

Version 23

Software Update Details

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.