



GerbTool V15.0 **Release Highlights (FINAL)**

May 15, 2006

New Features of Version 15.0

REVISED PRODUCT CONFIGURATIONS

Helping the industry strive towards a more intelligent transfer of data from Design to Manufacturing, WISE has revised the product configurations for GT-Communicator, GT-Inspector, and GT-Designer so that they now include support for ODB++, ODB++ (X), and IPC-2581. All of these interfaces will cover both importing and exporting. Using these interfaces will help you to be more compatible with CAD systems from vendors like Cadence and Mentor, who are now including output to ODB++. In addition, WISE is anticipating the adoption of IPC-2581 by various CAD vendors as their "default" intelligent data transfer format, and is making this interface available now to help you become more familiar with it.

Highlights:

- ODB++ Import and Export are now included at no additional charge within GT-Communicator, GT-Inspector, and GT-Designer.
- ODB++ (X) Import and Export are now included at no additional charge within GT-Communicator, GT-Inspector, and GT-Designer.
- IPC-2581 Import and Export are now included at no additional charge within GT-Communicator, GT-Inspector, and GT-Designer.

HYPERNETLIST GENERATION

Netlist generation is a "core" feature in many processes within GerbTool, so performance in this area is critical. You will benefit from performance increases of 15-20 times that of any previous GerbTool release. CAM350 users who have recently migrated to GerbTool will see a 5-10 times increase over their previous product. HyperNETLIST generation brings the performance of GerbTool on-par with high-end tools, such as Valor's Enterprise3000 and Mania Technologie's UCAM. This increase in performance will allow you to complete processes like netlist comparison in under 15-minutes total processing time, for both the netlist generation and the comparison itself. These extreme throughput increases translate directly to your ability to perform the checks and analysis you need, without negatively impacting your overall processing time, thus improving not only the accuracy of your design but also improving your time-to-market.

Highlights:

- Faster netlist generation means less impact on overall processing time, thus helping you to improve your accuracy and time-to-market.
- 5-10X faster performance than CAM350 netlist generation on most designs.
- 15-20X faster performance than any previous GerbTool product release, on most designs.

HYPERNETLIST COMPARISON

One of the primary reasons you want to generate a netlist from your Gerber data is so that you can then do a comparison against a CAD-generated IPC-D-356 netlist. Performance and accuracy increases being one of the keys to this release, netlist comparison has also been enhanced to operate at optimal levels. You can expect to see roughly a 5X improvement in comparison times over previous releases. Helping you to leverage the error results to their maximum benefit, we have now included a new Pin-Point Error feature designed to zero-in on any shorts or opens that were detected during the comparison process. This feature alone can tremendously reduce the amount of time necessary to troubleshoot a job, by eliminating the need for you to have to visually trace the offending nets involved within the short or open error.

Highlights:

- Netlist comparison performance that is comparable to Valor Enterprise3000 and Mania UCAM, on most designs.
- 5X faster performance than any previous GerbTool product release, on most designs.
- New Pin-Point Error technology is designed to help you zero-in on shorts or opens.

AUTOMATIC DRAWN PAD CONVERSION

You will enjoy the ability to process an entire data set “hands-off” with our new Automatic Drawn Pad Conversion feature. Automatic Drawn Pad Conversion will process your data significantly faster than ever before, with the added benefit of maintaining pads like rounded rectangles with standard intrinsic apertures – there is no need for custom apertures. In cases where a conversion could not occur, an error is logged within the Analysis tree, where you can review and correct as required.

Highlights:

- Converts drawn pads to flash pads quickly and efficiently in a true “hands-off” manner.
- Uses solder mask layer for accurate analysis of pads that need to be converted. The Solder mask layer does not have to be flashed.
- Supports the conversion of rounded rectangles with an intrinsic aperture; no custom apertures required.

STENCIL ENHANCEMENTS

We first introduced the Stencil Enhancement feature in GerbTool version 14. Working with some of our existing stencil users, we have further optimized the Stencil Enhancement feature to be more flexible in its ability to accommodate a wide range of processing techniques. Originally the Stencil Enhancement feature was designed to work in conjunction with the Paste Mask Generator/Optimizer; however there are times when some users need to work from an existing paste mask layer, without any optimization, and make enhancements to specific pad-pair openings that are necessary to comply with a customer’s requirements. To this end, now you can increase or decrease openings while simultaneously adjusting the shape configuration of a particular pad-pair. In addition, rounded corners can be applied to all enhanced shapes. You will also benefit from the ability to work with either percentages or physical amounts when developing your rules for pad-pair enhancement. You can also “justify” your enhanced openings along any edge of the copper pad that the opening is associated with.

Highlights:

- Added support for “justification” of the paste mask opening being enhanced.
- Revised data entry fields to support “increases” or “decreases” by using positive or negative numbers on either a percentage or physical amount basis.
- Increased enhancement control with support for rounded corners on all shapes as well as height and width control.