

Version 23 Software Update Details

Problem Fixes in Version 23.0.6 (08-Sep-2020)

3D View

- Some DIL-based packages were being drawn at right angles to their proper position.

Component Edit

- Symbol previews were not been updated after editing and saving the symbol if that symbol originally came from a different library file than the currently available symbol.

Component Search Engine

- Integration with Component Search Engine (Samacsys Library Loader) was no longer possible after the release of Version 24.

Connectivity Check

- The Connectivity Check could report that a ground plane was split into multiple regions when in fact it was complete.

Design Rule Check

- When using Net Match spacings there was no way to instruct the application whether to look for the smallest matching spacing or stop on finding the first match.

Display Net Name

- Depending on what was selected, it was not always possible to show net names on all relevant pins in one go.

Import Easy-PC DOS

- Some components could be imported 'flipped' to the wrong side of the board.

Plotting

- When plotting a hatched symbol shape with 'Break Silkscreen' enabled on the Layer Type, the shape could instead be plotted as filled.

Projects

- Adding an existing Schematic design to a Project was not clearing any Block name already defined on that sheet.

Problem Fixes in Version 23.0.5 (18-May-2020)

3D View

- Creating a 'user' 3D package from a built-in package on a PCB symbol with a large number of lines/shapes could result in a 3D package library that was no longer accessible by the application.

BOM Composer

- It was not always possible to get design- or component-level Associated Parts to be included in the output.

Plotting

- In a Schematic design, it was not possible to access the Settings tab of the Plotting and Printing dialog without the application closing.

Pour Copper

- If a copper shape in a PCB symbol was mistakenly assigned to a pad on a different layer to the copper (and thus could not be associated together), this shape might not be detected during Pour, resulting in copper being poured across that shape.

Styles

- When editing a Schematic or PCB symbol, the text styles for [Pin Names] and [Pin Numbers] were not being marked as used, and could therefore be deleted.

Changes in Version 23.0.4 (29-April-2020)

Design Rule Check

- Net Match spacings now has a checkbox to specify whether DRC should scan all matching spacings for the largest value, or whether it should stop on the first spacing rule that matches.

Problem Fixes in Version 23.0.4 (29-April-2020)

Add Component

- Stretching the Add Component dialog was not increasing the amount of space for the list of component Values.

Add Track

- The default action when clicking to insert a corner over an existing track segment on another layer was to ask to insert a via and join to that track, instead of offering this as the default only if both tracks were on the same net.

Colour Files

- Some elements of the colour settings available on the dialog were not being saved/restored using Colour Files.
- The Colour Files dialog would sometimes not list all or any of the available files.

Component Edit

- After saving edits on a symbol whilst having a component open for edit, sometimes the symbol preview would not update to show the newly-saved symbol until you closed and re-opened the component.

Component Values

- It was not possible to Import a CSV file that had just been Exported from the Components page of the Libraries dialog.

Delete

- When deleting a Pour area, the choices offered on the popup question about removing the associated copper did not match the actions that the application then took in response to your choice.

Design Technology

- Grids were not visible on Design Technology and various other dialogs on some high-resolution displays.

Layers Bar

- The pickable checkbox for Text was not properly saving and restoring its setting.

Library Dialog

- Where folder names are 'squeezed' by replacing some folders with '...', it was not possible to see the full names even by resizing the library dialog.
- When Copying items to another library, the application would prompt that the item already existed in the library you are copying 'from', not 'to'.

Menus

- The right-click menu when selecting a component had the command 'Display Net Names' on the menu twice.

ODB++

- In some situations, blind or buried vias were being output as through-board.
- In one particular PCB design, some copper was missing the cutouts needed to provide isolation from a plane.

PDF Output

- Where filled shapes were used in symbols, text items would sometimes be obscured by the shape filling when printed to PDF.

Plotting and Printing

- Sometimes the Plotting dialog would get 'stuck' asking you to save a Plot Job and you were unable to cancel.
- Redundant dialog controls for thermal isolation and spokes were still present on the Settings tab even though the values entered there were no longer used.
- Using 'Time' as a field for composing plot file names was causing all plots in a given 'run' to have different times, rather than all using the time of the first one generated in that run.

Pour Copper

- Copper could be poured too close to component copper (copper shapes defined in a PCB symbol) if the shape-to-shape spacing was larger than other spacings.

