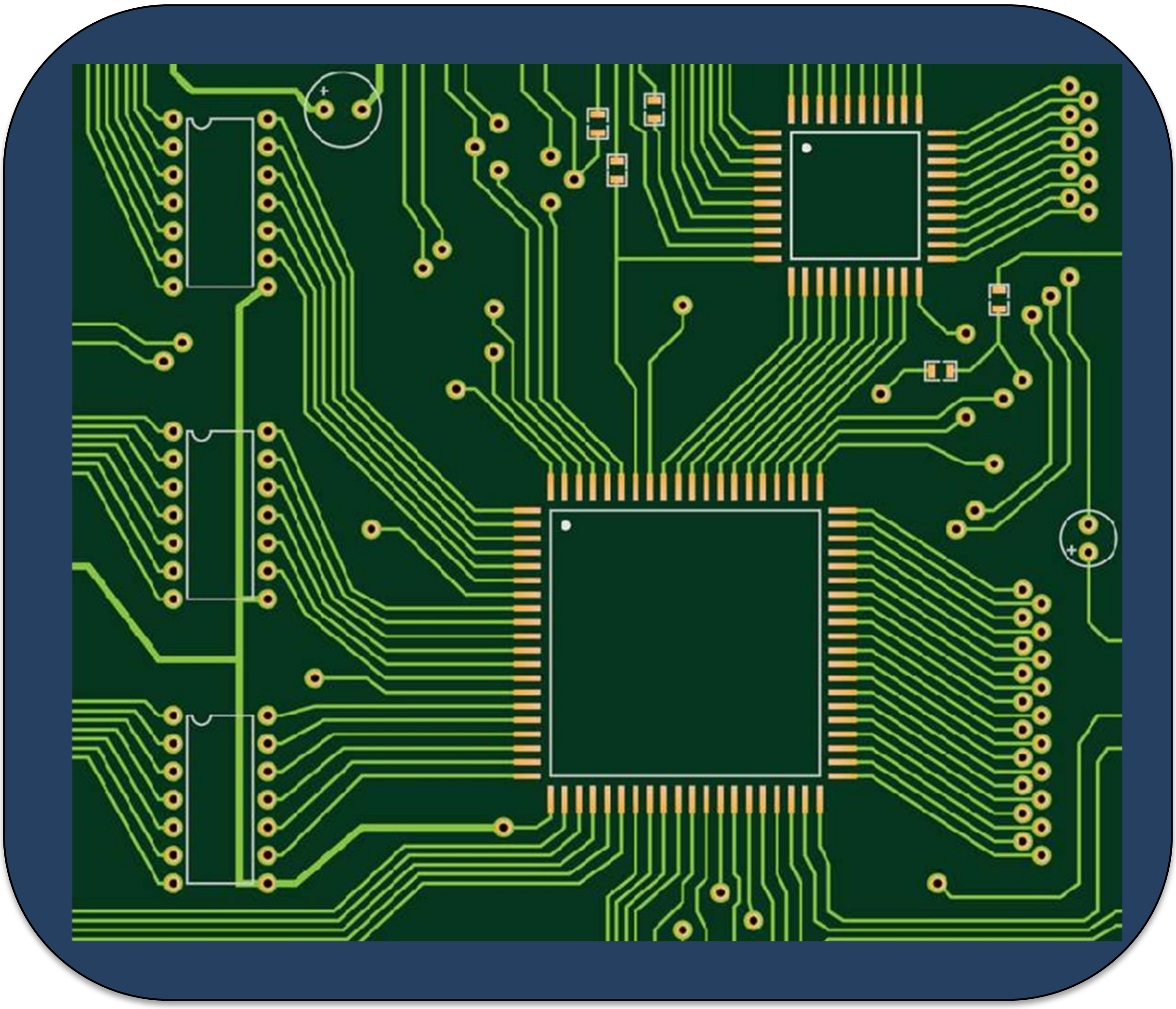
Printed circuit board-

the base of each Electronic Product



Dr. Ratan Sengupta

Printed circuit board - The base of each electronic product

Dr. Ratan Sengupta

BlueRoseONE.com DIY
Stories Matter

First Published by

BLUEROSE PUBLISHERS

www.bluerosepublishers.com

info@bluerosepublishers.com

Foreword

This book has been prepared, based on my 35 + years' experience on conducting training on Electronic PCB Assembly, both in the capacity of a Trainer and Consultant, solving assembly problems.

I have realized that PCB, which is the base of all electronic product manufacturing Industries, do not realize the fact that their system long term Reliability is a direct function of each PCB on which Solder joints are made. Hence, a basic Induction training on PCB as an important component in soldering technology , is absent from their Manufacturing Training Plan.

Through this guide book, I have tried to share my experience with more than 300 Electronic Manufacturing Units in India, with a massage that if you wish to wish to produce Long Term Reliable System to your clients, please do not forget to emphasis on identification and selection of basic Printed Circuit Board(PCB).

I am confident that this book will serve both as basic text book for Diploma in Electronics and Reference book for thousands of electronic assembly operators.

Ratan Sengupta
10-05-2023

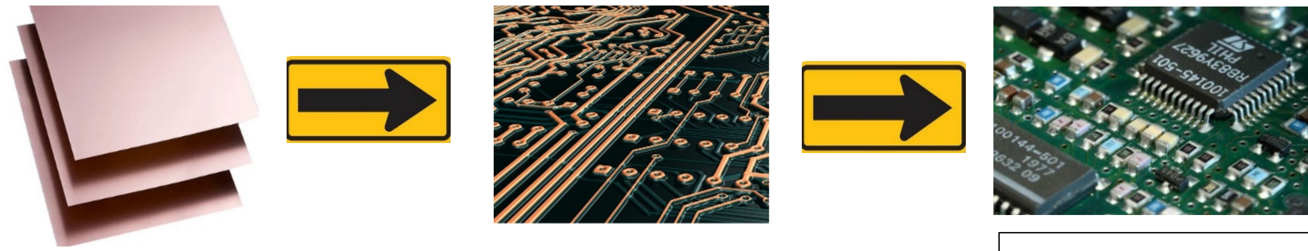
This E-book is dedicated to my wife Ajanta

Acknowledgement

Author is thankful to all references ,from which subject matter has been taken. While preparing a manuscript, the author has consulted various Standards, Articles and Company brochure, some of whose names are depicted in bibliography.

Last but not the least, my sincere thanks goes to our Digital publication executive, Ms. Payal Pansuriya, who has done noticeable effort in making the digital version of this E-book.

Dr. R. Sengupta
10-05-2023



PCB LAMINATE

ETCHED PCB

ASSEMBLED PCB

Content

1. Introduction & History of PCB Innovation
2. Common PCB Substrate
3. Single, Double sided and Multilayer PCB
4. Flexi & Flexi Rigid PCB
5. PCB Manufacturing
6. PCB Manufacturing defects with illustration
7. Bare Board PCB Testing
8. Bare Board PCB Repair Technique
9. PCB Legislation, Environmental Issues
10. Latest trend in PCB Technology
11. Glossary
12. References

Chapter 1:
Definition, Introduction, History
1.1 Definition

A printed circuit board (PCB) is an electronic circuit used in devices to provide mechanical support and a pathway to its electronic components. It is made by combining different sheets of non-conductive material, such as fiberglass or plastic, that easily holds copper circuitry.

Alternately, printed circuit board can be defined as a non-conductive material with conductive lines on it, onto which electronic components can be mounted and connected by traces to form a working circuit or assembly

PCB is also known as printed wiring board (PWB) or etched wiring board (EWB)

Fig.1.1: Printed Wiring Board

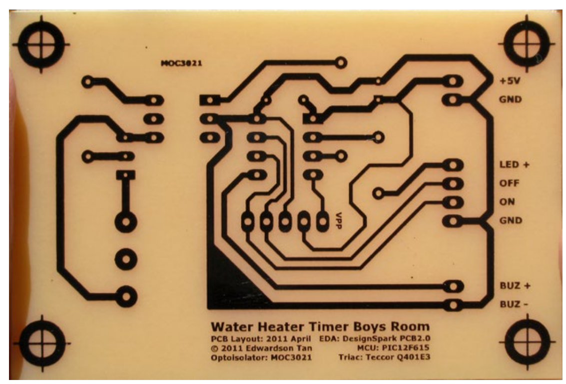


Fig.1.2: Etched Wiring Board

Alternatively, a printed circuit board, or PC board, or PCB, is a non-conductive material with conductive lines printed or etched. Electronic components are mounted on the board and the traces connect the components together to form a working circuit or assembly.

A PC board can have conductors on one side or two sides and can be multi-layer — a sandwich with many layers of conductors, each separated by insulating layers.

- The most common circuit boards are made of plastic or glass-fiber and resin composites and use copper traces, but a wide variety of other materials may be used. Most PCBs are flat and rigid but flexible substrates can allow boards to fit in convoluted spaces. Components are mounted via SMD (surface-mount) or through-hole methods.

A printed circuit board (PCB) is the board base for physically supporting and wiring the surface-mounted and socketed components in most electronics.

In applications where fine conductive traces are needed, such as computers, PCBs are made by a photolithographic process, in a larger scale version of the way conductive paths in processors are made.

Electronic components are typically placed by machine onto a finished PCB that has solder dabs in place. The PCB bakes in an industrial oven to melt the solder, which joins the connections. Most PCBs are made from fiberglass or glass-reinforced plastics with copper traces.

PCBs can be single-layer for simple electronic devices. Printed circuit boards for complex hardware, such as computer graphics cards and motherboards, may have up to twelve layers. PCBs are most often green but they can come in any color.

PCB or Printed Circuit Board is the traditional name for the bare board with the layout data and which you use to mount your components on once we have delivered it to you.

A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways,

tracks or signal traces etched from copper sheets laminated onto a non-conductive substrate.
When the board has only copper tracks and features, and no circuit elements such as capacitors, resistors or active devices have been manufactured into the actual substrate of the board, it is more correctly referred to as printed wiring board (PWB) or etched wiring board.
Use of the term PWB or printed wiring board although more accurate and distinct from what would be known as a true printed circuit board, has generally fallen by the wayside for many people as the distinction between circuit and wiring has become blurred.

Modern printed circuit boards are multi-layered with surface components being mounted by machine.

.

1.2 Introduction

- 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 in order 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 of the basics.

- 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.

Printed circuit boards (PCBs) are the boards that are used as the base in most electronics – both as a physical support piece and as the wiring area for the surface-mounted and socketed components. PCBs are most commonly made out of fiberglass, composite epoxy, or another composite material.
Most PCBs for simple electronics are simple and composed of only a single layer. More sophisticated hardware such as computer graphics cards or motherboards can have multiple layers, sometimes up to twelve.

Although PCBs are most often associated with computers, they can be found in many other electronic devices, such as TVs, Radios, Digital cameras and Cell phones. In addition to their use in consumer electronics and computers, different types of PCBs are used in a variety of other fields, including:

• **Medical devices**

Electronics products are now denser and consume less power than previous generations, making it possible to test new and exciting medical technology. Most medical devices use a high-density PCB, which is used to create the smallest and densest design possible. This helps to alleviate some of the unique constraints involved with developing devices for the medical field due to the necessity of small size and light weight. PCBs have found their way into everything from small devices, such as pacemakers, to much larger devices like X-ray equipment or CAT scan machines.

• **Industrial machinery**

PCBs are commonly used in high-powered industrial machinery. In places where current one-ounce copper PCBs do not fit the requirements, thick copper PCB can be utilized instead. Examples of situations where thicker copper PCBs would be beneficial include motor controllers, high-current battery chargers and industrial load testers.

• **Lighting**

As LED-based lighting solutions catch on in popularity because of their low power consumption and high levels of efficiency, so too does aluminum-backed PCB which is used to make them. These PCBs serve as heat sinks and allow for higher levels of heat transfer than a standard PCB. These same aluminum-backed PCBs form the basis for both high-lumen LED applications and basic lighting solutions.

• **Automotive and aerospace industries**.

Both the automotive and aerospace industries make use of flexible PCB, which is designed to withstand the high-vibration environments that are common in both fields. Depending on specifications and design, they can also be very lightweight, which is a necessity when manufacturing parts for transportation industries. They are also able to conform to the tight spaces that might be present in these applications, such as inside instrument panels or behind the instrument gauge on a dashboard.

- PCB is an acronym of **Printed Circuit Board** that helps in connecting the electronics components with pads, tracks and lines incorporated on a laminated copper sheet.

- It is considered as an insulating material which can be developed using epoxy on which copper layer is laminated.
- PCB design has been evolved in an amazing way and revitalized the latest technology. You can have a look at top 10 PCB design software.
- Before the inception of PCB, professional used laborious method of point-to-point wiring to connect the electronics components. This method was costly and lead to a most complicated design.
- In order to get rid of end-to-end wiring and make the circuit design hassle free, first PCB was developed by Australian Engineer Paul Eisler.

- With the passage of time demands of electronics became prevalent, this made professionals think they should come up with an ideal solution that made the electronics cheap and incorporated in a lesser space.
- This was the start of PCB that revolutionized the electronics industry with lots of innovation and productive ideas.
- Mostly, PCBs are composed of composite material, composite epoxy and fiber glass.
- These are the most common components used in electronics devices that makes the circuit design sophisticated and compact.
- PCBs come in different layers and multiple designs. PCBs used in simple electronics are composed of single layer. Most compact and advanced hardware like graphics card and motherboard are composed of multi layers PCBs.

1.3 HISTORY OF PCB INNOVATIONS

1.3.1 Evolution of PCBs

Over time, PCBs have evolved as an easy tool for optimizing the manufacturing of electronics products. What was once assembled easily by human hands soon gave the way to microscopic components which required the precision and efficiency of machinery. Consider the two circuit boards shown below. One is an older board for a calculator made within the 1960s. While the other is a typical high-density motherboard that you'll see in today's computers.

1.3.2 Introduction

Printed circuit boards are essential in a vast array of electronic devices and computer components. PCBs in their modern form have been around since the 1960s, when they were components in calculators, cash registers and other simple devices with electrical circuits.
In the 1970s, PCBs began appearing in digital watches and some of the world's first video games and personal computers. By the 1980s, PCBs were in alarm radios, video cassette recorders, Atari game consoles, CD

and laserdisc players and cordless phones. During the 1990s, an increasingly advanced and miniaturized PCB was responsible for the spread of desktop computers and peripheral devices like scanners and printers among everyday Americans.

The rapid evolution of computerized devices since the late 1990s has led to smaller and smaller PCBs with greater capacity than boards from prior decades. Smartphones, for example, have grown more compact, yet damage-resistant. A similar development occurred over roughly 12 years with portable MP3 players, which advanced from the heavy, cumbersome Arches players of the early 2000s to the sleek, lightweight iPod Nano models in favour among today's listeners on the move.

The development of chip components with greater versatility has facilitated the miniaturization of devices. With **micro ball grid arrays** and shorter boards, companies can now manufacture small computerized devices, knowing PCBs are sufficiently compact to fit into the enclosure at hand. This stands in contrast to earlier PCBs, where the size of the board would often dictate the size requirements of the enclosing device.

1.3.3 Evolution of PCBs

Over time, PCBs have evolved as an easy tool for optimizing the manufacturing of electronics products. What was once assembled easily by human hands soon gave the way to microscopic components which required the precision and efficiency of machinery. Consider the two circuit boards shown below. One is an older board for a calculator made within the 1960s. While the other is a typical high-density motherboard that you'll see in today's computers.

1.3.4 Chronological History

1850-1900

The spread of electrical power foreshadowed the introduction of telephones, light bulbs and consumer cameras, all of which hit the market during the final quarter of the 1800s. Even though the PCB itself did not exist within this timeframe, the technological foundations for its eventual development were almost solely rooted in the late 19th century.

1900-1950

This is the time that shows the primary patent for a PCB. In 1903, one of the famous German Inventor Albert Hanson filed a British patent for a device described as a flat, foil conductor on an insulating board having multiple layers.

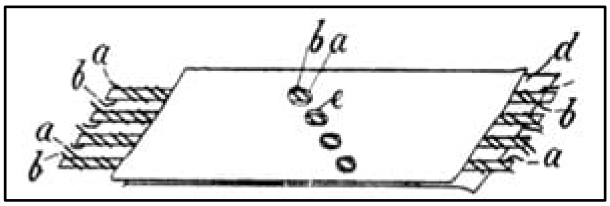
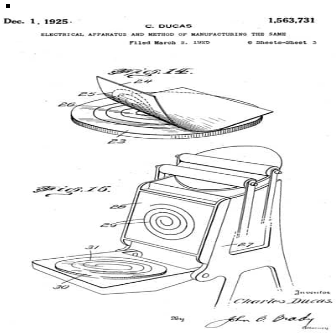


Fig. 1.3: The First Albert Hanson PCB patent drawing

The board did have through-hole construction and conductors on both sides, just like plated through-hole PCBs today. However, devices of this type would not see widespread use in the emerging technologies of the early 20th century, a period that saw the introduction of radios, phonographs, washing machines, dryers and vacuums.

.



In 1927, Charles Ducas patented a version of the circuit board. He used a stencil to print wires onto a board with conductive ink, placing an electronic path right onto an insulated surface. He called it printed wiring, and it was the precursor to today's electroplating.

Fig. 1.4 Ducas earlier PCB

The first significant adaptation of PCB-like technology occurred after the attack on Pearl Harbour in December 1941.

Paul Eisler had a background in printing, and was captivated by the idea of printing actual electronic circuits onto boards rather than soldering the wire on by hand. Unfortunately, the Jewish Eisler was distracted by the rise of Nazism, which forced him to flee from Austria in 1936.

In 1941, the U.K.-based Austrian inventor, Eisler, advanced the PCB concept with a device that used copper **foil on a non-conductive glass base,** which is considered to be the first modern printed circuit board. as the design would foreshadow modern-day top/bottom copper insulation on PCBs. In 1943, he also introduced a PCB-equipped radio that would later prove useful in military operations.

Fig. 1.5 Radio made by Paul Eisler that Uses first printed circuit board (PCB)

1950-1980

The first significant post-war use of PCBs came in 1947 with the Bell Labs transistor.

After World War II, the United States set its eyes on the final frontier: space. Printed circuit boards made space exploration possible in a way people had never considered before. Printed circuit boards significantly increase the efficiency of spacecraft, because they are low-weight and do not use a lot of electricity, even when performing very complicated tasks. In 1963 two developments occurred in PCB technology. One involved a plated hole-through technology. The second innovation at this time was the development of surface mount technology, courtesy of IBM. Both innovations were essential components of Saturn rocket boosters.

In 1980s, the decade saw PCB-reliant devices appear in the living rooms and pockets of everyday Americans. Gadgets such as VHS recorders, compact disc players, Walkman's, cordless phones and gaming consoles all transmitted signals via PCBs. Personal computers and EDA software also made inroads during this time.

.

1980-The Digital Era

The 80s, better referred to as the Digital Era, presented significant changes in how people viewed electronics. The demand for electronic gadgets increased significantly, and electronic shops struggled to meet the high demand for electronics.

In the 80s, circuit boards underwent a further decrease in size due to the invention of surface-mount technology (SMT). Due to lower manufacturing costs and high functionality, developers preferred SMT to the through-hole method, even with miniaturized board sizes.

By then, designers were still sketching circuits manually using light boards and stencils. But when the Computers and Electronic Design Automation (EDA) software was introduced in the 80s, developers quickly shifted to computer designing.

The 90s-PCBs for New compact SMDs

Even with the density and functionality of PCBs improving, production costs reduced significantly in the 90s. This enabled manufacturers to make various electronics to sustain the swelling demand.

In the 90s, silicon use became more familiar with the invention of the Ball Grid Array (BGA) method. This packaging technique offers more interlinking pins than the flat packaging method. Instead of just the edges, the whole lower part of the PCB becomes functional. Designers embraced Design for Test (DFT) techniques in their work. Instead of concentrating on solving the present needs, they considered future reworks in their designs.

During this period, electronics became more complicated, and consumers and businesses worldwide became more reliant on them. The global PCB manufacturing industry hit a market valuation of $7.1 billion in 1995.

2000 and beyond -The Internet Age

During this period, circuit boards became even smaller and denser. While five to six mil trace and space were assumed standard during this period, some high-tech companies fabricated PCBs with 3.5-4.5 mil traces and space. Elastic boards became more popular because they were a bit cost-effective; hence, more suitable for designs that required compact sizes.

