

# Pulsonix Version 9.1 Update

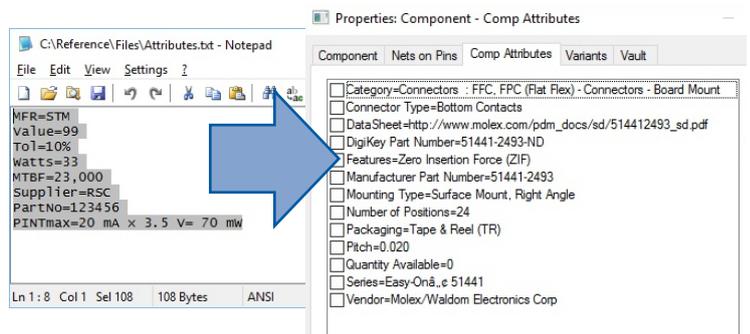
## Extended Capabilities

As an interim release, Pulsonix 9.1 extends the capability of the main 9.0 release at the start of 2017 with the introduction of new and enhanced features. Many features are refinements on existing functionality to make its use more efficient.

## Drag-and-drop Attribute text

Pulsonix V9.1 introduces the ability to use text created in a file and paste it into a Component in the design or library to populate attributes. By either using copy/paste or drag and drop for applications that support this, the addition of attribute values is straight forward and simple to do now. For multiple attribute entries, creation is maximised and ultimately speeds up your library creation process.

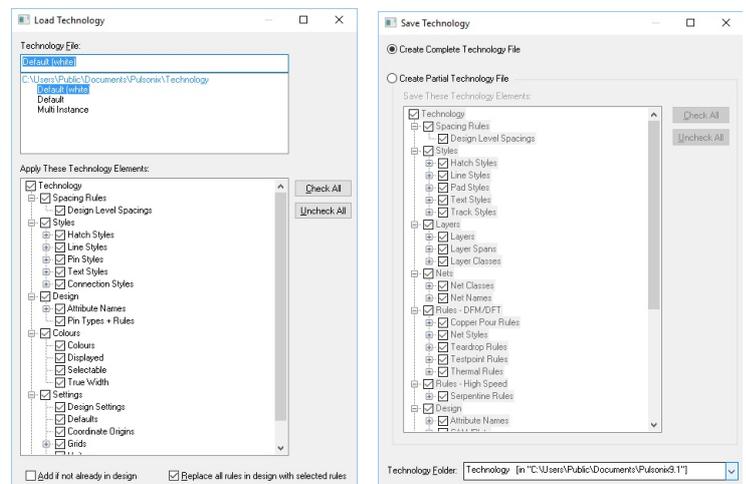
Quickly drag and drop text from a file onto a Pulsonix component to instantly create attribute properties



## Selective Load & Save Technology Files

When writing or importing Technology files, you can interactively select the facets of the technology file to use. Along with this change, new design facets for save and import have also been introduced, such as Units and extra entities within the Design Settings.

Technology files can be saved and loaded selectively to suit your company standards and design requirements



## Enhanced PCB Layers Bar

Within the PCB Layers bar, visual indicators will display the current Electrical and non-Electrical layers. When adding tracks or copper shapes or documentation shapes, these default layers will be used. The default can easily be changed by double clicking on it. This dialog can also be used to display the current status of a powerplane for powerplane layers. This will toggle the View Powerplane mode on or off efficiently.

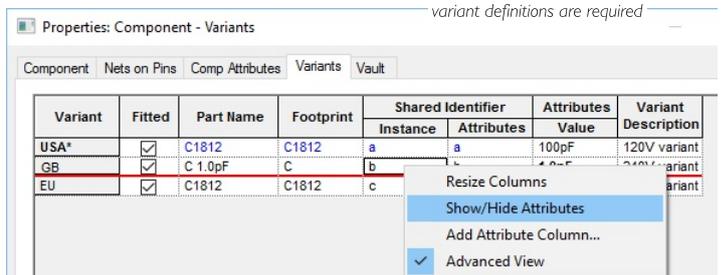


Fast selection of default layer usage and instant display of powerplanes increases design productivity