Preferences

- The default setting for 'Ignore Tracks fixed for Router' was 'checked', which meant that it could be difficult to unrout some tracks.

Properties

- Changing the layer span of a Via in the Properties dialog had no effect.

Reports

- In the custom report, it was not possible to choose to include only those components which had a particular Value defined using "Value <valuenam> Is Not" and entering a blank string.

Select Mask

- It was not possible to select Junction Points in a Schematic if the Select Mask for Components was turned off.

Teardrops

- The wrong shape was being generated for teardrops on T-shaped pads.

Problem Fixes in Version 23.0.3 (15-Jan-2020)

Design Rule Check

- When using multiple Net Match spacings with different spacings on different layers, checking of through-hole pads may not always find the worst (largest) spacing that applies to the pad.

Save New Version

- In some circumstances, saving a new version of a project to another folder could cause the original files to be moved rather than copied.

Problem Fixes in Version 23.0.2 (25-Nov-2019)

Apply Vias

- No account was being taken of the drill-to-drill spacing when calculating valid positions for adding vias, which could lead to some being later flagged with errors when running Design Rule Check.

Auto Fanout

- The dialog allowed you to type anything into the drop-down list for inside/outside/both, rather than restricting the input to only allow valid choices.
- The direction choices for inside/outside were producing results the exact opposite of what was chosen.
- If you selected a pad that had no unrouted connections, the menu command for Fanout Selected Pads was not enabled.
- In designs containing copper pour areas it was sometimes possible to generate spurious additional stubs that attempted to join to the copper shapes.

BOM Composer

- The sort order in the CSV file generated by BOM Composer was not the same as that shown in the on-screen display or the generated HTML/text report.

Configuration Files

- On the dialog for the separate Configuration Files program, the Help button gave an error 'Topic not found'.

Design Rule Check

- Spacing errors were not being detected for pads on the special layer [Bottom Layer] when Net Match Spacings were set up for the explicit bottom copper layer name.

Design Tool Tips

- Depending on the Zoom level, the on-screen design tool tip could show information for the wrong item.

Display

- In several places in the application the display of user interface elements (including some dialogs, and list boxes containing check boxes and/or icons) was incorrect or unreadable on high-resolution displays.

Drawing

- At some zoom levels and grid steps, drawing of screen grid dots could be very slow causing application to become unresponsive.

Gerber Output

- When generating Gerber data using the "X2" format, the aperture function 'SMDPad' in the file header was missing a mandatory second field ('CuDef' or 'SMDDef').

Project Files

- When saving a copy of a Project, an error message was being displayed that incorrectly indicated that the copied Project would not work properly.

Rename Component

- Renaming selected components in a PCB design could result in some of those components being given names that already exist on non-selected components.
- In some situations, components on some Schematic sheets in a Project were being overlooked.

- The setting of the 'By Stem' checkbox when using Auto Rename Components in a Schematic was not being remembered for next time the dialog was used.

Problem Fixes in Version 23.0.1 (01-Oct-2019)

Calculators

- In some situations, accessing or closing the Design Calculator dialog could cause the application to quit.

Design Rule Check

- DRC in a Schematic design or Project gave no indication of the checking results or the number of errors found unless you chose to view the detailed report.

Drawing

- Newly added Copper Pour Areas were always being drawn in a pink highlight colour until the design was saved and re-opened.
- Components added to a PCB design would sometimes display Values on the drawing even though they were not specified as visible in the Component library item.

Fix and Unfix

- The Fix/Unfix dialog had no method of unfixing tracks marked as 'fixed for router'.

IDF Export

- It was not possible to select the required Values for Mounting Hole Offset when trying to generate IDF 3 data.

Layer Spans

- Simply changing the name of a Span in the table in the Design Technology dialog did not enable the Apply button.

N. C. Drill Output

- When outputting coordinates with an explicit decimal point, a leading zero in the fractional part of the coordinate could be omitted.

Panel Wizard

- The preview picture showing you the panel you are creating needed to suppress the display of Notes from the PCB design as these could obscure the rest of the information in the preview.

Replicate to Layers

- The drop-down list for layers on the Replicate to Layers dialog was offering the special layers (e.g. [Top]) when they were not valid choices.

Split Sheet

- Using the Split Sheet command to move a portion of a Schematic drawing within a Project to a new design would result in the new sheet being added to the Project twice.