Pulsonix Design System

V3.0 Update Notes

Copyrights

Contents

Chapter 1. Getting Started With V3.0

About these notes

These update notes are provided for existing users as a supplement to existing Pulsonix Users Guide. These notes are to highlight new features in version 3.0 and to briefly describe their use.

Each chapter is broken down into logical functional descriptions based on the application type, Schematic design, PCB design etc.

Installing the new version

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

As with any other installation of Pulsonix, insert the CD and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen messages from the install wizard. You should install Pulsonix V3.0 on top of your existing installation, you do not need to uninstall any old version first.

New License

Version 3.0 requires a new license that will be supplied to you with the update or by email. For existing users it is recommended that you simply click the **No Change In Licensing** check box on the licensing page of the wizard. New licenses and changes to network licensing can be made after the installation using the License Manager.

New System Files

There is a change to the System files that are installed because of the updated development system used for Pulsonix 3.0. If you are installing from a previous version to version 3.0, (i.e. V2.x or V1.x), on the **Change Components** page of the installer, the **System** box will be automatically checked for you. **You must install the new system files in order to run Pulsonix V3.0.**

New Libraries

New libraries are supplied with version 3.0. These contain over 14,500 Parts with associated Symbols and Footprints.

New Parts are available from: Texas Instruments/Burr-Brown, ST, ON Semiconductor, Fairchild, Microchip, Altera, National Semiconductor, Analog Devices, International Rectifier, AVX, MuRata, Dallas/Maxim, Xilinx, Linear Technology, Cypress, Lattice, Actel, Quicklogic, Atmel, Molex, AMP, RF Devices, WAGO, Hamlin, 3M, Vishay, plus generic transistors, capacitors, resistors, diodes, connectors and fuses.

New Eagle ULP Conversion Files

In version 3.0, Pulsonix delivers new conversion ULP files for Eagle V4.11 and the previous ULP files for V4.09. To distinguish between the two sets of ULP files, the \EagleULP folder now contains two sub-folders for each version, \4.1 and \4.09 respectively.

Chapter 2. New Options

Introduction

This chapter covers all functional changes to Pulsonix. Specific application feature changes are shown in brackets after the general changes, e.g. (PCB).

Report Maker Improvements

New commands are available to support new functionality, such as Variants, and to expand the existing features.

List of Doc Symbols

This command is available for Schematic and PCB designs and will list all the Documentation Symbols used. Within this command you can also report various items about the **Pad** (or pin) and the **Pad Attribute**. The **Pin Count** reports the number of pins on a Documentation Symbol.

Within Schematics, all Doc Symbol types can be reported such as Signal References, Testpoints etc. You can refine the reporting of specific Doc Symbols by using the **Type** command and If.

List of Free Pads

This command is available for PCB designs and PCB nets only. It will list all the pads in the design or net, that have been added using Insert Free Pad. This does not include Component pads, vias or Mounting Holes.

Maximum layer

Used on a PCB design, this command will report the maximum number of electrical layers, no other layer types are included in the count. For example, for a design with a Top and Bottom side plus two inner powerplane layers, the maximum layer report will be 4.

Test Probe Side

In addition to the existing testpoint reporting, on a PCB design you can now report the test probe board side. The reported values are **top**, **bottom** or **all**.

Step & Repeat

Using this command will provide a repeat of all the following (indented)
commands for each copy, defined by a count in the X and Y directions.
Before the set of commands is repeated, all coordinates output are increased
by the **Step Offset** in the appropriate direction.

The **Step Offset** can be specified in the current design units or your own
units by using the **Coordinate Units** command.

The **Number of Copies** will repeat the command by the number specified in
the **X** and **Y** directions.

Default Footprint Name

This command appears for a Component or Parts or Component Group as a
selectable field. It is available in both Schematic and PCB design types. It is
used to report the name of the default footprint on the Part, regardless of
whether it is used or not.

By using this command, it is possible to list the Parts that have and haven't
used their default footprints (if changed by you in the design).

Transform Coordinate

The **Transform Coordinate** command us used to shift coordinates from their
original coordinate position in the design. This might be used to shift the
coordinates of everything reported by 5000 thou for example to align with
another set of coordinates. All coordinates following this command at this
level will use this transformation.
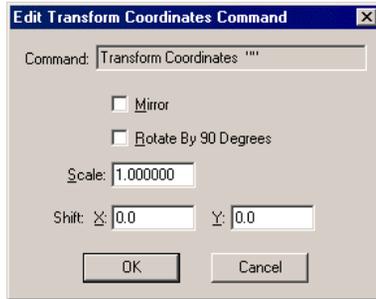
You can also specify a **mirror** and **90 degree rotation** of the coordinates.

Where a **Scale** has been defined, the scale will be applied before the shift is applied. The order in which the options are applied is: Mirror, Rotate, Scale and Shift.

List - New Line If Full

This command puts out a new line every 'n' items, where n is supplied by the you as the number required. For example, you might want to report the Name of a Net, then report the component and Pin Numbers that connect to it.
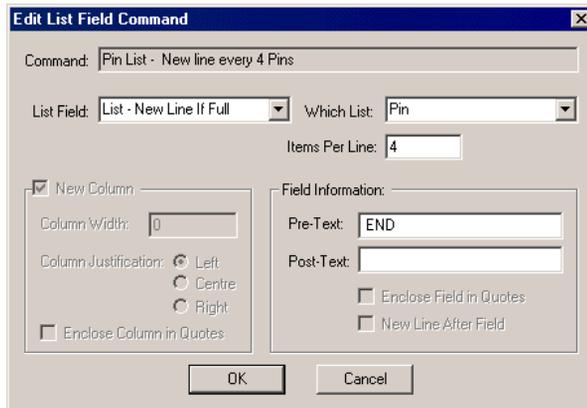
For some types of reports it is ideal to report the Component/Pin Numbers with up to 4 per line before starting a new line.
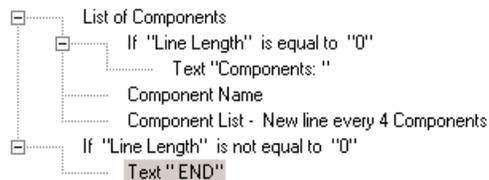

```
NET   GND
COMPONENTS U1.7 U3.7 IC1.8 R1.1 END
COMPONENTS C3.2 R4.1 U12.7 TR1.2 END
COMPONENTS X1.1 C11.2 END
```

Line Length

The Line Length command is used to get the current line length. As shown in the script below, you can test to see if the current line length is a specified size and then add text, for example, to the start of the line.

```
□---- List of Components
    □---- If "Line Length" is equal to "0"
        └---- Text "Components: "
    ---- Component Name
    ---- Component List - New line every 4 Components
□---- If "Line Length" is not equal to "0"
    └---- Text " END"
```
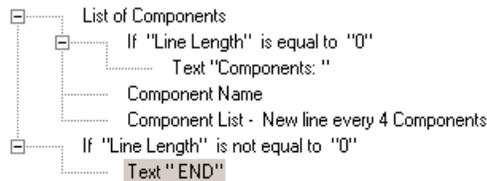
Comment

The **Comment** command is used to annotate the script for documentation purposes. When used, it adds a // image followed by a text string into the script. This string is ignored when the report is generated.

```
    ---- Prompt for Variable "NumberY" How many copies in the Y direction?
    ---- Set Variable "StepX" to 0
    ---- // This variable is used for checking for a positive number
    □---- If "Variable: NumberX" is more than "1"
        └---- Prompt for Variable "StepX" Enter the step in the X direction.
    ---- Set Variable "StepY" to 0
```
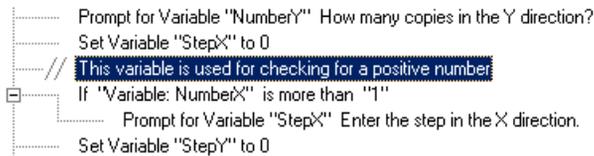
You may still use the text string command and disable it to also make comments in the script but this is now the recommended method.

Run from Cam/Plot

From within the CAM Plot option for a format file, you can now define a plot using the Scale, Rotation, Mirror, Position, Step & Repeat. This will also process the current variant as selected.