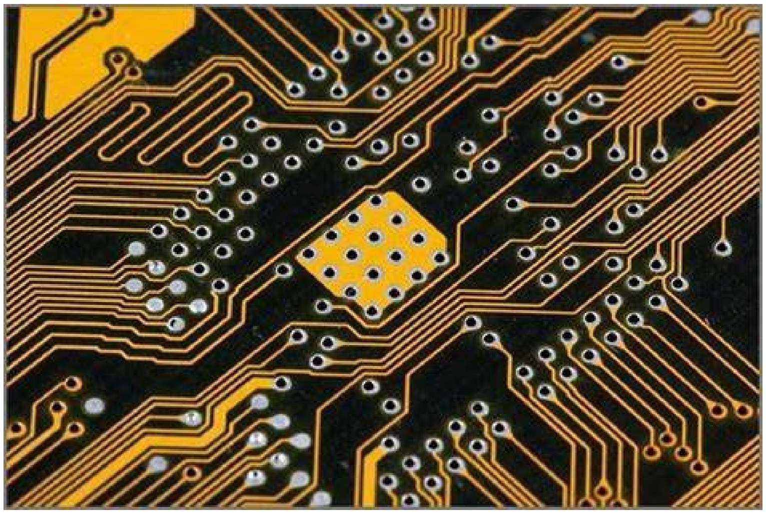
In 2006, Every Layer Interconnect (ELIC) process was introduced. It allowed developers to create links between stacked board layers. While this process raised the flexibility level and enabled designers to make maximum use of interlink density, ELIC boards were not widely applied until the beginning of the 2010s.

Modern Day PCBs

Though a PCB in the 60s might have contained almost 30 transistors, one chip on a contemporary motherboard can house over one million components. This has been enabled by cutting-edge technology, and electronic companies can effectively integrate high functionality into their PCB products.

Further, please go through Chapter 10, where the latest trend in PCB Technology has been defined, in length.

Chapter 2:
PCB Substrates & Materials



2.1 Introduction:

The primary material used to fabricate PCBs (Printed Circuit Boards) is of immense importance. It is essential to consider the performance of the material in terms of temperature resistance, adhesion, tensile strength, flexibility, dielectric strength, dielectric constant, and many other physical, electrical, and thermal factors. The performance and level of integration of the PCB is completely dependent on the material used to fabricate the PCB.

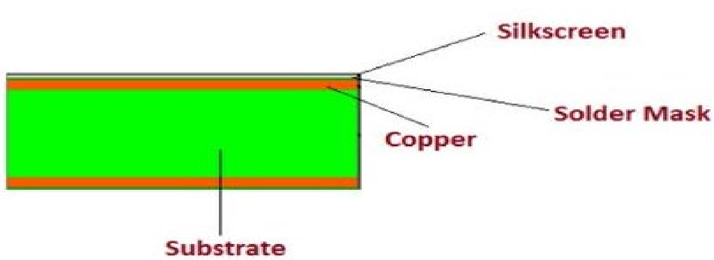
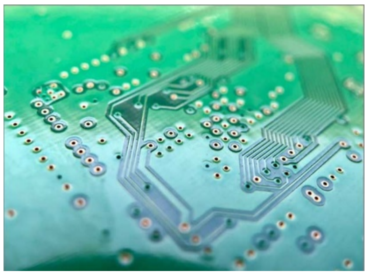


Fig. No.2.1

2.2 PCB Substrates

2.2.1 FR-1/FR-2



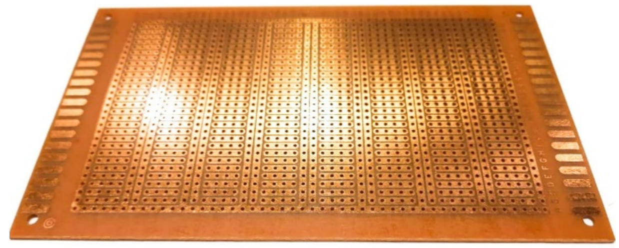


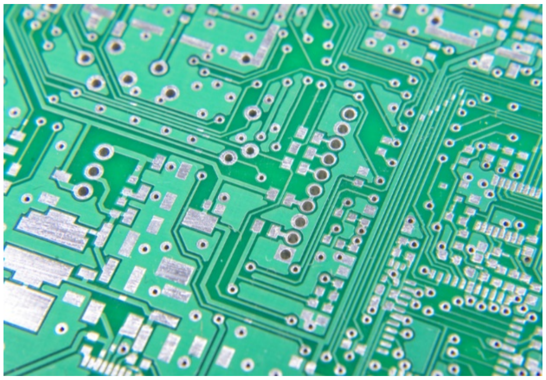
Fig. No.2.2: FR 2 Substrate

Fig. No.2.3:FR 1 Substrate

The family of laminates under the name FR-1 and FR-2 by the NEMA (National Electrical Manufacturers Association, USA) classification. These materials are made from paper and phenol compounds and are used only for the production of single side printed circuit boards. FR-1 and FR-2 has the similar parameters, the main difference is that FR-1 has a higher glass transition temperature T_{ag}. Because of the similarity of the parameters and applications of FR-1 and FR-2, many of the material suppliers produce only one type of laminates, most often FR-1. Laminates have a good ability to mechanical processing (milling, punching). Flammability rating is UL94-V0. FR-1/FR-2 laminates are divided into the following subclasses:

- standard;
- halogen-free, without phosphorus and antimony, non-toxic;
- with normalized index of CTI ≥ 400, ≥ 600;
- non hydrophobic;

2.2.2 Common PCB Material FR4



Most often printed circuit boards are made of standard glass-epoxy laminate namely FR4 type, with an operating temperature from -50 to + 110°C, glass transition temperature T_g of 135°C. The dielectric constant D_k can range from 3.8 to 4.6.

Fig. No.2.2: FR4 PCB Material

FR 4 Material Sub Classification:

Depending on the properties and applications, FR-4 laminates are divided into the following subclasses:

- standard, with a glass transition temperature T_g of ~ 130°C, with UV blocking or without it. The most common and widely used type, at the same time, the cheapest of FR-4;
- with a higher glass transition temperature, T_g ~ 170°C - 180°C, compatible with the lead free reflow technology;
- halogen-free, compatible with the lead free reflow technology;
- with normalized index of CTI ≥ 400, ≥ 600;

2.2.3Other PCB Materials

PTFE (Teflon)

PTFE is a kind of plastic material that does not provide any resistance, thus is used for high speed, high frequency applications. PTFE is extremely flexible, making it invaluable in applications with tight tolerances. It is also extremely lightweight, allowing it to be used across various industries. It is also flame resistant, exhibits high physical strength, provides temperature stability, and is versatile in application.

Metal

The traditional materials of copper, aluminum, iron, etc. are still used in PCBs. These materials allow the use of Surface Mount Technology (SMT) for the integration of components. They also provide mechanical durability. Thus, the product life of metal base PCBs is much longer.

2.2.4CEM

2.2.4.1 CEM-1

The family of laminates under the name CEM-1 by the NEMA (National Electrical Manufacturers Association, USA) classification. These materials are made from paper and two layers of woven glass epoxy and phenol compounds and are used only for the production of single sided printed circuit boards. As a rule, they have a milky white or milky yellow color. Laminates are incompatible with the process of metallization in holes; therefore, they are used only for the production of single sided printed circuit boards. Dielectric properties close to that of FR - 4, mechanical endurance is somewhat worse. CEM-1 is a good alternative to FR-4 when the price is the deciding factor. Flammability rating is UL94-V0.

CEM1 laminates are divided into the following subclasses:

- standard;
- with a higher T_g , compatible with the lead free reflow technology;
- halogen-free, without phosphorus and antimony, non-toxic;
- with normalized index of CTI ≥ 600;
- non hydrophobic, with good dimension stability;

2.2.4.2 CEM-3

The family of laminates under the name CEM-3 by the NEMA (National Electrical Manufacturers Association, USA) classification. Composite material based on glass-epoxy compounds, usually has a milky white color. Very widely used in the production of double-sided PCBs with plated holes. The properties are very similar to that of FR-4, except the lower mechanical endurance. CEM-3 is a cheaper alternative to FR-4 for most applications. Laminates have a good ability to mechanical processing (milling, punching). Flammability rating is UL94-V0.

Depending on the properties and applications, CEM-3 laminates are divided into the following subclasses:

- standard, with UV blocking or without it.
- with a higher T_g , compatible with the lead free reflow technology;
- halogen-free, without phosphorus and antimony, non-toxic;
- with normalized index of CTI ≥ 600;

2.2.5 RO3000

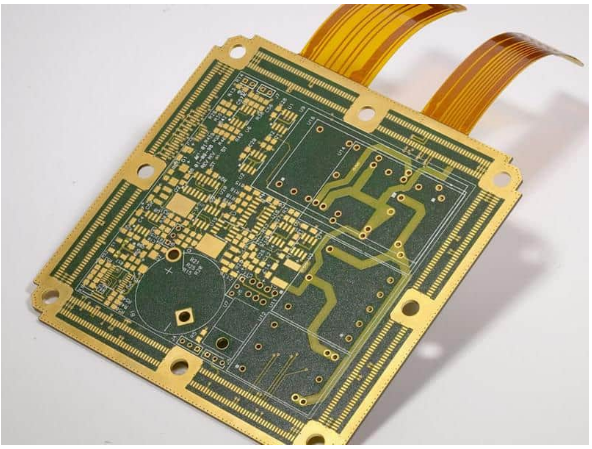
A family of laminates developed for wide use in the early 90s of XX century. These materials have excellent electrical properties at the high frequencies and high thermal stability. The CTE (coefficient of thermal expansion) along the X and Y axes is close to the CTE of copper and FR4, therefore it is possible to produce the reliable RO3000 / FR4 hybrid PCBs. Low dielectric losses (D_f = 0,0013 at a frequency of 10 GHz) provide great benefits when using these laminates in the applications for microwave range.

2.2.6 RO4000

This is a family of materials for a very high frequency range, which has been designed, on the one hand, to achieve the performance comparable with that of materials containing polytetrafluoroethylene (PTFE), and on the other hand, to simplify the technology of PCB production, that is, to make it more in line with the traditional technology used for reinforced laminates (FR4). Materials RO4000 contain reinforced fiber glass of a high glass transition temperature (T_g = 280 °C) with a thermosetting polymer as a bonding agent as well as additives ceramics.

2.3 Polyimide PCB:

So, polyimide is a flexible but rigid plastic-like substance that is durable enough to serve as a PCB substrate. The PCB Manufacturers widely use it due to its high cost to performance ratio.



Let’s review such a popular material as polyimide/polyamide, its different types, benefits of usage, and the scope of application. Besides, the comparison of polyimide and FR-4 materials will be provided as well. Polyamide is a polymer that comprises multiple imide monomers. It can be both synthetic and natural. The latter is silk and wool that may also be used for PCB

Fig. No.2.4:Polymide PCB

Polyimide vs FR4: What is the Difference?

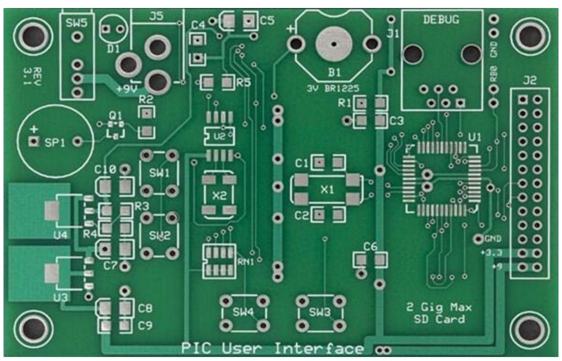


Fig. No.2.5:FR4 Material

Table 2.1

Factors	**Polyimide**	**FR-4**
Dimensional stability	Superior dimensional stability overall	Good dimensional stability
Flexibility	2nd and 3rd generation polyimides are flexible. Others – don’t	Has mediocre flexibility
Chemical resistance	Good resistance value to most chemical materials	Good resistance value to most chemical materials
Thermal resistance	It is highly resistant to temperature fluctuation. It can be used in operating environments in the range of -200°C to 300°C the 2nd generation polyimide does not withstand open flame though	It is moderately resistant to temperature fluctuations. It can be used in operating environments in the range of -50°C to 110°C

Factors	Polyimide	FR-4
Resistance to physical stresses	Superior resistance of filled and low-flow polyimides, and moderate resistance of 2nd and 3rd generation ones	Good resistance overall
Durability	Well durable overall	Well durable overall
Tensile strength	It can withstand up to 231 MPa	It can withstand up to 70 MPa
Thermal cycling	It has a superior thermal cycling capacity	It has moderate thermal cycling capacity
Cost	Low to medium-priced	One of the most affordable materials overall

2.3.1 Benefits of Using Polyimide PCBs

Polyimide is a versatile material having multiple strengths to offer. Some of its benefits to consider are the following:

a) Dimensional stability. The good thing about polyimide PCBs is that they do not change their dimensions under temperature fluctuations. It means that at -100°C and at 100°C, they will have nearly the same sizes. It makes the 2nd and 3rd polyimide types suitable for microelectronics.

b) Flexibility. 2nd generation polyimide comprises flame protection but offers superior flexibility overall. It is also an aspect of making such a material suitable for micro-PCBs.

c) Rigidity. Low-flow polyimides have superior rigidity. That makes them suitable for some applications that require electronics to withstand high stresses.

d) Chemical resistance. Polyimide PCBs withstand some inorganic and most organic acids, making them suitable for medical electronic. Polyimide have one of the broadest ranges of temperatures available. So, such boards fit most electronics operating in extreme environments.

e) Great tensile strength ratio. This property allows polyimide PCBs to withstand high levels of pressure and some forms of physical stresses.

2.4 Alumina PCB

Alumina PCBs is a type of ceramic PCBs that use aluminum oxide (Al_2O_3) as their substrate.

Alumina PCBs require high purity Al_2O_3 materials. Alumina has many isomorphic crystals, and alumina PCBs use α-Al_2O_3 . There are 75%, 96%, and 99% alumina substrates. The higher purity, the smoother surface, denser appearance, and lower dielectric loss. And of course, the higher price. To make a balance between quality and costs, PCBONLINE uses 96% alumina substrates for alumina PCB manufacturing.

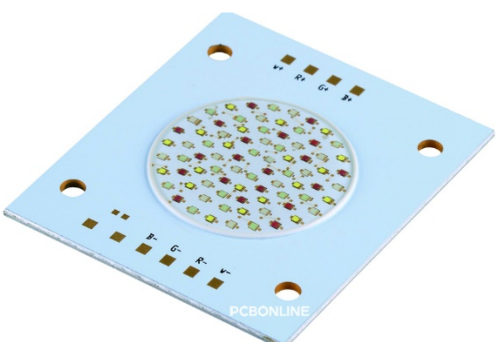


Fig. 2.6: Alumina PCB in LED Light

The most important characteristic of an alumina PCB is the **thermal conductivity, which ranges from 15W/mK to 50W/mK**. When the LED or IC generates a lot of heat, the heat is transferred to the alumina circuit board directly and quickly. Using the alumina PCB, there is no need for an insulating layer as it already achieves good thermal dispassion.

Another property of the alumina PCB is **the high thermal withstand**. The melting point of the 96% alumina is **1400°C**, and for the 99% alumina, it is **1600°C**. However, the melting point doesn't stand for the PCB maximum using temperature. If the alumina PCB is multilayer, and the manufacturer uses PP (polypropylene) to connect the two alumina layers, the PCB's overall maximum using temperature is reduced, as PP melts at lower temperatures. If the manufacturer adopts the high-temperature sintering lamination method, the two ceramic layers directly bond, and the thermal withstand is not affected. But in whatever lamination method or manufacturing process, an alumina PCB can work properly at 300°C. Compared with another ceramic PCB substrate, the beryllium oxide (BeO), alumina is safe. BeO has a much higher thermal conductivity than alumina, but it is toxic. While alumina has high biocompatibility, in other words, **safety**. Besides, alumina PCBs have excellent hardness and electrical insulation. For your reference, below is an alumina PCB.

Table 2.2 Specifications table (courtesy: PCB Online).

Characteristics	96% alumina	99% alumina
Color/Appearance	White/Dense	Beige/Dense
Density	3.6g/cm³	3.9g/cm³
Hardness	1500HV	1700HV
Warpage	≤0.3mm	≤0.2mm
Parallelism	±0.4%	±0.3%
Bending Resistance	3000Kgf/cm²	3500Kgf/cm²
Compressive Strength	25000Kgf/cm²	30000Kgf/cm²

Fracture Toughness	3-4Mpa m1/2	4Mpa m1/2
Thermal Conductivity	25W/(m·K)	31.4W/(m·K)
Voltage Resistance	18KV	18KV
Dielectric Constant [1MHz, 25°C]	9.4	10
Characteristics	**96% alumina**	**99% alumina**
Surface Roughness Ra	0.6-0.8μm	0.6-0.8μm
Water Absorption	0%	0%
Sintering Temperature	1689°C	1700°C
Volume Resistivity	/	/
Thermal Shock Resistance	200T°C	220T°C
Thermal Expansion Coefficient [20°C to 300°C]	6-7.5×10ˆ(-6)°C	8-10×10ˆ(-6)°C
Maximum Use Temperature	1400°C	1600°C

2.4.1 Pros and Cons of Alumina PCB

I find that the above-mentioned characteristics of alumina PCBs are the pros, but I need to list their pros and cons still. To be precise, the pros of the alumina PCBs are:

- Strong heat resistance and thermal conductivity,
- High strength and hardness,
- High electrical insulation, corrosion resistance, and biocompatibility.

In regards to the cons, alumina PCBs have the same cons as all of the ceramic PCBs do:

- Alumina PCBs are fragile and easy to break,
- Have higher costs than normal FR4 and metal core PCBs.

2.4.2 Application of Alumina PCBs:

Alumina is a material having balanced mechanical performance parameters, and it is cheaper than other ceramic materials such as aluminum nitride and BeO. So, alumina PCBs are used more commonly than other ceramic PCBs. Its applications include:

- Automotive products such as engines, sensors, and shock absorbers,
- LED lighting products,
- High-power semiconductor modules such as semiconductor refrigerators and electric heaters,
- Smart power modules such as high-frequency switching power supplies and solid-state relays,
- High-temperature industrial furnaces,
- Refrigeration sheets,
- Telecommunications dedicated exchanges, receiving systems, lasers, and other industrial electronics.

2.5 G10/G11 PCB Substrate

2.5.1 Advantages of G10 Epoxy Substrate

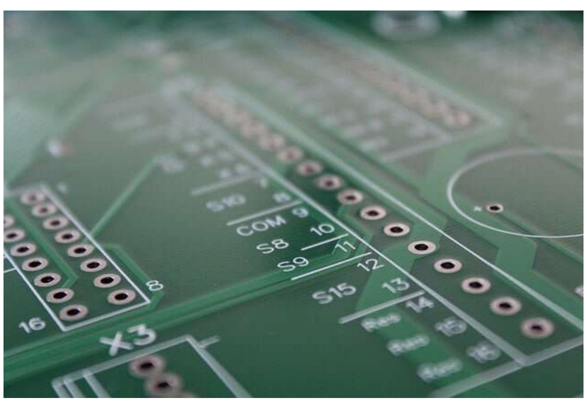


Fig. 2.7:G 10 Substrate

The PCB industry commonly uses substrates composed of glass fiber impregnated with resin, called G10. The G10 material offers very good insulation, and resistance to wear and corrosion. In comparison with regular glass fiber, epoxy-based glass fiber offers the PCB excellent properties such as high elastic modulus, high tensile strength, excellent impact resistance, very good chemical stability, high temperature stability and good fatigue resistance.

The G10 glass epoxy material with natural resins has color varying from light yellow to light green. The most popular G10 laminate grade is typically a continuous fiberglass fabric that has been impregnated with an epoxy adhesive. The electronics industry uses epoxy resin as the most versatile plastic, mainly because of its features of nearly zero water absorption, excellent insulation properties, perfect dimensional stability and exceptional adhesion properties, along with splendid electrical insulating features.

G10 offers extremely low dielectric loss performance, high mechanical strength, and superior electrical insulation, both when wet or dry. FR4 is a grade of G10 with flame retardant capabilities, and this is the main difference between FR4 and NEMA G10. Therefore, PCB manufacturers can easily and safely replace G10 substrates with FR4, but not the reverse.

G10 PCBs are used in aviation, agriculture, construction, aerospace, and chemical industries.

2.5.2 Comparison of G10 and FR4 Materials

- Both FR4 and G10 are high-pressure industrial PCB substrates consisting of glass fabric impregnated with epoxy resin.
- The major difference between FR4 and G10 is in their resin composition.
- G10 epoxy resin is a composite of epichlorohydrin and bis phenol A.
- In FR4 epoxy resin, tetra Bromo bis phenol A replaces a small part of bis phenol A.
- Because of the above replacement, FR4 laminate is more flame retardant.
- G10 and FR4 are very similar in properties.
- Glass transition temperature or Tg is relatively the same for both FR4 and G10.

