Volume 9, Issue 2, February 2024 International Journal of Innovative Science and Research Technology
ISSN No:-2456-2165

PCB Advanced Via/Holes Technologies

Jayasekar Micheal[1]
PCB Design Engineer, Intel Corporation, Santa Clara, CA, USA-95054

Abstract:- Printed Circuit Boards (PCBs) Via is the arrangement of different layers to make up a PCB. It involves determining the number of layers, their order, and the materials used for each layer. PCB Via is a plated hole in a circuit board that connects different layers of the board, allowing electrical signals and power to pass between them. They serve as electrical connections between different circuit board layers and help in efficiently routing signals and power. This article provides an in-depth understanding of PCB vias, discussing their types, design considerations, advanced Via technologies, and best practices.

Keywords:- *PCB Vias, Blind Via, Buried Via, Microvia, Sequential Laminations, Aspect Ratio, Via Filling, Via Plating, Annular Ring, Stackup.*

I. INTRODUCTION

In the ever-evolving world of electronics, Printed Circuit Boards (PCBs) remain the unsung heroes, and PCBs are the connectors of innovation. These tiny yet essential conduits are pivotal in ensuring our electronic devices are smaller, more efficient, and increasingly powerful. Understanding PCB vias' various types and functions and adhering to best practices in their design and implementation is essential for engineers and designers looking to create cutting-edge electronics.

As we delve further into the 21st century, the demand for smaller, more powerful, and energy-efficient electronic devices is ever-increasing. PCB vias are adapting to these demands through modern technologies and techniques. As consumer electronics become more compact and lightweight, Microvias use becomes increasingly prevalent. These incredibly small vias allow for high-density PCB layouts, making advanced features like high-speed network connectivity, AI processors, smartphones, other portable gadgets, and augmented reality Devices.

In addition to performance enhancements, modern PCB manufacturing also focuses on sustainability. Some manufacturers can fabricate HDI boards with microvias up to 60 layers. Eco-friendly materials and manufacturing processes are becoming more prevalent, ensuring that PCBs and their vias align with the global push for greener electronics. The importance of vias in PCB layout design cannot be overstated. Vias are the vital connectors that enable modern electronic devices' functionality, reliability, and miniaturization.

Vias are essential for connecting different layers within a multi-layer PCB, and Vias play a crucial role in PCB Stackup design, which involves planning the arrangement of layers in a printed circuit board. The proper use of vias in stackup design can significantly impact the PCB's performance, cost, manufacturability, and reliability.

They act as vertical bridges, allowing signals, power, and ground planes to flow between layers, ensuring the electrical functionality of the board. Designers must carefully consider where vias are placed to achieve the best combination of signal routing, thermal management, and overall board performance. Via placement can influence the choice of layer stack and component placement. How layers are stacked within the PCB affects signal integrity, power distribution, and overall performance.

This article will dive into PCB vias, exploring their types, functions, and best practices in design and implementation.

II. TYPES OF PCB VIAS

Vias are the copper-plated holes in the PCB that allow the layers to connect. There are several types of vias in PCB design and manufacturing. The major types of vias are Through Hole Vias, Blind Vias, Micro Vias, and Buried Vias. In addition to that, there are a few subcategories of via arrangements like stacked vias, staggered vias, skip vias, ELIC, and back-drilled vias. Some other common factors applying to vias are tented vias, plugged vias, Via stitching, and via-in Pad.

These various types of PCB vias provide designers with multiple options to tailor their designs to specific requirements, whether achieving high density, optimizing signal integrity, or enhancing overall reliability. The choice of Via type depends on the electronic device's particular needs and the design's constraints. Fig 1 shows the overall commonly used via types and filling.

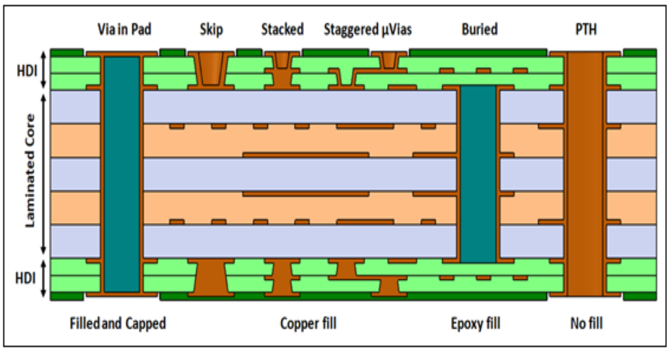

Fig 1 Types of Vias

ISSN No:-2456-2165

A. Plated Through Hole Vias

A through-hole via is a plated hole that traverses all layers of a PCB, connecting traces and components on different layers (Top to Bottom). These vias come in various sizes and shapes but share the same fundamental purpose: establishing electrical and mechanical connections between layers. Through-hole vias enable electrical signals to travel seamlessly between a PCB's different layers. This capability is fundamental for the interconnected world of modern electronics. As electronic devices evolve and demand higher performance, through-hole vias will remain the backbone of PCB design.

Beyond the electrical function, through-hole vias add mechanical and thermal stability to the PCB, which is instrumental in connecting these layers and providing the interconnectivity required for complex circuitry. Fig 2 Shows the through-via structure and thermal Via on a PCB. These vias also serve as anchor points for components. They help secure components on the PCB and maintain their alignment during manufacturing and use.

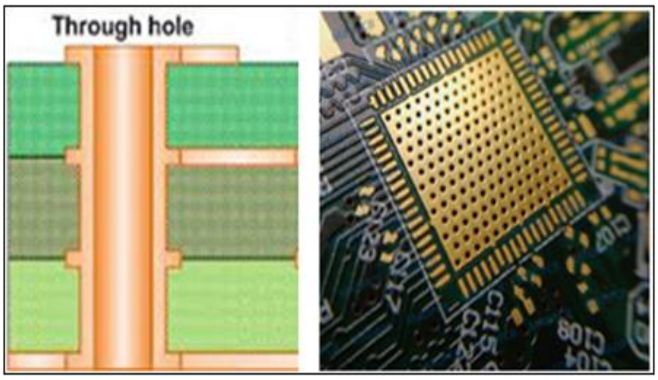

Fig 2 Through Hole Via/Thermal Via

Vias should be sized appropriately and spaced according to the PCB's specifications, ensuring that they do not interfere with each other or near the same net vias or different net vias. The copper area surrounding a through-hole via, known as the annular ring, should be sized correctly to ensure reliable electrical connections and sufficient mechanical support.

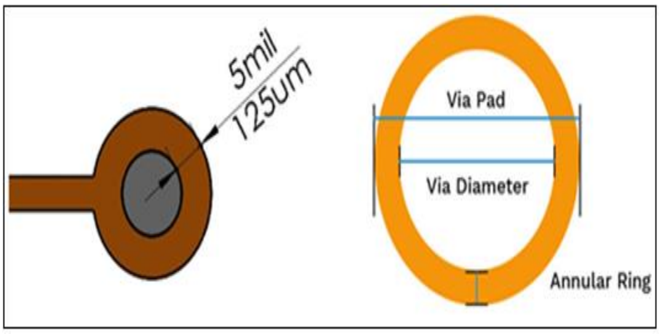

Fig 3 Through Hole Via Pad/Drill

As per IPC 6012 Claas2 Specification, a 5-mil annular ring or more is recommended. The PCB via size is also necessary in Vias, the most popular vias - the 20/10mil via. This means the via hole is 10mil, the pad is 20mil, and the annular ring is 5mil, as seen in Fig 3. A mechanical drill through-hole is the most cost-effective in a multilayer PCB. Laser drilling through holes in full-thickness boards will have limitations based on the hole diameter and the over-PCB thickness.

B. Blind Via

Blind Vias connect the outer layers to one or more inner layers, ensuring high-density routing without needing through-hole vias. A Blind Via connects an outer layer to one or more inner layers but does not go through all the layers on the board. Blind vias can be drilled with a laser or mechanical drill with an aspect ratio of 1:1 or less. Blind vias with laser drills must be filled with copper. Blind Via with mechanical drill can be filled with epoxy and plated over.

