

Version 18 Software Update Details

Problem Fixes in Version 18.0.9 (26-Nov-2015)

Back Annotation

- The Back Annotation command was not able to correctly handle name changes were the only difference was a change in the case of the name.
- In a project where schematic sheet names had been changed without first back-annotating any outstanding name changes, in some situations this could prevent the back-annotation process from completing.

Component Edit

- Attempting to save an incomplete Component to the library could cause the application to quit when attempting to display an error message about an invalid component pin net name.

Display

- It was not possible to change the 'Fixed for Router' highlight colour.

Gerber

- The default resolution for coordinates in Gerber output has been increased to five digits to reduce the potential for rounding errors in shapes with very small arcs.
- Plotting the corners of rounded rectangle pad shapes used poor arc approximation to draw each corner.
- Pads using the "T" pad shape were malformed on paste and resist plots.

Measure

- Distances measured between items on non-electrical layers could be incorrect.

Move

- When moving a track segment ending on Vias that are fixed, the vias were still being moved.

ODB++ Output

- Some designs with nested board outlines were not being correctly constructed in the output files.

Optimise Nets

- A connection would not be added to a through-hole pad on a power-plane net where that pad was marked as "isolate from plane".

Pro-Router

- On a few specific designs, the router would simply 'flash' the progress bar for a moment and make no attempt to route the board.

Replace Component

- Replacing a two-gate component with one having only a single gate could in some circumstances lead to the application getting stuck.

Trace-Router

- After selecting specific net classes to be routed, the router would also process nets on other classes.

Windows Plotting/Printing

- Bitmaps on Windows printouts could be inverted or incorrectly placed.
 - TrueType text at some orientations could be misplaced on the printout, depending on the version of Windows.
-

Problem Fixes in Version 18.0.8 (02-Jun-2015)

Change Layer

- Using Change Layer on a Height Check Area in a board with more than two layers and when the current layer is the bottom layer would cause the application to quit.

Design Rule Check

- Several spurious pad-to-shape errors were being flagged on one specific PCB design.

Display

- Changing the visibility of Pour Areas had no effect, they were being drawn using the Routing Area visibility.

DXF

- Some circles were not being imported at the correct location.

ODB++

- Rotated annulus pads were coming out as solid filled round pads.

Open

- On opening files from a previous version, some circular board outline cutouts were being lost.

Pour Copper

- If a copper shape or pour area was not pre-assigned to a net but was only assigned to the required net in the Pour dialog, any net-based spacing rules were not used on the first time that copper was poured.

Pull-Tight

- When using Pull-Tight on a track, and in some interactive editing, it was possible that the resulting track would not recognise a board cutout as an obstacle and could thus cause the track to go off the board edge.

Select

- It was not possible to select the pad of a single-pad component that had no other content (no shapes or text) so that Properties of that pad could be viewed or altered.

Shape Bar

- The width of a rectangle was incorrectly constrained based on the wrong coordinate of the rectangle, thus limiting the size to which the rectangle could be changed.

Star Points

- It was not possible to place a free pad star point over a component pad star point without getting DRC errors.

Windows Plotting

- Printing certain bitmaps to a Windows printer could cause the application to quit.

Problem Fixes in Version 18.0.7 (20-Mar-2015)**BOM Composer**

- When grouping components in the listing, attempting to group by Package (Footprint/PCB Symbol) had no effect, potentially resulting in incorrect component data being included in the report.
- Columns were still being included in the CSV version of the report even when 'hidden' in the readable version.

DXF

- Some DXF files use codes 41 and 42 as an alternative to code 43 to define a POLYLINE, which was not being detected in the DXF Import module.

Forward Design Changes

- If a net in the schematic included components marked as 'suppress from SCM/PCB' this could cause the net in the PCB design to become unrouted.

Trace-Router and Pro-Router

- On some systems, installing update 18.0.6 resulted in one or other Auto-Router causing the application to quit on completion of the routing task.

Problem Fixes in Version 18.0.6 (13-Mar-2015)**Intelligent Gerber Import**

- Coordinate format was not always being correctly retrieved from the incoming Gerber file, resulting in a spurious 'negative value' warning when attempting to import.
- The coordinates for drill files were not always set up properly if no incoming Gerber files were set to assign their pads as through-hole.

ODB++

- Some pads were being incorrectly isolated from their plane when the thermal connection width was over a certain size.

Pour Copper

- One specific design containing pads with overlapping copper shapes used to create 'custom' pads would not isolate those pads from the plane during Pour.

Pro-Router

- It was not possible to route a design containing a component with the name 'NETWORK'.

Reports

- When collating component reference names in a column/field in a report, one of the names would be missing from the output if the text had to be 'wrapped' onto a second line.

Styles

- When creating a T-shaped pad, it was not possible to set the stem width greater than the length.

