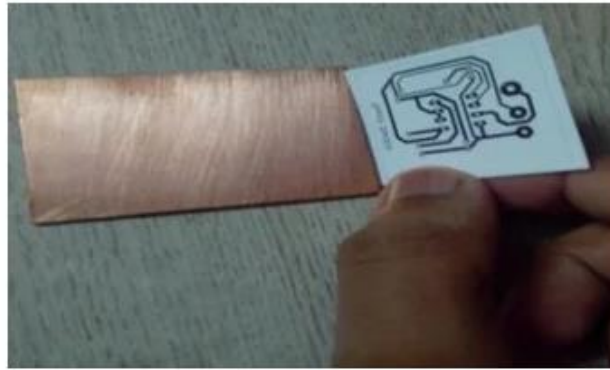
Rubbing away the top oxide layer

### **STEP 3: Transfer the PCB Print onto the Copper Plate**

#### **Method 1: Iron on Glossy Paper Method (For Complex Circuits)**

Transfer the printed image (taken from a laser printer) from the photo paper to the board. Make sure to flip top layer horizontally. Put the copper surface of the board on the printed layout. Ensure that the board is aligned correctly along the borders of the printed layout and use tape to hold the board and the printed paper in the correct position.





Place the printed side of the paper on the plate

### **Method 2: Circuit by Hand on PCB (For Simple and Small Circuits)**

Using the circuit as a reference, draw a basic sketch on the copper plate with a pencil. Once your sketch looks good, trace over it with a permanent black marker.



Using the permanent marker to sketch the PCB

### **STEP 4: Iron the Circuit from the Paper onto the PCB Plate**

- After printing on glossy paper, we iron it image side down to the copper side, then heat up the electric iron to the maximum temperature.
- Put the board and photo paper arrangement on a clean wooden table (covered with a table cloth) with the back of the photo paper facing you.
- Using pliers or a spatula, hold one end and keep it steady. Then put the hot iron on the other end for about 10 seconds. Now, iron the photo paper all along using the tip while applying a little pressure for about 5 to 15 mins.
- Pay attention to the edges of the board – you need to apply pressure and do the ironing slowly.
- Doing a long hard press seems to work better than moving the iron around.
- The heat from the iron transfers the ink printed on the glossy paper to the copper plate.



Iron the paper onto the plate



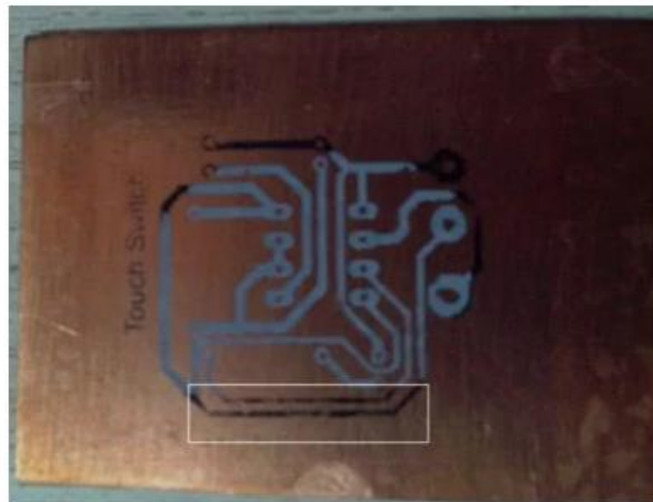
**CAUTION:** Do not directly touch copper plate, IT WILL BE VERY HOT FROM THE IRON.

After ironing, place the printed plate lukewarm water for about 10 minutes. The paper will dissolve, then you can remove the paper gently. Remove the paper by peeling it from a low angle.



Peeling the paper

In some cases, when removing the paper, some of the tracks get faded. In the figure below, you can see that the track is light in color, hence we can use a black marker to darken it as shown.



Light trace



Darkening the trace

## STEP 5: Etch the Plate

You need to be really careful while performing this step.

- First, put on rubber or plastic gloves.
- Place some newspaper on the bottom so the etching solution does not spoil your floor.
- Take a plastic box and fill it up with some water.
- Dissolve 2-3 teaspoons of ferric chloride power in the water.
- Dip the PCB into the etching solution (Ferric chloride solution,  $\text{FeCl}_3$ ) for approximately 30 mins.
- The  $\text{FeCl}_3$  reacts with the unmasked copper and removes the unwanted copper from the PCB.
- This process is called Etching. Use pliers to take out the PCB and check if the entire unmasked area has been etched or not. In case it is not etched, leave it in the solution for some more time.



Etching the plate



Etched copper plate

## STEP 6: Cleaning, Disposal, and the Final Touches for the Circuit Board

Be careful while disposing of the etching solution, it's toxic to fish and other water-based organisms! Don't even think about pouring it in the sink when you are done, it is **ILLEGAL** and might damage your pipes (hehe, who knew you could get arrested while making a PCB?!). Instead, dilute the etching solution and then throw it away somewhere safe.

A few drops of thinner (nail polish remover works well) on a pinch of cotton wool will remove completely the toner/ink on the plate, exposing the copper surface. Rinse carefully and dry with a clean cloth or kitchen paper. Trim to final size and smoothen edges with sandpaper.



Removing the ink