

PowerLogic 4.0

Release Highlights

Welcome to the latest release of PowerLogic. This release includes enhancements and new functionality as well as many defect corrections. PowerLogic 4.0 includes new functionality and enhancements in the following areas:

- **Increased Layers** – PowerLogic’s enhanced layer definition includes a default layer mode and an increased layer mode. Dialog boxes that provide layer information to the user now reflect any increase in layers.
- **ECO/ECOGEN** – Enhancements have been made for better handling of renames for components and nets. New functionality for gate and pin swaps has also been implemented. With the new enhancements users can annotate decal changes during ECO operations.
- **32 Colors** – PowerLogic’s color bar now provides 32 color choices.
- **Bus Enhancements** – Users can now define busses in PowerLogic with heterogeneous net names.
- **Saving Schematic Parts to Library** – PowerLogic can now save a part to the library from within the schematic editor.
- **Object Selection in Decal Editor** – Object select mode in PowerLogic has been expanded into the Decal Editor.
- **Change Part with Fewer Pins** – Users can now change a part type even if the new part type has fewer pins or gates.
- **Non ECO-Registered Parts** – The addition of non ECO-registered parts is supported in PowerLogic 4.0. Users can either include or exclude non ECO-registered in the PCB netlist and ECO compare operations.
- **Symbol Wizard Enhancements** – The Symbol Wizard has been updated to place component pins on all four sides of a CAE decal.
- **Library Conversion** – The Library Conversion utility converts existing version 3 libraries to the new version 4.0 library format. You **must** convert your libraries.

Increased Database Layers

PowerLogic's layer definition features now include a default layer mode and an increased layer mode. Dialog boxes that provide layer information have been changed to reflect any increase in layers. This increased layer information will pass directly to PowerPCB.

- **Increased Layer Count** – You can now have up to 250 total layers within the PowerLogic design environment. You can define up to 64 electrical layers and up to 186 non-electrical (documentation) layers.
- **Decreased Layer Count** – PowerLogic has new functionality for removing a layer from the design. This function enables users to remove unwanted layers from their designs.

Enhancements to ECOGEN

Enhancements have been made to support user requests for better handling of renames for components and nets. New functionality for gate and pin swaps has also been implemented. Enhancements now allow users to annotate decal changes during ECO operations.

- **ECO Decal Annotation** – You can now update decals using PowerLogic's incremental ECO operations for forward and back annotation to PowerPCB.
- **Improved Auto-Rename** – Enhancements have been made to ECO algorithms to improve auto-rename handling and operations where rename results in change part commands from ECOGEN comparison.
- **Pin and Gate Swap** – Support has been added for pin and gate swap operations.

Support for 32 Colors

- PowerLogic's color bar now provides 32 color choices.

Bus Enhancements

PowerLogic users have long requested the ability to define busses with heterogeneous names. That is, each net in the bus can have a totally unique name rather than adhering to "typical" bus naming conventions (DATA1, DATA2, DATA3, and so on).

- Any collection, or bundle, of net names may be drawn as a single bus. These bundles may also include groups of consecutive net names.

Saving Schematic Parts to Library

PowerLogic can now save a part to the library from within the schematic editor. This command will work on multiple selections including all parts on the schematic.

Object Selection in Decal Editor

The past two releases of PowerLogic have seen significant enhancements to increase its object select mode. One of the goals was to make PowerLogic behave more like PowerPCB for selecting objects. This object select mode evolution has now been expanded into the Decal Editor.

- Users will be able to select multiple terminals for copy, move, delete, set pin type, and set pin swap.
- Drawing items and text will have the appropriate equivalent object selection operations to that currently available in the schematic editor.
- A Query/Modify Terminal dialog box is provided that will work with either single or multiple selections of terminals.

Change Part with Fewer Pins

Users can now change a part type even if the new part type has fewer pins or gates.

- The connections going to the missing pins will not be deleted, they will be attached to automatically generated off-page symbols. The user will be notified of all disconnected pins.

Non ECO-Registered Parts

The addition of non ECO-registered parts is supported in PowerLogic 4.0. Users can either include or exclude non ECO-registered parts in the PCB netlist and ECO compare operations.

- Non registered parts without terminals will be allowed. This lets the user add parts such as nuts, bolts, and card extractors that can then be included in the BOM.
- Non registered parts can also be used for a sheet title block with the attribute labels on the part used to display sheet identification text.
- The Query/Modify Part dialog box will show the ECO registration status of a part.

Symbol Wizard Enhancements

The Symbol Wizard has been updated to place component pins on all four sides of a CAE decal.

Library Conversion

The Library Conversion utility converts existing version 3 libraries to the new version 4.0 library format. You **must** convert your libraries.

- **Conversion of version 3.0 libraries** – This utility can convert all version 3.0 formatted libraries to the new version 4.0 library format. Files with the following extensions are affected: *.pt, *.pd, *.ld, and *.ln.

All products mentioned herein are trademarks or registered trademarks of their respective companies.