Version 24
Software Update Details

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.