

Stitching vias and copper pours

1.what is stitching vias and copper pours :

Stitching vias : it is vias connected to the return reference plane to provide a path for current to return back to the source normally stitching via will be a via connected to the ground plane and different pcb layers also, it provides a reference for the electric field to avoid signal integrity and EMC issues and avoid field from propagation , without stitching vias the return current will spread across the layers and causes noise.

Copper pours : it is filling the empty area between the tracks with copper and usually this copper area is connected to the GND to reduce the cross talk between the signal tracks and reducing the noise and EMI

2.when to use stitching vias and when to use copper pours :

Stitching vias usually used when

1. The signal is changing the planes or changing the reference point for example when signal is changed from top layer to bottom layer then stitching vias is recommended providing path for the return current but when changing layers without changing the reference point, we don't need to use stitching vias at this case.
2. when high DC or AC current is needed to pass between layers multiple parallel connections of stitching vias could be used to provide a low impedance path between the layers.

Copper pours usually used when

- 1.power distribution in the board for example it might be used for 5v across the pcb also it is used to make a ground plane which is a large copper pour connected to the ground, and this provide a low impedance path for current

2.thermal management in the pcb especially with high power devices like voltage regulator as the copper pour helps to dissipate heat generated by the components across the copper area which acts like a heat sink to avoid localized heat in certain point

3.how to use stitching vias and copper pours properly :

Stitching vias

- Stitching vias should be placed close to the main via connecting between different planes with different references.
- When dealing with high current stitching vias can be placed along the track itself to be more effective.
- More stitching vias placed will limit the propagation of current in the layer even more so when high current is used more stitching vias in required to be effective .
- calculate an appropriate stitching via separation distance such that the via array can overcome the crosstalk and interference.

Copper pours

- copper pours should always be grounded and never leave it floating this help reducing the cross talk even more and be more efficient.
- if vias is used with copper pours they should be spaced close to provide shielding but also lower resonance to reduce the crosstalk and EMI.
- spacing between copper pours and trace is important as if the spacing is too large it will create micro strip behavior and when the spacing is too small it will create a coplanar wave guide behavior these behaviors affect the impedance of the trace so the closer the copper pours to the trace the trace impedance gets lower.

The image below shows the simulation for different kind of placement for stitching vias ,We can see who changing the number and placement of the vias the current propagation and distribution among the layer changing

The second images shows the losses of the signal traces corresponding to the copper pour spacing to the trace the red line is the closest and the relation is reciprocal closer copper pour make induce greater losses

