Version 24
Software Update Details

Problem Fixes in Version 24.0.4 (25-Mar-2021)

Component Values
o   When using the Filter controls to see components with a specific number of pins, the result was the opposite of what was intended as it would display only components which did *not* have that number of pins.

Cursors
o   With the cursor set to Small Cross, this would be replaced on the screen by the standard Windows arrow cursor in most situations.

Drawing
o   When using the commands for View | Net Connections, using Top or Bottom could result in the picture showing fewer connections than expected.

Hierarchy
o   When using Change Symbol to alter a generated block symbol, changing the number of pins had no effect on the resulting symbol.
o   Attempting to use Change Symbol would sometimes cause the application to become unresponsive.

Import CSV
o   When importing CSV pin data, leading spaces on the last field on the line could cause the data to be rejected as invalid.

Import Eagle
o   In some situations, the mapping of pin names was incorrect, leading to spurious results when running Integrity Check on the imported designs.

Import Easy-PC DOS
o   Positioning of mirrored components was incorrect for some designs.

Integrity Check
o   Values added to components in the Schematic were not always being recognised as missing or different when comparing with the PCB design.

Join Nets
o   On the Join Nets dialog, the button 'Other Net' did not swap the nets around to use the other net as expected.

ODB++
o   When generating the matrix of layer information, some layer names used for blind vias were incorrect, causing some third-party software to misread the drill layer data.

Plotting
o   When using the 'Pads Only' plot feature to auto-generate paste and resist outputs, if the option to define a Percentage oversize was chosen, the value for that oversize would alternate between positive and negative with each visit to the Plotting dialog.
o   In some situations, generally with designs created in older versions of the application, attempting to generate plots could cause a spurious message about the thermal isolation value being smaller than the copper to pad spacing rule.
o   When generating PDF or Windows output for a Schematic design, text and values were not always being drawn in front of filled symbol shapes, leading to some text being obscured.

Pour Copper
o   The gaps left between copper and tracks could be larger than expected for tracks that included width changes.

Variants
o   Copying a variant was not also copying the fitted state of components as it created the new variant.
o   Creating a new variant then altering the fitted state of components was not being remembered as separate steps for Undo, which could lead to Undo rewinding further than expected.

Problem Fixes in Version 24.0.3 (05-Jan-2021)

Apply Layout Pattern
o   Some of the dialog settings were not being remembered for next time the dialog was displayed.

Backup
o   If the 'work time' setting for backups was set to zero, no backup file was being created when the current design was saved.

BOM Composer
o   For columns that use more than one Value to populate the data, the application was not scanning through all possible Values to find a non-empty string.

Connectivity Check
o   Settings on the dialog were not being remembered for next time the dialog was displayed.

Design Rule Check
o   Unwanted pad-to-pad errors were being reported for pads that were linked as 'multiply mapped pins' in the component.

Display Net Name
o   For net names with the option to draw a 'box' around them enabled, hiding the net name was leaving the box visible on the design.

Generate Component
o   If an existing reference name was chosen, and the corresponding component was in the Component Bin, the selected free pads were not removed from the design when the existing component was placed.

Integrity Check
o   The message in the Integrity Check report for "Disconnect Pin XXX-1 from Net XXX" did not have a clickable hyperlink that would take you from the report to the pad in the design.

Nudge
o   Using the cursor keys to move an item left or right using 'Nudge' when 'Reversed View' was enabled would move the item in the opposite direction.

Pad Styles
o   When adding or editing a pad style exception in the Design Technology dialog, any value entered for 'Thermal' was not being retrieved from the dialog.

PDF Output
o   When the options to save plot files to a Zip file and remove them afterwards was enabled, if the PDF option to view the document on completion was also set then the PDF file was already gone by the time the PDF reader tried to display it.

Projects
o   When adding a design file to a Project, if that design file was already open in another instance of the application then the current application could quit unexpectedly.

Rename Component
o   The position of the dialog was not being remembered for next time the dialog was used.

Reports
o   The Output Reports dialog was mistakenly allowing you to select a BOM Composer (.BCT) report from the list and try to edit it as a standard 'custom' report, which had the effect of removing the BOM Composer information from the file.

Selection Mask
o   It was not always clear that any of the Select Mask options were enabled; when any Select Mask option is now enabled, the standard 'arrow' cursor will be altered to show a small 'filter' icon.

Start Page
o   If 'Live Content' is turned off, then cached startup page content from an earlier release could be displayed instead of reverting to the content installed with the current version of the application.

Status Bar
o   When selecting a component pad that has a Pad Style Override (checkbox renamed from 'Pad Style Exception'), the status bar will now show an asterisk (*) after the style name.

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.