



Pulsonix Design System V9.1 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2017

Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

Copyright in the whole and every part of this software and manual belongs to WestDev Ltd. and may not be used, sold, transferred, copied or reproduced in whole or in part in any manner or in any media to any person, without the prior written consent of WestDev Ltd. If you use this manual you do so at your own risk and on the understanding that neither WestDev Ltd. nor associated companies shall be liable for any loss or damage of any kind.

WestDev Ltd. does not warrant that the software package will function properly in every hardware software environment.

Although WestDev Ltd. has tested the software and reviewed the documentation, WestDev Ltd. makes no warranty or representation, either express or implied, with respect to this software or documentation, their quality, performance, merchantability, or fitness for a particular purpose. This software and documentation are licensed 'as is', and you the licensee, by making use thereof, are assuming the entire risk as to their quality and performance.

In no event will WestDev Ltd. be liable for direct, indirect, special, incidental, or consequential damage arising out of the use or inability to use the software or documentation, even if advised of the possibility of such damages.

WestDev Ltd. reserves the right to alter, modify, correct and upgrade our software programs and publications without notice and without incurring liability.

Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 13/07/17 iss 1

Pulsonix

20 Miller Court
Severn Drive
Tewkesbury Business Park
Tewkesbury
Glos, GL20 8DN
United Kingdom

Support Phone +44 (0)1684 296 570

Sales Phone +44 (0)1684 296 551

Fax +44 (0)1684 296 515

Email info@pulsonix.com

Web www.pulsonix.com

Contents

CONTENTS	3
VERSION 9.1 UPDATE SUPPLEMENT	5
Installing the New Version of Pulsonix.....	5
Licensing	5
Version 9.0 Update Notes	5
NEW IN VERSION 9.1	6
Net Class and Match Pair Level Spacing Rules.....	6
Variant Spreadsheet & Properties Changes	6
Advanced View	6
Column visibility.....	7
Variants Manager Changes	8
Attribute Editor - Hide Show Attribute Columns.....	9
Report Maker Changes	9
Shapes Area Command.....	9
List of Associated Parts Command.....	10
Is Locked Command for Components.....	11
Net Properties shows Rules.....	12
Load & Save Technology.....	12
Message to Save Technology	15
Other Changes to Technology Dialogs.....	16
Import Native Eagle Library files by dropping onto Library Manager	17
Minor changes to Design Settings dialog	17
Drag-and-drop Attribute text.....	18
PCB Layers Bar Changes.....	18
Toggle All Hollow Command.....	20
Net Name in Net Colour for Schematics.....	21
New Fast Synchronise option	21
Options for Area Track Style change	22
Minor size changes to Options dialog	22
Options dialog now resizable.....	22
Display page reorganised	22
Changes to General and Interactions pages	22
New categories added – Select and Tooltips.....	23
New Select Options	23
Back Annotation Changes for High Speed Design option	24
Snap Move to Orthogonal Only.....	24
‘Used’ Nets within Templates	24
Template Pour Order	25
Template Properties shows Rule used.....	26
Option To Hide Poured Templates	27
Options Highlight Isolated Copper.....	28
Options Move Cursor To Placement Origin	28
Options - Display.....	29
Selection Origin changes.....	29
Find XY	30
Find Bar Changes	30
DRC Errors Bar.....	31

4 Contents

Insert Connection – Type Pin Name improvements (PCB)	32
Apply Vias in Footprints.....	33
Changes to Apply Vias in PCB	33
Enhancements to the Add Polygon option.....	33
Net Type Attribute.....	34
Edit Value on double-click.....	34
Input values can now be expressions	35
Changes to PDF Output	35
Output Gerber X2 to Drill format	36
Changes to Scripting.....	37
Load configuration file.....	37
Rename items in library	37
Run script in other instance.....	37
Set CAM/Plot output template.....	37
Generate STEP output.....	38
Get extents of Board outlines	38
Generate ODB++ output	38
Generate IPC2581 output	39
Auto-generate plots.....	40
New 'InputDialog' command.....	40
Changes to Technology Files	40

Version 9.1 Update Supplement

Installing the New Version of Pulsonix

It is always recommended that you back-up all libraries, designs, technology files, profile files and report files before installing the latest version. Other than for any technical reason, this is good working practice, although you should already have a backup of this data!

To install Pulsonix, double-click on the download executable and wait for a short time. You may have been supplied a CD, in which case insert it and wait for it to run. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 9.1 on top of your existing V9.0 installation. If upgrading from V8.5, you can install it alongside the older version if you prefer. In any case, you do not need to uninstall the old version first unless you wish to remove it from your hard drive.

Licensing

Version 9.0 requires a new license if a new user or upgrading from an older version.

Version 9.1 does not require a new license if you already have V9.0. If you do not have V9.0 already, then you will require a new license.

If you are using a version of Pulsonix earlier than Version 8.5, you will require a new license, this would have been supplied to you under the terms of your maintenance contract.

For existing users upgrading from a previous version, it is recommended that you save the new license and overwrite the existing one. When requested during installation, simply click the **No Change In Licensing** check box on the licensing page of the installation wizard. The **License Manager** can be used to add new licenses and make changes to network licensing after the installation has been completed.

Version 9.0 Update Notes

Update Notes for Version 9.0 are available in the Pulsonix installation folder under \documents. Alternatively, select the direct link within the product itself: **Help menu | Online Manuals> Pulsonix V9.0 Updates Notes**

New In Version 9.1

Features documented here are specifically for Version 9.1. We have also included the update notes from Version 9.0 below this section for completeness. You may notice changes in dialogs between what is documented for V9.0 and that of V9.1. Improvements and new features have been added between the two releases based on customer feedback and review.

Net Class and Match Pair Level Spacing Rules

From within both **Net Class Level Spacing Rules** and **Match Pair Level Spacing Rules**, there is a new check box, **Symmetric Grid**. Selecting this option forces these spacing grids to be symmetric (i.e. Track to Pad equals Pad to Track).

The dialog below shows the option checked and the dialog greyed out in the non-editable portion (highlighted in red):

Match - <Net Class Name>=*	Match - <Net Class Name>=*						
	Track	Pad	Via	Testpoint	Mounting Hole	Copper	Text
Track	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Pad	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Via	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Testpoint	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Mounting Hole	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Copper	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Text	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400
Board	0.10160	0.10160	0.10160	0.10160	0.10160	0.10160	0.25400

Minimum Spacing: (except for explicit smaller net class value) Symmetric Grid

Variant Spreadsheet & Properties Changes

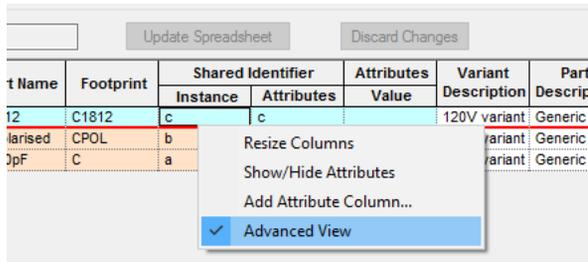
Advanced View

The variants grid in **Component Properties** and the **Component Variant Spreadsheet** now show all the **Component Attributes** and the actual **Footprint** name.

Component: C1		Update Spreadsheet		Discard Changes		
Variant	Fitted	Part Name	Footprint	Attributes		
				<Spice Device>	Category	Value
USA*	<input checked="" type="checkbox"/>	C1812	C1812	C	Capacitor/G	
GB	<input checked="" type="checkbox"/>	C Polarised	CPOL	C	Capacitor/G	22pf
EU	<input checked="" type="checkbox"/>	C 1.0pF	C	C	Capacitor/G	1.0pF

By default, the **Shared Identifier** columns are hidden unless needed, they can be shown using the context menu **Advanced View** option by clicking the right mouse button.

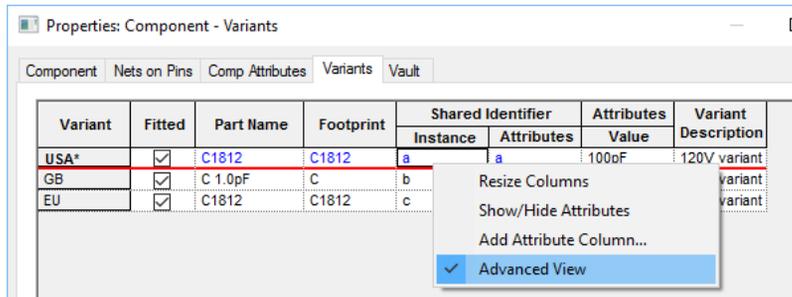
With the simplified view, instances are only split if the footprint is different, and attributes only split if there is a difference in the Part Name, Database identifier or some attributes are different, and the identifiers are allocated automatically.



This option would only be used where a difference 'instance' is required, for example, the same footprint but in a different location.

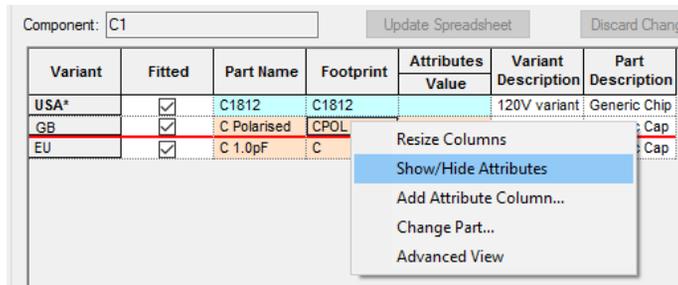
Properties

When using the **Component Properties** dialog and **Variants** page, you can also right click in the grid to reveal the context menu from where you can select **Advanced View**.



