Version 24
Software Update Details

Problem Fixes in Version 24.0.2 (15-Oct-2020)

3D View
o   Adding a blank "3DPackage" value to a Component to suppress any instance of that component from appearing on the 3D view had no effect, it was necessary to override this on the component instance itself to get it to work.

Add to Library
o   Adding symbols from a design to a library using the Add File button on the Library dialog was not preserving the original timestamps of those items from the design but was instead marking the items in the library with the current date.

BOM Composer
o   The CSV version of BOM files could contain extra unwanted numbers in value fields.
o   A value without a leading zero (for example ".2pF") was not being recognised as a valid number and hence was not included in the value-based sorting.

Design Rule Check
o   Checks for Acid Traps and Component Names were not adding sufficient information to the DRC report or the error marker descriptions.

Display (Colours)
o   The buttons on the right-hand side of the Display dialog were sometimes hidden or slightly misplaced as the dialog was opened or resized.

Gerber Output
o   Gerber files contained some constructs which did not conform to the newly issued strict syntax of the Gerber format. Whilst this did not cause any manufacturing issues, these deprecated (out of date) constructs have been corrected.

Library Tools
o   Trying to import a library data file from Ultra Librarian may display spurious messages about Eagle format before successfully importing the data.

Menus
o   If the only 'view' open in the application was the Start page, the Help menu did not include the command to save/load Configuration files.

Pad Styles
o   The ability to use right-click 'Apply To Column' was not available for the new Thermal Width column on the Pad Styles page of the Design Technology dialog.

Pour Copper
o   Little or no isolation gap was being created when pouring around hexagonal pads.
o   In some designs opened from earlier versions of the application, thermal spokes added when pouring could be extremely narrow.

Project
o   In a project with more than one instance of the same Block, pushing into one of those blocks and popping back out again could cause the arrangement of blocks in the Go To bar to be re-arranged to show the wrong relationship between blocks.

Properties
o   On the properties of a pad (terminal) in a Schematic design, the controls for setting thermal width and repositioning the pad were visible when they should have been hidden as they only apply to PCB.

Save to Library
o   Saving a new symbol to a library without first giving that symbol a name would add it to the library as an unnamed item which then could not be found again.

Problem Fixes in Version 24.0.1 (04-Sep-2020)

Add Connection

o When prompted for a new Net name on adding a connection or track, pressing OK without providing a net name would cause the application to quit.

BOM Composer

o When 'proper' electronic values are present in the Values column, spurious characters could be included in that column in the CSV version of the BOM.

Component Values

o Applying changes to the Filter on the Component Values dialog was not preserving the current width of the columns in the table.

Delete

o Depending on Preference settings, deleting a fixed component could cause the application to quit.

Design Rule Check

o Some DRC options could run noticeably slower than in the previous version of the application.

Flip

o When flipping a component to the opposite side of the board, the display was not updated to move any attached tracks to their new positions.

Generate Component

o The Generate Component dialog would let you create a Component without a Package name which is not allowed.

Import Library Data

o Attempting to import a .DSL file (DesignSpark format) from Ultra Librarian caused several spurious messages about 'Eagle import' before importing the data.

o Dropping a DSL or EPL file onto the PCB or Schematic symbol page of the Library dialog would import the Component definition instead of the symbol.

o Imported library data could have pin names or numbers being assigned 'user' text styles instead of the built-in styles [Pin Names] and [Pin Numbers].

Preferences

o The application was not remembering a change of setting for the checkbox 'Highlight Net being edited' in PCB Preferences.

Trace-Router

o On some systems, it was not possible to invoke the auto-router if the PCB design file was stored in a folder whose name contained non-ASCII characters.

View Power Plane

o Some items 'embedded' in a generated power plane were not being drawn correctly on the screen, with those items only being visible when selected.