

Version 27

Software Update Details

Problem Fixes in Version 27.0.2 (26-June-2024)

3DView

- Text was missing when outside the board outline with "exclude items outside the board" setting off.

Component Editor

- Symbol pin number is -2 instead of multi-pin within component editor.
- When editing a component, Using Colours highlights page from the context menu in a PCB preview did not show control for unassigned pins.

Design Rule Check

- DRC had incorrect component silkscreen text to vias errors.
- DRC was not using the Track to Track spacing from the Net Class when checking differential paired tracks.
- DRC was incorrectly reporting CN errors in a schematic where the items were not on different nets.
- DRC was putting error markers off the working area when components are in the bin.
- Moving component names onto an electrical layer with DRC enabled used to cause Easy-PC to quit.
- For a specific design, Design Rule Check was generating some incorrect Gap in Net errors and not finding some real gaps.

Framework

- On displays with high scaling, certain list boxes and grids were drawn too small.
- Component Search Bar was not returning any results in Easy-PC.
- Activating reverse view caused the design to move off screen.
- Using Next Error did not update the status bar.
- Support report was giving incorrect operating system edition.
- PCB Wizard used to add a non-existent design file to the currently opened project.
- HTML report links were not working.

Help

- Technology Connections dialog (in SCM) navigated to the wrong help page.
- Various help fixes and improvements.

Interaction

- Pads visible on certain layers were not pickable.
- Changing the net on a selected net name label using properties did not work.
- "Route connections at pair start" did not work when routing Diff Pairs.
- Auto Complete did not work in Diff Pair routing mode.
- Whilst adding a track from a Starpoint, cancelling out of the Select Starpoint Net dialog led to Easy-PC quitting.

Libraries

- The library item 74HC11 was not correct.
- Library page Tech Files button used to display a dialog with blank tech name if it could not find the last used tech file.

Plotting

- Plot preview was not scaling text correctly.
- Drill holes were being covered when plotting to PDF.
- Auto Mask did not work when generating solder mask Gerbers with components using copper shape.
- When plotting to PDF, some fonts were plotted too wide.
- All Colours Black switch in PDF plots was not staying unchecked to plot to colour.
- Duplicated D codes in Gerber output on specific designs caused manufacturing issues.
- Drill Ident Setup table grid changes were not being saved.
- Power plane plot ignored drill sizes.

Tools

- Running Integrity Check report on a specific Project caused Easy-PC to close.
- Activated ProRouter did not run correctly.
- Activating ProRouter did not add the option to the tools menu.

Problem Fixes in Version 27.0.1 (05-Sep-2023)

BOM Composer

- BOM composer reference names field was incorrect.

Help

- Various help fixes and improvements.

Install

- SnapEDA dockable bar menu link was not visible.
- Using the spice netlist used to display an error stating the spiceio.dll is missing.

Library Tools

- ESC key used to not close the floating library manager dialog.
- New components used to not save to library when using floating library manager.

Start Page

- Start Page links used to not work correctly.

Tools

- Hatched pour at a thermal relief used to cause Easy-PC to quit.
- Transfer to PCB used to not handle combinations of new pin range values.