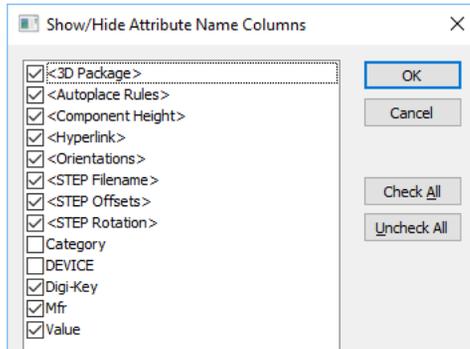
Column visibility

You can also choose the attribute columns to display by using the **Show/Hide Attributes** button on the context menu.



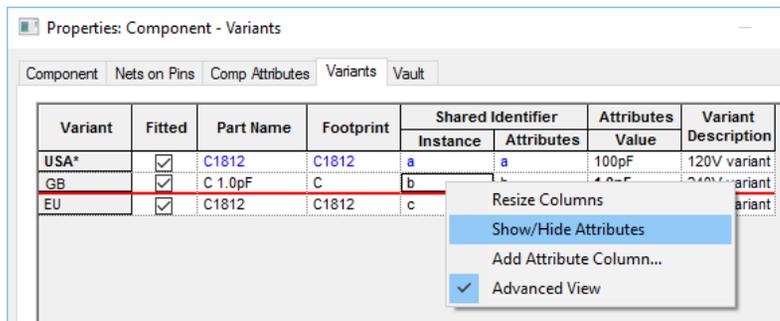
8 Pulsonix Version 9.1 Update Notes

On selection, this will display a list of attributes and a check box to select whether they are displayed (checked) or not. Additional buttons enable you **Check All** or **Uncheck All**.



Properties

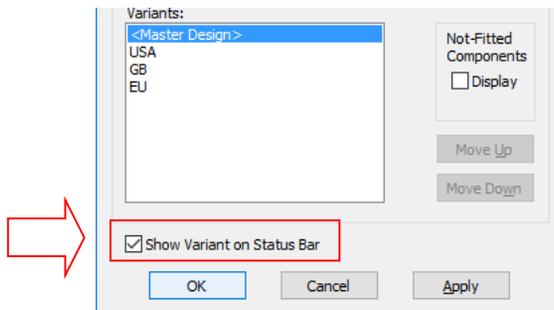
When using the **Component Properties** dialog and **Variants** page, you can also right click in the grid to reveal the context menu from where you can select **Show/Hide Attributes**.



Variants Manager Changes

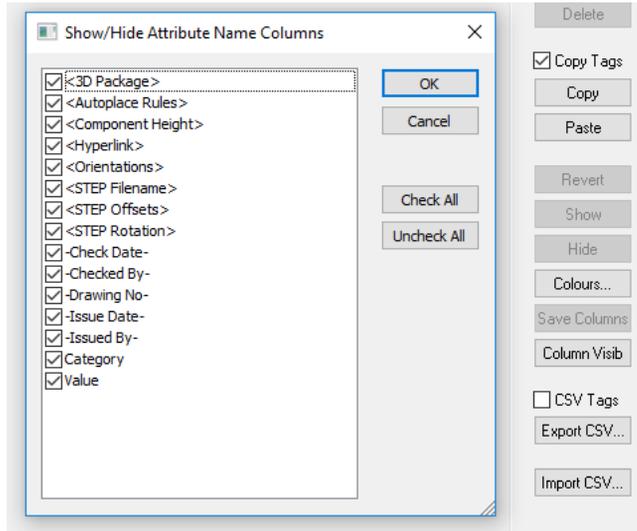
A minor change has been made to the **Variants Manager** following on from a new feature in Version 9.0. The ability to Show or Hide the **Variants** status on the **Status Bar** has been added as a check button in the Variants Manager. Use this switch to declutter the status bar.

Uncheck the **Show Variant on status Bar** check box to hide the current variant on the Status bar.



Attribute Editor - Hide Show Attribute Columns

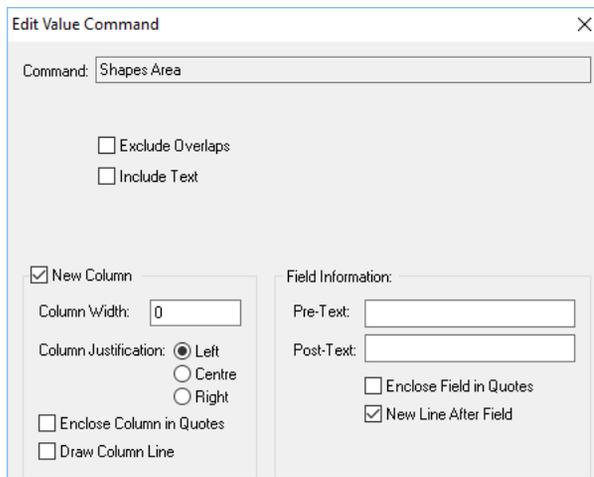
You can show or hide columns within the **Attribute Editor** or on drop down menu on the Part Editor Attributes Page. Check boxes controlling the display can be accessed by **Column Visib** button. Each check box represents the available attributes used in the appropriate designs.



Report Maker Changes

Shapes Area Command

There is a new **Shapes Area** command that reports the total area of all copper items or documentation shapes on the current layer.



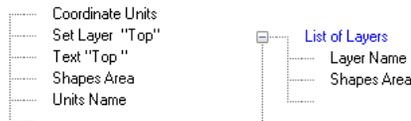
10 Pulsonix Version 9.1 Update Notes

If a shape is an open shape or closed and not filled, just the area of each segment in the outline will be included. This command uses the CAM Plot front end to decide which shapes and pad shapes to include. All information produced is based on the Layer Class settings for that layer.

If the current layer is of type non-electrical, pad shapes and board outlines are included if enabled by the layer class. Electrical layers include all copper shapes, tracks and pad shapes. Power plane layers include the copper area of the plane.

The command has two options: use **Include Text** to include the area of shapes that make up any text items on the layer. Use **Exclude Overlaps** to not count any areas that overlap, this is more accurate but can take a long time to calculate on large boards.

This command can be used at design level or within a **List of Layers** command. If at design level the current layer must be set using the **Set Layer** command.

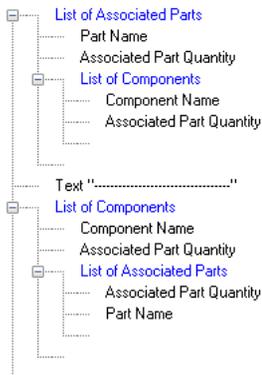


The report looks like this:

Top	824.78073
Solder Mask Top	1455.91249
Paste Mask Top	580.52632
Bottom	302.13025
Solder Mask Bottom	562.14975
Paste Mask Bottom	103.35725

List of Associated Parts Command

You can now use the **List of Components** command on an **Associated Part** (within a **List of Associated Parts** command). It reports the components that use the current associated part.



Reports looks like this when formatted:

```
Board (1)
Bolt 2.4mm (12)
  T1 (6)
  T2 (6)
```

```

Heatsink TO3 (2)
    T1 (1)
    T2 (1)
HEATSINK-5F (1)
Nut 2.4mm (4)
    T1 (2)
    T2 (2)
Plate (1)
Washer 6mm (4)
    T1 (2)
    T2 (2)
-----
PL1 (0)
T1 (11)
    6 x Bolt 2.4mm
    1 x Heatsink TO3
    2 x Nut 2.4mm
    2 x Washer 6mm
T2 (11)
    6 x Bolt 2.4mm
    1 x Heatsink TO3
    2 x Nut 2.4mm
    2 x Washer 6mm
    
```

The **Associated Part** command **Quantity** has now been renamed **Associated Part Quantity** so that it can be understood when used on a component.



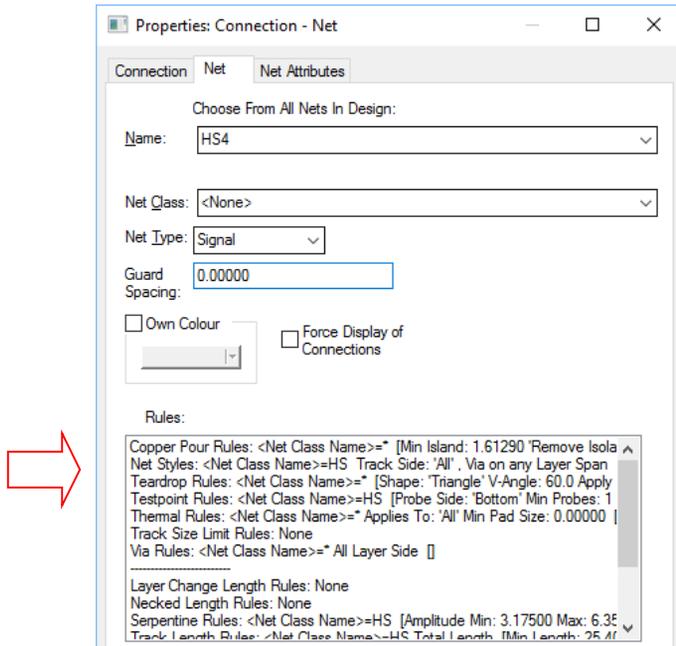
Is Locked Command for Components

You can now use the **Is Locked** command on **Components**. Previously, it was only available for error markers.



