DesignSpark PCB

V11.0 Update Notes

Issue date: 03/11/23

RS Components Ltd
Birchington Road
Corby
Northants
NN17 9RS
United Kingdom
Tel: +44 (0) 1536 444215

Contents

Chapter 1. Getting Started

Installation

Backing up your files

If you already have DesignSpark PCB installed, please remember to back up all your libraries, Technology files and any other data files before proceeding with the installation of the new version. The installer should not overwrite any of your own named files, but it can re-install new copies of our standard data files so if you have changed any of those files it is important to back them up first. If you are uncertain, check the time/date stamp on the file but in any case, make a back-up.

Of course, backing up your data is important not only for the upgrade but also at regular intervals during design.

Installation from a download link

A download link will be available to you on the DesignSpark website as soon as you start your subscription. Click the link to download the installer executable. You can choose to keep a backup of this file for future use. Any subsequent patches can be installed on top of this 'base' setup once it is completed.

Using Windows Explorer, find the executable in your *Downloads* folder and double-click it.

All other instructions should be followed until you click **Finish** to complete the installation.

The installation is the same for new and existing users alike. Existing users with versions prior to this latest version can install the new software over an existing installation without deleting the old one first.

Installing over existing DesignSpark software

If you already have an earlier version of DesignSpark PCB installed on your system and you wish to install the new version into the same folder as the earlier one, please note that you will then end up with both versions listed in the Windows Control Panel list of installed applications.

If you don’t want the earlier one to be listed in the **Control Panel**, you will need to un-install that version **<u>before</u>** you install the new one. If you install the new software into the same folder as the old version then try to un-install the old one, you will find that the new software will not run as the un-install will have removed many or all of the program files.

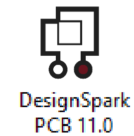
If you wish to install and use the new version without removing the old one, you will need to install the new version into a different folder. The two versions will then operate independently and either can be un-installed without preventing the other from running.

Data Files Location

There is a step in the **Setup** installation wizard that asks you where you want to place data files (for example, Libraries, Technology files, etc). The default is always to use the common documents folder, “Users\Public\Documents\DesignSpark” on Windows 10 or 11 (or the local language equivalents) if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running DesignSpark PCB V11.0

Once installed, an icon will appear in the **DesignSpark** folder and on your desktop. To start the program, double-click on the **DesignSpark** icon.



Chapter 2. Subscription Models in DesignSpark PCB

New subscription Models Introduced

Welcome to the latest version of DesignSpark PCB. With this exciting release, not only are there many new features to further improve your DesignSpark PCB system but we have also introduced new subscription models.

New models have been introduced: **Explorer**, **Creator** and **Engineer**

Explorer is the free version with essential tools to create schematics, convert to PCB and generate a Bill of Materials.

Creator is the paid version of Explorer with extra features to increase your productivity with DesignSpark PCB.

Engineer is the paid version with a professional and comprehensive feature set for manufacturing PCBs.

New Feature Descriptions

Within this document, features have been categorised to show the respective subscription model they belong to. This is denoted in the chapter heading with :

Explorer, Creator & Engineer

If your product is not listed in heading for a particular feature, then it will not be available.

- If the heading shows **Explorer**, then the feature is available in **Explorer**, **Creator** and **Engineer**.
- If the feature is listed under **Creator** and **Engineer**, then it is available in **Creator** and **Engineer** but not **Explorer**.
- If listed only under **Engineer**, then the feature is available for this model only and not **Explorer** or **Creator**.

You can upgrade to the next level of subscription at any time if you wish. Please see the DesignSpark web site for further information.

Chapter 3. New Features in DesignSpark PCB V11.0 (Explorer)

Highlight Net in a Project

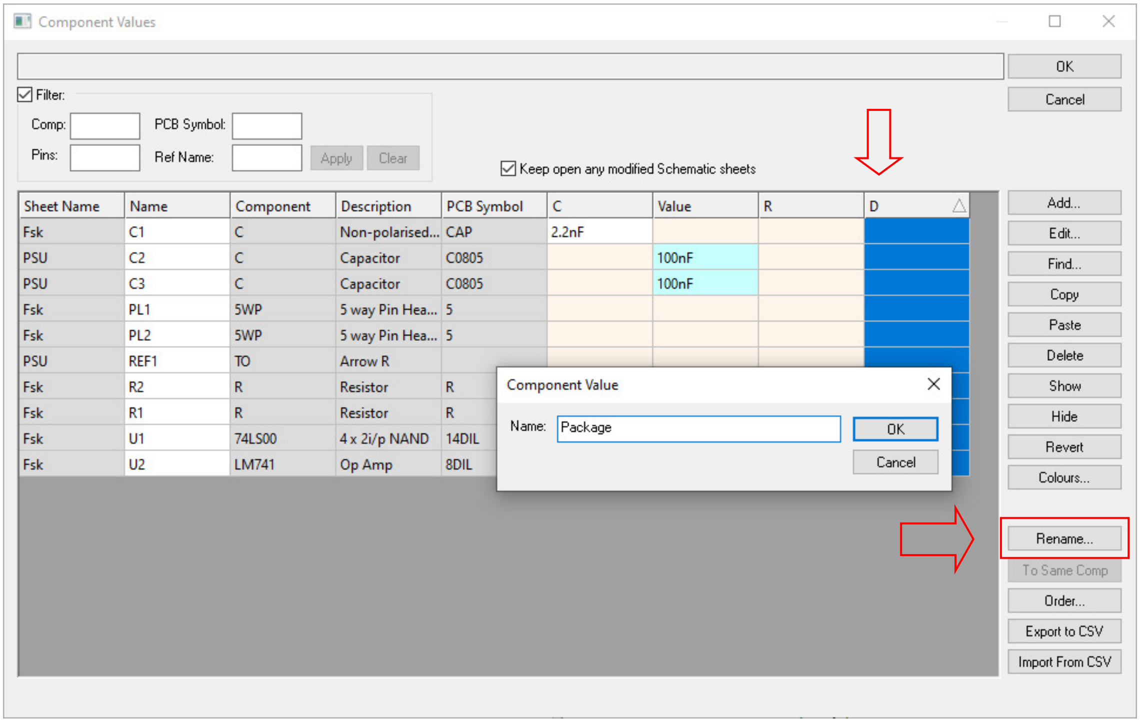
Available in subscription package: Explorer, Creator, Engineer

If you are working on a design in a **Project**, then **Highlight Net** will highlight named (user-defined) nets across all open designs from that Project so that you can easily see where that net is used. The highlight will be visible when you swap to another page that uses the net.

Rename button on Component Values dialog

Available in subscription package: Explorer, Creator, Engineer

The **Rename** button on the **Component Values** dialog enables you to rename an attribute for all Components in the design that use it. This is available for both **Library Components** and Components in a **design**.



By selecting the **Value** name in the column header, the **Rename** option becomes available. Selecting this provides you with a **Name** dialog from which to type the new name.

The result is like this:

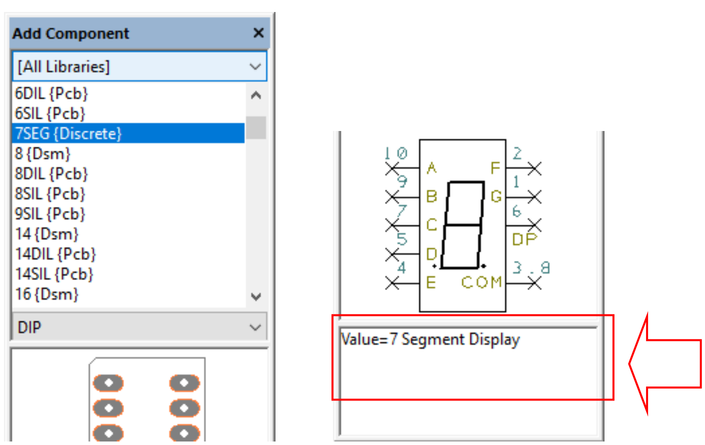
Sheet Name	Name	Component	Description	PCB Symbol	C	Value	R	Package
Fsk	C1	C	Non-polarised...	CAP	2.2nF			
PSU	C2	C	Capacitor	C0805		100nF		
PSU	C3	C	Capacitor	C0805		100nF		
Fsk	PL1	5WP	5 way Pin Hea...	5				
Fsk	PL2	5WP	5 way Pin Hea...	5				
PSU	REF1	TO	Arrow R					
Fsk	R2	R	Resistor	R			20K	
Fsk	R1	R	Resistor	R			10K	
Fsk	U1	74LS00	4 x 2i/p NAND	14DIL				SM
Fsk	U2	LM741	Op Amp	8DIL				DIL

Note, when used in the design and Values names are renamed, this is not reflected in the component Library.

Add Component Bar shows Values

Available in subscription package: Explorer, Creator, Engineer

The **Add Component Bar** now displays **values** of the selected component in a list below the PCB and SCM preview windows. This is in common with the Add Component and other similar dialogs.

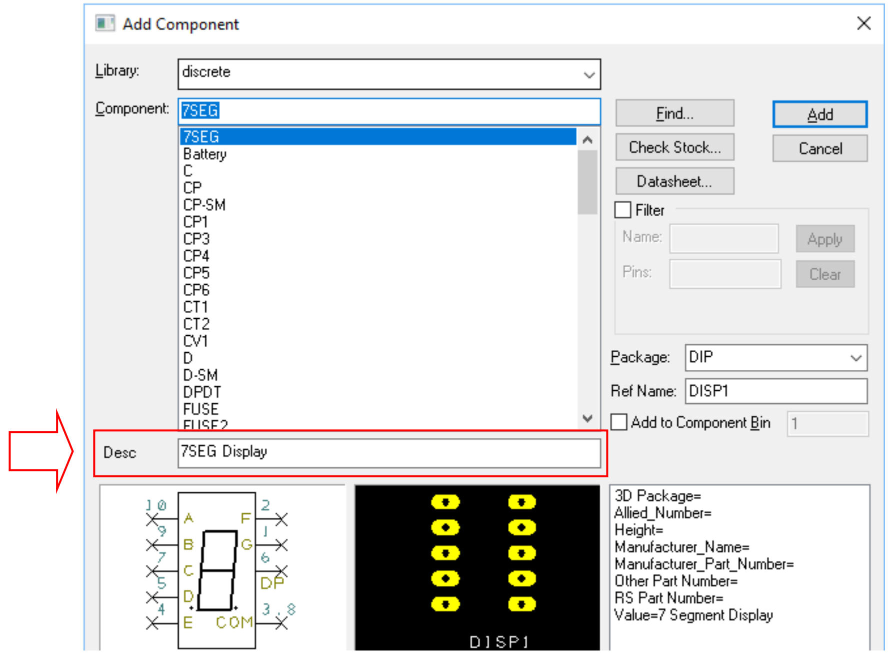


Component Description on dialogs

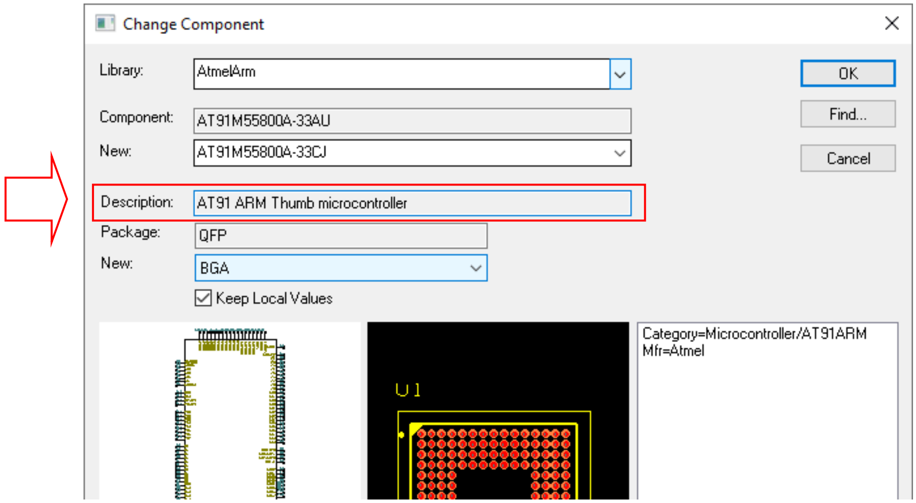
Available in subscription package: Explorer, Creator, Engineer

The **Description** assigned within a Component is now displayed on various dialogs. This change applies to **Add Component**, **Change Component** (from **Properties**) and **Replace Component**.

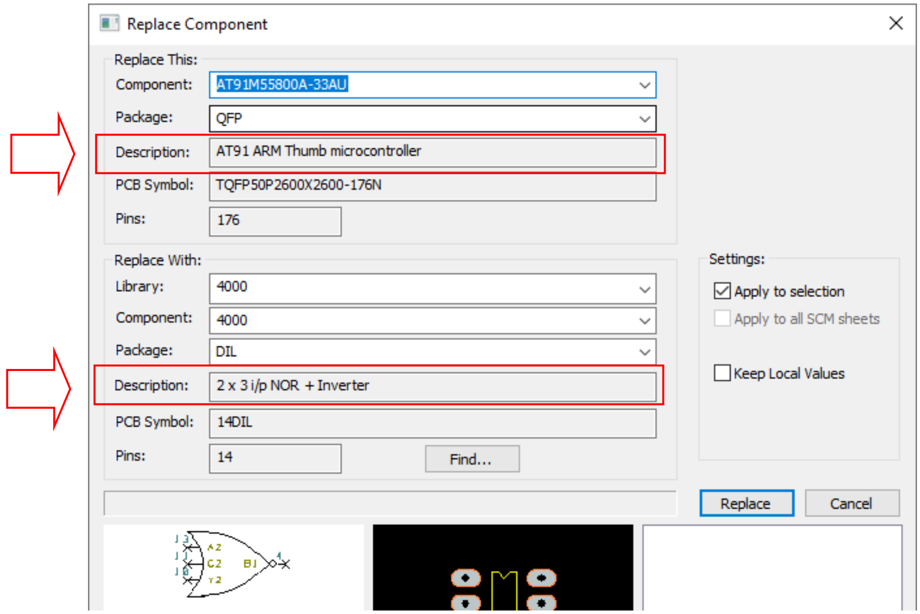
Add Component



Change Component



Replace Component

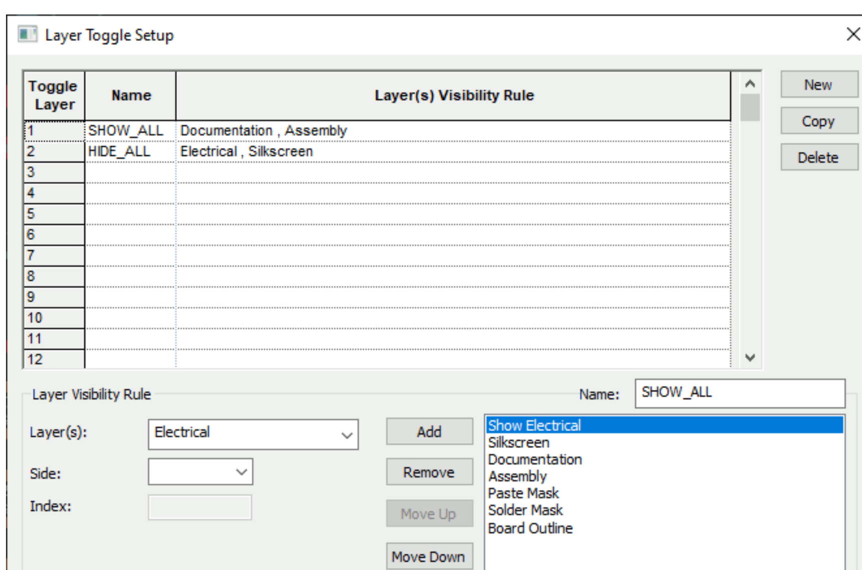


Improved Toggle Layers dialog

Available in subscription package: Explorer, Creator, Engineer

The **Toggle Layers** dialog has been improved to make it more user-friendly.

The changes reflect the **Design Technology** dialog layout and use a grid to contain the different toggles, this includes the **Toggle Layer**, **Name** and **Layer/Side**.



Navigation

The **New** button is used to create a new row and will take the first available grid.

The **Copy** button will be enabled when the currently selected row (or cell within a row) contains commands. When clicked, the currently selected row will be copied to the next empty (free) row and will be selected.

The **Delete** button will be enabled when the currently selected row contains commands and is not the first row. When clicked, the currently selected row will be deleted and the last row to have commands will be selected.

The command list within the **Layer Visibility grid** section will be automatically filled with the commands in the newly selected row and the other controls will be populated with the details of the first layer rule in the list. If there are no commands in the newly selected row, then a new default layer rule will be added to the list and their details populated in the relevant controls. This is now ready to be edited.

This dialog shows all the available **Toggle Layer** commands and their, if any, **Name** down the left side of the grid alongside a slot for specifying a command detailing the layer or layers to which each should apply. The commands can be modified by typing directly into the cell or, alternatively, constructed by using the set of controls at the bottom of the dialog.

Layer Visibility Rule Controls

This section consists of a list of layer rules shown on the right side and controls to create or edit a layer rule on the left side. The list of layer rules make up a toggle layer command. The controls on the left edit the currently selected (blue highlighted) rule in the list on the right.

Controls on the left side of this section will be enabled and disabled depending on the selection in other controls. For example, you can only define a side when a Layer Type has been selected from the Layer combination box.