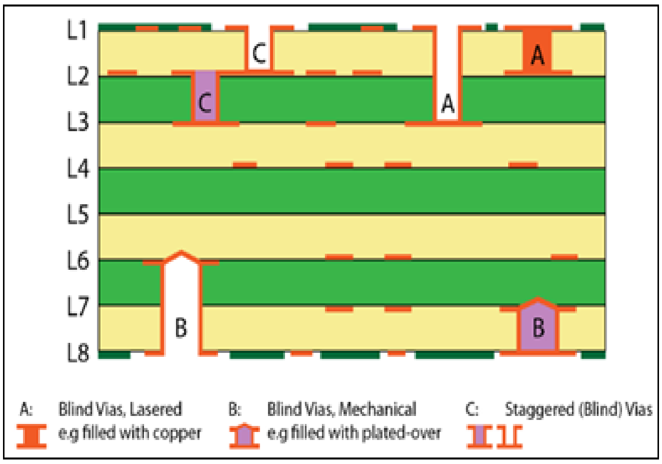

Fig 4 Blind Vias

Blind vias decrease the via stub length for high-speed signals, reducing the parasitic capacitance and producing good Signal integrity. Blind vias can be added to the boards by different methods based on the manufacturing process by vendors like Sequential lamination, Photo-defined, controlled depth, and laser Drilling. Fig 4 Shows the various types of blind via constructions.

➢ *Blind Vias with Sequential lamination*

A balanced Stackup is required for this process with very thin laminate starting from both outer layers. This is an expensive process due to multiple subassemblies laminated manufacturing steps.

➢ *Blind Vias with Photo defined*

This process involves laminating a sheet of photosensitive resin to a core. The pattern-covered photosensitive sheet is exposed to light, causing the residual material to harden. The material in the holes created is removed using an etching solution. Copper is plated in the hole and on the outer surface to create the outermost layer of the PCB. This is cost-effective when there are many blind vias on a PCB.

➢ *Blind Vias with Controlled depth*

This method drills the controlled depth through blind vias, like through-hole vias. In this process, the drill is set to penetrate only partway through the PCB. Generally, drilling is a cost driver, but this method is the least expensive way to create blind vias.

➢ *Blind Vias with Laser-Drilled*

This process is done after all layers of the PCB have been laminated but before the etching & and lamination of the outer layer. A via can be created by laser drilling the copper and the dielectric material between layers 1 and 2 in a single stage, and it could be better than mechanical-drilled vias to reduce cost.

C. Buried Via

Buried Vias (Commonly known as Conventional Buried via) exist entirely within the inner layers, connecting those layers while remaining invisible from the outer layers. Most often, buried vias are drilled separately in each layer before any board layers are combined. Buried vias connect the internal layers but don't extend to the top or bottom layer. There are several buried vias manufacturing processes in the PCB industry. Fig 5 shows the various types of buried via stacking like Standard Buried via, Stacked Buried via, and staggered Buried vias.

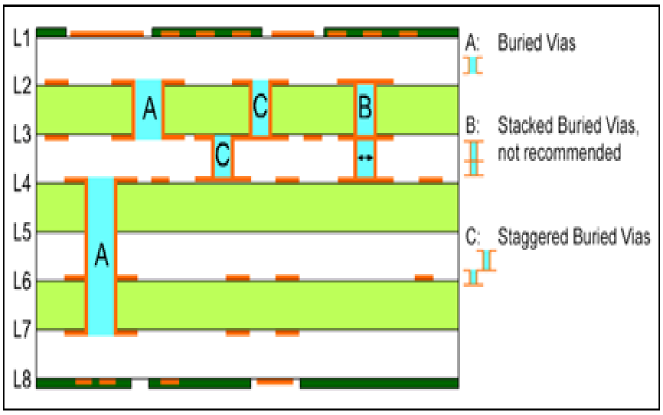

Fig 5 Buried Vias

Space Optimization is the primary advantage of buried vias. Buried vias can enhance signal integrity by avoiding the need to route signals through the outer layers. This is especially valuable in high-speed applications where signal quality is paramount. Buried vias are essential in high-density designs, such as advanced computing systems, servers, mobile devices, consumer electronics, medical devices, and networking equipment.

Buried vias help reduce the aspect ratio of the PCB. Aspect ratios for buried vias must be 1:10 or higher; 1:12 is recommended (Core Via with multiple Layers mechanical drill follows the through-hole aspect ratio rules). However, using buried vias can reduce the PCB thickness. By their nature, buried vias cannot be drilled after the board is entirely fabricated. Buried vias must be drilled at prior stages in the PCB's manufacturing process as a subset of sub-laminated metal layers. Buried vias are typically filled with conductive or non-conductive material to create a low-impedance connection between the different layers of the PCB.

D. Micro-Vias

As our devices become increasingly compact and powerful, the more complex technology that needs circuits fits in a smaller area of PCB. This is where advanced PCB Microvias come into play. These tiny, precision-engineered wonders are at the forefront of the miniaturization revolution, allowing designers to push the boundaries of what's possible in modern electronics. Microvias are a specialized type of Via used to construct high-density, ultra-compact printed circuit boards.

Microvias are essentially different from the other two types because they're not drilled with a regular drill bit. Instead, a laser is used to drill a Microvia, which is usually done before the lamination of layers occurs. Microvias have the smallest pad sizes, which allow for maximum routing channel width. Microvias can reduce the layer count for PCBs and ease the fine-pitch BGA breakout.

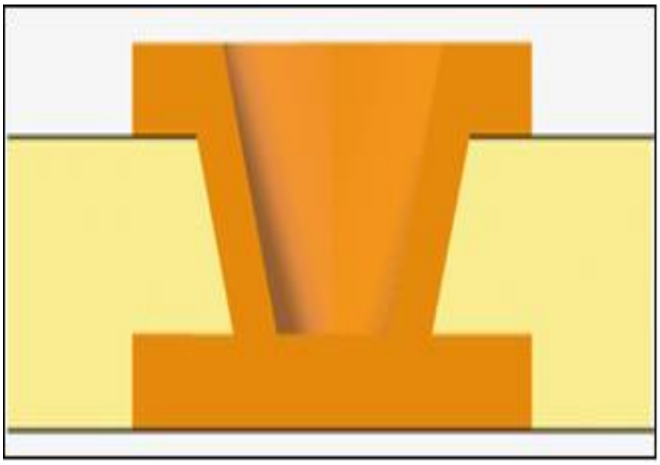

Fig 6 Micro Vias

To reduce layer count and the overall size of PCBs, micro vias contribute to cost savings in manufacturing. Smaller PCBs mean fewer materials used in production, leading to more sustainable electronics manufacturing. Subsequently, this layer is laminated with all the other layers of the PCB. The reliability of micro vias can be significantly impacted by the materials selected for use on the printed circuit board. For micro vias to be reliable, the materials used in their construction must be of a high grade.

IPC-T-50M and IPC-6012 define the structure of microvias. Microvia blind structure with a maximum aspect ratio of 1:1 hole diameter and depth, with a total depth of no more than 0.25mm when measured from the land foil to the target land. Usually, the micro via minimum drill aspect ratio is between 0.6:1 and 1:1. The Most common Microvia min drill size is 100 microns. With advanced technology, it can go down from 40 to 50 microns with fragile dielectric material.