Problem Fixes in Version 18.0.5 (16-Jan-2015)

Add Track

- When using Orthogonal mode, sometimes some track segments were being left in 'dynamic' mode, which meant they became invisible when the picture was redrawn.

Cross-Probe

- Some nets in a Project could not be cross-probed if schematic sheets had been removed from the Project then re-added.
- Attempting to cross-probe a net from Schematic to PCB in a single-sheet Project was not finding the net in PCB.

Cursor

- If the cursor type was set to Cross Hair, moving the cursor using the arrow keys on the keyboard would leave the cross hair in its original position.

Design Technology

- Sorting items in tables into order did not take account of values to the right of the decimal point.
- It was possible to set Isolation Gap to a negative value, causing issues with Pour Copper.

Drawing

- Redrawing the picture was slowly consuming GDI Objects, eventually causing the application to quit, usually after several hours of usage within one application session.
- Circles for drilled-out free pads were sometimes being drawn away from the pad to which they belonged.

Forward Design Changes

- Running Forward Design Changes after renaming a sheet in a Project would sometimes unrout more tracks in the PCB than strictly necessary.

Libraries

- Some footprints in Prolib.psl in the Pro Library had the Reference origin layer set to something other than Silk.

Library Manager

- Creating a new library would frequently offer to save it in a folder that was not in the library folders list.

Projects

- The list of Bus names in a Project was not always complete, sometimes one or more names could be missing. This could mean that the name shows up in the Goto bar but Properties on the bus thought it had no name.

Reports

- Sometimes the text version of a report included part of the hyperlink from the corresponding HTML version.

Select

- It was possible to select a surface-mount pad based on the size of a pad style exception defined for a layer on which that pad did not exist. Thus if there was a bottom-layer exception with a much larger size than the top layer or default size, it was possible to pick the pad well outside its boundary.

Text Callouts

- Moving the arrow of a text callout with Box=None could cause the application to quit.

Problem Fixes in Version 18.0.4 (24-Nov-2014)

Add Track

- Using Ctrl+Click to change layer while over a track on the same net on the opposite side of the board could cause the application to quit.

Auto Rename

- It was not possible to use Auto Rename Component to replace the name stem of components in Schematic designs.
- Auto Rename Nets across a project sometimes did nothing.

Back Annotation

- Some projects would not back-annotate name changes from the PCB.

Display

- Visible and Selectable flags for areas were not being separated into the three types of area, changing the values for one type were affecting the others.

ODB++ Output

- Pads and vias with a plane connection type of "Isolated" can be included in the ODB++ output file with a normal thermal relief connection instead.

Projects

- When producing a new copy of a project and its contents by using **Save As** on the project, cancelling the copy of one of the Schematic pages could cause the application to quit when attempting to finalise the copy.

Ultiboard Import

- When reading Ultiboard files into a pin-limited version of the application, if the incoming file exceeded the pin limit then this was not detected during import, causing the application to quit.

Problem Fixes in Version 18.0.3 (23-Oct-2014)

Add Track

- Attempting to add a track to a component that had unconnected Star Point pads could cause the application to quit.

Back Annotation

- Changes to component names or component values were being recorded for components marked as "Suppressed from Schematic to/from PCB", even though those name changes could not be back-annotated to the Schematic.
- Rename Components (the interactive point-and-click rename function) in PCB was not marking the start and end of the batch of renames, which could result in the wrong names being applied to components in the Schematic.

Component Edit

- The Values page of the property dialog would not display in some circumstances.
- PCB-only component pads (pads that exist on the PCB symbol but do not appear in the pin mapping in the Component) were sometimes acquiring the same pin number as was already used on a multiply-mapped pin. For example, a two-pin PCB component with a single schematic pin mapped as "1+2" would have its second pin numbered as "2" where it should have been "3".

IDF Output

- For some third-party software packages, the rotation value for mirrored components needs to be reversed. This has been achieved via a new checkbox on the IDF dialog, allowing you to choose whether or not to apply the reverse rotation.

Import

- Library and design data from Eagle was not correctly importing some rounded component pads, such as those on two sides of some QFN devices.

Move

- When moving large portions of a PCB design with 'Finish Markers' visible, the application could suddenly slow down as it re-calculated the possible positions for drawing the finish markers.

PDF Plot

- Producing a PDF output of a power plane from a particular PCB design was causing the application to quit.

Pour Copper

- The 'minimum copper area' value from the Pour dialog was not being remembered.

Power Planes

- Sometimes the wrong isolation gap was being used when generating power planes (either when output to manufacturing files, or when drawn using View Powerplane).

Projects

- Adding a new schematic page to a Project was allowing the option to ignore any duplicate component names, instead of renaming them to ensure that the entire project had no duplicates.
- Using the Rename option to change the name of an existing page in a project would not allow you to use a page name that included dots.

Properties

- Some combinations of Thermal Rules were not being properly stored back into the design when using the Area page of the Properties dialog on a Copper Pour Area.