2.5.3 Advantages of G10 Epoxy Substrates

a) Diverse Forms—G10 epoxy may contain a wide variety of resins, curing agents and modifier systems to achieve diverse shape requirements, suitable for almost any application, varying from low viscosity to high temperature solids.

b) Easy to Solidify—By using a variety of curing materials, it is possible to cure the G10 epoxy resin systems in temperatures from 0 to 180 C.

However, for specific environments with high temperatures, the recommendation is to use high Tg materials.

c) Strong Adhesive Force—G10 epoxy materials exhibit high adhesion to several materials, as they contain ether and hydroxyl bonds that are inherent in molecular chains of epoxy resins. G10 epoxy resins feature low shrinkage during the curing process and create very little inherent stress. This helps to improve the bonding strength.

d) Low Rate of Contractibility—G10 epoxy resins react to their curing materials. The reaction can be from direct addition of epoxy catalogs or open loop polymerization in the resin molecule. However, there is no release of water or any other volatile material. In comparison with unsaturated polyester resins and phenolic resins, G10 epoxy resins shrink less than 2% during curing.

2.5.4 G10 Epoxy Substrates—Technical Specifications

Technical Parameter	**Specifications**
Flame Retardant	HB
Bending Strength (normal), Perpendicular Layer	⩾340 Mpa
Bending Strength (150±2 °C), Perpendicular Layer	⩾340 Mpa
Impact Strength, Parallel Layer	66.7 KJ/m^2
Insulation Resistance (immersion in water D-24/23)	⩾5.0×10^2 Ω
Electrical Strength, Vertical Layer	⩾10.7 MV/m
Breakdown Voltage, Parallel Layer	⩾40 kV
Relative Dielectric Constant (1 MHz)	⩽5.5
Dielectric Loss Angle	3.3×10^{-2}
Water Absorption (D-24/23, Plate Thickness 1.6 mm)	6.5 mg

2.5.5 G11 Epoxy Board

a) Material:G10 epoxy board contains hard insulating material

b) Important Parameters

- Glass transition temperature 175 °C
- Resistance to water – high

c) Composition—Alkali-free glass-fiber cloth impregnated with epoxy resin, flame retardant, adhesive, and other additives added with hot pressing.

d) Applications—used for humid environments, manufacture of high-voltage switches, insulation oil, high-voltage GIS.

Conclusion

There are different materials used for the PCB design and fabrication, all offer a variety of advantages and disadvantages. The material is chosen according to the application, the result needed, environmental factors, and any other constraints the PCB will face.

Chapter 3:

Single, Double sided and Multilayer PCB

Part I: Single sided PCB

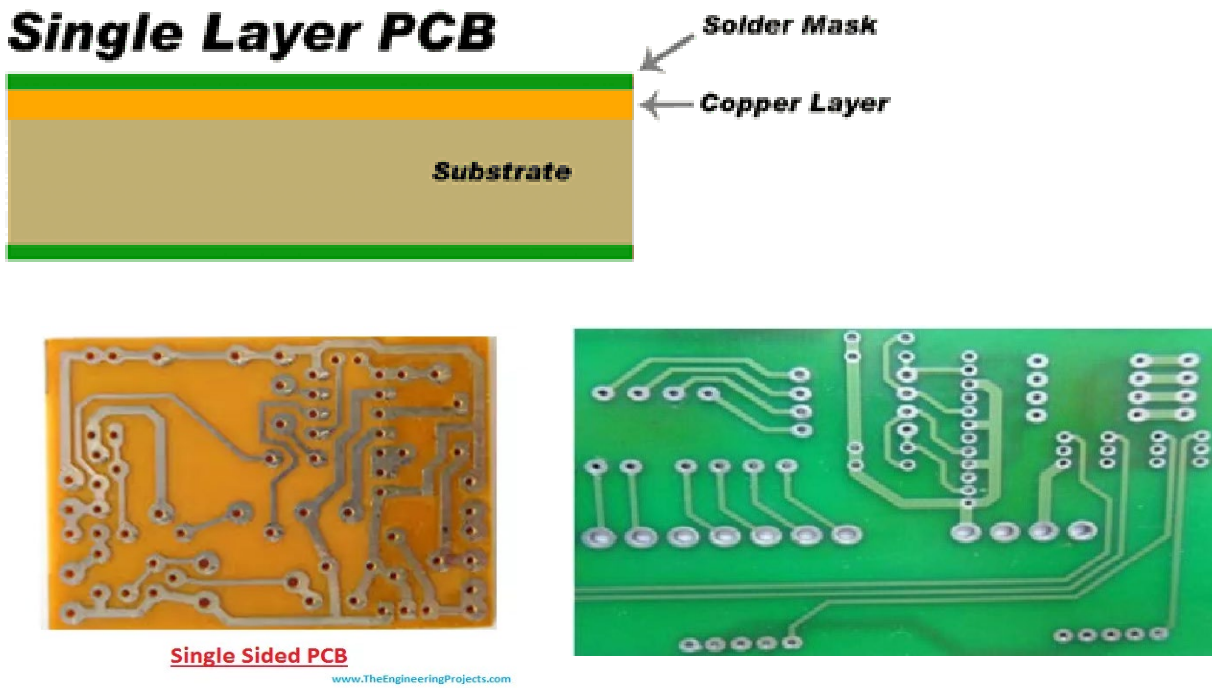


Fig.3.1: Single sided PCB

3.1 Definition: **Single Sided PCB** is the simplest printed circuit board, only have one layer of conductive material and are best suited for low density designs, Holes in the board are usually not plated through.

Component parts is layout on one side and the circuit is on the other side. As there is only layer conductor, it is called single sided PCB (Single-sided PCB or one layer PCB.

- Single sided PCBs are the start point of printed circuit board technology and still plays an important in electronic industry.
- These PCBs are usually cheaper to manufacture and are an ideal choice for low density designs.

- Single sided PCBs came into play in 1950s, manufactured in USA and still dominate the electronic market because of their simple design.

First time single sided PCB was made in 1950 and still have been using in market in a bulk quantity.

3.2 Features of Single Layer PCB :

- **High density** – For decades, high density printed boards have developed with the improvement of integrated circuit integration and the advancement of mounting technology.
- **High reliability** – Through a series of inspections, tests, and aging tests, the PCB can work reliably for an extended period (usually 20 years).
- **Designability** – For the various performance (electrical, physical, chemical, mechanical, etc.) requirements of the single panel, the printed board can be designed through design standardization in a short time and with high efficiency.
- **Producibility** – With modern production management, it can be standardized, scaled (quantified), automated, etc., to ensure product quality consistency.
- **Testability** – Complete test methods, standards, various test equipment, and instruments have been established to detect and appraise the eligibility and service life of the single PCB.
- **Assembled** – The circuit board facilitates the standardized assembly of various components and enables automated and large-scale mass production. At the same time, circuit boards and various component assembly parts can be assembled to form larger parts and systems, up to the complete machine
- **Maintainability** – Circuit boards and various component assembly parts are manufactured in standardized design and scale. If the system fails, it is convenient to replace components quickly; the system can be restored promptly with such flexibility. There are more examples, such as miniaturization and weight reduction of the system, and high-speed signal transmission.

3.3 Application:

Single sided PCBs are widely used in many electronic applications:

- Vending machines
- Camera systems
- Surveillance
- Calculators
- Printers
- Solid state drives
- Coffee makers
- LED lighting
- Packaging equipment
- Sensor products
- Power supplies
- Relays
- Radio and stereo equipment
- Timing circuits

Part II: Double sided PCB, with plated through hole (PTH)

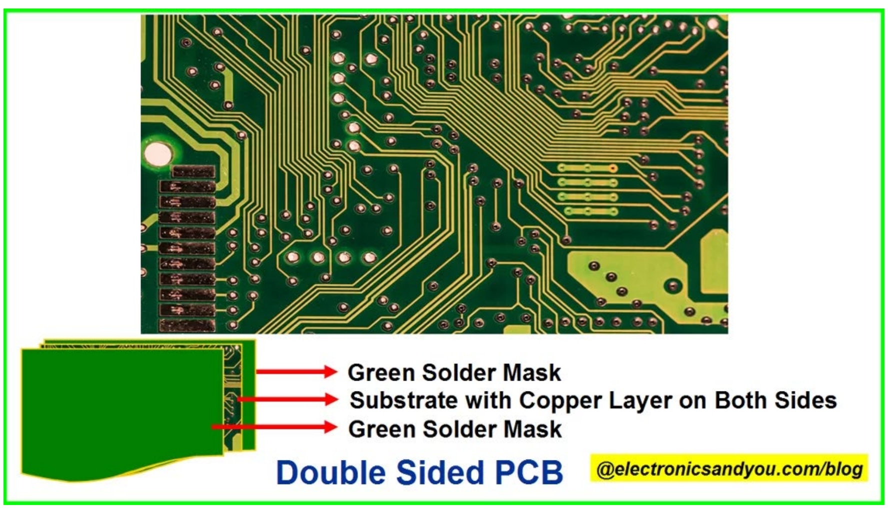


Fig. 3.1: Double sided PCB, with PTH

3.4 Definition:

A Double-Sided PCB Is a Board with Circuit Traces on Both Sides. The Two Faces Are Connected by Plated Holes. Today, Double-Sided Plated through Holes, Also Called Depth, are the principal type Of PCB in the Market.

Double Sided PCB uses both sides of the Printed Circuit Board. This is more usually supplemented with plated-through-holes where the copper connections go right through the connecting holes to the opposite side of the board. These PTH Connections either form simple electrical connection between both sides of the PCB (Via Holes), or electrical connectivity and mechanical support for leaded components.

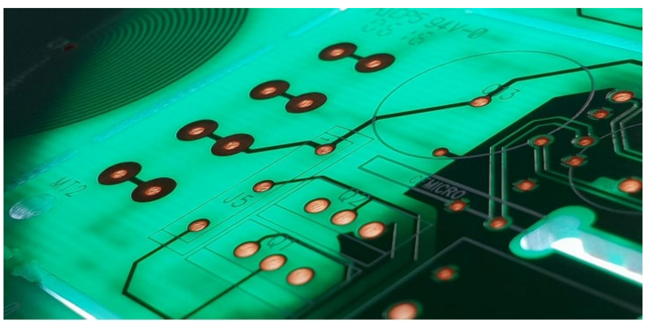


Fig.3.2a:Double sided PCB for Through Hole Boards

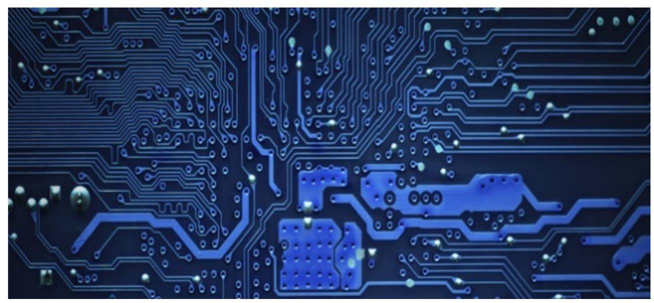


Fig.3.2b:Double sided PCB for SMDs

Both through-hole electronic components and Surface Mount Components (SMD) can be soldered on either side of this type of PCB.

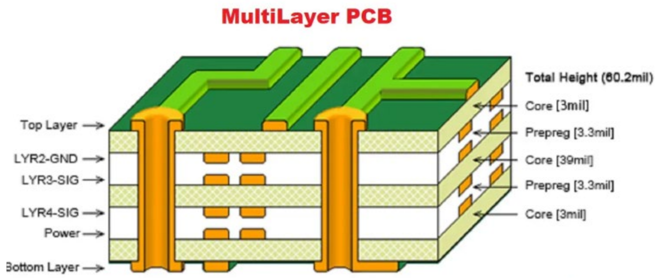
3.5 Application:

Fig. 3.3:Application of Double-Sided Board

Application of Double-Sided PCB

- **Aerospace**
- **Automotive**
- **Railways**
- **ATM**
- **Industry**
- **Consumer**
- **Metering**
- **Multimedia**

Part III: Multi-layer PCB



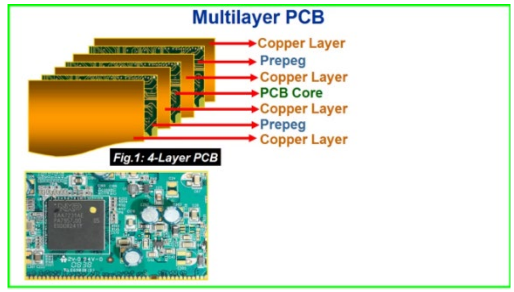


Fig. 3.4: Multilayer PCB

3.6 Definition:

Multilayer PCB is made with three or more conductive copper foil layers. These appear as several layers of double-sided circuit boards, laminated and glued together with layers of heat-protective insulation between them. The entire construction is arranged so that two layers are placed on the surface sides of the PCB to connect to the environment. All electrical connections between the layers are achieved with vias such as plating through holes, blind and buried vias.

Multilayer PCBs allow users to materialize denser designs, without compromising the performance.

Multilayer PCBs were invented due to the progressing changes in the electronics sector. The functions of most electronics have increased over time. Hence, they are more complex PCBs than single-sided PCBs or double-sided PCBs.

For numerous industries, multilayer PCBs have become the preferred option for a variety of applications. Much of this preference derives from the continuous push across all technology toward mobility and functionality. Multilayer PCBs are the logical step in this progression, achieving greater functionality while reducing size.

3.7 Application areas:

- **Consumer Electronics:** Consumer electronics covers a wide range of electronics equipment intended for everyday use. This tends to include devices used for communication such as smartphones, laptops, etc.; entertainment including smart TVs, music players; and kitchen appliances – microwaves, washing machines, and more. Most of these devices are equipped with multilayer printed circuit boards than standard single-layer PCBs. Why? This is because multilayer PCBs contribute to their functionality and compact designs.

• **Computer Electronics**:
Everything from servers to motherboards uses multilayer PCBs, primarily for their space-saving attributes and high functionality. With these applications, performance is one of the most essential characteristics of a PCB, whereas cost is relatively low on the list of priorities. As such, multilayer PCBs are an ideal solution for many technologies.

Fig. 3.5 :Computer Electronics

• **Telecommunications**:
Telecommunication devices often use multilayer PCBs in numerous general applications, such as signal transmission, GPS and satellite applications. The reason for this lies primarily in their durability and functionality. PCBs for telecommunications applications are often either used in mobile devices or towers outdoors. In such applications, durability is essential while still maintaining a high level of functionality.

Fig. 3.6: Telecommunication Equipment

Industrial Electronics: Electronics equipment used in an industrial environment is quite different from consumer electronics. The reason being they are exposed to harsh environments such as high temperature, shock, vibrations, pressure, dust, and humidity. Some common examples of industrial electronics equipment include automated assembly lines, packaging conveyors, spray painting robots, and so on. They are designed for long time functioning and they may be made to work throughout the day. In such cases, durable PCBs are required, and multilayer PCBs exactly meet this requirement. Multilayer PCBs are more durable than other alternatives available on the market today.

Fig. 3.7: Industrial Equipment

• **Medical Devices**: Multilayer PCBs are particularly favored in medical industry for their small size, lightweight nature and impressive functionality compared to single-layer alternatives. These benefits have led to multilayer PCBs being used in modern X-ray equipment, heart monitors, CAT scan equipment and medical testing devices etc.

Fig. 3.8: Medical Devices

• **Military and Defense**: Favored for their durability, functionality and low weight, multilayer PCBs are useful in high-speed circuits, which is becoming an increasing priority for military applications.

Fig. 3.9: Defense Equipment

• **Automotive**: Cars are relying on electronic components more and more in the modern era. This's why many auto manufacturers start to favor multilayer PCBs over other alternatives.

Fig. 3.10: Automotive Electronics

• **Aerospace**: Aerospace PCB applications must be reliable, able to handle the stresses of atmospheric journeys while simultaneously making enough room for the rest of the surrounding equipment. Multilayer PCBs present an ideal solution in this case. Their higher quality and functionality also contribute to this utility in the aerospace industry.

Fig. 3.11: Aerospace Electronics

• Miscellaneous Electronic products:

Multilayer PCBs are used in a wide variety of other industries, including the science and research industry and even home appliances and security. Everything from alarm systems and fiber optic sensors to atomic accelerators and weather analysis equipment uses multilayer PCBs, taking advantage of the space and weight savings offered by this PCB format, as well as their heightened functionality.

Fig. 3.12: • Miscellaneous Electronic products

Owing to its lightweight weight and compact designs, multilayer PCBs are considered to be the best choice for various industrial applications. The following are some important applications of multilayer PCBs:

- Some best examples of medical devices using multilayer PCs include X-ray equipment, CAT scanning equipment, blood pressure, and sugar testing device, MRI, CT scan, infrared temperature monitoring, and so on.

3.8 Conclusion:

With high-speed and high-performance technologies evolving, the demand for multilayer PCBs will continue to increase. So, if your company wishes to invest in those technologies, it needs to focus on building quality multilayer PCB solutions.

Chapter 4:
Flexi & Flexi Rigid PCBs

Part I: Flexible PCBs

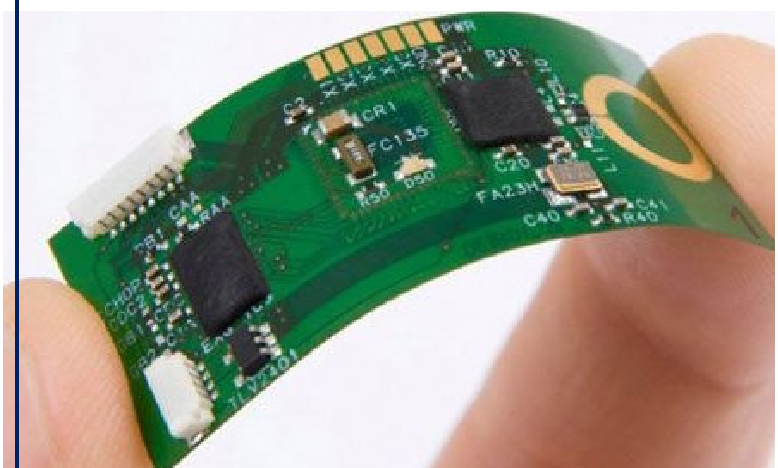


Fig. 5.1 Flexi PCB

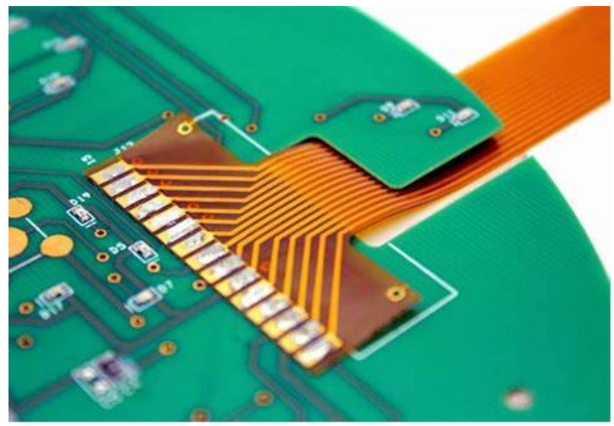


Fig. 5.2 Flexi Rigid PCB

5.1 Definition:

Flexible PCB (also called **Flex PCB/Flexible Printed Circuits (FPC)**) is an incredible addition to the electronics, because of its ability to bend, twist or fold into any shape, occupies less space and is very lightweight The IPC defines a flexible printed circuit as “a patterned arrangement of printed circuitry and components that utilizes flexible base material

- A rigid PCB is relatively easy to handle while processing a flexible PCB requires extra protocols (special clamping, material handling etc.), that's why Flexible PCB is normally costly as compared to other types of PCBs.
- Flexible PCBs are normally used in wearable electronic products as they can easily stick with the body.

The material used in flex PCBs can be just a few microns thick, yet still, be reliably etched. This often makes them preferable to rigid PCBs in certain applications, in addition to the benefit of removing traditional wiring harnesses and ribbon connectors, in particular contexts.

5.2 The History of Flex PCBs

At the beginning of the 20th century, early researches in the telephone industry saw the need to alternate layers of conductors and insulator to produce standardized, flexible electric circuits. An English patent from 1903 describes coating paper with paraffin and lying flat metal conductors to provide the circuits. Around the same time, Thomas Edison's notebooks suggested coating linen paper with cellulose gum, then tracing circuits on the gum with graphite powder. The late 1940s brought in mass production techniques, resulting in a number of patents for photo- etching circuits on flexible substrates as a way of replacing wiring harnesses. More recently, the addition of active as well as passive components to flexible circuits has introduced the term "flexible silicon technology", referring to the ability to integrate semiconductors (using technologies that include thin-film transistors) onto the flexible substrate. The combination of traditional advantages found within flexible circuit construction combined with onboard computing and sensing capability has led to exciting developments in several areas, most especially in applications in the aerospace, medical, and consumer electronics fields.