Like the controls on the left, the list buttons will be enabled and disabled based on the content in the list.

> The **Add** button starts a new layer visibility rule and adds a default rule to the list on the right. This is now ready to edit.
>
> The **Remove** button deletes the currently selected rule in the list. It will then select the next rule above (or below if nothing above) and automatically fill the controls on the left with their new values.
>
> The **Move Up** button moves the currently selected layer rule up one position.
>
> The **Move Down** button moves the currently selected layer rule down one position.

Layers, Layer Type and Side

The set of controls provided at the bottom of the dialog allow a new Layer Toggle command to be constructed or an existing Layer Toggle command to be edited.

The **Layer** combination box contains all the layer names and types in the current PCB design grouped together by whether it is a layer name or layer type. A **Layer** must be selected to create a valid toggle layer command and specifies what layer(s) the command is to act on.

As layer names can vary between designs it may be preferable to associate layers by characteristics rather than their explicit names.

Side provides the additional optional field which allow for more detailed control when a Layer Type toggle is being created.

Omitting Side and just specifying a Layer Type implies ALL layers of that type regardless of where they appear in the layer stack.

Index is available if the **Side** is selected as **Inner**. This will be a numeric value that refers to the Inner layer in the sequence it appears (in the Inner Layers list). It might be a more suitable alternative choice when specifying a specific inner layer (rather than all inner layers) to use a named layer command instead.

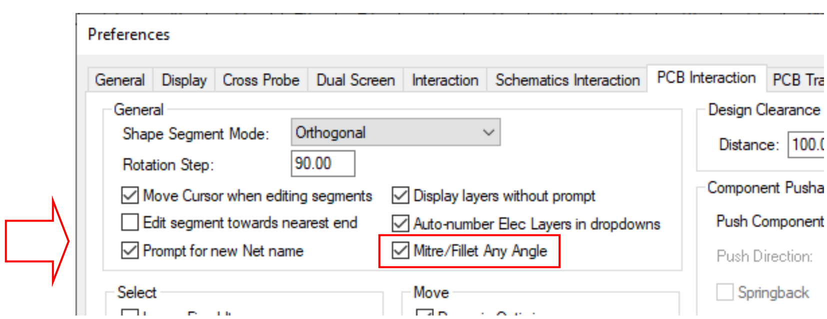
Omitting the Index when specifying **Inner** implies ALL inner layers, so "Electrical Inner" will toggle all the inner electrical layers.

The Layer Name, Layer Type and Side are defined in the **Design Technology** dialog.

Interactive Mitre/Fillet Any Angle in PCB

Available in subscription package: Explorer, Creator, Engineer

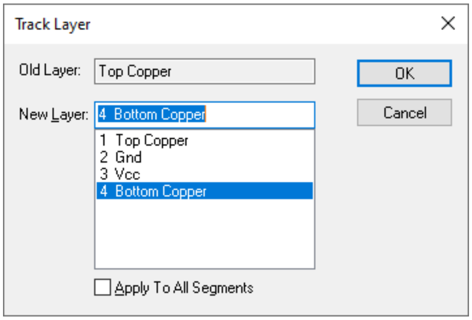
In PCB, you can now interactively add/edit Mitres and Fillets for track that are at any angle instead of just right angles. To enable this option, use the **Mitre/Fillet Any Angle** switch on the **Preferences PCB Interaction** dialog. With this option disabled, you can only mitre orthogonal corners.



Updated Change Layer and Track Layer dialogs

Available in subscription package: Explorer, Creator, Engineer

The **Change Layer** combo box is now a list and matches **Change Style,** it is a simple selection rather than a drop down. This new style is used on **Change Layer** and **Change Track Layer**.



Chapter 4. New Features in DesignSpark PCB V11.0 (Creator)

Component Content Import From SnapEDA to DesignSpark

Available in subscription package: Creator, Engineer

A new collaboration between RS Components and SnapEDA brings you a dynamic interface from where you can search parts and download them directly into your DesignSpark designs and library.

From the **View** menu, **Dockable Bars>** you can select the **Component Search** option. You can also select this option by right clicking on the DesignSpark framework and selecting **Component Search** from the menu.

Prerequisites

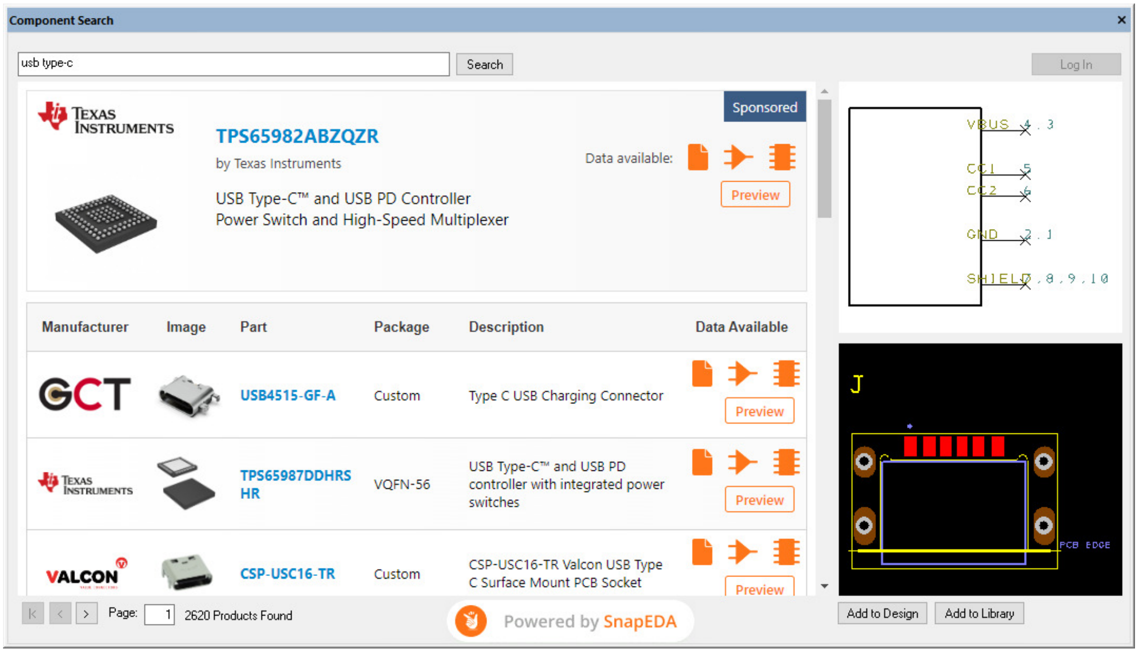
You must have a SnapEDA account in order to download searched contents using this facility. Go to the SnapEDA web site to create yourself an account. You only need to do this once. Without an account, you can search and view content but not download it without the login.

Running the Component Searcher

Once enabled, this will display a browser window, ready for searching.

Use the **Log In** button to access your login details page for the SnapEDA web site. You must have a login to download library content from this web site. You should enter your details for the first time of use, then these will be remembered for subsequent use. You can search and view content but not download it without the login.

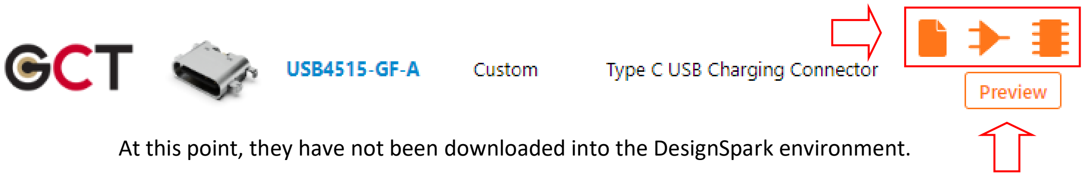
Once logged in, type your required part name into the **Search** field and press <Enter>:



You will be presented with a list of found results, or a warning message if it doesn't find any.

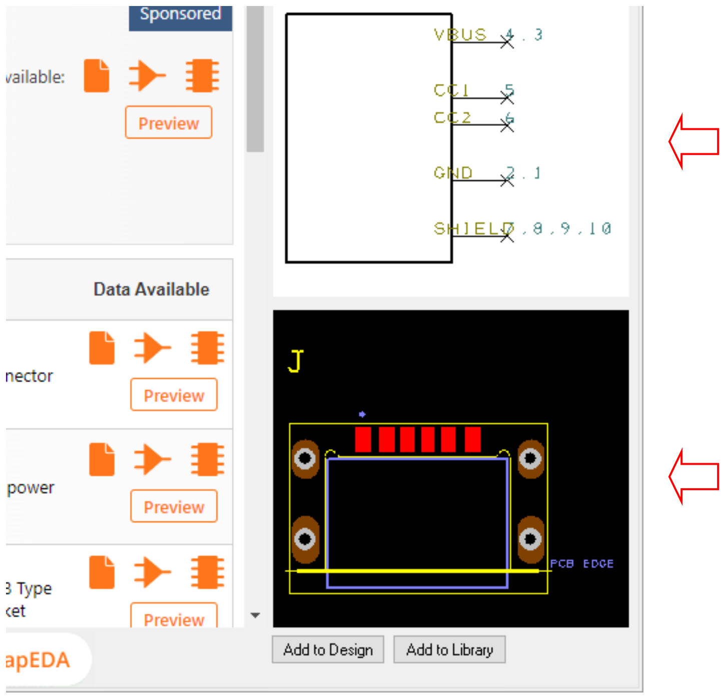
Once items are found based on your search criteria, the results window will display an image of the manufacturer for each item, an image of the item plus icons for the Part **Datasheet**, **Schematic Symbol** and **PCB Footprint** if these are available as downloads in DesignSpark format.

By selecting any of the orange icons to the right side of the results will display a quick preview of the item.



At this point, they have not been downloaded into the DesignSpark environment.

If you select the **Preview** button, this will download the Part data and display it in the Preview windows in the **Component Search Bar**.



Preview Windows

After the Preview button has been used, the symbol and Footprint will be displayed in the Preview windows to the right side of the dialog. Once displayed, you can drag and drop the symbol into the design, or you can use the **Add To Design** button.

Add To Library

Use this button to add the downloaded the **Component**, **Schematic Symbol** and **PCB Footprint** into your own local DesignSpark library. Doing this means that you can also edit the symbol or footprint if you wish to refine it to suit your own needs. Remember also, these library items can be used on another design without having to download them again, whether you modify them or not.

The downloaded library contents will be placed in a newly created library named **DownloadedLibrary** under your **Library** folder.

Add to Design

Use this button to add the downloaded Component into your design. This will place the Component at the end of your cursor in the design ready for placement plus the relevant symbol depending which editor you are currently using; Schematic or PCB. Another way to add the symbol is to select it in the preview window, then simply drag and drop it into your design.

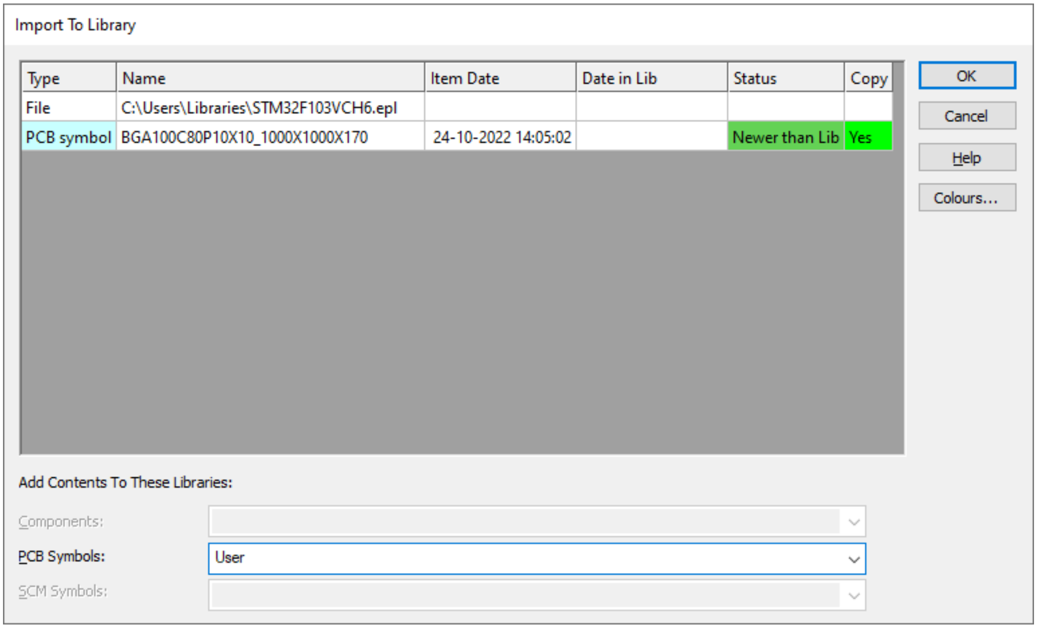
Drag and Drop .dsl Files using the Import To Library Feature

Available in subscription package: Creator, Engineer

From the Windows Explorer, you can drag a library file created in an external content resource, such as Samacsys, PCB Libraries or SnapEDA with the extension .dsl onto the Library Manager where it will open the **Import To Library** dialog.

The **Import To Library** dialog will be run for the same .dsl file for each instance of the library type required, so for the Component, Symbol or Footprint.

Note, this feature is not available for **Explorer**, the **Add** button on the **Library Manager** must be used in this version of the product.

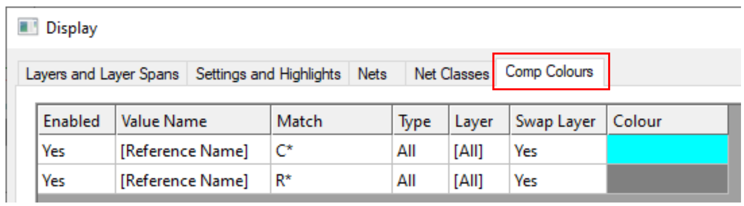


Rules for Colouring Components

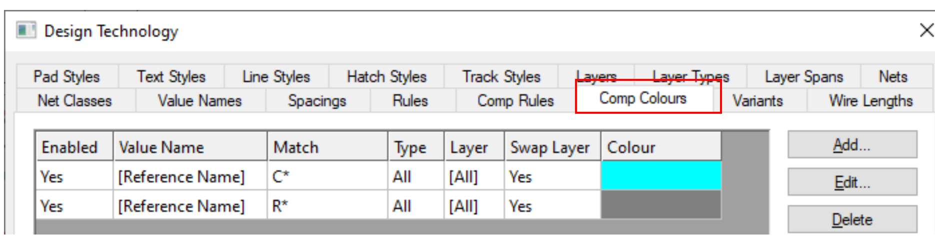
Available in subscription package: Creator, Engineer

A **Component Colours** tab is available in both the **Design Technology** dialog and **Colours** dialog to allow settings for over-riding the colours used to draw Component shapes and/or text.

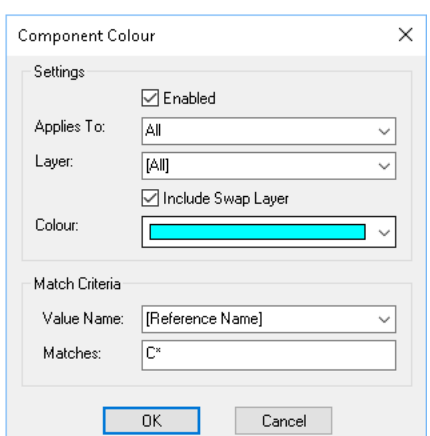
Match components by wildcard on chosen property/attribute and specify colours to be used for matching components by side/layer. This can be used for things like producing a colour-coded assembly drawing, highlighting components with an 'obsolete' value, etc.



The **Design Technology** dialog reflects the same colours as the **Colours** dialog:



Using the **Add** button displays the **Component Colour** dialog:



Applies To: is used to select the item the colour applies to, choose **All**, **Shape** or **Text**.

Layer: choose the layer that the colour is applied to. This can be any layer in the design.

Include Swap Layer will automatically do the same colouring on the equivalent layer on the bottom side as you've set on the top. When set to No, the colour you've defined will only apply to the layer you have chosen

Colour: choose the alternative colour required for the item.

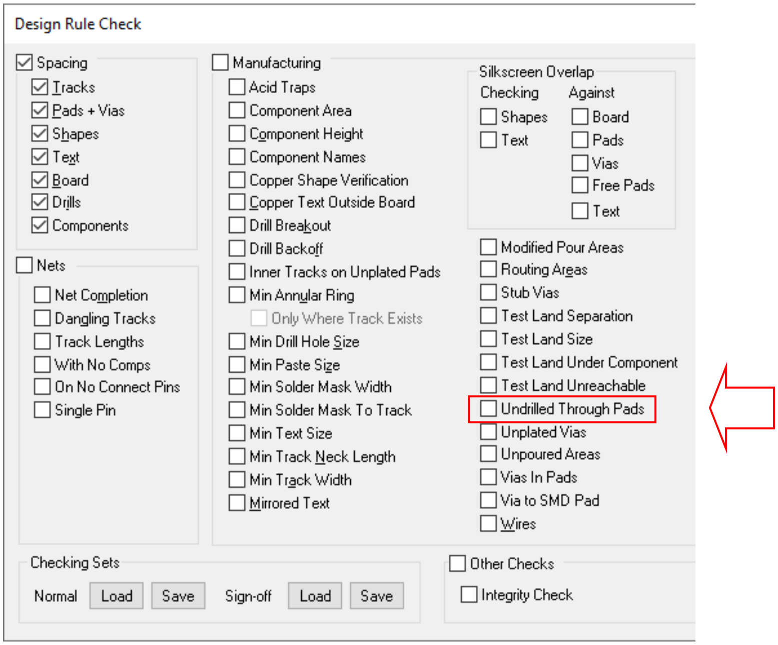
Match Criteria: match based on a Value Name, for example [Reference Name]. The **Matches:** can be specific names or wildcard names.

