

NEW FEATURES and COMMANDS

NEW - Via Analysis checks

Full complement of DRC checks specific to vias. Via pads and drills are known when importing from intelligent databases such as IPC-2581, ODB++ and GenCAD and can now have separate DRC rules from regular pads and drills. These new checks include:

- **Via/Via** Minimum spacing allowed between vias.
- **Via/Pin** Minimum spacing allowed between vias and component pins.
- **Via/Trace** Minimum spacing allowed between pads and traces.
- **Via/Border** Minimum spacing allowed between vias and the border.
- **Via Min Size** Minimum via size allowed.
- **Via/Drill Annulus** Minimum annular ring required between vias and drills.
- **Via Drill/Direct to Plane** Checks for via drill locations on plane layers that represent a direct connection.
- **Via Drill/Adjacency** Minimum spacing required between adjacent via drill locations.
- **Via/Top Mask & Via/Bot Mask** Minimum annular ring required between vias and the top/bottom solder mask.
- **Via Drill/Top Mask & Via Drill/Bot Mask** Minimum annular ring required between via drills and the top/bottom solder mask.
- **Via/Top Silk & Via/Bot Silk** Minimum clearance required between vias and the top/bottom silkscreen layer.
- **Via/Missing Top Mask & Via/Missing Bot Mask** Checks for vias without openings on the top/bottom solder mask.

NEW - Ability to ignore specific analysis errors on subsequent runs

The ignore error option allows you to ignore individual analysis errors, hiding the error from the normal error list and causing that error to not be reported during future analysis runs with the same settings. DRC, DFF, Netlist Compare and Layer Compare errors can be tagged to be ignored. Right click on an error in the navigator and choose "Ignore Error". The selected error is moved to the ignored error list and will be ignored in any future analysis that generates that error. When there are ignored errors defined, the "Ignored Errors" list will be the first list in the analysis navigator.

NEW - Add Barcode

This command allows you to define and add a barcode to your design. Create Code 39, Code 128 and Dot Matrix barcodes. Use Inverted option of the barcodes to create the barcode in a copper area. Once created your barcode will be added to the aperture table as a custom.

NEW - Create Mill Paths from Tabs

Added a new command to the Tools/NC menu to create mill paths from break tabs. You can now easily create new mill paths on a separate NC Layer so the routing machine can remove the tabs.

NEW - Flatten Customs on Export of Gerber

Choosing this option in the export settings for Gerber prevents the use of aperture macros in the exported Gerber file. With this option on the custom aperture macros will be flattened and output as polygons.

NEW - Support for Peck Drilling, Pinless Rout and Depth Control

- Added support for peck drilling (Sieb & Meyer G81/G80, Excellon M62/m63) and Sieb & Meyer pinless rout (G11) command.
- Added support for depth control parameters Z, K and H. The "Sieb & Meyer Properties" dialog has controls to set the following depth control options:
 - Absolute Working Plane (Z): Distance between the table surface and the absolute working plane.
 - Relative Working Plane (K): Distance between the board surface and the relative working plane.
 - Absolute Traveling Plane (H): Distance between the table surface and the absolute traveling plane.

VisualCAM Stencils Enhancements

New Built-in Paste Shape types

- New built-in shapes of V-notch and Perimeter Band.
- Array shape now supports Square dots with optional corner radius or chamfer corner. Also added the ability to control the minimum space between dots.

Flatten Customs for Gerber Export

Choosing this option in the export settings for Gerber causes all apertures that would be created as customs to be output as polygons instead. This creates compatibility with laser machines that do not support custom apertures.

Decimals allowed in percentages

When modifying a shape by percentage, more precise values up to three digits after the decimal are now supported.

Import NC data

You now have Import NC and NC Tool list menu commands

- Fixed issue with Width and Length reductions on customs
- Inside Corner Radius are now saved with library for MELF shape
- Fixed dialog issue with pins set to "No Paste" changing to "None"
- Fixed Area Ratio calculation problem on customs
- Fixed issue loading ODB++ files into the Shape Library
- Fixed issue identifying specific part at correct angle
- Fixed issue with Delete/Rename in the Footprint Library
- Library and Quick Convert can now undersize customs by width and length independently

IPC-2581 Enhancements and Fixes

- REVB1 Optional Export
- Added support for B1 Name attribute in the Property element
- Export now supports Modes and optional layer inclusion/exclusion
- IPC2581 Specs and Designer Notes are now displayed in the Stackup section of the navigator
- Fixed incorrect application of the angle attribute

