

Pulsonix Design System V7.5 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2012 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: 24/01/12 iss 2

Pulsonix

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

Support Phone	+44 (0)1684 296 570
Fax	+44 (0)1684 296 515
Email	info@pulsonix.com
Web	www.pulsonix.com

Contents

CONTENTS	3
CHAPTER 1. VERSION 7.5 UPDATE SUPPLEMENT	5
Installing the New Version of Pulsonix	5
Licensing	
Windows 7 Support	
Report Maker Enhancements.	6
Snap Cursor to Grid	
Multiple File Extensions	
CAM Plot Changes	25
Enable a Named Group of Plots from Context menu	
Move Additional Plot Items In CAM Plot Option	
Copy Plot	
Crop Plot Output	
PDF Output Bookmarks	
PDF Output Variants	
Gerber Verification Plot to PDF	
Associated Parts – Execute Hyperlink	
Local Pin Types	
Component Variant Spreadsheet	
Delete Variant Changes	
Attribute Editor shows Variant ID	
Swap Component Positions	
Part Edit Changes	
Footprint Dialog Filter	
Ability to Resize Footprint/Symbol Dialogs	
Change Connector Part Type option	
Delete Segment on Unextended Delete	
Net Class - Mark All Nets As One	
Replace Part Changes	
Insert Multiple Item Changes	
Auto-Rename in Schematic Editor	41
Schematic Bus can have Own Colour	41
Show Net Name on Change Net	42
Pad Style Spacing Shapes	
Changes to 3D Viewer	46
New 3D Package Shapes	46
Axial Components - Rounded Ends	46
Colour-coded Bands on Components	
3D View - Print With White Background	
3D View - Save As Image	
Layer Type Choice for Component Body	
Find Bar – Drill Sizes in Table Units or Design Units	
Spacing Wildcard Match	
Wildcard Strings.	
Spacing Rule Class to Class Match	
Copy Pad Style	
Suppress Pads on Powerplanes	

Highlight Locked Tracks	51
Draw Pads in Top Most Layer Colour	51
Net Styles Dialog Changes	
Via Protection Against Automatic Changes - Default Settings	53
Add Track Option – Use Style From Picked Track	53
New Context options in Layers Bar	54
New Commands for Change Layer	54
Component Pushing uses Accurate Shapes	55
IDF Output Enhanced	55
DXF Import Enhanced	57
Scale Import Plot	57
Align DXF 0,0 to Origin	57
Apply to Entire Column	57
Copy Shape Layer Map to Text Grid	58
STEP I/O Improvements	58
Design Rules Check – Check for Missing Teardrops	60
Online DRC – Check Via in Pad	60
Online DRC – Check Comp to Comp	60
High Speed Option - Differential Pairs - Spurs Allowed Check	61
Changes to Calculators	62
Ariadne Import (Cost Option)	62

Chapter 1. Version 7.5 Update Supplement

Installing the New Version of Pulsonix

It is 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, insert the CD or double-click on the download executable and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 7.5 on top of your existing installation or along side if you prefer; however, you do not need to uninstall the old version first.

Licensing

Version 7.5 uses a new license. This will be supplied to you by email under the contract conditions of your current maintenance agreement.

If you are using a version of Pulsonix earlier than Version 7.x, you will require a new license.

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.

Windows 7 Support

As further confirmation, since the release of Pulsonix 6.1 (and for later versions) the release version of the Windows 7 operating system in both 32 and 64 bit variants is fully supported. No earlier Pulsonix versions can or will be supported. Also, Beta or pre-release versions of Windows 7 cannot be supported.

Report Maker Enhancements

New commands – Is Blank and Is Not Blank

When using an **If** Command, it can now have the test **Is Blank** and **Is Not Blank**. These only work on text fields. These commands also exist when using the **While** command on a text variable. They can be used to test for empty fields.

List of Components List of Attributes If "Attribute" is not blank
Edit If Command
Command: If "Attribute" is not not blank
If: Attribute Attribute:
Use: OX Coord OY Coord O Value O Name O Inherited Value
Test the Value field length
Is: not blank

New command - Is All SM Pads

The new Is All SM Pads command is used on the List Of Components command to test if all pads on the component are surface mounted, it reports True or False. It can also be used within an If "Is All SM Pads" command.

 List	of Components
	Is All SM Pads
	Component Name

The new Is SM Device command is similar to the new Is All SM Pads command but looks at Mounting Holes as well.

 List c	of Components
	Is SM Device
	Component Name

The new **Is SM Pad** command can be used on a single component pad to test if it is surface mount or not. This would be used within a **List of Pins** command or **If "Is SM Pad"** command to test the pad condition.



List of Component Groups by side

You can now sort **List of Component Groups** by **Side**. The **Side** command appears under the **Sort By:** drop down list on the **Edit Sort Criteria** page. If you want to do this, ensure that Side is also part of the Component Group definition. This way, all the components on the top side will be reported separately to all the components on the Bottom side.

Edit List Co	ommand	
Command:	List of Component Groups	Edit List Command Sort Criteria
	Sorted List Edit Sort Criteria	Command: List of Component Groups
- Componer Compo	nt Group Definition: onents are in the same group if the following parame	Side Attribute: Attribute: Image: Organization Attribute: Image: Organization

Ability to pass variables from one report to another

New commands allow common report format files to be shared or run as sub-routines.

To do this, the **Run Report** command dialog now has 8 optional parameters (supplied either by variable or fixed value) that can be passed to the report being run. There is also an optional variable to contain a return value from the run report (**Place Result in Variable**).

iort Nam	e: compsBySid	e			 ~ <		
	Write Rej	port To S	eparat	e File			
arameter	rs Passed To Rep	port:				_	
V 1.	Fixed Value	~	top				$ \langle $
2.	Fixed Value	~					N
3.	Fixed Value	~					
4.	Fixed Value	~					
5.	Fixed Value	~					
6.	Fixed Value	~					
7.	Fixed Value	~					
8.	Fixed Value	~			 		

An example Report Maker script below running this (*compBothSides*) demonstrates how this command may be used:

 Set Variable "topCount" to 0
 Set Variable "bottomCount" to 0
 Run Report "compsBySide" with parameters "top" result in "Variable: topCount" <
 Blank Line
 Run Report "compsBySide" with parameters "bottom" result in "Variable: bottomCount"
 Blank Line
 Variable "topCount"
 Variable "bottomCount"

In the report being run (*compBySide* in our example above), the parameters can be picked up by setting a Variable to their value. e.g. **Set Variable 'side' to Runtime Parameter 1**, and you can save the value of a Variable to the report result for passing back to the parent report. For example, **Save Variable 'count' to the report result**.

The example script being called from the first script (*compBySide*) demonstrates how these new commands may be used:

Variable Name:	side		~			
Variable Tupe:			inn I an abh	F -l		
valiable rype.		Action Details	gritengin 🔾 Hu	e ur raise		
Action:		Action Details.				
• Set Value	0:	Fixed Value:				
O Append Va	Iue With:	Field Contents:	Attribute			~
O Multiplu Va	value by:		Attribute:			
O Divide Val	ie Bir		Use: O X Coo	ord O Y Coord		
Ask User F	or Value:		🔿 Value			
O Extract Fiel	d From:					
O Replace T	ext In Value:	🔽 Runtime Param	eter: 1			
O Save To R	eport Result					
🔿 Show Valu	e In Message Box					
🔿 Write Value	e To Report:					
Set'	/ariable "count" to /ariable: side" to R (ariable: side" is m Prompt for Variabl (ariable: side" is m Set Variable "side	0 Runtime Parameter atching '''' le "side" Type the atching '''' atching '''' '' to all	1 required side (to	p, bottom or all):	
Set' Set' If ''\ If ''\ If ''\ If ''\ Else	/ariable "count" to /ariable: side" to R (ariable: side" is m Prompt for Variabl (ariable: side" is m Set Variable "side (ariable: side" is m Text "All compon	0 Runtime Parameter atching '''' atching '''' '' to all atching ''all'' ents''	1 required side (to	p, bottom or all);	
Set's Set's If ''s If ''s If ''s Else Und	/ariable "count" to /ariable: side" to R (ariable: side" is m Prompt for Variabl (ariable: side" is m Set Variable "side" (ariable: side" is m Text "All compon Variable "side" erline "."	0 tuntime Parameter atching ''' etching ''' '' to all atching ''all'' ents''	1 required side (to	p, bottom or all):	
Set' Set' If ''\ If ''\ Else Und /// List	/ariable "count" to /ariable: side" to R /ariable: side" is m Prompt for Variabl /ariable: side" is m Set Variable "side" /ariable: side" is m Text "All compon Variable "side" erline "-" of Components Set Variable: side" If "Variable: side" Set V	0 Runtime Parameter atching ''' le ''side'' Type the atching '''' atching ''all'' atching ''all'' ents'' htComponent'' to tr '' is not matching 'Va anable ''wantComp	I required side (to ue "all" ariable: side" nonent" to false	p, bottom or all):	

If the script above is run stand alone, i.e. does not have the Variable Side passed in from a prompt report it prompts the user for a a side value. This new command is documented further down in the Report Maker changes section *User Input Prompted with Drop Down List*.



The **Save To Report Result** selection within the **Variable** command is used to save the current value of this variable as the report result. This is only relevant if the report has been run by a parent report using the **Run Report** command. The parent report can then retrieve this result and place it into its own result variable to use in its report.

Report pin to pin track lengths

E

The ability to report pin to pin track lengths has been added. For a net you can now report or place into a variable the **Maximum Pin to Pin Length** and **Minimum Pin To Pin Length** value from its net class.

Eist	of Nets
	Net Name
	Minimum Pin To Pin Length
	Maximum Pin To Pin Length

You can now report a list of the component pin pairs for a net using the **List of Pin Pairs** command. Each unique pair of pins will be reported.

List of Nets	•
List c	f Pin Pairs
	Net Name
	Start Node Node Name
	End Node Node Name
	Distance
	Track Length
	Minimum Pin To Pin Length
	Maximum Pin To Pin Length
	Chain Links Count

The **Chain Links Count** for a **Pin Pair** is the number of pads you pass through to get from one to the other, plus 1. This enables you to test for the shortest path between pins and report it.

In the example report below, the Chain Link Count is tested to see if it is equal to 1, this means there is a direct link between the pins. If it does (in the design) then it sets two variables for the Max and Min Pin to Pin Length. With this set, the first variable (Min Pin to Pin Length) is checked to be greater than 0. The Track Length for that net is then checked to be less than the variable just set and if it is (equal to 0), then it reports the Net Name and start Node Node Name etc. It then spins through the Max Pin to Pin length to run the same check.



When the **Maximum Pin To Pin Length** and **Minimum Pin To Pin Length** commands are used on pin pairs they take into account the net class subgroups defined within the net class by pin attribute. By comparing these values against the pin to pin track lengths you can report length fails.

The Pin to Pin Connection rule defines the Maximum Track Length and Maximum Number of Vias in the shortest path between each component pin and the nearest other pin in the net.

An alternative method to check the Max and Min Pin To Pin Lengths is by running the script below. This sets up three variables for Max and Min Pin to Pin length, and a rule check for whether the rule is set (havePinToPinRules). It then checks to see if the rule is not equal to 0, if it isn't then it sets the havePinToPinRules to True. If this is true, the List of Pin Pairs command is run. Each rule check (Max Pin to Pin and Min Pin to Pin) is run after checking the variable maxpin2pin and minpin2pin.



When run, the report looks like this:

```
net: HSE1
A1.2 U13.1 len: 5.652 min: 8.000
net: HSE2
A1.1 U13.2 len: 5.828 min: 8.000
```

You can also use the **Is Selected** on a pin pair to only report the track to track lengths on selected pins in the design.



If Differential Pair

There is now a **Has Differential Pairs** command that can be used in a design (at the top level) or within a **List of Nets** command.

List of Nets
 Net Name
 Has Differential Pairs

This example might be reported like this with a true or false status:

```
$11 false
$12 false
+5V false
GND false
HSE1 true
HSE2 true
NPRESET false
OUTPUT false
```

Has Differential Pairs can also be used within If to test if Nets have Differential Pairs. It can also be can be used within a List of Pin Pairs command.



This example might be reported like this:

Net=HSE1 Net_Class=HSE U13.1 U13.9 Net=HSE2 Net Class=HSE U13.2 U13.8

Report Maker List of Differential Pairs

You can now output using the **List of Differential Pairs** command for PCB and Schematic designs. Within this, you can also do a **List Of Pin Pairs** to report each of the two pin pairs in each differential pair, and can report the paired length rules and actual paired lengths using additional commands.



Ability to replace substrings of text items

The ability to replace substrings of text items has been added, e.g. to replace spaces in part names (which old pick-and-place machines don't like) with another character before they are reported.