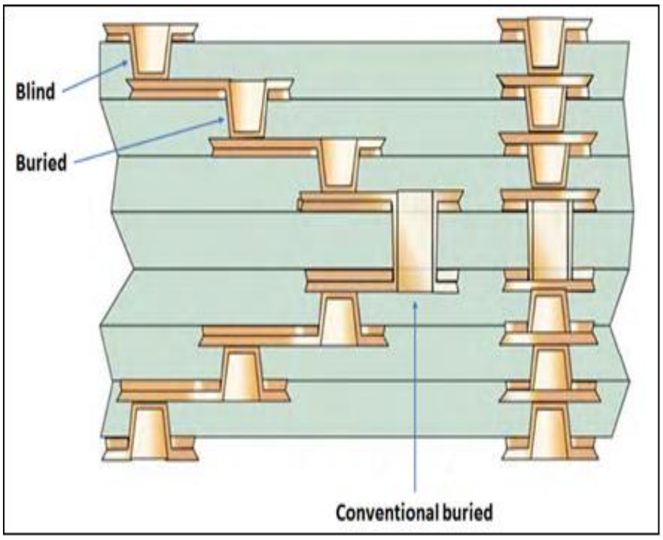

Fig 7 Blind and Buried Microvia

Is Microvias can act as Blind Via? Yes, that can serve as Blind Vias when only running from the external layer to the next layer. Microvias can be either Blind Microvia or Buried micro via. Fig 7 shows the difference between both.

- *Blind Microvia:*

Blind microvias start in the surface layer and terminate one layer below the surface. However, they could terminate to 2 layers below the surface layer if the aspect ratio is kept low.

- *Buried Microvia:*

Buried Microvias have the same structure as blind vias, span between two interior layers, and do not reach either circuit board surface.

- *Microvia Stacking Types*

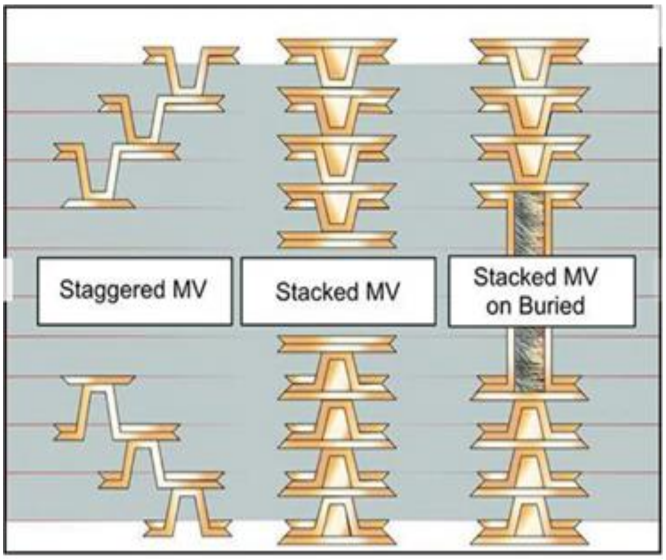

Fig 8 Micro Via Types

There are two types of Microvia stacking in PCB: Stacked Microvia and Staggered Microvia. Fig-8 shows the different structures of Microvias such as Stacked, Staggered Vias, and Stacked Microvias on buries.

- *Stacked Microvias*

Stacking microvias are microvias that layered directly on top of each other. Stacked Microvias, compound structures, are stacked one on top of another to achieve the highest possible routing density, which can further optimize PCB designs by improving density and performance.

- *Stacked Microvias on Buried Via*

Stacked Microvias over buried Vias create more space for the high-density wiring and fanout from low-pitch BGAs. It requires an additional Cu cap plating process for the buried vias. Stacking vias on the Buried vias is not advisable; it causes reliability issues and increases the manufacturing cost.

- *Staggered Microvia*

Staggered Microvias are microvias where the position on each layer is not directly on top of each other. Staggered via also means the vias from 2 layers are connected after copper plating, but the two vias are not precisely at the same place. Staggered microvias are also very common in HDI PCBs, which is a highly reliable and cost-effective method.

- *Skip Microvia:*

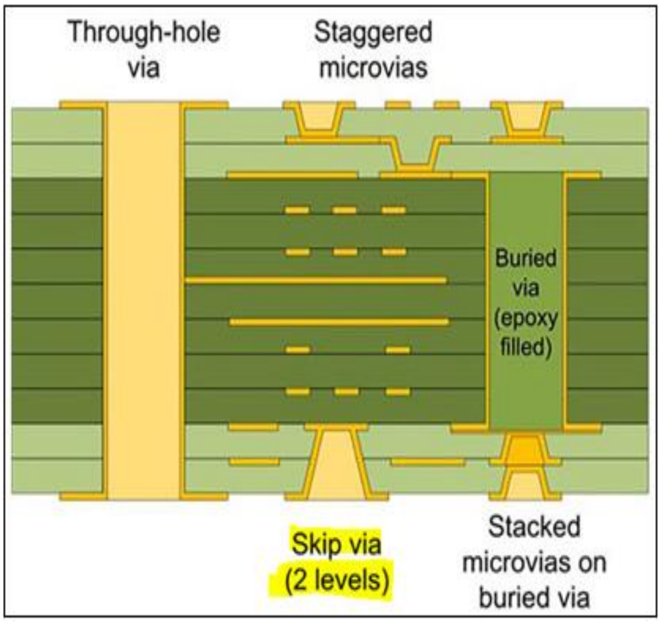

Fig 9 Skip Microvia

A skipped via (known as variable depth Via) connects at least two lay-up layers directly connecting the outer layer with an inner layer electrically without connecting the intermediate layer, for example, a Laser hole from layer 1 to layer three without connecting layer 2. Fig 9 shows the typical 2-level Skip via.

III. HDI VIA STRUCTURE

Microvias are commonly used in high-density interconnect (HDI) PCBs to connect very fine-pitch surface-mount components, such as ball grid array (BGA) and chip-scale package (CSP) devices. The small size of micro vias allows for higher routing density and better signal integrity in these demanding applications.

The standard method of Vias structures is Blind/Microvia, Staggered, and Stacked Vias, stacked via over buried via, and Skip Via, which is used in PCB fabrication. Designing HDI (High-Density Interconnect) PCBs or Substrate/Package Design requires a more precise and complex design with a fine pitch Pad, trace Spacing, and vias.

A. Sequential Lamination Configuration

Sequential lamination is one of the critical technologies used in HDI PCBs. This process utilizes advanced technologies like the sequential stacking and bonding of the various layers of the PCB. This is achieved through Microvias drilling each layer and bonding all layers added at a time. Its precision, flexibility, high density, and mechanical and thermal stability make it an essential tool for modern PCB design and manufacturing.

The first lamination cycle in HDI PCB sequential lamination is the core lamination. This process involves the lamination of the core material, typically a glass-reinforced epoxy or polyimide material. The second lamination cycle is the prepreg lamination. This process consists of the lamination of a layer of prepreg material, typically a glass-reinforced epoxy or polyimide material pre-impregnated with a curing agent.

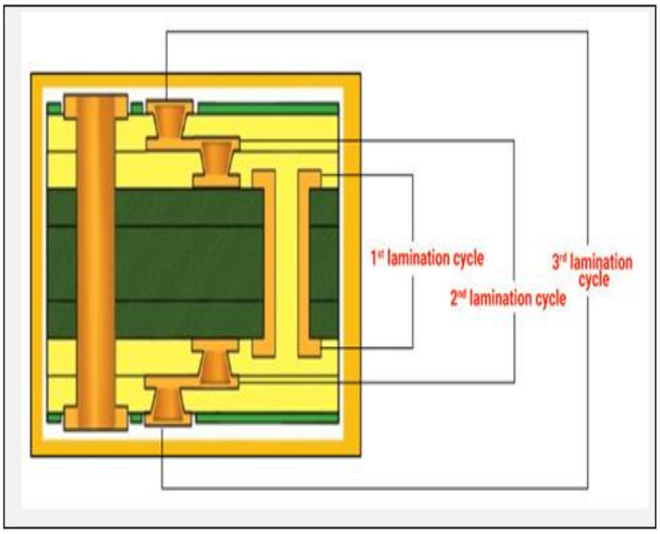

Fig 10 Three Lamination Cycle

The sequential lamination cycle is one of the most used methods in HDI PCB fabrication. This method involves laminating two or more laminate layers in a specific sequence to achieve a final product with the desired properties. Based on the complexity, the lamination cycles go up to 9 levels with advanced manufacturing technology (Higher cost). A commonly standard cost-effective method for an HDI board is four lamination cycles. The Microvias and Buried Vias determine the lamination cycles used with the X+N+X configuration in the PCB.