Net Properties shows Rules

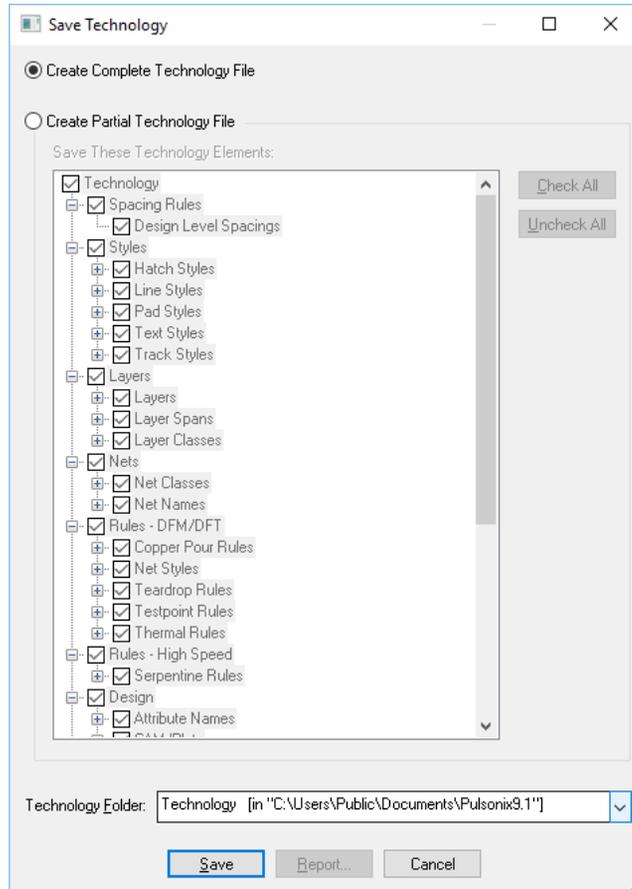
Properties of a **Net** now shows list of matching rules associated with the selected net. Rules cannot be changed from this list, only viewed.



Load & Save Technology

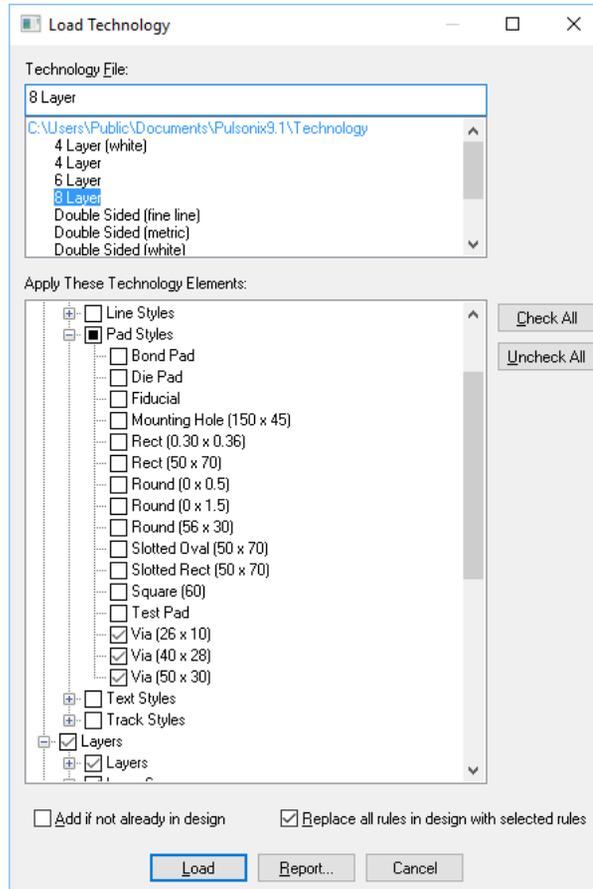
Save Technology now has a resizable dialog that allows you to choose between saving a **Complete Technology File** and saving a **Partial Technology File** with just some items in it. Both options save to a Schematic, PCB or Part Tech file.

For a **Partial Technology** you can use the element tree to check the items you want saved. This tree will open to individual item level so you can, for example, create a file with just three Pad Styles. You can choose which technology folder to save in, allowing you to keep the partial tech files separate. Choose the sections required by checking or unchecking them. When Save is pressed, you will be promoted with a **Save Technology File** name in the folder designated on this dialog.



The **Save Technology** option can now also be used when editing a Technology file.

Load Technology can load either a **Complete** or **Partial Technology File**. Choosing a partial Technology File will only show items that were saved into it. The “new” element tree now opens to item level, so you for example you can choose to load a specific Pad Style. The dialog has a list for choosing the required tech file.



There are some new element sections as follows:

Spacing Rules – Now has sub sections **Design Level**, **Net Class Level** and **Match Pair Level**. The latter two have sub sections with the individual items.

Drill Sizes – Drill Size Settings has been split off as a separate section

Colours – Displayed is renamed from Visibility and Selectable and True Width are new.

Design Settings – Defaults, Coordinate Origins and DRC Acceptance Rules have been split off as separate sections.

Units – new

Design Level Attributes – new

Dialog switches

Apply These Elements: The tree control in the middle of the dialog displays the available technology elements from the chosen file in a structured manner. The check boxes allow you to selectively copy technology information from the chosen technology file. Clicking on a check box to the left of a section header will toggle between checking and unchecking all items within the section. If you select just some items in a section the header check box will show as a partial selection. The "Check All"

and **Uncheck All** buttons are provided as an aid in setting up the choices you need. Click on the plus and minus symbols to open and close branches of the tree. The tree can be enlarged to show more elements, see **Resizing the Dialog** section at the bottom of this page for details.

Add if not already in design: If left unchecked, only those technology elements already in the current design will be updated from the technology file. If this button is checked, any elements not already in the design will be added to the design.

Replace all rules in design with selected rules: This only applies to any Rules sections being loaded, including Sub nets and Match Pair Level Spacings. If checked, all rules in the design of the type being loaded will be removed and replaced by the chosen rules from the technology file. If left unchecked, matching rules in the design will be updated and new rules in the technology file will be added depending on the **Add if not already in design** switch. Note: new rules may need to be repositioned above or below the existing rules to take effect.

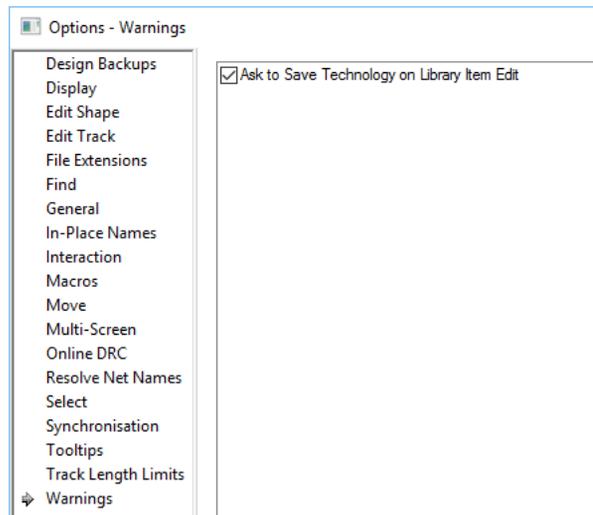
Load: Press this button to load the information from the technology file into the design according to the settings on the dialog. No report will be given at this stage, so if you want to know what technology elements will be loaded press the **Report** button first.

Report: Press this button to view a report of the changes that would be made if you were to press **Load**. The report includes details on different values found and items not yet defined in the design. An extract from a report is shown below, in which you can see that a new track code would be added, and the Bias on the Top and Bottom Copper layers would be reversed.

Message to Save Technology

You can now turn off the message to save modified **Technology** when editing **Footprints, Symbols** and **Parts**. If you turn it off, no save will be done.

If you wish to switch this option back on, use **Options** and **Warnings**. Select the check box on the option **Ask to Save Technology on Library Item Edit**.



Other Changes to Technology Dialogs

Sorting Pad Styles

Within the **Technology** dialog, you can now sort **Pad Styles** by clicking on the column headers.

Below shows the list sorted by **Shape**:

	Name	Layer	Shape	Width	Length
Y	Pad (0.5mm)		Octagon	0.48260	
	PadStyle21		Oval	1.52400	2.540
	61		Rectangle	0.76200	3.048
Y	Rect (0.2mm x 1.5mm)		Rectangle	0.24994	1.500
Y	Rect (0.2mm x 1.5mm)1		Rectangle	0.19990	1.500

Below shows the list sorted by clicking on **Name**:

	Name	Layer	Shape	Width
	Mounting Hole		Round	3.8100
Y	Pad (0.5mm)		Octagon	0.4826
Y	PadStyle1		Round	1.5240
Y	PadStyle2		Round	1.5240
Y	PadStyle3		Square	1.5240

Adding New Rules

Using the **New** button in any of the **Technology Rules** dialogs, it now inserts the item at the current position. If, for example, the top row is selected it will insert the item at the top of the list.

Apply To Selected Nets

Within the **Technology Nets** dialog, right clicking on the columns for Check Same Net, Use Own Colour, **Colour and Display Connections**, you can now select options to **Apply to selected cells** and **Apply to all with same Net Class**. Previously, this was only available for Net Class and Guard Space columns in V9.0.

	Name	Net Class	Type	Guard Space	Check Same Net	Use Own Colour	Colour	Display Connection	Att
Y	+5V		Power	0.00000	<input checked="" type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A		Signal	0.00000	<input checked="" type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	ame>
Y	ADD1		Signal	0.00000	<input checked="" type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	ame>
Y	ADD2		Signal	0.00000	<input checked="" type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	ame>
Y	ADD3		Signal	0.00000	<input checked="" type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	ame>
Y	ADD4		Signal	0.00000	<input checked="" type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	<Net Names

Multiple Row Selection

For **Differential Pairs**, **Signal Paths** and **Sub Nets**, as with the **Nets** dialog, you can now use **Ctrl-select** or mouse drag to select multiple rows. With multiple rows selected, the current item details at the bottom of the dialog is the top most item selected. With multiple rows selected, you can now also right click on the cell which now has the **Apply to selected cells** option available if it is allowed.

Name	Chain Link Name	Use Own Colour	Colour	Display Connection	Edge Coupled
C12.2-R41.1 C13.2-R39.2		<input checked="" type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>
C18.1-U17.A6 C17.1-U17.A5		<input type="checkbox"/>		<input type="checkbox"/>	<input type="checkbox"/>
C18.2-P5.5 C17.2-P5.6		<input type="checkbox"/>		<input type="checkbox"/>	<input type="checkbox"/>
C20.1-U17.A10 C19.1-U17.A9		<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>
C20.2-P6.3 C19.2-P6.2		<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>

Apply to entire Column
Apply to selected cells

Import Native Eagle Library files by dropping onto Library Manager

Eagle native library files (.lbr) can now be dropped onto the Pulsonix **Library Manager**. The previous method of using the Import button can still be used but the drag and drop methodology is far more convenient.