New Design Rule Checks

DRC for Undrilled Pads

Available in subscription package: Creator, Engineer

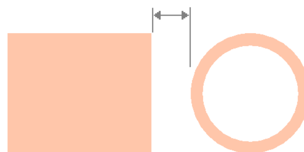
The new **Undrilled Pads** check in the **DRC** dialog looks for through-board pads (on layer [All]) that use a pad style with no drill hole. This avoids the mistakes where pads have not been specified for drilling and the final manufactured PCB has no drills or plating when required. This check is highly recommended on all boards that have any plated-through holes.

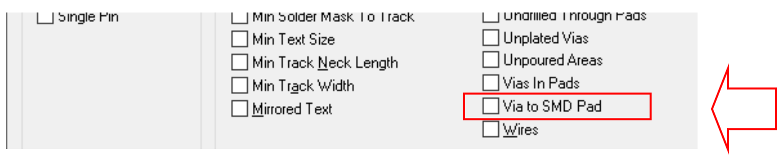


DRC for Via to SMD Pad distance

Available in subscription package: Creator, Engineer

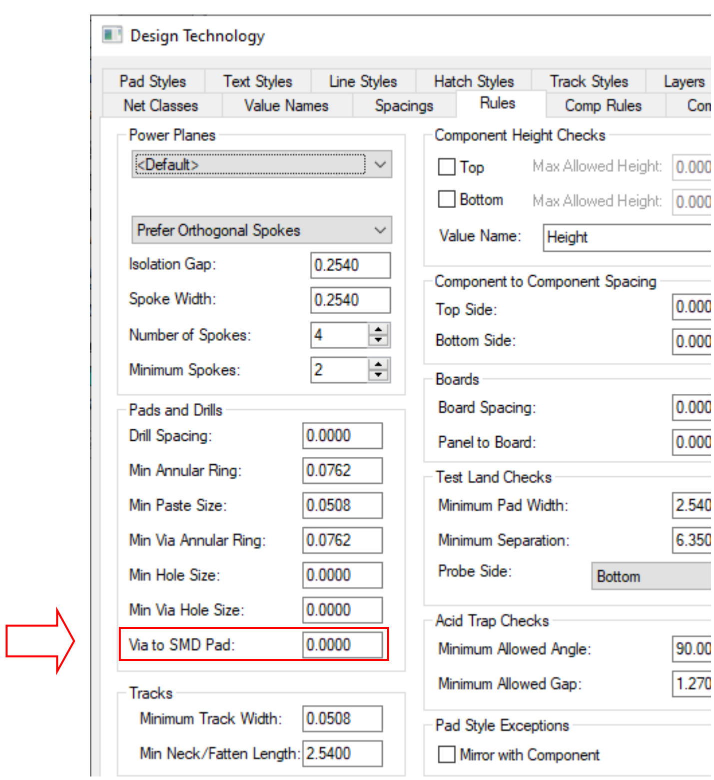
There is a new check in **DRC** that looks for a minimum distance from a **Via to an SMD Pad** on the same net. This means you can set a value that is acceptable by the manufacturer to ensure that solder bridges don't exist.





Available in subscription package: Creator, Engineer

To use this option, it has to be enabled and a value defined on the **Rules** tab of the **Design Technology** dialog. This value specifies the distance from the SMD pad edge to the pad edge of the via.

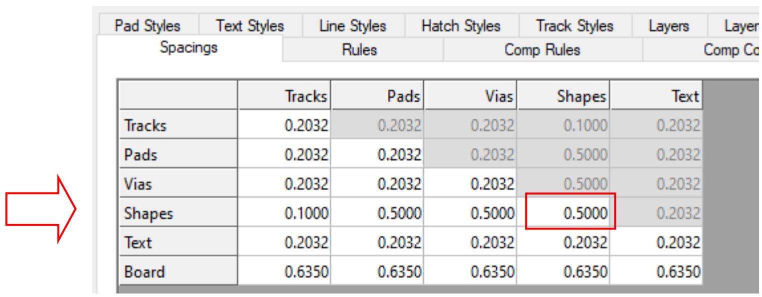


DRC to use Track Spacings for Teardrops

Available in subscription package: Creator, Engineer

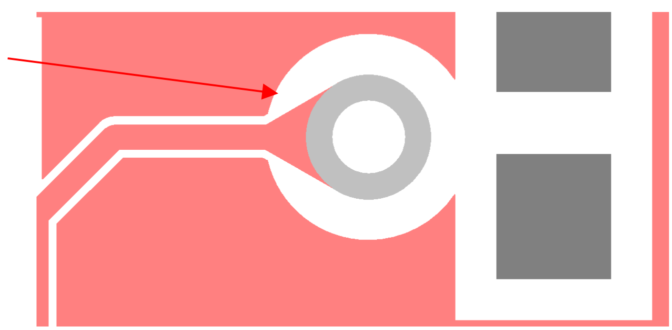
A new check box on **Spacings** page of **Design Technology** dialog instructs the program to use the spacings defined for tracks when dealing with teardrops. This is used instead of the default setting which is to use the Copper shape spacings.

The pouring will use the **Shape to Shape** spacing, like this if not checked:

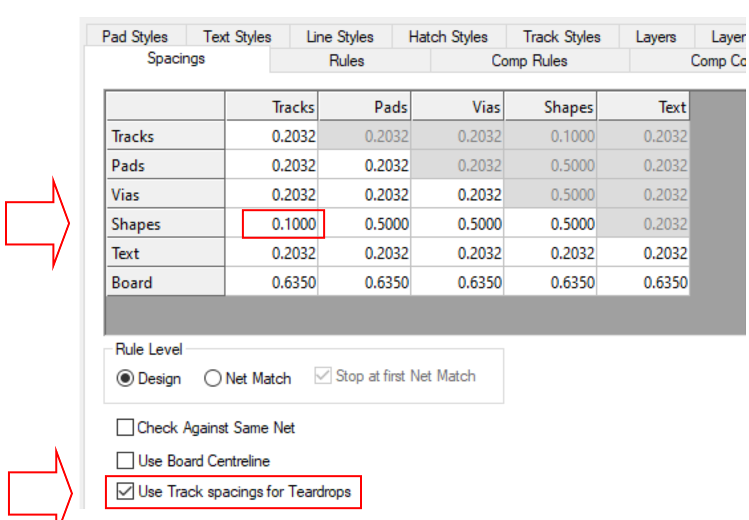


	Tracks	Pads	Vias	Shapes	Text
Tracks	0.2032	0.2032	0.2032	0.1000	0.2032
Pads	0.2032	0.2032	0.2032	0.5000	0.2032
Vias	0.2032	0.2032	0.2032	0.5000	0.2032
Shapes	0.1000	0.5000	0.5000	0.5000	0.2032
Text	0.2032	0.2032	0.2032	0.2032	0.2032
Board	0.6350	0.6350	0.6350	0.6350	0.6350

The result will look like this:

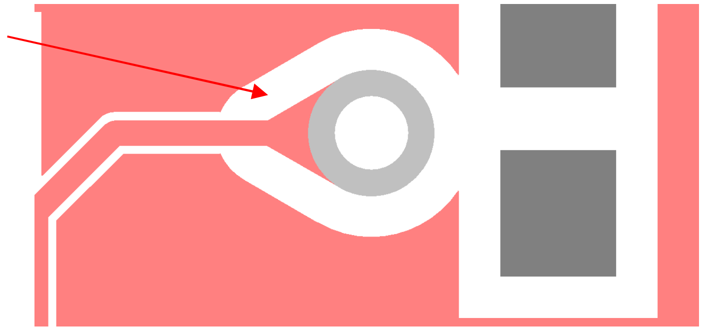


If the **Use Track spacing for Teardrops** check box is selected, it will use the **Shape to Track** spacing instead:



	Tracks	Pads	Vias	Shapes	Text
Tracks	0.2032	0.2032	0.2032	0.1000	0.2032
Pads	0.2032	0.2032	0.2032	0.5000	0.2032
Vias	0.2032	0.2032	0.2032	0.5000	0.2032
Shapes	0.1000	0.5000	0.5000	0.5000	0.2032
Text	0.2032	0.2032	0.2032	0.2032	0.2032
Board	0.6350	0.6350	0.6350	0.6350	0.6350

In this example, the Shape to Track spacing is larger but equally, it could well be smaller.

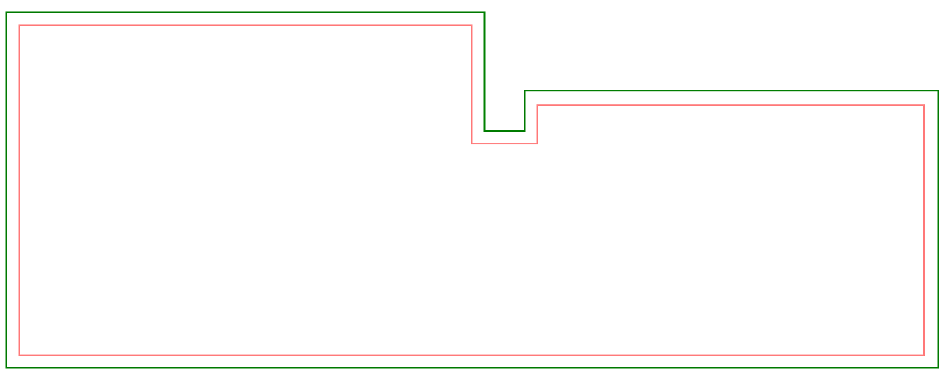


Resize Shape

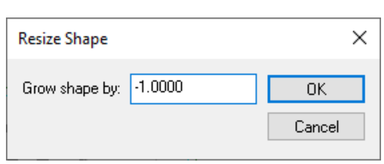
Available in subscription package: Creator, Engineer

The ability has been added to **grow** or **shrink Area**, **Board**, **Copper** or **Doc shapes** using the **Resize Shape** command.

This flexible new option enables you to select a Shape, the board outline for example, copy it, change its Type to a Copper Pour Area and then shrink it by a set value. An exact copy of the board outline will then be made but slightly smaller. This is shown in the example below:



To use this option, select the shape and right click, select **Resize Shape** from the context menu.



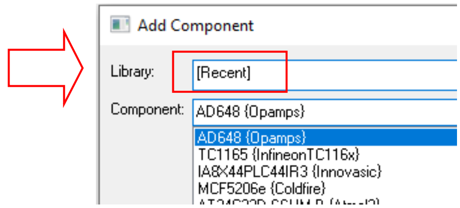
Type in the value required. A negative value using the **minus** – sign will **shrink** the shape. A positive value (with **no minus sign**) will **grow** the shape by the value typed. This will use the current design units as displayed in the bottom right-hand corner of the **Status** bar.

This option can be used in the Schematic design and symbol editors too.

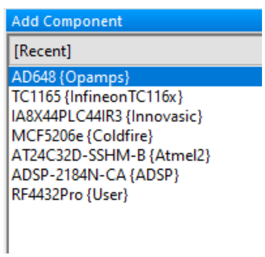
Add Component – List of Recent items

Available in subscription package: Creator, Engineer

The **Add Component** dialog now has the ability to remember the most recent components you have added to your designs. It will present those as an easily accessible list when you select **[Recent]** from the library list in the Add Component dialog:



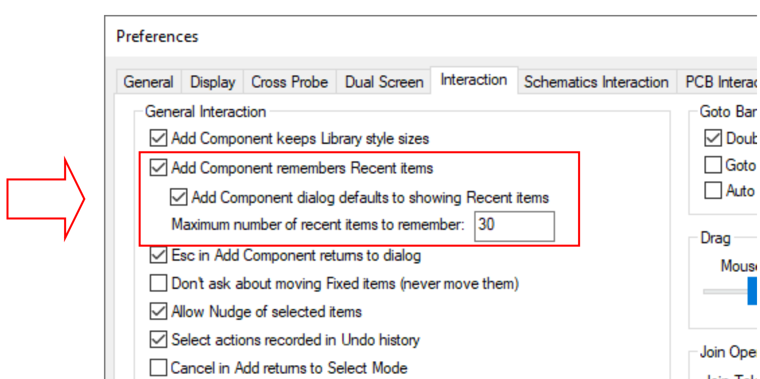
The same list is available in the **Add Component Bar** when you choose **[Recent]** from the top of the library list:



The **[Recent]** list is also available in the **Change Component** dialog accessed from **Component Properties**.

Each component you add or use from any of these locations will be automatically added to the top of this list, so you will always see your most recently added or used components right at the top.

The operation of this feature is controlled by settings on the **Interaction** tab of the **Preferences** dialog.



These allow you to enable/disable the whole ‘recent items’ feature, control how many items are to be remembered (the default is 30) and to specify whether to always display the [Recent] list when you open the Add Component dialog.

You may find it useful to use the Add Component dialog and bar together like this: in **Preferences**, **Interaction**, uncheck the box for showing recent items on entering the **Add Component** dialog. This allows you to use the dialog as before, browsing different libraries or using **Find** to locate items. On the **Add Component Bar**, set the library name to [Recent] to see your recently added items.

Now your recent items are available to drag into your design from the **Add Component Bar**, but if you want to add a component that you haven't recently accessed then open the Add Component dialog to find it. When you add that to your design it will automatically appear at the top of the Recent list in the dockable bar so it is readily accessible next time you want it.

IDF Export – New Output Options

Available in subscription package: Creator, Engineer

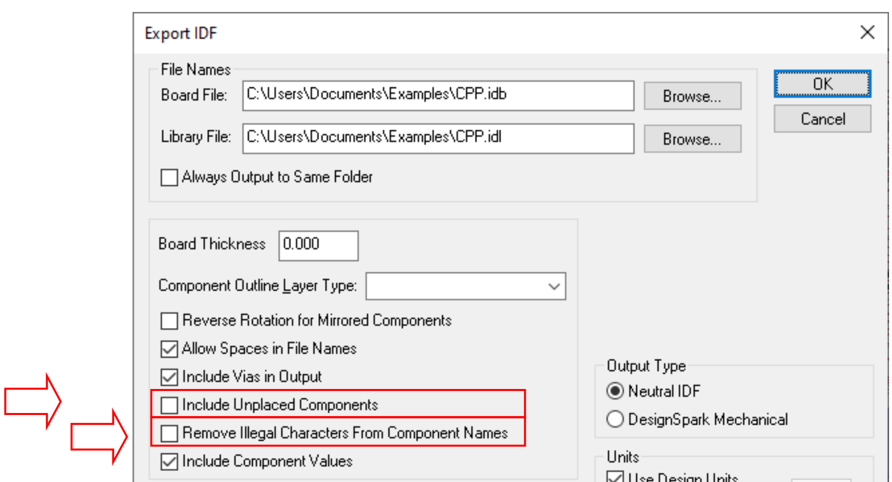
Remove illegal characters from Component Names

Include/Exclude Component Values

There are two new options in IDF export dialog:

Remove illegal characters from Component Names

Include (Exclude) Component Values



Remove Illegal Characters From Component Names - check this box to remove characters that are illegal in Component names for some programs, such as Inventor 3D. They will be replaced with an underscore. These characters include: \, /, :, *, ?, ", >, <, |

Note: The IDF export option already did this for Version 3.0 formats but this new option lets you switch it off if required.

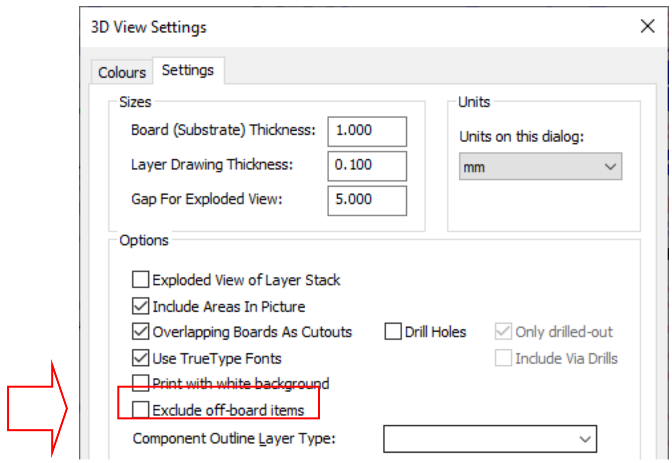
Include Component Values - check this box to includes Component Values in the output file. This can only be for Version 3.0 formats.

Exclude Off Board Items option for 3D View

Available in subscription package: Creator, Engineer

There is a new check box on the **Edit** menu, **3D Settings** dialog called **Exclude off-board items.** When selected, this will prevent components and other items that are not entirely within a board outline will be excluded from the view.

This can help for example if you have a standard drawing outline with title blocks, checking this box will exclude those portions that are not on a board.