B. Lamination Types

This structure is generally known as an X+N+X or i+N+i stackup, where the outer sections consist of “X” sequentially laminated layers connected with microvias. The inner portion of the layer stack is linked to the outer sections at the top and bottom ends with a buried via, and the buried via portion (called a core via) also connects to the other inner layers. If the fabrication house can produce it, we could conceivably use any sequentially laminated layers outside the Stackup. For example, 3+N+3 and 4+N+4 layer stacks are standard options HDI PCB fabrication houses provide.

2+N+2 Stackup is the standard option for fine-pitch BGA with a high pin count. The manufacture of a PCB consists of layering an epoxy pre-impregnated fiberglass sheet between each copper layer and laminating together under high temperature and pressure using a hydraulic press. The IPC-2226 standards defined the HDI stackup/lamination in six types, from Type I to Type VI.

- *Type: I Lamination*

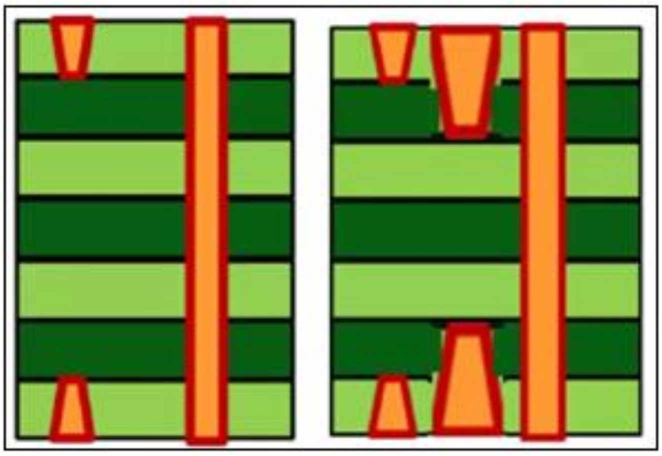

Fig 11 Type-I Lamination

The HDI Stackup type I has a single layer of micro vias on one or both sides, with a laminated core. Type I Stackup only permits blind vias and PTH. They can originate from either the top or bottom surface. These are the simplest HDIs that can be designed or fabricated and, consequently, the most limited type of stackup. It’s also the cheapest to manufacture.

- *Type: II Lamination*

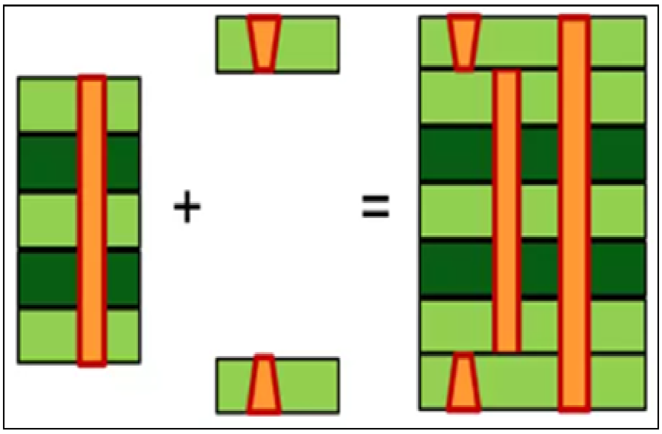

Fig 12 Type-II Lamination

It has a laminated core and is constructed using micro vias, blind vias, and buried vias. It requires a single layer of micro vias on both sides or at least one side. Type II Stackup allows buried vias in the core, but the blind vias are limited to just one HDI layer. The use of buried via helps reduce the overall board thickness, which allows for smaller via diameters.

➢ *Type: III Lamination*

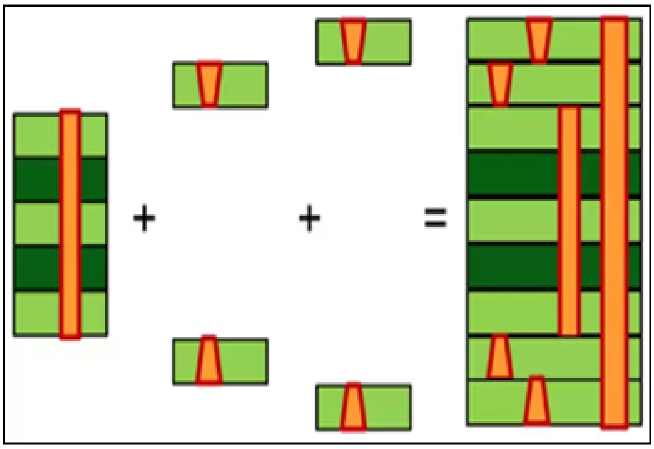

Fig 13 Type-III Lamination

It is also constructed using blind vias, micro vias, and blind vias but requires two layers of micro vias (Stacked or staggered vias) on one or both sides. Type III Stackup is the most used HDI Stackup, especially for high-density requirements. It allows for two or more HDI layers, with blind microvias that can be staggered or stacked in all HDI layers.

➢ *Type: IV Lamination*

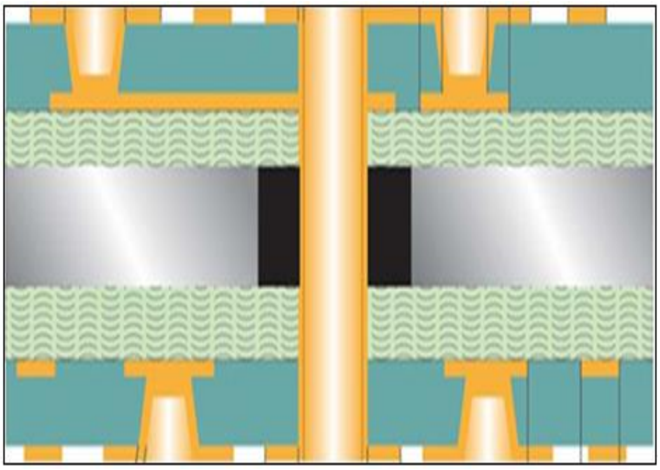

Fig 14 Type-IV Lamination

Type IV uses a passive core, which means the core has no electrical properties. It can, however, still be used for heat dissipation, shielding, or CTE management. Over-core involves depositing dielectric over an internal core layer and is less common among HDI Stackup.

➢ *Type: V Lamination*

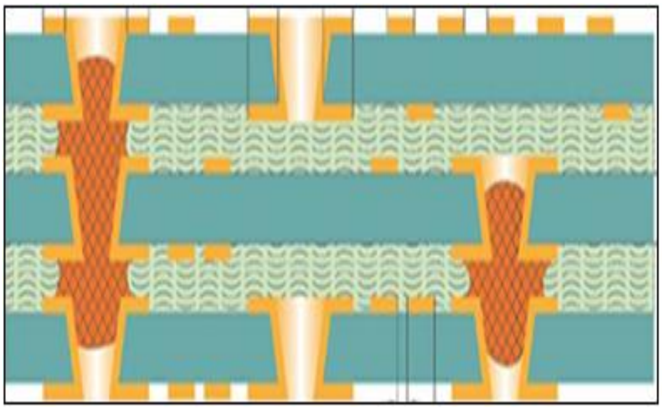

Fig 15 Type-V Lamination

Type V is a coreless Stackup. It uses plated Microvias and a conductive paste layer interconnect. The layers are co-laminated, so there is a single lamination step and no buildup or sequential lamination. This is the rarely used expensive manufacturing process.

