

Version 20 Software Update Details

Problem Fixes in Version 20.0.3 (20-Dec-2016)

Apply Layout Pattern

- In some situations a track pattern was not being applied from the original pattern.

Auto Rename

- When auto-renaming a PCB design on both sides of the board at the same time, it was possible that this could cause duplicate component names to be assigned when the renames were later back-annotated to the Schematic.

Forward Design Changes

- Groups containing only Schematic-only components were listed as differences to be applied, but no changes could be made to the PCB as these were Schematic-only items, so the same differences were reported next time.

Gerber Output

- In two specific cases a filled copper shape overwrote a pad rather than leaving a hole for it.
- The application would occasionally create an unwanted output file called PLOT.GER.

Library Manager

- When using the Copy button to copy items to another library, the target library did not become the currently selected library as it used to do in the previous release.

OBD++

- In a specific case the outline of a complex filled shape was breaking out from the shape and overwriting an adjacent item.

Problem Fixes in Version 20.0.2 (08-Oct-2016)

3D View

- The 'Top Coloured' setting for 'Can' packages had no effect, causing the top to always be drawn in the same colour as the body.

Add Component

- The preference setting 'Add Component Keeps Library Style Sizes' was not having any effect on how styles of newly-added components were matched to styles already in the design.

Add Square

- Holding the Ctrl key when adding squares did not alter the shape to be centred on the cursor position.

Add Text

- Attempting to add a free text item containing "=" was incorrectly invoking the popup coordinate calculator.

Add Triangle

- Using Change Style to alter the style of a shape during Add Triangle would ignore that style change on finishing the shape.

Auto Place

- The Auto Place dialog mistakenly allowed the placement grid to be set to zero, which then caused the application to loop forever.
- In some circumstances, Auto Place could position components so that DRC would then report component-to-component errors.

Component Edit

- After selecting a package in a component that was not the 'Current' package, any change in the PCB symbol was not being saved to that package. This meant that 'Make Current' had to be used to change the current package before making any changes, thus involving more steps than were necessary.
- Pressing F1 after invoking "Import Pin Data" was not displaying the help page for importing pin information.

Design Rule Check

- Component-to-Component checking was in some circumstances generating a spurious error indicating that two components were overlapping when in fact they were not.

Drawing

- Screen grid sometimes failed to draw when panning or zooming, reported when creating or editing symbols with no technology file selected.

Drill Table

- Inserted drill tables were not using the correct coordinate precision from N.C. Drill output.

Forward Design Changes

- Groups containing items other than components were not being removed by Forward Design Changes despite being listed as removed in the report.
- When new components were added to an existing PCB design, they were placed inside the board outline on top of other components, instead of being arranged outside the board where they were easier to access for placement.

Eagle Import

- Oval (finger) pads were sometimes being imported as round pads when reading 'native' Eagle data files.
- Some imports into Schematic Symbol libraries were not bringing in all the symbols.
- Data containing component or symbol names with ")" or ")" in their name would not import.

Gerber Output

- TrueType text items were being output using a zero-width aperture for the perimeter of text characters, which was not guaranteed to be correctly interpreted by all manufacturing front-end software.

Goto Bar

- The default setting for 'Search All Project' was Off, which meant that many users did not realise that Goto could be used to find items across all sheets in a project.
- It was not possible to find a net that was used only by Copper Pour Areas.

Layers

- Sometimes it was not possible to delete unused layers from a PCB design.

Plotting

- Plot report no longer included the project name.

Setup

- No Windows Explorer association was being made for Panel files during installation.

Spacings

- Changes to spacing values in net class match pairs were not being saved to the design.

Symbol Wizard

- The spinners for some values (including 'width across symbol' and 'length of pin leg') stepped in the wrong direction (clicking up arrow caused value to be reduced).
- Drop-down lists for pin and line styles were not being populated with styles from the selected technology file.

Update Component

- The Update Component menu command was not always available on the right-mouse menu when selecting Schematic-only components.

Problem Fixes in Version 20.0.1 (15-Aug-2016)

Fonts

- The output resolution for TrueType fonts was too low to accurately represent the shape of some text characters.
- Program quit producing Gerber output using True Type fonts.
- Drawing net names and pad names inside tracks and pads using True Type fonts was too slow.

Integrity Check

- Dimensions in a PCB design were confusing Integrity Check.

Pour Copper

- In one particular design using a Pour Area around two separate board outlines, no copper was generated.
- Copper pour was often producing copper to copper errors due to a change in the previous patch.
- Was adding a spurious cut across the poured copper.