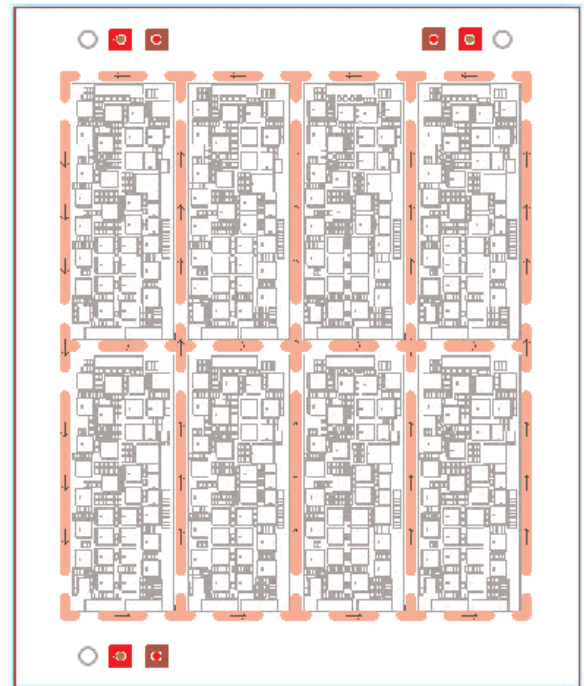


Pulsonix 8.5 Release

Panel Editor

The new Panel Editor feature allows customers to take control over pre-fabrication processes. It enables multi-PCB designs to be panelised ready for plotting and on into the manufacturing stage. The Panel Editor is more than just a plotting station; it allows panelisation of designs, addition of test coupons, fiducial markers, documentation and fabrication details. New facilities within it allow tab-routing paths to be defined. With control over the addition of tab direction, tab thickness, tab spacing and mouse bites, it enables accurate manufacture-ready breakout tabs to be created. V-Score paths can be added using the shape tools and then processed for manufacture, all within the Panel Editor environment.

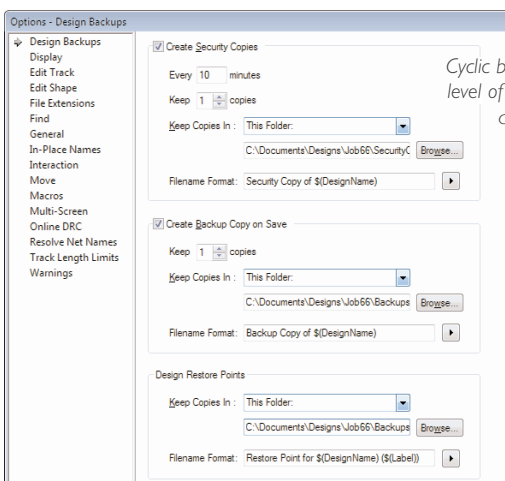
Powerful new tools within the Panel Editor allow manufacturer-ready panels to be fabricated



Cyclic Backup Methodology

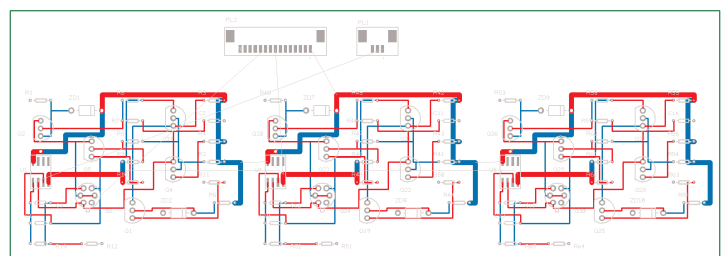
Cyclic design backups and security saves have been added for Pulsonix 8.5. The style of naming and backup criteria allows customisation of this critical process. Users are able to choose backup styles to allow incremental or decremental naming or infinite naming. Additionally, a Restore point can be chosen during the design process which can be recovered at any point. Multiple Restore points can be chosen either manually or automatically.

Cyclic backups add another level of reassurance to your design process



Group Pattern Reuse

Multiple group selection for design reuse has been added to enhance the current reuse placement algorithm. When a selection has been made, this can now be applied to multiple groups in the design. Once placement has been made, multiple positional rearrangement options can also be applied to the existing design using a new placement pattern from the original design. These two new options significantly save precious design time and ensure inaccuracies are removed from the process.



Multiple group selection and multiple position rearrangement options lead to enhanced accuracy

IPC-2581 Netlist Export

The IPC-2581 netlist export is an intelligent vendor-neutral format that will serve as an alternative to traditional manufacturing outputs such as Gerber and ODB++. Pulsonix 8.5 will support this format in both its product and as a corporate consortium member.