➢ *Type: VI Lamination*

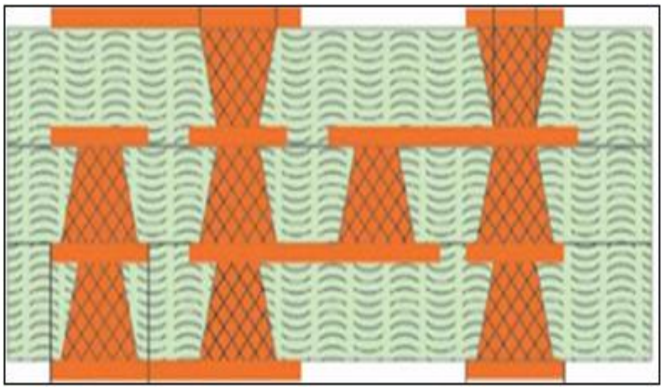

Fig 16 Type-VI Lamination

The most complex is Type VI, better known as every layer interconnect (ELIC), where stacked/staggered Microvias are placed throughout the Stackup. The sequential or core lamination process involves inserting a dielectric between a layer of copper and an already laminated sub-composite using an antistrophic film.

C. ELIC (Every layer Interconnect)

ELIC is commonly known as any layer HDI PCB. To adapt to the development of CSP and inverted chip packaging (FC), it is necessary to use high-density PCB with an internal via-hole (IVH) structure.

To break through the limitation of traditional HDI high-density interconnection laminates, it is necessary to import higher-order any-layer interconnection technology so that any layer can be arbitrarily connected to another layer to form internal conduction.

The interconnection structure of via hole (IVH) is designed for higher-level HDI products to achieve lightness, thinness, shortness, and trimness.

This is the most complex HDI PCB design structure where all the layers are high-density interconnection layers, which allow the conductors on any layer of the PCB to be interconnected freely with copper-filled stacked microvias structures (Fig-17). This structure provides a reliable interconnect solution for highly complex large pin-count devices, such as CPU and GPU chips utilized on handheld and mobile devices, while producing superior electrical characteristics. Examples include smartphones, ultra-mobile PCs, Smartwatches, AR Glasses, GPS, Memory cards, and small electronic devices.

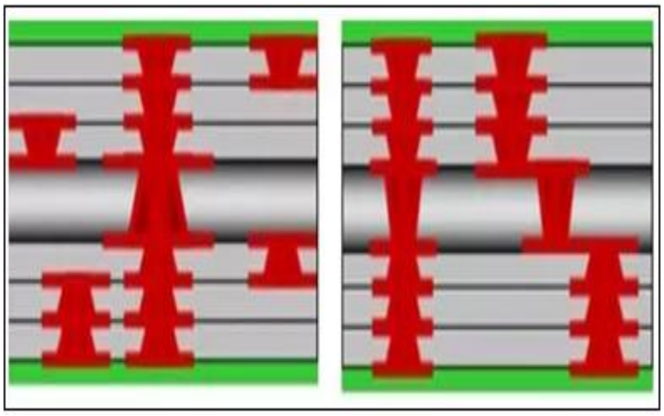

Fig 17 ELIC Via Structure

ELIC is one technology that lets PCB designers create very thin, Rigid-Flex, flexible PCBs with high functional density. When using ELIC on an HDI board, each layer has its own copper-filled, laser-drilled microvias. ELIC uses only stacked copper-filled Microvias to make connections through each layer. Once the layers are stacked, this allows connections between any two layers in the PCB. The ELIC manufacturing process starts with two significant things. These are ultra-thin cores with laser micro vias and a solid copper base. The micro vias are filled with copper internally. The next step is to include a dielectric layer during the lamination process.

Laser drilling is on the recent layer to complete the ELIC PCB stack. The process is conducted in a loop unless the desired PCB stack that contains copper-packed micro vias develops. The benefit of using copper filling is the structural integrity of the board. Another reason to use this filling is to prevent dimpling/voiding in the interior micro vias. For portable devices like Smartwatches, ELIC PCB is the best solution. It has the option for flexible mounting, running faster, excellent thermal performance, high reliability, short lead time, and low power consumption, but the cost is higher.

IV. VIA FILLING

PCB Via hole filling is a process where the via holes are filled with a solder mask or resin. Via holes that are filled/plugged improve the PCB reliability by decreasing the possibility of trapped air/liquid in the Via holes, thus ensuring good yield when PCB assembly. There are two methods of Via filling: Conductive fill and Non-Conductive Fill.

- *Conductive Fill:*

Conductive via filling involves filling the vias with a conductive material such as conductive epoxy or copper to establish electrical continuity between layers.

- *Non-Conductive Fill:*

This involves using a non-conductive material such as epoxy or resin to seal the vias. This method is suitable when electrical isolation between layers is required, and it helps prevent signal interference.

- *Copper-Filled Vias:*

Copper-filled vias are filled with either pure copper or epoxy resin with copper. Many kinds of via can be filled with copper, including standard vias, micro vias, blind vias, and buried vias. Designs with Microvias in BGA pads or similar small soldering pads can benefit from copper filling since this will eliminate the possibility of air voids in the solder joints.

- *Buried Via Filling:*

Filling buried vias in a Printed Circuit Board (PCB) is a common practice that involves sealing the vias with a material to enhance reliability and protect against various environmental factors.

- *Via Filling Types*

IPC-4761 guided for approval via filling and protection for Printed circuit boards on various options.

- *Type-I*

Via is tented using a non-conductive dry film mask on one or both sides. It is not recommended for the modern PCB SMT assembly process. Masking one side is Type I (a), and the dual side is Type I (b).

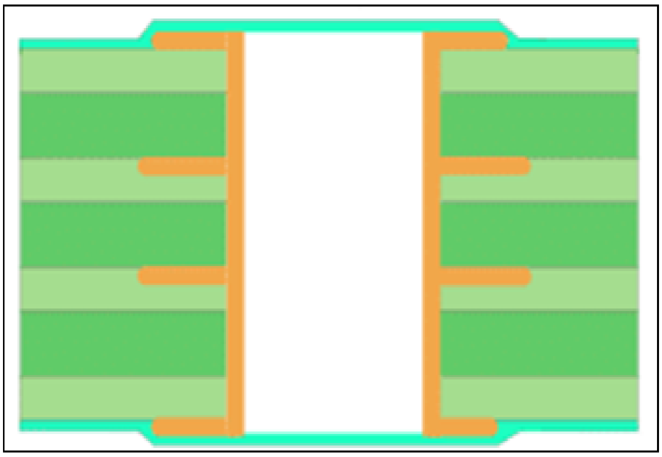

Fig 18 Type I Via Dry Film Mask

- *Type-II*

Via is tented and coated on one or both sides using a non-conductive Soldermask. Tenting/Masking: one side is Type II (a), and the dual side is Type II (b).

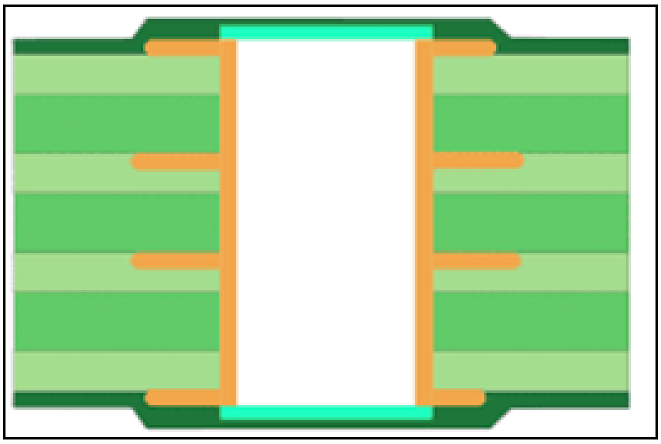

Fig 19 Type II Via Tenting/Masking

- *Type-III*

This Via is partially filled or plugged with nonconductive epoxy paste to prevent solder flow through holes on one or both sides. Filling one side is Type III (a), and the dual side is Type III (b).