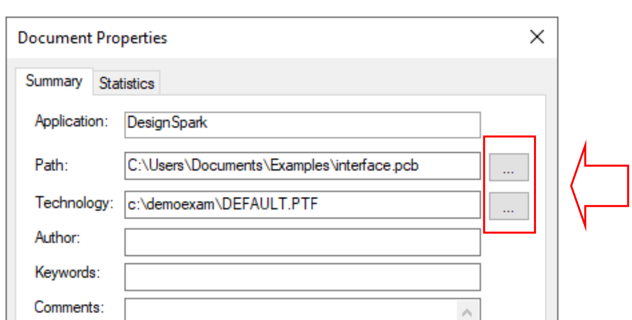


Navigate to Folder feature in Document Properties dialog

Available in subscription package: Creator, Engineer

Two small [...] buttons have been added to the **Document Properties** dialog on the **File** menu, alongside the Path name of the design/project and the **Technology** file.

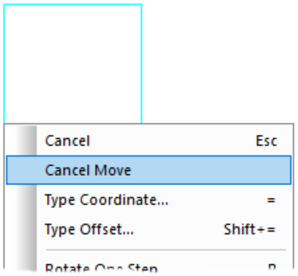
Selecting these will open the corresponding folder in Windows File Explorer.



Cancel Move After Paste

Available in subscription package: Creator, Engineer

There is a new option on the context menu called **Cancel Move**.



This can be used when an item is copied and after paste. This will cancel the move of the item and release it back in its original position.

If a section of design is cut from one design and pasted to another, the offset is not preserved. That means that if the original section has been critically positioned, the pasted section must be manually repositioned to the required location without the ability to use properties and type in exact co-ordinates. The **Cancel Move** option allows the original position to be preserved.

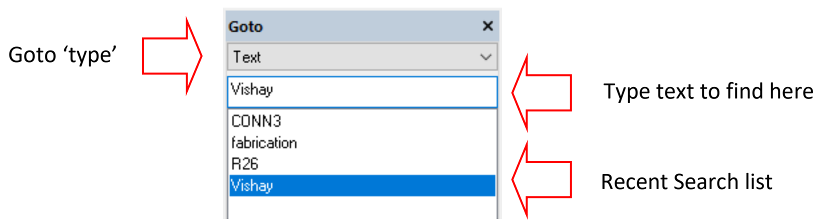
If the item is copied using **Duplicate** instead of **Copy**, the menu entry is still present but only while the item is on the cursor and dynamic, it disappears once the item is placed.

Enhancements to the Goto Bar

Available in subscription package: Creator, Engineer

Goto Text

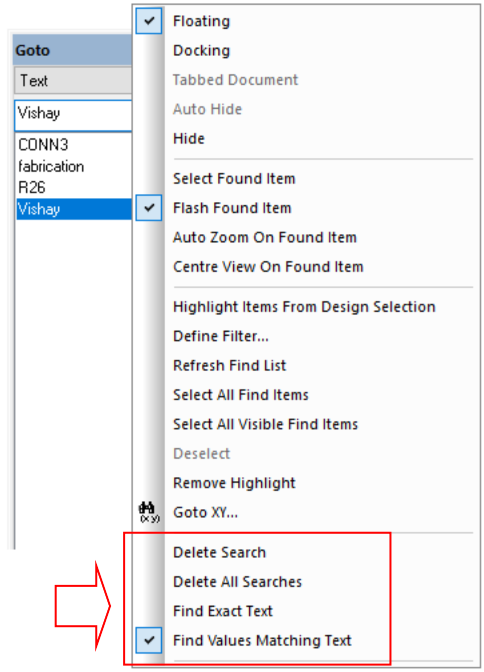
A **Text** search has been added to the **Goto** Bar, this can find all 'free' Text and Values text in your current design.



Choose **Text** in the Goto Bar as the '**type'** and enter the text to find in the box. As you search for different text items in your design, the searched list will be populated. This can be refined using the **Delete Search** option from the context menu on the selected text string.

Note, to find Component Names, like, R1, C1 etc. choose Values Text.

New options on the context menu are available for **Goto Text**.



This option can also be used to search for Values by selecting **Find Values Matching Text** from the context.

Context menu commands

Delete Search - Deletes the selected search from the list.

Delete All Searches - Deletes all previous searches from the list.

Find Exact Text - Only text matching the exact search string will be found.

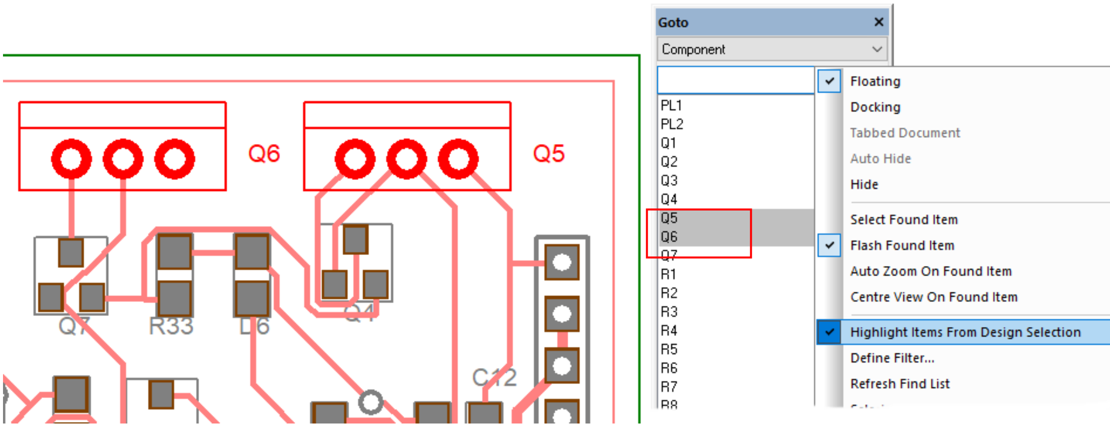
Find Values Matching Text - Use this to find items with Values containing text matching the search string.

Highlight Items in GoTo Bar

Available in subscription package: Creator, Engineer

There is a new option on **Goto Bar** context menu called **Highlight Items From Design Selection**.

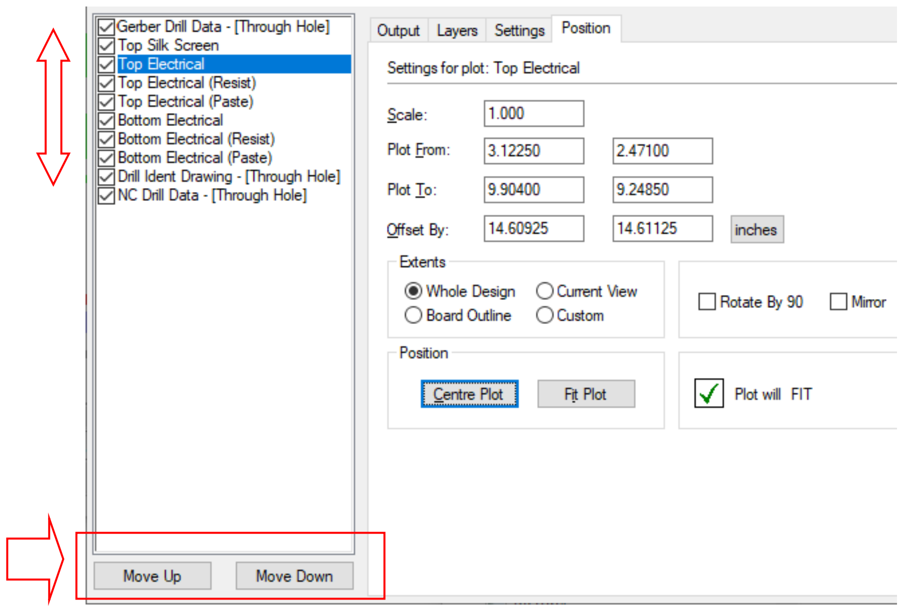
If you select items in the design, they will be highlighted in the Goto Bar (if it is pinned). It will highlight the relevant item e.g. if you select a Component and have Nets select in the Goto Bar, it will highlight the component net(s).



Plotting & Printing - Move plots up and down in the Plot List

Available in subscription package: Creator, Engineer

There are two new buttons on the **Plotting & Printing** dialog available from the **Output** menu - **Move Up** and **Move Down**. These buttons allow you to reorder plots in the list, to sort them by type (Gerber etc.) for example.

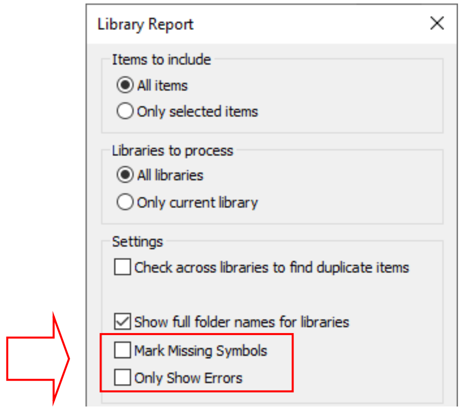


Library Manager Report

Mark Missing Symbols & Only Show Errors

Available in subscription package: Creator, Engineer

There are two new settings in Library report dialog - **Mark Missing Symbols** and **Only Show Errors**. This report is available from within the **Library Manager** on the **Symbol** and **Component** tabs.



Mark Missing Symbols, will add the word, ‘Missing’, to the report if the symbol can’t be found.

Only Show Errors, only reports missing symbols and reports where a symbol is defined, but can't be found in an accessible library.

Track Editing - Finish Track on Start Pad not allowed

Available in subscription package: Creator, Engineer

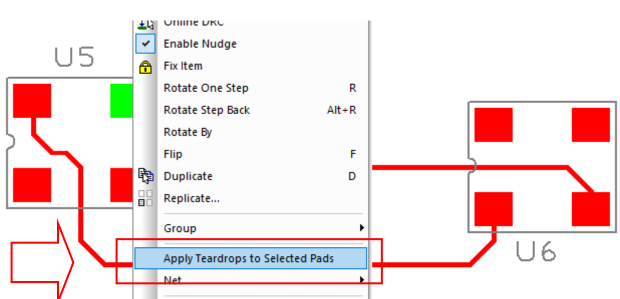
You can no longer finish a track when the track still inside the pad that it starts on. There are no practical situations where a track ever needs to start and finish within the same pad area, so the editing function has been tightened up to avoid this. Generally speaking, these are called dangling tracks and should be avoided.

If you have created this situation on existing designs, they can be detected using the **Design Rules Check** option and subsequently resolved by manually editing the offending track. Use 'Next' from the context menu to select buried track (key <N>) or hold <Alt> when left clicking.

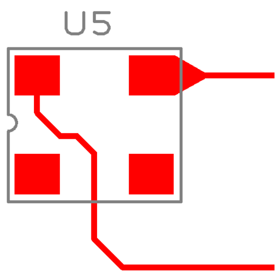
Add Teardrop to Selected Pad only

Available in subscription package: Creator, Engineer

If you have a pad selected, you can use the new option to **Apply Teardrop to Selected Pad**.



The result will look like this:



Likewise, once the teardrop is added, you can select the pad and remove the teardrop using the **Remove Teardrop from Selected Pad** option.

BOM Composer

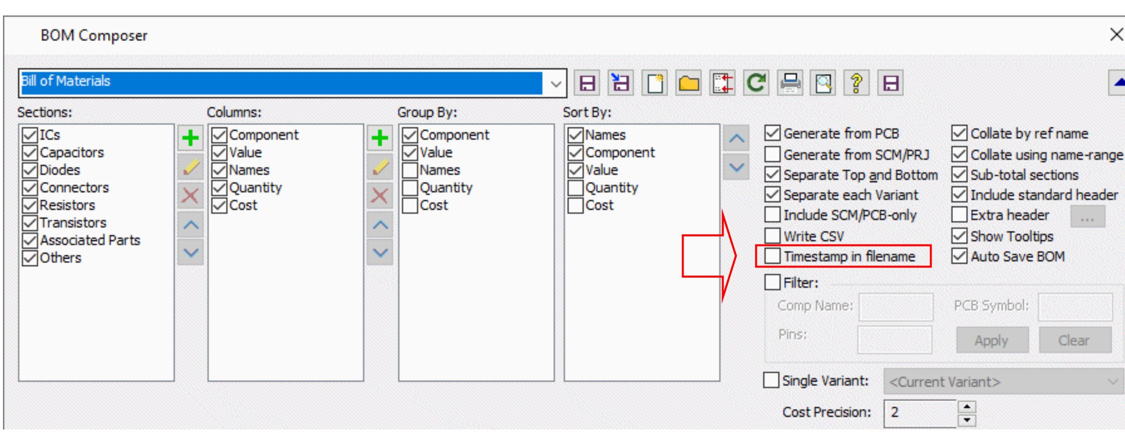
For **Creator** and **Engineer** users, the BOM **Composer** feature is available on the **Output** menu and is designed to simplify the task of producing the Bill Of Materials ("BOM") you need as part of your production documentation. Using example templates, a built-in default setup, or building your own template from scratch, the whole process is quick and easy. What's more, you can immediately see the effect of each change you make to the template, right in front of you on the screen.

The features below are new to version 11.0:

BOM Timestamp

Available in subscription package: Creator, Engineer

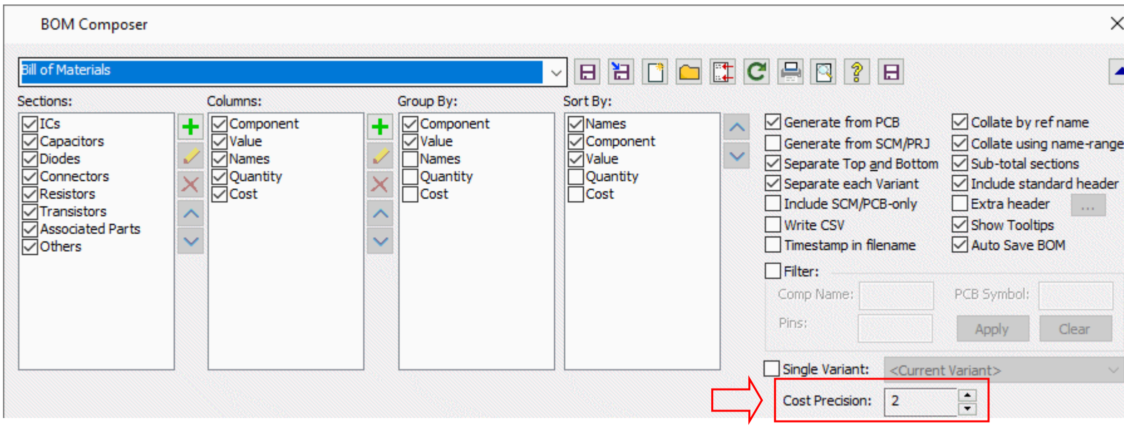
You can now add a **timestamp** to the BOM filename to make it unique.



BOM Cost Precision

Available in subscription package: Creator, Engineer

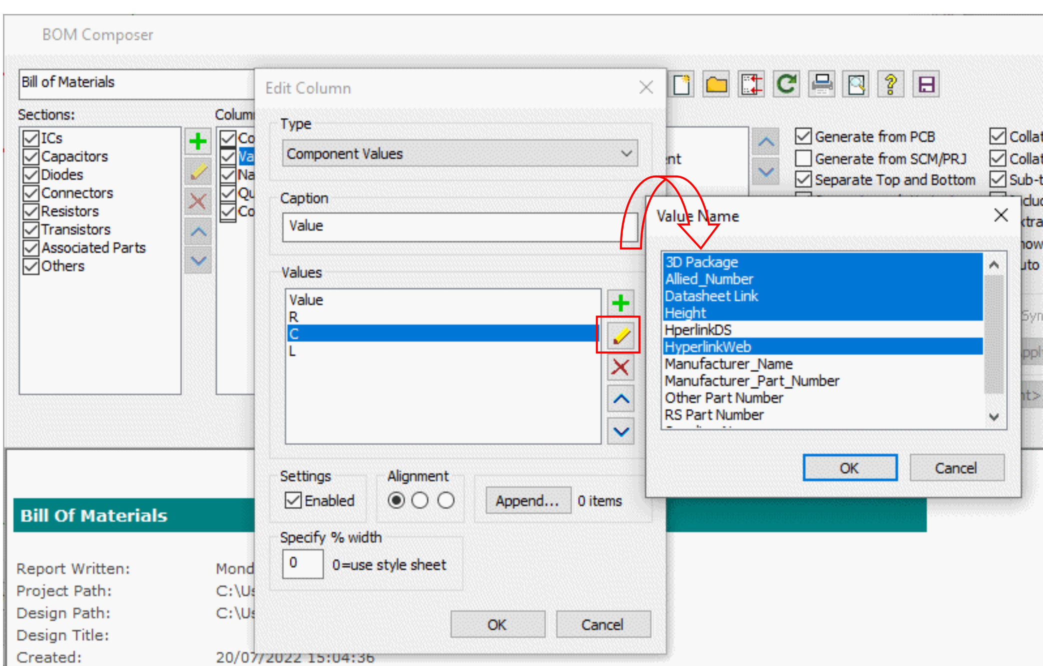
You can now change the precision of the component cost.



