



## Don't trust your Gerbers!

**Warning: The Gerber files exported from your PCB Design software may be Incorrect!**

*By Simon Garrison, Numerical Innovations*

Most PCB design software packages are very liberal in their interpretation of the Gerber format, and usually output CAM files (i.e. Gerber, drill, ODB++, etc.) that are not to specifications. There's a very good possibility your Gerber files have not been exported correctly, and may be interpreted incorrectly by your PCB Fabricator's CAM software.

This is why almost every PCB designer has to use a Gerber viewer afterwards to look at their Gerber files before sending them out... but unless you have eyes like a hawk, simply staring at Gerber files inside a Gerber viewer (especially with hundreds of signals & thru holes) is just an invitation for mistakes. It may have been understandable in the past for a designer to rely solely on a Gerber viewer but nowadays in these competitive economic times you can't afford to just stare at Gerber files and assume everything will be okay.

### **CAM Software is the Perfect Solution**

CAM Software is to PCB designs what Print Preview is to Word Processor documents. Just as you wouldn't think of printing your documents without previewing them first, the exact same should be held true with your PCB designs. Remember Murphy's Law "If something can go wrong, it probably will." Make sure that you eliminate any possibilities of something going wrong by using CAM software to View & Verify your PCB designs are ready before sending to manufacturing.

***Every Year PCB Designers waste over \$30 Million (USD) on failures & repairs that could have been prevented with CAM Software. That averages to \$500 (USD) lost each year per PCB designer!***

**Avoid the Mistakes** - When there are shorts, weak track/pad connections, wrong part footprint used, silkscreen over mask openings, etc.

**Avoid the Frustration** - When PCB boards come back wrong or they fail out in the field.

**Avoid the Embarrassment** - When projects are delayed and thousands of dollars are wasted.

Be sure to load all your Gerber & drill files into CAM Software, view & run DRC/DFM, and then output accurate Gerber files; with the peace of mind your design is now ready for manufacturing! Using CAM software will not only prevent unwanted problems hidden in your Gerber files, it will uncover design flaws not seen in CAD, improve yields, lower manufacturing costs, and increase your time to market.

Numerical Innovations, LLC (NI) has developed the industry's first comprehensive Viewing & DFM checker tool that runs directly on the user's desktop for Free. FAB 3000™ with Free DFM Checker provides PCB designers worldwide the added confidence in knowing that their Gerber & Drill data have been pre-checked by the FAB 3000 DFM Checker - and they will not have any unwanted delays in getting their boards produced. The DFM Checker is extremely easy to operate by simply importing the Gerber & Drill files into FAB 3000, assigning layer types, and clicking a button to begin; if manufacturing violations are detected a description and count will be displayed. For more information please visit the NI website: <http://www.numericalinnovations.com>.