5.3 Application:

Flexible printed circuit boards are suited to a variety of situations where rigid circuit boards can't be used.

a) **Commercial Electronics** –Flexible printed circuits are used widely in computer technology – everything from printers, scanners, smart devices, and more. With the market driving for smaller and more advanced handheld technology, flex PCBs are essential for heavy computing power without sacrificing weight restrictions

Fig. 5.1: Commercial Electronics

b) Wearable Devices -Flex PCB is normally used in electronic wearable devices i.e., wristwatches, lockets, spy microphones, smart cameras etc.

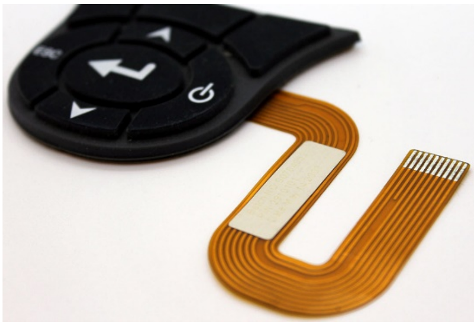


Fig. 5.2: Wearable Devices

c) Electronic Gadgets- Flexible PCB is also used in electronic gadgets i.e., mobile phones, computer keyboards, CD/DVD drives etc.

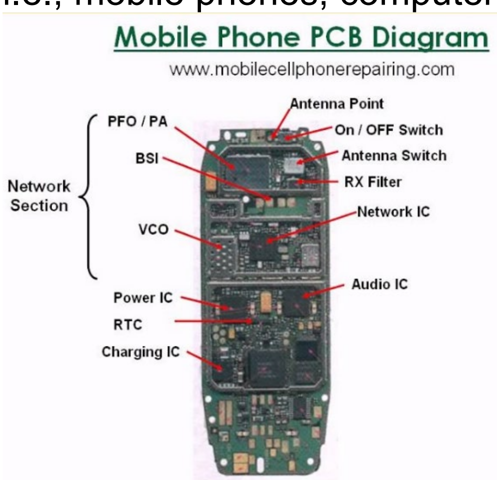


Fig. 5.3: Mobile Phone PCB

d) Automotive – The second largest sector for flex PCBs is in vehicles. The average car looks significantly different from even the beginning of the decade, and their hardware has grown more advanced in turn. As cars begin to include more sensors and electronics for everything from climate control and GPS software to dashboards and rear-view cameras, flex PCBs allow for both space allotments and thermal performance.

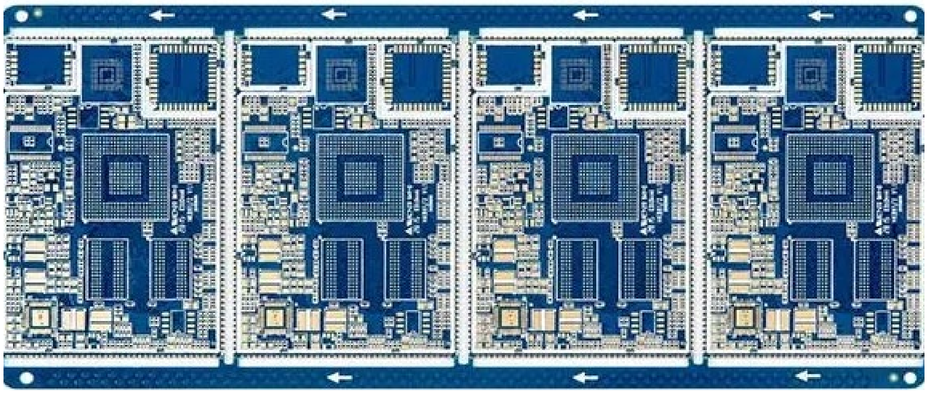


Fig. 5.4: Automotive PCB

e) **Aerospace** – Thanks to their remarkable ability to handle the stress of extreme temperatures and vibrations, flex PCBs are uniquely suited for aerospace applications such as satellites. They also can take on unusual shapes that are well suited for spacecraft and substituting connectors that are more prone to failure.

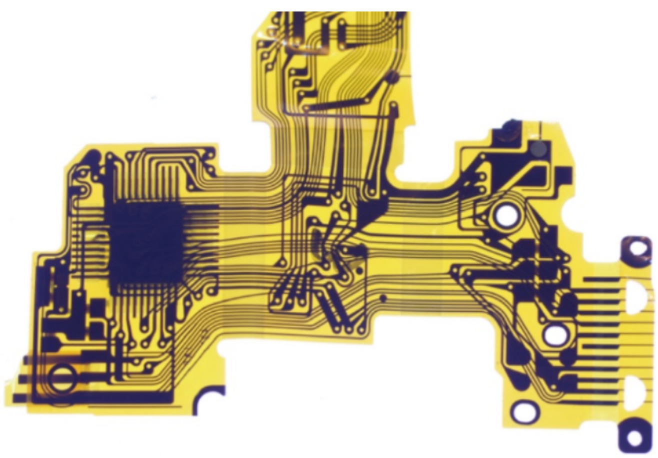
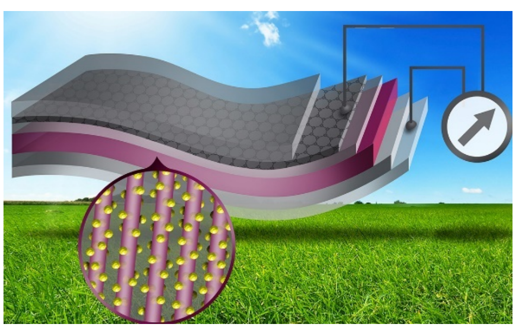


Fig. 5.5: Automotive PCB

f) Medical Devices – As medical technology evolves; wearable devices have become a key feature in the past few decades. Flexible printed circuit boards offer greater durability for these devices, as they can mirror shapes found in the body. Hearing aids, vital sensors, and more all use flexible printed circuits to leverage high performance that can stand up to wear and tear.
A medical device company utilizes flex PCB designs as important components of a new class of hearing- assist devices, providing higher range and resolution.

Fig. 5.6: Flex PCB: Medical Device

g) Solar Technology



A flexible solar cell is another addition in the field of flexible electronics which are lightweight, can be folded or twisted into any shape, and are mainly used for power satellites.

Fig. 5.6: Flex PCB: Solar Cell

Part II: Flexi Rigid PCBs

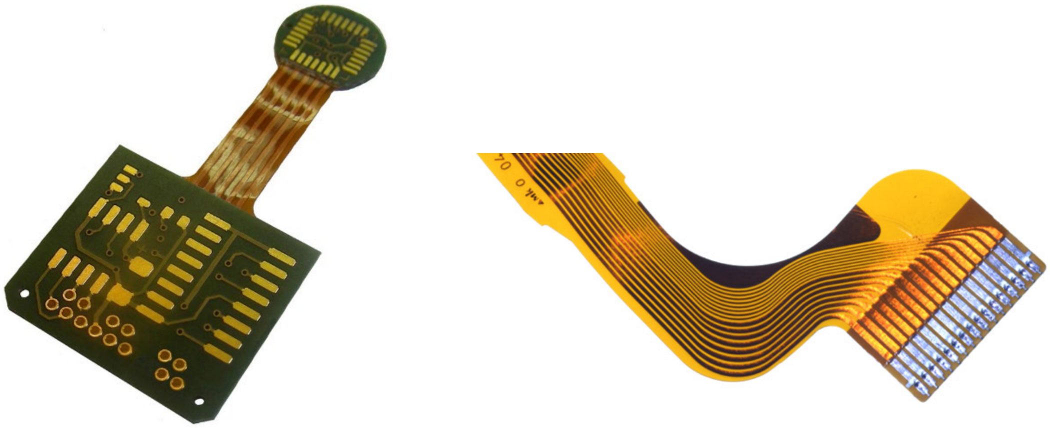


Fig. 5.3 : Rigid Flex PCB

5.4 Definition:

Miniaturization of electronic systems demands highly flexible PCB boards. **Rigid-flex PCBs** contribute largely in this regard. **Rigid-flex PCBs** several advantageous factors that add to its popularity for a wide range of industrial and commercial applications.

Rigid flex printed circuit boards are boards that use a combination of flexible and rigid board technologies. Most rigid flex boards consist of multiple layers of flexible circuit substrates attached to one or more rigid boards externally and/or internally, depending upon the design of the application. The flexible substrates are designed to be in a constant state of flex and are usually formed into the flexed curve during manufacturing or installation.

Rigid flex designs are more challenging than the design of a typical rigid board environment, as these boards are designed in a 3D space, offering greater spatial efficiency. By being able to design in three dimensions designers can twist, fold and roll the flexible board substrates to achieve their desired shape for the final application's package.

Perhaps the most common rigid PCB is the computer motherboard. The motherboard is a **multi-layer PCB** designed to distribute power from the power supply while allowing communication between all components of the computer, such as CPU, GPU, and RAM.

Advantages of **Rigid-flex PCBs:**

- Reduced manufacturing cost as fewer parts are needed for the entire assembly.
- Extreme reliability due to the assembly with minimal number of wires, cables, plug-in connections, and solder joints.
- Occupy less space when compared to traditional boards.

5.5 Application:

COMPUTER MOTHERBOARD

A computer motherboard is a perfect example of rigid PCB, which is a Multilayer Rigid PCB, used to distribute electricity from the power supply and creates a conducting path between CPU, GPU, and RAM.

CONSUMER ELECTRONIC PRODUCTS

Some low-cost products use rigid PCBs like toys, electronic gadgets, desktop devices, and solid-state devices.

MANUFACTURING

Test equipment, electrical switches, industrial automation systems, control panels, industrial air conditioners, and CCTV surveillance systems.

MEDICAL DEVICES

Pacemakers, cochlear implants, handheld monitors, imaging equipment, drug delivery systems, wireless controllers, among others.

MILITARY

Weapons guidance systems, communication systems, GPS, aircraft missile-launch detectors, surveillance or tracking systems, and others.

AEROSPACE

Radar equipment, GPS, radio communication systems, control tower systems, sensors, noise and vibration testing systems, motion sensors, environmental and climatic test chambers.

TELECOMMUNICATION

Base stations, handheld units, communication satellites, wireless communication systems, signal processing systems, transmission media, routers and servers, online signal expansion systems and so on.

AUTOMOTIVE

Electronic control module, transmission controls, LCD displays, comfort control units, air conditioning systems, music systems, traction control systems, entertainment systems, and navigation systems.

Chapter 5

PCB Manufacturing Process

6.1 Introduction

PCBs are the foundational block of a majority of modern electronic devices. Electronic Components are mounted on that to communicate with each other through the printed circuit board tracks.

With the development of advanced technology in recent years, the PCB manufacturing process can now be fully automated. It consists of four steps: designing, manufacturing, assembly, and testing.

6.2 PCB Manufacturing Process Steps

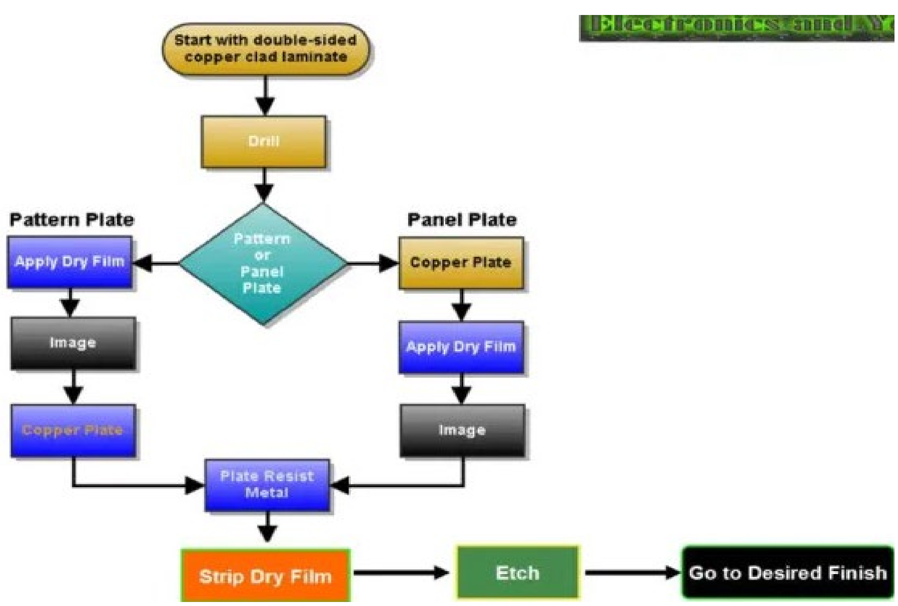


Fig.5.1 PCB Manufacturing Process Flow Chart

Step-1: Patterning or Etching

Majority of printed circuit boards are manufactured by applying a layer of copper over the entire surface of the PCB substrate material either on one side or both sides. This creates a blank printed circuit board, with the copper everywhere on the surface. From here the unwanted copper is removed by subtractive methods.

Fig.5.2: Copper Clad Laminate

Step-2: Photoengraving

The photoengraving process uses a mask or photomask combined with chemical etching to subtract or remove the copper areas from the circuit board substrate.

The photomask is created with a photoplotter which takes the design from a CAD PCB software. Lower resolution photomasks are sometimes created with the use of a laser printer using a transparency.

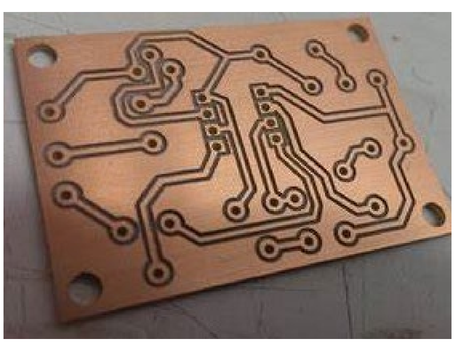


Fig.5.3: Photoengraving

Step-3: Lamination

Many printed circuit boards are made up of multiple layers; these are referred to as **multi-layer** printed circuit boards. They consist of several thin etched boards or trace layers and are bonded together through the process of lamination.

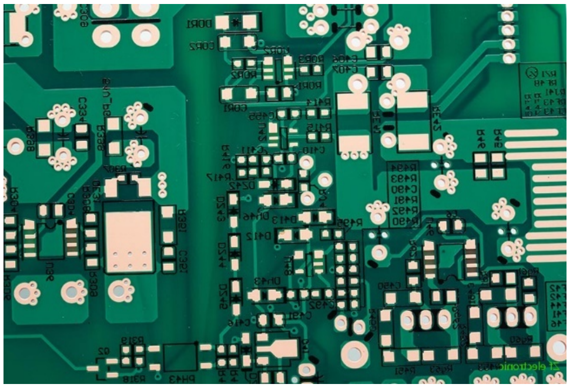


Fig.5.4: Lamination

Step-4: Drilling

Each layer of the printed circuit board requires the ability of one layer to connect to another, this is done through drilling small holes called “*VIAS*“. These drilled holes require precision placement and are most commonly done with the use of an automated drilling machine. These machines are driven by computer programs and files called numerically controlled drill or NCD files also referred to as excellon files. These files determine the position and size of each drill in the design.

Controlled depth drilling can be used to drill just one layer of the circuit board rather than drilling through all the layers. This can be accomplished by drilling the individual sheets or layers of the PCB prior to lamination.

- **Blind Vias**: When the holes connect a layer to the outside surface
- **Buried Vias**: When the holes only connect interior layers and not to the outside surface.

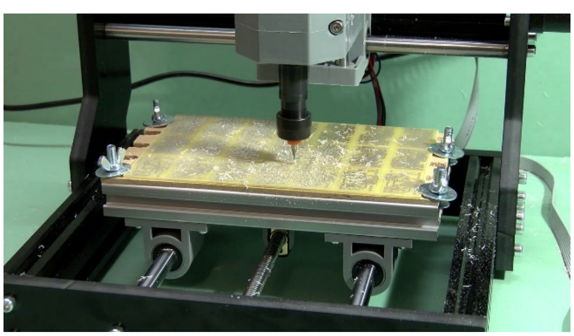


Fig.5.5: Drilling

Step-5: Solder Plating (Solder Resist)

Pads and lands which will require electronic components to be mounted on are plated to allow solderability of the components. Bare copper is not readily solderable and requires the surface to be plated with a material that facilitates soldering. In the past a lead based tin was used to plate the surfaces, but with RoHS (*Restriction of Hazardous Substances*) compliance enacted newer materials are being used such as nickel and gold to both offer solderability and comply with RoHS standards.

Areas that should not be solderable are covered with a material to resist soldering. Solder resist refers a polymer coating that acts as a solder mask and prevents solder from bridging traces and possibly creating short circuits to nearby component leads.

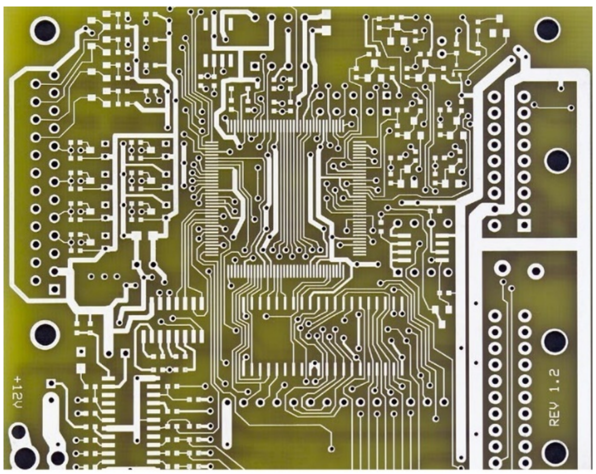


Fig.5.6: Drilling

Step-6: Silk Screen

When visible information needs to be applied to the board such as company logos, part numbers or instructions, silk screening is used to apply the text to the outer surface of the circuit board. Where spacing allows, screened text can indicate component designators, switch setting requirements and additional features to assist in the PCB assembly process.

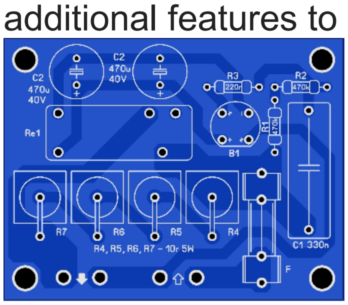


Fig.5.7: Silk Screening

Step-7: Testing

Unassembled circuit boards are subjected to a bare board test where each circuit connection is verified as correct on the finished circuit board. In high volume circuit board production, a bed of nails tester or fixture is used to make contact with the copper lands or holes on one or both sides of the board to facilitate testing. Computers are used to control the electrical testing unit to send a small current through each contact point on the bed of nails and verify that such current can be detected on the appropriate contact points.
For small to medium volume production runs, a flying probe tester is used to check electrical contacts. These flying probes employ moving heads to make contact with the copper lands and holes to validate the electrical connectivity of the board being tested.

Fig.5.8: Automated PCB testing

Step 8: Packaging and Shipping

This stage is about packaging and shipping the PCBs to their intended destinations. The standard packaging design protects PCBs from dust and other environmental factors. However, packaging may change based on the customer's specifications and also the PCB manufacturer.

Fig.5.9.: Bare PCB Packing

Chapter 6
PCB Manufacturing defects & their description

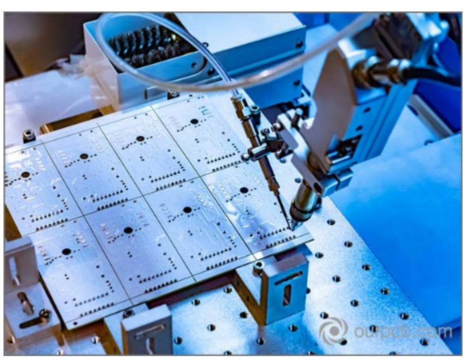
6.1Introduction

Fault minimization is the topmost priority of PCB manufacturers. To do that, understanding common faults and following PCB manufacturing guidelines is necessary. If you cannot prevent cheap PCB manufacturing, the final product will be subpar and prone to malfunctioning. May lead to rework, waste time, and materials. and increase production costs.

Things can always go wrong in manufacturing. When this happens, it’s important to know where the error or defect took place and how to fix the problem. Pinpointing the specific reason for failure can be difficult, so it is important to understand the most common PCB defects and their impact on product quality.

