

Version 15 Software Update Details

Problem Fixes in Version 15.0.8 : 17-Oct-2012

3D View

- Using a CHIP or DIL package for a PCB symbol with only one pin would cause the application to quit.

Component Edit

- Deleting and re-adding the same Value while editing a component would cause the application to quit.

Design

- Some PCB nets created by Translate to PCB or Forward Design Changes were being incorrectly initialised to use 'own colour' and hidden connections.

Design Rule Check

- Some pads placed inside copper cutouts were being incorrectly flagged as errors.
- Some 'net match' spacings with smaller values than the default spacings were not being used.

Drawing

- When moving or adding free pads, other free pads in the design could disappear from the screen.

IDF Export

- The component height value was always being taken from the Component definition even if over-ridden with a different value on the component instance in the design.

Import Eagle

- Some PCB designs were being imported with vias created where they did not exist in the original Eagle file.

Import Netlist

- Some OrCAD netlists were being imported without creating appropriate Net Class information, potentially leading to a program crash when later trying to access Nets.

Library Tools

- Attempting to import an Easy-PC Library Transfer (ELT) file that contains library items having forward slashes in their name would cause the application to quit.

N.C. Drill

- The lack of an explicit 'drill mode' (G81) command at the start of the file was causing some CAM viewers and front-end software to reject the file as invalid.

ProRouter

- Restoring track information after routing a single connection could cause existing curved corners to be distorted or lost.

TraceRouter

- One particular design failed to route without indicating any reason for the failure.
-

Problem Fixes in Version 15.0.7 : 30-May-2012

3D View

- PCB symbols with the only closed shapes on the silkscreen layer being very small could end up with no component body being generated on the 3D view.

Change Net

- Attempting to change net on one sheet in a multi-sheet project could result in a 'Failed to open document' error message being displayed.

Delete

- Deleting a track to a PCB pin which was multiply-mapped in the Component definition could remove that pin from the net even though the other mapped pins were still connected.

Dimensions

- The dimension lines were not being correctly adjusted when the dimension text was manually moved from its default position.

DXF Import

- If an imported DXF file contained an instance of a block when the block definition was missing from the DXF file, the import process would loop indefinitely trying to find the missing block.

DXF Output

- Very short bullet pads (with width > length) were producing DXF data that would then not read into AutoCAD.

Flip

- Flipping components in a Schematic was not always positioning attached text/names correctly.

Gerber

- One particular copper shape caused a line to be drawn across the board when the options for hardware fill and hardware arcs were turned off.

Integrity Check

- The 'Suppress from SCM/PCB' switch on a component was being ignored and the component was being added to the PCB anyway.
- One particular pair of designs would cause an incorrect set of changes to be made that were reversed the next time the function was run.

PDF Output

- When processing a schematic with blocks that had been flipped, the incorrect bounding box was calculated and therefore some items were missing from the output.

Technology

- The ability to right-click in the Spacings grid on the Design Technology dialog and Apply to Row, Column or All was missing since the initial release of V15 but has now been restored.
- Incorrect handling of re-ordered layers meant that in some cases the chosen layer order was discarded on exit from the dialog.

Problem Fixes in Version 15.0.6 : 27-Feb-2012

Delete

- The option 'Remove Isolated Pads From Net' was mistakenly being applied to Schematic designs as well as PCB.

Design Rule Check

- DRC was intermittently failing to apply net Guard Spacing when checking items.
- Surface-mount pads using pad styles with non-zero drill holes were not being flagged as errors where the resulting drill hole violated the spacing rules against items on other layers.

Display

- The Nets tab on the Display dialog could sometimes show Net Classes instead of Nets.

Electrical Rules Check

- The check for Schematic-only components was not ignoring components with no pins such as sheet outlines.

Grids

- Choosing the 1/40th grid snap using the popup Grid snap menu was having no effect.

Optimise Nets

- Optimising nets in a design containing component copper assigned to the net of a component pin would cause the application to quit. This problem could show up at any point when Optimise Nets is run, including after Integrity Check, Forward Design Changes, and Translate to PCB.

Save To Library

- Attempting to use 'Save To Library' either from the Library manager dialog or directly from a design file could cause corrupt library data to be saved if the source library item or design was not in the current design format.

