STACK-UP STRATEGY

PART 2: DFM CONSIDERATIONS

SEGMENTS OF SANITY FROM THE GUIDE

EMA Design Automation®

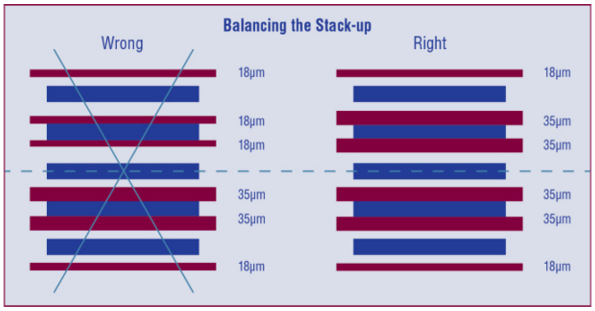
Selecting a Sound PCB Stack-Up Strategy: DFM Considerations

Due to its effect on both a PCB's electrical and mechanical performance, selecting a sound PCB stack-up strategy is vital to creating a reliable PCB. Part One of this two-part series discussed cost adders when doing so. Here, we discuss some added DFM considerations to ensure a manufacturable, flat PCB.

Balancing the Stack-Up

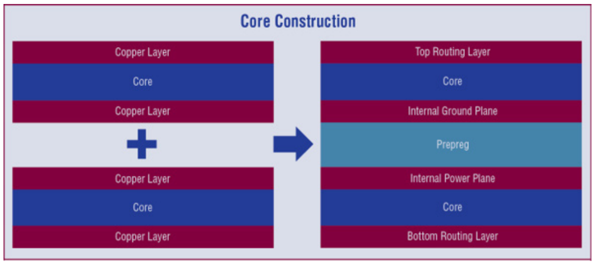
PCB warping can occur in a board when a designer is not aware of and/or does nothing to mitigate mechanical stresses. These can build up when laminating dissimilar layers of copper with various thickness combinations of glass-epoxy laminates.

Ensure the materials on each side of the PCB stack-up centerline are matched. If the stack-up is laminated with disproportionate materials and material thicknesses, the unbalanced stack-up will experience warping due to built-in stresses incurred after heated lamination and cooling.



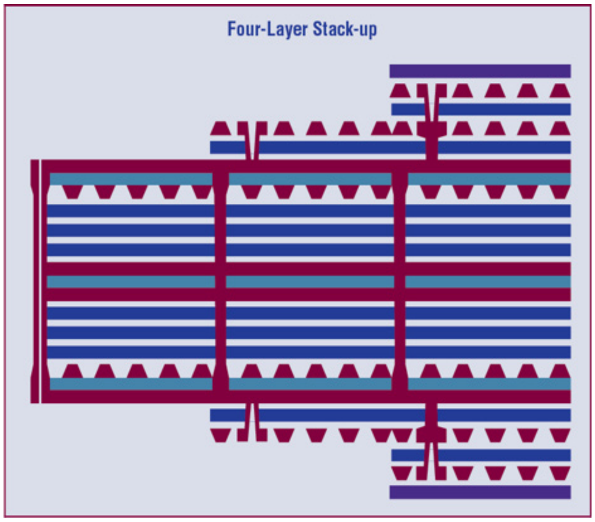
Core Construction

Core construction builds up a multi-layer PCB. Double-sided cores are printed and etched, then bonded together with a material called 'prepreg' or 'B-stage' material. It remains tacky as the board layers are stacked up in a press, curing when heat and pressure are applied. The process is mostly utilized on microwave PCBs where very expensive, high-tech microwave laminates are combined with low cost laminates to achieve a compromise solution for performance and price. Made up of core materials on the outer layers, this type of stack-up is called a "core build."



Foil Construction

The most common method for laminating multi-layer designs is the foil construction method combined with a sequential lamination process. In sequential lamination, cores may be individually printed, etched, and even drilled and plated prior to being laminated together with other cores. Individually selected sheets of copper and prepreg laminate materials create a diverse combination of layer interconnects. The foil construction methodology also has the advantage of "dialing in" distinctive combinations of glass-epoxy weaves which use unique resin-to-glass ratios, yielding exceptional dielectric properties to help control trace impedance as required.



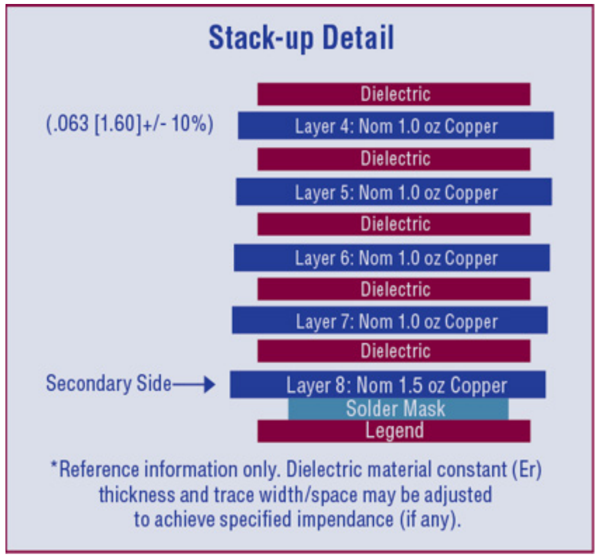
Specifying Impedance

It is not enough to simply specify an impedance on PCB traces. The impedance must be achievable in manufacturing, based upon good stack-up design. The PCB fab shop is allowed three ways to adjust trace impedance accurately on a PCB: **dielectric and value of the material, trace distance from reference plane(s),** and **trace width and spacing.**