You might use this to align the output with other plots using transformations. By applying the transformations to the format file plot, you can generate an appropriate report without changing the Format File itself.

New If condition "Exists" for attributes

When using the **If** statement as a command, if the If is checking an attribute, it will now make the check based on whether the attribute exists on the item or not. If the If has been used on an X Coord or Y Coord then it checks if the item has a visible attribute position.

This way, the presence of an attribute on an item can be used as a condition to what is reported.

List of Variants

With the introduction of Variants to Pulsonix 3.0 (see later section), you can now report various items on Variants. Variants can be listed for a design or when reporting a component. This applies to both Schematics and PCB.

Within the items available under List of Variants, you can report **Variant Name** and **Variant Description**. These are taken from the Variants Manager dialog as entered by you.

Has Variants

For the support of Variants, a new command, **Has Variants** has been added. This can be used on a design to see if the design has any Variants at all, or used on a Component to see if it is participating in a Variant. If a variant exists, it reports True or False for the item.

This command is better used when combined with the **If** statement, for example, If Has Variants is equal to True do…. Else…

Set Variant

For the support of Variants, a new command, **Set Variants** has been added. When the command is added you are requested to select the variant name, this will be one of the names entered in the Variants manager dialog. By default, if Variants are in use in a design, a Report Maker report will act on the Variant selected as 'Current' in the design. Set Variant uses this command to base the report on the specific design variant named in the script.

An extra check box on this dialog also allows the variant selected to be processed as a **Variable Name** for use with the **Variable** command.

Is Fitted

This is a check used on a Component and it reports the fitted status as being True or False. This allows the Variant name to be supplied during the running of a report for the Variant that is current in the context of the report being generated. This is best used in conjunction with the If command to test that a Component is fitted or not.

Report Improvements

Each of the standard PCB reports that contain design lengths or angles can now be defined with their own units. On the **Reports** dialog, select a report and click on the **Own Units for Standard Report** check box, use the **Units** button to change the units.
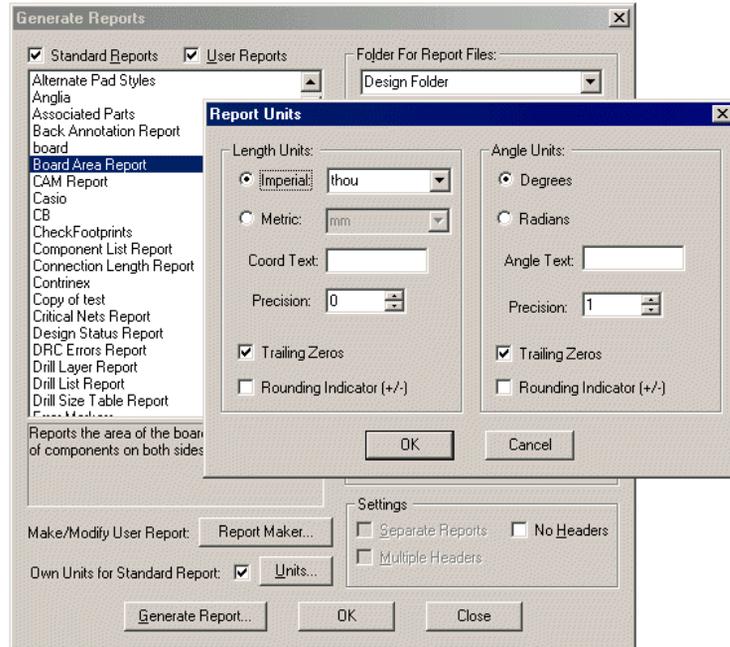
New and Changed Reports

Two new reports have been introduced – **Critical Net Report**. This reports any nets that use a Net Class that has a Net Class Rule defined, it also reports significant items such as track length etc. where that particular rule has been defined.

Variants Report. This is discussed later on in this section under Variants.

In version 3.0 there is a change to the report summary in the **Net Report**, **Net Completion Report** and **Critical Net Report**. The summary now appears at the top of the report so that it is visible without scrolling to the bottom of the report.

Star Points

Star or Delta **Points** allow you to connect multiple named nets together at a common point. This might be the GND and Digital_GND nets for example, where they have the same input point but need to be isolated from each other except for the source point.

Star points are used in both Schematic and PCB designs and are recognised by Translate to PCB and Synchronise Designs.

New Documentation Symbol Type

A new **Doc Symbol** type has been added to the Doc Symbol definitions - **Star Point**.
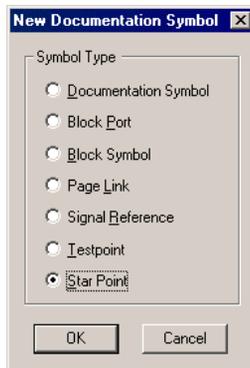
The Schematic Doc Symbol can contain single or multiple terminals depending on your requirement but the PCB Doc Symbol can only contain a single pad.

When used in the design, all the connections or tracks connected to the star point are connected together, but the netlist will still report separate nets.

Insert Star Point

There is a new option on the **Insert** menu called **Insert Star Point**.
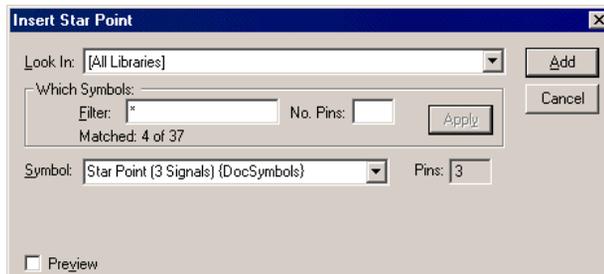
When selected, this will search the Schematic or PCB Doc Symbol libraries for the Doc Symbol type of Star Point. This options works in the same way as Insert Component etc. You can Preview the star point by selecting the **Preview** check box.

Schematics

The process of using a star points in Schematics is straightforward. The Doc Symbol is defined as being a Star Point type. This can have any number of terminals but the nets connecting to them remain independent regardless of whether a single terminal or multiple terminals have been defined.
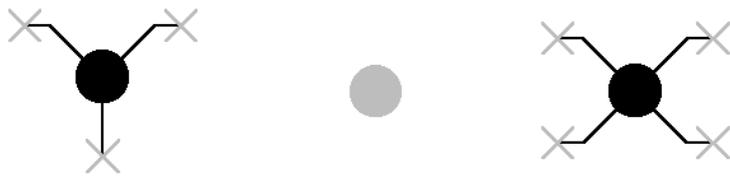
The illustration above shows 3 different star point types, others are available.

These could be used like this:
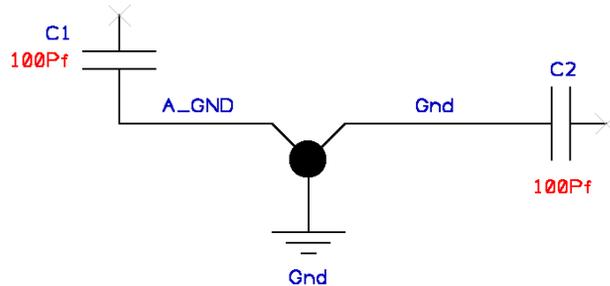
The **Properties** dialog for the star point will show the following:
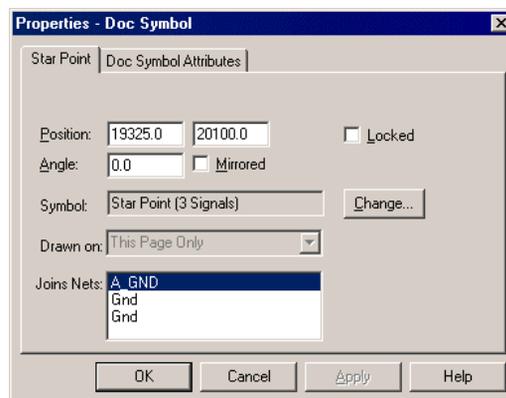
You can use the Properties dialog to **Change** the Doc Symbol and to display the net names (**Join Nets**) of nets connected to the star point.

Selection of the star point symbol will display the multiple net names on the status bar using the range format separator as defined in the **Design Settings** dialog.