6.2 Significant errors in the PCB manufacturing process

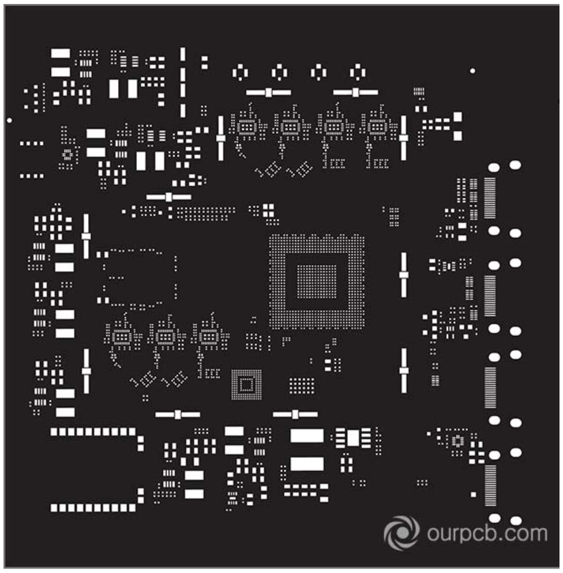
a) Plating Hole Defects



If the copper deposition is incorrect, it will create plating voids, leaving gaps in the wall with no copper coating. It could happen due to air bubbles, contamination in the holes, contaminated materials, and other similar reasons. You can prevent this from happening by following the manufacturer’s direction and cleaning the equipment as instructed.

Fig.6.1: Holes in PCB fabrication

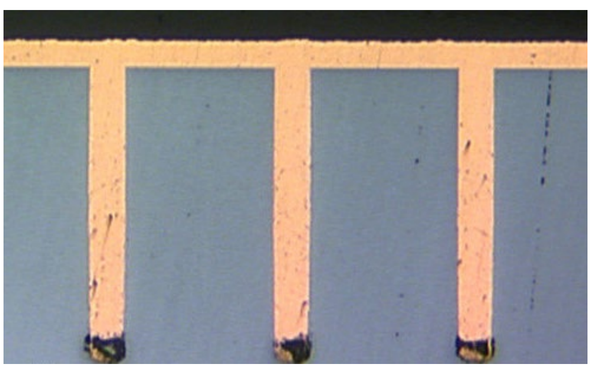
a) Lack of Solder Mask Between Pads



It is a common problem that accompanies PCB manufacturing . This defect occurs when one solder crosses one lead to the other, and ends up with an abnormal connection with multiple traces. If these shorts go undetected, they can damage the complete assembly by burning up the components. One standard solution is adding a solder mask between pads, so there is no gap between the stencil

Fig.6.2 : Lack of Solder Mask Between Pads

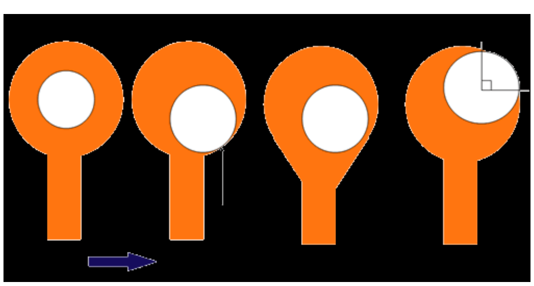
b) Plating Voids



Plating voids happen when the material does not coat evenly during the deposition process. This may be because of contamination of the material, air bubbles caught in the material, insufficient cleaning of the holes, insufficient catalyzation of the copper, or rough hole drilling.

Fig.6.3 : Plating Voids

d)Insufficient Copper-to-Edge Clearance



Since copper is a conductive metal, soft and vulnerable to corrosion, the copper will be covered with other materials to prevent the corrosion from occurring. However, if copper is too close to the edge after trimming, the copper layer will be exposed and can cause Shorts, corrosion.

Fig.6.4a: Insufficient Copper-to-Edge Clearance

c) Blowholes

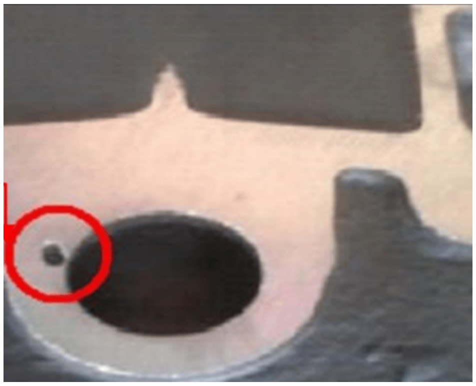


Fig.6.4b: Plating Voids

These holes usually show up at via locations and may extend into or through the entire length of the PTH. The cause of the blowhole may be gas attempting to escape due to internal moisture build up.

f) Missing Solder Mask Between Pads

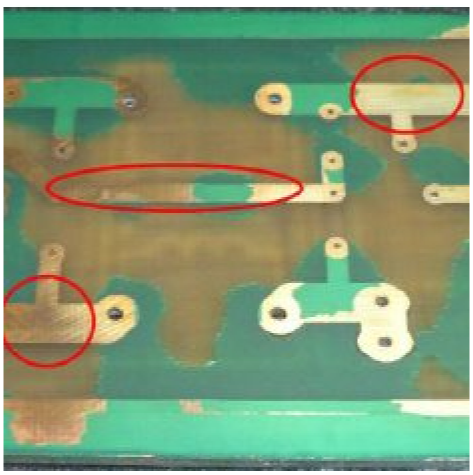


Fig.6.5: Missing Solder Mask Between Pads

The solder mask layer insulates the copper layer and prevents contact between the copper and other metal, solder, or conductive bits. It also prevents corrosion of the copper layer. When this layer is missing between pads, the copper can be exposed and may result in a short or corrosion, which will impact the functionality and longevity of the PCB.

g) Acid Traps

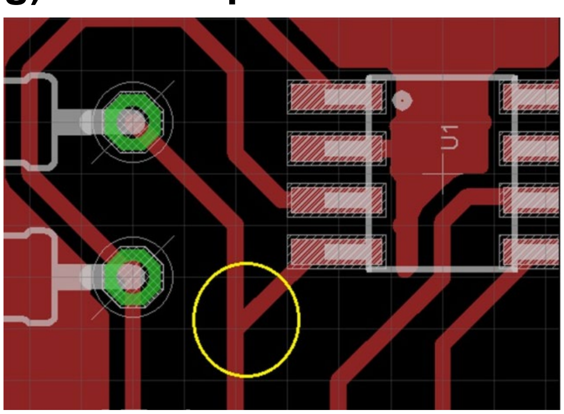


Fig.6.6 : Acid Traps

Acid traps are pocket spaces on the PCB, usually sharp corners, in which etching solutions could get trapped. These etching solutions are used to strip excess copper from a board during manufacture. If they are trapped, there is a risk of it tunneling through the board, causing corrosion to the traces and leading to faulty traces.

h)Plating Hole Defects

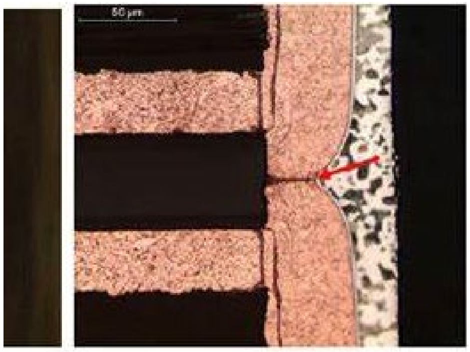


Fig.6.7 : Acid Traps

Holes carry electricity from one side of the board to the other. Plating of the hole wall is done during manufacture. During this, If the copper deposition is incorrect, it will create plating voids, leaving gaps in the wall with no copper coating. It could happen due to air bubbles, contamination in the holes, contaminated materials, and other similar reasons.

h) Lack of Solder Mask

It is a common problem that accompanies PCB manufacturing at home or production PCB manufacturing. The error is small, therefore challenging to identify. If these shorts go undetected, they can damage the complete assembly by burning up the components. One standard solution is adding a solder mask between pads, so there is no gap between the stencil and PCB.

i) Copper Foil Etching Defects

Sometimes a part of the substrate' copper foil is etched away due to the restrictions because of changing the substrate for producing dimensional changes when stress is relieved. The best solution for the problem is during the process of designing the circuit board, it must be tried to make the board in even distribution. If this is not possible, then some space for transition must be left which would not affect the circuit. This is due to the reason that plank structure of the cloth of glass and density difference of weft is leading to the differences in intensity of longitude to latitude of plank.

j) Electromagnetic Compatibility Issues in Cheap PCB Manufacturing

Electromagnetic interference (EMI) and Electromagnetic compatibility (EMC) are two standard terms linked with the PCB manufacturing process. The former term is generally used to produce and transmit electromagnetic energy, whereas the latter one talks about the damaging effect of EMC. These issues arise because of possible design flaws and can be decreased simply by lowering the board's ground area.

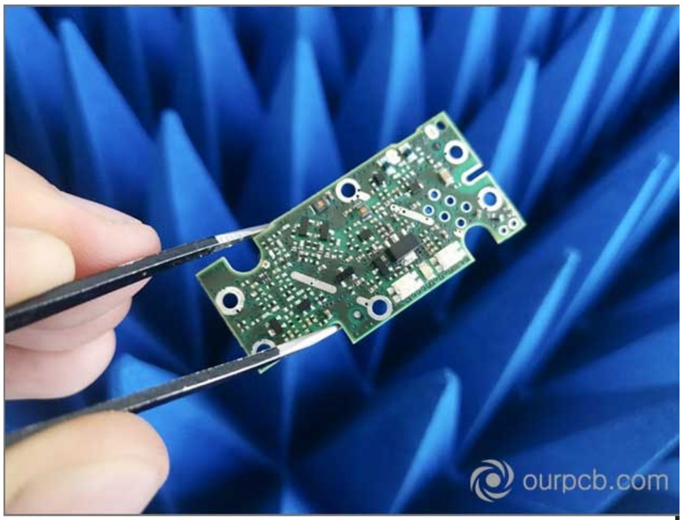


Fig.6.8: PCB Board in front of Electromagnetic compatibility measurements absorbers

K) Burned Circuit Defects in Cheap PCB Manufacturing

When the PCB passes high temperatures during the manufacturing process, circuits usually can get burned, if there is not enough space around the component. To avoid this problem. It's critical to ensure that components have adequate space around them to allow air to circulate.

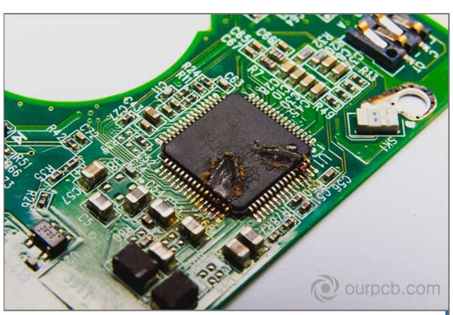


Fig.6.9: Blow up IC on circuit board after short circuit

l) Chemical Leak Problem in Manufacturing Of PCB

Corrosion of PCB boards is a very troublesome problem. Compared with the issue of PCB burnout, their appearance is different. But the result is the same, and it will also cause the PCB not to work properly.

To prevent this issue, thoroughly clean the board, as the smallest residue can cause corrosion of the boards and trigger a short circuit.

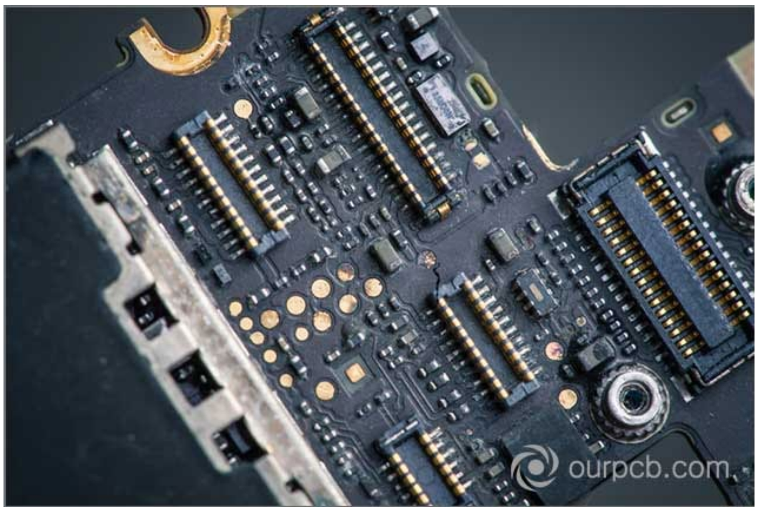


Fig.6.10: PCB board corrosion

j) Change in Substrate Size

Substrate size stability should also be excellent for substrate material. The size of the PCB substrate can be changed for the following reasons:

- The release of the shearing effect causes the substrate to shrink in size.
- Tension causes the size of the base plate to change.
- The Multi-layer board lamination process results in dimensional changes.

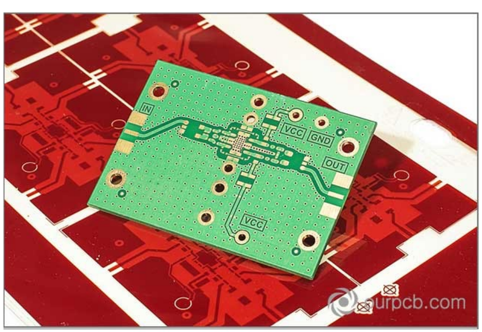


Fig.6.11: PCB board corrosion

k) Bending and Warping in SMT PCB

We get distressed when the PCB is bent. If we want to restore the original shape, some components will be damaged, and we will lose many costs. Here are the common issues in PCB prototype manufacturing or multilayer PCB manufacturing:

- Cools too quickly after hot melt, causing shape changes
- Substrate curing
- Different thicknesses of copper foils and different structures cause changes in shape.

l) Problems with Drilling in Cheap PCB Manufacturing Services

Problems observed/Solutions are:

- Hole deviation (Increase the drilling speed or lower the feed speed rate.)
- A white ring appears on the edge of the orifice (When thermal stress causes a fracture, adjust the abrasion, and replace the bit).
- Rough wall (Re-sharpen the drilling bit before use and follow manufacturer direction.
- The wall of the hole is too large and exceeds the standard size (use the recommended drill bit size).
- Holes are not fully penetrated (Set the drilling program according to manufacturer suggestion, before using it).

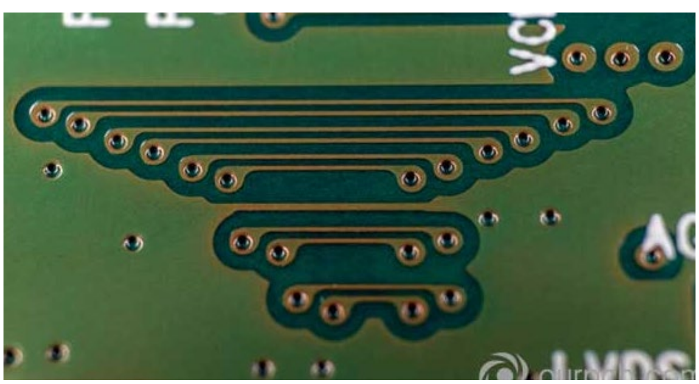


Fig.6.12: PCB board corrosion

m) Defects on Copper Substrate Surface

Problem description:

- The copper surface is uneven
- There is glue on the surface pits
- A gluing point on the surface of the substrate

Fig.6.13: PCB board corrosion

n) PCB Board Has White Spots

White spots could quickly generate if lead-free or contamination of the RMA solder happens. Changes in the materials used can cause white spots. If white spots are evenly distributed, this is usually from using the wrong solvent for the flux. For example, if a heavy-duty flux remover is used on a no-clean flux – this will leave white residues.

The cleaning process is also crucial in the case of white spots The ideal guideline is following the four steps of the cleaning process. Start with wetting, then scrubbing, then rinsing and lastly drying.

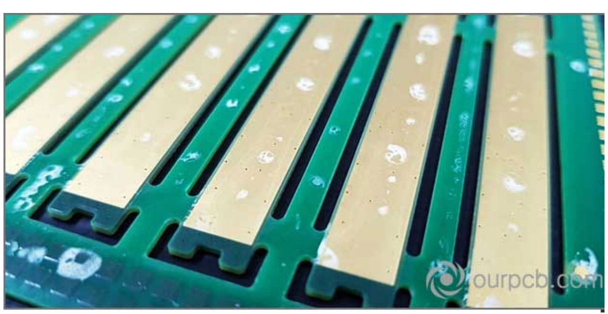


Fig.6.14: PCB Board with White Spots

s) Solder Webbing

Solder mask webbing can prevent shorts or solder bridges between adjacent pins, typically for multi-pin ICs. Solder webbing, which is slightly different, is caused by pollutants and can present problems with the solder joint quality.

Chapter :7

Bare Board PCB Testing

8.1 Introduction:

Bare board PCBs represent the pre-assembled state of a PCB before components like microcontrollers, logic ICs, transistors, and regulators are soldered onto it. When bare board PCBs are fabricated based on a design, they will have vias, pads, through-holes, component designators, and electrical paths formed by copper traces.

PCB manufacturing is a complicated process where copper layers are etched and hundreds of holes are drilled according to a design. Even with the aid of machines, the results aren't always perfect. Occasionally, you'll have issues on a few PCBs that go undetected until they are assembled and deployed.

While bare board PCBs are a necessity for mass volume production, they are also a preferred option for prototyping. *Bare board PCB testing helps detect post-fabrication issues.*

A bare printed circuit board has no through-holes or electronic components. This board comprises a substrate, conductive pathways and patterns, solder mask, and metal coating. These conductive pathways and patterns direct the flow of electricity in the circuit board.

The PCB serves a lot of functions which includes providing mechanical support to electronic components, the traces, and pads.
An Etching process helps to form the conductive traces in these pathways.

A bare PCB is also known as etched wiring boards of printed wiring boards. Printed circuit boards mean bare boards or blank boards. Contract electronic manufacturers and OEMs attach electronic components on this bare boards to produce printed circuit assemblies (PCBAs). A printed circuit board assembly is different from a printed circuit board. PCBAs feature several electronic components and through holes while PCBs have none of these.

Bare PCBs are manufactured with the sole purpose of populating them with electronic components. A bare PCB is needed for the manufacturing of a PCBA. The presence of electronic components on a PCB makes it a functional PCBA. Also, the bare board is the foundation of a printed circuit board assembly.

The absence of electronic components on the bare board helps in conducting a wide range of tests. These tests ensure suitability of the desired PCB design, and as well evaluate the bare board to correct any preference problem.

8.2 Testing Bare Board PCBs

Testing a bare board is crucial before approving the board for final production. Some boards can be defective and early detention of any defects helps in reducing cost and time spent on manufacturing. Manufacturers need to test this bare board to verify its functionality. The best way to guarantee the functionality of your bare is board through testing.

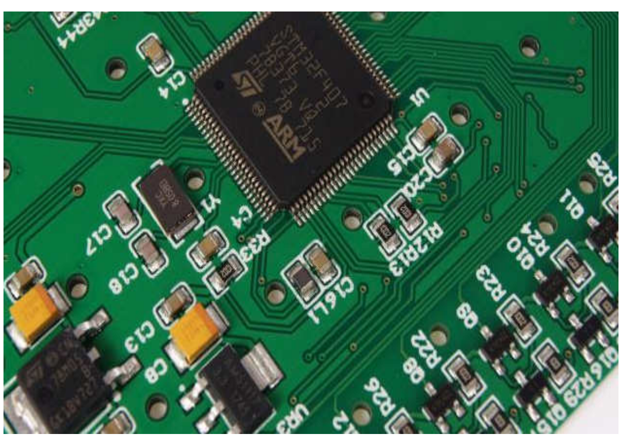


Fig.7.1: PCB Board with White Spots

Bare PCB testing is carried out to verify the isolation and continuity of a bare board's electrical connection. The manufacturer verifies resistance between two separate electrical connections during isolation testing. On the other hand, continuity testing ensures there are no open points on a circuit.

8.2.1 Pinned Fixture Test

This is also known as Bed of Nails and In-Circuit testing, and is suitable for large scale manufacturing. The method tests every connective surface on the PCB board at the same time, using spring-loaded pins or probes. Top and bottom plates apply pressure to a custom pin fixture to ensure a connection across the board.