Problem Fixes in Version 15.0.5 : 07-Feb-2012

3D View

- The elements of some MELF packages were being drawn at the wrong orientation depending on the relative position of the two pads on the footprint.
- Some MELF packages were using the wrong calculation for the overall size of the component.
- If the pads in a PCB symbol were not numbered in the order in which they were added, this could cause the 3D image for the corresponding component to be rotated by 90 degrees.

Apply Layout Pattern

- In some circumstances it was possible to cause the application to quit when applying a pattern from one group to another.

AutoRouter

- The presence of copper terminals (tracks attached to copper shapes) could cause the application to quit when that net was re-routed.

Component Values

- New columns added to the Component Values dialog using the Add button would display the text of the value itself as the column heading instead of the value name.

Cross Probe

- With "Bring Probed Design to Front" off and "Probe Corresponding Design" on, sometimes the window for the design in which items were being picked was covered over by the target design window.

Design Calculators

- It was not possible to type the Precision on the Scientific calculator.

Design Rule Check

- Spurious Drill-to-Drill errors were being reported for coincident vias that used layer spans that did not overlap.
- Online DRC was not enabled in Panel designs.

DXF

- The positioning of component names could be wrong when outputting designs to DXF, causing them to be placed a long way outside the board.

Find Library Item

- The Find dialog was not able to find any components when attempting to match by user-defined component values. This was caused by a problem with the generation of library index files. If you discover that your Find option is not seeing your component values, delete all the ".CMX" files from your library folders and they will be re-generated by the program with the correct information.

Flip

- A component near the edge of the working area could not be flipped. It should now flip over as requested, and then shift back to a valid position within the working area.

Generate Component

- Existing footprints with pads numbered non-sequentially could prevent the correct assignment of the footprint to a set of pads.

Groups

- It was not possible to assign an ungrouped component to an existing group.

Import Eagle

- Data could not be transferred into Easy-PC from Eagle Version 6.
- Pin numbers could be invisible after being placed on a specific layer during import.
- Track ends were not always being connected to their pad if the track end was not exactly on the pad position.
- Importing a multi-sheet schematic did not set up net classes on the nets.
- Component names were not being correctly positioned.
- Component "Value" was not being displayed in an imported Schematic.

Intelligent Gerber Import

- Attempting to import a drill file from Allegro could cause the application to quit.
- Some surface mount pads were being incorrectly removed from the imported result.
- Some pads were being assigned invalid layers which then prevented them from being converted to vias.
- Very small arcs were being converted into straight lines, leading to loss of 'serpentine' patterns.

Layers

- Toggling the state of some 'visibility' checkboxes on the Layers bar could result in some categories of item being displayed when they were supposed to be hidden.

Library Manager

- Attempting to import some ELT (composite library data files) could fail partway through when attempting to create an invalid temporary file.

NC Drill

- Some arcs for milling board outline shapes were being output as minor instead of major arcs.

Online DRC

- Moving an item inside the oversized clearance due to a net guard spacing was not being flagged as an error.

Optimise Nets

- Connected component copper within footprints was not being included in the optimisation.

PDF Output

- Some text was not being plotted in the correct position.

Plotting & Printing

- The Drill Symbol dialog could not be closed if more than one entry existed in the drill list that showed the same rounded-off value for drill diameter.
- Pads with no width but a non-zero drill were not being output.
- When plotting a Panel design, a separate error message was being displayed for each missing PCB design.

- An incomplete filled shape was being generated for poured copper on one particular design.

Pour Copper

- Incorrect results when pouring in two specific PCB design files.

Print

- The board outline was not being included on the printout.

Properties

- Changing the style of a component pad using the 'Pad Style Exception' checkbox had no effect if the only difference between the two styles was in 'by layer' exceptions set up on one or other pad style.

Reports

- It was possible to create a component report that would cause the application to hang.
- The IPC-D-356 output had the component reference name in the wrong position on the line.
- The short-form IPC-D-356 output was not applying the coordinate system origin to XY positions in the file.

Sketch Track

- When using Sketch Connection in a Schematic design, attempting to drop the connection to finish in empty space would cause the application to quit.

