

Version 17 Software Update Details

Problem Fixes in Version 17.0.7 (23-July-2014)

3D View

- Prism shapes in user-defined packages were not being inverted for a component that had been placed on the back of the board.

Change Style

- Changing the track style of a zero-length track between a via and a pad at the same location could cause the application to quit as the track is tidied out after making the style change.

Component Edit

- Changing the symbol of a gate on the Gates tab to a symbol with a different number of pins could cause a corrupt component to be generated.

Design Rule Check

- Copper Shape Verification check on a particular design would not complete, causing the application to run out of memory.
- Component-to-Component checking was being done using the 'edges' of the component bounds, rather than treating the bounds as a 'solid' box, thus leading to fewer errors being reported than should have been the case.
- Inconsistent results were being reported when checking components in a particular multi-channel design laid out using Apply Layout Pattern.

DXF Import

- Certain commands were not being interpreted correctly in DXF files in recent formats, potentially leading to nothing appearing in the design after import.

DXF Output

- Long layer names could cause the DXF file to be unreadable in other software such as AutoCAD and TrueView, despite the published length limit of layer names being 255 characters.
- Text items could be misplaced from their proper locations.

Easy-Router

- Problems with circular keepout areas would sometimes cause the router to leave connections drawn on the screen when in fact those connections had been routed.

Gerber Import

- The holes defined by drill files were not always being matched to the corresponding pads during Intelligent Gerber Import, resulting in pads without holes.
- If the folder where the import process tried to write its report file was not writeable, the application would quit.

Goto

- The Goto bar would sometimes retain information (such as Component names) from a design that had been closed or reloaded, which would cause the application to quit if you clicked on any of those items.

Library Manager

- The presence of a comma in the name of a 3D package could cause the application to quit as it encountered that name when reading the 3D library.

Plotting

- Plotting positive 'plane and items' power planes for a panel design could cause the application to quit.

Pour Copper

- Incorrect copper shapes were generated on two specific PCB designs.

Security Copy

- Security copies were being saved even if the checkbox controlling this was unchecked in Preferences.

Update Component

- Replacing single-gate components which originated from early versions of the software was not protected against those components having incorrectly numbered gates, thus causing the application to quit as it attempted to access a gate that did not exist.

Problem Fixes in Version 17.0.6 (23-May-2014)

Forward Design Changes

- In designs containing some types of PCB-only component, running Forward Design Changes or Integrity Check could cause the application to quit as it attempted to generate the report.

Startup

- On a tiny number of machines, usually running Windows 7, the application would 'block' on startup after a Windows API function call failed to return. The Epcwin.exe process would be visible in Task Manager but nothing would be displayed on the screen.

Update Component

- In some circumstances when more than one package of the same component is used in the same design, updating one of those components would result in an internal fault in the design data, potentially leading to the design file becoming unreadable.

Problem Fixes in Version 17.0.5 (8-May-2014)

3D View

- When package type is set to DILSwitch, if it happened to match with a PCB symbol containing only a single pin, the application would quit while attempting to generate the 3D impression.
- When creating a new 3D package, sometimes the details would not be saved the first time round, requiring the details to be entered a second time before the new package would be saved to the library.

Apply Layout Pattern

- When applying a pattern between Groups where the 'from' group contained components having copper within their footprints, the application could quit while applying the pattern.

Back Annotation

- Processing back annotation data was causing the application to lock up for one particular design.

Design Rule Check

- Attempting to check using the option to check 'Selection' was not in fact checking anything. Full design-level checking was unaffected.

Drill Tables

- When the drill ID was more than just a single letter, the table could become misaligned so that the drill symbols no longer lined up properly.

Duplicate

- A net with a name of "N" was erroneously recognised as a default (unnamed) net, causing the joining of nets during duplicate to leave the net as unmerged with the original.

Edit Track

- Using backspace to delete previous segments was not changing the current active layer when backspacing through a Via.

Integrity Check

- A malformed warning message was being output to the report when ignoring PCB-only component pads.

Plotting

- The preview on the Plotting dialog did not include Vias.

Projects

- If the procedure to add a PCB design to a Project was cancelled partway through (for example at a prompt to save an older format design file), the file could be only partially linked to the project and would then cause the application to quit.

Properties

- The arrowheads on Dimensions could become unfilled after altering text settings in Properties.

Rename

- Attempting to rename nets across a Project was locking up if it encountered a name change that differed only in the case (e.g. renaming mux25 to MUX25).

Reports

- The Layers report contained error messages in place of the expected information about Layer Types.

Startup

- On a very small number of machines, Windows would refuse to let the application appear on the screen. An entry in the Task Manager for the 'epcwin.exe' process would be created, but would then be blocked due to a failure in a particular Windows API function.

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".

Copyright © 1997-2013 WestDev Ltd.

Number One Systems and Easy-PC are trademarks of WestDev Ltd.

All trademarks acknowledged to their rightful owners. E&OE.