When designers select trace widths based upon values determined by an impedance calculator, the manufacturing team can easily adjust the dielectric strength and thickness of the prepreg materials in stock. The different materials can be used to adjust trace distance from the reference plane to achieve the specified impedance value, and trace width and spacing may be dialed in through the CAM department. However, the PCB fabricator can only achieve impedance in this way if it is noted on the PCB fabrication drawing. The note must state that the three specific characteristics may be varied.

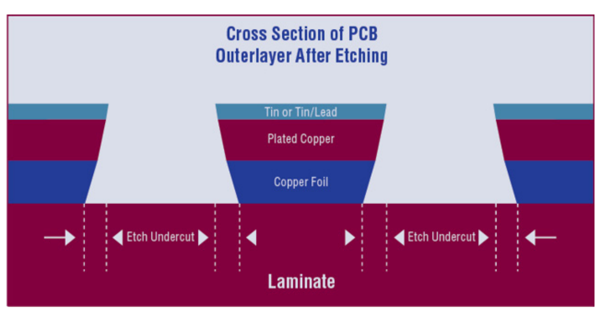
There are sometimes specialized customer engineering requirements which will not tolerate variations in physical composition or structure. Hybrid material-controlled stack-up specification is less common and usually specified when PCBs will be required to fly into space or other specialized circumstances.

This methodology is very expensive and results in long lead times due to the need for specialized materials and processes. If material-controlled stack-up methodology is not required for performance, it must be eliminated from the design process as it will mostly yield higher-cost parts or a no-bid on a request for quote.



Trace Width vs. Copper Thickness

Sometimes making the decision to add layer pairs to a PCB stackup is based upon a designer running out of routing space. One way to handle this is to examine whether shrinking the trace widths and spacing would allow all the routing to fit. However, this option must be checked in with the PCB fabrication stakeholder. The print-and-etch factors for trace widths must stay proportionate to each other, and must be reflected in the PCB stack-up detail. Due to the nature of printing and etching processes, the acids attack the sides of the traces in contact with the substrate material more aggressively causing a trapezoidal effect. If copper thickness is not reduced, the copper width at the base may become too narrow and fail:

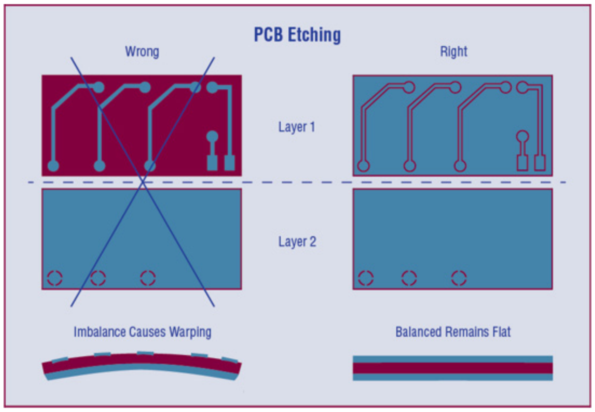


Selecting a Sound PCB Stack-Up Strategy: DFM Considerations

Balanced Etching Between Layers

Besides etching considerations for trace routing, imbalanced etching of larger copper areas of the PCB can also contribute to layer stress and warpage and will cause an automatic balancing challenge for the PCB fabricator. Commonly, PCB core laminates are supplied with copper foil laminated evenly to both sides. The laminated copper expands and contracts at a much different rate than the glass-epoxy laminates, but when the copper exists on both sides of the glass-epoxy equally, the stresses are evened out, allowing the core to remain flat.

Challenges are introduced when a large percentage of the copper is only etched off one-side of the core. If there are dense copper patterns on one side, try including copper ‘fill flooding’ by using ground fills on the opposite side to balance out the copper on each side. In the case of multi-layer PCB stack-ups, do not randomly flood every layer with copper flooding for the cause of balancing the stack. In general, every signal layer deserves a solid return path laminated adjacently to it in the stack-up for signal integrity purposes.



PCB flatness is achieved by sound design, not brute force. While it is easy for the PCB fabrication engineers to mechanically warp boards in the opposite direction to meet any flatness specifications for problem designs, it would also be very unethical. Upon experiencing heat in the wave solder or reflow oven, an unbalanced PCB will again warp to its unconstrained equilibrium, and the problem will be once more exposed.

Conclusion

Keeping DFM considerations in mind when creating a PCB stack-up will help decide critical issues in a project. Properly balancing the copper in the design stack-up during layout is the best preventative method to achieve PCB flatness and a manufacturable board.

For more, download The Hitchhikers Guide to PCB Design.