

Version 17 Software Update Details

Problem Fixes in Version 17.0.4 (18-Mar-2014)

3D View

- Each successive redraw would increase memory usage.

Auto-Router

- When using the basic internal auto-router (Easy-Router), some track and via keepout areas were being ignored.

BoardMaker Import

- Duplicate pin names in the incoming data were not being intercepted during import.

Duplicate

- Duplicate didn't copy pad style exceptions from the original component instances.

Frame Select

- Multiple uses of Frame Select would lose the previous selection, and became stuck in Deselect mode.

Next Error

- While moving a component dragged from the Component Bin, pressing the hotkey for Next Error can cause the application to quit on releasing the moving component.

ODB++

- Tracks with width changes could result in incorrect data in eda\data file.

Paste

- It was not possible to Copy/Paste between Schematic Symbol and Schematic Design.

PDF Output

- Missing fonts were causing PDF output to fail.

Pour Copper

- Thermal spokes were being added even when inside a copper keepout area.

Reports

- Presence of test for 'IsScmOnly = false' in a report would cause PCB-only components to be omitted from the report.

Startup

- The application was failing to start on one specific Windows 7 computer.

Styles

- Unhelpful error message given if corner radius is too large for pad width in Pad Properties.

Trace Router

- Designs containing guard spacings would achieve lower completion rates following an earlier fix.

Translate to PCB

- There were some issues with Schematic-only and PCB-only components following an earlier fix.

Problem Fixes in Version 17.0.3 (10-Jan-2014)

Add Via

- When adding 'stand-alone' vias, each via being added would temporarily disappear from the screen while the next via was being placed.

Design Rule Check

- If a PCB design contained a Value Position that had somehow not been assigned a text style, attempting to run the Silkscreen Overlap check would cause the application to quit.

DXF

- When outputting DXF files from a PCB design, the section for plated holes was only being included if both plated and unplated checkboxes were ticked.

Library Manager

- If the 'Styles Tool' encountered a symbol that it could not read (perhaps one from a more recent version of the application), it would still attempt to access the information from that symbol and thus cause the application to quit.

ODB++

- Use of some pad styles including those using the 'Offset Bullet' shape could cause the resulting ODB++ file to be rejected by some third-party applications as unreadable.

Optimise Nets

- When removing existing connections to copper shapes during Optimise, the 'nodes' present in the copper shape for the end-points of the connection were not always being removed. This could result in layers being marked as 'Used' in the Design Technology dialog when no visible design items remained on those layers.

Pour Copper

- Geometry issues relating to several specific design files were causing incomplete or incorrect copper to be created.

Reports

- When generating the HTML version of reports for the Report Browser, if the contents of any column overflowed the designated size of that column, the formatting of the report would break down.
- When collating information into a single column in a report, the column formatting could break down or some items would not be included in the text for that column.
- When generating quoted output (usually used for producing CSV files), any text string that itself contained a quote was not being intercepted and dealt with (by doubling-up the quote character). This could result in the CSV file being rejected or malformed when read into other software such as OpenOffice.

Styles

- It was not possible to define a pad style using the Triangle pad shape where the width of the triangle was greater than the length.
- No warning was being given on opening a design containing text styles that referenced Fonts that were not present on the current computer. This situation will typically only happen if a special or custom Font is used, and that Font is not supplied when sending the design file to a user on a different computer.

TraceRouter

- Routing was not taking into account any Guard Spacing defined for nets in the PCB design.

Translate to PCB

- Components with no pins were not being added to the PCB. This is a historic issue where zero-pin components were designated as 'schematic only' even if they had a PCB package assigned to them. This restriction based on the number of pins is no longer required, as any components that should not transfer to the PCB can instead be marked using the 'Suppress' option.

Update Component

- In some circumstances, using Undo to remove changes made by Update Component could cause the application to quit.
- Some pad style exceptions were not being applied to an existing component when an updated version of the symbol was loaded from the library.

Problem Fixes in Version 17.0.2 (01-Nov-2013)

Add Shape

- Using Type Position (shortcut key "=") to define start point or corners when adding a closed shape in Orthogonal segment mode would simply redefine the start point of the shape each time.