O Extract Field From: Image: Construction of the state o	Ask user für value.	
• Replace Text In Value: • Save To Report Result • Show Value In Message Box • Write Value To Report: • Write Value To Report: • Vite Value T	O Extract Field From:	
O Save To Report Result With Text: O Show Value In Message Box ✓ Replace All Occurrences O Write Value To Report: ✓	Replace Text In Value:	Replace Text: Whitespace Characters
O Show Value In Message Box ○ Write Value To Report:	O Save To Report Result	With Text:
O Write Value To Report:	O Show Value In Message Box	Replace All Occurrences
	◯ Write Value To Report:	

For a variable command, there is a new **Replace Text In Value** action. Type in the text sequence to be replaced, or check the **Whitespace Characters** box to easily replace all spaces, tabs and newline characters. You can then provide the text sequence to replace them with, or leave this blank to simply remove them. If the **Replace All Occurrences** box is left unchecked, only the first occurrence of the text will be replaced.

Ability to insert information into reports from external file data

The ability to insert information into reports from external file data has been added. To do this a new **List of ASCII File Lines** command has been created. This command can have the file name provided by a fixed text string, or by the contents of a text variable at run time. This file name may contain a relative path, in which case it will be applied relative to the current design folder or library folder when run.

List of Asc	äiFile Lines from file: "C:∖Project∖Data.txt" ⊨Text		
Edit Ascii File Line	es List Command	×	
Command: List of Ascii File Lines from file: "C:\Project\Data.txt"			
Ascii File To Report Lines From			
Fixed Name:	C:\Project\Data.txt Browse)	
🔘 Use Variable:			
Note: When the re	port is run, any relative path provided will be applied to the current design folder or library folder		

Each ASCII file line can then be reported or saved to a variable using the Line Text command.

This command could be used in an inserted report to display the current contents of an ASCII file in the design. The file path could be provided by an attribute of the inserted report symbol.



This feature can also be used along with the new feature **Extract a String into sub-strings** (documented below). This is used to insert a table from an excel spreadsheet into your design. See the example below.

Ability to extract a string into sub-strings

The ability to extract a string into sub-strings and loop with a variable set to each sub string has been added. For example the first part of the report may build up a comma separated list of fields in a variable based on some criteria. Then the second part of the report could extract each field in turn and report it.

To do this, add a **variable** command and set its action to **Extract Field From**. Choose the variable the field is to be extracted from and the separator character (or string) that defines the end of the field. When this command is run, the variable will be set to the field text, and the variable it is extracting from will have the field (and its separator) removed from it.

The example below uses this feature to insert a CSV file into your design as a table.



Edit Variable Command			
Command: Set Variable "Field" to field extracted from "Variable: Line"			
Variable Name: Field	▼	Cancel	
Variable Type: 🔘 Integer 💿 Text	Number O Design Length O True or False		
O Ask User For Value:			
 Extract Field From: 	Extract Field from start of Variable: Line		
O Replace Text In Value:	Field Separator:		
O Save To Report Result			

Ability to show a simple message box

The ability to show a simple message box has been added. For a **variable**, there is a new action **Show Value in Message Box**. When the report is run this command will display the current contents of the variable in a message box to the user.

Set Variable "var" to Hello World	
Show Variable Var in a message box	
Edit Variable Command	X
Command: Show Variable "var" in a message box	ОК
Variable Name: var	Cancel
Variable Tune: O Integer III Tevt O Number O Design Length O True or False	
O Save To Report Result	
⊙ Show Value In Message Box	
Write Value To Report	

The report output continues when you press the **OK** button. In our example, this would display like this:



User Input Prompted with Drop Down List

You can now offer a drop down list of possible values when prompting for user input for the report. For example, a list of variant names in the design. Offering a prompt is achieved using a **variable** command with action set to **Ask User For Value** like before, but now you can check a box to **Show List of Values of Type** and choose a type of design item to present in a list when the prompt command is run. Names will be extracted at runtime from the design (if there is a design and it is of the correct type).

Edit Variable Command			
Command: Set Variable '''' to	ОК		
Variable Name:			
Variable Type: 🔘 Integer 💿 Text	Number O Design Length O True or False		
Action:	Action Details:		
◯ Set Value To:			
O Append Value With:			
O Decrement Value By:			
O Multiply Value By:	Prompt: Choose from Variant list:		
O Divide Value By:	Show List of Values of Type: Variant		
⊙ Ask User For Value:	Allow Users Own Value		
O Extract Field From:	Extract Value List from Variable:		
O Replace Text In Value:	Field Separator: Field can be Quoted		
🔿 Save To Report Result			

There are two special value types.

Prompt: Choose from Variant list:		
		1
Show List of Values of Type:	Variant 🗸	Edit Value List
🖌 Allow Users Own Value	<user list="" specified=""> <variable list="" specified=""></variable></user>	
Extract Value List from Variable:	Attribute Component	~
Field Separator:	Layer Layer Layer Span	uoted
	Net Page	
	Part Variant	

Choose **<User Specified List>** and use the **Edit Value List** button to add your own list of acceptable values. Or choose **<Variable Specified List>** to extract a list of acceptable values at runtime from another variable (using a specified **Field Separator** between acceptable values). This is useful to present a list of items that satisfy a certain criteria, e.g. all components on the top side

Multiply Value By:	Prompt: Choose from Variant list:
O Divide Value By:	Show List of Values of Type: Variant
⊙ Ask User For Value:	Allow Users Own Value

When **Variable Specified List** is used, you can select the variable name from which to extract the data:

Prompt: Choose from Variant list:	
Show List of Values of Type: Variable Specified List> Edit Value List	
Allow Users Own Value	N
Extract Value List from Variable: ChoiceList	
Field Separator: 🔎 🗌 Field can be Quoted	

You can also use a check box to specify whether to Allow Users Own Value or to only let them choose a value from the drop down list.

Ability to make a report not-insertable into a design

You can now make a report not-insertable into a design. To do this insert the new command **Not Insertable** at the top of the report script. The interactive **Insert User Report** option will not list it.

```
List of Nets
```

Note: If an already inserted report is changed to be not insertable - it will still run..

Ability to output angles as clockwise rotation

The ability to output angles as clockwise rotation has been added. A switch has been added to the **Angle Units** command **Report Clockwise Angle Direction**.

Edit Angle Units Command
Command: Angle Units
Unit Type: 💿 Degrees 🛛 🔿 Radians
Angle Text: deg
Precision: 1
Decimal Point Character:
Report Clockwise Angle Direction

Ability to output coordinates relative to the relative origin

The ability to output coordinates relative to the Relative Origin has been added. The **Transform Coordinates** command now has new parameters **Coordinate System Origin** and **Relative Origin**.

Edit Transform Coordinates Command	×		
Command: Transform Coordinates "sys origin" ""			
Before Transformation - Base Coordinates On:			
Coordinate System Origin (Default)			
Relative Origin (if Defined)			

If **Relative Origin** is checked and the relative origin is placed in the design the coordinate will be output relative to it. Else, if the **Coordinate System Origin** box is checked and the system origin is defined it will be output relative to the system origin, otherwise it will be output using its exact value.

Τ	ansform Coordinates "rel origin" ""		
ė Li	List of Components		
	Component Name		
	Position		

Note: use of this command stops any CAM Plot transformation and if the relative origin is at (0, 0) and not switched on, it will not be used.

New commands for reporting the placement area for each footprint in a library

Enabled commands for reporting the placement area for each Footprint in a library. This allows you to run a report on a library which gives you the Footprint Name and the total sq mm (or sq inch) size of any body Area and placement Area. To do this **List of Symbols** now has the following field commands enabled:

Is All SM Pads Is Embedded Component Is SM Device Position (symbol origin) Extents Area Top Placement Extents Area Bottom Placement Extents Area

Top Placement And Pads Extents Area

Bottom Placement And Pads Extents Area

Is All SM Pads and Is Embedded Component report True or False as the test status.

The **Position (symbol origin)** reports the position of the Symbol origin in the Footprint relative to the coordinate system origin.

Each of the Extents commands allow you to choose the Value from a drop down list:

Edit Command		
Command: Extents		
Value:	Area 🗸	
	Area	
Use: () X Coord (Bottom Left	
Vew Column	Height	
	Top Right	
Column Width: 0	Width	

Extra field on an attribute to get the "original" value from the part or symbol

There is now an extra field on an attribute to get the "original" value from the Part or Symbol. This allows you to produce a report of the attributes that have been changed in the design. When you edit an **Attribute** command, or an **If** command on an attribute field or a variable command using an attribute field, there is now an extra **Use:** option of **Inherited Value**. It reports an empty field if there is no inherited value.

⊫ List	of Components
	Component Name
	Attribute "Value"
	Attribute "Value" Inherited Value

Command:	Attribute "Value"		
Attribute:	/alue	💌 🗖 Or	
- 1100:			

Note: A test of Exists on the inherited value will be true if there is an inherited attribute, even if it has no text.

When run, it could report like this:

```
A1 10uF Inherited Value: 1uF
A2 100uF Inherited Value: 1uF
A3 1uF Inherited Value: 10uF
```

Ability to report groups

The ability has been added to report Groups.



Use List Of Items within List of Groups to report each item in the group. For each item you can use Item to report a description of it and can test whether it Is Group Master Item. You also can use the following commands:

```
Group Name
Is Tight Group
List Of Components
Comp Name, etc
Is Group Master Item
List Of Gates
Gate Name, etc
Is Group Master Item
```

Is Group Master Item - Use this command within List of Items to report whether the item is the groups master item. This can also be used within List Of Components and List of Gates commands when they themselves are within a List Of Groups command.

Use List Of Items within List of Groups to report each item in the group. For each item you can use Item to report a description of it and can test whether it Is Group Master Item.

```
List Of Groups
List Of Items
Item - [reports textual description of the item]
Position
Is Group Master Item
Is Selected
```

When run, the report could report like this below, you could also test and change the conditions reported:

```
Component: A2 Group1 (524.000,535.000) true false
Component: A3 Group1 (524.000,530.000) false false
Copper: Layer:Top Group1 (518.500,533.500) false false
```

Used within List Of Components, List Of Groups can be used to show components which belong to multiple groups.

```
List Of Components
Comp Name, etc
List Of Groups
Group Name
```

You can also use List Of Groups within List Of Gates.

```
List Of Gates
Gate Name, etc
List Of Groups
Group Name
```

Added ability to set the report file name and folder within the report script

The ability to set the report file name and folder within the report script has been added. There is a new command **Report File Name** that replaces the old **Report File Extension** command. This now has optional Folder, Filename and Extension parameters. If a relative folder path is provided, it will be applied to the current design folder or library folder.

Edit Report Fi	ile Name Command	
Command: Rep	oort File Name - Folder: "o:\temp\" Name: "my_report_name" Extension: "txt"	
When the re folder path is	sport is run, any unchecked fields will use the default values, and if a relative s provided it will be applied to the current design folder or library folder.	
If text of the is run by the	form \$(Param1) is included in your names, it will be replaced when the report corresponding Runtime Parameter.	
Folder:	c:\temp\ Browse	
🗹 Filename:	my_report_name	
🗹 Extension:	txt	
	OK Cancel	
E List c	ort File Name - Folder: "c:\temp\" Name: "my_report_name" Exten: <mark>f Components</mark> Is All SM Pads Component Name	sion: "txt"

Text Substitution In Report Filenames

The **Report File Name** command also allows runtime substitution of text of the form \$(Param1) with its corresponding runtime parameter.

If you are using nested reports, you can build up your filename using information passed through to this report from its parent report. If you include text of the form \$(Param1) anywhere in the report name fields provided in this dialog, it will be replaced at run time by the corresponding runtime parameter passed from the parent report. If the runtime parameter was not set up, the text will simply be removed from the name. Note, there are only eight parameters so \$(Param8) would be the highest that can be used.

	Set Variable "count" to 0	
	Report File Name - Name: "\$(Par-	am1)"
	Set Variable "side" to Runtime Pa	ameter 1
.	If "Variable: side" is matching ""	
	Prompt for Variable "side" T	Command: Report File Name - Name: "\$[Param1]"
ė	If "Variable: side" is matching ""	When the report is run, any unchecked fields will use the default values, and if a relative
	Set Variable "side" to all	folder path is provided it will be applied to the current design folder or library folder.
.	If "Variable: side" is matching "a	If text of the form \$(Param1) is included in your names, it will be replaced when the report
	Text "All components"	is run by the corresponding Runtime Parameter.
.	Else	Folder: Browse
	Variable "side"	
		Filename: \$(Param1)
	Underline "-"	
//		
-	List of Components	
	Set Variable "wantComponer	nt" to true
=	If "Variable: side" is not mai	iching "all"
	If "Side" is not match	ing "Variable: side"
	Set Variable "wa	ntLomponent" to faise
	If Wariable: wantComponen	P is equal to "true"
	Component Name	r is equal to true
	Increment Variable "co	unt" hu 1
	Save Variable "count" to the repor	t result

Added List of Symbol Names inside List of Part Gates

A List of Symbol Names command has been added for use inside the List of Part Gates command. This allows access to alternative symbol names when running a Library Report.



