

Version 18 Software Update Details

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

Add Track

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

BOM Composer

- o 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

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

Display

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

Keys

- o 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

- o 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

- o 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

- o Component Value Positions placed on electrical layers were not being avoided when pouring copper.
- o The presence of a Pour Keepout could prevent thermal spokes from being added to nearby pads outside that Keepout area.
- o 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.
- o The value specified for Minimum Copper Area was not being correctly interpreted, sometimes leading to insufficient copper being generated during Pour.

Printing and Plotting

- o 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.
- o 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

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

Reverse View

- o 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

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

Styles

- o 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

- o 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

- o 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

- o The default template did not include 'subsidiary' value names R, C and L in the column for Value.
- o Component Package and Description were not available to be chosen for a column.
- o There was no ability to save the generated BOM as a CSV file.
- o 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.
- o Changing the order of items in the "Append To Value" dialog was not being saved when closing the dialog.
- o The Grand Totals section still included an entry for Cost even after unchecking the Cost column.

Bus Properties

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

Drawing

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

Duplicate

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

Gerber

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

Grid

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

PDF Output

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

Plotting

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

Print

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