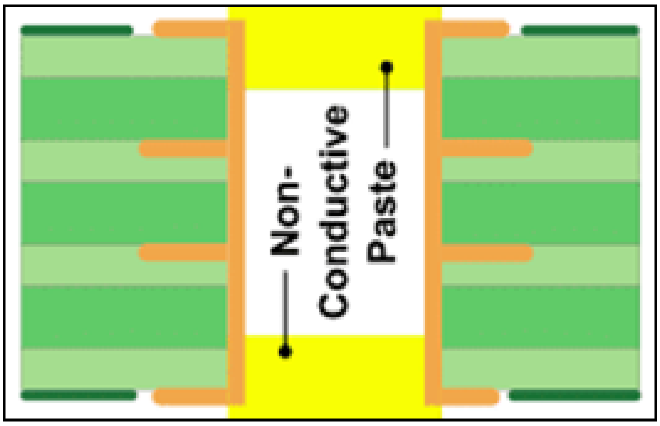

Fig 20 Type III Via Filling

- *Type-IV*

This Via is partially filled or plugged with nonconductive epoxy paste to avoid solder flow through via holes on one or both sides with an additional solder mask layer on one or both sides. Filling/Masking one side is Type IV (a), and the dual side is Type IV (b).

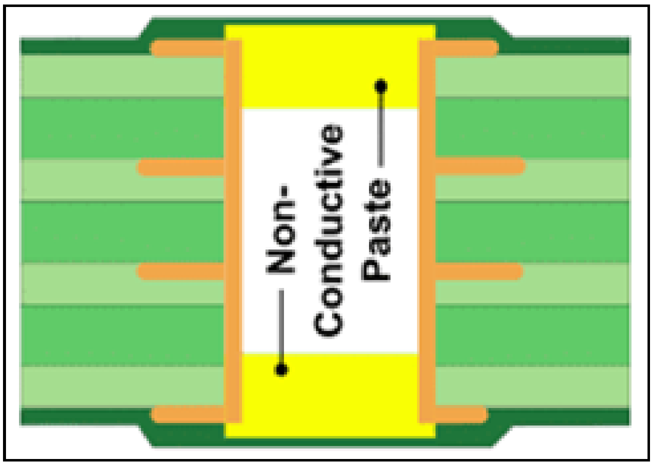

Fig 21 Type-IV Via Filling/Masking

- *Type-V*

This Via is fully filled with conductive or non-conductive material.

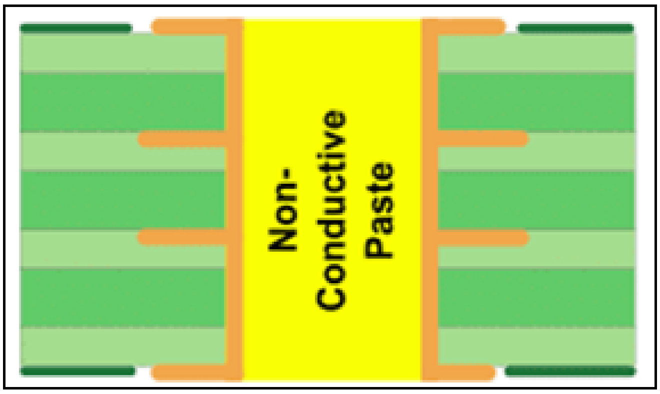

Fig 22 Type-V Via Filling

- *Type VI*

This Via is fully filled with conductive or non-conductive material and covered with Soldermask on one or both sides. Masking one side is Type VI (a), and the dual side is Type VI (b).

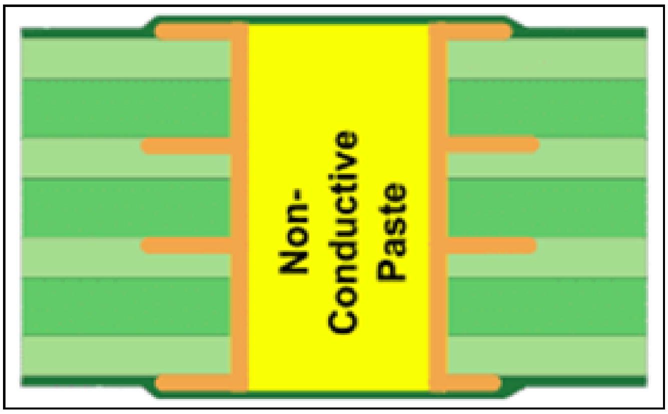

Fig 23 Type-VI Via Filling/Masking

- *Type VII*

This Via is filled with a conductive or non-conductive material and capped with a metalized coating on both sides. Caping one side is Type VI (a), and the dual side is Type VI (b). This method is commonly used for Via-In-Pad for SMT assembly.

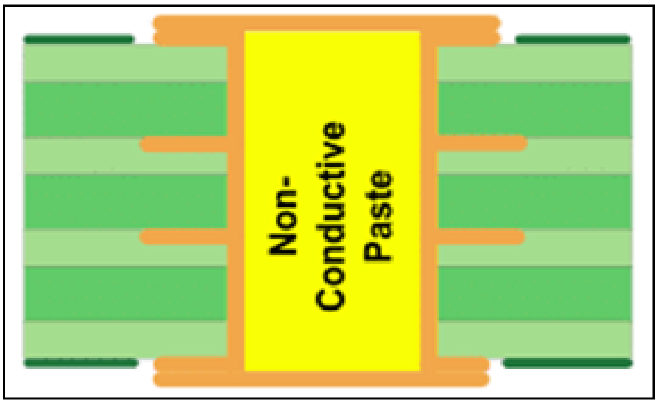

Fig 24 Type-VI Via Filling/Capping

ISSN No:-2456-2165

V. VIA DRILLING

PCB drilling is the process of creating holes in bare circuit boards. This process is carried out to facilitate component placement and the interconnection of electrical nets between various PCB layers. Drilling is the most expensive and time-consuming step in PCB manufacturing.

The PCB drilling process acts as the foundation for vias and the interconnectivity of different layers. A key element in this quest for miniaturization is the drilling of micro vias, tiny holes that allow for high-density interconnections on PCBs. Once the lamination process has been completed, the PCB enters the drilling phase, following the specifications in the PCB NC drill files. The drilling process of adding holes in the PCB is carried out to facilitate the positioning of the creation of vias, and the electrical connection between the different layers of the PCB.

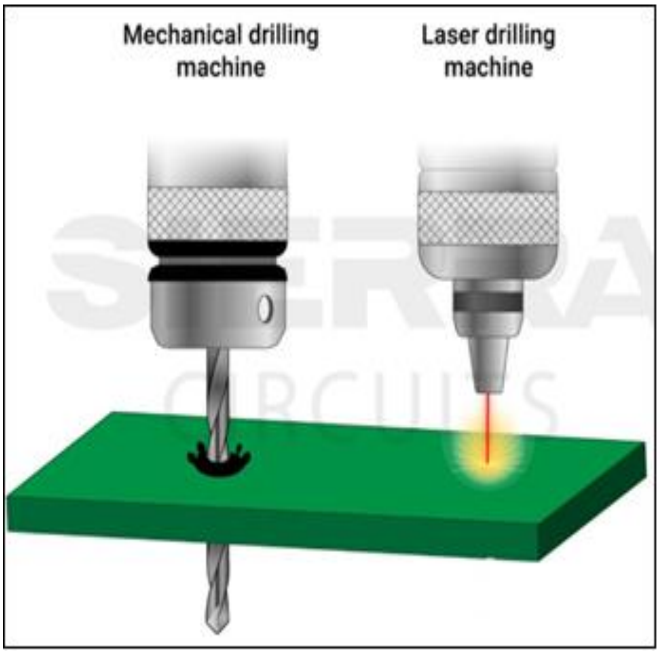

Fig 25 Mechanical/Laser Drilling

There are two types of PCB drilling available in PCB manufacturing; one is mechanical drilling, and the other one is laser drilling. The choice between laser drilling and mechanical drilling in PCB manufacturing depends on the specific requirements of the design, budget, and intended scale of production.

A. Mechanical Drilling

Mechanical drilling is the most cost-effective method for drilling holes for high-volume PCB manufacturing PCBs. The drilling process is controlled by Automatic pre-programmed Computer Numeric Control (CNC) machines. Depending on the Via holes and aspect ratio type, holes can be drilled like Blind, Buried, and through-hole vias. Mechanical drilling may not achieve the same level of precision as laser drilling, particularly for extremely small vias.