List of Symbols contains additional List commands

For use within a Library, the List of Symbols command now has List of Mounting Holes, List of Bond Pads and List of Vias enabled for Footprints (in a library).



New command for Is Default for Symbol and Footprint Names

Within the **List of Symbol Names** and **List of Footprint Names** commands, you now have an **Is Default** command to report if the current symbol name or footprint name is the default that will be used by the Part Gate (in Schematics) or Part (in PCB). You can use this to report if a Part in the design is using the default symbol or not. The following example lists the alternative Footprints for a Part:



Snap Cursor to Grid

There is a new check box on the **Options** dialog and **Preferences** page named **Snap Cursor to Grid**. Select this option to make the cursor move only on the current grid when in the design area.

Low High	et 🧧 💌 enuies
C Par:	(Note : Needs program restart to take effect)
Pan Sensitivity: Reversed Mouse P	an
Low High Res	et Cursors:
	Style: Small Cross 🖌
Relative Coordinates	Snap Cursor to Grid
Show relative coordinates in dialogs when use	d
- Rack Annotation	Dialog Bride:

Gridded cursors will still be shown when modal cursors are displayed.

This option can only be used with the three cross cursors, it will not work with the standard windows cursor.

Toggle Cursor Gridding Command

There is also a new command available in the **Customise** option and **Keyboard** named **Toggle Cursor Gridding**.

Multiple File Extensions

The ability to have multiple extensions associated with a file type has been added to Version 7.5. For example for STEP Models, you could specify .stp and .step.

From within the **Options** dialog and **File Extensions**, use the **Edit** button to add multiple file extensions as required.

Display Edit Connection Edit Shape File Extensions Find General Interaction Move Macros Multi-Screen Online ERC Warnings	.dch .dcs .drl .dxf .dxf .gbr .gbr .gbx .hp .idb .idb .idb	Database Connection Search Files Database Connection Search Files Drill Output Files DXF Output Files Gerber Import Files Gerber Output Files LPKF Output Files Pen Plot Output Files IDF Output Board Files IDF Output Board Files IDF Output Library Files
	.stf .stp;.s .stp .txt .uni	Schematic Technology File tep STEP Models STEP Output Files Report Files UniDAT Output Files

The Edit dialog allows you add the extensions using the semi-colon ; character as the extension separator.

Edit File E	xtension	
Туре:	STEP Models	
Extension:	.stp;.step	
	ОК	Cancel

Multiple file extensions are used for both browsing of data and writing data out. These multiples would be shown under the **File Of Types** as legal extensions to use for writing data.

An example of multiple file extensions would be where browsing for STEP models. Models with the .stp and .step extension would be selected in the Contents: list.

Folders Schematic Syr	nbols Schematic Doc Symbols	PCB Footprints	PCB Doc Symbols	Parts 3D Vie	w STEP Mo
Model Folder: C:\S	TEP Models				
Contents 2520-6002_step.stp 6002.step c-1.102395-2-m-34.s c-1.1734037-04-34.s c-13222-1.41-34.stp c-1473005-11-j3d.stp c-1473005-11-j3d.stp	1220.stp tp tp	<u>R</u> enan <u>C</u> opy <u>M</u> ove	ne		
c-5102321-1-d-3d.st C-CAN, C-CAN_34.st cap-elec-430x430k	p 19 90.stp				

CAM Plot Changes

Enable a Named Group of Plots from Context menu

From within the **CAM Plots** dialog, you can now enable just a named group of plots with a single context menu command **Enable Selected Group**.

N	Ex abla d	C	Davis	D	Carla	Detet (
name Tan	Enabled	Group	Device	Process	Scale	Rotat	<u>N</u> ew
Power		Gerber	Enable Sel	ected Group	6	Auto	<u>E</u> dit
Bottom		Gerber	Apply To C	iolumn K	6	Auto F	Сори
Silkscreen Top		Gerber	Apply to pl	lots using same devid	Auto F	Gobà	
Solder Mask Top		Gerber	Gerber	Layer Solder Masł	1.000	Auto F	<u>D</u> elete
Paste Mask Top		Gerber	Gerber	Layer Paste Mask	T 1.000	Auto F	Delete All
Silkscreen Bottom		Gerber	Gerber	Layer Silkscreen B	9 1.000	Auto F	Delete All
Solder Mask Bottom		Gerber	Gerber	Layer Solder Masł	1.000	Auto F	
Paste Mask Bottom		Gerber	Gerber	Layer Paste Mask	1.000	Auto F	Up
Documentation			Windows	Layer Documentat	i 1.000	Auto F	28
<through hole=""></through>			Excellon	Layer Span <thro< td=""><td>u 1.000</td><td>No Ro</td><td>Do<u>w</u>n</td></thro<>	u 1.000	No Ro	Do <u>w</u> n
Drill Drawing			Gerber	Laver Span <thro< td=""><td>u 1 000</td><td>Auto F</td><td></td></thro<>	u 1 000	Auto F	

Move Additional Plot Items In CAM Plot Option

You can now move additional plot items within a plot on the CAM Plots grid. The currently selected row is now drawn in bold and can be moved **up** or **down** the grid using the buttons on the right side of the dialog.

s						
lot Settings	Drill Sizes					
Enabled	Device	Process	Scale	Rotate	Mirror	Posi
	Windows	Layer Top Electrica	1,000	Auto Rotate		Auto
		Layer Top Silk 💙	1			
		Layer Bottom Silk S				
	Windows	Layer Bottom Electr	1.000	Auto Rotate		Auto
	Windows	Laver Bottom Electr	1.000	Auto Rotate		Auto
	s lot Settings	5 lot Settings Drill Sizes Enabled Device Windows Windows Windows	5 Iot Settings Drill Sizes Enabled Device Process ✓ Windows Layer Top Electrica Layer Bottom Silk S Layer Bottom Electr Windows Layer Bottom Electr	S Iot Settings Drill Sizes Enabled Device Process Scale ✓ Windows Layer Top Electrical 1,000 Layer Bottom Silk S Layer Bottom Silk S	S Iot Settings Drill Sizes Enabled Device Process Scale Rotate Windows Layer Top Electrica(1,000 Auto Rotate Layer Bottom Silk S Windows Layer Bottom Electr 1.000 Auto Rotate	S Iot Settings Drill Sizes Enabled Device Process Scale Rotate Mirror Windows Layer Top Electrical 1 000 Auto Rotate Layer Bottom Silk S Windows Layer Bottom Electr 1 000 Auto Rotate Windows Layer Bottom Electr 1 000 Auto Rotate

Copy Plot

From the **CAM Plots** dialog, you can now copy a plot using the **Copy** plot button. This copies the currently selected plot and all additional plots. The copied plot will be displayed with the original name plus the word **Copy** next to it, as shown below with **Bottom Electrical Copy**.

Generate Plot - CAM Plots											
Plot Preview CAM Plots Plo	t Settings	Drill Sizes									
Name	Enabled	Device	Process	Scale	Rotate	Mirror	Position	Area	Step Repeat	Exclusions	<u>N</u> ew
Top Electrical		Windows	Layer Top El	1.000	Auto Rotate		Auto Shift	<design extents=""></design>		Y	
Bottom Electrical		Windows	Layer Bottom	1.000	Auto Rotate		Auto Shift	<design extents=""></design>		Y	<u> </u>
Bottom Electrical Copy		Windows	Layer Botto	1.000	Auto Rotate		Auto Shift	<design extents<="" td=""><td></td><td>Y</td><td>Conv</td></design>		Y	Conv
Top Silk Screen		Windows	Layer Top Sil	1.000	Auto Rotate		Auto Shift	<design extents=""></design>		Y	<u> </u>
Bottom Silk Screen		Windows	Layer Bottom	1.000	Auto Rotate		Auto Shift	<design extents=""></design>		Y	<u>D</u> elete
<through hole=""></through>		Excellon	Layer Span	1.000	Auto Rotate		Auto Shift	<design extents=""></design>			Delete All
											Delete All

Crop Plot Output

In the **CAM Plot Wizard**, you can now define a **Crop area** which will suppress items totally outside the given area.



Use **Crop Output To:** a defined area to prevent items outside this area from being plotted. The cropping is only done crudely, so items that cross the area will be plotted. In general, this would be used for a set of items which form a coherent subset of the design which you would want to position within the overall plot.

You can choose the **Board** and **Design** extents from the list. If there are **named areas** in the design, these are also shown on the drop down list for selection.

PDF Output Bookmarks

When you output to PDF using the built-in PDF writer supplied with Pulsonix, you can now output a list of Components and Nets as bookmarks.



The Components and Nets will be displayed within a list on a per-page basis that can be opened using the small + box. These lists are hierarchical, components show the Component Name, the Gate and the pin numbers on the Gate. Nets will show the net name and the component name with all pins on that net.

The bookmark now also lists Blocks and Block Ports from the Schematic editor.

🖃 🧊 ТОР	
🖻 🦵 Blocks	
🗉 🦵 CTRL	
🖻 🦵 LD1	
- 🖓 SE	G
🗉 🦵 LD2	
🗄 🦵 LD3	
🗉 🥊 CTRL:Page	1
🗉 🦵 Block Po	rts
🗄 🕕 Compon	ents

PDF Output Variants

When outputting a PDF plot using the build-in PDF driver within the CAM Plots dialog, you can now list variants and fitted status of each component by selecting it and using options on the context menu.



Gerber Verification Plot to PDF

The verification plot can now be output to PDF as well as the existing Windows output. Choose PDF or Windows from the drop down list box presented in the Gerber Setup dialog within CAM Plots.

Setup Gerber		X
Plotting Area	Units: inch 💌	ОК
Lower Left (X,Y):	Upper Right (X,Y):	Cancel
0.0000 0.0000	10.0000 10.0000	
- Registration Point		Apertures
Centre 🗸	0.0000 0.0000	
Produce Verification Plot	Fixed Aperture File	e
PDF V Scaled	To Fit emp\GerberAperture	s.csv Browse
PDF		
0 Windows	Format:	
Hardware arcs: 1955	Units: inches Tupe: Absolute	

Associated Parts – Execute Hyperlink

In the **Edit Associated Part** dialog from the **Design Properties** dialog, an **Execute Hyperlink** button is available for hyperlink attributes on the associated part. The Associated Part must have a Hyperlink attribute already added in the **Part** definition using the **Part Editor**.

Properties: Po Attributes Associa	CB Design - Asse ated Parts Journal	ociated Part		
Name	Min Quantity	Quantity	Add	
Board	0	1		
Plate	0	1	Delete	
Bolt 2.4mm	12	12		
Heatsink TO3	2	2	Edit	
Washer 6mm	4	4		
Nut 2.4mm	4	4	Edit Associated Part	
	0		Part: HEATSINK-SF O Quantity: 1 Minimum Quantity: O Description: Heatsink 5F Series DIP/TO-5 Thermalloy Car Attributes: Category=Hardware/Heatsink Execute Value=12.7mm Value=12.7mm Execute	K 1cel Hyperlink

Local Pin Types

The **Properties** dialog can now be used to override a **Pin Type** on a component pin in a Schematic design. On the **Pin** page, there is a new **Override** check box. When checked, it allows you choose a pin type for the pin from a drop down list. This pin type will then be used when running the **Electrical Rules Check** (ERC) option.

🗖 Propertie	s: Pin - Pin		
Comp Pin	Attributes Pin Attributes G	Net Nate Component	et Attributes Ungated Pins
<mark>. №</mark> ame: <u>P</u> osition: <u>A</u> ngle:	3 114800.0 134900.0 0.0	Logic Name: IN+	
Alternativ Name: <u>W</u> idth:	Pin Type: re Pin Style Cross60 60.0 S <u>h</u> ape:	BiDirectional <undefined> BiDirectional Ground Input No Connect Open Collector</undefined>	Verride
Length:	60.0 Thickness:	Open Emitter Or-Tieable Output Passive Pob Connect Power Terminator Tri-State	

You can also use **Properties** to override the **Pin Type** on **Ungated** pins. Check the **Override Pin Type** check box to then gain access to the **Pin Type** drop down list.

Prope	erties: Pin - Ungat	ed Pins			
	Comp Attributes	Net			Net Attributes
Pin	Pin Attributes	Gate		Component	Ungated Pins
Pin	Override Pin Type	Pin Type		Net	
4		<undefined></undefined>	0	∋nd	
8		≺Undefined>	V V	'DD	
		Bi-Directional Ground Input No Connect Open Entitler Or-Tieable Output Passive Pcb Connect Power Terminator Tri-State			

Reloading Local Pin Types

When reloading components from the library there is now a **Keep local pin types** option to prevent the local pin types being changed.