When drag and drop is used, the following dialog is used once the Technologies dialog has been cleared:

Save Items from Dropped Designs X

Parts

PCB Footprints

Scm Symbols

Parts: 4 PCB Footprints: 2 Scm Symbols: 2

DIL40 HCTL-1100-40 HCTL-1100-44 PLCC44	DIL40 PLCC44	HCTL-1100-40 HCTL-1100-44
---	-----------------	------------------------------

Note: This can only be done with Eagle files at V6.x or later. Older files are not stored in XML format and must be converted using ULP files supplied in Pulsonix.

Minor changes to Design Settings dialog

Resizable, position remembered and size optimised

The **Design Settings** dialog is now resizable using the grabber handle in the bottom right hand corner.

The position and size is now remembered on close and reopen.

Overall, the **Design Settings** dialog has been reduced in height for a more optimal fit on some PCs.

A new menu option, **Settings**, now categorises existing options but this style of heading enables the Settings page to be folded away.

The dialog list order for **Settings** and **Defaults** has been swapped to ensure that **Synchronisation** is more prominent.

Drag-and-drop Attribute text

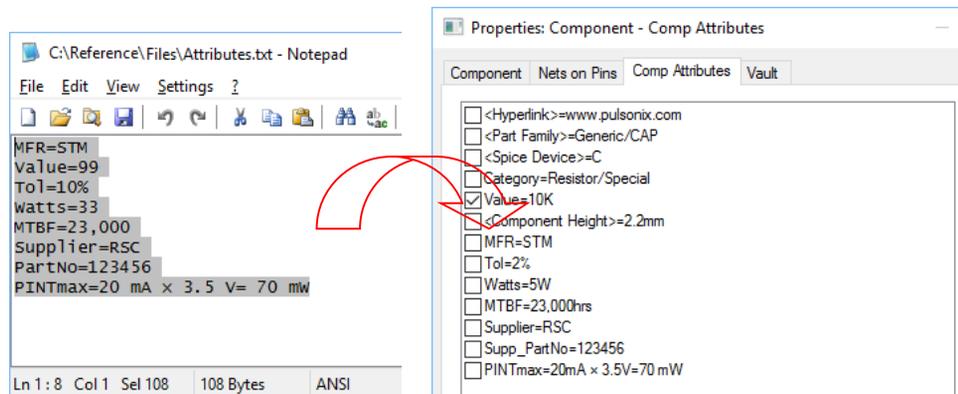
Drag-and-drop one or more text files, or a block of text from another application, and drop it onto the application to add attributes to Component or Part. This is available from the **Attributes** tab on the **Properties** dialog for Components and the **Attributes** page in **Part Editor**.

This facility is hard-wired to expect the attribute text in **name=value** form, using = as the separator. Any line of text that does not fit with this form will be ignored. Any name that already exists, will have the value replaced without prompt or warning.

Using Drag & Drop

There are two methods: select the file within the Windows Explorer and drop it onto the **Component Attribute Properties** or **Part Editor Parts Attributes** pages.

Alternatively, select the required text in the file itself and drag out of the text file and onto the attributes pages, either **Component Attribute Properties** or **Part Editor Parts Attributes**.



Note: Copy Paste does not work for this function, the text must first be selected, then dragged to the appropriate Pulsonix dialog.

Also, the standard Windows Notepad editor does work for drag and drop because of its own limitations. Other text editors such as Word or Notepad2 do however work.

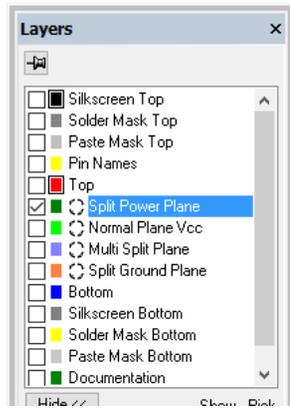
Also, some system attributes defined as part of the Part definition, such as <Part Family> or <Part Description> cannot be added or changed using drag and drop.

PCB Layers Bar Changes

New functionality has been introduced to the **PCB Layers Bar**:

- Ability to define the current (default) electrical and documentation layers
- Ability to show and hide powerplane layers
- Display notation of power plane layers
- 'Details' pane for Categories in Footprint Layers Bar

The amended layers bar now looks like this:



Defining Current Elec and Doc Layers

The colour squares are slightly smaller now allowing a black square around both the default electrical and documentation layers. Left click on the squares to set that layer as the default. If there is no common default (i.e. all categories do not have their default set to the same layer) the black square is not drawn.



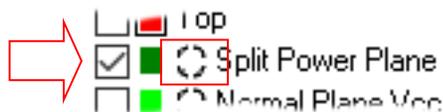
When starting a new track or adding a new doc shape, the default layer will be used.

On the context menu, the selections for the default layer change depending on the layer type selected:



Showing and Hiding Powerplane Layers

Powerplane layers are indicated by a small white thermal pad icon to the right of the coloured square.

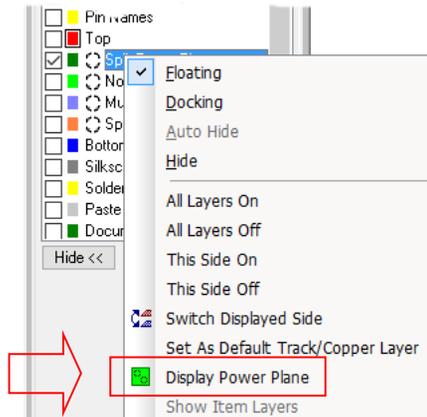


20 Pulsonix Version 9.1 Update Notes

If the powerplane is being displayed the icon is in negative. Left click on the icon to display or hide the powerplane.



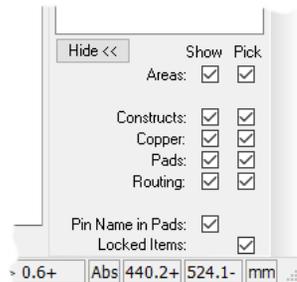
The functions to display, regenerate and hide the powerplane are also on the layers bar context menu.



Layers Bar in Footprint Editor

Footprint editor now has check boxes for fast display and picking changes to categories.

The **Pin Names** check box has been renamed to **Pin Name in Pads** to avoid confusion with pin name attributes.



Toggle All Hollow Command

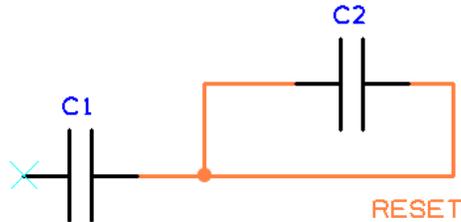
There is a new **Toggle All Hollow** command that can be assigned to a keyboard shortcut key and will toggle all hollow segment options on or off simultaneously.

To set this command to a shortcut key, go to **Tools** menu, **Customise** dialog, **Keyboard** page.

This command is in addition to existing commands for toggling the Hollow display of individual items (Shapes, Tracks and Pads). This function is to be used in conjunction with the True Width display on/off.

Net Name in Net Colour for Schematics

In the Schematic design, when a net is assigned its own colour, any net names displayed will also now be drawn automatically using the same colour.



New Fast Synchronise option

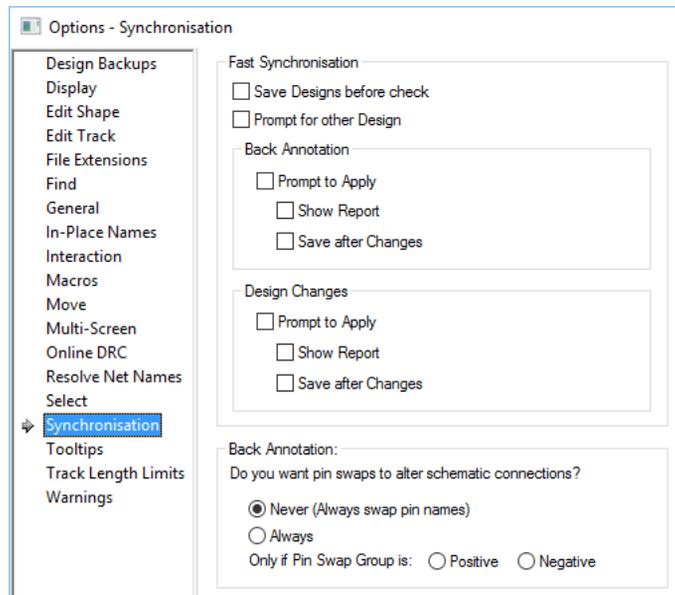
A new command is available for a **Fast Synchronise**. This does the same as **Synchronise Designs** but using pre-defined option switches. It is now possible to reduce synchronisation to a 'single click', with no further interaction.

This option is not on the menu though and can only be used once assigned to a command.

Changes to Options dialog

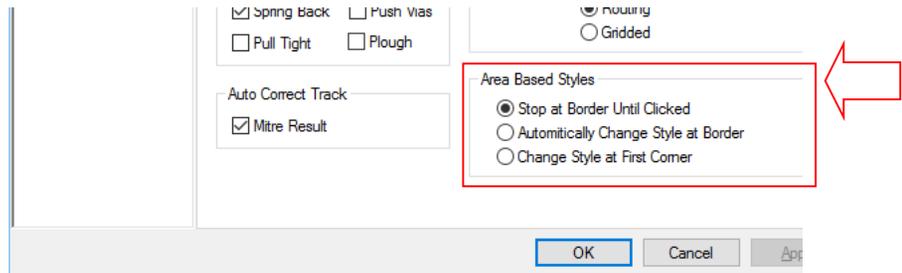
There are options available on the **Options** dialog, under the new **Synchronisation** tab which allow you to use the default options in most cases.