Mechanical drilling uses physical drill bits to remove material and create holes in the PCB. It's a contact-based process. It's often faster than laser drilling, making it suitable for large-scale production. Mechanical drilling is versatile and compatible with a wide range of PCB materials. Mechanical drilling involves physical tools, such as drill bits, which can wear out and require regular replacement. The contact-based nature of mechanical drilling can sometimes lead to material stress and potential damage, especially in delicate PCB layers.

B. Laser Drilling Technology

This is not like mechanical drills, the laser drilling process doesn't physically contact the PCB material it works with. The high influence beam of the UV laser machine can drill through copper and organic dielectric to create the tiny Via hole. The UV laser has excellent reflection, absorption, and transmission capacity on different materials. Laser drilling is a non-contact method that uses high-intensity laser beams to ablate material, creating micro vias. It is precise and controlled, making it ideal for tiny holes in PCBs.

➤ *Standard Laser Drilling:*

Laser drilling offers exceptional precision and control, producing highly accurate micro vias, even with diameters 50 to 150um. The flexibility of laser drilling allows for creating complex and intricate patterns. Laser drilling removes minimal material, reducing the risk of damaging delicate PCB layers. Since it's a non-contact process, laser drilling does not involve physical tools that wear out.

Laser drilling equipment can be expensive, making it less cost-effective for large-scale PCB manufacturing.

Laser drilling may not be suitable for all PCB materials, and it may require specific materials and coatings for optimal results. The process can be slower than mechanical drilling, particularly for large PCBs or high-via-count designs.

➤ *Laser Direct Imaging Technology:*

A new technique that can be used to drill holes in the printed circuit board is direct exposure (also known as direct imaging). Based on the image processing principle, this new method increases accuracy and speed by creating a digital image of the PCB and converting it into a map of positions used by the laser as references for drilling holes.

C. Back Drilling Vias

It is a process to remove the unused section of plated through holes/Vias. Via Stub is a non-functional part of Via, causing significant signal integrity in high-speed design. Stubs are the source of impedance discontinuities and signal reflections, which become more critical as data rates increase. To resolve this, a fabricator can employ back drilling, where most via stubs are removed by re-drilling with a slightly larger drill bit. Fig-26 Shows that back-drilling.

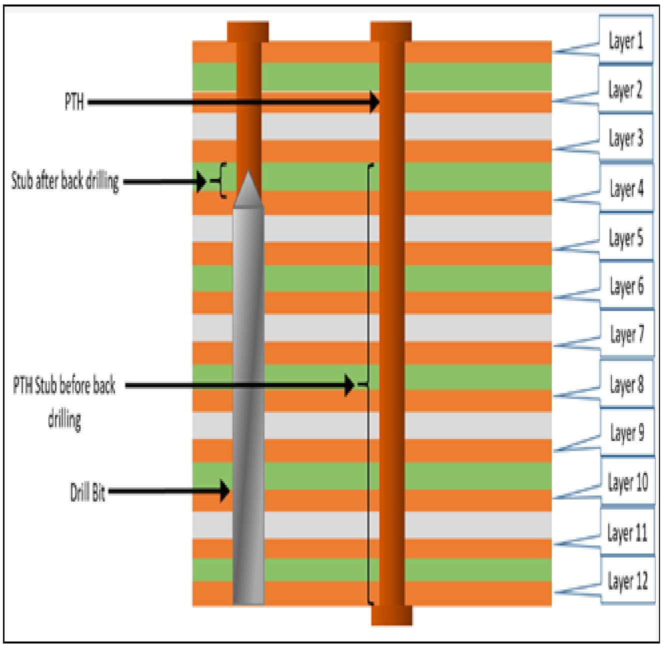

Fig 26 Back Drilling

Back drilling or controlled depth drilling (CDD) is a method to remove stubs of copper from a through hole. It is the preferred method in High-speed designs.

D. Aspect Ratio

The aspect ratio in the context of PCBs refers to the ratio between the thickness of the PCB and the diameter of the drilled Via holes. The aspect ratio (hole diameter to board thickness) plays a role in manufacturability, and an appropriate aspect ratio is necessary to ensure proper plating and reliability during manufacturing. The aspect ratio is determined by the overall thickness of the printed circuit board during primary drilling without plating applied and the diameter of the smallest drilled hole.

This is an important ratio due to its effect on the plating that is within the vias, and also affected by the annular rings.

➢ *Through Via Aspect ratio*

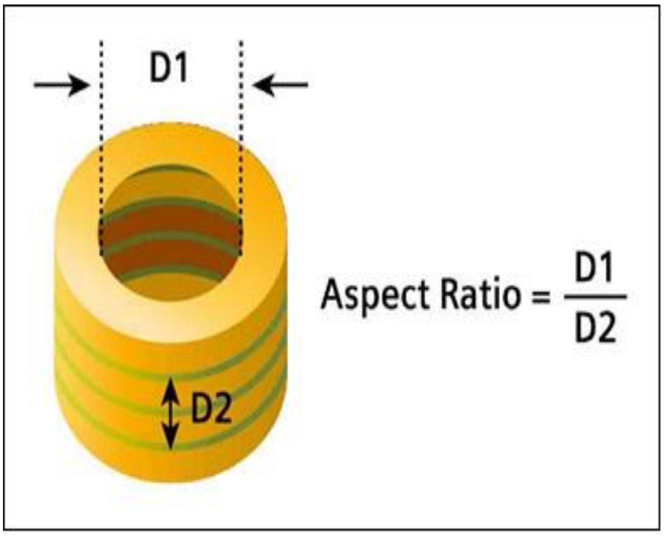

Fig 27 Through-Hole Aspect Ratio

The aspect ratio of a through via in a Printed Circuit Board (PCB) refers to the ratio between the length of the via (from the top to the bottom of the Board) and the diameter of the drilled hole.

TH Aspect ratio = Hole height/drilled hole Diameter

The hole height is the same as the PCB thickness if the start and finish layers of the hole are the top and bottom layers of the PCB. Most manufacturers can do a Vias aspect ratio from 6:1 to 12:1; the preferred is 8:1.

➢ *Blind and Buried Via Aspect Ratio*

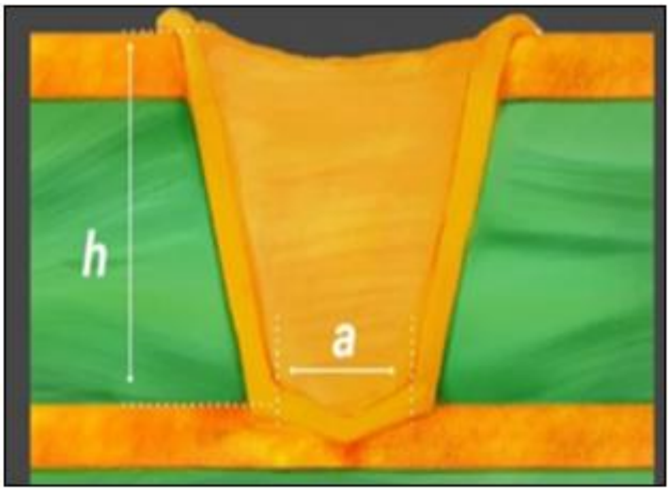

Fig 28 Blind and Buried Microvias Aspect Ratio

For Blind and Buried Microvias, it is the ratio between the hole depth and the diameter of the drilled hole. It is calculated through the formula AR=h/a where h=hole depth (dielectric thickness + Copper foil thickness). The preferred and cost-effective aspect ratio for Microvias is 0.75:1, and the larger aspect ratios, such as 1:1 to 2:1, can be fabricated, but they have reliability concerns.

➢ *PCB Via Plating*

PCB Via plating process starts after the PCB has been laminated in the heat press. It refers to coating the inside walls of drilled holes, such as vias or through holes, with a conductive material to establish electrical connections between different layers of the printed circuit board. The total amount of copper is now electroplated on the exposed metal areas of the board, including in the via holes.

This is done by connecting the board to an electrical charge so that the boards act as cathodes for the electroplating process and then dipped in chemical baths. Copper plating thickness is usually about 20um.

