



# GerbTool 16.4 Release Highlights

---

March 20, 2014

## NEW FEATURES:

### Cross Probe with Cadence Allegro/OrCAD

Cross Probing allows GerbTool and Cadence Allegro/OrCAD to synchronize views, layers, and the highlighting of specific database items.

- Cross Probe using Zoom Commands View Window, Zoom In, Zoom Out, Pan Window and View All.
- Cross Probe Items using Query Commands Query Item and Query Net.
- Cross Probe Analysis Errors - Locate the desired Analysis entry in the Navigator and double click it. The Allegro system will zoom to the location of the error.
- Right click an item in the main window and select Cross Probe or Cross Probe Net.
- Cross Probe from Navigator.
- Cross Probe nets from the Navigator. Locate an interesting net and either double click on it or select Highlight from its right click menu. The Allegro system will zoom to the item.

### Cadence NetShorts

- Added support for Cadence NetShorts in both IPC2581 and IPC356.
- DRC/NLC analyses now suppress errors that are associated with the specified nets & locations.

### Windows 8.1 and Windows Server 2012 Operating System Compatibility

WISE Software has always been committed to insuring the highest level of compatibility with the Windows™ operating system. For this release we have made the necessary changes to ensure that GerbTool is compatible with the Windows 8.1 and Windows Server 2012 environments. This includes making sure that all necessary drivers, for components like hardware keys and FlexLM™, have been updated and tested for compatibility.

### GenCAD Import

- Added import of GenCAD files.

## **Stencil Tools Additions**

- Quick Convert command gains the ability to reduce size by percentage and convert to window pane.
- Added secondary built-in stroke font for stencil text along with three TrueType stencil fonts.
  - Stroke-Stencil
  - Advanced Dot Digital-7
  - Enhanced Dot Digital-7

## **V-Score Support (Included with Advanced NC Option)**

New layer type of V-Score added to support the New command Add/V-Cut to easily add V-Score lines and properties to your design.

## **Consolidate Vector Items**

New command Consolidate Vector Items converts all vectored items of the same line width into a minimum set of polylines. Items included are draws, arcs, and polylines. This command is very useful in creating a clean Border layer to be used in several processes within GerbTool.

# **USER REQUESTED ENHANCEMENTS AND FIXES:**

## **IPC2581**

- Now imports PhyNet physical net section into external net data to allow net list comparison.
- Added support for net names with "." and "+" characters.
- Increased support for via tagging.
- Export no longer outputs empty "set" elements.
- Increased use of step and repeat nx and dx elements.
- Fixed character width problem with import of text entities.
- Fixed metric export rounding issue.
- Added V-Cut support.

## **Advanced Panelization**

- Added support to choose the panel border as the border instead of simple bounding box.
- Added "Negative Plane Clearance" option on Venting & Thieving tab.
- Added "Clearance from" options on the Robber Bars tab.
- Added "Clearance from" options on the Venting & Thieving tab.
- Fixed panel image not updating the extents after image file changes.
- Improve locating image files when opening a panel file.
- Fixed Venting & Thieving Negative Dot Pattern clearance problem.
- Fixed panel layer sets that were not being updated when a layer is deleted or moved.
- Image file changes are not being updated in layer mapping information.
- Fixed the Panel Layout margins to keep the last settings.
- Fixed Layer Mapping issues.

## **Macro Enhancements**

- Added print/page setup macro command.
- Added IPC 2581 import in macros.
- Added ODB import in macros.
- Added command to support a list of choices.
- Added macro command to return the length of an item.
- Added GETTEMPLAYER and FREEMPLAYER macros.
- Added GETARRAYSIZE and ISARRAY macro commands.
- Added FILEGETNAME and GETAPPDATAFOLDER macro commands.

## **Other Enhancements**

- ✓ Added display of analysis error properties in Item Properties dialog.
- ✓ Added ability to see the total number of errors next to each type in the error lists.
- ✓ Added Direct to Plane DRC check with option to include or exclude plated, unplated, vias.
- ✓ Added export of PNG, JPG and TIFF formats to the bitmap export command.
- ✓ Added ODB++ Version 8 support
- ✓ Edit Clip command enhancements:
  - Clip by Circle, choose from center, diameter or size
  - Clip "outside", either Circle or Rectangle and by center, diameter or size.
- ✓ Added alternate key mapping for keyboards without Home/End keys.
- ✓ Toolbar view commands redraw, view all, and view filmbox now work as nested commands like their keyboard equivalents.
- ✓ NC import dialog now saves the Max/Min default settings.
- ✓ Fixed bitmap export problem with large horizontal bands of data missing from advanced panel design.
- ✓ Fixed Fire9000 import for these specific files.
- ✓ Fixed importing of Gerber file with extremely flat arcs. (Fixed import and suggested Allegro export setting that would eliminate such arcs).
- ✓ Added ability to remove border text layer name field without layer number during print.
- ✓ Added CURRENTFOLDER in the print page setup to allow current Windows folder name to be displayed in the border text.
- ✓ Fixed print border text: \$SCALE not updating after fit to page has been turned off.
- ✓ Added warning message while entering of a Mill Path when the tool size doesn't fit an inner contour.
- ✓ Improved the Pin-Point error success rate.
- ✓ Added auto map of imported apertures over 10,000.