BOM Multi-select Value Names

Available in subscription package: Creator, Engineer

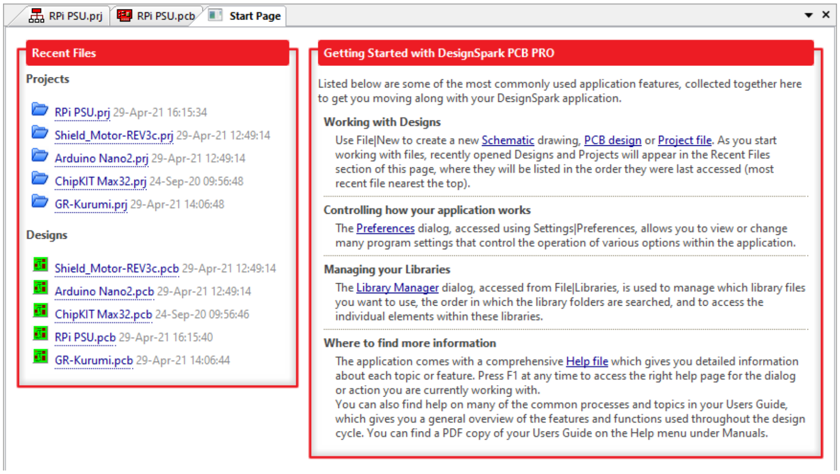
When setting up the **Values** column, you can now select and add multiple Value names using the **Shift** and **Ctrl** keys.



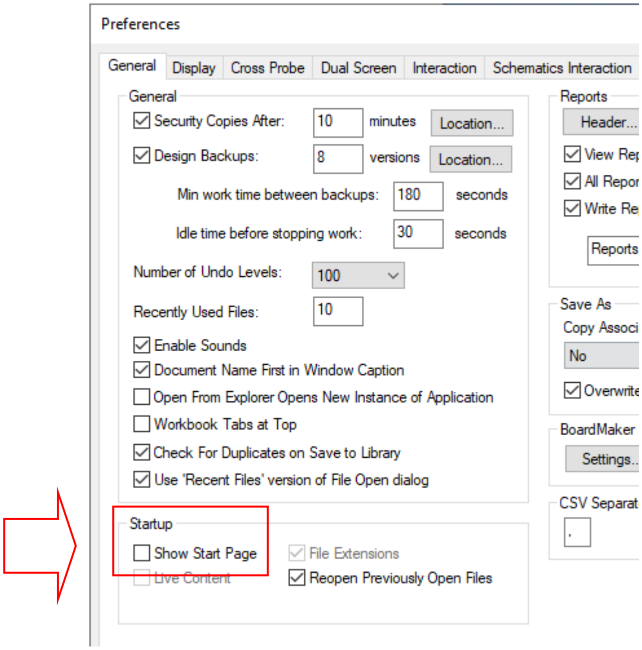
Enhanced Start Page

Available in subscription package: Creator, Engineer

As an addition to **Creator** (it is already available in **Engineer**), there is a more enhanced **Start** page where more information is displayed about DesignSpark. The information is automatically taken off the DesignSpark web site.



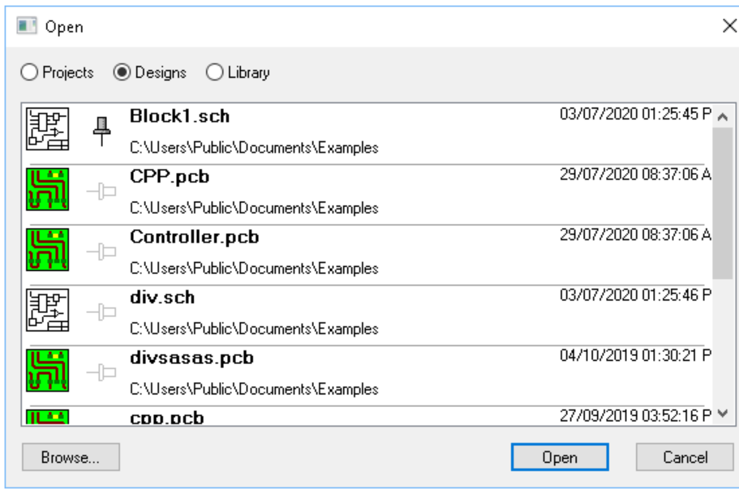
This page can be disabled or enabled using the **Settings**, **Preferences** dialog under **Startup**.



Improved Open Files Dialog

Available in subscription package: Creator, Engineer

As an addition to **Creator** (it is already available in **Engineer**), the **File Open** dialog that provides access to **recent Projects**, **Designs** and **Library** items has been added. This can also be optionally changed back to the standard **Open** dialog using **Preferences** if required (see below).



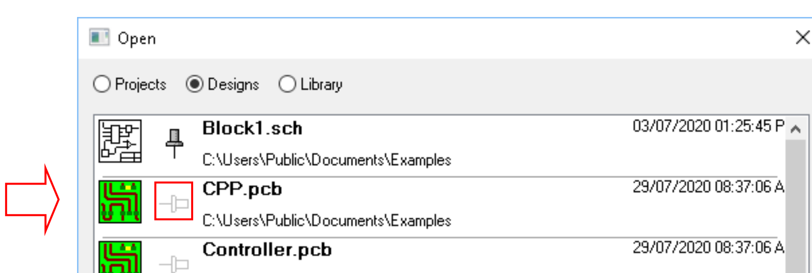
By using the **Ctrl** key, you can select or deselect more than one file at a time to open.

Once the file required is selected and press the **Open** button.

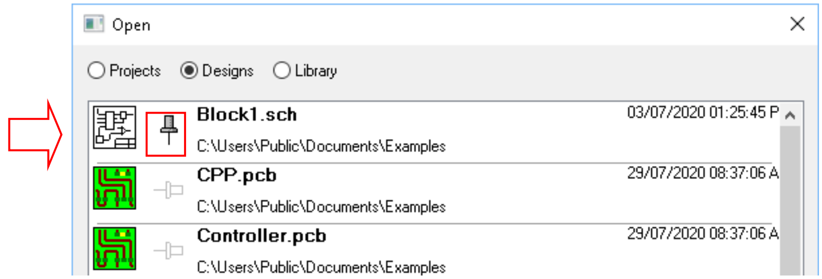
The **Browse** button will enable you to call up the standard **Open** dialog to search for your required design file.

'Pinning' items in the list

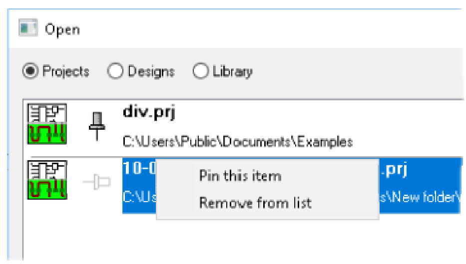
This new dialog includes the ability to 'pin' favourite items to the top of the list. This can be done by either clicking on the small pin next to the item name or by selecting the item once and right clicking to reveal the context menu.



On selection, the pin changes from unpinned to pinned:

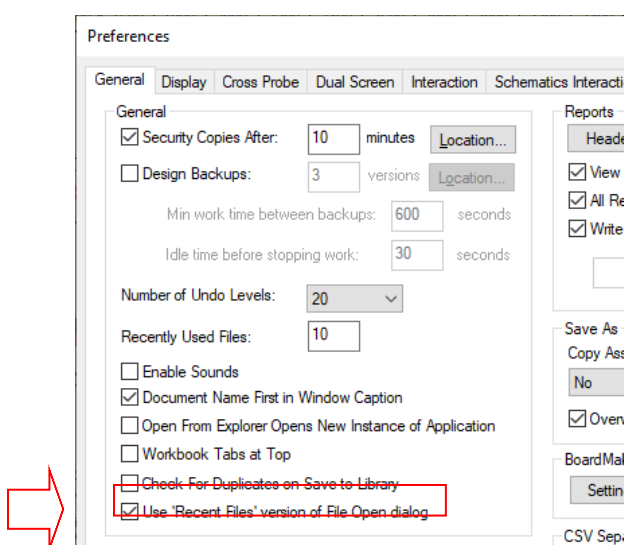


As well as **Unpinning this item** from the list, the context menu also enables you to **Remove** the item from the list (remember, the list is all recently used not a whole list of design items). There's no maximum number of entries for this feature, so it's up to you to remove any files that are no longer relevant. Use the **Remove from list** option on the context menu to do this.



Disabling this option

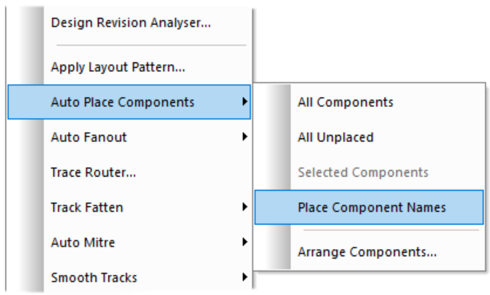
If you wish to disable this option and just use the standard dialog, uncheck the option **Use Recent Files version of the Open dialog** check box in **Preferences** and **General**.



Place Component Names in PCB

Available in subscription package: Creator, Engineer

There is a command on the **Tools** menu, under the **Auto Place Components** option, called **Place Component Names**. This will run through all the components in the design (or only the selected ones) and will rationalise the orientation and position of all the value positions to avoid other obstacles, such as Pads or Vias. Doing this is desirable because names are usually printed in silkscreen and the paint will potentially block the drill holes or pads.



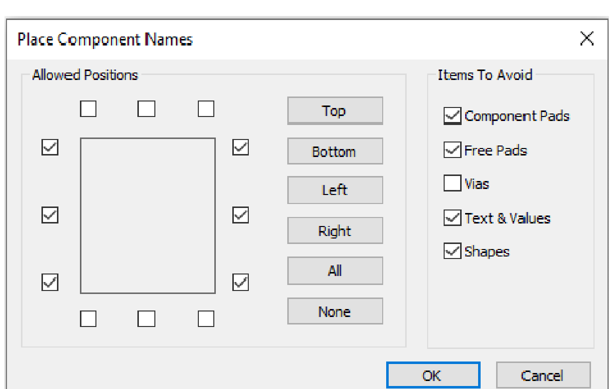
The means you can place all of your component names in a PCB, based on the settings in this dialog. If you have rotated components, the component name will also be rotated. This dialog enables them all to be repositioned and rotated automatically.

Use the check boxes in **Allowed Positions** to choose from the 12 available positions around the component for the names. The buttons to the right provide you with a quick way of setting multiple choices. The component position is the one used on the component in the design and not the symbol origin.

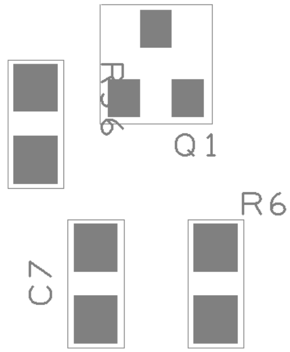
Use the **Items To Avoid** settings to control which items are avoided or ignored while trying to place the names.

Positioning Names

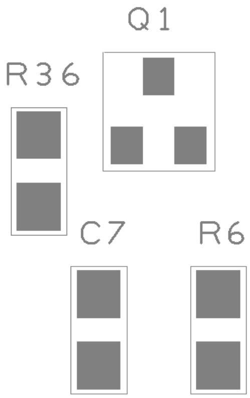
Each component in turn is processed, rotating the name back to a standard horizontal angle and then attempting all chosen possible positions to find a free space. This doesn't guarantee that all names will be repositioned as space may be restricted, but on most boards, it should manage to standardise the readability of most names.



In the example below, the component position setting has been set to the top right corner (the Allowed Position), below is the starting design with a good mix of component names with various rotations and pad violations:



The image below shows the affect after running this tool:



Chapter 5. New Features in DesignSpark PCB V11.0 (Engineer)

Differential Pair Track Editing

Available in subscription package: Engineer

Overview

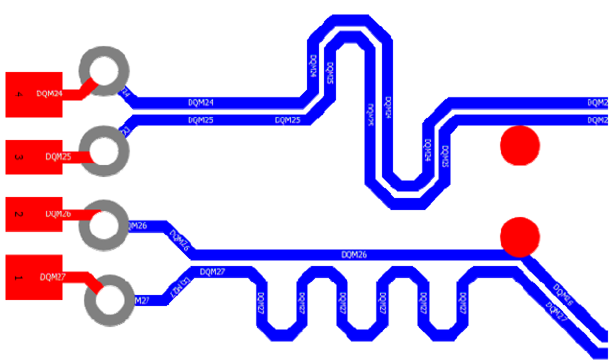
You can now use **Diff Pair** from the **Edit** menu to add **Differential Pairs** to your design.

Diff Pairs are enabled and controlled from the **Design Technology** dialog and **Net Class** tab.

Controls are available for separate **Diff Pair Gap** and **Skew Length** values for pairs on a **Net Class**.

What is a Differential Pair?

A Differential Pair is a term used in PCB design to define a pair of tracks that carry signals that are transmitted down them. These are usually tightly coupled tracks; one carries a positive signal and the other carries the returning signal, this tight coupling balances the pair.



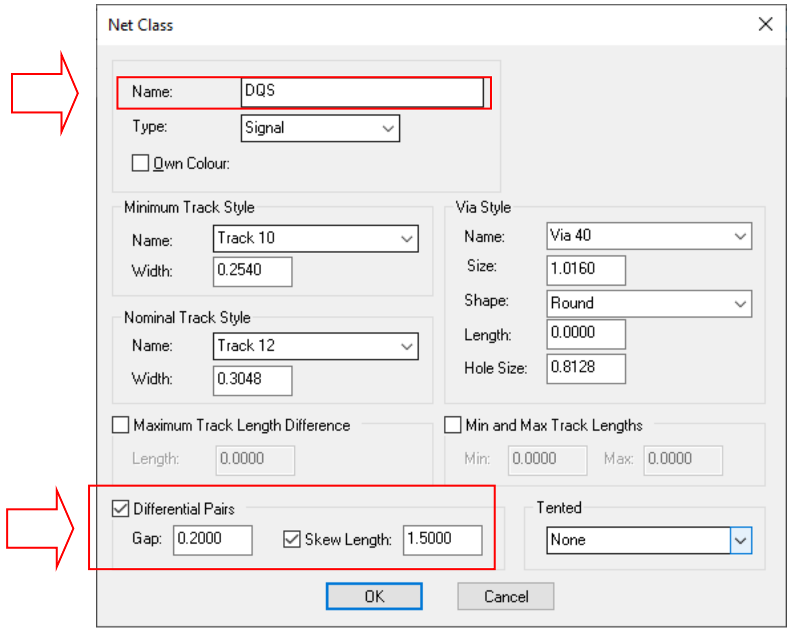
Setting up Diff Pairs

Two items to enable Diff Pairs:

Diff Pairs are defined using **Net Class** settings within your **Design Technology** and they must be enabled for use by selecting the **Differential Pairs** check box.

Once **Differential Pairs** are enabled on a **Net Class**, the application will automatically detect pairs of nets using this Net Class and then their **Net Names**. It searches for mathcing pairs of nets with the suffix **_P** and **_N** and automatically designates those as a pair, for example, DQS_P and DQS_N. If you had DQS1_P and DQS2_N for example, these would not match as a legal Diff Pair.

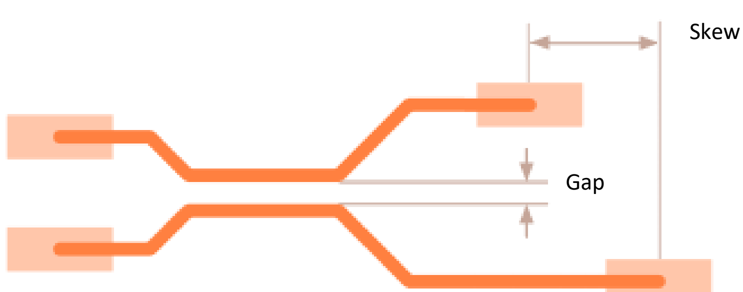
The Net Class dialog allows you to enable **Differential Pairs**:



Once enabled, controls are available for separate **Diff Pair Gap** and **Skew Length** for pairs on the **Net Class**.

A **Gap** different to the main Track to Track Spacing rule may be specified for specific Diff Pairs. This is the gap between the edges of the tracks of a pair and overrides the normal Track to Track Spacing rule.

The **Skew Length** value between each pair can be set, this effectively enables you to set a maximum delay distance. This can be as long or as short as required depending on your timing requirements.



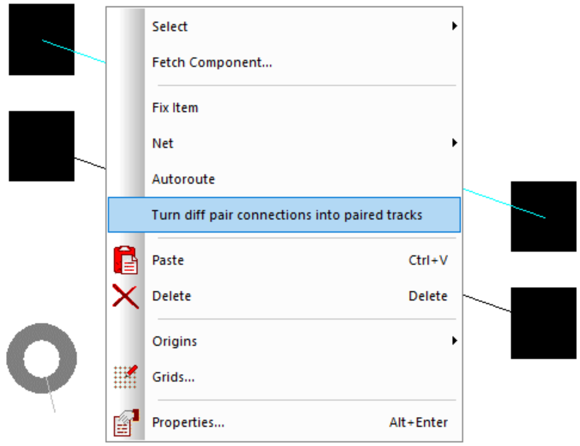
Other length rules for a Net Class can also be utilised within a Diff Pair, such as **Min/Max Track Lengths**.

Creating Diff Pairs

When a Diff Pair is added to the design, it will be the actual centre of the two lines that is on grid, not the Diff Pairs themselves. This means you should try and use grid multiples when routing so that odd segments are avoided.