Reload From Library 🛛 🛛 🔀							
Only reload if different version in the	library						
Reload control							
Keep <u>a</u> ttribute styles & positions	Replace gate symbols with defaults						
Keep local attribute values	Reload nets assigned to ungated pins						
Replace <u>f</u> ootprint with default	Update predefined nets on symbol pins						
Replace <u>p</u> in networks	Match pins by position						
Re-apply name stem from part	Keep local pin <u>types</u>						
Match position of the first pad							

Local Pin Types In Report Maker

In the **Report Maker** option on the **Output** menu, there is now a **Has local pin type** command to be used on a schematic component pin.

Component Variant Spreadsheet

You can now view and edit variants in a spreadsheet style interface. The **Component Variant** editor is available on the **Edit** menu in both the Schematic and PCB design editors.

Part	Name	UK*	Norther	n Europe	USA	Asia	1	A	^	OK	
С											
CP3				C	Polarised3	C Polarise	d3			Apply	
DIN41612-	PLG-64		DIN41612	-SKT-64							
3WP										Lancel	
5WP											
26WDP											
26WDP										Lopy	
3WP										Dasta	
BC184L										Faste	
BC184L											
R 0.063W	SMTF 1.6	ĸ		F	2			R 0.063W S	L G	le Verienter	
R									ľ	NU Valiarits.	
R											
R									,	Not Fitted:	
R8C						4308R-101	1				
R											
R		R 0.063W SMTF 1	.5K			R 0.063W	SMT	R 0.063W 5	9	Shared:	
R		R 0.063W SMTF 1	.5K			R 0.063W	SMT	R 0.063W \$			
R		R 0.063VV SMTF 1	.5K			R 0.063W	SMT	R 0.063W \$			
R									[Different Attribu	
R											
SPDT											
74LS251									L	Jifferent Instand	
74LS251									~		
								>			
				D: 101						Attributes:	
enc HI		Update Spreadsh	eet	Discard Unar	iges					Add Column	
	F 244 - 14	Dent Name	Shared	Identifier	A	ttributes			ributes		
ananc	Filleu	Falthanic	Attributes	Instance	C C	R	٧	Faitbest	sut.	Del Column	
	Image: A start of the start	R 0.063W SMTF 1.5K	а	а	0.2pF	1.5K		Thick Film Su	urfe 🛛		
n Europe		R 0.063W SMTF 1.5K	a	а	0.2pF	1.5K		Thick Film Su	urfe	Delete Value	
		R	b	b	0.2pF	3.3K		Resistor			
				a	0.2pF	1.5K		Thick Film Su	urfe 🛛	Hevert Value	
		R 0.06377 SMTF 1.5K									
		R 0.063W SMTF 1.5K R 0.063W SMTF 2.2K	d	a	0.2pF	2.2K		Thick Film Su	urfe 🔄		
Imerica		R 0.063VV SMTF 1.5K R 0.063VV SMTF 2.2K R	d b	a b	0.2pF 0.2pF	2.2K 3.3K		Thick Film Su Resistor	urfe C	Description:	
Imerica	✓✓✓	R 0.063VV SMTF 1.5K R 0.063VV SMTF 2.2K R	d b	a b	0.2pF 0.2pF	2.2K 3.3K		Thick Film Su Resistor	urfe (Description:	
	Part C CP3 DIN41612-1 3MP 5V/P 26WDP 26WDP 26WDP 26WDP 26WDP BC184L BC184L R <t< td=""><td>Part Name C CP3 DIN41612-PLG-64 3MP 5MP 26MOP 3MP BC184L BC184L BC184L R</td><td>Part Name UK* C C CP3 DIN41612-PLG-64 SWP SWP SWP SWP 26WDP SWP 26WDP SWP 26WDP SWP 26WB SWP SCH4L SCH4L BC184L BC184L R R0.063W SMTF 1.5K R R R8C R R R SPOT SVSTET 74LS251 J 74LS251 J Fitted Part Name Image: P R OR3W SMTF 1.5K R</td><td>Part Name UK* Norther CP3 DIN41612-PLG-64 DIN41612 SWP SWP DIN41612 SWP SWP SWP SWP SWP DIN41612 SWP SWP SWP SWP SWS SWP SOUTH SW SWE BC184L SWE SWE R R.0.063W SMTF 1.5K R R R.0.063W SMTF 1.5K SPDT 74LS251 SWE SWE T4LS251 SWE SME Grain Fitted Part Name Shared Attributes R R SME</td><td>Part Name UK* Northern Europe C C C CP3 C C DIN41612-PLG-64 DIN41612-SKT-64 C SWP 26WDP C 26WDP 26WDP C 26WDP 26WDP C 26WDP 26WDP C 26WAP C C SWE C C 80C184L C F R R 0.063W SMTF 1.5K R R 0.063W SMTF 1.5K R SPDT C 74LS251 C C 74LS251 C<</td><td>Part Name UK* Northern Europe USA C C C C C C Polarised3 DIN41612-PLG-64 DIN41612-SKT-64 DIN41612-SKT-64 DIN41612-SKT-64 DIN41612-SKT-64 SWP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP<</td><td>Part Name UK* Northern Europe USA Asia C C C C Polarised3 C C Polarised3 C Polarised3 C Polarised3 C Polarised3 C Polarised3 C Polarise SWP SWP</td><td>Part Name UK* Northern Europe USA Asia C C C C C Polarised3 Polarised3 C Polarised3 C Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 P</td><td>Part Name UK* Northern Europe USA Asia A C C C C C Polarised3 Polarised3</td><td>Part Name UK* Northern Europe USA Asia A C C C C C Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised</td></t<>	Part Name C CP3 DIN41612-PLG-64 3MP 5MP 26MOP 3MP BC184L BC184L BC184L R	Part Name UK* C C CP3 DIN41612-PLG-64 SWP SWP SWP SWP 26WDP SWP 26WDP SWP 26WDP SWP 26WB SWP SCH4L SCH4L BC184L BC184L R R0.063W SMTF 1.5K R R R8C R R R SPOT SVSTET 74LS251 J 74LS251 J Fitted Part Name Image: P R OR3W SMTF 1.5K R	Part Name UK* Norther CP3 DIN41612-PLG-64 DIN41612 SWP SWP DIN41612 SWP SWP SWP SWP SWP DIN41612 SWP SWP SWP SWP SWS SWP SOUTH SW SWE BC184L SWE SWE R R.0.063W SMTF 1.5K R R R.0.063W SMTF 1.5K SPDT 74LS251 SWE SWE T4LS251 SWE SME Grain Fitted Part Name Shared Attributes R R SME	Part Name UK* Northern Europe C C C CP3 C C DIN41612-PLG-64 DIN41612-SKT-64 C SWP 26WDP C 26WDP 26WDP C 26WDP 26WDP C 26WDP 26WDP C 26WAP C C SWE C C 80C184L C F R R 0.063W SMTF 1.5K R R 0.063W SMTF 1.5K R SPDT C 74LS251 C C 74LS251 C<	Part Name UK* Northern Europe USA C C C C C C Polarised3 DIN41612-PLG-64 DIN41612-SKT-64 DIN41612-SKT-64 DIN41612-SKT-64 DIN41612-SKT-64 SWP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP 26WDP<	Part Name UK* Northern Europe USA Asia C C C C Polarised3 C C Polarised3 C Polarised3 C Polarised3 C Polarised3 C Polarised3 C Polarise SWP SWP	Part Name UK* Northern Europe USA Asia C C C C C Polarised3 Polarised3 C Polarised3 C Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 P	Part Name UK* Northern Europe USA Asia A C C C C C Polarised3 Polarised3	Part Name UK* Northern Europe USA Asia A C C C C C Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised3 Polarised	

The dialog is broken down into two distinct sections with controls for each.

The top section is the component variant spreadsheet, with a row for each component. The bottom section allows you to view and edit the variant details for the component currently selected in the top half.

Compone	ent Variant Spread	sheet	· · · · · · · · · · · · · · · · · · ·	
Name	Part Name	USA*	GB	EU
C1	C1812	ĺ	C Polarised	C 1.0pF
C2	C1812		C 1.0pF	
C3	С			
C4	С		10-16-1031	
C5	0603YC104KATDA		<u> </u>	

The main grid in the top left of the dialog shows a row for each component and a column for each variant in the design.

Cells are colour coded to show the state of each component in a particular variant. The colour indicates whether the component is not fitted, is a different part, uses different attribute values or uses

a different physical appearance. This makes the spreadsheet ideal for quickly seeing which components in the design use variants and which are not fitted in certain variants.

The first column shows the component name and the second column shows the main part used for the component. The variant columns only show a part name for a component if it is different to the main part. The part name cannot be edited in this grid, but can be changed by editing its value in the component variant grid at the bottom section of the dialog and then updating the spreadsheet with the changed component. The height of the row may be adjusted if you wish to accommodate longer variant names but keep the width of the columns to a minimum so many variants can be viewed on a single screen.

The **Name**, **Part Name** and **Variant** columns can be sorted by clicking on their column headers. A variant column is sorted by variant **State** or **colour**. Clicking on any cell apart from the column headers will make the component the row represents the current component and will change the lower grid to show its variant details. The current component is indicated in the spreadsheet using a bold text style in the first two columns. When clicking on a different row to change the current component, and the lower grid has variant changes pending, you will be asked if you want to update the spreadsheet with these changes prior to switching the current component.

You can select multiple components in the spreadsheet by dragging the left mouse button vertically in the **Name** column. The only reason for doing this is to use **Paste** to copy the same variant information from another component to multiple components.

Down the right-hand side of the top of the dialog is a set of buttons that allow you to copy variant information between components and define the colours used for the different variant states.

OK - This will Apply any component variant changes to design and exit the dialog.

Apply - If this button is enabled it means there are component variant changes in the spreadsheet that have not yet been committed to the design. Pressing this button will apply these changes to the design, but stay in the dialog for more editing.

Cancel - This will exit the dialog without changing the design at all.

Close - The **Cancel** button becomes **Close** after using **Apply** for the first time. Pressing this button will exit the dialog and throw away any changes you have been made since the last **Apply** was performed.

Copy - This will remember all the information shown in the Component Variant grid for the current component. This information can be subsequently copied to another component by selecting the target components row in the spreadsheet and pressing the **Paste** button. The shortcut keys **<Ctrl-C>** can also be used.

Paste - This will copy the remembered variant information from the previous **Copy** action to the current component, or multiple selected components, in the spreadsheet. Part names, fitted state and sharing information will all be copied, so this feature should only be used on components that are to be the same part and want to appear the same for all variants in the design. The shortcut keys <**Ctrl-***V*> can also be used.

Cell Colour Coding

The next set of buttons show the colours that will be used for each variant 'state' in the dialog. Click on a colour block to change its colour. The colours are retained in the system registry, so that your choice of colours is preserved for future use. **No Variants** - This colour is used for all cells of a component that does not vary for any variant.

Not Fitted - This colour is used for cells where a component is not fitted in the variant the cell represents.

Shared - This colour is used for cells where a component is fitted in the variant the cell represents and it shares the same part name, attributes and physical component instance as the 'normal' component.

Different Attributes - This colour is used for cells where a component is fitted in the variant the cell represents, but it has a different set of attributes to the 'normal' component.

Different Instance - This colour is used for cells where a component is fitted in the variant the cell represents, but uses a different physical component instance to the 'normal' component.

No Variants:
-
Not Fitted:
· ·
Shared:
-
Different Attributes:
-
Different Instance:

Component Edit Pane

The bottom part of the dialog is occupied by a grid or table showing the variant details for currently selected component in the spreadsheet.

omponent: R1		Update Spreadshe	et I	Discard Chang	es				Attributes: -
Variant	Fitted	Part Nama	Shared	Identifier		Attributes		Part Decorir	Add Colum
variarit	Titteu	Faithanic	Attributes	Instance	С	R	٧	raitbescrit	Del Colur
ЈК	 Image: A set of the set of the	R 0.063W SMTF 1.5K	а	а	0.2pF	1.5K		Thick Film Surfa	
Northern Europe		R 0.063W SMTF 1.5K	a	a	0.2pF	1.5K		Thick Film Surfa	Delete Va
USA		R	b	b	0.2pF	3.3K		Resistor	
Asia		R 0.063W SMTF 1.5K	с	a	0.2pF	1.5K		Thick Film Surfa	Hevert Val
Africa		R 0.063W SMTF 2.2K	d	a	0.2pF	2.2K		Thick Film Surfa	
South America		R	b	b	0.2pF	3.3K		Resistor	Description:
									Part Part
<								>	🔄 🗌 Varia

This is similar to the **Component Variant Properties** dialog for a selected component, but with the added feature of being able to include attribute value information. It contains a row for each variant in the design and columns allowing you to change the components fitted state, part name, whether it has different attributes and whether it has a separate physical component instance. Cells in these columns are coloured depending on their 'state' using the same system as the spreadsheet. The name of the variant currently active in the design is shown in **Bold** text in the appropriate row. The table can be enlarged to show more variants, see **Resizing the Dialog** section at the bottom of this page for details.

