Easy-PC

Number One Systems

Version 19
Software Update Details

Problem Fixes in Version 19.0.6 (15-Aug-2016)

Pour Copper

- Copper pour was often producing copper to copper errors due to a change in the previous patch.
- Was adding a spurious cut across the poured copper.

Add Text

Problem Fixes in Version 19.0.5 (05-Aug-2016)

3D View

- After adding items to a 3D package library, the buttons on the Library Manager dialog were not updated to reflect the changes.
- On the Library Manager dialog, the description showing the list of aliases for a 3D package was being truncated rather than wrapping round to another line, making it difficult to see all the aliases for the package when browsing through the list.

Add Text

- It was not possible to add text to a layer that used the 'special effect' layer usage.

Apply Layout Pattern

- In one particular design, one component in the pattern was being placed a long way from its proper position.

Component Bin

- In a multi-sheet project, if a schematic sheet had lost its reference back to the containing project, then moving a spare gate to the component bin in that sheet would cause the application to quit.

Component Edit

- After adding a Value to a component, if that Value was then deleted before saving or closing the component, the application would quit.

DXF Import

- Importing a DXF file containing circles that had no width defined could result in some shapes being over-sized.

Design

- If a Bus name had somehow been placed outside the working area there was no way to retrieve it.
- Bus names were not being included in the calculation of the design 'extents', so View All may not always show the entire design.

Duplicate

- Duplicating Net Name items that used one of the 'Box' shapes could cause the application to quit when the design was saved.

Fonts

- The output resolution for TrueType fonts was too low to accurately represent the shape of some text characters.

IDF Output

- The coordinates in the output file were not using the System Coordinate Origin.

ODB++

- Pads using the 'offset rectangle' pad shape were not always being output at the correct orientation.

PDF Output

- When outputting multiple plots from a PCB design to separate PDF files, the names assigned to those plot files were sometimes wrong, and thus did not reflect the actual contents of the file (silkscreen containing copper, copper containing paste mask, etc.).

Pour Copper

- In one particular design using a Pour Area around two separate board outlines, no copper was generated.

ProRouter

- In some circumstances, the router could place tracks across a board cutout, or across copper shapes attached to component pads.

Projects

- Attempting to add a sheet to a project where that sheet was already open in another instance of the application would cause the application containing that project to quit.

Rename Components

- Automatic component rename was not always following the settings of the Rename dialog, making it difficult to rename separately on top and bottom of the board.

Reports

- The library report showing the Components that use the selected PCB Symbols was not using a case insensitive search for the matching symbols and could thus produce an incomplete report.
- Some fields from the standard report header (author, keywords, comments and subject) were not being included in reports even after they were enabled in the Preferences dialog.

Spacings

- When setting up Match Net Spacings, the square brackets [ ] around net class names were being lost as soon as they were entered, thus making the entries refers to those names as nets rather than net classes.

TraceRouter

- In some circumstances, the router could place tracks across a board cutout, or across copper shapes attached to component pads.

Problem Fixes in Version 19.0.4 (22-Mar-2016)

3d View

- One specific design would cause the application to quit when the 3D View was opened.

Automatic Fan-out

- Attempting to create a fan-out on a pad with no net caused the application to quit.

Auto-Place

- It was not possible to make components flip to the back of the board.

Change Net

- Attempting to change the net of a schematic connection which is not attached to any terminals or junctions would cause the application to quit.

Design Rule Check

- If the option to treat board outlines as 'centre line' was enabled, DRC was still using the line style width for its calculations.
- In DRC for a Schematic with the option for 'If no visible net name' enabled, pins assigned to a net but with no connection on them would still be marked as an error.

Dimensions

- The precision for the dimension value was always being initialised as 2 when creating the dimension regardless of the Defaults dialog setting.

Gerber

- Solder paste plots for pads using the Annulus pad shape would have their inner hole filled in.

Layers

- The 'Areas' checkbox on the Layers dockable bar was only affecting routing areas, ignoring copper pour and height check areas.