## Variant Spreadsheet Advanced View

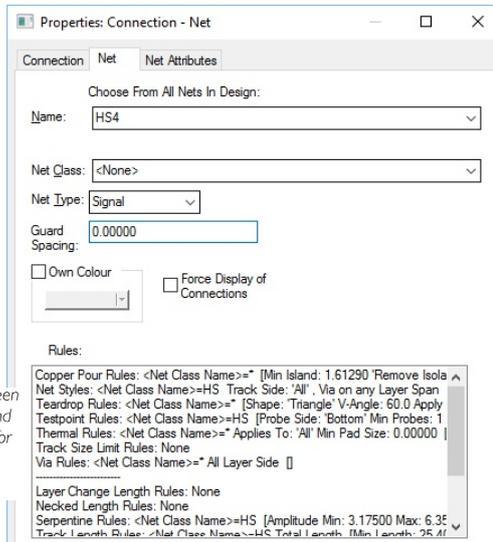
The Advanced View allows you to create complex Variant rules within a Part. Where there is a requirement for the same Part but where a variant of it uses a different attribute value or footprint, this new mode would be used. Where the Part variant has the same instances in each, the standard mode can be used.

The Advanced View is utilised when complex component variant definitions are required



## Shows Rules in Net Properties

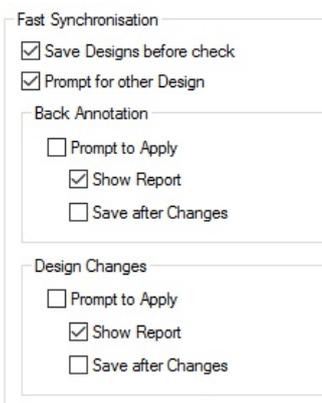
The Net Properties dialog now displays all the rules associated with a selected net. This enables you to verify the nets currently assigned rules in a convenient dialog.



Where complex rules have been defined for the Technology and constraints, they are shown for the selected net using the Properties dialog

## New Fast Synchronise option

A new 'single click' mode for the Synchronise Design option enables a much faster update of the PCB with no confirmation questions or reports of changes to be made. This new option will just do it based on a set of pre-defined user selections.



Single click design synchronisation provides fast design update to help speed up your design process

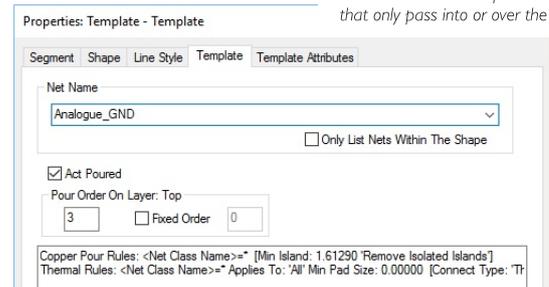
## Import Native Eagle Libraries using drag/drop onto the Library Manager

Native Eagle EDA libraries can be dragged from your desktop and onto the Pulsonix Library Manager to import library content. This mode will display the library chooser dialog from where you can selectively import content and select the library to use.

## 'Used' Nets within Templates

After a Copper Pour Template has been defined, for nets that pass through this area, the new check box within the Properties dialog allows you to just show the nets that pass through the area and not all the nets in the design. This helps simplify the net choice to speed up the selection and definition process.

To aid net selection for a new template, the Net Name listed can be pre-loaded with nets that only pass into or over the Template



## Feature Summary:

- Symmetric Net Class and Match Pair Level Spacing Rules
- Variant Spreadsheet & Properties Column visibility
- Variant visibility on Status Bar
- Attribute Editor - Hide Show Attribute Columns
- New Report Maker commands
- Column sorting in Technology dialogs
- New Template Pour Order dialog
- Template Properties shows Rule used
- Option To Hide Poured Templates
- Option to Move Cursor To Placement Origin
- Option to Highlight Isolated Copper
- Selection options for Component Origins
- Brighten items mode during Find
- Net Type Attribute available for Technology Rules
- PDF Output now shows Part Description
- Output Gerber X2 to Drill format
- Extended command set for Scripting
- Input values can now be expressions
- Apply Vias now available in Footprint editor
- Apply Vias now available for non-electrical shapes
- Edit Attribute Value on double-click
- Find XY command in neutral mode
- New Toggle All Hollow command
- Own Net Name Colour in Schematics
- New Snap Move to Orthogonal Only mode
- DRC Errors Bar uses find parameters
- Typed Pin/Net Name Insert Connection improvements