Update Spreadsheet - This button becomes enabled when you have changed one of the cells in this grid. Press the button to update the main spreadsheet with the changes for this component. The changes will not be committed to the design until the **Apply** or **OK** buttons are pressed to apply the changes to all the components in the spreadsheet.

Discard Changes – This button also becomes available when you have changed one of the cells in this grid. Press the button to discard the changes to the component and restore it to the values last saved in the spreadsheet.

Column Headers

Variant – This column shows the variant name. The name highlighted in **Bold** is the current variant.

Fitted - This column shows if the current component is fitted in that variant (components are fitted in all variants by default). Use the checkboxes to set the fitted status.

Part Name - This column shows the part used for the current component in that variant. You can use this column to change the part used for this component, but if you want a different part for a particular variant you must first use the **Shared Identifier** columns to give the component a different shared **Attributes** identifier. Changing the part for a particular variant will also change it for all variants that have the same shared attributes identifier. Double click in the cell to change the part used by a component. The **Change Part** dialog will be presented for you to choose the required part. If the chosen part for a particular variant uses a different symbol or footprint to the components main part, then you will also have to change the shared instance identifier to a different value before updating the spreadsheet.

If the Database Connection is enabled, the component's variants dialog will display an additional **Database** column; This displays the component's database key field information for each variant and is useful for seeing if database part variants are being used even if the Pulsonix Library part is the same across variants.

Shared Identifier - An advanced form of variants is to use a different component instance in some variants. You may want the component to have different attributes, but still be represented by the same physical symbol or footprint. Alternatively, you may want a completely different part with a different symbol or footprint, that can be positioned and connected independently. To do this, you need to assign a **Shared Identifier** (also referred to as the Variant Id), this can be any character or word.

There are two columns under the **Shared Identifier** heading. You can share **Attributes** between variants by entering the same identifier in the first column, and you can share physical **Instances** (symbols) by entering the same identifier in the second column. A blank Id is a valid identifier and initially, by default, all columns are blank indicating the same instance and attributes are shared across all variants. You cannot share attributes without also sharing instances across the same variants. Instances can be shared provided they have compatible parts and footprints in the variants in which they are to be shared.

For each Identifier, there must be at least one variant where the **Attributes** and **Instance** Identifiers are the same. The colour coding of the cells mentioned above helps indicate which variants have different attributes and which use different physical component instances.

Attributes:

You can add a column showing a particular attribute name and all the values for that attribute for each variant. This enables you, for example, to change the value of a generic resistor for each variant. If you want a different value for a particular variant you must first use the **Shared Identifier** column to give the component a different shared **Attributes** identifier. Changing the value for a particular variant will also change it for all variants that have the same shared attributes identifier. To change the value, left click in the cell and type the required text.

(Component:	C1			Update Spreadsheet			nanges	
	Variant	Fitted	Part Name	Database	Shared	dentifier	Attributes	Variant	Part Description
	• an rank	Hatta	. are manto	Databaoo	Attributes	Instance	Value	Description	r are booonpaon
	USA	>	C1812		с	с		120V variant	Chip capacitor
	GB	V	C Polarised		b	b	22pf	240V variant	Capacitor Polarised
	EU	V	C 1.0pF		а	a	1.0pF	220V variant	Capacitor

The buttons to the right-hand side of the grid provide you with some tools to edit the attributes and their values. Some buttons may only be enabled if appropriate cells are selected in the grid.

Add Column - This will add another column (Attribute Name) to the Attributes section of the grid, allowing you to change existing attributes more relevant to the current component, or to create a new attribute name to use for attributes be added to the component. An error message will appear if you enter an attribute name that already exists in the grid. The attribute names are retained in the system registry, so that your choice of attribute names for columns is preserved for future use.

Delete Column - Use this to remove an attribute column from the grid. This does not delete existing attributes for the component, removal of the column simply means you will not change the values of this attribute when the spreadsheet is updated with the component changes.

Delete Value - Depending on what is selected this will either be **Delete** or **Reset**. Delete will remove the attribute from the selected cell. Reset will reset the selected cell to its inherited value.

Revert - Use this button if you have accidentally modified the wrong cell but do not want to cancel the whole dialog. Pressing Revert will return just the selected cell to its original value from the design. This button will only be enabled if the selected cell has been modified.

Description:

There are two optional columns that can be used to show the description of the variant and part used in the row. Use the check boxes to the bottom right of the dialog to show or hide these columns.

Attributes:
Add Column
Del Column
Delete Value
Revert Value
Description:
🗹 Part
🗹 Variant

Heverr value
Description:
🗹 Part
🗹 Variant

Additional Database Connection Column

These additional features are available as part of the Pulsonix Database Connection (PDC) product.

Component: C1					
Variant	Fitted	Dart Hame	Database	Shared	dentif
Varianc	ritteu Part Hame	Database	Attributes	Ins	
USA		C1812		¢	с
GB	V	C Polarised		b	b
EU	V	C 1.0pF		a	а

If the **Pulsonix Database Connection** is enabled, the component's variants dialog will display an additional **Database** column; This displays the component's database key field information for each variant and is useful for seeing if database part variants are being used even if the Pulsonix Library part is the same across variants.

Note that if the Part Name is changed for a variant, unless it is chosen using the **Database Bar**, any existing database information will be lost and it will revert to being a 'local' component for that variant. For a 'local' component, the Database column will be blank.

Resizing the Dialog

This dialog can be resized by dragging any of its edges or corners. The spreadsheet grid at the top of the dialog will be resized by the amount you have resized the dialog by, and the other controls will be stretched or moved to fit around it.

If you hold the **Shift** key down whilst resizing, the Component Variant grid in the bottom half of the dialog will be resized, and the spreadsheet grid in the top half of the dialog will only be horizontally altered. This is useful if you have a lot of variants to enlarge the lower grid to be able to see them all at the same time.

Hevert value	
Description:	
Variant	
C	$\land \land \neg$
	シ 、――

Delete Variant Changes

When using the **Variant Manager**, if deleting the variant would cause a Component to become **Unfitted** in any variant, you are now prompted to decide if these components should be deleted.

	Variants Manager	×	
	🖉 Use Variants		
	Current Variant	ר ר	
	Europe	2	
	Description:		
	230v European variant Copy		
	Delete		
	Variants:		
	<master design=""> Not-Fitted USA Component</master>		
	Europe	-	
Pulsonix			X
D D	eleting variant 'Europe' would cause some components to become u	nused i	in any variant.
🕘 _D	o you want to delete these components?		
	Yes No		

Attribute Editor shows Variant ID

The **Attribute Editor** now shows the **Variant ID** for variant components in the design. Selecting a variant ID cell will display information about the component and variant it is fitted (or not fitted) in.

	🔲 Edit	Attributes					
	Attribute C1(a) F	is of: Component itted In Variants: E	ts iU				
V	Name	Part Name	Family	Footprint	Variant Id	Category	
	C1	C 1.0pF	Generic/CAP	С	a N	Capacitor/Generic	
	C1	C Polarised	Generic/CAP	CPOL	bК	Capacitor/Generic	
	C1	C1812	Generic/SMC	C1812	с	Capacitor/Generic/	
	C2	C1812	Generic/SMC	C1812	а	Capacitor/Generic/	
	C2	C1812	Generic/SMC	C1812	C	anacitor/Generic/	

Swap Component Positions

There is a new interactive mode, **Swap Component Positions**, in the **Schematic** and **PCB** design editors on the **Utilities** menu. In Schematics it can be used on **Gates** and **Connector** pins.

When you enter this mode, components are selected as you pass over them.



If you click to select the two components to swap positions. In PCB there is an option **Swap Routed Components** on the context menu. If enabled the last segment of a track attached to the component will be unrouted prior to swapping the component positions. Component **Rotation** and **Side** is also swapped. If two components are pre-selected they will be swapped in a single shot action.

If the components are different sizes, their positions are swapped such that their centres are as close as possible but with their positions still on the component grid.

If the component you want is underneath the currently selected one you can select it by using the **Select Next** or **Select Previous** command on the context menu. When the correct component is selected click on the selected component that you wish to swap.

Part Edit Changes

Footprint Dialog Filter

In the **Footprint** dialog in the **Part Editor** it is now possible to use a wildcard filter of the footprint name. The number of pins is also shown but is fixed at the number of pins defined for the footprint unless it is a new footprint for which the number of pins is as yet undefined.

E Footprints		\mathbf{X}
In Library:	In Part:	
C (footprints) C0402 (sm footprints) C0504 (sm footprints) C0603 (sm footprints) C1206 (sm footprints) C1201 (sm footprints) C1812 (sm footprints) C1825 (sm footprints) C2512 (sm footprints) C3216 (sm footprints) C3216 (sm footprints) C3218	▲dd>>> C (footprin CO803 fsr C1206 dsr C1206 dsr C1210 fsr Down	nts) m footprints} m footprints} m footprints} m footprints}
C3527 (sm footprints) C3528 (sm footprints) C6032 (sm footprints) C7227 (sm footprints) C7243 (sm footprints) C7257 (sm footprints)	Filter	s: 2 Apply
CAPP2// (footprints)	С	Cancel
Pre <u>v</u> iew		

Once the naming and wildcard have been used, use the Apply button to update the selection list.

Ability to Resize Footprint/Symbol Dialogs

The **Footprint** and **Symbol** dialogs in the **Part Editor** are now resizable. These dialogs are available when choosing a Footprint or Symbol to add to a Part.

Footprints	
In Library:	In Part:
15-43-6563 (footprints) 15-43-8541 (footprints) 15-43-9510 (footprints) 21-0020A (maxim) 221-0249A (maxim) 42410-6456 (footprints) 42878-9370 (footprints) 43202-4104 (footprints) 43202-6107 (footprints) 43202-6107 (footprints) 43202-6107 (footprints)	Add >> C (footprints) COB03 {sm footprints} COB05 {sm footprints} Up C1206 {sm footprints} Down C1210 {sm footprints}
520470-4 (footprints) 53047-02 (footprints) 53048-02 (footprints) 53261-02 (sm footprints) 53398-02 (sm footprints) 555165-5 (footprints) AMP-MTAP0ST-2 (foo	Filter No. Pins: 2 Apply OK Cancel
1 0-	 0 2

Resize the dialog by picking and dragging the bottom right hand corner of the dialog using the gripper.

Change Connector Part Type option

On the **Edit** menu in the **Part Editor**, it is now possible to change an existing **connector** part to be a **normal** part using the new **Change Part Type** option.

Change Part Type 🛛 🛛 🛛				
Erom:	Connector	ОК		
<u>T</u> o:	Normal Part 🛛 👻	Cancel		
70.		Cancel		

Once selected, the Gates page appears and normal part functionality within the Part Editor is restored.

Delete Segment on Unextended Delete

Display Edit Track Edit Shape File Extensions Find General ♥ Interaction Move Macros Multi-Screen Online DRC Warnings	Select Select Tight Groups Minimum Pick Tolerance Drag Along Shape Selects Path Between 2 Points Control Drag Does Duplicate Frame Select Select If Completely Framed Alt Drag Does Frame Select Select Error Markers
	Copy Copy Single Cutout As Shape Delete Unextended Delete - Deletes Segments Optimise After <u>D</u> elete

Unextended Delete now has an entry to **Delete Segments** on the **Interaction** page of the **Options** dialog.

When used, this will remove just the selected segments and leave the rest. This can be used on **Tracks**, **Schematic Connections**, **Busses**, **Copper**, and **Documentation shapes**.

Deleting a segment on a **Closed** shape will make the shape **Open**. Deleting a segment in the middle of an Open shape will split it into two shapes.

When a Schematic Connection is split in two, the normal delete options determine the net naming of the two halves.

Net Class - Mark All Nets As One

In **Technology**, you can set a flag on a **Net Class** so that when one net using that net class is marked in the design (using the $\langle \mathbf{H} \rangle$ key), all nets using that net class are also marked. One use of this is to enable you to mark the full net path even when the path is split over a resistor for example.

Edit Net Class
Name: HSE Ţype: Signal ♥ ♥ Mark All Nets As One Styles Net Rules Size Limits Special Routing Attributes
C Track Length

This flag is also copied across from the Schematic when synchronising with a PCB design.