Star Point | Nets: A_GND,Gnd | Grid: <Workir

PCB

The main difference between the Star Points used in Schematics and PCB designs is that the ones used in PCB always only have a single pad.

During the **Translate to PCB** stage the nets defined on star points are connected together at the star point. They still retain their unique net names but all connections emanate from the same Doc Symbol pad.

For example, the design might look something like this, the shape shown for the star point here is actually the silkscreen shape, the pad on the electrical layer is the same size as the tracks.
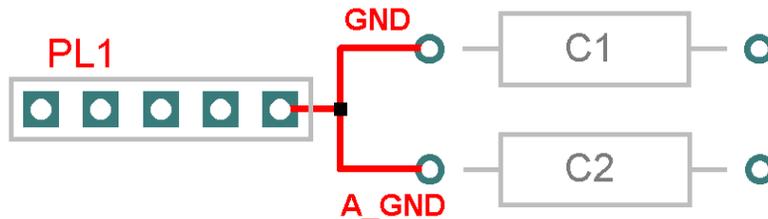
Even though the nets are physically joined at the star points, the nets remain separate and not merged. A special case for DRC is where the track segments attach to the star pad, and errors between these segments are not reported.

Groups

Items in the design can be grouped together for moving etc.

When more than one item is selected, the **Group** option appears on the context menu. Selecting this option causes all the items to be added to a new group. If the items are deselected and then one of the items is selected, all other members of the group become selected.
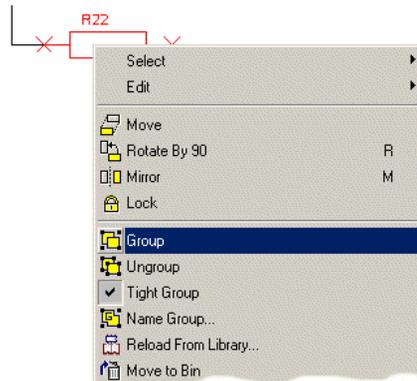
If the group is to be named, select the group and select **Name Group** from the context menu.
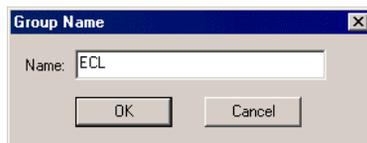
Groups can be named and the group and name passed through from Schematics to PCB designs. If the group is to be translated to PCB, it must be named. Named Groups can be managed using the **Technology** dialog and **Groups** tab.

Name	Tight	Context
ECL	☑	All Designs
Processor Chips	☑	All Designs
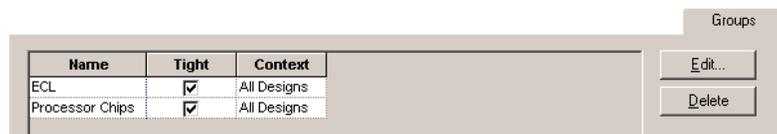
Groups are created as **Tight** groups, that is all members of the group act as one. If you want to make changes to individual members of the group, change the group type to **Loose**. This can be done by selecting the group and unchecking **Tight Group** on the context menu. Any group member when selected will then act individually. To change back to a tight group, select any member of the group and check **Tight Group**.

The **Context** must be **All Designs** for a group to be transferred from a Schematic to PCB.

Hyperlink Attributes

You can add hyperlink attributes to library items or designs using the Insert Attribute option. This attribute is shown as a built in one with the name <Hyperlink>.

Where the hyperlink is used you can add a link to various file types. With the hyperlink displayed, you can select the **Execute Hyperlink** command from the right mouse menu where Pulsonix will attempt to invoke the appropriate application to read the link. This will work the same as if you had selected a link from your Windows Explorer program.

You can use filenames, such as PDF files etc., and folders in the hyperlink name, or you can use Internet URLs.
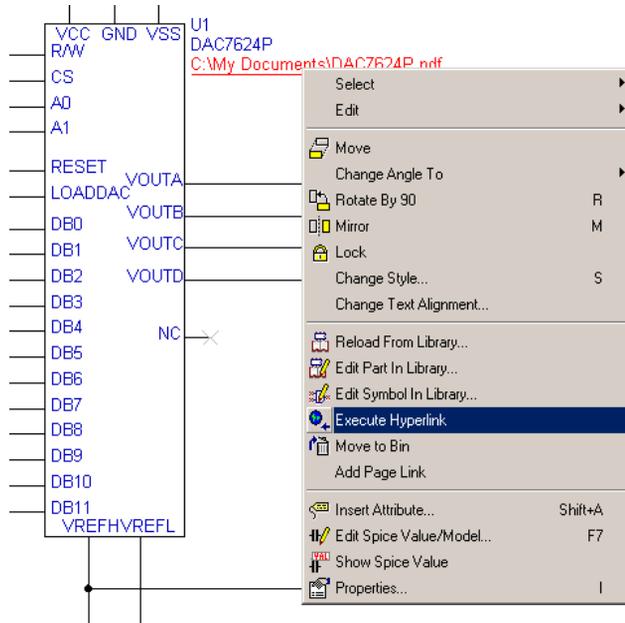
Adding Multiple Components

After selecting the **Insert Component** option, if you then right mouse click while the Component is dynamic on the end of your cursor, you can select the **Insert Multiple Items** option from the context menu. This is available for both the Schematic and PCB design editors.
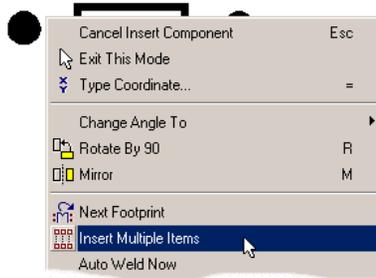
From this you are presented with the **Insert Multiple Components dialog**:
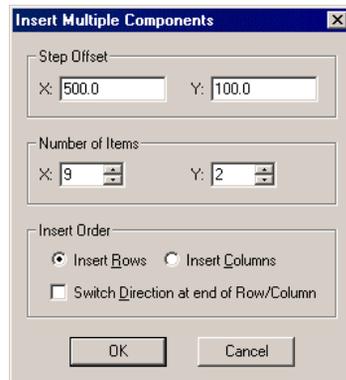
The **Step Offset** defines the distance each Component is offset from the previous one. There are separate values for X and Y and they may be negative to step in the reverse direction.

The **Number of Items** defines the number of items in the X & Y directions. These numbers include the original as the first item (so they start from 1).

Use **Insert Order** to define the order that the items are added. First select **Insert Rows** or **Insert Columns** to specify the items are added in rows or columns, and check the **Switch Direction** box to specify that the next row is started above the end of the previous row, or leave unchecked for each row to be added in the same direction.

Press **OK** to add the multiple items. They will all be dynamically displayed on the cursor for you to click in the required position. Use **Insert Multiple Items** again to alter the item counts or gaps between them whilst placing the items. Set the counts to 1 to go back to adding a single item.

Adding Multiple Pads/Pins

After selecting the **Insert Pad** option, if you then right mouse click, you can select the **Insert Multiple Items** option from the context menu. This is available for both the Schematic and PCB design editors.

From this you are presented with the **Insert Multiple Pads** dialog described in the previous section.

If you hold down the **Shift** key while clicking the Insert Pad option, you will be put straight into the **Insert Multiple Pads** dialog above without having to select it from a menu.

Part Families

A new built-in Part Attribute **<Part Family>** has been introduced to link alternate Parts together, thus allowing the new **Alternate Part** option to be used (see below).

When used in the **Part Editor**, the attribute is easily added using the **Part Family** edit box rather than having to add an attribute by name.

Once this name is defined, the Part can be easily swapped with any other Part that has the same Part family name using the **Alternate Part** option.

Other options now have this Part Family box for defining this attribute or displaying the Part Family, **New Part**, **Properties** of a Component, **Add Component**, **Next Alternate Part** and the **Parts Wizard**.

To set up alternate Parts in your own library, use the **Attribute Editor** in the **Library Manager** to add **Family** names in the <Part Family> column.

Part Alternates

The **Part Alternates** option enables you to swap a Component's Part very quickly. This uses the **Part Family** attribute defined for a Part to identify other Parts with which it may be swapped.

