

NEW FEATURES and COMMANDS

NEW – 64-bit architecture

Moving to a 64-bit architecture allows GerbTool to make full use of all the memory available in the Windows 64-bit address space. This removes the restrictive 4GB memory limit imposed on 32-bit applications. This allows us to support significantly larger databases with increased accuracy.

NEW – Enhanced View Tabs

View tabs now save/restore all view settings including viewed layers.

- We have added the optional ability to automatically load default View Tabs for ease of viewing and working with your data
- Customize your own working environment by adding additional view tabs and controls
- View Tabs support the new WSSI_SITE environment variable so your whole team can use the same environment setup

NEW – Site wide environment control

Control of several configuration options at a network level is now available via the new WSSI_SITE environment variable. Some of the settings available are:

- View Tab Settings
- Loading Macros
- User Menu
- DRC/DFM Rule Sets including dynamic rule sets
- Advanced Panel Patterns
- Custom Apertures Loaded

NEW - Options for object snapping

Using the HOME key during operations which require a coordinate, now snaps to the nearest end or center of lines, arcs, and polyline or polygon segments. Using SHIFT+HOME allows snapping to the nearest point or centerline on the edge of an item. CTRL+HOME and CTRL+SHIFT+HOME continue to pop the Absolute coordinates dialog when snapping.

NEW – Fill Closed Polylines

It is now quite simple to turn an entire design worth of drawn outlines into filled polygons with voids. Options for cleaning up poorly drawn outlines help to solve some of the toughest data manipulation needs.

NEW – Updated Dimensioning Tools

Modeless dialogs for Drawing/Dimensioning allow on-the-fly switching of parameters for a faster, smoother workflow.

- Changed ordinate dimensioning to use a single base point per drawing layer
- Automatically create a drawing layer if one does not exist during Add/Dimension command
- Dimension changing to Horizontal/Vertical ask for the position of the text
- Note Balloon shape now surrounds the full text

NEW – Test Point Analysis “Optional Module” for GerbTool

Design for Test (DFT) analysis checks have been implemented for Test Point locations defined in IPC-2581 files. This is an optional module for GT-Inspector and GT-Designer.

New checks include:

- Test Point to Test Point
- Test Point to Exposed Copper
- Test Point to Mill Path
- Test Point to Border
- Test Point to Component
- Test Point to Mask
- Missing Test Point

IPC-2581 Enhancements and Fixes:

- Added support for Xform on Standard Primitive for Zuken files
- Fixed issues loading IPC2581 files from Altium
- TestPoint attribute in PhyNet is now imported into GerbTool
- Fixed ColorTerm attribute ordering
- Now Exports "Internal Layer" type None as Internal
- Fixed exporting of blind and buried layers when dielectric layers are present
- Layer Type "Other" Missing in 2581 export
- Layer Type of Immersion Tin Top and Bottom set wrong on import
- Losing Layer "Immersion" on export of 2581
- Dimension layer is now exported to 2581
- Method to bypass 2581 import dialog - added check box to the dialog if checked will only display the dialog if Shift key is pressed
- Fixed issue where step names were changing to lower case on export

ODB++ Enhancements and Fixes

- Fixed import problem producing bad pad angles
- Fixed this file import with unrecognized fields
- Added support for "Poynting for Microwave" software
- Fixed import of file with filenames that are all numeric

Macro Developer Enhancements

- NEW - DefineForm macro command to enable the design and use of a custom dialog with the following types of form controls:
 - EditControl
 - MultiLineEditControl
 - DropListControl
 - ListControl
 - ButtonControl
 - FileBrowserControl
 - StaticText
 - AnalysisErrorReport
 - GetItemArea
- NEW - Macro command for Hole Charts both one up and panel designs
- NEW - Macro command to access FHS via GET and PUTNCTOOLDESIGNDATA
- Fixed - DCEXPAND macro command no longer requires the layer to be visible
- Fixed - TextReplace macro command problem fixed
- Fixed - Macro Crash removing layers
- Fixed - Stacked Pads macro command no longer requires the layer to be visible
- Fixed - FILEWRITE command parser issue

DRC-DFM Enhancements and Fixes

- Solder Bridge checks now run much faster
- Added "Single Error Highlight" option when viewing DFM errors
- Fixed false Copper Sliver error
- Fixed Netlist error with extremely long arcs
- Fixed duplicate errors produced by Border check
- Added support to ignore external nets named "unused" during netlist comparisons
- Fixed "Drill to Copper" error issue on plane layers
- Added refresh of "ERR" tab after loading in a new file
- Data items with NonstandardAttribute name="TEXT" are now ignored during Acid Trap, Copper Sliver, and Stubs detection

Other Misc

- View > Restore Toolbars command will now load factory defaults
- License Activation Email now works with 64-bit Outlook
- Fixed "Custom to Intrinsic" not creating intrinsic when expected
- Consolidate vectored items now adds a segment rather than snapping endpoints
- Added support for new constructs found in CAM350 file
- Fixed issue creating Hole Charts that have more than 12 legends
- Added support to keep rotation of pad shape during Drawn Pad Conversion
- Added support to analyze and consolidate custom apertures when the same graphics are found at different rotations
- Added ability to control layer visibility after multi-layer format import
- Fixed missing clearance in custom aperture that contains cutline polygons
- Isolated Pad Removal using Drill Pads Only now removes same size
- Fixed Convert Drawn Pads alters these window panes in unexpected ways
- Fixed Dimension issue not updating aperture used
- Fixed GerbTool Stencil Setup Shapes issue
- Fixed GerbTool Paste Shape orientation wrong on export of 2581