Replace Part Changes

You can now **Replace Part** by a specific representation, and also specify the resulting representation during replace part. The footprint and representation are now always used as part of the selection criteria for the components to be replaced (in previous versions, they were only relevant if the Part was not changing).

	Replace Part 🔀				
	Replace This:				
	Part: Part Representation				
	Desc: Part Representation Pins: 16				
	Family:				
	Footprint: <any footprint=""></any>				
\square	Rep: KAny Representation>				
	<pre></pre>				
	Part: Part Hepresentation Change Database				
	Desc: Part Representation Pins: 16				
	Familin				

Insert Multiple Item Changes

Some improvements have been made to aid the insertion of multiple items, mostly to ease the adding of **Groups** of Schematic **Connector Pins**. Some of these changes affect **Arrange Multiple Items** and **Replicate** as well.

Define Offset and Stagger Values as Grid steps

You can now define the **Offset** and **Stagger** values as **Grid Steps** (using the **current grid** in the current interactive operation).

Insert Multiple Connector Pins
Step Offset X: 300.0 Y: 100.0 O Distance Grid Steps
Number of Items × 1 Y: 10 Vi All Items In Component 10
Insert Order Insert Rows Insert Columns Reverse Order Switch Direction at end of Row/Column
Stagger Alternate Row/Column Stagger Pitch: 50.0 • Distance Grid Steps

Reverse Order

You can check a box **Reverse Order** to make the offsets reverse. You can still use negative offset values but this makes it more obvious.

Insert Multiple Connector Pins	X
C Step Offset	
X: 300.0 Y: 100.0	💿 Distance 🛛 Grid Steps
Number of Items	
X: 1 🗘 Y: 10 📚	All Items In Component 10
- Insert Order	
Insert Rows O Insert Columns	Reverse Order
Switch Direction at end of Bow/Col	LIDD

All Items In Component

You can select an **All Items In Component** check box to place all remaining Schematic Gates or Connector Pins in the current component being added. The number of items left in the component is shown in the edit box next to it. Checking this option will change the item counts to have a row or column with the correct number.

Insert Multiple Conne	ector Pins		X
X: 300.0	Y: 100.0	 Distance 	O Grid Steps
Number of Items	Y: 10 🗢	All Items In Com	ponent 10

Rearrange Multiple Items – Total Count

Using **Rearrange Multiple Items** you can now see a total **Item Count** of the number of items in the selection.

Rearrange Multiple C	onnector Pins	
X: 300.0	Y: 100.0	⊙ Distance ◯ Grid Steps
Number of Items	Y: 3 🗘	Item Count: 10

Change angle and mirrored state

In the **Insert Multiple** and **Rearrange** dialog, you can now change the individual **Angle** and **Mirrored** state of each of the items being added. This allows you rotate the connector pins to be the correct direction.

Item Orientation Angle: 0.0	Stagger Pitch:	50.0	Uistance	U una steps	
Angle: 0.0 <u>M</u> irrored	Item Orientation				
	Angle: 0.0	<u>M</u> irrored			

Auto-Rename in Schematic Editor

The Auto-Rename feature now supports rename **By Location** for a Schematic design as well as a PCB design. This option is available when also using the schematics **By Page** option and will rename items on the same schematic page sequentially according to their position on that page.

For multi-gate components the most relevant gate, as dictated by the current **By Location** settings, will be used to determine the item's position in the rename sequence.

Auto Rename 🛛 🔀
Component 🗸
Which Items: All Items 🔽
Include Locked Items
This Stem Only
Stem: B
Rename to: B
Start At: 1
☑ By Page
Rename Current Page Only
Bange: 1
Increment By Range Un All Pages
By Location
Direction
Left to Right 🗸 🗸
then:
Top to Bottom
Channel <u>W</u> idth: 100.0

By checking the **By Location** box, you can rename Components or Test Points by location, within each page. This enables you to produce a name sequence based on the position of these items on the schematic pages. This style of rename is commonly used for the service engineers convenience when attempting to diagnose problems using a schematic diagram as reference.

Schematic Bus can have Own Colour

A **Bus** can now have it's own colour. By checking the **Own Colour** check box and selecting a colour, you can override the general Bus colour defined in the Colours dialog.

Properties: Bus - Bus	
Segment Bus Bus Attributes	
□ <u>N</u> ame: ADD [0-9]	▶
Apply the style to all connected or same na	me busses
Line <u>S</u> tyle: Bus	V Uwn Colour
<u>W</u> idth: <u>50.0</u> L <u>e</u> ngth: 725.0	

The **Bus Properties** and **Change Style** dialogs can propagate the **Style** and **Colour** changes to all connected or same name busses by using the new check box.

There is a default in the Design Settings dialog, Bus defaults for this option.

Design Settings - Defaul	ts - Bus
 Defaults Attribute Block Bus Component Connection Doc Shape Embedded View Error Junction Net Origin Pin 	Line Style: Bus Name Style: Attributes Bus Terminal Offset: 100.0 Bus Terminal Direction: Up (on vertical bus segments) V Right (on horizontal bus segments) V In Dialogs - Apply Styles to All Connected or Same Name Busses

Show Net Name on Change Net

Defaults dialog - Net Name positioning

The **Defaults** dialog has a new option on the **Nets** page for **Net Name positioning.** These define the required **Offset** of the name from the pin, junction or bus terminal end, whether it can be **Auto-rotated** to lie along a vertical connection, and if the **Text Alignment** can be altered to be closest corner of the name to the pin.

Design Settings - Defaul	lts - Net	
Component	Net Class: Signal Name Style: Net Names	
Connection Doc Shape Embedded View Error Junction Wet	Net Name Position X Offset 100.0 Y Offset 100.0	
Origin Pin Report Symbol Star Point Testpoint	✓ Auto Rotate ✓ Auto Adjust Text Alignment	

The **Show Net Name**, **Add To Net**, **Add Connection to Bus** and **Apply Pre-defined Signals** options are other interactive operations that add Net Names, and they will also obey the new default name positioning rules.

Show Net Name on Change Net

A new option on the **Change Net** dialog to **Show Net Name** has been added. This is available if the Net Name is a user defined name. If checked, the name will be displayed on the nearest node to the last picked position on the selected sub-net. If the node already has a hidden net name it will be used, otherwise it will add a new net name and its offset from the pin will be determined by the new net name defaults mentioned above.

Change N	let 🔀
	Choose From All Nets In Design:
Net <u>N</u> ame	ADD9 🔽
	✓ Local Net
Net <u>C</u> lass:	Signal 🗸
	Change Name Of <u>S</u> ubnet Only
	Show Net Name

If there are no attached connections to resolve the name direction, and the nearest node is a pin, it will work out the direction from analysing the symbol to see which side the pin is on. The new Net Name will have its offset extended if it needs to avoid the displayed Pin and Logic name.

The **Show Net Name** option still has two modes of adding a net name that will vary slightly from **Change Net**; if a single connection is selected it will attempt to add a new net names at the selected point on the connection, and if you pick the free end of a connection, the Net Name will be added to a new junction in a direction away from the connection (rather than along it).

Pad Style Spacing Shapes

New Spacing Shape Pad Exceptions within Technology allow additional spacing around pads to be defined. This could be used, for example, to allow extra spacing around pads on small components which may slip when being placed on the board during manufacturing.

To use this facility, edit the **Pad Style** and add a pad exception using **By Layer:**. Edit its **Type:** to **Spacing Shape**.

Note, in the example below, Micro-Via entries would only be available to you if you have the Advanced Technology cost option.

Edit Pad	Style By Layer 🛛 🛛 🔀
<u>N</u> ame:	Round (125)
<u>T</u> ype:	Spacing Shape
Layer:	By Layer Spacing Shape
-Shape:	Micro-via Entry Pad k Micro-via Stop Pad

When viewed in the list of **Pad Styles**, this is displayed as a **Spacing Shape** and the layer it represents. You could have two exceptions on the same layer, one for the **Pad Shape/Size**, and one for the **Spacing Shape**.

		Name	Layer	Shape	Width	Length	Drill Hole	Plated
1	Υ	Oval (60 x 110)		Oval	60.0	110.0	32.0	<
Ī	Υ	Round (60)		Square	60.0		32.0	
	Y	Round (70)		Round	70.0		42.0	
ľ	Y	Round (125)	_	Round	125.0		50.0	V
ſ		Spacing Shape	Тор	Round	225.0			
+	¥	Square (60)		Square	60.0		35.0	V
ľ	Y	Via (40)		Round	40.0		24.0	~
Ī		Via (50)		Round	50.0		32.0	V
Ī	Υ	Via (60)		Round	60.0		32.0	

Once the exception has been added, all options that can use additional spacings, such as **Copper Pour**, **DRC** etc. will use the **Spacing Shape** as well.

Spacing Shapes Colours

In the Colours dialog, Others, there is now a new option for Spacing Shapes.

Colours - Others					
Layers Elec Shapes	Name	Displayed	Selectable	True Width	Colour
Doc Shapes	Connections	>	Image: A start and a start		
Pads	Symbol Origin		V		
Boards & Areas	Relative Coords Origin	V	V		
Vias	Coordinate Origin				
Text	Background				
Attributes	Bitmaps				
Highlights	Spacing Shapes				
Others					
Nets					

The example below shows the three big pads in the centre of the design after pouring copper and with the Spacing Shapes displayed. Additional clearance (using the Spacing Shape) has been applied to this pad style, copper pour has taken this into consideration.



User Defined Pad Shapes and Spacing Shapes

Spacing Shapes can also be used within the **User Defined Pad Shapes** option. For a selected area, from the context menu, select the **Toggle Spacing Shape** function to make the shape a **Spacing Shape**. Spacing Shapes should only be defined on an Electrical layer or the through hole layer, and there should be only up to one shape on each layer. This will be checked when you attempt to save the pad style.

		Select	•	
		Edit	•	
	8	Move		
	¥.	Type Coordinate	-	
	đ× đ¥	Type Offset Shift+	= \	۱.
	a	Lock		Ł
(()		Change Layer	L	1
		Change Shape Type		l
λ				1
		Toggle as Spacing Shape		
$\langle \rangle$		Resize Shape		
\sim		Edit Circle		
		Enter Radius		
		Enter Diameter		
	H.	Highlight Selection		
	8	Properties	I	

Properties and Spacing Shapes

When using **Properties** of the Pad, Via or Mounting Hole, the **Connect Type** can now be a new item called **Thermal [Ignore Spacing Shape]**.

If the **Pad Style** has a **Spacing Shape** defined for the current layer, then this will be used to determine the shape around the pad, instead of the actual pad land shape. This can be disabled by selecting the connection type **Thermal Pad (ignoring Spacing Shape)**, this does not disable the **Spacing Shape** for any other spacing or rule checking process.

Properties: Pad - Pad 📃 🗖 🔀
Pad Pad Attributes Test Component Comp Attributes Net Net Attributes
Name: 2 Logic Name:
Position: 2100.0 3175.0
Angle: 90.0
Layer: <through board=""></through>
Alternate Pad Style
Name: Round (125)
Width: 125.0 Shape: Round
Length: 125.0 Drill: 50.0 Id:
✓ Plated
Power Plane Connection: Default
Default
Not Isolated
Thermal Pad (ignore Spacing Shape)

This rule is also available in the Technology option, DFM/DFT page and Thermal Rules.

Measure and Spacing Shapes

If one of the items is a pad, via or mounting hole and its style has a **Spacing Shape** defined the **Use Spacing Shapes** check box will be shown. If checked, and the spacing shape is on the current layer being measured, the spacing shape will be used instead of the actual pad land shape. if this is the case, the pad information will also be changed to show the spacing shape size.

Width: 3.43 Height: 3.43 Drill Hole Size: 1.27 Layer: <through board=""></through>	<	
To: Pad R41.2 on net N016		
Width: 3.43 Height: 3.43 Drill Hole Size: 1.27 Layer: <through board=""></through>	<	
Layer: <through board=""></through>	∨ pes	<u>/</u>
Dist. Short Gap	Drill	N

Changes to 3D Viewer

New 3D Package Shapes

New 3D package shapes have been added for BEAD, MELF, DILSwitch and LEDdisplay.



Axial Components - Rounded Ends

For Axial components you can choose to round off the ends of the components using the Rounded Ends check box.

_				
E	dit 3D Package			
	—Package Definiti	on:		Applies to Sy
	Package Style:	Axial	~	R
	Height:	0.00 🛟	mm	Ales Applies
	Inside:	0.50 🛟	mm	AISO Applies
	Standoff:	0.50 🛟	mm	
	Body Colour:	Goldenrod	~	
	🗹 Top Closed		Rounded Ends	
	Top Coloured			Preview -
	Top Colour:			

Colour-coded Bands on Components

Package Styles of **Can** and **Axial** can now be given dynamically colour-code stripes (bands) on the component body. The colour-code bands are based on the Attribute value of the component.

