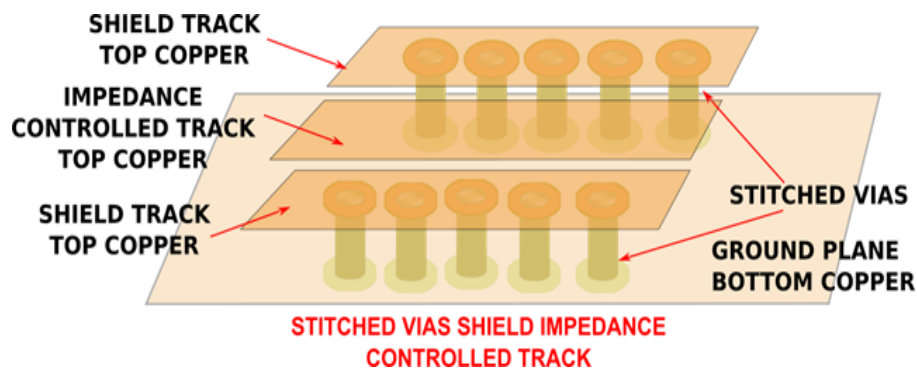


Stitching Vias

1. Introduction:

Stitching vias are something you often see spread around the surface layer of a PCB. If someone is using copper pour correctly, then they will ideally calculate an appropriate stitching via separation distance such that the via array suppresses crosstalk/interference. The other option is usage as multiple parallel connections between layers that can provide low resistance and impedance, and therefore they can provide high current in DC or AC.



2. The Typical Usage: Connecting Ground

The standard usage of stitching vias is to connect grounds across multiple layers. In a Multilayer PCB, it's common to have more than one region of copper assigned to the same ground net. Stitching vias are a useful tool for connecting these everywhere and ensuring minimum possible impedance for any return current propagating along the reference plane in the PCB.

3. Advantages of Stitching Vias

- **Improved Signal Integrity:** By providing a low-impedance return path, stitching vias help to reduce crosstalk between adjacent signals. This is particularly important for high-speed signals where even small amounts of crosstalk can degrade performance.
- **Reduced EMI:** Stitching vias can help to reduce electromagnetic interference (EMI) by providing a controlled path for return currents. This can improve the overall noise immunity of a circuit.
- **Improved Power Delivery:** Stitching vias can be used to create a more uniform distribution of power to the various components on a PCB. This can help to prevent voltage drops and improve the stability of the power supply.
- **Increased Mechanical Strength:** Stitching vias can help to improve the mechanical strength of a PCB by providing additional bonding points between the different layers.

4. Disadvantages of Stitching Vias

- **Increased Cost:** Adding stitching vias to a PCB design can increase the manufacturing cost due to the additional vias and the associated drilling and plating processes.
- **Reduced Yield:** The process of drilling and plating vias can introduce defects that can reduce the overall yield of a PCB.
- **Increased Design Complexity:** Designing a PCB with stitching vias can be more complex than designing one without them. Careful consideration must be given to the placement and spacing of the vias to ensure that they do not interfere with other components or signals.