When a Part is added using Insert Component and it has a Part Family attribute, you will be able to use the new **Next Part Alternate** command from the right mouse menu.

When setting up your own libraries, you could provide Part Family names where equivalent Parts are available or where whole families or popular Part are used. There is no restriction to how Part Families are named or used.

Library Manager Improvements

On the **Parts** page of the **Library Manager** option, you now get the **Preview** and **Details** tabs. This gives you a choice between showing graphical Preview or Part Details (as it was).
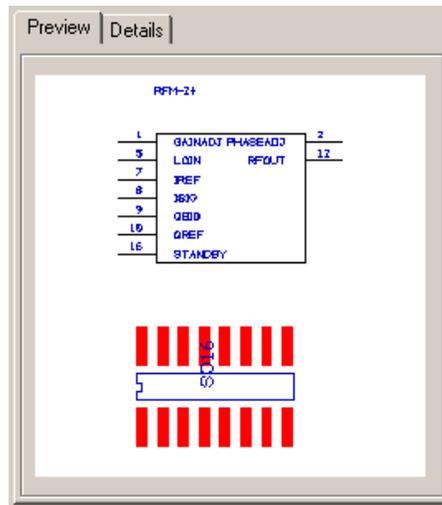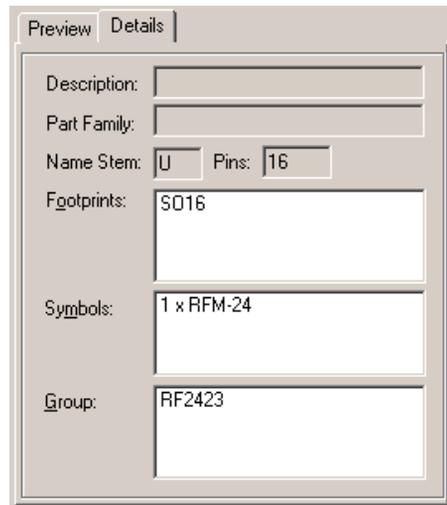
New Preview mode   Original Details mode

Save to Library Improvements

When editing SCM Symbols and PCB Footprints, using **Save To Library** will now include a **Create Part using footprint/symbol** checkbox.
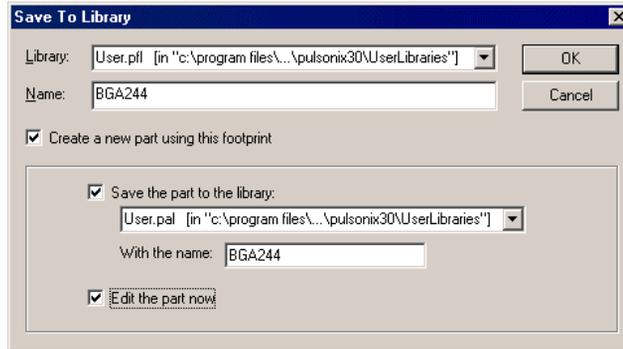
Checking this will drop down more library details which you can use to extend the save to the Parts library. You can enter or select the name of the Parts library and can opt to **Edit** the Part after a save.

When using this option, the Part definition will contain as much detail from the Symbol as possible but will still need editing to contain everything required for a complete Part. It will give you a basic 'shell' with which to work with.

Part Editor Changes

New Part

There are two changes to the **New** page for the Part editor, these have been introduced to make it quicker to start a new basic Part.

Previously, you could only create a (normal) Part, Connector or Associated Part, this has now been expanded to include **PCB Only** and **Schematic Only** Parts.
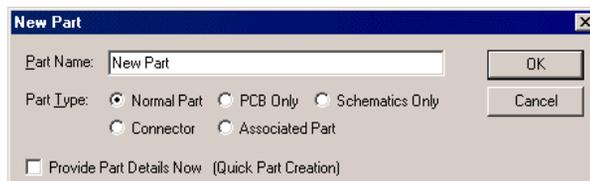
A further check box on the dialog allows you to enter more information about the Part than you used to be able to add to create it much quicker.
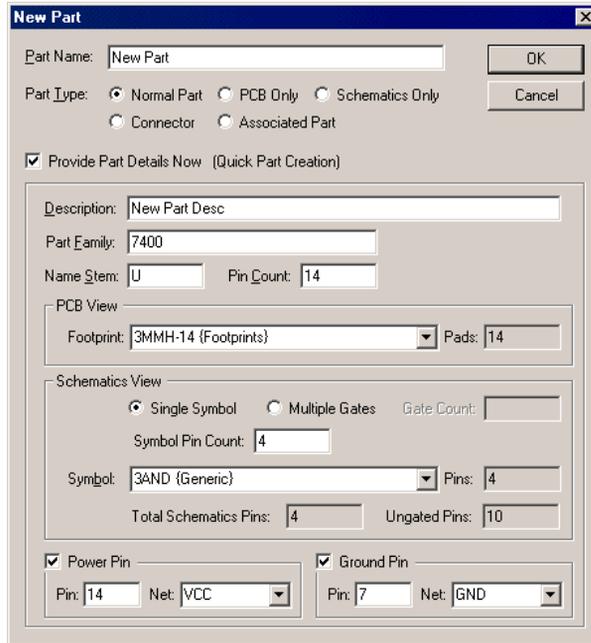
This dialog is very flexible in that it will make selections for you from the relevant library based on your input. Items such as the number of pads that the PCB Footprint will have will filter the items in the PCB footprint libraries to only display that number. If you want all of the footprints displayed then make the **Pin Count** 0.

You can choose all the fields to complete or enter partial information and complete it later on in the Part Editor.

The **OK** button will save the Part as entered and put you into the Part editor as before.

Close Dialog On Edit

On the main **Library Manager** dialog, a new button enables you to close or keep the Library dialog open when selecting a Part to edit.
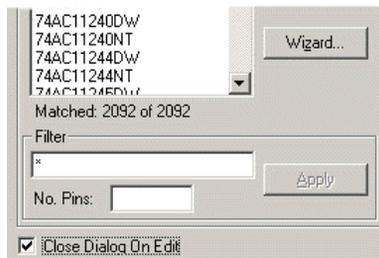
Library Wizard Improvements

Other than specific changes made to the Part Wizard (detailed below), the Symbol and Part wizards have had the **Save To File** option removed from the Finish pages. These have been removed to save confusion as to what was required during the final stages of the Wizards use. If you wish to save the library item to a file, please edit the item and use the Save As option from within the editor itself.

Part Wizard Changes

The **Part Wizard** has been significantly reworked to make it much easier to use and understand.
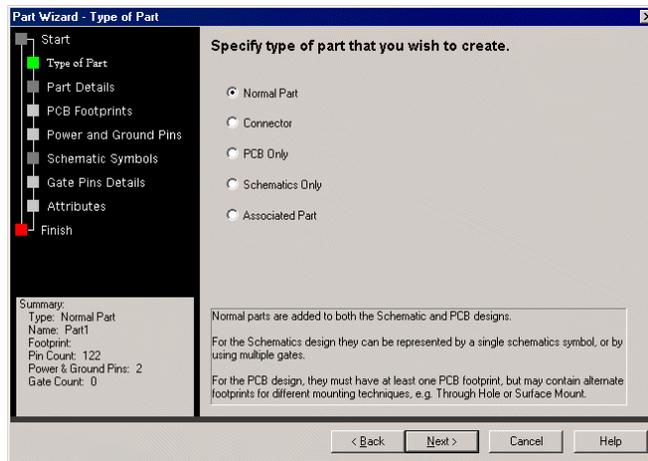
Below is a summary of the changes:

1) Summary of the Part is included on the lower left corner to make it consistent with footprint editor.

2) An information box is included on most pages, this is context sensitive on the Part Type and Part Details pages.

3) Some of the pages have been renamed to make clearer.

4) The page list only changes when you change the Part Type, it also does it straight away without any delay.

5) A preview window is included on the Footprints, Symbols and Gate Pin Details pages.

6) The Technology file has been moved from the Part Type page to the Details page.

7) The Pad Count has been moved from the Details page to the Footprints page. This allows picking of footprints to define the number of pads.

8) The Signal Pins count has been removed from the Details page. You now have Add & Delete buttons for Signal Pins on the Power and Ground page.