Paste

- When copying and pasting sections of a PCB design to a new design, some of the pads could be connected to the wrong nets.

Pour Copper

- Some thermal spokes were not being added to some off-grid pads.
- The 'minimum pour area' on the Pour dialog would reset itself to zero or a very small value.

Power Planes

- Some power planes could be generated with the wrong thermal relief spoke width.

Select

- Using Alt+Click to view the pick list in a Schematic with DRC errors would cause the application to quit.

Problem Fixes in Version 19.0.3 (12-Nov-2015)

Component Edit
- Changes to property checkbox settings were not always being saved with the component.
- Editing several components in the same application 'session' could cause the application to become unstable.

Forward Design Changes
- The Goto bar was not being updated with changes to the PCB design caused by Forward Design Changes.

Gerber Output
- The values output to Gerber files to identify the layer number when using X2 format were incorrectly starting from zero instead of 1.
- Solder/paste over/under-sizing was not being generated correctly for T-shaped pads.
- Using a dashed line style was creating spurious extra lines on the plot.

IDF Output
- Angles were only being output to the nearest whole degree, thus causing loss of accuracy in positioning items.

Library Manager
- If no library folders are defined, attempting to create a new library could cause the application to crash.

PDF Output
- Spurious extra arrow-heads and lines are plotted for radial dimensions. Also affects other output device types.
- Inaccurate vertical placement of some TrueType text items.
- Some bitmaps at particular combinations of rotation and mirror were not being correctly positioned on the output.

Pour Copper
- Calculation of optimal positioning of thermal spokes was sometimes failing to place some spokes where they would overlap with existing spokes.

Properties
- When changing the properties of a Panel PCB Item to alter it from absolute to relative path was not re-mapping the layers.

Rename
- Using point-and-click Rename in the PCB could create a set of renames that causes duplicate component names in the Schematic when back-annotated.

Reports
- One particular design was causing the application to hang when trying to generate a BOM.

Support
- The support report was not showing the correct Windows version when running on Windows 10.

Problem Fixes in Version 19.0.2 (16-Sep-2015)

BOM Composer
- Components which were marked as unfitted in a variant were still being included in the output.

Component Values
- A CSV file created by exporting from the Component Values dialog would fail to read back in.

Gerber Output
- Some filled copper shapes were being generated in the output file without being filled.

Import Eagle
- Designs containing components with names longer than 31 characters could cause the application to quit during the import process.

Import Netlist
- When importing an EDIF netlist containing BGA data, the import was failing with an error message about failing to recognise device pin numbers as it was not correctly handling the alpha-numeric pin numbers.

ODB++ Output
- In some cases, nesting several board outlines inside each other on a power plane could cause invalid output data.

PDF Output
- Some rotated and/or mirrored bitmaps were not being output in the correct position in the PDF document.

Plotting
- The presence of certain characters including ">" in layer span names could cause the application to quit while generating Drill files.

Pour Copper
- Several design files were failing to pour.

Pro Router

- One specific design would fail to start routing, simply returning without doing anything.

Properties

- It was possible to set a Pour Area as filled, potentially leading to the mistaken assumption that this created copper on the final board.

Save Project Version

- Applying new version numbers in some combinations of project files could cause the wrong changes to be applied to the PCB when Integrity Check was next run.

Problem Fixes in Version 19.0.1 (03-Aug-2015)

Component Editing

- When editing a component, the drop-down list of library names on the Packages tab was empty.

Gerber Output

- The default setting for "X2 in G04 comments" should be "True" so that all Gerber data being generated can contain X2 commands. Placing X2 in comment fields allows compatible readers to detect the X2 commands without causing issues in non-compliant readers.

Import CSV

- Importing component placement CSV files was not using component positions as being relative to the design coordinate origin.

Pour Copper

- Several design files were failing to pour due to the same internal issue with the internal geometry package.

Tool Bar

- Two incorrect icons were being displayed on the standard Edit toolbar.