The **Fast Synchronise** feature also has an option to save designs before performing synchronisation if you wish this to happen.



Options for Area Track Style change

There are now options within the **Options – Edit Track** dialog to decide how **Area Based Styles** are applied. This determines what happens at the border of an area marked for **Area Based Styles** when a track style change is defined in the design technology **Net Styles** entries. These actions are applied with the **Online DRC** enabled.



You can choose one of three actions:

Stop At Border Until Clicked: Using this mode, the track addition stops and a corner is placed on the border. The style changes at that point once the mouse is clicked to continue.

Automatically Change Style At Border: By moving the track through the area, a corner is automatically placed on the border and the style changed to that of the Alternative style defined.

Change Style At First Corner: You can continue into the area and a style change is only made when you place a corner.

Minor size changes to Options dialog

Options dialog now resizable

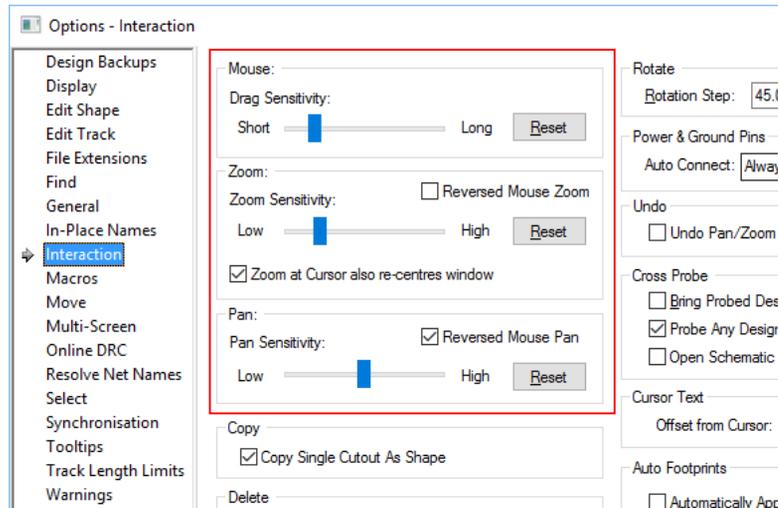
The **Options** dialog is now resizable using the grabber handle in the bottom right hand corner.

Display page reorganised

The **Display Options** page sections have been reorganised to reduce the overall dialog height.

Changes to General and Interactions pages

The **Mouse**, **Zoom** and **Pan** sections have been moved from the **General** page to **Interaction**. These changes are intended to rationalise the General dialog.



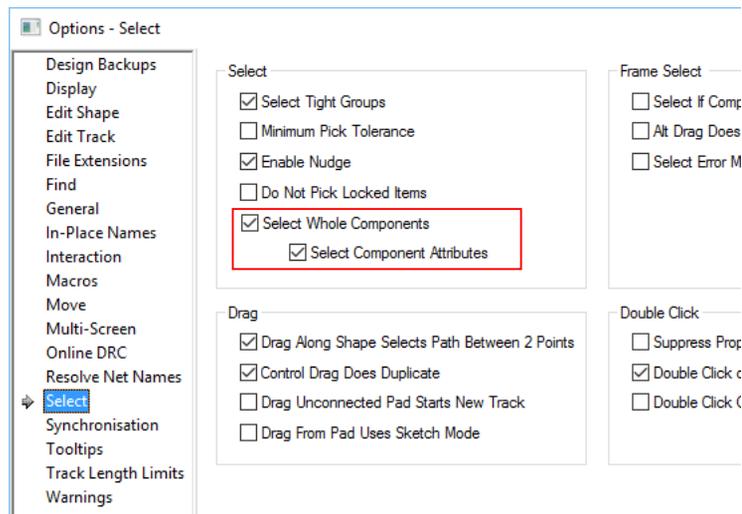
New categories added – Select and Tooltips

There are new option categories for **Select** and **Tooltips**.

Both sections have been removed from the **Interaction** page and placed into their new respective pages.

New Select Options

Within the **Options** and **Select** dialog, there is a new option **Select Whole Components**.



Checking this will mean selecting any part of a component except pads and attributes will select the whole component, subsequent shift select on the component part will change the selection to just be that part (this is how 9.0 schematic editing works). Unchecking the option means just the picked

component part will be selected, subsequent **Shift-select** will select the whole component (this is currently how 9.0 PCB editing works). The choice is now yours for both PCB and Schematic (defaults as per 9.0).

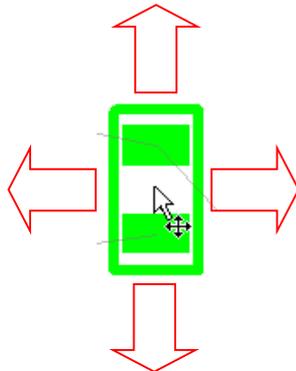
With **Select Whole Component** enabled, a further option is available. Check the **Select Component Attributes** option to allow selecting of component attributes (like in 9.0 Schematic editing). Uncheck it to have the whole component selected when picking attributes, subsequent **Shift-select** on the attribute will change the selection to just that attribute.

Back Annotation Changes for High Speed Design option

For the **High Speed** option, in version 9.0 a change was made so that you cannot change the name in **Safe Mode** on a **Differential Pair** or **Signal Paths**. In version 9.1, you can now change the names in safe mode. Any of these changes will now be back annotated from the PCB to the Schematic.

Snap Move to Orthogonal Only

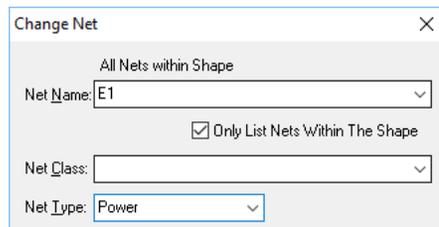
Whilst in move, holding the **Shift** key down will restrict movement to horizontal or vertical relative to the original items' position.



'Used' Nets within Templates

When doing **Add To Net**, **Change Net** or **Properties** on a **Template**, you can choose to only list nets which are 'used' within the template.

On selection of a **Template**, once in the **Change Net** option, select the **Only List Nets Within The Shape** check box.



Change Net

All Nets within Shape

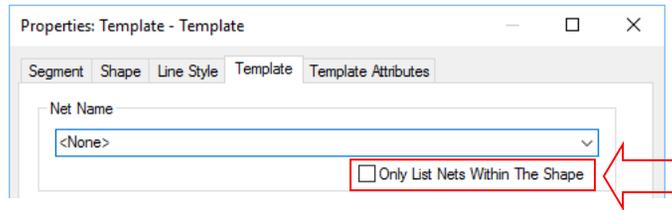
Net Name: E1

Only List Nets Within The Shape

Net Class:

Net Type: Power

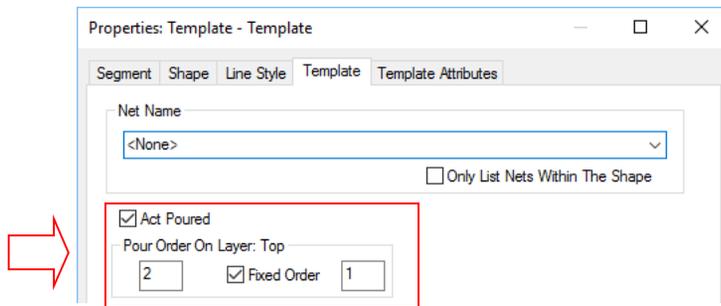
Within **Properties** of a **Template**, the Net Name selection can be filtered using the check box **Only List Nets Within The Shape**



Template Pour Order

Pour Order in Properties

By default, Templates will now be poured according to area size, (smallest to largest, without subtracting cutouts), the order is determined at the time of use. You can change this behaviour by marking some templates using a **Fixed Order** in **Template Properties**.



Templates which are marked as **Fixed** will be poured in the predetermined fixed order (with unfixed templates fitting in by area). You can use **Set Pour Order** from the **Utilities** menu (see below) to set the order in a dialog, **Ordered Template Pour** will allow you to set it interactively, and you can also change the order using Template Properties.

Older format files will have all existing templates marked at **Fixed Order**, so they will still pour in the same order.

With **Act Poured** check box selected, **Pour Order On Layer: XXX** shows the position in the order that this template will be poured in, when pouring all templates. (Obviously, this order makes no difference if you pour them all individually). The order is within the templates on the same layer, as there is no interference between templates on different layers.

You can make a template have a **Fixed Order**, and specify the position within all the fixed order templates on that layer. By default, templates are poured from smallest to largest by area (excluding cutouts); if there are templates with fixed order, these are poured in that fixed order, regardless of area. Use **Ordered Template Pour** to interactively view and define the order.

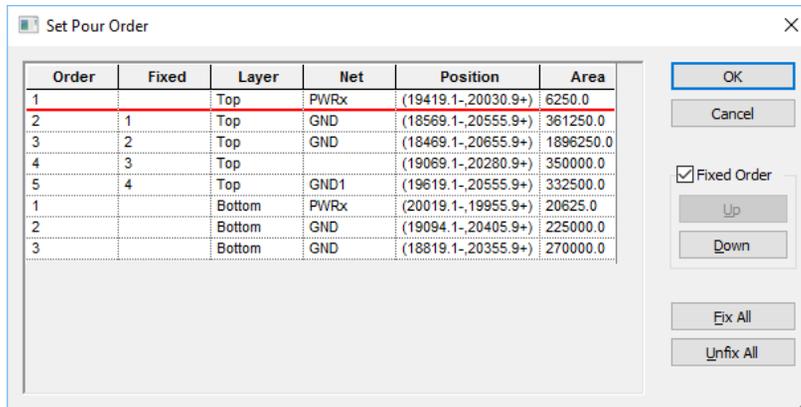
Note: Templates on Split Power Planes define the splits, they do not have an order and are not poured.