9) The Gate Count and number of pins per gate has been moved from the Details page to Schematic Symbols page. On this page you only need to type number of gates if using homogenous Parts (Parts with one Symbol type) to let the wizard know how many gates to add when add is pressed.

10) Save to file on the Finish page has been removed.

Part Browser Changes

You can now select more than one **Category** to list in the **Parts Browser**. The previous **Change Part Category** dialog has been replaced with the **Part Browser Categories** dialog. This uses a grid where each row defines a category name. This also allows you to Filter the Parts using wildcards on an attribute value, or specify the required Pin Count (**No. Pins**).
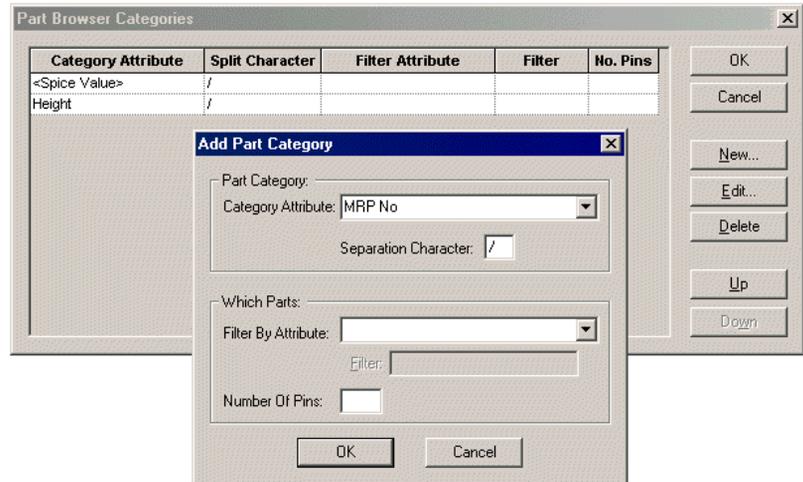
Once the **OK** button on the **Part Browser Categories** dialog is clicked, the **Part Browser** will be updated with the new selection.

Assembly Variants

Assembly Variants has been added to Pulsonix 3.0 for both the Schematic and PCB design editors and for the Technology Files. Assembly Variants enable you to create a master design, but then categorise Components depending on their final build status. You can elect to have Components fitted or not fitted in any particular variant.

A new menu option **Variants** has been added to the **Setup** menu, this displays the **Variant Manager** dialog.
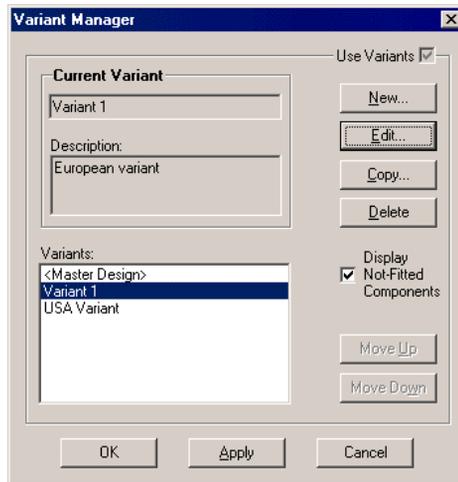
This allows variants to be enabled in the current design. Buttons for **New**, **Edit**, **Copy** and **Delete**, allow creation and manipulation of variants. The variant list order may be modified using the **Move Up** and **Move Down** buttons. Selecting from the list of variants will set the one selected as the current variant in the design. This is displayed in the dialog as the **Current Variant** and is also displayed as a suffix, enclosed in brackets, to the design name shown in the design window title bar.
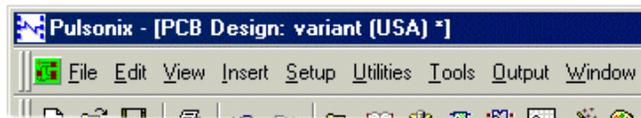
Once a variant is set as 'Current' and the Variant Manager dialog closed, it is then possible to set Components as not fitted in this particular variant. By default, a Component is fitted unless explicitly set not fitted. Selecting the **Display Not-Fitted Components** check box in the **Variant Manager** will force Components to be fully displayed in the design view even when not fitted.

Selecting **<Master Design>** as the current variant temporarily disables the use of variants by treating all Components in the design as being fitted.

Component Properties

Components can be chosen as being fitted or not fitted in the design by selecting them and using the **Properties** dialog.
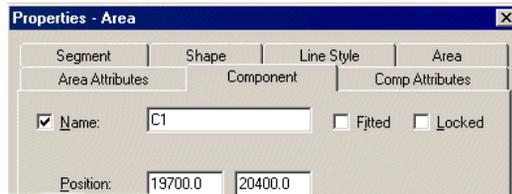
A **Fitted** check box has been added to the Component Properties page (and the Gate Component Properties page) this is only displayed if Variants are in use in the design and there is a 'Current' variant selected, (but not if Master Design is selected).

Newly added components are initially fitted in ALL variants, regardless of whether a particular variant is 'Current', when it is added and must be explicitly set unfitted by un-ticking the 'Fitted' check box in the appropriate variants.

Display of Not-Fitted Components in the Design View Window

In Schematics, not fitted Components/Gates together with any connections to their pins are not displayed (unless the **Display Not-Fitted Components** option has been enabled).

In PCB, the display of **not fitted** Components is determined by the layer class setting. All items with an electrical layer class will continue to be displayed even when the Component is not fitted, e.g. Pads. By default, all other items will not displayed for not-fitted Components, e.g. Silkscreen shapes, although this can be changed by checking the **Visible In All Variants** option for the appropriate layer class. (As with Schematics, the **Display Not-Fitted Components** option provides a global override to force not-fitted components to be always displayed fully.)
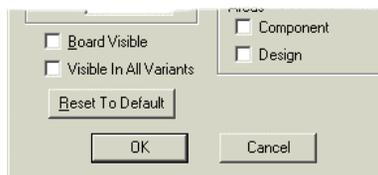
Colours

There are 2 new highlight colours: **Variants** and **Not Fitted**. The Variants colour can be used to highlight any Component that participates in variants, i.e. is not fitted in at least one variant. The Not Fitted colour can be used to

highlight details of not fitted components that are still visible and is best used in conjunction with the **Display Not-Fitted Components** option.

The **Variant** and **Not Fitted** colours appear on the **Highlights** page of the **Colours** dialog.
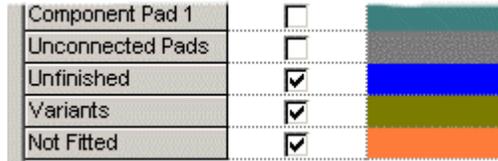
LineStyle

It is possible to specify a line style to be used for drawing any shapes that are still visible for not fitted Components. This may be set by selecting the **Variant** category on the **Defaults** tab on the **Design Settings** dialog. A dashed line style used in conjunction with the **Display Not-Fitted Components** option gives the best effect.

CAM/Plots

If Variants are being used in a design, the **CAM/Plot** tab of the **Generate Plot** and **Technology** dialogs will display an additional column; **Variant**. This defines the particular variant for which each output will be produced. This may be a named variant or "Master Design" in which case the output will be produced for the entire design independent of any particular variant. It may also be set to "Current Variant" in which case the output will be produced for the variant that is selected as 'Current' at the time the plot is run.

Name	Enabled	Device	Process	Variant	Scale	Rotate
Top	✓	Gerber	Layer Top	<Master Design>	1.000	Auto Rot
Bottom	✓	Gerber	Layer Bottom	<Master Design>	1.000	Auto Rot
Silkscreen Top (USA)	✓	Gerber	Layer Silkscr	USA	1.000	Auto Rot
Silkscreen Top (ASIA)	✓	Gerber	Layer Silkscr	ASIA	1.000	Auto Rot
Silkscreen Top (EUROPE)	✓	Gerber	Layer Silkscr	EUROPE	1.000	Auto Rot
Solder Mask Top (USA)	✓	Gerber	Layer Solder	USA	1.000	Auto Rot
Solder Mask Top (ASIA)	✓	Gerber	Layer Solder	ASIA	1.000	Auto Rot
Solder Mask Top (EUROP	✓	Gerber	Layer Solder	EUROPE	1.000	Auto Rot

The **CAM/Plot Wizard** Output page now also allows specification, if necessary, of the variant for which the plot is to be produced.