Edit 3D Package		
Package Definit	on:	Applies to Sy
Package Style:	Axial 💌	R
Height:	0.00 🛟 mm	Also Applies
Inside:	0.50 🛟 mm	AISO Applies
Standoff:	0.50 🛟 mm	
Body Colour:	Goldenrod	
Top Closed	Rounded Ends	
Top Coloured		Preview
Top Colour:	✓	- jonen
🔽 Stripe	✔ Near Pin 1	
Stripe Colour:	Charcoal	
Pins:		

CAN shapes will get a stripe up the side of the body near to pin 1 or pin 2 depending on the pin 1 checkbox. **Axial** components get a single stripe near one end or the other (Capacitors, Diodes etc.), except for **Resistors** and **Inductors** which will get the correct set of coloured stripes depending on the value of each component.

The **Value** attribute is read and converted to a standard colour coding. Values in formats such as 2.2K or 2K2 or 10u are acceptable.

For **Resistors**, a **Value** or **R** attribute is read and converted to a stripe, and for **Inductors**, the **Value** or **L** attribute is read and converted to a stripe. For both types, the **Tolerance** or **Tol** attribute is used for the tolerance band.

3D View - Print With White Background

The background of the image saved (using **Save As Image**) will be white if the setting for **Print with** white background is enabled in the **3D Settings** dialog.

3D View Settings - Settings			×
3D View Settings - Settings Colours Settings Sizes Board (Substrate) Thickness: Layer Drawing Thickness: Gap For Exploded View: Options ✓ Exploded View of Layer Sta	0.275 0.001 3.000	Units Units on this dialog: mm	
 Transparent Substrates Include Areas In Picture Show Drill Holes 			
Show Wires Use TrueType Fonts Print with White Backgroun	d		

3D View - Save As Image

From within the 3D view screen, you can now write the current 3D view out as an image file using the option **Save As Image** from the **File** menu. **Save As Image** supports Bitmap Files (*.bmp), JPEG Files (*.jpg) and TIFF Files (*.tif).

Layer Type Choice for Component Body

There is a new selection the **Settings** page of **3D Settings** dialog to choose the **Component Outline** Layer Class.

This specifies the name of a **Layer Class** to be used when determining the 'size' of the body of the component. If this Layer Class name is specified, only shapes in the footprint that live on a layer of that class will be considered. The default value is to have no Layer Class name defined, which then allows any shapes to be matched and used for generating the 3D component bodies. To remove a previously specified Layer Class name, simply delete the text from this box.

Find Bar – Drill Sizes in Table Units or Design Units

There is a new option on the **Find** bar context menu to choose between displaying **drill sizes** using the **drill table units**, or using the **design units**.

Find		- д Х
	x	× 🕅 🗂 🛔 👷 🖼
Drill Size		~
23.6 thou		
23.6 thou 35.0 thou	~	Eloating Docking Auto Hide
	~	Linde Use Drill Table Units
	X	Elash Found Item <u>Gentre View on Found Item</u> Add Found Item to Favourites

The units text is now also shown after each drill size.

This option is also available on the Find page of the Options dialog allowing it to be pre-set.

Options - Find		
Display Edit Track Edit Shape File Extensions Find General Interaction Move Macros Multi-Screen Online DRC Warnings	Action On Found Item Select It Flash It Centre View On It Allow Find Under Sliding Bar Add To Favourites List Flashing Items Flash For: 1 seconds	Finding Nets ✓ Include Default Nets Finding Error Markers ✓ Include Locked Errors Only List Errors In The Design Finding Drill Size ✓ Use Drill Table Units

Spacing Wildcard Match

The wildcard match string for **Net Class** names in the **Technology** dialog and **Spacing Rules**, can now be a blank string. This means it matches just items not on a net or on a net with no Net Class. You can therefore define specific rules for items that are on no net, items such as Copper or Free Pads with no Net Name or Net Class.

	Edit Class To Class Spacing Ru	le	
/	Match Classes HSE		JK Incel

Warnings	×
An empty class name will match nets with no net class and items not on	OK
	Cancel
	Report
	Warnings <u>O</u> n/Off
×	Do not tell me again

When you add a rule using a blank string, a warning dialog is displayed:

Spacing Wildcard Match Rules

With the blank string rule now available, below is a reminder of the string rules available to you:

* - matches everything on a Net or not.

?* - matches everything that has a Net Class.

Blank string - matches anything with no Net or no Net Class.

Named string - matches a specific Net Class.

Wildcard Strings

All wildcard strings can now use | for or. So when used, power|gnd* would match power or gnd*.

Spacing Rule Class to Class Match

Technology | Spacings

From within the **Technology** dialog and **Spacings**, **Match Net Class Pair**, you can now specify a single rule which matches both **Outer** layers (Top and Bottom).

New Class To Class Spacing Rule	X
Match Classes	OK Cancel
On Layers Side:	
Vame: Outer Top Unner Within Areas: Bottom	

Technology | Net Styles

From within the **Technology** dialog and **Net styles**, you can now specify a single rule for **Track Styles** which matches both **Outer** layers (Top and Bottom) for the **On Side:** style.

Edit Net Styles
For Nets with Net Class: GND
Within Areas Named: *
Define Default Track Styles For Tracks:
On Side: Duter or www.selfacture.com On Layer: Outer
Top VS Default Track St Inner Name: Track (30)
<u>₩</u> idth: 55.00

Copy Pad Style

There is a new button on the **Technology** dialog and **Pad Styles**. A **Copy** button has been added, this copies the currently selected pad style and all the **By Layer** exceptions, if the list is expanded.

	Name	Layer	Shape	Width	Length	Drill Hole	Plated
Y	PadStyle1		Round	60.00		32.00	 Image: A set of the set of the
Y	PadStyle2		Oval	60.00	110.00	32.00	Image: A state of the state
Y	PadStyle3		Square	60.00		35.00	V
Y	PadStyle4		Round	70.00		42.00	~
Y	PadStyle5		Round	125.00		50.00	~
Y	Rect (0.6mm x 2.2		Rectangle	23.62	86.61	0.00	
	Star Point (25)		Square	25.00		0.00	
Y	Via (40)		Round	40.00		24.00	Image: A state of the state
Y	Via (50)		Round	50.00		30.00	V

Suppress Pads on Powerplanes

There is a functional change to be aware of, this effects existing designs as well as new designs but is controlled using a switch. All processing, **Powerplanes**, **Copper Pour** and **DRC** option now take suppressed pads into account. All checking uses the drill size (if there is one) rather than the pad size. This applies even for unmodified existing designs, this will result in a smaller ring around suppressed pads on powerplane layers. There is no interface change as such for this update. Pad suppression is set up on individual layers.

By default, the switch is set to work as it did previously. If however, you uncheck the switch, it works in the new way described above.

Text Styles	Maximum: 0.0	
Hatch Styles		
🔁 Rules	State of the second	
Spacing Rules	Drill Holes	
🔶 Design Rules	Drill to Drill Space	Drill to Board
DFM/DFT Rules	Minimum 10.0	Minimur
Differential Pairs	Minimum: 10.0	PHILING
🔁 Nets	Allow Coincident Holes	- Micro-via Dr
Net Names	Only If Some Size	Minimum
Net Classes	Uniy ir Same Size	Minimu
Net Styles		
Pin Networks	Check suppressed pads as thou	ugh pad is there.
🚔 Lavers	l.	

Highlight Locked Tracks

There is a new option on the Colours dialog, Highlights tab named **Locked Track Segments**. By selecting the check box and choosing a suitable colour, locked track segments in the design can be highlighted.

In the Footprint editor, this feature is shown as Locked Breakout Segments.

Elec Shapes	Name	Displayed	Colour
Doc Shapes	Selection		
Pads	Highlight		
oards & Areas	Marked Net		
ias	Online DRC		
'ext	Clearances		
Attributes	Weld Spot		
Highlights	Weld Spot (Direct Hit)		
Others	Test Points (Top)		
Nets	Test Points (Bottom)	 Image: A start of the start of	
1005	Test Points (All)		
	Component Pad 1		
	Unconnected Pads		
	Unfinished		
	Variants		
	Not Fitted		
	Locked Track Segments		
	Differential Paired Tracks	V	
	Attached Dimensions/Callouts		
	Highlight 'Pass'		
	Highlight 'Warning'		
	Highlight 'Fail'		
	Highlight 'Linchecked'		

In the Schematic editor, this option is shown as Locked Connection Segments.

Draw Pads in Top Most Layer Colour

There is a new switch in the **Display** dialog under **Options**, **Draw Pads in Top Most Layer Colour**. This causes pads to be drawn in the top most visible colour instead of the *Side* or *Through Board* colour. A through-hole pad might be drawn in the Top Electrical layer colour if this were the *top most* visible layer.

	PCB Drawing	
	Draw Drill Holes: Always	✓ Translucent Copper Transparent Opaque
	Merge Colours	Reset
	 Step Urthogonal PCB Connections Highlight Tracks Using Stripe 	Draw Hollow Segments when 'True Width' off
<u> </u>	Simulate TrueType Fonts (True Scale)	Copper Segments
>	☑ Draw Pads in Top Most Visible Layer Colour	Track Segments
V		Other Shape Segments

Net Styles Dialog Changes

Stop Spans Being Used For Vias in Net styles dialog

Using the **Technology** dialog and **Net Styles**, you can now specify that a particular layer span should not be used for automatically inserting vias during manual routing.

When editing a **Net Style**, you can check the **Vias Not Allowed** box. This is used when automatically inserting a via (or changing its span) by changing track layers the spans marked with **No Vias** in the **Net Styles** entries will not be used and the next best via span will be used.



Layer Spans set to Any Spans

There is also a slight change in **Net Styles** in that the layer span can be set **as <Any Spans Not Through Hole>**, so the via style, via protection flags or **Vias Not Allowed** flag can be applied to all spans except the through hole span in net styles entry.

▼ × For Nets of Type: <a< td=""><td>ny> 🔽</td><td>ОК</td><td></td></a<>	ny> 🔽	ОК	
× (* wildcard characters	**' and '?' allowed)	Cancel	
🗹 Define Via Defaults			
For Vias with Layer Span:	<any></any>	~	4
Vias Not Allowed	<any> <any hole="" not="" spans="" through=""></any></any>		
Define Via Protection	Die Core Top > Die Core Bottom		Ň
✓ Define Default Via Sty	Inner 2 > Die Core Top Top > Die Core Top Top > Inner 2		
Name: Via (40)		v	

The combination of these two features enables you to easily specify for a particular net class, or in a particular area, that vias can only be added with through-hole spans.

Via Protection Against Automatic Changes – Default Settings

From the Setup menu, the Defaults dialog and Via Defaults page now has a Via Protection Against Automatic Changes section with switches Allow Delete if not Routed and Allow Reduce Layer Span. These will be used for vias added whilst changing track layers.

Design Settings - Default	s - Via			X
Defaults Area Attribute Bitmap Board Component Construction Line Copper Dimension Dimension Units	Layer Span: Pad Style: Via Protecti	<through hole=""> Via (40) on Against Automatic Changes — V Allow Delete if not Routed V Allow Reduce Layer Span</through>		

Note: the Add Via and Add Testpoint functions will still always add fully protected vias, and there is still an interactive option End On Via – Adds Protected Vias, but if you switch it off the new defaults will be used.

The via protection for new vias can also be overridden for particular **Net Classes**, or for particular **Layer Spans**, or within specified areas using the **Net Styles** dialog. You will see a new **Via Protected** column, which will have text in if the protection has been set for an entry. To change the via protected state **Edit** the row and in the **Edit Net Styles** dialog, check the **Define Via Protection** check box and choose the required protection.

	For Vias with Layer Span: (Any)	
	Vias Not Allowed	
	☑ Define Via Protection: ☑ Delete if not Routed ☑ Reduce Span	
1	🔽 Define Default Via Stule	

Add Track Option – Use Style From Picked Track

In the Options dialog under Edit Track, there is a new option Use Style From Picked Track.

Options - Edit Track	
Display	Segment Mode: Angled (45)

If enabled, when adding a PCB track and <u>starting on another track</u>, the new track will take its style from the track segment being picked.

New Context options in Layers Bar



Two new options have been added in the Layers browser on the context menu to set default layers.

Use **Set As Default Track/Copper Layer** when an electrical layer is selected. Note it will also set the default template layer. Use **Set As Default Documentation layer** when a documentation layer or non-electrical layer is selected. Setting the default documentation layer will set the default layer of most items that are added for documentation purposes, except **Dimensions** and **Construction Lines** that are usually kept on their own layers.