Clearances

- When working in a Panel design, the clearances drawn around a panel outline when moving a board outline were incorrect as they did not use the correct spacing.

Component Edit

- After editing the Description, changes would not be saved unless you first clicked to another control or page in the component editor.
- Depending on which 'tab' in the component editor was last used, creating a new component using File, New could sometimes cause the application to quit.
- The interpretation of the 'all packages' checkbox and the result of adding values and whether or not they were added or altered in all packages was not consistent.

Cross-probe

- Cross-probing a component pin from Schematic to PCB could cause the application to quit if the PCB was not already open.

Integrity Check

- Swapping component reference names around in Schematic between components using different packages could trigger Integrity Check to change packages on components in the PCB rather than simply altering component names.

Intelligent Gerber Import

- Importing files in some situations without using a Technology file could cause the application to quit.

Libraries

- After using 'Copy To' to make a copy of an item within the same library, the new item did not show up in the list of library contents until you switched to look at a different library and back again.

Library Databook

- The application would quit while generating databook pages if it encountered a library item containing a forward or backward slash.

Nudge

- The DX,DY incremental values shown on the status bar did not appear when nudging selected items unless you started dragging them with the mouse first.

Support Report

- The report generated for sending technical information to the support team did not include information about some of the Microsoft runtime files.

Technology

- The Values Names page was not present on the Design Technology dialog when editing symbols, where it would be useful to manage value names for creation of Value Positions on the symbol.

Trunk Routing

- Using the command 'Push Tracks Further Apart' could sometimes cause the application to quit.

Functional Changes in Version 17.0.1 (11-Sep-2013)

Rulers

- The background colour of the horizontal and vertical rulers will change depending on the state of the "Snap to Ruler Stops" option. With this option enabled (the default setting), the background of the ruler will be light yellow. When snapping is turned off, the background colour will be white. This can be used as a useful indication of the current state of the snapping option without having to use the right-mouse popup menu on the ruler to examine the state of the switch.

Problem Fixes in Version 17.0.1 (11-Sep-2013)

3D View

- LED packages, wire pins and square pins were being drawn incorrectly.

Component Edit

- Invoking the 'Name Pins by Row/Column' command for a Schematic-only Component would cause the application to quit.
- When editing a multi-package component with the 'all packages' checkbox enabled, editing an existing Value or changing the visibility state using the checkbox in the list on the Values tab was not applying the same changes to other packages in the component.
- When adding a new package to a component with the 'all packages' checkbox enabled, values were not being copied across from the current package.
- Attempting to use shortcut keys to control the symbol previews (Zoom, Pan, etc.) did not always work, sometimes they had no effect, sometimes they went instead into the current cell in the Pin Assignments grid.

Component Heights

- When checking component heights, the height calculated from a 3D-View package was incorrect for some types of package.

Drawing

- Very shallow arcs, arcs drawn near the edge of a reduced working area, or arcs which were large in proportion to the extents of the working area, were sometimes being drawn on the screen 'approximated' as a series of straight lines.

Edit Track

- The values shown on the Head-Up Display were incorrectly resetting to zero on changing layer while adding a track.

Gerber Output

- One specific design with a large amount of copper was producing copper shapes in the Gerber file which covered areas that were supposed to be left clear.

Integrity Check

- The setting 'Design uses Variants' in the PCB was not being updated to match that setting in the Schematic.

PDF Output

- Popup information for signal references was only showing the component details.
- Some drill holes were being filled in by tracks.

- Attempting to produce a PDF output from a PCB contained in a Project would cause the application to quit.
- Mirrored TrueType text was being printed slightly undersized.

Pour Copper

- One specific design where the board outline was surrounded by an unfilled copper shape produced incorrect results when pouring copper within the board.

Projects

- It was not possible to move a gate on a multi-gate component from one Schematic sheet to another.

Reports

- Attempting to rename a user report would cause the application to quit.
- After creating a new user report, it would not appear in the list of available reports.
- Running Net Completion Report would cause the application to quit if it encountered a pad outside the board.

Setup

- On first-time installation, the path to the user libraries folder was being added to the configuration as "UserLibrary" when the folder is actually called "UserLib".