The **Auto Generate CAM/Plot** dialog now has an additional option **Generate For Each Variant** which, when checked, will create a plot per defined variant, for each process type that is affected by variants. By default in PCB, this will be the non-electrical layers and in Schematics will be each page in the design.

The information included in each plot will be affected by the selected variant and the Components not fitted in that variant. In PCB, the Component items plotted are determined by the layer class of each item with electrical items, such as pads, always included, whereas other items, such as silkscreen shapes, are excluded although this can be changed by checking the **Visible In All Variants** option for the appropriate layer class. The items included in plots are not affected by the **Display Not-Fitted Components** option which applies to screen display only.

If variants are in use in a design, a CAM/Plot report will show the name of the variant for which the plot was produced.

Translate to PCB

The variants and not-fitted Component information defined in a Schematic design will be automatically transferred to the PCB design when the **Translate to PCB** facility is used.

Synchronise

Synchronise Designs will now additionally check and update variant and not-fitted Component information as necessary.

Optimise Gates

The **Optimise Gates** feature in Schematics has been made variant-aware in that a gate will only be moved to another Component if it is fitted and unfitted in exactly the same variants.

DRC

Design Rule Checking has been made variant-aware in that certain types of spacing errors that would otherwise be reported are suppressed if the Components involved are never fitted in the same variant. The spacing rules affected are component-to-component, pad-to-pad and drill-to-drill.

Associated Parts

The **Associated Parts** tab in the **Design Properties** dialog will reflect the design's 'Current' variant. Quantities of associated Parts added for Components that are not fitted in the current variant will not be included. Additional quantities of associated Parts added directly in to the design will be included in all variants.

Quantities of associated Parts relating to not-fitted parts will also be excluded from the Parts List Report.

Auto-Rename

The **Auto-rename** option is now variant-specific. If a design has a 'Current' variant, not-fitted Components will be excluded from the rename.

New Current Variant Attribute

A new **Current Variant** attribute can be added to a design, this will display the name of the 'Current' variant selected in the design.

Copy and Paste/Duplicate

Copying not-fitted Components will retain the not-fitted information only for named variants that already exist in the destination design. Duplicate operations takes place within the scope of a single design so duplicated Components will always preserve the not-fitted state of the original.

New and modified reports

Variants Report

This is a new report that lists all variants and Components in a design in grid format and shows which Components are fitted in which variant.

Part List Report

This report is now variant-specific. If the design has a 'Current' variant, its name is displayed at the start of the report and not fitted Components are excluded.

Part List CSV

This report is now variant-specific. If the design has a current variant, not fitted Components are excluded.

Other Reports

Several other reports have been made variant specific and when the design has a 'Current' variant, will show its name at the start of the report and exclude not fitted Components and their pads. These include:

Alternate Pad Report	JTN Netlist
Board Area Report	Net List
Component List Report	Pick and Place Report
IPC-D-356-full	Pick and Place CSV
Power and Ground Report	Power and Ground CSV
Unconnected Pins Report	Un-Gated Pins Report

Report Maker

This now contains additional syntax for reporting variants and associated information. The new commands are detailed in the section above for Report Maker changes.

Other Outputs Affected

Various other Output formats have been made variant-specific and will omit Components that are not-fitted for the 'Current' design. These include:

DXF Export, LPKF Export, BETASoft Output, IDF and ODB++

Insert Bitmap

The **Insert Bitmap** option can be used to include pictures or logos into the design. This can be used within the Schematic and PCB design editors, it is not available for use with Symbols or Footprints but is for Doc Symbols.

The bitmap types available are for Windows Bitmap, J-PEG and TAG Image Bitmaps.
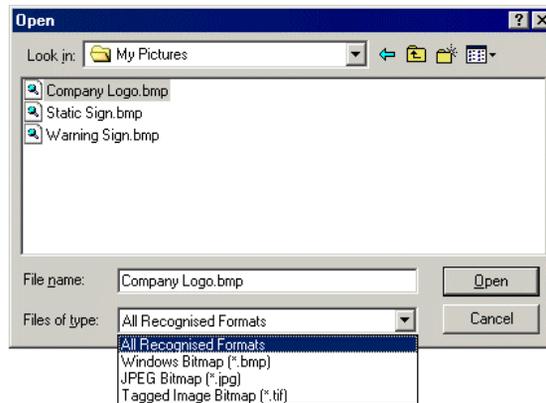
Once the picture is in the design, you can resize it as you would any other shape by selecting an edge or corner and dragging it. The Mirror, Move and Rotate options can be used to manipulate the shape. The shape can also be edited using the **Properties** of the selected bitmap.

When used in the PCB design, the bitmap can be placed on any non-electrical layer. The layer change can be made using the **Change Layer** option. Design with Bitmaps inserted can only be plotted through the Windows output device they cannot be plotted to Gerber or ODB++.

The display of the inserted bitmaps is controlled through the **Colours** dialog. You can elect to Display it, make it Selectable or switch its True Width on/off.

Name	Displayed	Selectable	True Width	Colour
Symbol Origin	☐	☑		
Relative Coords Origin	☑	☑		
Coordinate Origin	☑	☑		
Background				
Bitmaps	☑	☑	☑	

Tabs: Shapes | Connect Points | Text | Attributes | Highlights | Others | Nets
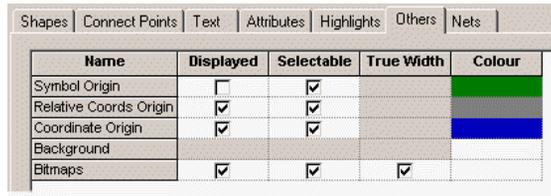
Design Setting Changes

The Design Settings has been changed to allow the modification of the Component Pin Name separator. For example, the 'dot' in U11.6, separating the U11 and the pin number 6.

Pin Names
Component to Pin
Name Separator:

Synchronisation

During Synchronisation, it now reports a warning if the Parts version in the Schematic design is different from the version in the PCB design. If this situation arises it lists the offending Parts.

```
Synchronise Designs Report
--------------------------
Checking integrity of Schematic
'C:\PCB\Pulsonix\Display\display_3.sch'
with PCB 'C:\PCB\Pulsonix\Display\display_3.pcb'

Warning: The following parts in the Schematic are older
versions than those in the PCB.

Part 8x1
Part 20Way Connector
```

Select Profile

Select Profile has been added to the **Setup** menu (in Schematics it appears as **Select Drawing Profile**). This can be used to select all the items that make up a profile when the **Insert Profile** option has been used. This is useful if you wish to select the Profile and all it's associated items in one selection, you could then delete them for example.

Toggle Units

There is a new function **Switch Units**. This allows you to easily toggle between **Imperial** and **Metric** units without having to open the **Units** dialog. It is assigned by default to **Ctrl+U** shortcut keys.

When toggling, the dialog will remember the previous 'other' setting. If toggling for the first time, the default settings will be used.

Edit Arc Changes

Three new **Snap modes** and **Enter Angle** have been added for version 3.0.