- Import Stackup speed improvement
- All True/False booleans are now lower case in 2581 Export
- Step and Repeat enhancement
- 2581 file from Eagle now imports
- FHS (finish hole size) in 2581 output
- Xform handling updated
- IPC-2581 BOM Mechanical section now passes through even if there are no Footprints
- Fixed BOM LayerRef problem
- Fixed IPC-2581 - TolPercent should be True not Yes
- Fixed Zuken IPC2581 REVB file import issues
- Fixed B1 Thermal "gap" should be "spoke width"

Macro Commands

- GENERATEVENTINGTHIEVING – Advanced Panel venting/thieving
- GETINSTANCECOUNT - gives you a count of how many of a given instance are in a panel design
- ADDVIEWTAB – allows you to add View Tabs through a macro
- ASSOCIATEPARTS - This command associates component part pins with existing layer pads
- ADDRECTROUNDED to support radius corners
- DELETEANALYSYSERRORS – Allows you to delete all lists from the analysis error section of the navigator
- STRLWR - converts all characters in a string to lowercase
- STRUPR - converts all characters in a string to uppercase
- DELETEALLPARTS - this command deletes all parts in a design
- Enhanced CLIP macro command
- Enhanced PADREMOVAL macro now supports a Tolerance if you want all pads that are offset from each other within a certain amount to be considered Stacked Pads
- Fixed \$\$DSNNAME System Variable to update after SAVEAS command
- Fixed \$\$MAXLAYERS to update whenever layer table size is increased
- Fixed issues running Quick Panel Macro on data that includes parts

Advanced Panelization

- Macros commands added to support automation of Advanced Panelization
 - Generate Venting and Thieving: GENERATEVENTINGTHIEVING
 - Get image instance Count: GETINSTANCECOUNT
- Import 2581 Panel now defaults to "Use Panel Border"
- Layer Type of Other now auto maps in Advanced Panel when name matches
- Importing Panel steps into VisualCAM from ODB++ or IPC-2581 files now creates .vcam images

Other

- Added ability to copy data to a V-Score Layer using copy commands or through the navigator
- Fixed load time on files with large number of segmented arc's
- Fixed issue with Pin Point errors during Save
- Fixed Chamfered corner issue coming out rounded after intrinsic conversion
- Fixed issue of unassociated parts during Isolated Pad Removal
- Added methods for determining footprint type and relative pin location for footprints with a mixture of round and rectangular pins
- Fixed Load and Save time with files that have very large netlist data sections

- Isolated Pad Removal Enhancements:
 - Added option to allow only isolated pads with a drill to be removed from inner layers
 - Changed to allow isolated pads that are completely drilled out on top/bot layers to be removed
 - Layers no longer have to be visible
 - Valid netlist is no longer required
 - Updated macro command to include new parameters DRILLEDPADSONLY and THRUHOLELAYER
- NC Change Tool table from the Navigator now works properly
- Fixed issue with NC Optimize by NC Group
- Fixed typo during tab load if existing tabs with the same name already exist
- Fixed Design Compare of Text showing false error
- Parts that become unassociated are no longer deleted when Top/Bottom Layers are deleted
- Fixed duplicate footprints issue when importing ODB designs and then merging libraries
- Fixed Show Unused Footprints issue when loading a Footprint Library
- ODB++ files will now load in even if the job folder is missing the standard font file
- Panelized designs opened in a macro will now automatically map new image layers
- Fixed issue of merging Footprint Libraries with same shape names
- Added option to combine D-Codes on Consolidate Vector Items command
- Fixed issue with Layer Compare error report not showing error type
- Export NC new verifies the NC Tool and Link Tool are not using the same Tool number
- Dimensioning Enhancements:
 - Added support for adding dimensions to data within panel instances. Dimensioning between data within an instance and panel data or data within other instances is permitted
 - Added support for polylines
 - Fixed problem with adding or modifying an existing dimension that resulted in other dimensions being corrupted
- Fixed issued with loading CAM350 Composite data
- Fixed Custom to intrinsic issue on these Gerber files during import
- Fixed issue with slow Paste generation and missing paste with this design.
- Fixed issue of importing this ODB++ design with EDA option
- Request for ability to copy a shaperset with its assignments
- Added FHS, Tol Plus, and Tol Minus to NC Tool Table and Hole Chart dialogs
- Added support to set FHS when setting Tolerance from the IDF file
- Fixed Custom to Intrinsic oblong shapes
- Added Tolerance to IDF Import
- Fixed issue of Mill path joining together even when user selected No in the dialog
- Fixed the Add Mill path changing the start point
- You can now change the active D-Code during Add Rectangle
- Fixed Custom to Intrinsic processing when processing arcs on this rotated rectangle
- Fixed issues with automatic Convert Drawn Pads
- Fixed rotated custom apertures coming from CAM350 data set
- Hotkey Y - Layer Setup now updates the Color Bar
- View Highlights Toolbar button has been updated to behave like a Hotkey