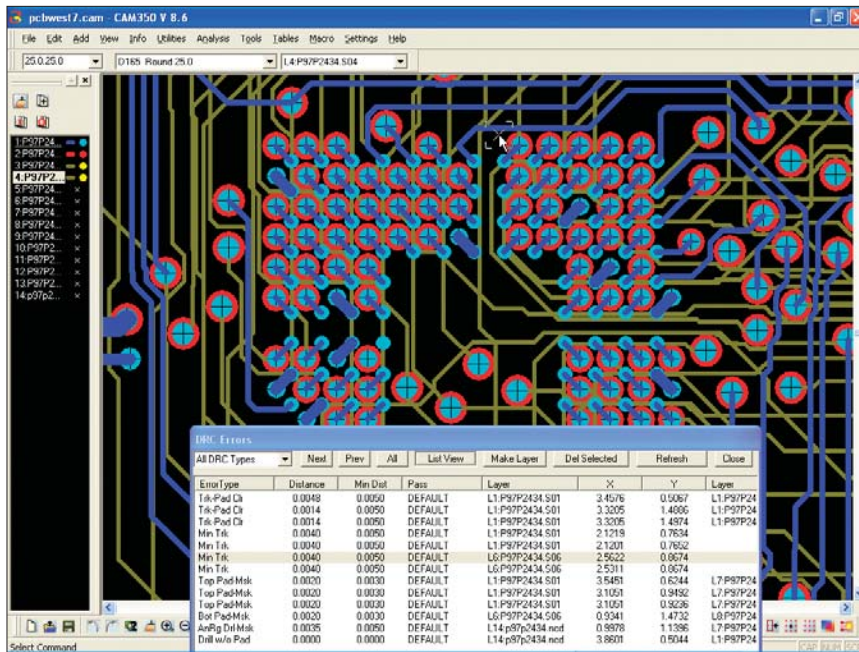


Design for Fabrication



Design for Fabrication is exactly as it sounds, the process of checking a PCB design before it goes to manufacturing for fabrication complications. Performing this function better prepares the design for fabrication.

During this process, potential fabrication issues are flagged allowing the designer the opportunity to fix or address them.

Implementing DFF functionality into the process ensures PCB designs will be manufactured more effectively, efficiently and successfully, while maintaining the original design intent and database integrity.

DFF Audit

DFF Audit performs over 80 essential bare-board analysis checks – automatically checking for acid traps, soldermask slivers, copper slivers, silk screen, power and ground, signal layers, starved thermals, soldermask coverage, and many more.

Making sure the soldermask data is generated using proper clearances, ensuring that there are no potential solder bridge conditions, and fixing potential acid traps will eliminate bottlenecks in the CAM department of any fabrication shop.

- Analysis checks take only seconds... even on complex boards
- Analysis results are graphically displayed for trouble-free viewing
- Violations can be easily inspected and potential problems quickly avoided
- Data integrity is maintained by making alterations directly to the database

Design for Fabrication

When a design issue is found in fabrication, the fabricator must attempt a “work around” and sometimes changes the design. Performing DFF functions during design ensures that the original design intent remains intact.

Example

If an acid trap is detected in fabrication, copper will be dropped into the crevice, which will solve the manufacturing issue. But, the designer might have preferred to reroute the trace completely, removing the chance of the acid trap altogether.

By addressing this issue in fabrication, the design database may not get updated with this vital change and as a result, the integrity of the design may be compromised. If the design is re-spun in the future, this issue might be overlooked resulting in scrap, or worse, faulty boards.

In Summary

When a PCB design is created with fabricator's requirements in mind, quality design files are the result. This saves time, resources, and money, while getting the product to market the fastest way possible.

Locate, identify, and quickly fix manufacturing violations before submitting your design files for fabrication, not after.