You can now select the new snap modes from the right mouse menu once an arc has been selected, **Snap Angle**, **Snap Radius** or **Snap None**. With the **Angle** mode, this means that the arc mode calculations are done whilst moving the arc, to 'snap' the arc angle to the nearest degree. With the **Radius** mode set, this sets the calculations done whilst moving the arc, to 'snap' the arc radius to the current working grid. For **Snap None**, This sets the calculations done whilst moving the arc, to allow the arc to be positioned without 'snapping' the arc angle or radius.
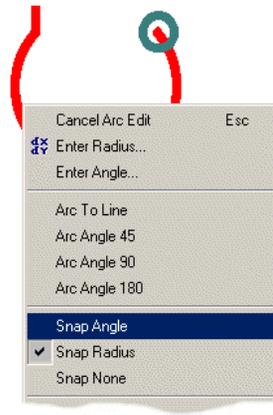
Enter Angle is used to type in an exact angle.
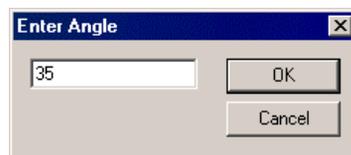
Find Group

You can now find items in named **Groups** using the **finder**.
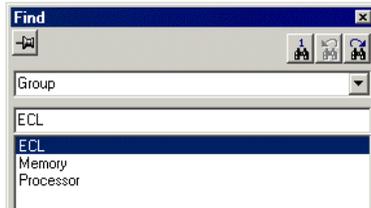
Critical Nets Report

The **Critical Nets Report** has been added, this is available in both the Schematics and PCB design editors. It lists nets that have Net Class Rules on them and critical rules and design information, for example the pin to pin lengths, max track width, track lengths between the rules etc. It will only report information if a rule exists for it.

```
Critical Nets Report
--------------------
Units: thou
Summary
-------
 Total Number of Critical Nets: 2
 Total Number of Rule Fails: 1  (each marked by an
asterisk '*')

Critical Nets
-------------
Net Class: PCI Clock
    Rules:
        Maximum Pin To Pin Via Count: 0
        Minimum Track Length: 2400.0
        Maximum Track Length: 2600.0
        Minimum Track Width: 6.0
        Maximum Track Width: 6.0
        Track Not Allowed On:  Inner Bottom
    Net: /DEVSEL
        Pin To Pin Length
            U3.62    P1.B37    441.2+
        Pin To Pin Via Count
            U3.62     P1.B37   0
        Track Length: 441.2+
        Minimum Track Width: 6.0
        Maximum Track Width: 8.0 *
```

Apply Net Class on Net Name page in Technology

In the **Technology** dialog, you can now do right mouse **Apply to entire Column** on the **Net Class** on **Net Name** page. This change has also been to some of the other pages in the Technology, e.g. Class Type on Net Class page.
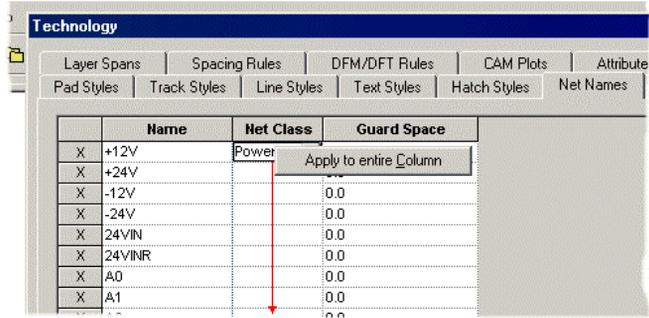
Import UltiCAP hierarchical files (SCM)

With Pulsonix version 3.0, it is now possible to fully import UltiCAP Schematic designs that contain hierarchical blocks. The blocks are recreated in Pulsonix along with block ports, connectivity and bus pins.

Symbol Alternates (SCM)

Symbol Alternates has been introduced in version 3.0. This allows you to switch between one Schematic symbol and another.
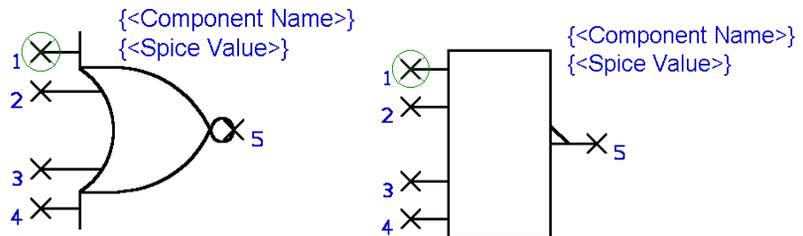
The alternate symbols have to be defined in the Part definition. From the **Gates** page, by selecting any Symbol name you can then select **Change Gate Symbol** from the **Edit** menu or **Change Symbol** from the right mouse menu.
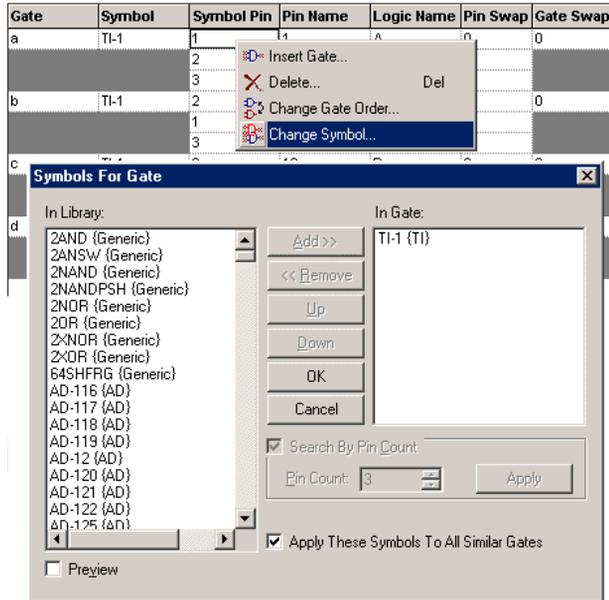
You can add additional gates to the list by using the **Add>>** button. The order in which these are listed will affect the order in which they appear in the Schematic design editor on the Insert Component dialog for example. The first one in the list will always be the one that is shown in these dialogs first.

Once the Part has the alternate Symbol defined, it can be selected when that Part is used in the design.

One way to choose the alternate Symbol is by using the **Properties** dialog after selecting the Component to change. From the drop down list next to **Symbol:** choose the symbol required.
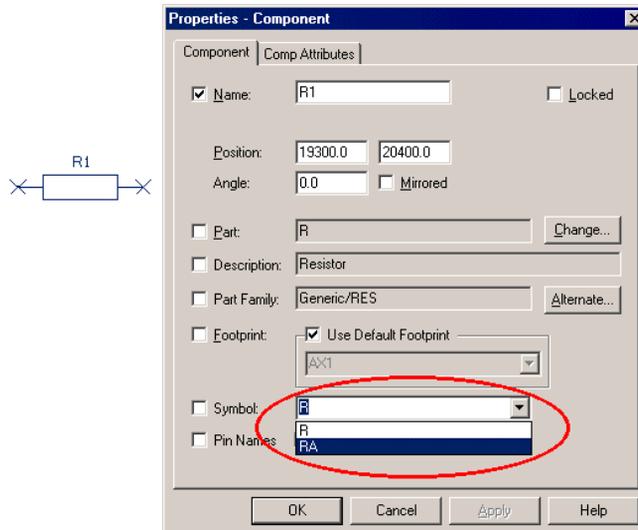
ODB++ (PCB)

ODB++ is an intelligent CAD/CAM data exchange format which helps overcome the inherent ambiguity of the Gerber and Excellon formats. This is used by the manufactures as an alternative format. ODB++ has already become widely acknowledged as the new de-facto industry standard for PCB CAD data in the manufacturing environment. In fact, some PCB manufacturers are offering more attractive PCB fabrication prices if the PCB design data is delivered in ODB++ format.

The ODB++ export option is located on the **Output** menu. It offers Compressed and non-compressed (Folder) versions. Make your choice using the **Compressed** check box.
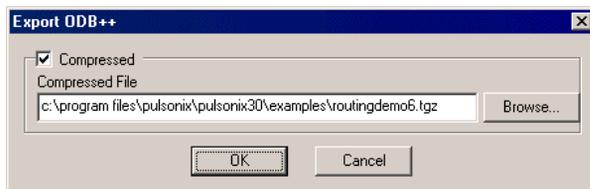
If Compressed is selected, it uses the TAR format and produces a single compressed file.

Un-compressed uses a Folder structure based on the **Browse** path setting. When used you must send the whole folder structure to the people plotting the design. For some of the viewers you must use the folder structure to view the files but some will only work with the compressed version output, you must check which type is required before using this option. Check with the person plotting or manufacturing the design.

Measure Tool (PCB)

The **Measure Tool** is used for measuring the distance between points or items in the design. This is available in the PCB design editor and Footprint editor application in Pulsonix. It can also be used to show the gap between the edges of two items and the required minimum spacing between them. This includes the Drill-to-Drill distance between selected pads.

The tool also provides information about the dimensions of the picked items, and allows you to record the measurement in a report.