Spacings

- The value of the drill spacing on the Rules page of the Technology dialog could not be changed.

Symbol Wizard

- The units used in the Wizard were always stuck at mm.

TraceRouter

- The router would not allow or handle powerplanes on outer layers.
- In some circumstances the router would place a track across copper that belonged to a different net.
- The presence of one or more net classes without assigned via styles would prevent the router from running.

Update Component

- Rotated value positions on components could end up in the wrong position after the component was updated.

Problem Fixes in Version 15.0.4 : 07-Oct-2011

Back Annotation

- The report produced during Back Annotation did not list any Group renames.

Design Calculators

- Errors in the formulae used for Asymmetric and Differential Stripline calculations.

Forward Design Changes / Integrity Check

- Forwarding design changes from Schematic could result in the wrong changes being made in the PCB.

Group

- The application would allow more than one group to have the same name.

Reverse View

- Reversing the view for a PCB design would reverse the view for all open designs including Schematics.

Type Coordinate

- Coordinate changes made on a Group using Type Coordinate or Type Offset were taking a long time to apply.

Problem Fixes in Version 15.0.3 : 20-Sep-2011

3D View

- The application would quit attempting to preview a Dipswitch 3D package for a PCB symbol which had no pads.

Add Net

- Adding a new net to a pin on schematic sheet in a Project could cause the application to quit.

Auto Correct Track

- Portions of the original track could be left behind as the corrected track is inserted.

Edit Mitre

- When starting to mitre a corner using double-click the edited segments were drawn incorrectly making it difficult to see what you were editing.

Integrity Check

- After renaming components in the schematic, applying those changes to the PCB could cause incorrect changes to be made to components.
- In some circumstances, the presence of very similar components in the Schematic could cause Integrity Check to repeatedly swap components in and out of the PCB.

Plotting

- The Drill Ident setup dialog would mistakenly report duplicate entries when the rounding that occurs due to unit precision resulted in more than one row in the table showing the same hole size.

Pour Copper

- Pouring was using the pad-to-copper spacing rather than the thermal isolation gap specified in the design rules for pads/vias on the same net as the copper.

Properties

- Changes to the 'Show Connections' checkbox on the Net properties tab were not being saved.

Save To Library

- Symbols saved to the library using the Save To Library dialog could be incomplete.

Sketch Track

- The track pattern produced by Sketch Track was sometimes incorrect, with many small segments twisted over each other.

Symbol Edit

- Editing a PCB symbol would not display the pads if no technology file was used.

Technology

- Changes to the 'Show Connections' setting on the Nets page of the Design Technology dialog were not being saved.

TraceRouter

- The TraceRouter was not enabled unless the 'Easy-Router' optional feature was installed, whereas it should also be available if ProRouter was installed.
- Some designs with large Board spacings would not route.
- Attempting to route a design containing a Net Class that did not have a Via Style assigned to it would cause the application to quit.

Problem Fixes in Version 15.0.2 : 02-Aug-2011

Align

- Aligning segments of shapes would sometimes move the segments too far.
- Track segments were not being aligned at all.

Drawing

- When editing connections into tracks, inactive layers were not being lowlit. Lowlighting would only occur on changing layers.

Move Segment

- Moving a segment of a rectangular shape such as a Board Outline was no longer keeping the adjoining segments orthogonal.

N.C.Drill Output

- Shapes from electrical layers were being output to Excellon drill files as milling/routing commands.

ODB++ Output

- The presence of a netname with >64 characters was preventing the ODB++ file from being readable by other tools.

Open

- If a schematic design contained multi-gate components where some but not all of the gates in a particular component were in a Group, the application could hang on attempting to open the design file.

Problem Fixes in Version 15.0.1 : 26-Jul-2011

Add Trunk Route

- It was not possible to change the Track Style used for the tracks inserted by Add Trunk Route.
- Automatic routing of the start/end portions of trunk routes could result in track segments being added on the incorrect layer for the pad to which they were joined.

3D View

- The list of available package types included a 'blank' entry which if used could cause unpredictable results in the 3D drawing.

Open

- o Some design files created in earlier versions of the application could not be opened in V15.

Copyright © 1997-2012 WestDev Ltd.

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

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