

Version 21 Software Update Details

Problem Fixes in Version 21.0.5 (25-Apr-2018)

3D View

- Single-sided boards were being displayed in 3D view without a substrate.

Integrity Check

- Problems with duplicate components in a Schematic sheet were not being reported by Integrity Check if no other differences between Schematic and PCB were found.

Open

- Opening some Schematic designs belonging to Projects was causing the application to quit if the option to open all Schematic sheets at once was not set.

Plotting

- Paste shapes for some footprint shapes set as "auto mask" were omitted from plots.

Pour Copper

- For some square pads, angled (45 degree) spokes were being generated pointing in the wrong direction.
- When nesting Pour Areas of same net and layer inside each other, the outer area could pour across the inner one and thus ignore any over-ride of thermal rules on that inner area. The outer area pour will now stop at the edge of the inner area, allowing each one to apply its own rules.

Properties

- Attempting to change the name of a Bus by selecting its existing name and accessing Properties would cause the application to quit.

Trace Router & Advanced Router

- Auto routers could place tracks over pads defined using offset pad shapes (offset rectangle or bullet).
-

Problem Fixes in Version 21.0.4 (03-Apr-2018)

Board Maker

- Import of PCB designs from Board Maker was sometimes creating tracks where there should be copper shapes.

Net Sheets

- Display of Net Sheets in a Schematic was listing empty nets.

Plotting

- In some designs using the setting for 'break silkscreen' in a layer type, the application would quit when attempting to plot the silk screen.

Pour Copper

- Nesting one copper pour area inside another (on the same net and layer) was not being detected, causing the outer pour area to flood over the inner one even if the rules defined on the inner area were different.

Pull-tight Routing

- Attempting to use Pull Tight Routing would sometimes fail to alter any tracks even when there was space to pull them tight.
-

Problem Fixes in Version 21.0.3 (23-Jan-2018)

Back Annotation

- When back-annotating a net name change from PCB to Schematic, the net was not always being updated on all sheets in a multi-sheet Project if the net was used in a bus on multiple sheets.

Calculators

- Scientific calculator performed the wrong calculation when a sequence of operators was entered.

Component Rename

- Auto Rename of components was not obeying the setting 'Reverse Left/Right on bottom side'.

Eagle Import

- Some paste shapes could be missing when reading XML (Eagle 6+) format.
- Rotated rounded rectangle pads were getting the wrong corner radius.
- Multi-line text could be rendered as a single line.
- Bottom documentation layer was imported on the Top side.
- Component keepout shapes could be created as fragmented shapes.

Gerber Output

- When using custom file extensions, the wrong extension was being assigned to auto-generated pads-only plots.

IDF Output

- Spurious data was sometimes being written to the output file for designs with components in the Bin.

ODB++ Output

- In one specific design, a missing copper cutout caused the copper to over-run other items.

Pour Copper

- In some circumstances, Pour would leave out thermal spokes which could legally be included.

Preferences

- Additional settings now available for controlling the operation of Unroute.

Text Styles

- The drop-down list used to show available fonts did not include 'Open Type' fonts.

Problem Fixes in Version 21.0.2 (11-Sep-2017)**Dynamic Align**

- It was not always possible to place items close to other items or properly obey the grid snap settings, in some cases causing an item being added (such as a pad) to snap back on top of the previously added item.

Pour Copper

- The positioning of thermal spokes was not always following the rules defined in the Design Technology. In particular, the application would insert angled spokes even if the rules stated only Orthogonal should be used.

Properties

- Attempting to view the Properties of a free pad in a PCB design could cause the application to quit.

Pull Tight

- The 'Pull Tight' commands were not ignoring fixed tracks, resulting in those tracks being modified.

Problem Fixes in Version 21.0.1 (01-Sep-2017)**Add To Net**

- Attempting to add an existing net to a Shape using 'Add to Net' could cause the application to quit.

Auto Rename

- The 'Goto' bar was not being refreshed to reflect the changes made to Component Names.

Delete

- Accepted DRC Errors were not being treated as if they were fixed, so they were simply being deleted.

DRC Errors

- Changing an error to an accepted error using the context menu command was not redrawing it, so it was not applying the appropriate colour and visibility settings from the Display dialog.

IDF Output

- When choosing to use your own units from the IDF output dialog, those units were not being applied to the component heights.

Import Eagle

- The import of Eagle native (XML) format data was not correctly handling the situation where the same-named footprint was present in the file from different source libraries.

Import Pin Data

- When importing CSV pin data into a Component, any line which contained an empty field at the end (i.e. a line ending with the separator character) was being ignored.

Library Manager

- When previewing a Component with multiple packages, the PCB symbol library name shown was the name for the package selected for the graphical preview, and not the name of the source library for each individual PCB symbol.
- The report from the 3D Packages page did not include the 'Package to PCB Symbol' information if all three of the checkboxes were selected on the report options dialog.
- When saving library items (components and symbols) to libraries, no error message was being displayed if the attempt to save failed for any reason other than the whole library being read-only.
- Clicking the Rename button to rename library items simply did nothing if the library was read-only, rather than displaying an appropriate error message.

Merge Shapes

- Depending on the item being selected, the context (right-click) menu did not always include the Merge Shapes command.

N. C. Drill

- The Excellon drill section of the plot report did not properly format the drill sizes and could thus show misleading information, even though the drill file itself contained the correct data.

Nudge

- Using the cursor keys to move an item did not always move it if either 'Snap to Ruler Stops' or 'Dynamic Align' was enabled.

Plotting

- The Settings page for a power plane plot had the 'spinner' control for the number of spokes incorrectly attached to the checkbox for Pads, causing some odd-looking effects on the dialog.

Properties

- The Alignment drop-down list for the properties of a text item (including value position) was not allowing you to change the text to be left-aligned.

Setup

- The product installer was not allowing the install to proceed until something was entered into the Company Name.

Test Land Reports

- The short-form IPC-356 and 'human readable' test land report were not applying the 'Neutral Net Names' option.

Unroute

- The 'Ignore Fixed Items' checkbox under Unroute on the PCB Interaction page of the Preferences dialog was not working, which resulted in the Unroute commands being unable to unroute any sections of track which were marked as fixed for any reason.

Values

- There was no way to display just the name of the current Project as a value position, only the complete path.