New Commands for Change Layer

A new set of commands are available to change the layer of the selected item directly to an electrical layer. In the Customise dialog, the commands are labelled **Change To Layer 1** to **Change To Layer 10**, layer 1 being the Top Electrical layer. You can assign keys to these commands (such as 1 through to 9).

Customise					
Commands Toolbars Tools Menu	Keyboard Application Look Options				
Category:	Filter				
All Commands 🗸 🗸	change to				
C <u>o</u> mmands:	Context: <all></all>				
Change To Layer 2	Annly				
Change To Layer 1					
Change To Layer 2	Assigned Shortcuts:				
Change To Layer 4					
Change To Layer 5	2 <u>Remove</u>				
Change To Layer 5	Reset All				
Description:	Press <u>N</u> ew Shortcut:				
Change to physical layer 2	Assign				

Once assigned to a key, use **Change To Layer 1** to change to the Top Electrical layer, **Change To Layer 2** to change to the next electrical layer and so on. These commands are only available if the selected item can exist on an electrical layer.

Component Pushing uses Accurate Shapes

When pushing components, it takes the actual component shapes and pads into account. This is particularly useful for round components, and through-hole versus surface mount components where accurate shapes are used for the pushing boundaries.

IDF Output Enhanced

There is a new check box on the IDF Output dialog **Include Non-Round Holes as Board Cutouts**. Check this to output pad, mounting hole and via slots as cutouts of the board outline. Slots in vias will only be output if existing option **Include Vias in Output** is also checked.

IDF Output			
	◯ Version <u>2</u>	• Version <u>3</u>	
Board Filename			
C:\Project\HSE.idb		Browse	Cancel
Library Filename			
C:\Project\HSE.idl		Browse	
Allow Spaces in <u>F</u> ilenames	:	Include Vias in Output	
Software <u>A</u> rcs		☑ Include <u>N</u> on-Round Holes as Board Cutouts	5
		Include Unglaced Components	-
Board Thickness: 1.600		Default Component Height: 5.000	

There is now an option in the dialog to **Include Unplaced Components**. In previous versions, it always included unplaced components.

IDF Output			X
	◯ Version <u>2</u>	• Version <u>3</u>	
Board Filename			
C:\Project\HSE.idb		Browse	Cancel
Library Filename			
C:\Project\HSE.idl		Browse	
Allow Spaces in <u>F</u> ilenames	;	Include ⊻ias in Output	
✓ Software Arcs		✓ Include Non-Round Holes as Board Cutouts	
		✓ Include Unplaced Components	
Board <u>T</u> hickness: 1.600		Default Component Height: 5.000	

You can now enter a **Default Component Height** value in the IDF Output dialog to be used for components that don't have a height value. In previous versions, the output set the height to zero if it didn't exist in the component. The Units used will be design units.

IDF Output				
	O Version 2	• Version <u>3</u>		
<u>Board Filename</u>				
C:\Project\HSE.idb			Browse	
				Cancel
Library Filename				
C:\Project\HSE.idl			Browse	
Allow Spaces in <u>F</u> ilenames		Include ⊻ias in Output		
Software <u>A</u> rcs		✓ Include Non-Round Holes as E	loard Cutouts	
		Include Unplaced Components	:	
Board <u>Thickness</u> : 1.600		Default Component Height: 5.000)	

Component Shapes

There is now more choice for choosing what shape is output for components. It allows you to choose the **Component Body Area**, and lets you disable use of any shapes that you don't want, for example outlines that are not on the **Layer Class** selected.

	ise Autoute in Flace of Flanname.		Y
Co	mponent Shapes		
U	se the following to find the component shap	pe (applied in order):	
	🗹 Component Body Area		
	Component Outline with Layer Class:	Silkscreen	•
	✓ Largest Component <u>O</u> utline		
	Component Extents		

Multiple Board Outlines

If multiple board outlines, the main one (containing the others) is output in the BOARD_OUTLINE section, and the other boards are output as additional shapes in OTHER_OUTLINE sections with identifier BOARDn.

Overlapping Cutouts

During output, all overlapping cutouts in the board will now be merged together.

Import IDF Other Outlines

Import IDF will now read OTHER_OUTLINE shapes with identifiers starting with BOARD as additional board outlines.

DXF Import Enhanced

Scale Import Plot

The ability to Scale a DXF during import has been added. This control will help some DXF files where the scale is not as expected.

bon import	
Units Use Map File	
O Imperial: thou Vo DXF Mapping file selected.	
Metric: mm Browse	Save To
Scale: 1.000	sian Coordinate Origin

Align DXF 0,0 to Origin

Normally, the imported DXF is placed at the coordinates defined in the incoming DXF file. If you prefer, you can choose instead to align the DXF to the **Design Coordinate Origin** or the **Relative Origin** (if defined).

DXF Import			×
Units O Imperial:	thou	No DXF Mapping file selected.	
 Metric: 	mm	Browse Save To	
Scale:	1.000	Align DXF 0,0 to Origin: Design Coordinate Origin	~
- Shape Items		Design Coordinate Origin Relative Origin	
DVELours	Transfer Here	n Tumo I aver Cutente I aver	

To do this, simply check the **Align DXF 0,0 to Origin** check box, and choose the origin from the dropdown list. This feature can be used for example to load a DXF board outline into an existing design which already has a sheet outline outside where the board needs to be positioned. By placing your relative coordinate origin at the desired location for the board, the incoming DXF data will be shifted to that point.

Apply to Entire Column

When the cursor is over the **Item Type** or **Layer** columns, you can choose **Apply to Column** from the context menu to apply the current type or name to the whole column.

DXF Impor	rt							
Units O Imperi O Metric Scale	al: thou	 ✓ Use Map No DXF Ma ✓ Align DXI 	Use Map File No DXF Mapping file selected. Browse Save To Align DXF 0,0 to Origin: Design Coordinate Origin					
DXF Lay	/er Transfer	Item Type	Layer	Cutouts Layer				
0	Image: A start of the start	Doc Shape	l a on o Ista antina Caluma					
ALIGNME	ENT V	Doc Shape	iy to entire Column					
	-141	Dog ougbo						
_BOARD		Doc Shape	Top Silk Screen					

Copy Shape Layer Map to Text Grid

When the cursor is over an item in the Layer column, you can choose Copy Shape Layer Map to Text Grid from the context menu.

XF Import						
Units		Use Ma	p File			
OImperial: thou		No DXF M	No DXF Mapping file selected.			
Metric:	mm	~	Browse		Save To	
Scale:	1.000			Desi	Construction Optimized	
		Align DA	r o,o to ongin.	Desi	gri coordinate origin	
Shape Items						
DXF Layer	Transfer	Item Type	Layer		Cutouts Layer	
0	~	Doc Shape	Top Silk Sc	Annly	to entire Column	
ALIGNMENT		Doc Shape	Top Silk Sc	Copy	Share Layor Map to	Tout Crid
_BOARD		Doc Shape	Top Silk Sc	Сору	Shaper Map to	rext Griu

This maps the text layers to be the same as the shape layers.

DXF Layer	Transfer	Item Type	Layer	Cutouts Layer	
0	>	Doc Shape	Top Silk Screen		
ALIGNMENT	V	Doc Shape	Top Silk Screen		
_BOARD		Doc Shape	Top Silk Screen		
_THROUGH	V	Doc Shape	Pin Names		
_TOP_SIDE_	V	Doc Shape	Top Silk Screen		
⊙ Cutouts on S	ieparate DX	FLayers 🔘	Use DKF Cutouts	Placement Sites	Settings
 Cutouts on S Text Items 	ieparate DX	FLayers 🔘	Use DKF Cutouts	Placement Sites	Settings
 Cutouts on S Text Items DXF Layer 	ieparate DX	FLayers 🔾	Use DKF Cutouts	✓ Placement Sites	Settings
Cutouts on S Text Items DXF Layer 0	ieparate DX	FLayers O	Use DKF Cutouts Laver k Screen	✓ Placement Sites	Settings
Cutouts on S Text Items DXF Layer O ALIGNMENT	ieparate DX	FLayers O	Use DKF Cutouts Laver k Screen k Screen	✓ Placement Sites	Settings
Cutouts on S Text Items DXF Layer 0 ALIGNMENT _BOARD	Tran	Sfer	Use DKF Cutouts Laver k Screen k Screen k Screen	✓ Placement Sites	Settings
Cutouts on S Text Items DXF Layer ALIGNMENT _BOARD _THROUGH_BC	Tran	Sfer Signal States Stat	Use DXF Cutouts Laver k Screen k Screen k Screen nes	✓ Placement Sites	Settings

STEP I/O Improvements

Export Improvements

Component identifier tagging is now done automatically in files exported in STEP format from Pulsonix. This allows for identification when used in the MCAD system. Both Component Name and Part Name are exported.

Model Colours used in Preview

If the model you are referencing in the design has its own colours defined, the STEP Previewer will now use these colours. When the STEP file is written, the model colours will also be included in the output file for further use in the MCAD system.

STEP outputting Mounting Drill Holes

There is now an extra check box on the **STEP Output** dialog which allows **mounting drill holes** to be output irrespective of the **Advanced settings**.

	Output STEP
	STEP Filename: c:\project\design.stp Browse
Ν	Board Thickness: 0.627
	Mounting Drill Holes
V	Advanced Settings (these are likely to increase generation time and file size)
	Pads Vias Drill Holes
	□ Iracks □ Copper □ Doc Shapes □ Text
	Output Preview Cancel

STEP Export – No Model Transformations

The **No Model Transformations** setting is used if all your STEP models have been previously aligned to match your footprints rotation etc. This allows the transformations to be ignored. This option is selected in the **Settings** menu, **Design Settings** and **General**. This setting is made on a perdesign basis.

Report Symbol Star Point Template Testpoint Text Callout Track Variant Via Wire General Coordinate System Naming Placement Sites	Synchronise with Schematic Apply All Spacing Rules Apply Footprint Changes Ignore Attribute White Space Allow Update of Schematic to match PCB	STEP No Model Transformations
---	---	----------------------------------

STEP Positional Changes Imported

When Component positions have been changed in the mechanical system, these positional changes can now be imported back into the Pulsonix PCB design. It will compare the current positions of the components with positions of the components as described in the STEP file.

To import positional changes, from the **Utilities** menu select **STEP Positional Changes**. If there are any changes you will be asked if you to apply the changes.

STEP Positional Changes	×
2 Positional Changes have	e been found
Apply Changes	View Report
Can	ncel

Select **View Report** to see which components have changed. Select **Apply Changes** to move the components.

Design Rules Check – Check for Missing Teardrops

You can now check if a teardrop is missing due to a track corner preventing the teardrop from forming.



For example, teardrops may not be added because a track entering a pad has sufficient space or clearance to add it, or a corner into a via has insufficient length.



When DRC is run, any errors found will be reported with a Td error marker like this:

Missing Teardrop (Td) At (15849.0 4400.0). Layer 'Bottom'. Missing Teardrop (Td) At (16400.0 5400.0). Layer 'Bottom'. Missing Teardrop (Td) At (15025.0 5202.4-). Layer 'Top'.

Online DRC – Check Via in Pad

Online DRC – Check Comp to Comp

Two new Online DRC Options, **Check Via In Pad** and **Check Comp To Comp** to enable switching off these checks when performing online DRC checking.

Options - Online DRC	
Display Edit Track Edit Shape File Extensions Find General Interaction Move Macros Multi-Screen iv Online DRC Warnings	Online DRC Online DRC Online DRC Online DRC Online Drop (Check changed data after interactive operations) Online Drop Check Attached Iracks & Wires On Drop After Edit (Check item after having its properties edited) Add Error Marker (to show first error) Check Poured Copper Online DRC Check Via In Pad
	Check Comp To Comp

Check Via In Pad - Uncheck this option if you do not want Online DRC to check for vias within pads.

Check Comp To Comp - Uncheck this option if you do not want Online DRC to check the spacing between Components.

High Speed Option - Differential Pairs – Spurs Allowed Check

Within the **Technology** dialog and **Differential Pairs**, the Differential Pairs dialog now has a flag to say if **spurs are allowed**.

A spur is defined as a single path to an electrical item with no sub-spurs.

Differential Pin Pair			
First Pin Pair			
Net: HSE2			
Pin: U13.2			
Top Layer: 0.000			
Bottom Layer: 0.350			
Allow Track Spurs: 🗹			
OK Cancel			



This allows you to add spurs off without error as is shown in the two examples below:

When running **Design Rules Checks**, the **DRC** and **Differential Pairs Reports** are more consistent in reporting spur errors.

Changes to Calculators

The **Track Width**, **Resistive** and **Heat Sink** calculators can now take an ambient temperature that can be set below zero, in other words, a negative value.

Ariadne Import (Cost Option)

As part of the Advanced Import cost option, you can now import Ariadne Schematic designs, PCB designs and Libraries. Please contact sales or your local Pulsonix distributor for more information.