Settings and using Pour Order

By default, Templates are ordered by area (the area of the outline shape - without subtracting cutouts). This order is determined dynamically at the time of use. Changing the area of a template, or adding a new template may change the order. In many cases this is sufficient. However, you may wish to resolve certain conflicts and this is done by setting a **Fixed** order on some templates. This fixed order determines the order amongst the fixed order Templates on a given layer, those without fixed order will slot in to the order according to area. You therefore need at least two fixed order Templates on a layer before this has any effect on the order.

The setting of the Pour Order is achieved through the **Utilities** dialog and **Set Pour Order** option.

The dialog shows the Templates in your design (excluding Templates on powerplane layers which define a split plane and should not be poured). The currently selected Template is highlighted in the design. All other design items are then low-lighted (dimmed) to make the visibility of Templates more obvious.



The list is ordered firstly by Layer, then by the actual order the template will be poured in.

Each entry in the list shows the layer on which the template appears, and the net (if any) to which it is assigned. Also the position of the top left corner of the template bounds, and the area of the outline (without subtracting cutouts - this is the area used to determine the default order).

Use the **Fixed Order** check box on the dialog to fix or unfix the currently selected Template.

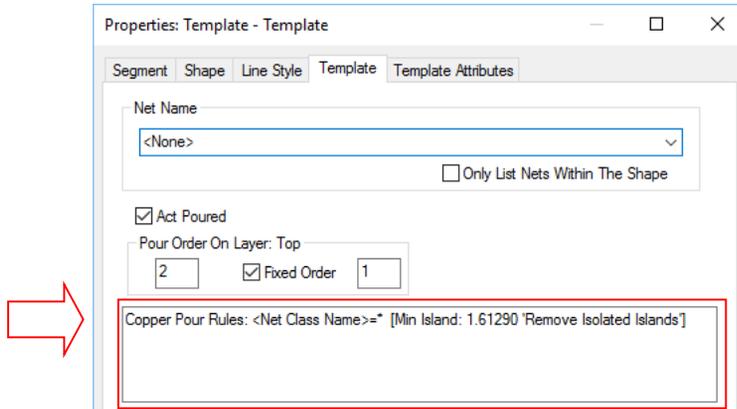
You can use the **Up** and **Down** buttons to move the currently selected **Fixed** template through the **Fixed** order. You will notice that the Template will move to its new position in the pour order.

Use **Fix All** to make all the Templates **Fixed** in the current order, and **Unfix All** to reset to the default order by size.

Template Properties shows Rule used

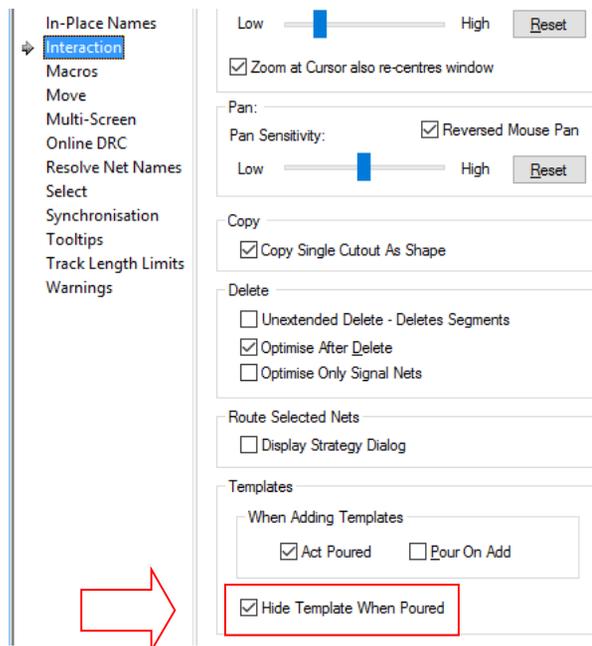
The **Template Properties** dialog now displays the rules associated with the selected Template. Rules can only be viewed here and not edited.

The rules displayed for the Template are: **Copper Pour Rules** and **Thermal Rules**.



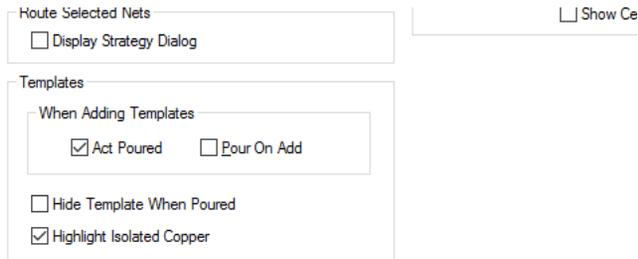
Option To Hide Poured Templates

A new option within the **Options** dialog, **Interaction** page enables **Templates** to be automatically hidden when poured. A Template will not be drawn if it contains poured copper but re-appears when the Template is cleared. Check the selection **Hide Template When Poured** to enable this option.



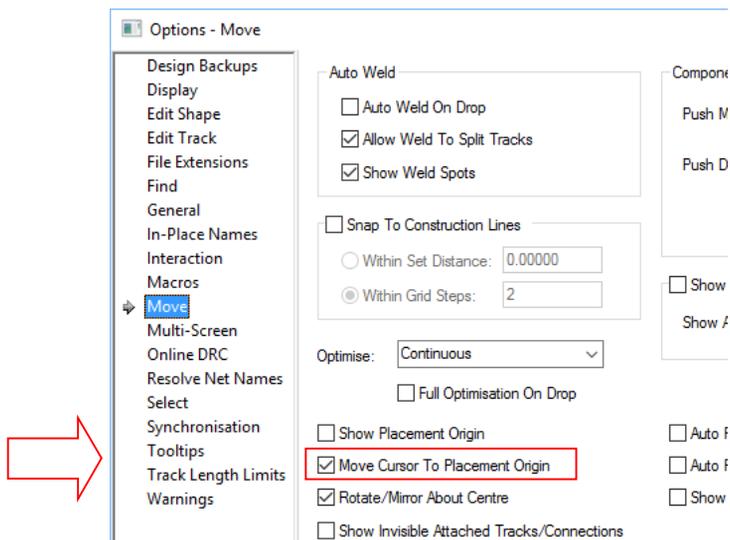
Options Highlight Isolated Copper

Within the **Options** dialog and **Interaction**, there is now an option (**Highlight Isolated Copper**) to allow you to choose if isolated copper is highlighted (previously, it was always isolated).



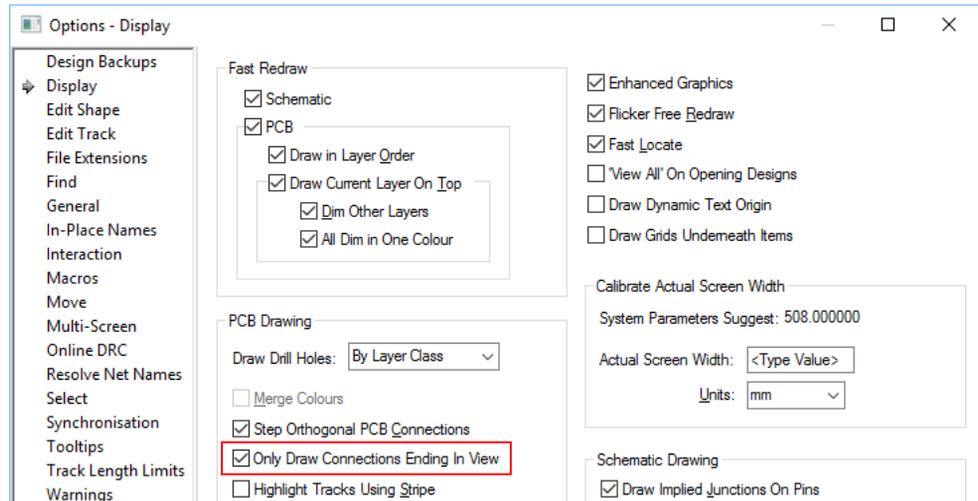
Options Move Cursor To Placement Origin

There is a new options in the **Options, Move** page called **Move Cursor To Placement Origin**. When enabled, as you move a selection the cursor will be initially moved to the original selection origin. This will only apply if the picked item in the selection has an origin, i.e. a Text item or uses a Symbol (e.g. Component, Signal Reference, Doc Symbol, Block Instance etc.) or if a temporary selection origin has been explicitly created for the move.



Options - Display

Within the **Options** dialog and **Display**, there is a new option **Only Draw Connections Ending In View**. This option declutters the screen by not drawing connections which just cross the view and don't connect to items within it.



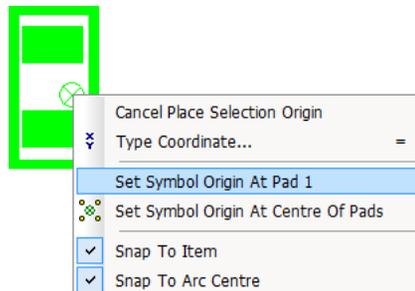
Selection Origin changes

Place Option Origin option renamed

The **Place Option Origin** has been renamed to **Place Selection Origin** on the context menu.

Options for select mode

There are new options available on the context menu when in **Select** mode.



This allows you to place an origin, snapped to any item that will be used as the origin of the selected item when Moving, Rotating and Mirroring it.

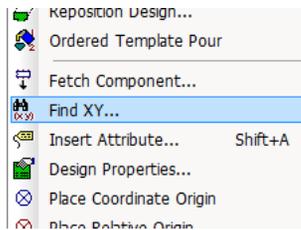
Whilst placing the origin you can use the context menu to place the **Symbol Origin** on the selected component **Pin 1**, or at the **Centre of the Pads**. If there is no pin named "1", it will use the first pin on the symbol, for example A1.