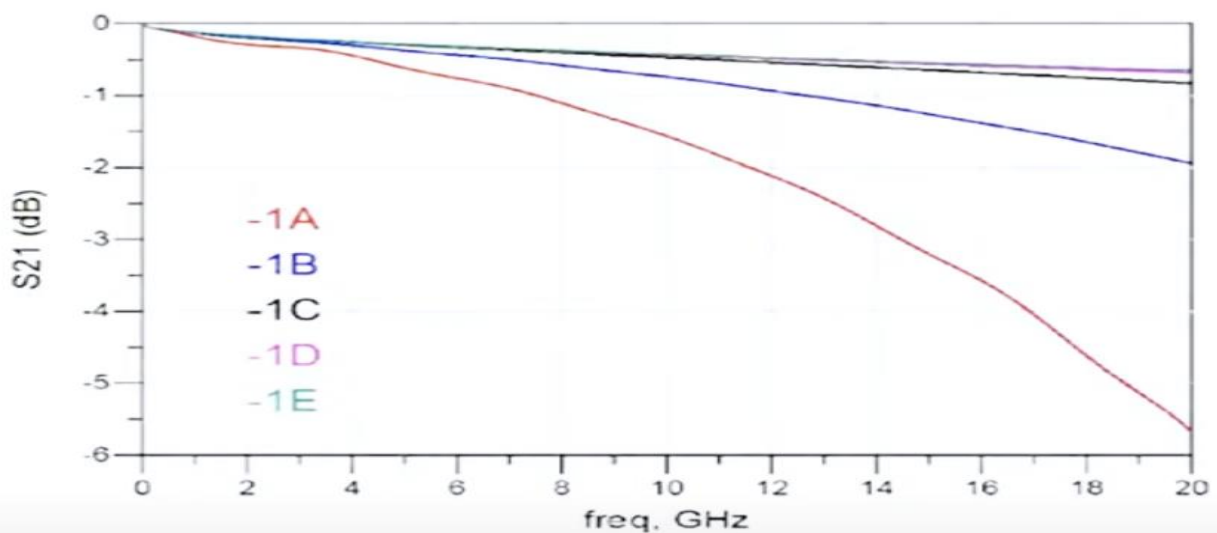
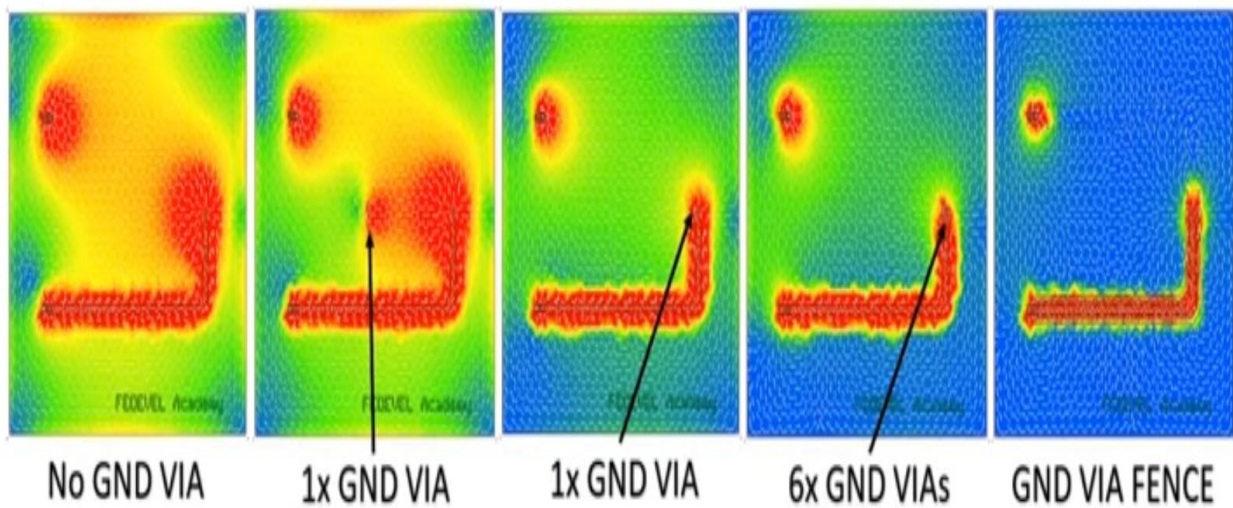


Figure 3b: Insertion loss plots for models listed in Table 1.

3. Best practice and applications :

1.Stitching via

- **connecting ground**

the main purpose of stitching vias is connecting ground across multiple layers in multi-layer pcbs , this is done by connecting the regions of copper assigned to the GND together.

- **layer transition**

when making a layer transition in digital or RC circuits it is common to use a stitching via near the transitioning via to provide a clear ground reference for the control signal

- **shielding**

Stitching vias can be used to prevent coupling of electromagnetic waves this is done by having a stitching via close to the signal via to provide a noise reduction that happen as the loop inductance in the stitching via is lower cause the via is closer to the GND

3. Advantages and disadvantages for using stitching vias and copper pours :

1.Advantages of stitching vias

Reduced Ground Loops: makes the ground plane solid and consistent which help reduce noise and improve the performance of the board

Improve thermal management: stitching vias helps transfer heat from top layers to the bottom layers and this is important when working with high density integrated circuit

Enhance signal integrity: done by providing low impedance return path for high frequency signals

2. Disadvantages of stitching vias

Reduce routing space: by limiting the available areas for routing makes it difficult to route freely

Short circuit problems: wrong placement of stitching vias can make short circuit between layers

Complex design: adding many stitching vias to the design can make it more complex to manufacture and higher cost

3. Advantages of copper pours

Better thermal management: by dissipating the heat more effectively Across the pcb

Reduce impedance for power and ground: low impedance ensures stable power delivery and lower voltage drop

4. Disadvantages of copper pours

Skin effect: when the spacing between copper pours and traces is small this will reduce the impedance of the trace and induce the current to transfer across the sides of the trace which is well known effect called skin effect this can produce more losses to trace

Ground loops: this happens when there is more than one path to the ground which can cause unwanted noise in the circuit especially in RF and AUDIO circuits