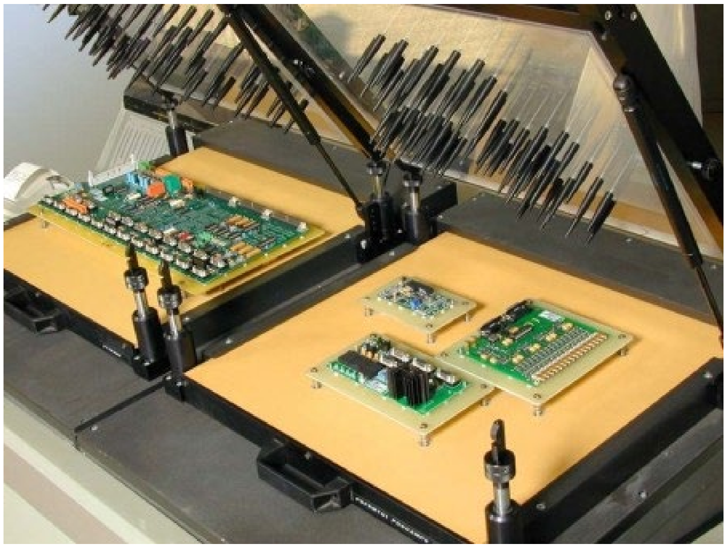


Fig.7.2: Pinned fixture test Apparatus

The pinned fixture test is highly efficient. It is very easy to carry out and only takes a few seconds to perform. However, there are some downsides like lack of flexibility and high expenses since each PCB design needs the creation of a new fixture and as well as a custom array of pins.

This test involves developing a fixture containing multiple pins that match the number of nets on the circuit. It means that you'll have two bed-of-nails fixtures pressing the PCB from the top and bottom, with the hundreds of probes making a connection with the testing points of the PCB. It's a very fast testing method, but expensive, and it must be remade if the board undergoes any changes after the production of the fixture.

This test uses two or more probes to test all the nets on the PCB. The probe moves along the X-Y axis according to the layout programmed on the testing panel. There is no need to develop a fixture for the flying probe test, making it the cheaper option. However, the flying probe test is comparatively slower.

8.2.2 Flying Probe Test

Fig.7.3:Flying Probe Tester

Fig. 8.1: Flying Probe Test

The flying probe test is a common way of testing bare circuits. This test makes use of two or more robotic arms. These robotic arms have poles which glide across the PCB surface to test each net and verify the connections of the circuit board. Also, the flying probe testers take instructions from software application to travel across the connections. These testers check for capacitance, diode, and inductance issues. They also check for any open circuit or short circuit.

At first, the flying probe test was only used to test the bare board (board without components). After some technological advancements, now the test is used for bare boards as well as assembled PCBs. It can detect short circuits, open circuits, wrong component placement, reverse polarity, and component failure.

7.2.3 Conclusion

Bare PCBs are manufactured with the sole purpose of populating them with electronic components. A bare PCB is needed for the manufacturing of a PCBA. There is a higher risk of producing dead boards if a manufacturer fails to test a board. A bare PCB must go through stringent tests before components can be assembled on it.

Chapter 8:

Bare Board PCB Repair Techniques

8.1 Precautions to be Taken to Avoid Mechanical and Electrical' Damage during Repairing

Careless handling of PCBs and hand tools can lead to further damage during the repair process. The following precautions should be taken:

a) Remove the board from the equipment during the repair process.
b) Great care should be taken to avoid mechanical damage to the board when removing the component which has been stuck to the board. Cleaning solvents must be used carefully to avoid damage to the components on the board.

c) Mechanical damage to the copper loll is most likely to occur when stress Is applied to the component leads in a direction that would force the copper away from the board. Avoid applying such a force.

d) During the soldering process, when the joint is at the soldering temperature, the strength of the bond between the track or land and the board is less. So, the soldering process must be correctly carried out and the joint should be allowed to cool and regain the strength of the bond.

e) Excessive heating of the board should be avoided and care must be taken to avoid unnecessary damage to any protective coating.

f) In multilayer boards, care must be taken during repairs to avoid damage to any of the PTH, as these interconnect the internal conductors.

g) While replacing the components, it must be checked to see whether special handling techniques are required. If so, those instructions must be followed strictly. The components which are sensitive to static electricity, should be handled carefully so that they are not damaged.
h) The components can easily be damaged, if leads are bent very close to the body.

8.2.2 PCB PAD AND TRACE REPAIR

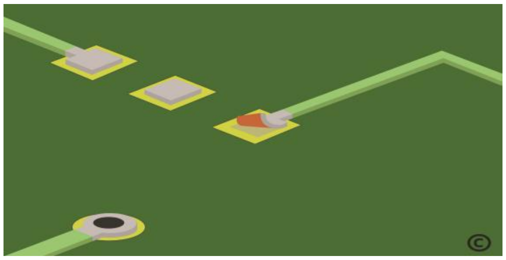


Fig. 8.1

1.The board area requiring repair is first cleaned in and around the pad/trace to be repaired. See Fig. 8.1

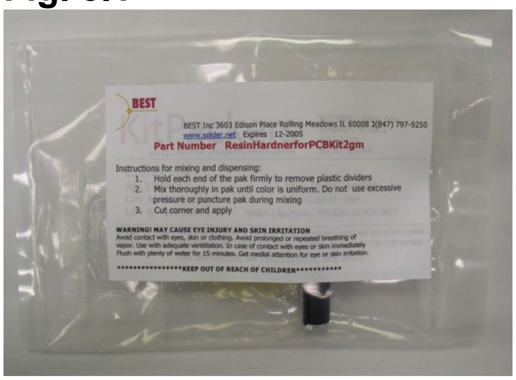


Fig. 8.2

2. This epoxy (Fig. 8.3), is designed to withstand the chemicals, heat and can adhere to materials common in PCB assembly. It has been used since 1998 and can be used in a variety of applications including but not limited to pad and trace repair, solder mask repair, baseboard material repair as well as numerous others. The mix ratio is already taken care of for you in the pre-mixed bags and therefore is easy to remember and easy to mix. It is a 3:1

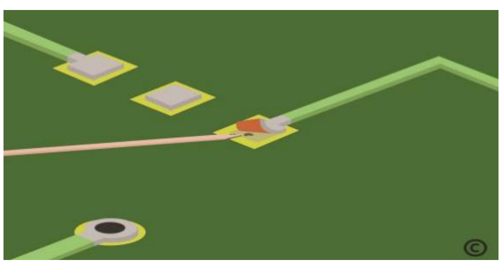


Fig. 8.3

3. The board area requiring repair is first cleaned in and around the pad/trace to be repaired.

4. Carefully apply a small amount of epoxy **(Fig 8.3)** under the entire length of the lifted circuit. The tip of an orange wood stick may be used to apply the epoxy.

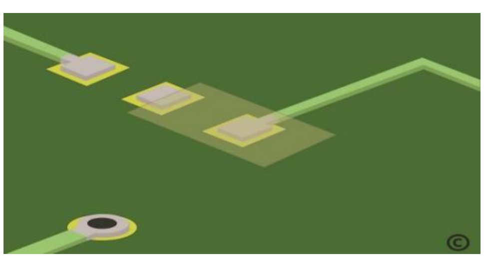


Fig. 8.4

5. Press the lifted pad/trace down into the epoxy and into contact with the board.

6. Apply additional epoxy to the surface of the lifted circuit and to all sides as needed for additional strength. Tape the repaired pad/trace into position while curing **(Fig 8.4)**

7. Cure the BEST epoxy per the instructions.
8. Re-apply conformal coating to match prior coating if it required.
9. Perform a visual examination per IPC A-610 acceptability criteria or customer requirements.
10. Perform a tape test per IPC-TM-650.10.Perform continuity and other electrical tests as applicable.

8.2.3 Gold Finger Repair

Gold fingers are prone to damage as they are on the periphery of the circuit board and they are also prone to solder splash during the wave soldering process. When edge contacts are contaminated with solder, get scratched, torn or damaged, technicians can bring them back into specification. These contacts can either be replaced or repaired. The repair methods will restore your edge contacts to a high level of performance and reliability.

When the gold fingers get scratched, contaminated with solder or if the plating is worn, they can be re-plated. The contaminated contacts are stripped down to the bare base metal and replaced to meet the thickness specified. Our electroplating process is reliable and will restore damaged contacts to "like new" condition with full functional conductivity and durability.

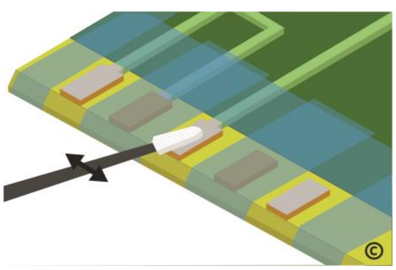


Fig 8.5

Generically for a replating process the first step is to clean and inspect the area to replated. At this point BEST will determine whether or not a replating process will be appropriate based on the extent of the damage or splash on the gold-plated surface (**Fig 8.5)**

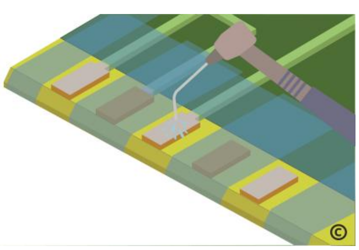


Fig 8.6

Areas in and around those with solder splash are taped off and the solder is wicked off and then a solution helps to strip off the remnant solder from the surfaces. If a replacement pad is required then a new gold-plated pad is installed instead of spending time re-plating the existing pad (**Fig 8.6)**

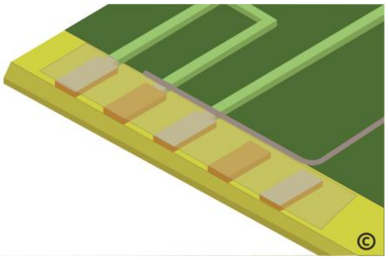


Fig 8.7

Areas in and around those with solder splash are taped off and the solder is wicked off and then a solution helps to strip off the remnant solder from the surfaces. If a replacement pad is required then a new gold-plated pad is installed instead of spending time re-plating the existing pad (**Fig 8.7)**

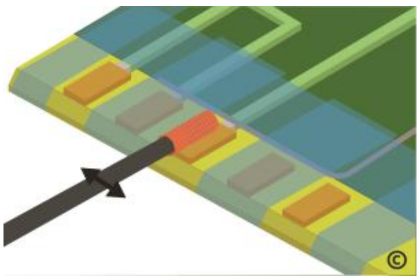


Fig 8.8

A bus wire is then attached in close proximity to the re-plating area. Using a small power supply with swabs connected to the electrodes the plating process is undergone using a nickel (if required) then gold plating solution. After plating the area is inspected and tested **(Fig. 8.8)**

8.2.4 SOLDER MASK REPAIR PROCEDURE

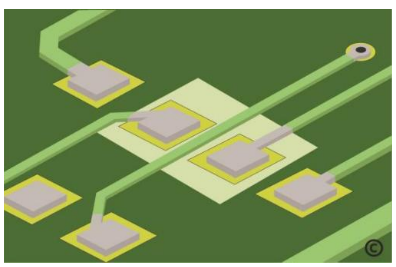


Fig 8.9

1. Make sure that the area to have the solder mask repaired is cleaned and inspected (**Fig 8.9)**

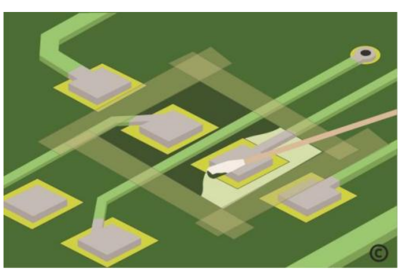


Fig 8.10

2. If required, apply high temperature tape in order to outline the area where the replacement solder mask will be applied **(Fig 8.10)**

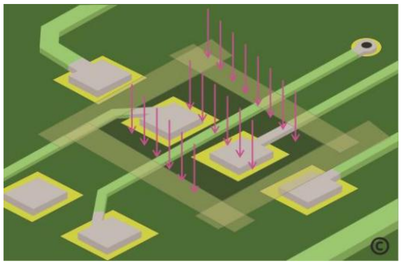


Fig 8.11

3. using the replacement UV-curable solder mask material apply this to the areas requiring repair. If desired, add color agent to the mixed epoxy to match the circuit board color **(Fig 8.11)**

4. Cure the replacement solder mask by setting the PCB underneath the UV curing lamp as per the manufacturer's recommendations. Watch out for heat sensitive components.

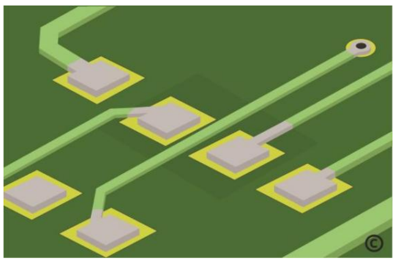


Fig 8.12

5. Perform a visual inspection for color match and adhesion per the IPC TM-650 specifications **(Fig 8.12)**

6. Inspect per the latest IPC-A-610 requirements.

8.2.5 REPAIR OF A CIRCUIT TRACK

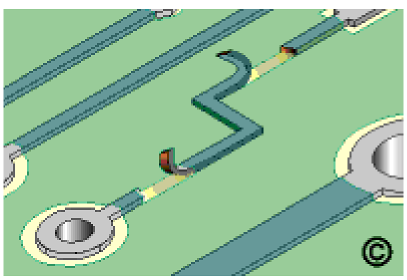


Fig 8.13

1. Evaluate damage and measure conductor width (Fig 8.13)

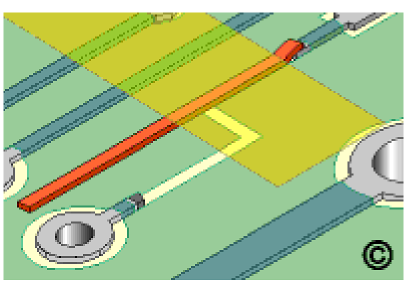
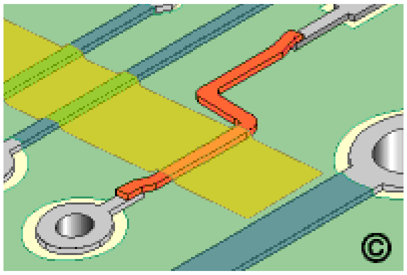


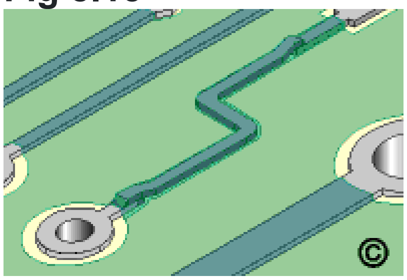
Fig 8.14

2. Remove damage, place new track and solder



3. Form circuit track to match conductor path.

Fig 8.15



4. Make final bends and solder
5. Bond in place and overcoat with epoxy

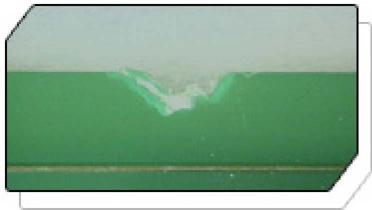
Fig 8.16

8.2.5 PCB Corner and Edge Repair

Corners and edges are often damaged by poor handling procedures. When the base laminate delaminates, blisters, measles, or is routed improperly. PCB REPAIR SECTION technicians can repair these defects. When this happens to critical prototype, development or high value boards such as servers PCB REPAIR SECTION repair technicians can make repair that laminate so that it is returned to "as new" condition.

Whether it is a minor or major repair, PCB REPAIR SECTION technicians use a high strength, thermoset epoxy, along with the skill set to make sharp corners after rebuilding or consistent edges to bring the laminate surface back into specification.

PCB REPAIR SECTION technicians use a variety of different IPC repair techniques along with the "tricks of the trade" they have learned over many years in order to accomplish high quality repairs. Only a trained eye could discern that a repair had been performed.



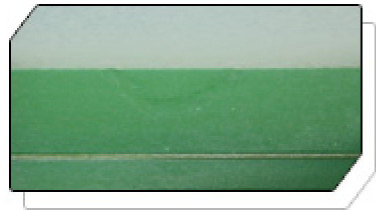


Fig 8.16(a): Damaged PCB edge **Fig 8.16(a): Repaired PCB edge**

8.2.6 Replating Process

Generically for a replating process the first step is to clean and inspect the area to replated. At this point PCB REPAIR SECTION will determine whether or not a replating process will be appropriate based on the extent of the damage or splash on the gold-plated surface.

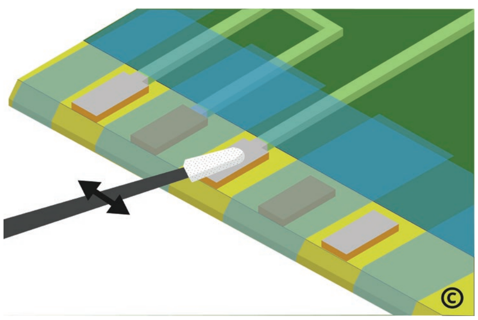


Fig 8.17

Areas in and around those with solder splash are taped off and the solder is wicked off and then a solution helps to strip off the remnant solder from the surfaces. If a replacement pad is required then a new gold-plated pad is installed instead of spending time replating the existing pad.

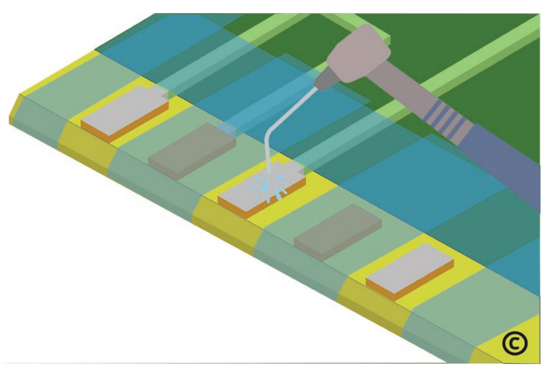


Fig 8.18

The solution remnants are then rinsed from the board. The board is then dried and inspected.

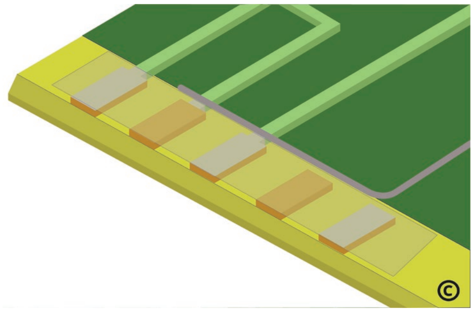


Fig 8.19

A bus wire is then attached in close proximity to the replating area. Using a small power supply with swabs connected to the electrodes the plating process is undergone using a nickel (if required) then gold plating solution. After plating the area is inspected and tested.

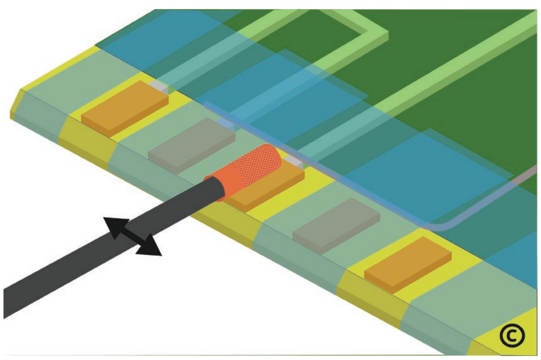


Fig 8.20

CHAPTER 9:
PCB Legislation-
(Environmental issues, ROHS)

9.1 Background:

The PCB Waste Generated decades ago will harm our environment in the years to come. These discarded materials consist of hazardous substances such as lead. Mercury, cadmium, and several other chemicals, that adversely affect life on Earth. There is a need for environmentally responsible manufacturing for RoHS – complaint PCBs across the entire consumer electronics sector in today's scenario.
RoHS (Restriction of Hazardous Substances) restricts the use of hazardous materials, particularly found in electrical and electronic components.
Electronic waste mostly consists of hazardous substances such as heavy metals and other chemical materials with absolutely no chemical control. The problem persists further as the acidic rainwater dissolves Lead and other harmful substances. This contaminated rainwater will be added to the water bodies. Thus, severely causing harm to aquatic life and human well-being.

9.2 ROHS-An Introduction

RoHS 3
COMPLIANT
EU 2015/863

In January 2003, the European Union Parliament passed two legislations that impacted the electronic industry. The first one was Waste Electrical & Electronic Equipment, commonly (WEEE) it is making manufactures responsible for the end-of-life disposal of their finished goods. The second one is more important which is the Restriction of Hazardous Substances (RoHS).

The RoHS directive makes it illegal to manufacture any electrical or electronic equipment that contains restricted materials, materials that are harmful to the environment. The legislation took full effect on July 1, 2006.

9.3 Restricted Materials

The RoHS compliance has restricted the use of eight materials in the PCBs and electronic items. They are Mercury (Hg), Lead (Pb), Cadmium (Cd), Hexavalent Chromium (CrVI), polybrominated diphenyl ethers (PBDE), Polybrominated biphenyls (PBB), and phthalates such as BBP, DEHP, BBP, and DIBP.

