Stitching vias and copper pouring in PCB Design

1. Introduction

Via Stitching and Copper Pour can be like acid: quite useful if implemented properly, but also dangerous if used indiscriminately.

2. Stitching vias

2.1 Usage

- Commonly used to tie grounds together
- Routing the PCB as it is very important for maintaining ground continuity.
- Shielding as it provides sufficient shielding against specific RF frequencies, but they must be spaced close to be effective so that you don't excite a resonance that increases crosstalk/EMI.
- Suppress noise particularly for slow digital protocols.
- With high-speed digital and RF, having the stitching via structure close to the signal via provides noise reduction

2.2 Advantages

- Improved grounding
- If power is provided, closely spaced vias can provide high current with minimal temperature rise
- Suppress noise
- Confining Electromagnetic field by providing a low impedance path for high frequency signals
- If a reference across layers is provided, stitching vias are convenient for low-speed signals to minimize EMI

2.3 Disadvantages

- Adds complexity to the PCB design.
- Adjusting the stitching via placement could affect a differential via.
- For high-speed signal transitions, the randomly placed stitching vias can't be relayed on to guarantee signal integrity
- shielding as an outstanding issue regarding the sizing, spacing, and placement of stitching vias

2.4 Best practices

- A constant grid pattern is used for via placement
- The only way to accurately determine via impedance and the effect of stitching on via impedance is to use an electromagnetic field solver.

- Thermal relief pads are used to connect vias to copper pours
- Maintaining appropriate spacing to avoid short circuits

3. Copper pours

Copper pours are large areas of copper that cover significant portions of the PCB

3.1 Usage

- Providing a low impedance path for power distribution
- Improving grounding by creating a large ground plane
- Heat dissipation

3.2 Advantages

- Efficient power distribution
- Thermal management like vias help in dissipating heat from the components
- Also, improved grounding like vias

3.3 Disadvantages

- Problems like signal integrity, careful planning is essential
- Could make Manufacturing obstacles as by making the copper pour wider, you reduce the mutual inductance and the field strength reaching the victim trace.
- It dissipates heat and becomes a DFM/DFA problem when the pour region is large.

3.4 Best practices

- Providing sufficient space between signal traces and copper pours
- For better efficiency combined with stitching vias
- Grounding the copper pour not leaving it floating

4. Conclusion

Copper pour with stitching vias will not magically solve every EMI problem.

whatever problem you're trying to solve with copper pour with stitching vias was probably caused by something else that is a lot more important (routing, stack up, etc.).

If you can solve that problem first, then you might find that you don't need copper pour.