This origin is temporary, it will remain whilst you use the **Ctrl** key to add and remove items from the selection, but will be removed when all are deselected or a new selection is made.

The selection origin is remembered when copying a selection to the clipboard, and restored on paste of that data. This gives another more simplistic method of adding components in a circle by selecting the item or point you want as the centre of the circle as the selection origin.

Find XY

The **Find XY** option is now available on the shortcut menu when nothing is selected in the design. This has been added to aid the precise positioning of your cursor in the design if required.

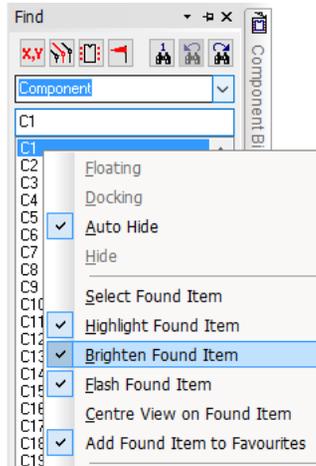


Find Bar Changes

New Find Bar Options

There are new options **Brighten Found Item** and **Highlight Found Item** on the context menu in the **Find bar** and in the **Options** dialog under **Find** page (Actions On Found Item). Previously, you could see found items by having them selected or highlighted (if **Select Found Item** was not enabled). Now you can choose how to see them by combining the four options, Select, Highlight, Brighten and Flash.

When using **Brighten Found Item**, the items are **brightened** by uniformly greying out all other items. The design remains greyed out until you click or drag in select mode within the design, or exit select mode for another operation. While in this mode, it enables you to zoom in and out, and pan around to view the item better, whilst the design is greyed out.



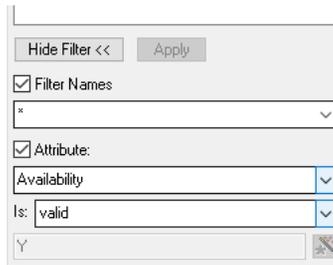
New Find Bar Categories

New **Find Associated Part** category. This command finds components that have an **Associated Part**.

New **Find Net Class** category. This command finds nets that have a Net Class defined.

The **View Selection** command is now on the context menu.

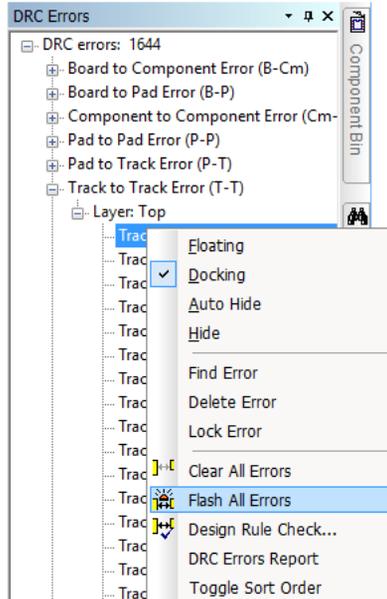
Within the attribute filter are fields for **Valid** and **Not Valid** for selecting whether an attribute matches its validation setting in the **Technology** dialog and **Attributes Validation** field. When using this, the validation value is displayed as a greyed out value on the dialog.



DRC Errors Bar

The **DRC Errors bar** now uses all find parameters (select, highlight, brighten & flash) when double-clicking on an error in the bar.

The context menu function **Flash Errors** has been renamed to **Flash All Errors**.



Insert Connection – Type Pin Name improvements (PCB)

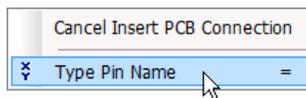
In PCB, when using the **Insert Connection** option and using the shortcut menu **Type Pin Name** to add connections by Component/Pin Number, a number of improvements have been made:

- If you type a pin name that is off the screen the view will be panned to include it.
- The **Type Pin Name** dialog title says if Start Node or End Node required.
- There is a new **Apply** button. This does same as pressing the <Space> bar; where it adds a connection to a pin but keeps the dialog ready for the next pin on the net.
- New **Finish** button. This does same as pressing “=”; where it ends the connection on a pin but keeps the dialog ready for a start pin for a new connection.
- The status bar now shows what to do when using **Apply**, **Finish**, the Space bar and “=”.

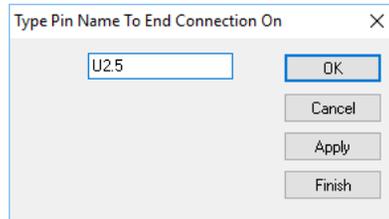
Functionality Recap

This option is existing functionality but to refresh its use for users unfamiliar with it:

When adding a connection using the **Insert Connection** option, you can right click to reveal the **Type Pin Name** option. As you will see, the shortcut key, = has been setup by default for its use.



When selected, this will display a small dialog box in which to type the connection list, for example C1.2 then <Enter> (for **OK**).

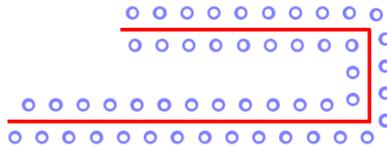


The second connection point would then be added using **Type Pin Name** again (use the shortcut = to access the menu).

Alternatively, after entering the first pin name, instead of pressing **OK**, press the **<Space>** key and type the second pin name. Press the **Finish** button to stop or keep pressing the **<Space>** bar to add more connection points to the net.

Apply Vias in Footprints

As with PCB designs, you can now **Apply Vias** to shapes in Footprints.



Changes to Apply Vias in PCB

The **Apply Vias** options (on the **Utilities** menu) has now been changed to allow vias to be applied to **non-electrical shapes** as well as the existing shapes allowed.

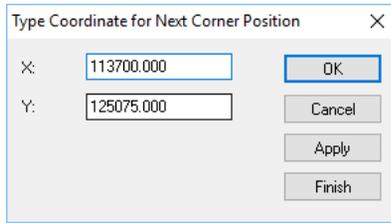
Enhancements to the Add Polygon option

Some small improvements have been made when adding **shape polygons** to a design:

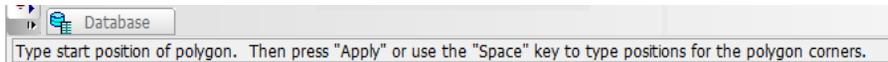
- If you type a start position the view will be panned to include the start point if it was off the screen.
- If you type a corner position or offset that is off the screen, the view will be changed to include the whole shape.
- A shape origin is drawn at the start position.



- **Type Offset** is now available before the start point is created. It will add a start point as coordinates, like **Type Coordinate**, but then differs by entering offsets for subsequent corners.
- The **Type Coordinate** and **Type Offset** dialogs have been changed when adding shapes as follows:
 - Title says if **Start Position** required or **Next Corner** position.

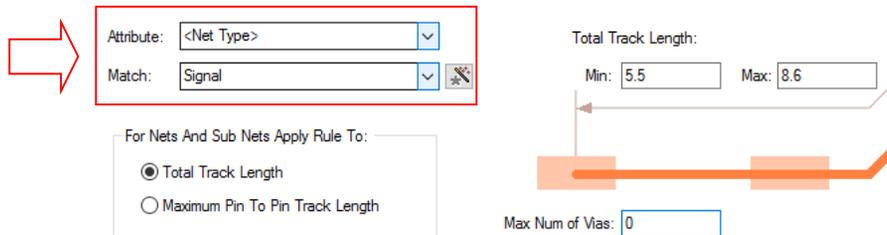


- A new **Apply** button. This does the same as pressing the <Space> bar; it adds a corner but keeps the dialog displayed ready for the next corner.
- A new **Finish** button. This does same as pressing “=”; it adds a corner and then finishes the shape.
- The **Status** bar now mentions using **Apply** and **Finish** and <Space> bar and “=” during use.



Net Type Attribute

You can now use the new net attribute **<Net Type>** on a rule to apply the rule to all nets of that type.

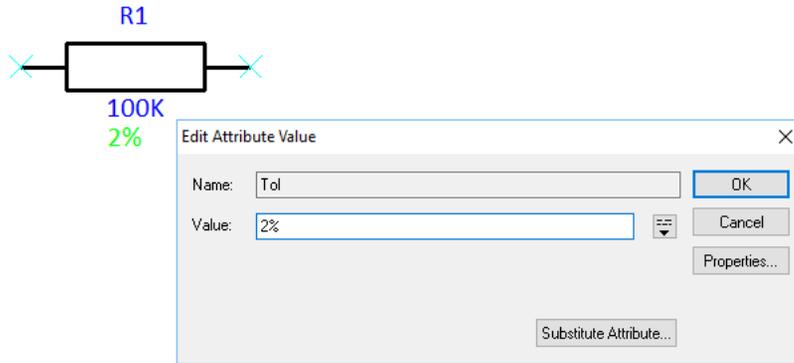


NB <Net type> is used on a net to describe its type. Three types are available, Power, Ground and Signal.

This feature was also back fitted to version 9.0.

Edit Value on double-click

On selection of an attribute that can be edited, a new simplified dialog is available on the context menu. Use the **Edit Value** option to display this option. This allows you to quickly change an attribute value of a displayed editable attribute.



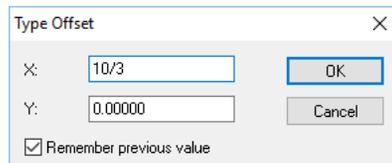
By default this option is assigned to double-click to make editing easier. The default action can be changed using **Options** and the **Select** page, **Double Click on Attribute Edit Value**. This can be disabled and have it go into **Properties** instead. If the attribute is not directly editable (like a Part Name for example), double-click will enter the **Properties** dialog instead.