The compliance also specifies the maximum levels of these restricted materials in the PCBs and other electronics items.

1. Mercury (Hg): < 100 ppm

2. Lead (Pb): < 1000 ppm

3. Cadmium (Cd): < 100 ppm

4. Polybrominated Biphenyls (PBB): < 1000 ppm

5. Hexavalent Chromium: (Cr VI) < 1000 ppm

6. Polybrominated Diphenyl Ethers (PBDE): < 1000 ppm

7. Bis(2-Ethylhexyl) phthalate (DEHP): < 1000 ppm

8. Dibutyl phthalate (DBP): < 1000 ppm

9. Di isobutyl phthalate (DIBP): < 1000 ppm

10. Benzyl butyl phthalate (BBP): < 1000 ppm

9.4 Other impacts of RoHS

a) Component Solderability

The majority of components require the use of some type of plating, somewhat close to what is used for PCBs, to provide solderability and shelf life. Before the RoHS directive, the components, surface finish in the majority was tin-lead. The conversion to Pb-free has resulted in the dominance of 100 percent Sn as the solderable Surface finish.

It is not simple to switch from a tin-lead-plated part to a lead-free finish. It may seem relatively simple to move from tin-lead to 100 percent tin, but a 100 percent tin deposit is more vulnerable to line issues such as pH control rinsing. Its effect on the surface finish is also greater than tin-lead has ever been. Similarly, converting from base metal to an exotic such as nickel palladium gold would entail the procurement of certain pricey specialized measuring instruments.

Heavy metals like Lead and Mercury have an adverse effect on the human body organ and can disturb the entire central nervous system. These metals also affect the functioning of kidneys and the reproductive system. A higher intake of lead through contaminated water will reduce the reaction time of body organs such as finger, wrists, and ankles.

RoHS initiatives will provide necessary relief for all of us, compliance with RoHS directives by PCB manufactures will support our fight against climate change and global warming

b**) Does adhering to RoHS standards lower the standard of quality?**

RoHS is not a compromise. In fact, the solder and other lead-free components used can result in a better product lifespan. This is because of the toxicity of the lead, which can leach out from the solder and other components into the PCB itself, gradually weakening it over time. Research into lead-free alternatives to traditional solder was only reasonably successful at first, but today, mixtures that include copper, tin, and silver have been proven to be reliable as well as environmentally safe. The fact that a fit for purpose alternative to lead has been found has meant that an ever-increasing number of EMCs are choosing to embrace RoHS standards, both for environmental concerns and to trade within the European Union.

9.5 WEEE Regulations

WASTE ELECTRICAL AND ELECTRONIC EQUIPMENT
WEEE

New directives under the waste electrical and electronic equipment (WEEE) and restriction and use of certain hazardous substances (RoHS) regulations have dictated that components and assemblies must be ‘lead free’. That is in this context defined as less than 0.1% lead by weight.

The WEEE directive was introduced into UK law in January 2007 by the Waste Electronic and Electrical Equipment Regulations 2006. Key aims are to reduce the amount of electrical and electronic equipment being produced and to improve the environmental performance of businesses that manufacture and supply such equipment.

9.6 Features of RoHS-Compliant Assembly Services

Today, many PCB assembly services are utilizing processes and technologies that help them manufacture RoHS-compliant PCB assemblies. The following are a few features that help distinguish these services from others.

• **Lead-free Surface Finishes:** Many manufacturers are nowadays using lead-free surface finishes such as lead-free HASL, ENIG, Electroplated Nickel and Soft (Bondable), Hard Gold Immersion Tin (White Tin), and Organic Solderability Preservatives (OSP).

• **RoHS Compliant Substrate Materials:** The choice of RoHS material depends on the requirement of the application. There are materials such as Isola IS410, and Polyclad 370HR, that assure standard electrical performance. However, materials such as Isola FR415 and Polyclad Gutek assure high electrical performance at low loss.

• **Primary and Secondary Services:** It has been observed that many RoHS PCB assembly service providers specialized in primary and secondary services such as lead trimming, lead-free wave soldering, and many more.

9.7 Benefits of Using RoHS-Compliant PCBs

The benefits offered by lead-free and RoHS-compliant PCBs are various. The following are a few of them:

• **Helps Reduce Metal Poisoning:** As the technology landscape is fast evolving, many customers are discarding their obsolete electronic equipment in landfills. Equipped with various hazardous materials, this equipment led to severe poisoning. Although recycling is being performed, still the equipment may have hazardous substances in them. The RoHS directive has forced OEMs reduce their dependence on hazardous substances. This has helped reduce the impact of these materials on the environment, and people working with them.

• **Improved Product Safety:** With RoHS compliance being adopted by many prominent electronics manufacturers in EU and the US, now users can be assured that the product they are using is lead and mercury free. As a result, the sales and popularity of the products from these manufacturers has tremendously increased.

• **Improved Thermal Properties:** It is proven that lead-free PCBs have excellent thermal properties than their leaded counterparts. They can easily withstand temperatures between -45°C to 145°C. Nowadays, PCB manufacturers are using specialized halogen-free laminates that have further helped improve their thermal stability up to 300 °C.

Chapter 10:
Latest trend in PCB

10.1 Introduction:

Both consumers and businesses demand more innovative products with advanced technologies and abilities. Therefore, there is a rapid progression and development in electronic technology. Printed Circuit Boards, or PCBs, are one of the most necessary components of every electronic device in today's world. Launching any technology involves PCBs, and PCBs should adapt to the rising needs of the technologically advanced world. Electronic Device should keep these trends in mind when designing their products.

10.2 The Future of Printed Circuit Boards

The PCB manufacturing process presents adequate room for growth. Manufacturers worldwide are faced with various challenges in their daily activities. The following trends will shape the future of PCB:

1. Wearable Technology

Currently, wearable gadgets are widely used and are expected to hit new levels soon. These gadgets seem to be shifting towards smaller, more powerful equipment every year. They are mostly implanted into clothing and other flexible accessories, like smart shoes and wristbands.

Fig.10.1: Wearable Technology

2. High-Speed Capacities

In the era of 5G, the need for Speed has greater impact on the PCB design. To meet this growing demand for speed, Hardware design engineers must ensure that PCB Design, Materials and Manufacturing improves the Signal Integrity of the design. PCB designers should understand the impact of Dielectric material, vias, trace length, cross talk, impedance discontinuities on the Signal Integrity and design their PCB. PCB Designers should also make use of new features in EDA tools that helps design right the first time.

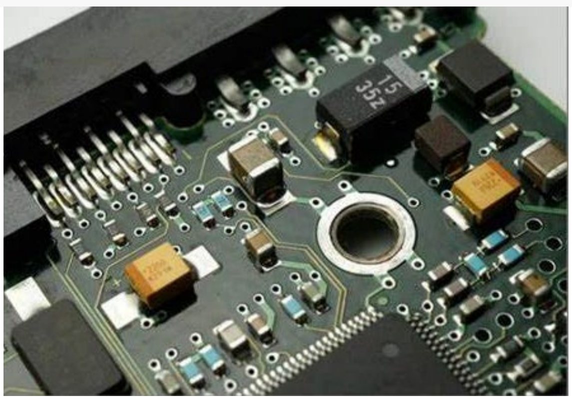


Fig.10.2: High-Speed Capacities

3. Camera Technology

Tiny cameras that are mounted into PCBs are transforming the way people use electronics. Equipped to capture high-resolution images and videos, these cameras are found in smartphones and medical equipment. The technology behind them is continually tweaked to make them smaller, more powerful, and efficient.

Fig.10.3: Camera Technology

4. Eco-Friendly Materials

Climate change is a trending topic globally, and the amount of electronic waste is a matter of concern. Most PCB materials are non-degradable and contribute to vast hips of the chunk in landfills. Bearing this in mind, environmentalists have recommended significant changes to PCB manufacturing – using biodegradable laminates and assembling with less harmful chemicals. Furthermore, authorities worldwide want the precious metals in electronic waste to be extracted and recycled for use.

5. High-Density Interconnect PCBs

There are rapid developments in automation in almost every industrial landscape. New advancements are common in every sector, from military communications and aerospace applications to medical diagnostics tools and wearable technology. Therefore, the need arises for HDI PCBs. High-density interconnects (HDI) PCBs are smaller and lightweight than other PCBs and offer high-speed and reliable signals.

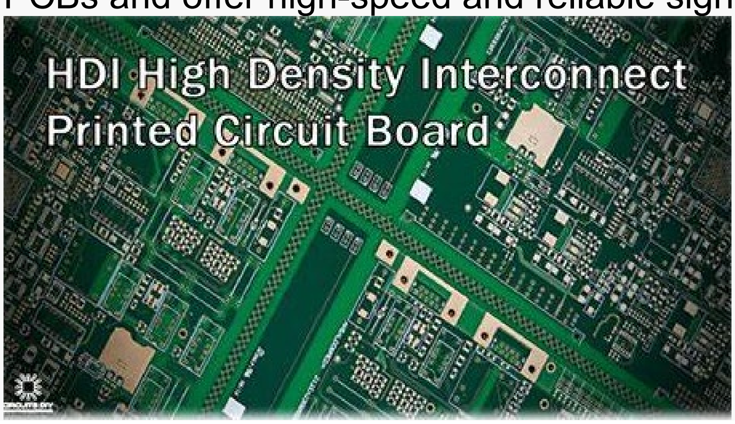


Fig.10.4: High-Density Interconnect PCB

6. Internet of Things (IoT)

Communication is necessary for all products, and it has been made possible through IoT in various ways. The Internet of Things defines the network of physical objects infused with software, sensors, and other technologies for exchanging and transferring data from one device or system to another over the internet. IOT parts are included while manufacturing PCBs, and any IOT gadget requires power, wireless connectivity, and sensors. To satisfy these criteria, it is essential to change the design of PCBs. For instance, smart watches are getting smaller; thus, they need to fit more functionality within a smaller space. Users also expect the devices to run for a long time, so the PCB designs should have long battery life.

Fig.10.5: IOT

7. High Power Boards

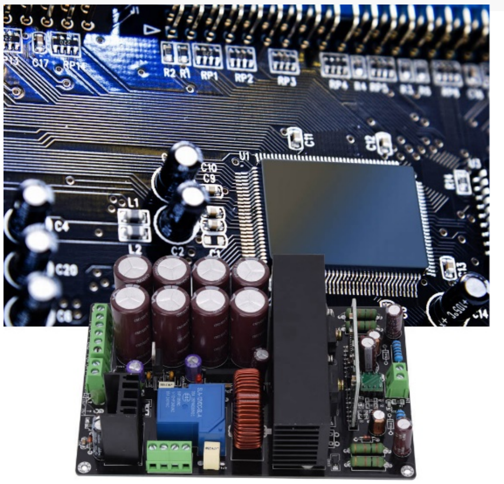


Fig.10.6: High Power Boards

The need for high power boards is increasing endlessly. This is due to the growth of electric devices, which demand high voltage boards like 48V and more elevated. There is also a great demand for solar panels that function at 24V or 28V.

8. Biodegradable PCBs

Fig.10.7:Biodegradable PCBs

Electronic waste problems create a global issue, and biodegradable PCBs are manufactured to minimize these issues. Previously, the problem of disposing of PCBs started with environmental issues as they had chemicals in them and were also undegradable. These issues were resolved when biodegradable PCBs came to be manufactured. Further, the metals such as gold, silver, palladium, gallium, and tantalum are extracted from the chips and reused.

9. Commercial-Off-The-Shelf (COTS) Components

The application of the COTS technique is considered to bring regulation and reliability to the parts used in essential space-based arrangements. Traditionally, electronic parts applied in space production undergo stringent scrutiny from various government agencies and quality inspection units. Nevertheless, the commercialization of this sector may minimize the regulation of space-based components.

Fig.10.8:COTS Components

10. High Power Boards (48V and higher)

There is a significant thrust towards higher power PCBs. This includes boards with up to 48V supplies. These voltage levels are in response to the growth in solar energy, where panels typically operate at 24V or 48V, and electric vehicles (EVs) where voltages may be in the hundreds. These high-power boards require PCBs to mount larger components like battery packs while being able to deal with interference issues effectively.

Fig.10.9:COTS Components

11. Conclusion

We have discussed the top PCB trends that will considerably shape the electronic industry in 2021 and beyond. These trends will continue the evolution of printed circuit board manufacturing technology. The future of the circuit board industry is luminous and will be shaped by product performance and miniaturization. Additionally, being environmentally conscious and regulating production expenses will play a big role in shaping the future of this industry.

PCBs today are tiny, multilayered, complex systems that hardly resemble their earliest ancestors. They're also produced at a much higher and more efficient rate than ever before thanks to sophisticated design software and manufacturing processes. Even 10 years ago, micro vias, HDI and FPGAs were only seen in the most expensive designs, yet are now readily available to designers worldwide.

As technology and consumer demand grows and develops, however, so must PCBs. As the basis of all electronic devices, PCBs feel intense pressure for development and growth. With consumers pushing for slimmer and faster devices, and with industries seeking improved functionality, the PCB must continue developing into the future.

Chapter 11:

Glossary of Terms- Printed Circuit Board Terminology

11.1 Introduction:

Having a basic understanding of printed circuit board terminology can make working with a PCB manufacturing company much faster and easier. This glossary of circuit board terms will help you understand some of the most common words in the industry. While this isn't an all-inclusive list, it is an excellent resource for your reference.

11.2 Glossary of Terms

<A>

A1 Active Components: This term refers to a type of component that is dependent on the flow direction of an electrical current. For example, a transistor, rectifier or valve would be considered active.

A2 ALIVH: Short for any layer inner via hole, this is a type of technology used to build multi-layer BUM PCBs. This method uses a solder to create an electrical connection between PCB layers. ALIVH often replaces traditional vias and is a useful production method for creating high-density BUM PCBs.

A3 Analog Circuit: It refers to circuits processing analogue signals (continuous and variable signal). The output is non-binary within this type of circuit.

A4 Annular Ring: This term refers to the copper pad area that is left after a hole is drilled through it. This ring is measured from the edge of the pad to the edge of the hole and is an important consideration in PCB design, as it allows an electrical connection to be made from one side of the hole to the other.

A5 Anti-Solder Ball: This type of technology is commonly applied in SMT production lines with the goal of limiting the amount of tin involved in the stencil process. This is done by making a stencil on the board and creating openings at places where the solder ball tends to be produced so that the tin paste will flow to the openings.

A6 AOI: Short for automated optical inspection, AOI refers to a type of inspection method used to find potential problems concerning soldering performance in multi-layer PCBs with components mounted on. The AOI equipment finds these issues by capturing images of the inner PCB surfaces, looking for any possible issues in terms of displacement, polarity etc.

A7 AQL: Short for acceptance quality limit, AQL refers to the acceptable number of defective boards produced within a production run. These are identified, counted and removed during inspection. AQL is an important figure for monitoring the quality of an assembler's production practices.

A8 Array: This word refers to the combination of multiple copies of the same PCB into a connected matrix of boards. An array may also be referred to as a panelised, stepped out or palletized PCBs. By assembling boards this way, the assembly process can be completed much more quickly. The Array # Up, in turn, refers to how many PCBs are included in the array.

A9 Aspect Ratio: Aspect ratio refers to the ratio between a PCB's thickness and diameter of its minimum via. It's best to keep aspect ratios low to improve plating quality and minimize potential via failures.

A10 Assembly: A process involving a series of procedures where components and accessories are placed on a PCB, resulting in a functional board.

A11 Assembly Drawing: An assembly drawing is a reference depicting the assembly requirements of a PCB. These drawings will usually include the placements of components as well as the construction technologies, methods and parameters needed to make it happen.

A12 Assembly House: A name used to refer to a manufacturing facility where PCBs and components are assembled. These houses will usually contain PCBA equipment such as a printer, mounter, reflow oven, and more.

<B>

B1 Back Drilling: Primarily applied in multi-layer PCB fabrication, back-drilling helps improve signal integrity by removing stubs from plated through-holes. These stubs are unnecessary portions of via that extend into the hole, potentially causing reflections and other disturbances that damage signals.

B2 Backplane: This is a supporting plane on a circuit board that plays an insulating role.

B3 BGA: Short for ball grid array, this is a type of component packaging used in integrated circuits (ICs) for surface mounting. They can ensure high-speed efficiency since they use columns of balls instead of pins. BGAs are usually used to mount devices like microprocessors on PCBs permanently.

B4 Bare Board: This term refers to a circuit board with no components mounted on it.

B5 Blind Via: A blind via is a through-hole that connects inner layers, but it can't be seen from the exterior of the PCB.

B6 Board: This is a shortened term for printed circuit board. This word also indicates the substrate upon which the PCB is printed. The board is an important electronic part, acting as a carrier for an electric connection between electronic components.

B7 Board House: This is another name for the facility where PCB boards are fabricated.
B8 Board Type (Single Unit and Panel): This indicates the manufacturing method of a PCB in terms of volume. Usually, a board is classified into one of two types: single unit or panel. In single unit manufacturing, PCBs are fabricated one by one. In panel manufacturing, on the other hand, multiple units of PCBs are manufactured in a single panel.
B9 Body: A word used to describe the central section of an electronic component. It does not include the component's pins, leads or accessory parts.
B10 Buried Resistance Board: The term refers to a printed circuit board with resistors buried inside. This design improves the integrity of resistant components to improve the overall function and reliability of the PCB.
B11 Buried Via: This term is used to refer to a via connecting a top layer to one or more inner layers. In other words, a buried via can only be seen from one side of the board when looking at it from the outside.

< C >
C1 Cable: Another word for a wire that is capable of transmitting electricity or heat.
C2 CAD: An acronym for computer-aided design, CAD refers to a designer's use of computer and pattern equipment to develop and implement a PCB layout. The result is a three-dimensional graphic of the design, which, in this case, is the layout of a PCB.
C3 CAE: An acronym for computer-assisted engineering that refers to schematic software packages used to develop and visualize PCB designs.
C4 CAM Files: CAM is an acronym for computer-aided manufacturing, and the files produced by this software are used for PCB manufacturing. There are multiple types of CAM files, including Gerber files for photoplotters and

NC Drill files for NC Drill machines. These files are usually sent off to board and assembly houses for refinement and eventual manufacturing.

C5 Carbon Mask: This is a type of conductive carbon paste that is added to the surface of a pad. Made with a combination of resin and carbon toner, carbon masks are heat-cured and are typically applied to jumpers, keys, etc.

C6 Ceramic Substrate Printed Board: This type of board is made with a ceramic substrate, to which other materials are bonded with alumina or aluminium nitride. The primary selling points for ceramic substrate boards are their excellent insulation capabilities, thermal conductivity, soft solderability and adhesive strength.

C7 Check Plots: This is a list of check items that are based on which quality control inspection or test is implemented.

C8 COB: Shorthand for chip-on-board, this term is a type of bare chip SMT technology. COB involves directly mounting integrated circuits to a PCB instead of packaging them first. Common in mass-produced gadgets and toys, COB can be identified by a black glob of plastic on a PCB, called a glob top. Underneath the glob, the chip connects to the board with fine wires.

C9 Circuit: It refers to a conductive loop composed of metal leads and electronic components. It falls into one of two categories: DC circuits and AC circuits.

C10 Coating: A coating is a solid continuous film that either protects, insulates or decorates the PCB.

C11 Component: Alternatively called electronic components or parts, components are basic pieces that can be used to build electronic equipment and devices. Examples include resistors, capacitors, potentiometers, valves, radiators, etc.

C12 Component Hole: This is a plated hole in a PCB that is made for a component. These holes are intended to facilitate either a component pin, termination or wire with an electric connection.

Component Library: It's a collection of components as represented in a CAD software system. It's stored in a computer data file for later use.

C13 Component Side: This refers to the side of a PCB that contains components. The opposite side contains soldering points for components.

C14 Connector: This term refers to a transmitting component that connects two or more active components in an assembly. Usually, connectors consist of a plug and receptacle, which can be easily joined and separated.

C15 Copper Weight: This term is used to indicate thickness of copper foil on each layer of a PCB. It's typically expressed in ounces of copper per square foot.

C16 Countersink Holes: These are cone-shaped holes that are drilled into a PCB. To allow a countersunk screw to sit flush with the PCB surface.

C17 Counter bored Holes: These cylindrical holes are meant to be used with a fastener so that the fastener sits flush with the PCB surface.

C18 Cut out: This is a groove that is dug on a PCB.

< D >

D1 Daughter Board: The "daughter" of a "mother" board, a daughter board contains plugs, pins, sockets and connectors and plays a big role in internal connections for electronic devices and computers.