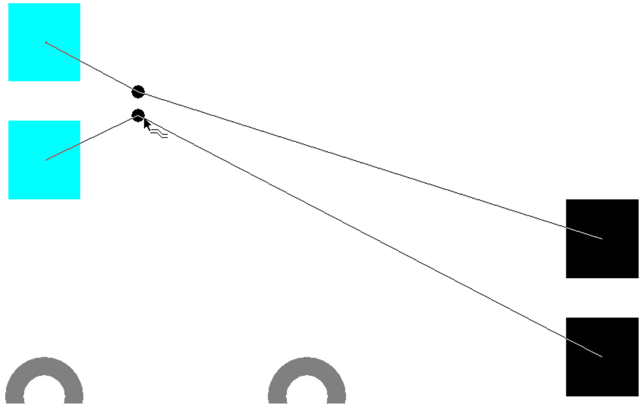
To start a Diff Pair, you can choose a number of options:

- Pick one of the pads of the Diff Pair connections and choose **Turn diff pair connections into paired track** from the context menu.



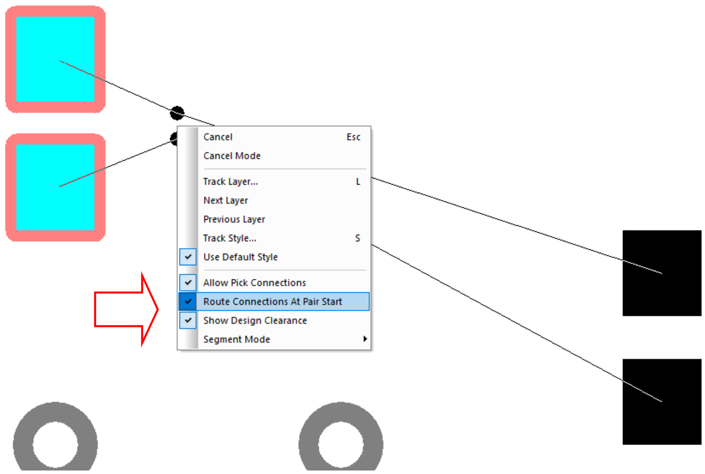
- Pick either the connection or Pad of the Diff Pair connections and select **Diff Pairs** from the **Edit** menu.

Once the **Diff Pair** option is selected you will see two ‘dots’, these present the start of the Diff Pair, a modal cursor indicated you are in ‘pairing’ mode.



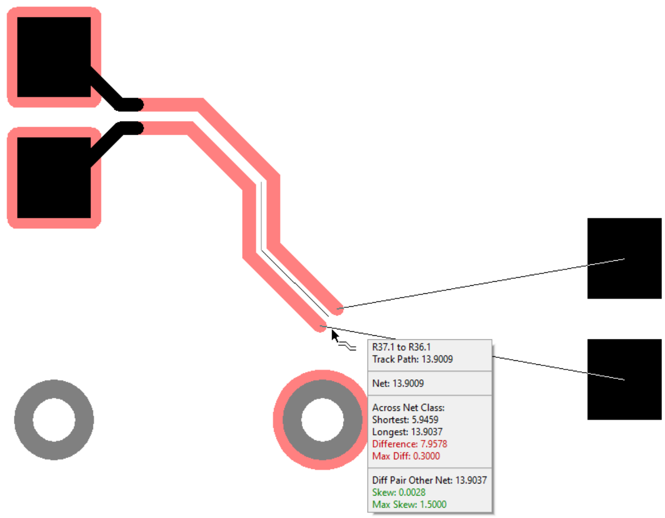
From here, you can ‘draw’ in your pair. This operates the same as when adding tracks with most tracking modes available; **Change Layer**, **Segment Modes, Display Clearances** etc. These are available on the context menu.

When you see the start points, you can also immediately right click to view the context menu. This contains the **Route Connection At Start Point** mode.



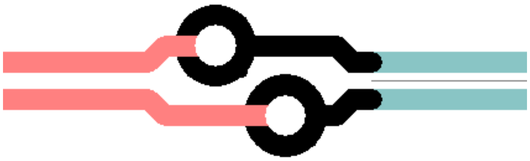
When this is selected, the start of the Diff Pair will be taken from the pads and will automatically route out (where it can) to the Diff Pair tracks.

Draw the tracking in as required:



During routing, this uses the **Diff Pair Gap** (on the Net Class, if defined) or if not defined, it uses the **Track to Track spacing** for the current Net Class and Layers (this way you can have different gaps on different layers).

During editing, you can use **Change Layer** using the context menu option or <L> key. Two vias are inserted along with suitably spaced tracking. When using **Change layer,** if not **All Segments** then it inserts a 45-degree via pattern where vias are separated using the **Via to Via Spacing**.

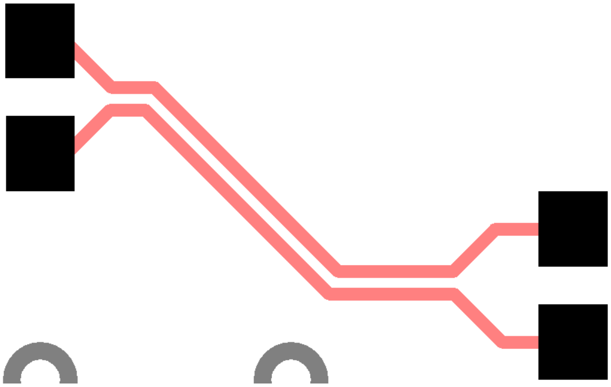


Diff Pairs – Double Click mode

While editing a Diff Pair, it uses the **Edit Track** options in **Preferences** to decide what to do when **double-click** is used to finish, i.e. **Finish Here** or **Auto Complete**.

Use the **Show Legal Completion Path** option to identify the path that will be taken on completion.

Using **Auto Complete** will enable the pairing to finish on the target pads for you:

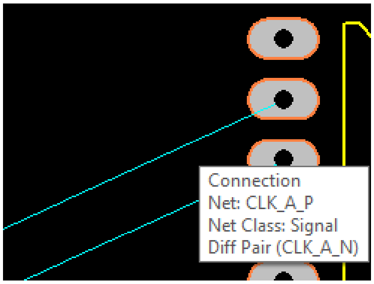


Diff Pairs – Partial routing

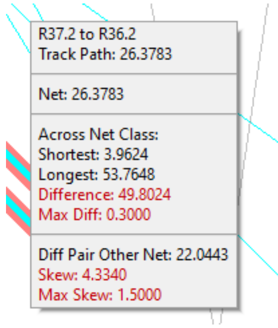
If a diff Pair has been partially completed, within Add Diff Pair mode, you can pick the two junctions at the end of the paired section to start from. Move the gather point back over the junctions to add paired segments to the existing ones and continue Diff Pair routing.

Diff Pairs – Design Tooltips

When selecting a connection that forms part of a Diff Pair, **Design Tooltips** will now show information about the Diff Pair if it detects a match on the Net Class name:

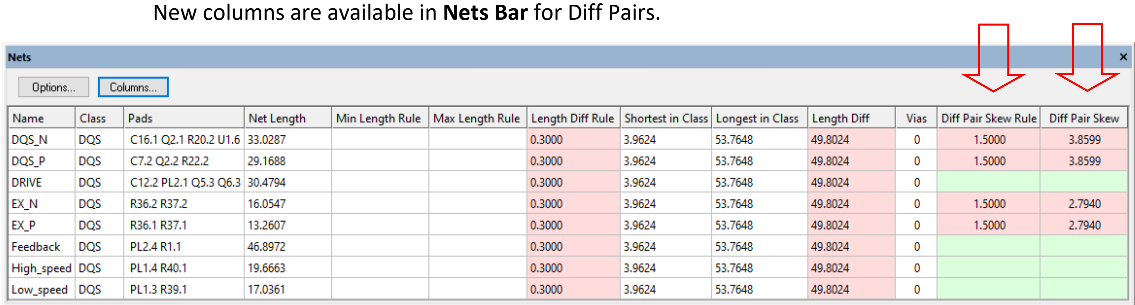


When editing any track that is in a Diff Pair the track limits show the length of the other net and, if defined, the skew and max skew values. The skew is measured as simply the difference in total net lengths.



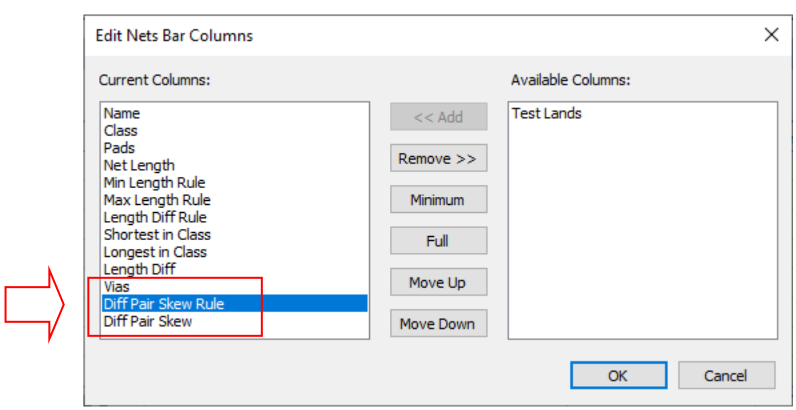
Diff Pairs – Nets Bar

New columns are available in **Nets Bar** for Diff Pairs.



Name	Class	Pads	Net Length	Min Length Rule	Max Length Rule	Length Diff Rule	Shortest in Class	Longest in Class	Length Diff	Vias	Diff Pair Skew Rule	Diff Pair Skew
DQS_N	DQS	C16.1 Q2.1 R20.2 U1.6	33.0287			0.3000	3.9624	53.7648	49.8024	0	1.5000	3.8599
DQS_P	DQS	C7.2 Q2.2 R22.2	29.1688			0.3000	3.9624	53.7648	49.8024	0	1.5000	3.8599
DRIVE	DQS	C12.2 PL2.1 Q5.3 Q6.3	30.4794			0.3000	3.9624	53.7648	49.8024	0		
EX_N	DQS	R36.2 R37.2	16.0547			0.3000	3.9624	53.7648	49.8024	0	1.5000	2.7940
EX_P	DQS	R36.1 R37.1	13.2607			0.3000	3.9624	53.7648	49.8024	0	1.5000	2.7940
Feedback	DQS	PL2.4 R1.1	46.8972			0.3000	3.9624	53.7648	49.8024	0		
High_speed	DQS	PL1.4 R40.1	19.6663			0.3000	3.9624	53.7648	49.8024	0		
Low_speed	DQS	PL1.3 R39.1	17.0361			0.3000	3.9624	53.7648	49.8024	0		

The **Edit Nets Bar Columns** dialog includes **Diff Pair Skew Rule** and **Diff Pair Skew.**

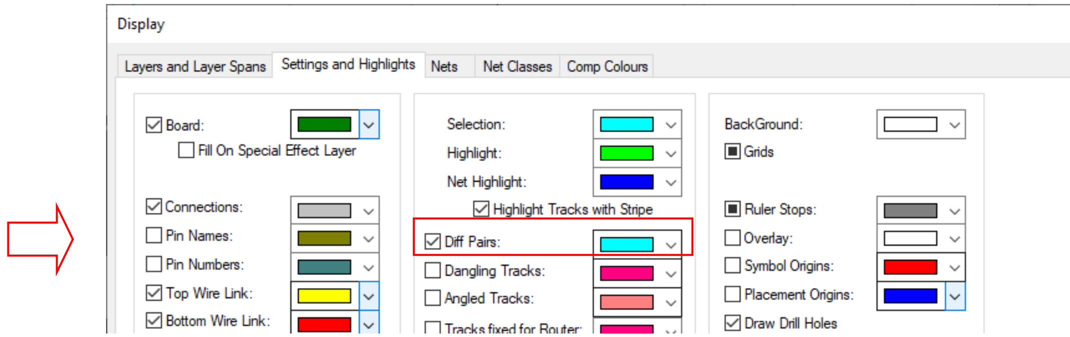


For a Diff Pair net, **Diff Pair Skew Rule** shows the rule on a net if its Net Class has a skew rule defined and **Diff Pair Skew** shows the actual difference between the lengths of the two nets forming the Diff Pair.

As with all Length columns in this dialog, the cells change colour to display whether the Skew is within the limits defined.

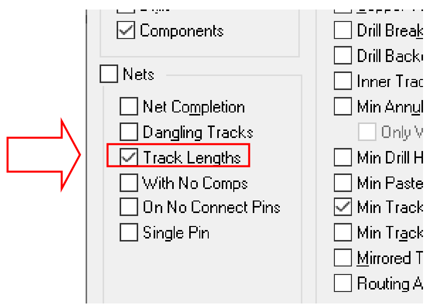
Highlight Colour of Diff Pairs

There is a highlight colour in the **Colours** dialog that allows you to see paired connections and tracks (using the centre stripe) for diff Pairs using a highlight colour to make them more visible.

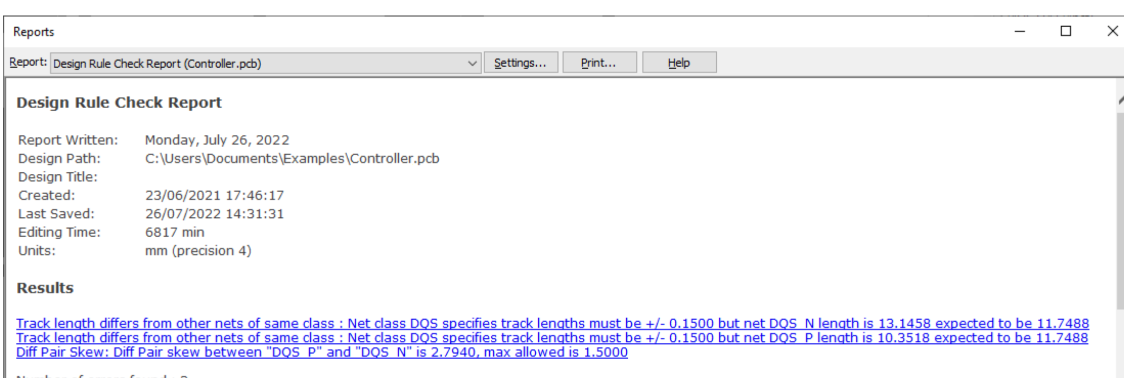


Design Rules Checking of Diff Pairs

The DRC **Track Lengths** check now checks for Diff Pair skew errors.

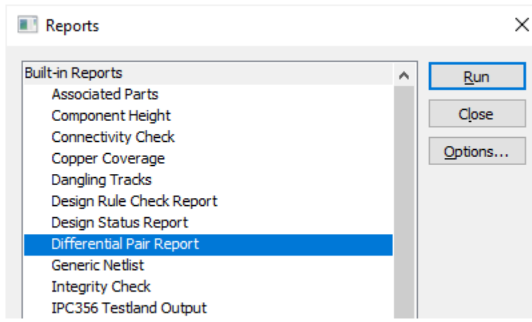


When run, the DRC report shows Skew differences:



Diff Pair Report

A new report is available in the **Output** menu and **Reports** called **Differential Pairs Report**.



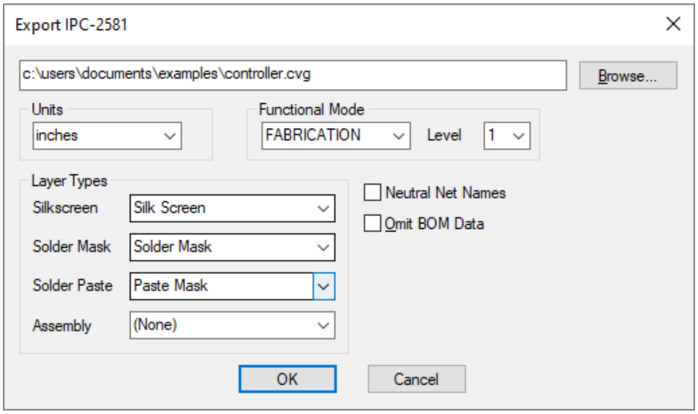
When run, the report will display values relating to the Diff Pair nets in your design:

IPC-2581 Output

Available in subscription package: Engineer

There is an **IPC-2581** export option on the **Output** menu. This option is available in the PCB Design Editor and in the PCB Panel Editor.

IPC-2581 is a modern CAD/CAM data exchange format which can be used as an alternative to Gerber, Excellon or ODB++ for manufacturing your PCBs. IPC-2581A format is exported.



Use **Browse** to change the destination of the output file name and location.

Choose the **Units** required for the IPC-2581 export. From the drop down list, choose **Inches** or **mm**.

Functional Mode

The IPC-2581 export is capable of outputting all or partial design files based on your requirements. From the drop down list, choose between **FULL**, **DESIGN**, **FABRICATION**, **ASSEMBLY** or **TEST**. Each of these modes will give you different portions of the design to suit the process it is intended for.

The **Level** drop down list is available for all outputs except FULL. This enables you to choose the detail level for each functional mode selected. Choose between levels 1, 2, or 3.

Layer Types

In the IPC-2581 format the **Silkscreen**, **Solder Mas**k, **Solder Paste** and **Assembly** layers need to be identified. Use the appropriate drop down list to select the **Layer Types** which will identify these layers. You can select **(None)** if you are not interested in the layer type.

If there are no power planes in the design these options are suppressed.

