

Version 19 Software Update Details

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.