Support for Gerber X2

Pulsonix 8.5 will support extensions to UCAMCO's Gerber format to produce Gerber X2 compliant files. The Gerber X2 format supports essential design attributes which themselves more closely support the transfer of PCB designs to manufacturing.



Library Technology Partners

In order to provide access to enhanced libraries to its users, agreements have been signed with specialist library companies; PCB Libraries, Accelerated Designs and SnapEDA. Each vendor providing different and unique libraries to enhance Pulsonix. Working with these vendors further cements the relationship by enabling Pulsonix users direct access to manufacturer's parts on their web sites.

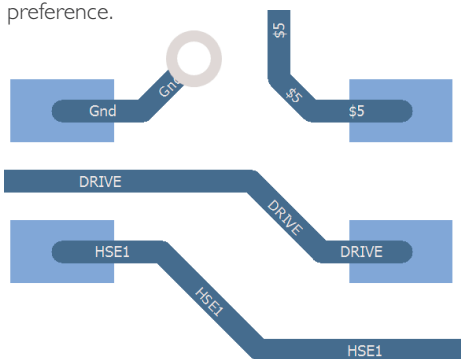


New Native Library Format

To facilitate the new Library Technology partnerships, the Pulsonix library manager has been enhanced to allow drag and drop methodology directly into the library using its new neutral format. Users simply drop the selected parts onto Pulsonix making them ready to use in one simple operation.

Net Names Displayed in Tracks

The visibility of Net Names in tracks has been added. As well as the display, control of the auto-hide zoom rate and font used have also been added. Visibility can be toggled on or off as a user preference.



Net Names in tracks can be toggled to display

Additional Features Summary

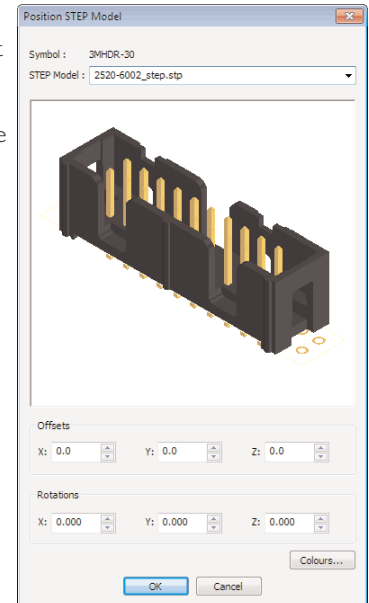
- Drag & Drop Native Format to Parts Library
- Reverse Engineer - match PCB placement
- 'Spread Out' feature
- Set Symbol Origin At Centre Of Pads command
- New enhanced accelerated graphics mode
- Add Part To Design from Library Manager
- Expanded Design Tooltips
- Grids Displayed as X-Crosses
- Draw Current Interactive Grid Only
- New View Highlighted option
- Advanced User-defined Report Naming
- Override Component Pad Position on Components
- Cross probe of Star (Delta) Points
- Import Placement CSV
- Cyclic Design Backups
- Additional advanced DRC checks
- Select STEP Filename Model from drop down list
- Attribute <STEP Filename> available for Doc Symbols
- Hatch Lines Using Percentage Width and Offset
- Floating Network Licenses - Disconnect Client

Position STEP Model

The introduction of a dynamic graphical positioning tool for adding STEP models to the footprint enables accurate visual association between the two items. By changing values within the dialog, instant visual feedback takes the guesswork out of positioning the STEP model.

As an additional enhancement to all dialogs that use the STEP Filename attribute, the STEP Filename Model can be selected from a drop down list from the library.

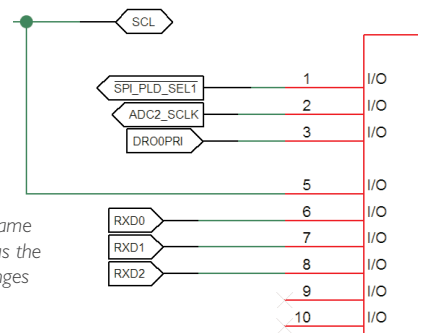
A new option, Save STEP Models, has been added to enable the 'collection' of all STEP Models used in the design and saved to a folder location. The dialog reports any items that have models or are missing models.



Position STEP Models in a footprint using the graphical dialog

Dynamic Net Name References

Add dynamic net name markers using this new feature. The text callout box changes size to fit the size of the net name being added.



Dynamic net name markers resize as the net name changes