Neutral Net Names

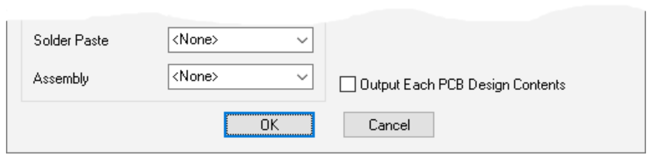
If this is checked then all user defined net names will output as numerical net names. This can be used to hide company specific net names, this will help protect your design IP.

Omit BOM Data

The **Omit BOM Data** switch is available with any **Fabrication Level** set. Use this to remove the BOM data from the IPC-2581 file to hide company specific Parts names, this will help protect your design IP.

Panel Editor Specific IPC-2581 Switches

When running the **Panel Editor**, there is a switch that is only visible when running the IPC-2581 option:



By selecting the **Output Each PCB Design Contents** check box, a copy of each of all the PCBs in the panel are exported. With it not selected, only the contents of one PCB is exported.

IPC-2581 Viewers

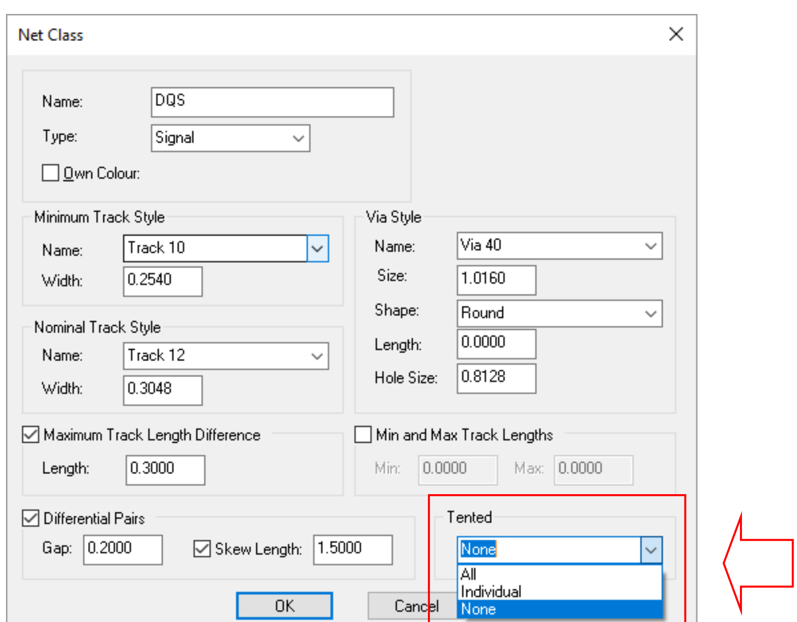
There are IPC-2581 viewers available on the web, these enable you to verify the output before sending to external sources, this is highly recommended.

Tented Vias

Available in subscription package: Engineer

This is a simple method of specifying whether or not vias are to be 'tented'. A tented via is one that is covered with solder resist during the manufacturing stage. In order for this to happen, the via that is to be tented must be identified in the design and a specific output generated.

The **'tented'** property is defined in the **Net Class**. When vias in the design use this Net Class, the output file generated in the **Plotting and Printing** dialog can use this flag as a selection option.



Each Net Class has a new property of **Tented** which can be set to one of three values:

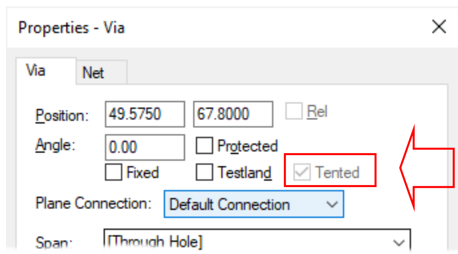
None: vias on this Net Class are automatically not tented so they will appear on plots as normal.

All: all vias on this Net Class are automatically classified as tented.

Individual: the Tented flag is set by you as required. This is set on **Via Properties** for each via

Via Properties

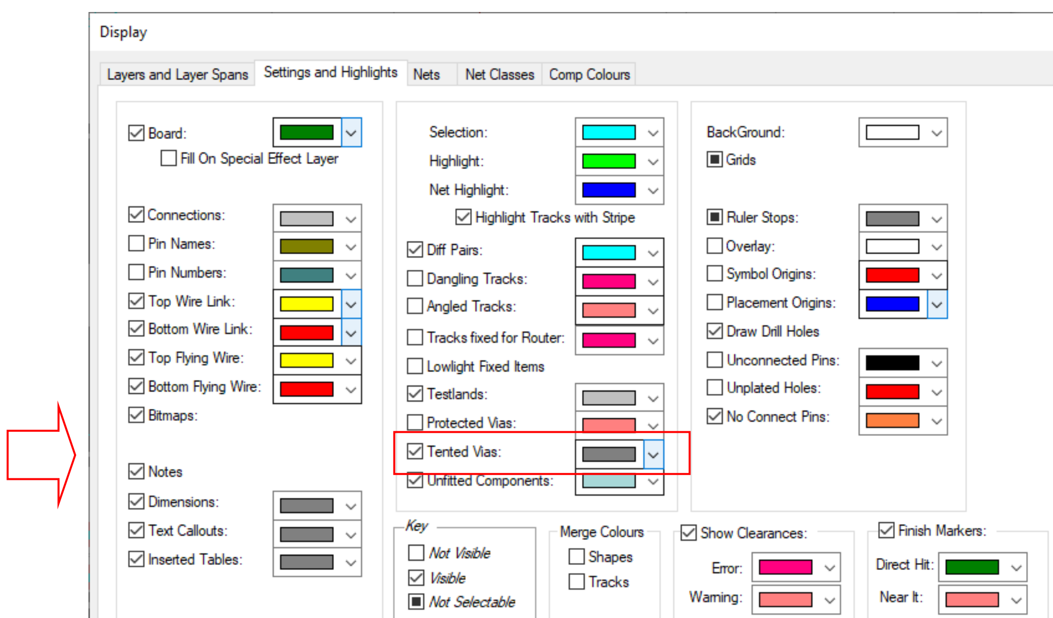
When **Tented** is set to **None** or **All** on the Net Class, the tented state of a via is displayed on the **Via Properties** dialog but the check box is greyed out because it is inheriting its setting from the Net Class.



When the **Tented** property is set to **Individual**, this check box then becomes available for setting.

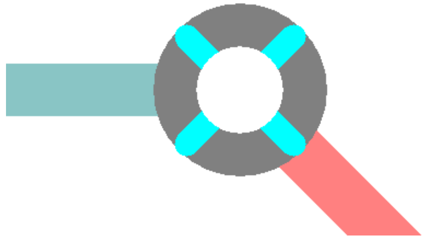
Tented Via Colours

On the **Colours** dialog there is a new check box and colour setting for **Tented Vias**:



Tented Vias in the Design

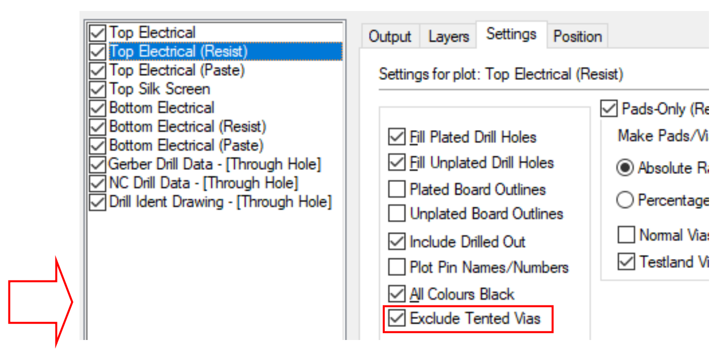
If this highlight is enabled (checked) in the **Colours** dialog, any via that is tented (either individually on **Properties** or from its **Net Class**) will be marked with an X (cross) in the same way as we do to highlight unconnected pads:



Note, this is the only visual indication of the 'presence' of the via shape on a resist layer, the rest of the processing is done at the plotting stage. In other words, if you display only your top resist layer, vias that are marked as tented will still be drawn.

Plotting Tented Vias

In **Plotting and Printing**, there is a new check box on the **Settings** tab for each plot that allows you to decide whether or not to exclude tented vias from that plot.



The default setting for this is unchecked (plots include tented vias) but when auto-generating plots it will try to determine which plots are Resist and will thus tick this box for you. This setting is also saved in **Plot Jobs** so it is easy to prepare a preferred set of plots and settings and save it for future use.

Highlight Net across Project

Available in subscription package: Engineer

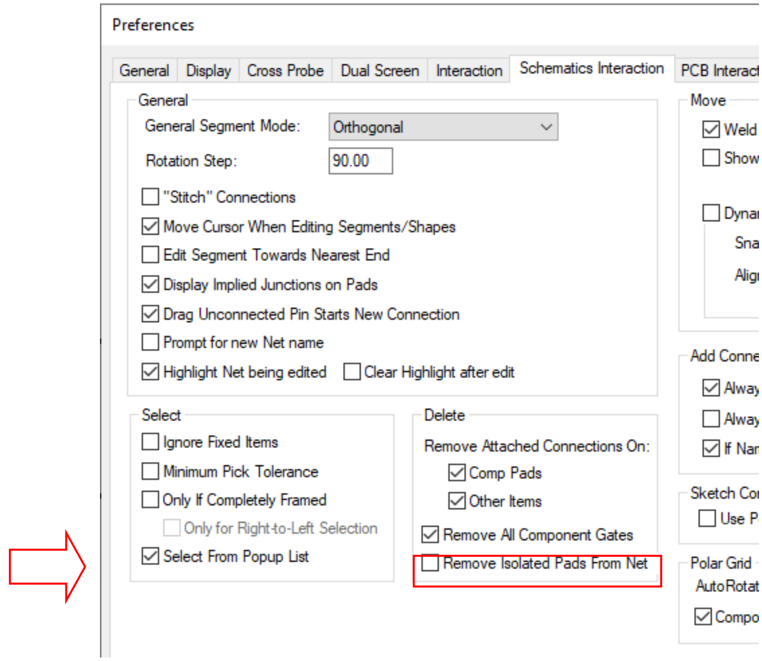
The **Net Highlight** command has been enhanced to work across **all open designs** in a **Project** for user-defined net names.

Delete in SCM, option to keep Net

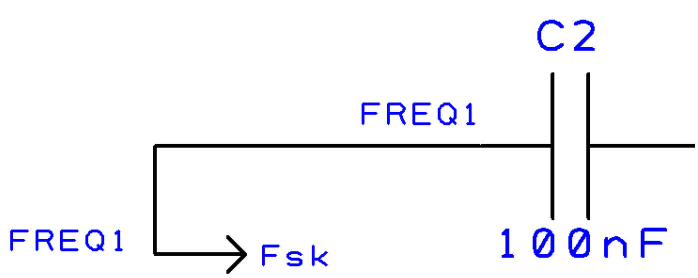
Available in subscription package: Engineer

There is a new check box **Remove isolated pads from net** in the **Delete** settings for **SCM Interaction Preferences**. Previously, this was only available in **PCB Interaction**. The default value is checked (to match current behaviour where net names are removed if isolated).

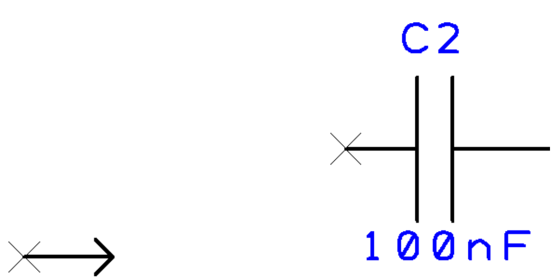
This option is used to (optionally) retain a net name on a pin when deleting the connection to it. The default action is for the net name to be removed.



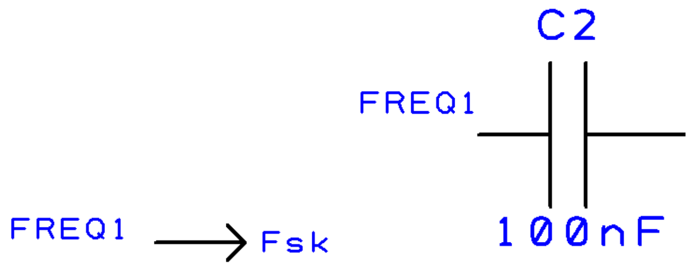
To demonstrate this, we start with the FREQ1 net between two pins:



With the option **checked**, the net and net names are fully removed.



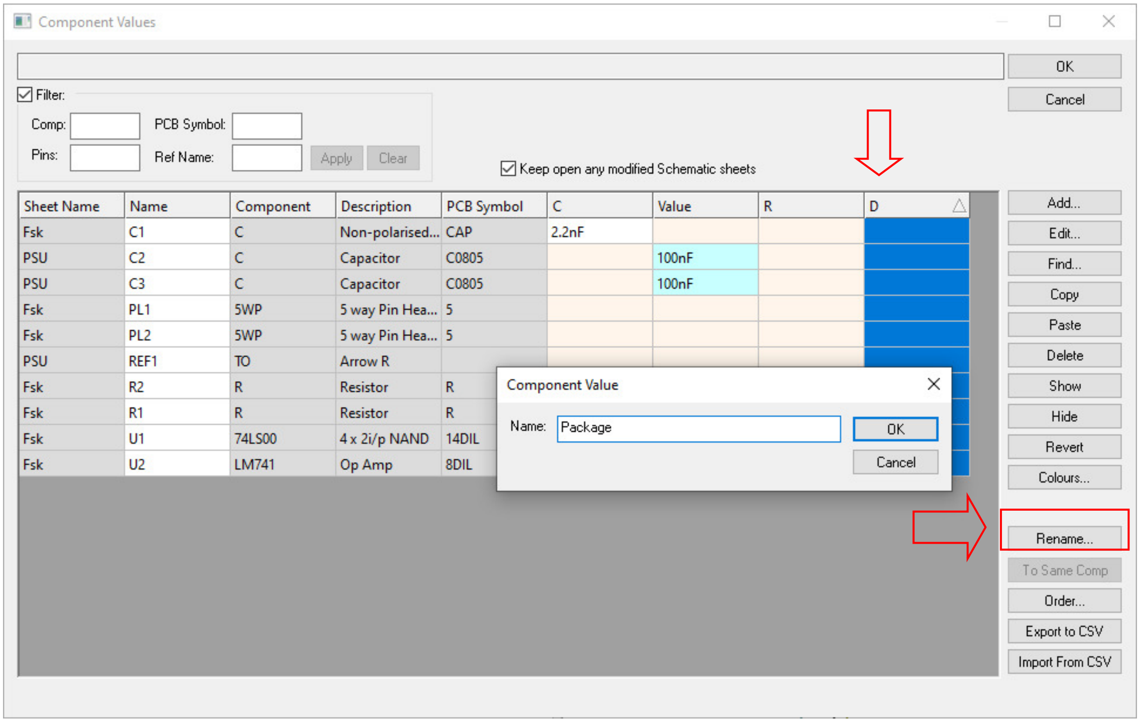
With the option left **unchecked**, the net names on the pins are retained when the connection is deleted.



Component Values – Rename option

Available in subscription package: Engineer

In the **Component Values** dialog, there is a new **Rename** button that allows you to change the name of an existing attribute (value). This is available for both **Library Components** and Components in a **design**.



By selecting the **Value** name in the column header, the **Rename** option becomes available. Selecting this provides you with a **Name** dialog from which to type the new name.

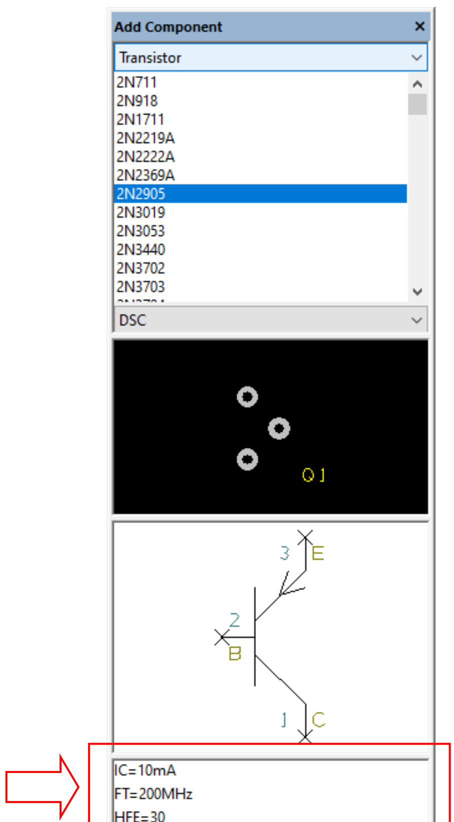
The result is like this:

Sheet Name	Name	Component	Description	PCB Symbol	C	Value	R	Package
Fsk	C1	C	Non-polarised...	CAP	2.2nF			
PSU	C2	C	Capacitor	C0805		100nF		
PSU	C3	C	Capacitor	C0805		100nF		
Fsk	PL1	5WP	5 way Pin Hea...	5				
Fsk	PL2	5WP	5 way Pin Hea...	5				
PSU	REF1	TO	Arrow R					
Fsk	R2	R	Resistor	R			20K	
Fsk	R1	R	Resistor	R			10K	
Fsk	U1	74LS00	4 x 2i/p NAND	14DIL				SM
Fsk	U2	LM741	Op Amp	8DIL				DIL

Note, when used in the design and Values names are renamed, this is not reflected in the component Library.