The **Measure Results** dialog displays information about the measurement made:
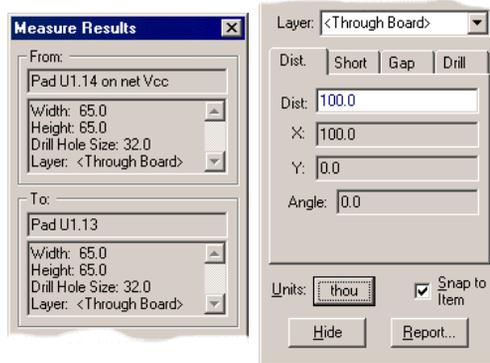
The dialog is shown split here for clarity. The top half displays dimensional information about the picked Items, or positions that the measurement was taken **From** and **To**. If there is more information that can be displayed, the vertical scroll bars are enabled to view all the data.

The bottom half shows the **Layer** being used. This shows the layer that the **Gap** between the items is measured on. If the two items are on completely different layers, 'Different' is shown. In this case the gap is still shown, but may not be that relevant. If a picked item contains different shapes on different layers, a pad with layer reassignments for example, you will be offered a drop down list of common layers between the items to choose from. When choosing a different layer, the measurement information tabs and the line displayed on the design will be updated to show the measurement on the chosen layer.

Under **Layer** are the **Measurement Information Pages**. These pages show different measurements between the two items. Picking a measurement tab will change the information page and alter the measurement line displayed in the design. The different measurements are as follows:

Distance - Shows the distance between the picked points, and the angle of the line between these points.

Shortest - Is only displayed after two items have been selected. It shows the shortest distance between the two items. Widths of lines are not considered for this measurement; use the Gap tab for edge-to-edge measurement. If a component is selected the shortest distance is measured to its placement area on the relevant layer.

Gap - Is only displayed after two items have been selected. It shows the smallest gap between the edges of the two selected items. If both items are electrical items, it shows the required **Spacing** between these items. This spacing is set to zero if the two items are on the same net. If two components are selected the spacing is the minimal component spacing. This page can be used to check two items to see if they are in error. If the gap is less than the required spacing, it will be drawn in red. If the layer is "Different" then the gap is not relevant unless one of the items changes layer.

Drill - Is only displayed if one of the two selected items has a drill hole. The distance shown is from the edge of the drill hole on these items. If both items have drill holes, the spacing shown is the minimum spacing required between drill holes.

Units
Use this to change the units used to display the lengths shown in the dialog. The button shows the units currently being used, and will show "Design" if the design units are being used. When pressed the Units dialog will be displayed.

During use of the Measure Tool, the right mouse menu offers the **Snap To Items** and **Show Measurement** options.
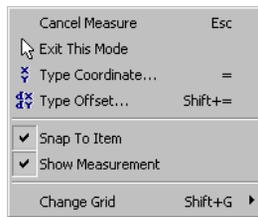
Snap To Item allows less accuracy on the picking of an item. This would be the normal mode of use. Leave it unchecked to restrict picking to positions in the design.

Show Measurement enables you to toggle the display of the **Measure Results** dialog. This provides more information about the items being measured and is designed to be placed to the left or right of the screen. If you don't want the dialog in the way then hide it as the basic distance information is still displayed on the status bar.

A line is drawn between the two items representing the distance being reported.

Ordered Template Pour (PCB)

You can now pour all the templates in the design without individually selecting them, and you can select the order in which they are poured. This option is only available from the right mouse menu if selected in free space, and from the **Run Command** option on the **Edit** menu.

From the right mouse menu you can select **Order Template Pour**. From this you can change the order in which templates are poured by clicking on them.
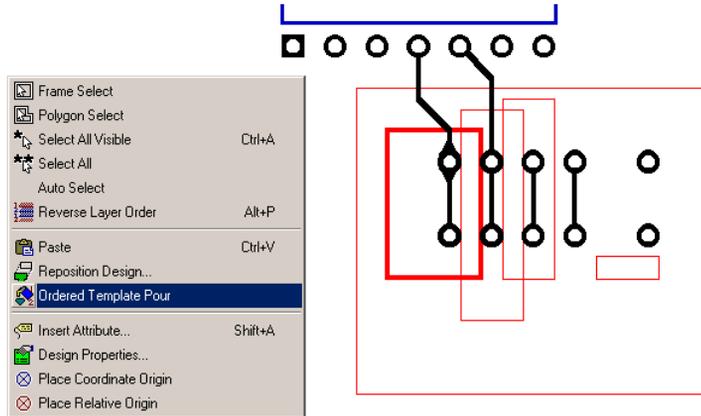
If no specific order is chosen, the templates will be poured in the order as added to the design. The order numbers are shown clearly for all templates in the design when this option is selected.
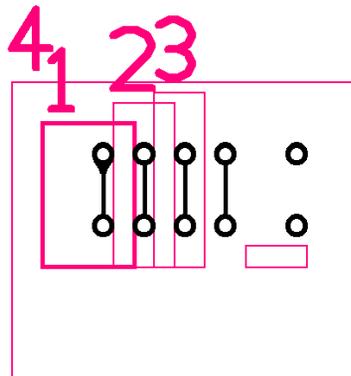
If you wish to change the order then select the template that you want to bring higher up the order. If you double click on one of the templates then this will start the pour.

If you right mouse click at this point, another menu is displayed. This allows you to start the pouring of all the templates, or without pouring, allows you to save the poured order if it had been changed.
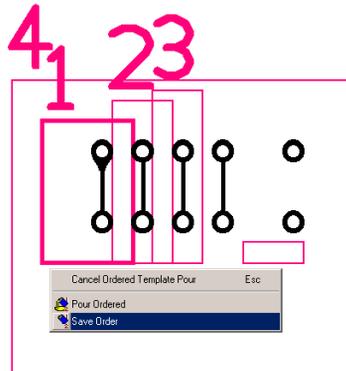
Staircase Removal (PCB)

As part of the automatic router, the **Optimisation** routine has been enhanced to more efficient in its staircase removal. Where a 'staircase' is referred to, this means the track sections which move across the board in short X and Y steps.

New Symbol Definition (Library Toolkit Option)

The Library Toolkit option now contains a new section for the generation of Schematic Symbols. The updated Users Guide for this option is available in the \Documents folder of both the installation folder and the on the distribution CD.

Appendix A Default Files

This section details new default files and changes made to existing files.

Format Files

New or changed format file(s) include:

- **variants.rff**   Output formatter for use with the new Variants.

Standard Reports

New reports

- **Variants Report**   A report for use with the new Variants option, this the uses variants.rff format file.

- **Critical Nets Report**   Reports all nets that have Net Class rules defined for them in the SCM or PCB design

Changes made to:

- **Net Completion/Nets/Critical Nets Report**   These reports now have the report summary at the start of the report to make identification easier.

Formattable Parts/Net List files

Previously created Formattable Parts and Net List files are no longer supported in Pulsonix Version 3.0. You will no longer be able to view or edit them. If you have format files in this old format then you will need to rewrite them using the Report Maker option. If you wish to get help replacing your old format files, please contact your local technical support centre or distributor.

Eagle ULP Conversion Files

New ULP conversion files are supplied for Eagle V4.11 as well as Eagle V4.09. These can be located under the \EagleULP folder in the Pulsonix installation folder.

Appendix B Libraries

New Libraries

The whole library set has been changed. There are many new Parts (approx. 14,500) and associated Symbols and Footprints.

These are categorise by manufacturer, or by Part type such as Diode.

AD.pal
Altera.pal
AMP.pal
Actel.pal
Atmel.pal
AVX.pal
Capacitor.pal
Diode.pal
Fairchild.pal
Fuse.pal
Generic.pal
Hamlin.pal
Hardware.pal
IR.pal
Jumper.pal
LT.pal
Maxim.pal
Microchip.pal
Molex.pal
Murata.pal
NatSemi.pal
Omron.pal
OnSemi.pal
Resistor.pal
RFMicro.pal
Spice.pal
Spice_Examples.pal
Spice_Models.pal
ST.pal
TI.pal
Trans.pal
Vishay.pal
Wago.pal
Xilinx.pal