Reports

- The header section of Reports was not outputting anything for the Units of the design.

Rules

- The rules for how copper should be poured (with or without thermals on pads and/or vias) were not always being translated correctly to the new Thermal Rules scheme when reading designs produced in an earlier version of the application.

Star Points

- When trying to connect to (or route connections to) Star Points directly in a PCB design (rather than the usual method of bringing the connectivity over from the Schematic), the fact that the Star Point pads are on top of each other in the PCB meant that it was very difficult to select the correct pad, and hence choose the correct net.

Technology Files

- New Thermal Rule settings added to a PCB technology file were not being transferred to a new PCB which used that technology file as its starting point.

Problem Fixes in Version 18.0.2 (22-Sep-2014)

Add Track

- Choosing the Add Track command to begin adding tracks directly from the Add Via command could cause the application to quit.

BOM Composer

- When the general Report setting for HTML output was not enabled, BOM Composer would not generate the CSV version of the BOM even when this was selected on the BOM Composer dialog.

Change Layer

- Using Change Layer on a track to a via that is within a pad could cause a spurious additional via to be added.

Display

- The settings to control the colour and visibility of Symbol Origins was not available on the Display dialog when editing a symbol.

Keys

- Sometimes when attempting to move an item using the arrow (cursor) keys, the item would not move, or would move only in certain directions, unless it was first moved using the mouse.

Library Manager

- It was not possible to simply alter the case (switch between upper and lower case) of a name or part of a name using the Rename command in the Libraries dialog.

Measure

- When measuring the gap between pads, the gap displayed would always show the distance calculated on the electrical layer even when a non-electrical layer (such as Solder Mask) with an oversize was selected in the Measure dialog.

Pour Copper

- Component Value Positions placed on electrical layers were not being avoided when pouring copper.
- The presence of a Pour Keepout could prevent thermal spokes from being added to nearby pads outside that Keepout area.
- Hatch settings from the Area Properties were not being correctly carried forward to the Pour command, resulting in solid rather than hatched filling being produced.
- The value specified for Minimum Copper Area was not being correctly interpreted, sometimes leading to insufficient copper being generated during Pour.

Printing and Plotting

- When outputting a schematic design to Windows or PDF via Print or Printing and Plotting, when Autofill of symbol shapes was enabled, would sometimes print the filling in the wrong order causing it to obscure other detail within the symbol.
- When generating plots for a Panel design, no warning was given if the setting 'Plot Each PCB Design Contents' was not checked, potentially leading to incomplete manufacturing data being produced.

Rename Component

- Using the 'Selected Board' setting to rename components only within a specific board outline would cause the application to freeze.

Reverse View

- When drawing names inside tracks and pads in Reverse View, the names were drawn mirrored instead of being flipped back so that they were still readable.

Save Technology File

- Saving a PCB design containing Copper Pour Areas as a technology file would lose the association between Areas and their assigned Nets.

Styles

- When defining a T-shaped pad, it was not possible to define the size (depth) of the T-bar (crossbar) to be larger than the width.

TraceRouter

- The router was not always using the 'best fit' Layer Span when routing stubs on a multi-layer design with planes.

Problem Fixes in Version 18.0.1 (20-Aug-2014)

Add Component Panel

- The list of libraries used to populate the Add Component dockable view was not always being updated when the library Folder settings were altered.

BOM Composer

- The default template did not include 'subsidiary' value names R, C and L in the column for Value.
- Component Package and Description were not available to be chosen for a column.
- There was no ability to save the generated BOM as a CSV file.
- When collating component names onto a single line for one type of component, there was no option to output all the individual component reference names instead of using name ranges.
- Changing the order of items in the "Append To Value" dialog was not being saved when closing the dialog.
- The Grand Totals section still included an entry for Cost even after unchecking the Cost column.

Bus Properties

- Applying an existing bus name to a new bus in a multi-page project could cause the application to quit.
- The name of the selected bus was not being displayed on the Status Bar.

Drawing

- Automatic filling of component symbol shapes was not filling all possible shapes.

Duplicate

- Using Duplicate when placing a component in Add Component could cause the application to quit.
- Duplicating a selection was causing the application to hang when attempting to increment a net name finishing with "-1".

Gerber

- When plotting a rotated design containing bullet pads in a panel, some of the pads could be plotted at the wrong orientation.
- A 'positive plane and items' output could fail to isolate certain items with the correct thermal relief.
- Two specific designs failed to generate Gerber data for their planes.

Grid

- When adding tracks or schematic connections, snapping to grid sometimes jumps sideways.

PDF Output

- Auto-fill schematic symbol shapes were not being filled on PDF outputs.
- Auto-fill was not being suppressed for drawing outlines and other non-component items.

Plotting

- Closing a design after plotting, then opening another design to be plotted, could cause the application to quit.

Print

- Printing some multi-page schematic projects to PDF could cause the application to quit.