D2 Decal: Another word for a graphic representation of an electronic component, which can also be called a footprint.

D3 Digital Circuit: The alternative to an antilog circuit. Digital circuits operate in a binary fashion like a switch, exhibiting one of two results as a consequence of an input. This is a typical circuit for computers and similar equipment.

D4 DIP: An abbreviation for a dual in-line package, a DIP is a kind of housing for integrated circuits. This housing will typically come in the form of a moulded plastic container with two rows of attachment pins.

D5 Double-Sided PCB: A type of PCB that features traces and pads on both sides, rather than a single side.

D6 DRC: An acronym for design rule check, this is a software verification of a PCB layout. These are often used on PCB designs before production to ensure the design doesn't contain any potential sources of error, like small drill holes or traces placed too close together.

D7 Drill Hits: This is another way to refer to where holes will be drilled in a PCB design.

D8 Dry Film Solder Mask: This is a type of solder mask film that is applied to a printed board that results in a higher resolution mask with finer line designs. This method tends to be more expensive than liquid solder masks.

< E >

E1 Edge Connector: This type of connector is designed for the edge of a PCB, and it is most often used to facilitate an add-on card.

E2 Edge Plating: This is a term used for copper plating that stretches from the top to the bottom of a surface and along the edges of a board, allowing for edge soldering and connections.

E3 Electro conductive Paste Printed Board: This term is used to describe PCBs that are manufactured using a silkscreen printing method. The process involves applying an electroconductive printing paste to set traces and to implement stable through-hole connections.

E4 EMC: An acronym for electromagnetic compatibility, EMC refers to the capability of a piece of equipment or system to run without producing excessive electromagnetic interference. Too much electromagnetic interference can interfere with or damage other pieces of equipment within the same electromagnetic environment.

E5 ESD: A shorthand for electrostatic discharge, which is caused by static electricity.

E6 External Layer: Also called an outer layer, an external layer is a layer on the outside of copper to which components attach.

< F >

F1 Fabrication Drawing: This drawing is a way for designers to communicate a PCB design to engineers and workers. It will typically include an illustration of the board, locations and information about holes to be drilled, notes about the materials and methods involved, etc.

F2 Fine Pitch: This term refers to a class of chip packages with micro-spacing between leads, typically below 0.050 inches.

F3 Finger: These are metal pads found along the edge of a board. These are typically used when trying to connect two circuit boards together to expand the capacity of a computer, for example.

F4 First Article: This is what the first manufactured board is called. First articles are usually produced in small groups before volume production begins so that designers and engineers can inspect the product for potential errors or performance problems.

F5 FR4: This is a material rating for a flame-resistant material. It also refers to the most commonly used PCB substrate material. The name specifies that the resin material is capable of automatically extinguishing when it is aflame.

F6 Functional Test: Alternatively called behavioural test, functional test is designed to determine how well a product's attributes meet design demands.

< G >

G1 Gerber File: A type of CAM file used to control a photoplotter. It's a standard way of communicating board specifications with manufacturers.

G2 Glob Top: This refers to a "glob, " a small ball of non-conductive plastic used to protect the chip and wire bonds on a COB. The glob is usually black in colour and is resistant to thermal expansion, which prevents temperature changes from damaging the connection between the glob and the board.
G3 Gold Fingers: These are connectors found on the edge of a PCB after the board has been plated with gold. Hard, smooth and flat, these fingers are excellent conductors, supporting edge-to-edge connections.
G4 Grid: "Grid" is another term for an electrical grid, an interconnected electrical network that transmits power.

<H>
H1 Half-Cut/Castellated Holes: This refers to holes that are drilled on the edge of a board and plated, resulting in a half-circle hole on the edge of the PCB. This is common for PCBs designed for microchip testing.
H2 HDI: An acronym for high-density interconnector, an HDI is a type of PCB fabrication technology. It uses micro blind via technology to manufacture PCBs with high trace density.
H3 Header: The portion of a connector assembly that mounts directly to the printed circuit.

< I >
IC: Short for integrated circuit, an IC is also called microcircuit, microchip or chip. Essentially, IC describes a method for miniaturizing circuits, especially for semiconductor devices.
Internal Layer: This term refers to the inner layers in multi-layer PCBs. These inner layers are mostly signal layers.

IPC: An abbreviation of Institute of Printed Circuits, a worldwide non-profit association dedicated to the design of PCB wiring. The group helps enterprises achieve greater business success by helping them meet rigorous manufacturing standards, which, in turn, improve overall quality standards.

< K >

Kapton tape: Alternatively called polyimide tape, this electrically insulating tape has numerous useful features, including heat resistance, inextensibility and thinness.

< L >

Laminate: This term refers to the combination of different materials through heating, adhesive and welding methods to create a new material with multiple layers. The resulting material has greater strength and stability than the individual materials combined to create the laminate.

L2 Laser Photoplotter: Alternatively called a laser plotter, this type of photoplotter creates a finely-lined raster image of the end product. The result is a high-quality, highly accurate plot.

L3 Layer-to-Layer Spacing: This is the distance between PCB layers. The lower the spacing, the more difficult the manufacturing process will be.

L4 Lead: Another word for a terminal on a component.

L5 Legend: This is a shorthand guide for marking component names and positions. Legends help ease the assembly and maintenance processes.

L6 LPI: Shorthand for Liquid Photo imageable, an LPI is a liquid solder mask that is sprayed on a PCB. This method is more accurate, thinner than a dry film solder mask and more affordable.

< M >

Mark: A term used to refer to a set of patterns for optical localization. Marks can be classified into PCB Marks and local Marks.

M2 Membrane Switch: A membrane switch is applied to the front of a finished PCB. It indicates functions of the PCB and components, such as key functions, indicators and other parts. The membrane also provides protection for the PCB in the form of waterproofing and humidity protection.

M3 Metal Base/Core Printed Board: Metal core PCB refers to a type of PCB with a core material made of metal instead of plastic, resin or FR4 material.

M4 Mil: A "mil" is another way to say a thousandth of an inch. It's also the equivalent of a "thou. "

M5 mm: "mm" is another way to express a millimetre or a thousandth of a meter.

M6 Motherboard: This is the main board in a computer or electric device. The motherboard carries key interconnections and components that support the primary functions of the device.

M7 Mounting Hole: This hole is intended to secure the PCB to its final location in a device. To ensure there is no interference, all mounting holes are non-conductive and unplanted.

M8 Multi-Layer PCB: This is a type of PCB with at least three conductive layers of trace and components.

M9 Multi meter: A testing tool used to measure electrical values like current, resistance and voltage.

M10 Multi-Wiring Printed Board: An equivalent to a multi-layer printed circuit board, this term refers to PCBs with multiple layers of trace, with dielectric layers between each.

< N >

N1 NC Drill: This is a more common name for a Numeric Control drill machine. This type of machine is what assemblers use to drill holes in PCBs.

N2 Node: This is a pin or lead that is connected to at least one wire.

N3 NPTH: An acronym for non-plated through hole, NPTH refers to a hole with no plated copper on the hole wall. This means no electric connections can be made using the walls of this hole.

< O >

O1 Open: This is a short way of saying "open circuit, " which is a break in an electrical circuit's continuity. This prevents current from flowing and can disrupt the proper function of a PCB.

< P >

P1 Pad: This is one of the most basic composition units of a PCB assembly. A pad is a contact point used to connect components with a via and is the point to which the components are soldered.

P2 Panel: A panel is a combination of boards produced simultaneously to improve efficiency during the manufacturing process. Once the process is finished, these panels are typically broken apart into their singular units before being used.

P3 Panellise: This is the act of grouping multiple PCBs into a panel to improve manufacturing efficiency. An alternative term is penalization.

P4 Part Number: This is an identification method used in industry to differentiate parts from one another. It's also used to identify specific parts, which is helpful in identifying problematic assembly batches and preventing incorrect product applications.

P5 Part: This is another word for a component, or a basic piece of electric equipment, such as a resistor, capacitor, potentiometer, valve, radiator, etc.

P6 PCB Base Material: The material upon which the PCB is built. The PCB base material is typically composed of resin, metal, ceramic or another material with thermal and electric properties that support the PCB's final function.

P7 PCB Database: All the data that is or could be used for a PCB design. This data is usually stored in a computer file.

P8 PCB: An abbreviation of Printed Circuit Board, a PCB is a board that contains a conductive material and components, which act in synchrony to produce a designed response. PCBs rely on electrical circuits, which are either printed or soldered onto the board to elicit the desired result. Printed circuit boards are available in a wide variety of shapes, sizes and purposes to suit any industry or application.

P9 PCBA: This is an acronym for Printed Circuit Board Assembly, where a company solders components to boards.

P10 Peelable Solder Mask: A solder mask or layer of solder mask that can be peeled from the board.

P11 Photoplotter: A device used in manufacturing to produce artwork onto film by plotting objects instead of images.

P12 Pick-And-Place: A method of SMT assembly where a machine automatically picks up SMDs and places them in the correct positions on the board.

P13 Pin: A terminal on a component. It is also called a lead.

P14 Pitch: The distance between pin centres of SMDs.

P15 Plated-Through Hole: Alternatively called a PTH, this is a procedure in which a through-hole is plated so that the hole wall can be conductive. This is often used as a contact point for through-hole components and can be used as a via.

P16 Prepreg: Also called PP, is the key material for multi-layer PCB manufacturing. it is primarily composed of resin and strengthening material that is then classified into glass-fibre cloth, paper base, compound material etc.

P17 Press Fit Holes: This is a hole through which a contact terminal can be pressed into a PCB.

P18 Printed Wiring: A process where a design is etched into conductive metal on a board, producing a wire design for the PCB.

P19 Printing: Part of the PCB manufacturing process where a circuit pattern is printed on the board.

P20 PWB: An acronym for Printed Wiring Board, which is another name for a PCB.

<R>

R1 Reference Designator: Alternatively called "Ref Des, " this is the name of a component on a PCB. Typically, the component name begins with a letter or two, indicating the component class, followed by a number. These designators are usually printed on the silkscreen to help identify each component.

R2 Reflow: This is the process of melting solder to create a joint between a pad and a component or lead.

R3 RF: Short for radio frequency, RF is an electromagnetic frequency ranging between 300KHz and 300GHz. RF can also be a type of high-frequency electromagnetic signal.

R4 RoHS: Alternatively known as the Restriction of Hazardous Substances, RoHS is a European environmental protection law. Many global companies must follow RoHS standards to sell products in the EU.

R6 Route/Track: This is the layout of a PCB's wiring structure, which is important for the proper function of the PCB. As a verb, the act of routing means designing such wiring structures.

<S>

S1 Schematic: A technical drawing that illustrates the connections between PCB components. Schematics will often include abstract representations of components instead of pictures and is an important first step in PCB design.

S2 Short: This is an alternative way to say "short circuit, " which is a connection with low resistance, resulting in excess current at the connecting point. This can cause serious problems in the PCB, including failure.

S3 Silkscreen: This is a layer of epoxy ink applied to a PCB that contains component names and positions. The labels included on silkscreens help to direct workers through the assembly process. Typically, silkscreens are white, which helps the labels stand out against the PCB's solder mask.

S4 Single-Sided PCB: A PCB design with traces and pads included on only one side of the board.

S5 Slot Hole: Non-round holes on a PCB that may or may not be plated. These are often required for specific components but are costly due to the labour needed to cut them.

S6 SMD: Short for surface mount devices, it refers to components designed to be soldered on the surface of PCBs, rather than through a thru-hole.

S7 SMT: Short for surface mount technology, this type of assembly technology directly solders SMDs to the surface of a PCB, rather than running components through thru-holes. This allows the board to function without drilling holes through it and also helps improve component density on the surface of the PCB.

S8 Solder Mask/Solder Resist: This is a layer of material, usually consisting of an epoxy resin, which isn't compatible with solder. This material is applied to the entire PCB, except those areas where content needs to be soldered. This process helps to physically and electrically insulate traces, preventing shorts. Solder masks are often green in colour, though red and black are also common.

S9 Solder Side: This is the opposite of the component side and is usually regarded as the bottom side.

S10 Spacing: This term refers to the distance between wires on a PCB.

S11 Substrate: This is another word for "PCB base material", the primary material for PCB fabrication. Generally, this material can be flexible or rigid and can be made of epoxy, metal, ceramic or other materials. The function of the end PCB will usually determine which substrate will be used for the project.

S12 Supported Hole: This is a via with pads on both sides of the PCB. It's also plated inside the via. This means the entire hole can support functions relating to thermal or electrical conductivity.

S13 Surface Finish: Since copper tends to oxidize in natural environments, a surface finish protects the layer from doing so. Oxidation can cause the tin paste to fail or solder incorrectly. The primary types of surface finishes include HASL, ENIG, IMAG, OSP and others.

<T>

T1 Tented Via: This is a type of via that has a dry film solder mask covering both its pad and its plated-thru hole. This solder mask insulates the via completely, protecting the PCB against shorts. Some vias are tented only on one side to allow for testing on the other.

T2 Thou: This is shorthand for a thousandth of an inch. It's another way to say "mil. "

T3 Through-Hole/Thru-Hole: This refers to a hole passing through at least two layers of a multi-layer PCB. It's also used as a descriptor for components with parts or pins that run through a board to be soldered to another side.

T4 Trace/Track: This refers to the copper path printed on a PCB. It functions similarly to an electrical wire, connecting components on a PCB board. The word "trace" is also used to refer to a segment of the path.

T5 Tracing: This term refers to the width of a PCB's wires.

<U>

U1 UL: UL stands for Underwriter's Laboratories, Inc., a renowned company specializing in establishing safety standards and independently assessing products according to these standards.

U2 Unsupported Hole: This type of hole has a pad on the solder side, but no pad on the component side. There is also no metal layer inside the hole. This means the hole has no conductive reinforcement.

<V>

V1 Vector Photoplotter: Alternatively called a vector plotter or Gerber Photoplotter, this type of photoplotter draws a plot line by line using light manipulation technology. This method can produce larger plots, but it is also much slower than the more modern laser photoplotter method.

V2 Via: This term refers to plated through-holes that connect signals between traces on different layers of a PCB. These holes have conductive copper interiors to maintain an electrical connection.

V3 Via Filled with Resin/Via Plugged: This is a via that is filled with an epoxy resin. Once filled, copper can be soldered to the surface of the resin without influencing the final product.

V4 Via in Pad: Also called a thru-hole on the pad, a via in pad functions as an electric connection between layers. It is useful for multi-layer components or for fixing the positions of components.

V5 V-Scoring: This is an incomplete cut through a panel, which is often used to help break apart panels of PCBs into single units.

<W>

W1 Wire: This refers to a conductive cable that can transmit electricity or heat. It also refers to a route or track on a printed circuit board.

CHAPTER -12
PCB REFERENCE WEBSITES & BOOKS

12.1 PCB Legislation (Environmental issues, ROHS)

1. Ref: https://www.protoexpress.com/blog/rohs-compliant-PCBs-a-step-towards-environmentally-responsible-manufacturing/
2. **Ref:** https://worldsway.com/service/rohs-restriction-of-hazardous-substances-circuit-board-manufacturing/
3. **Ref:** https://www.clarydon.com/lead-free-rohs-weee-regulations
4. **Ref:** https://www.acceleratedassemblies.com/blog/how-are-rohs-compliant-PCB-assembly-services-different-from-others
5. **Ref:** https://www.enventure.com/blog/regulatory-compliance/rohs-weee-and-how-it-relates-to-PCB-design/

12.2. Latest trend in PCB

1. **Ref:** https://www.tessolve.com/top-PCB-industry-trends/#:~:text=HDI%20PCBs%20have%20improved%20wiring,the%20latest%20devices%20and%20technology.
2. **Ref:** https://www.tempoautomation.com/blog/top-6-PCB-industry-trends-and-the-manufacturing-challenges-they-present/
3. **Ref:** https://www.analogictips.com/PCB-development-trends-for-2022-and-beyond/

4. **Ref:** https://www.embedded.com/top-PCB-trends-and-challenges/
5. **Ref:** https://www.PCBcart.com/article/content/future-of-PCBs.html

12.3 PCB Fabrication

1.Ref: mktPCB.com/pub-manufacturing-process

12.4 PCB manufacturing process

12 Ref: PCBcart.com

13 Ref: https://www.PCBcart.com/PCB-fab/standard-PCB.html

14 Ref: https://www.PCBcart.com/PCB-fab/led-PCB.html4

15 Ref: https://www.PCBcart.com/PCB-fab/flexible-PCB.html

12.5 G-10 Woven Glass And Epoxy Definition And Applications

13 Ref:https://www.bevi.com/@/File/Get/?id=954&download=0#:~:text=Page%201,Glass%20woven%20epoxy%20G10,high%20mechanical%20strength%20is%20sought.

14 Ref.: https://www.norplex-micarta.com/structural-materials/what-is-g-10/

15 CEM-1 (cotton paper and epoxy

16 Ref: https://www.multi-circuit-boards.eu/en/glossary/CEM_1.html#:~:text=Description%3A,in%20the%20document%20IPC%2D4101

17 Ref: https://www.rayPCB.com/why-is-cem-1-PCB-material-still-popular-in-the-circuit-board-market/

18 CEM-1 (cotton paper and epoxy

19 Ref: https://hillmancurtis.com/cem-1-PCB/

20 CEM-3 (non-woven glass and epoxy)

21 Ref: https://www.PCBmay.com/cem-3-PCB/

22 Characteristics, details and types of PCB substrates

23 Ref: https://www.proto-electronics.com/blog/characteristics-details-and-types-of-PCB-substrates

23.5 Flexi PCB

24 Ref: https://www.PCBnet.com/blog/what-is-a-flex-PCB/#:~:text=The%20IPC%20defines%20a%20flexible,made%20of%20a%20flexible%20film.

25 Ref: https://www.protoexpress.com/blog/the-history-of-flexible-PCBs-and-their-modern-applications/

12.7 Rigid Flexi PCB

12 Ref: https://www.sfcircuits.com/PCB-production-capabilities/rigid-flex-PCBs#:~:text=Rigid%20flex%20printed%20circuit%20boards%20are%20boards%20that%20use%20a,the%20design%20of%20the%20application.

13 Ref: https://www.PCBcart.com/article/content/flexible-and-flex-rigid-PCB-applications.html

14 Ref: https://www.PCBelec.com/PCB-manufacturing/rigid-PCB

15 Ref: https://www.midwestPCB.com/flex.html

16 Ref: https://www.rigiflex.com/faq/rigid-flex-PCBs-applications

12.8 Multilayer PCB

1. **Ref:** https://www.PCBcart.com/article/content/multilayer-PCB-benefits-and-applications.html
2. Ref: https://www.acceleratedassemblies.com/blog/a-brief-discussion-on-six-common-applications-of-multilayer-PCBs
3. Ref: https://embetronicx.com/tutorials/PCB/benefits-of-multilayer-PCB-and-applications/

12.9 Double sided PCB, with plated through hole (PTH)

12 **Ref:** https://www.icape-group.com/PCB/PCB-types/double-sided-PCB/plated-through-holes-PCB/

13 **Ref:** https://www.clarydon.com/manufacturing/single-double-sided-PCB

14 **Ref:** https://www.electronicsandyou.com/blog/double-sided-PCB.html

12.10 Single sided PCB

1. **Ref:** https://www.rayPCB.com/single-sided-PCB/
2. **Ref:** https://www.theengineeringprojects.com/2018/03/single-sided-PCB.html
3. **Ref**: https://www.mokotechnology.com/what-is-single-sided-PCB/
4. **Ref**: https://microcontrollerslab.com/single-layer-PCB/

12.11 PCB References Books

1) Printed Circuit Boards: Design and Technology
CEDT Series McGraw-Hill, 2006 - Printed circuits - 691 pages

2) High Performance Printed Circuit Boards
McGraw-Hill professional engineering
Professional Engineering Series Boss Hart Tata McGraw-Hill, 1983 - Printed circuits - 467 pages

3) Generation of Precision Artwork for Printed Circuit Boards
A Wiley-interscience publication
Preben Lund Wiley, 1978 - Printed circuits - 353 pages

4) Christopher T. Robertson,Prentice Hall Professional, 2004 - Electronic books - 264 pages

5) Printed Circuit Assembly Manufacturing
Volume 21 of Manufacturing Engineering and Materials Processing
CRC Press, 03-Feb-1987 - Technology & Engineering - 384 pages

6) Printed Circuit Assembly Design
Professional engineering Leonard Marks James Caterina McGraw Hill Professional, 27-Jul-2000 - Technology & Engineering - 368 pages

7) Digital PRINTED CIRCUIT Design and Drafting, Darryl Lindsey Bishop Graphics, 1986 - Electronic circuit design - 423 pages