The larger the aspect ratio is, the more complex the plating process becomes. A simple rule of thumb is thicker boards need larger vias. Longer plating time and processing can increase the chances of cracks in the hole wall due to expansion. The lower aspect ratio has more muscular bonded hole walls and less chance of cracking.

ISSN No:-2456-2165

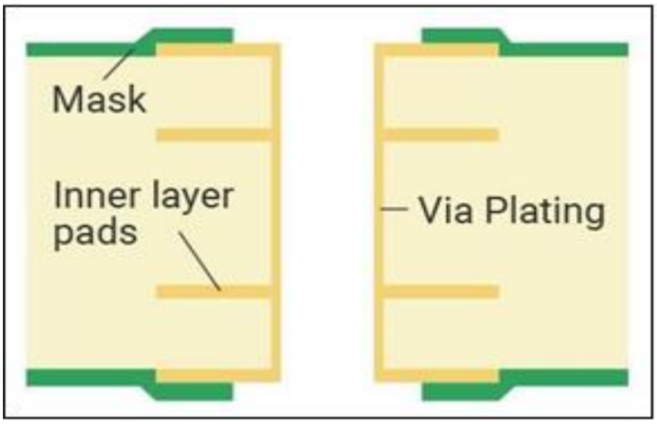

Fig 29 Via Plating

➢ *Annular Rings:*

The copper area surrounding a through-hole Via, known as the annular ring, this should be sized correctly to ensure reliable electrical connections and sufficient mechanical support.

$$\text{Annular Ring} = \frac{(\text{Diameter of the Copper Pad} - \text{Diameter of the Via/Hole})}{2}$$

The size of the annular ring is equal to half of the difference between the diameter of the pad and the diameter of the Hole. Ex: 20 mil Pad 10 mil Drill (20-10/2=5mils Annular ring).

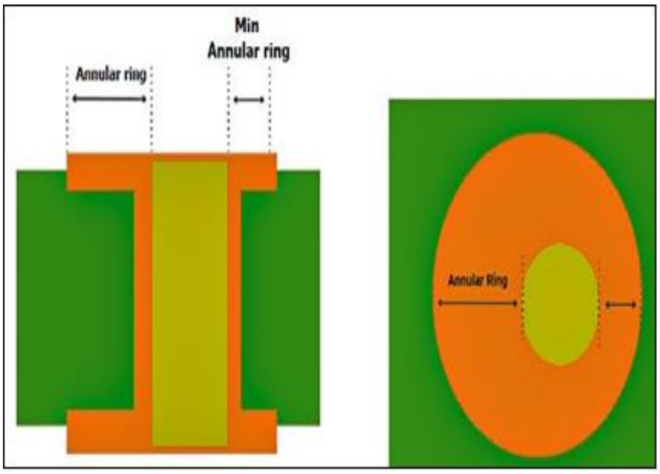

Fig 30 Annular Ring

The size of the annular ring, including its diameter and width, is critical for ensuring proper electrical and mechanical performance. The width of the annular ring is determined by factors such as the via or hole diameter, the minimum clearance requirements, and the manufacturing capabilities of the PCB fabrication process. A wider annular ring provides increased mechanical stability and solder joint reliability.

➢ *Teardrops*

Teardrop is named because of its droplet-shaped structure, and it is an extra future that adds additional copper at the Junction between Via and trace. One common problem during fabrication is misaligned holes and undesirable breakouts due to drill wander. PCB teardrop yielding on the vias, and landing pads helps ensure a missed drill hit can still form a reliable connection while preventing track separation.

Fig 31 Teardrops

The purpose of this teardrop is to reinforce the mechanical strength of the PTH and prevent it from breaking or being damaged due to stress during installation or use.

➢ *Via-in-Pad*

PCB via-in-pad (VIP) technology revolutionizes PCB design by allowing vias to be placed directly within component pads of Surface Mount Components. VIP technology is particularly beneficial in high-density designs where space optimization and signal integrity are paramount, especially on BGA, which allows place Vias to SMT Pads directly.

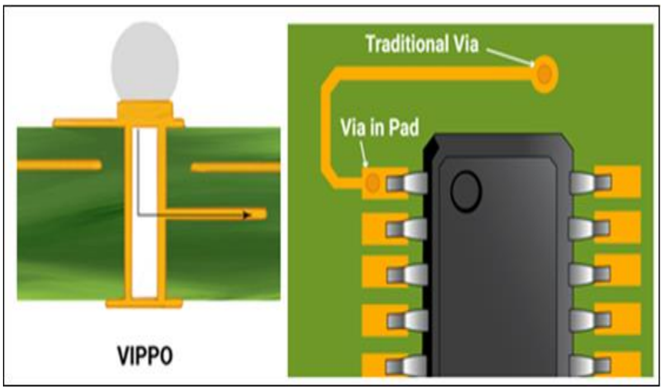

Fig 32 Via-in-Pad

In the via-in-pad process, the vias are typically filled with a non-conductive epoxy, capped, and then plated over to avoid short circuits or empty soldering caused by tin leakage. This method ensures the vias are filled, reducing the risk of voids and improving plating consistency.

➢ *Size and Spacing*

The spacing between a PCB Vias and a nearby trace is a critical design parameter that influences the PCB's electrical performance, manufacturability, and reliability. Vias should be sized appropriately and spaced according to the PCB's specifications, ensuring they do not interfere with each other or nearby copper. With advanced technology, the through vias hole/Pads/Trace can be 125/250um spacing, and laser drilled micro Vias can be 50/150um spacing.

VI. CONCLUSION

PCB vias are the lifelines of PCB layout design. They enable the interconnections that power our modern electronic world. One essential component of PCBs is the VIA, a seemingly humble but crucial element that bridges the gaps between different layers of a PCB.

Understanding the types of vias, their functions, and best practices for design and implementation is critical for PCB designers looking to create efficient, reliable, and innovative electronic devices. As technology advances, vias will remain at the heart of PCB layout design, adapting to meet the evolving demands of the electronics industry.

PCB vias are the unsung heroes of modern electronics, enabling the functionality and miniaturization of our favorite gadgets. Their diverse types and crucial roles in signal routing, power distribution, and thermal management make them a fundamental element in PCB design. By understanding their functions, following design best practices, and leveraging advanced technologies, engineers can continue to push the boundaries of what's possible in the world of electronics. As PCB technology evolves, vias will remain a cornerstone in connecting the dots of innovation.

In conclusion, PCB vias are the connective tissue of our increasingly interconnected and compact world. They enable the technology that drives innovation and makes our lives more efficient, convenient, and entertaining. As new technologies continue to emerge, PCB vias will remain at the forefront of modern electronics, adapting and evolving to meet the demands of the ever-advancing electronic landscape. Whether it's enabling lightning-fast data transmission or supporting the latest advancements in artificial intelligence, PCB vias will continue to play a crucial role in the relentless march of technological progress. So, the next time you marvel at your smartphone, tablet, or any other electronic marvel, remember the unsung hero – the PCB Via – quietly but indispensably connecting the dots of modern technology.

REFERENCES

[1]. PCB Stackup Design for Modern Electronics https://ijisrt.com/pcb-stackup-design-for-modern-electronics
[2]. PCB Material Selection for High-Speed Application https://ijisrt.com/pcb-material-selection-for-highspeed-application
[3]. In-Situ De-embedding (ISD) by AtaiTec Corporation. www.ataitec.com
[4]. Acceptability of Printed Boards, IPC-612, IPC-4101, IPC-2581, IPC-2221, IPC-4761, IPC-D-317A, http://www.ipc.org
[5]. Proto-Electronics: https://www.proto-electronics.com
[6]. NCAB Group: https://www.ncabgroup.com/ microvias/
[7]. CAMPTECH: https://camptechii.com
[8]. Rush PCB https://rushpcb.com
[9]. Sierra Circuits https://www.protoexpress.com/blog/
[10]. Hemeixin PCB, https://www.hemeixinpcb.com
[11]. Altium PCB Design, https://resources.altium.com