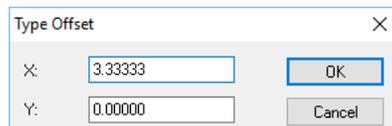
Input values can now be expressions

When typing in a value, such as in **Type Offset**, or any dimension, you can now use simple expressions with + - * / and brackets. For example, (11/2)+(4.6*5.1)

For example, typing 10/3 will give you 3.3333

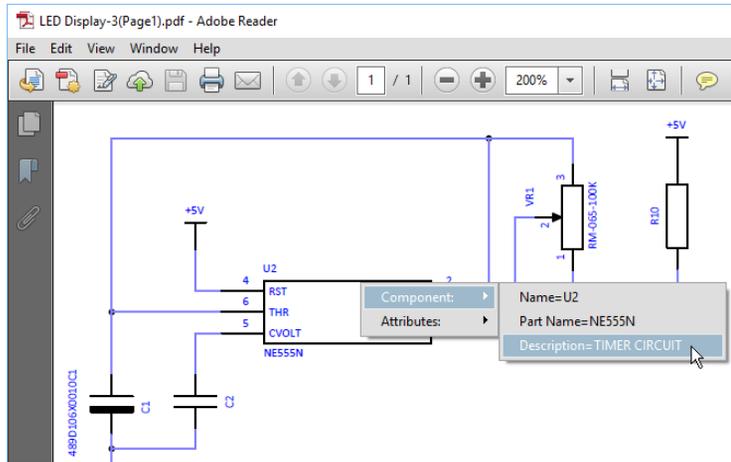


The remembered value will be this:



Changes to PDF Output

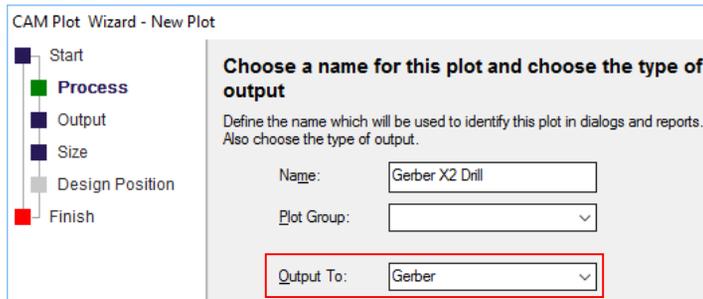
When generating PDF plots from the CAM Plot dialog, the PDF plot now shows the Part description in the pop-up window when hovering over a Component.



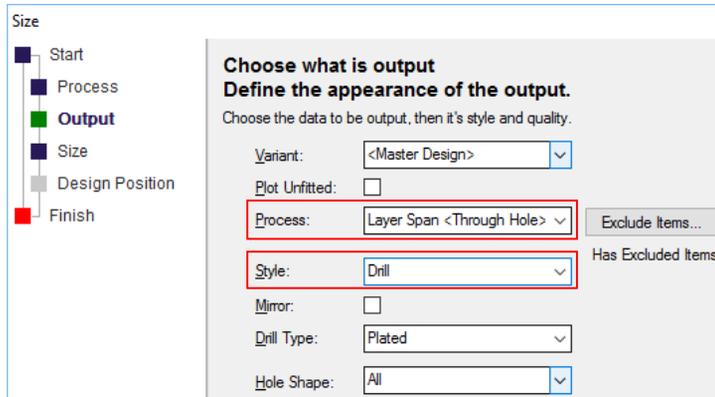
Output Gerber X2 to Drill format

As part of the Gerber X2 improvements, you can now output drill data in Gerber by selecting a layer span and the plot style **Drill** (as opposed to **Drill Drawing**).

Using the **CAM Plot** dialog, for a **New** plot, with the output format set to **Gerber**:



The **Output** page of the Wizard allows you to choose a **Layer Span** as the **Process**. With the Layer Span set, the **Style** can be now set to **Drill**.



Changes to Scripting

Load configuration file

A new function in the Application object which loads the specified configuration file. If running dual screen then loaded configuration is applied to both instances of the application.

```
call Application.LoadConfiguration("c:\temp\psx.cfn")
```

Rename items in library

Ability to rename existing library items (parts, footprints and symbols). This is a new function available in a PartLibrary or SymbolLibrary object. Note the second parameter (ReadOnly) of the function call to open the library must be False to allow the rename to work.

```
set PartLib = OpenPartLibrary("User.pal",False)
if Not PartLib Is Nothing then
    if PartLib.Rename("New21", "ATX4175PL") then
        ' it worked!"
    end if
end if

set SymLib = OpenSymbolLibrary("User.pfl", False)
if Not SymLib Is Nothing then
    if SymLib.Rename("SOT203-proto", "SOT203-M") then
        ' it worked!"
    end if
end if
```

Run script in other instance

This command will run the named script in the other application instance when running in dual screen mode. The second parameter determines whether or not to wait until the script is finished in the other instance before continuing.

```
If RunScriptOtherInstance("C:\ Pulsonix\Scripts\check.vbs", true) then
    ' do something here
end if
```

Set CAM/Plot output template

This application function allows direct setting of the output file name 'template' as shown on the CAM/Plot dialog **Settings** page.

```
strPath = "$ (DesignName)$(PRE)-$(Variant)$(PRE)$(PlotName)$(POST)"
```

```
if Application.SetPlotOutputTemplate(strPath) then
    ...
end if
```

Generate STEP output

This new function on a Document object will (for a PCB design) generate a STEP output file without invoking the STEP dialog or other user interface. The settings for the dialog are supplied to this function as an XML file as described below instead of passing a long list of parameters in the function call, which also allows flexibility in adding settings in future releases without changing the functional interface.

```
Set Design = Application.ActiveDocument
if Design.IsPcb then
    if Design.GenerateSTEP("c:\test\test.tgz", "c:\test\STEP.xml") then
        end if
    end if
end if
```

The names of the XML fields should be easy to understand as they correspond closely to the entries you can see on the Output dialog in the application. Any settings not provided in the XML file will use the current default values as would appear on that dialog. Boolean settings are 1 = true, 0 = false.

BoardThickness	numeric
MountingDrillHoles	boolean
NoComponents	boolean
OutputPads	boolean
OutputVias	boolean
OutputDrillHoles	boolean
OutputTracks	boolean
OutputCopper	boolean
OutputDocShapes	boolean
OutputText	boolean
SolidWorks	boolean

For example:

```
<MountingDrillHoles>1</MountingDrillHoles>
<OutputPads>1</OutputPads>
```

Get extents of Board outlines

This new function on a Document object will (for a PCB design) return a rectangle object which defines the total bounding box of all Board Outline items in the design.

```
Set Design = ActiveDocument
Set Ext = Design.GetBoardExtents
```

Generate ODB++ output

This new function on a Document object will (for a PCB design) generate an ODB++ output without invoking the ODB++ Output dialog or other user interface. The settings for the dialog are supplied to this function as an XML file as described below.

```
Set Design = Application.ActiveDocument
```

```

if Design.IsPcb then
    if Design.GenerateODB("c:\test\test.cvg", "c:\test\ODB.xml") then
        end if
    end if
end if

```

The names of the XML fields should be easy to understand as they correspond closely to the entries you can see on the Output dialog in the application. Any settings not provided in the XML file will use the current default values as would appear on that dialog.

Compressed	boolean
UseCamPlot	boolean
IncludeWindows	boolean
SilkLayerClass	string
MaskLayerClass	string
PasteLayerClass	string
OutputPositive	boolean
OutputComposite	boolean
NeutralNetNames	boolean
OutputCompDoc	boolean
PinNamesInPackages	boolean
OutputPanelPCB	boolean

Generate IPC2581 output

This new function on a Document object will (for a PCB design) generate an IPC-2581 output without invoking the IPC2581 Output dialog or other user interface. The settings for the dialog are supplied to this function as an XML file as described below.

```

Set Design = Application.ActiveDocument
if Design.IsPcb then
    if Design.GenerateIPC2581("c:\test\test.cvg", "c:\test\2581.xml")
    then
        end if
    end if
end if

```

The names of the XML fields should be easy to understand as they correspond closely to the entries you can see on the Output dialog in the application. Any settings not provided in the XML file will use the current default values as would appear on that dialog.

UseCamPlot	boolean
IncludeWindows	boolean
FunctionalMode	string *
FunctionalLevel	numeric
NeutralNetNames	boolean
SilkLayerClass	string
MaskLayerClass	string
PasteLayerClass	string
AssemblyLayerClass	string
Units	string @

* FunctionalMode is one of FULL, DESIGN, FABRICATION, ASSEMBLY, TEST
 @ Units is one of the available Unit names (e.g. cm, mm, mil, etc.)

Auto-generate plots

This new function on a Document object will invoke the CAM/Plot 'auto generate plots' without displaying any user interface. The settings for the dialog are supplied to this function as an XML file as described below.

```
Set Design = Application.ActiveDocument
if Not Design Is Nothing then
    Call Design.AutoGenPlot("c:\test\autogen_scm_plots.xml")
end if
```

The names of the XML fields should be easy to understand as they correspond closely to the entries you can see on the Output dialog in the application. Any settings not provided in the XML file will use the current default values as would appear on that dialog.

XML settings (PCB):

DeleteExistingPlots	boolean
ForEachVariant	boolean
DeviceNameElec	string
DeviceNameNonElec	string
DeviceNameDoc	string
DeviceNameDrill	string
DeviceNameDrillDrawing	string

XML settings (SCM):

DeleteExistingPlots	boolean
ForEachVariant	Boolean
DeviceName	string

New 'InputBox' command

There is a new 'InputBox' command to display a prompt dialog to get user input. This command takes 3 parameters; a caption for the dialog, a prompt message, and a default value for the text.

```
Result = Application.InputBox("Version", "Enter version number", "1")
```

Changes to Technology Files

There have been many changes to the standard PCB Technology files supplied. The changes are small in detail to improve your default starting position if using them. Their names have also been slightly modified to more reflect their purpose and to make them more identifiable.

Please be aware of these changes if you use the default supplied Technology files as your old Technology files may still exist on your system.