Case Two - Optics Component Company

An established optic component company understands and stresses the importance of Design Rule Checking

Luminous output their Gerber files without analysis or question. The Gerber file, not to their knowledge, had a specified decimal accuracy, but the drill file required a different decimal accuracy.

The Gerber files were sent to fabrication...

- Drill files created with different decimal accuracy from Gerber files
- Fabricator holds up bare board production due to misalignment of drills on artwork pads

The fabricator would **NOT** build the boards due to their CAM350 system flagging the discrepancy as mis-aligned drills.

Rules and Requirements Come Hand-in-Hand

Verifying the design complies with both internal design rules and PCB fabrication guidelines will ensure successful board production. Fabrication Design Rule Checking is vital to this process. Implementing this will prevent:

- Accidental pad/drill misalignment caused by limitations of CAD system
- Altering of artwork to meet fabrication requirements, changing design intent
- Discontinuity in finished PCB when using multiple fabricators for prototype and production
- Insufficient clearance from drilled hole to copper on high voltage layers

The Result

After much wasted time disputing the Gerber files, the fabricator finally used CAM350 to show Luminous the issue. CAM350 identified the problem quickly and easily. Luminous's claim that the Gerbers were fine was quickly negated.

Objectively solving the issue, the fabrication facility saved both themselves and Luminous money by avoiding building bad boards. This facility still services Luminous.

In Conclusion

Luminous is now in serious evaluation of CAM350 as part of their post layout verification process.