Add Component Bar shows Values

Available in subscription package: Engineer

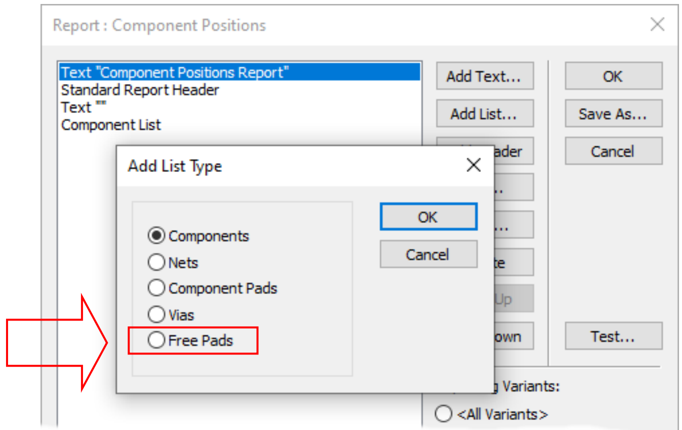


The **Add Component Bar** now displays values of the selected component in a list below the PCB and SCM preview windows. This is in common with the Add Component and other similar dialogs.

Free Pads in Custom Report

Available in subscription package: Engineer

For **User Reports**, you can now produce a report showing the properties of **Free Pads**:



This is available when you use the **Add List** option.

When run, the report can be used to produce a list of free pads that may have been marked as use for Testlands or fiducial markers and suchlike.

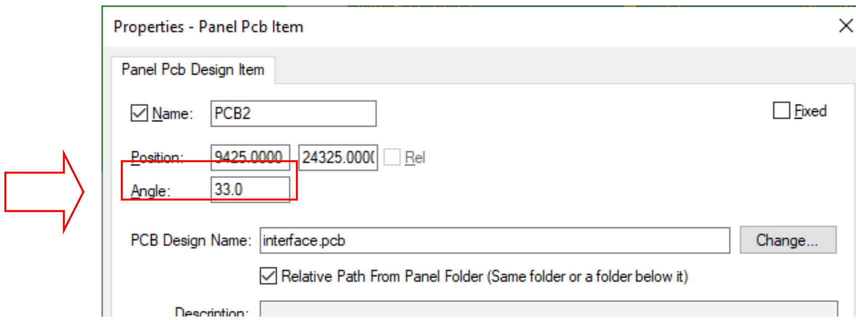
Report: Free Pad Positions CSV (chipKIT Max32.pcb)

Position X	Position Y	Testland
66.04	101.60	N
66.04	53.34	N
140.97	101.60	N
147.32	53.34	N
24.50	115.05	N
27.20	115.05	N
30.27	115.05	N
29.75	112.72	N
29.75	109.65	N

Angle of PCB Item in a Panel Design

Available in subscription package: Engineer

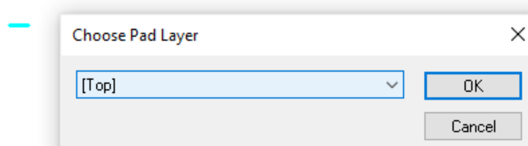
In a **Panel** design, in **Properties** for a **Panel PCB Design** item, you can now set the **Rotation** to be any angle you want, you are no longer restricted to multiples of 90 degrees.



Shape To Pad

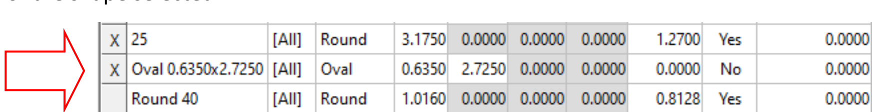
Available in subscription package: Engineer

The new context menu command **Shape to Pad** in **Select** mode is available when there is at least one **single line shape** or **unattached track** selected.



This command will ask you to choose the layer for the pads (based on the layer of the first valid selected line), then it will take each line and replace it with an oval pad of the same dimensions on the chosen layer.

For each pad created, a new **Pad Style** will be added to the **Design Technology** using the dimension of the shape selected.



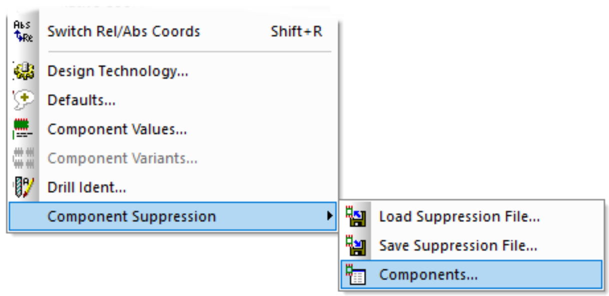
X	25	[All]	Round	3.1750	0.0000	0.0000	0.0000	1.2700	Yes	0.0000
X	Oval 0.6350x2.7250	[All]	Oval	0.6350	2.7250	0.0000	0.0000	0.0000	No	0.0000
	Round 40	[All]	Round	1.0160	0.0000	0.0000	0.0000	0.8128	Yes	0.0000

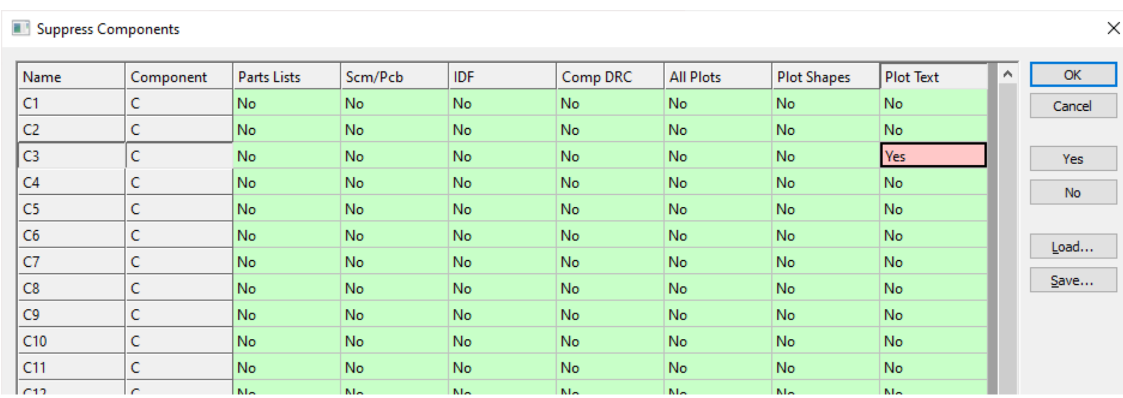
This function is intended for use after importing Gerber files and the need to convert short track segments into pads. These would be isolated segments that would have originally been pads in the host system before being written to Gerber. During import, this cannot be done automatically and is ideally suited to a manual 'by eye' observation, this new option will aid this.

Component Suppression dialog

Available in subscription package: Engineer

A new command is available on the **Settings** menu and **Component Suppression** option called **components**. On selection, this displays a dialog with a table of components and their current suppression settings to make it easy to manage them all in one location. In previous releases, you are restricted to doing them through the **Component Properties** dialog.





The **Yes** and **No** buttons toggle the status of a selected cell. The cell can be toggled by double clicking it also.

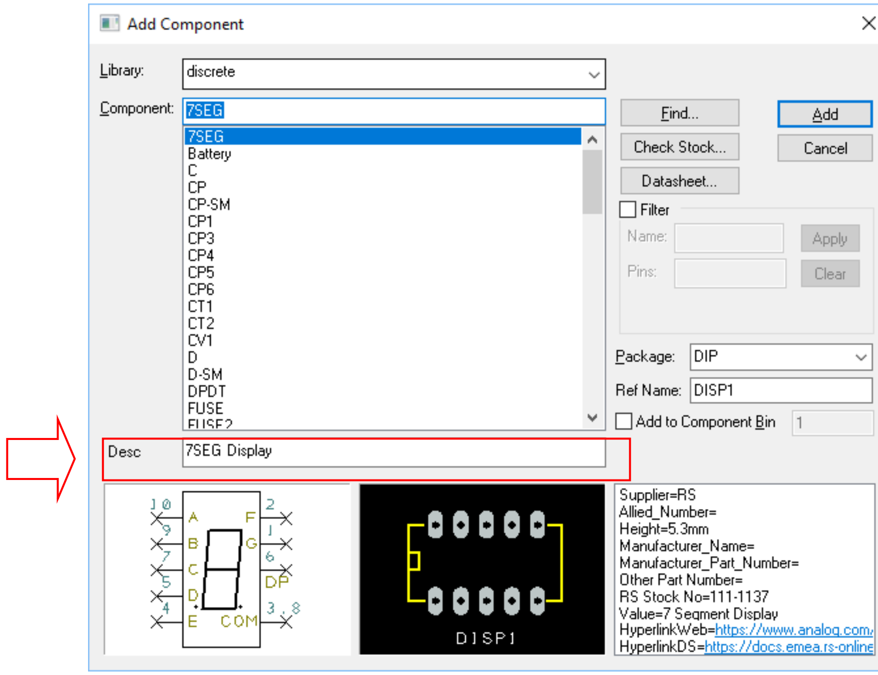
The **Load** and **Save** buttons on this dialog do the same as the existing Load/Save Suppression File commands that were available in previous version. This allows you to save settings to a text file, manipulate them, and read them back in.

Component Description on dialogs

Available in subscription package: Engineer

On dialogs where components are **Added** or **Replaced**, the **Description** from the component is now displayed along with the existing information (package etc.).

This change applies to **Add Component**, **Change Component** (from **Properties**) and **Replace Component**.

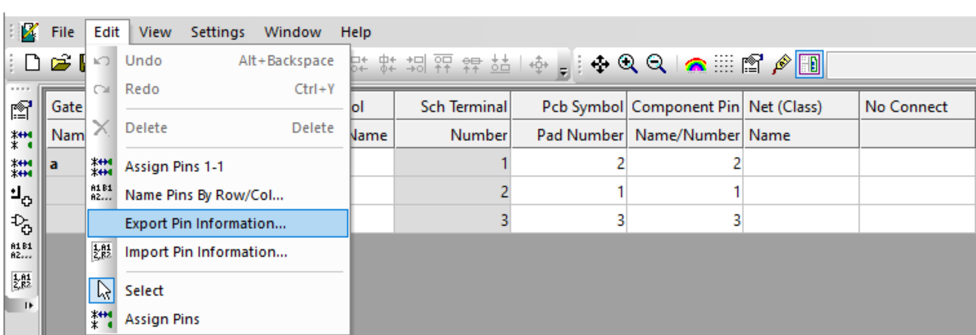


Export Pin Information from Component Editor dialog

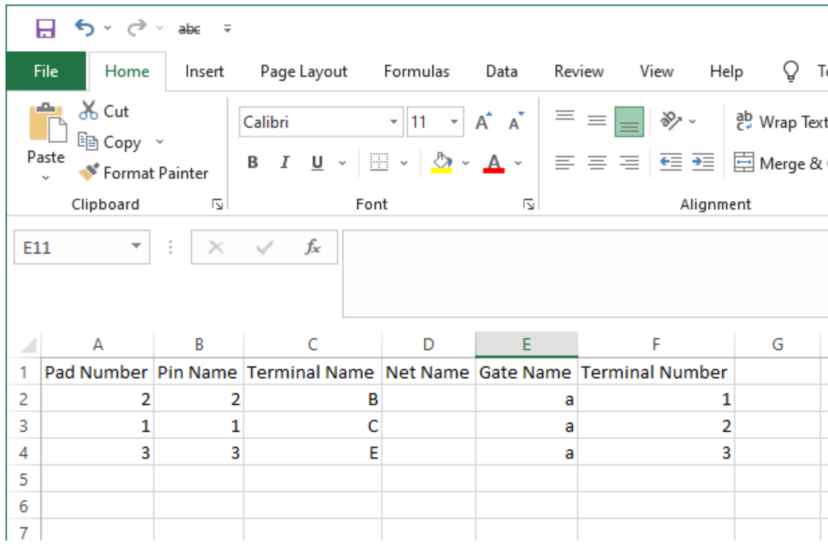
Available in subscription package: Engineer

In the **Component Editor**, it is already possible to import a CSV file of pin information to help assign names and numbers to pins in the component. You can now **Export Pin** information in a CSV.

The new **Export Pin Information** command is available in the **Component Editor** on the **Edit** menu.



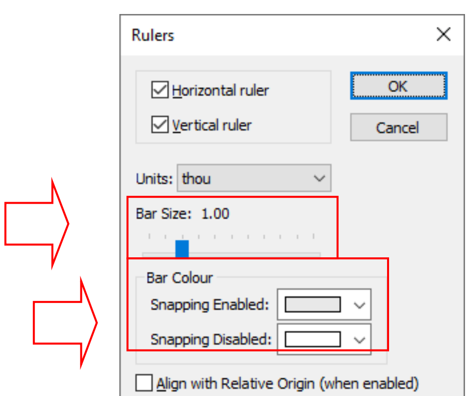
This helps you prepare this data by outputting a CSV file containing the existing pin details from the component, which you can then edit in a program like Excel, and read back into the component using the existing **Import Pin Information** command.



Changes to Rulers

Available in subscription package: Engineer

There are two new settings on the **Ruler Settings** context menu for **Bar Size** and **Ruler Colour**.



Bar Size

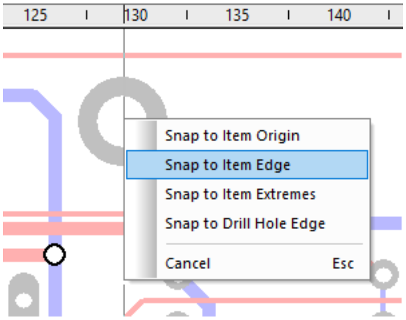
The **Bar Size** slider is used to increase or decrease the size of the ruler bar. This can be useful if you have a 4K monitor or if you wish to view it larger on a regular monitor.

Ruler Colour

Use the **Bar Colour** to change the background colour of the Ruler. Care should be taken when changing this so that you don't loose the text colour on a dark background.

Interactive Ruler Stops

When you drag a ruler stop you can now see it move dynamically. You start the drag from the bar with menu items for additional functionality available on the context menu if you right click the mouse:



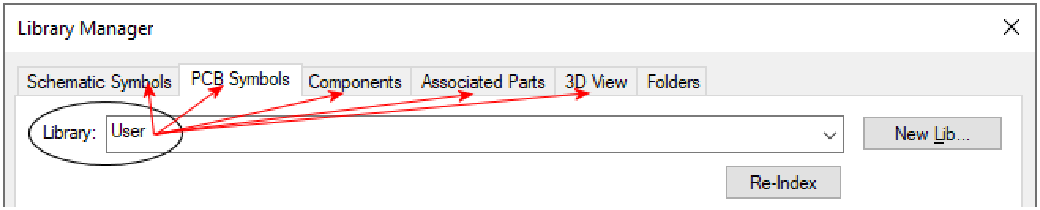
There are options on the context menu to snap the ruler stop to the item you finish on. The snap to origin, extremes and drill edge work the same as the existing options from the ruler bar. The snap to item edge option will snap to the closest edge on the item.

There are options on the context menu to snap the ruler stop to the item you finish on. Use the **Snap To Origin**, **Snap To Item Edge**, **Snap To Item Extremes** and **Snap To Item Drill Edge Hole** to snap to exact item positions.

Library Editor Synchronise Library Names

Available in subscription package: Engineer

When using the **Library Manager**, if a specific library is selected, **User** on the **Component** tab for example, when you now swap to another tab, **Schematic Symbols** for example, the program will try and choose the corresponding User symbol library if the same name exists. The Library Manager synchronises the library name across all tabs including 3D Packages and Associated Parts.



Copper Coverage Report

Available in subscription package: Engineer

From the **Output** menu | **Reports** you can now write a **Copper Coverage** report.

Balancing the use of copper across each layer and between layers of your PCB can help avoid manufacturing issues such as warping or twisting of the PCB. The **Copper Coverage** report can assist you in making an assessment of the copper balance in your design.

The report shows you the approximate area of copper on each layer of the board, along with a figure for the percentage of the board area that is covered by copper. The area of copper is calculated from the area occupied by open or closed/filled copper shapes, and routed tracks, so it won't be a precise “geometric” figure for the total copper but it will be close enough for you to determine whether there are potential issues to address before going to manufacture.

A sample report might look like this:

Board Area

5419.34

Copper Coverage

Name	Filled Copper	Hatched Copper	Copper Lines	Tracks	Total	Coverage
Top Copper	248.85	0.00	0.00	602.10	850.95	15
Layer 2	4803.78	0.00	0.00	22.62	4826.40	89
Layer 3	4739.14	0.00	0.00	0.00	4739.14	87
Bottom Copper	322.74	0.00	0.00	462.28	785.02	14

Coverage values are approximate total area of copper items for each electrical layer. Items designated as copper for the purposes of this report include shapes (both open and closed) and routed tracks. Power planes assumed to be fully covered. Coverage shows approximately what proportion of board is covered by this copper to assist you in managing copper balance across the board. All measurements in mm squared.

Note: it should be noted that as stated in the report, the coverage values are approximate as a best calculation. Power plane layers are assumed to be full coverage.

There are further online resources about copper coverage and balancing available here :-

1. Copper balance: https://www.multi-circuit-boards.eu/en/pcb-design-aid/copper-balance.html

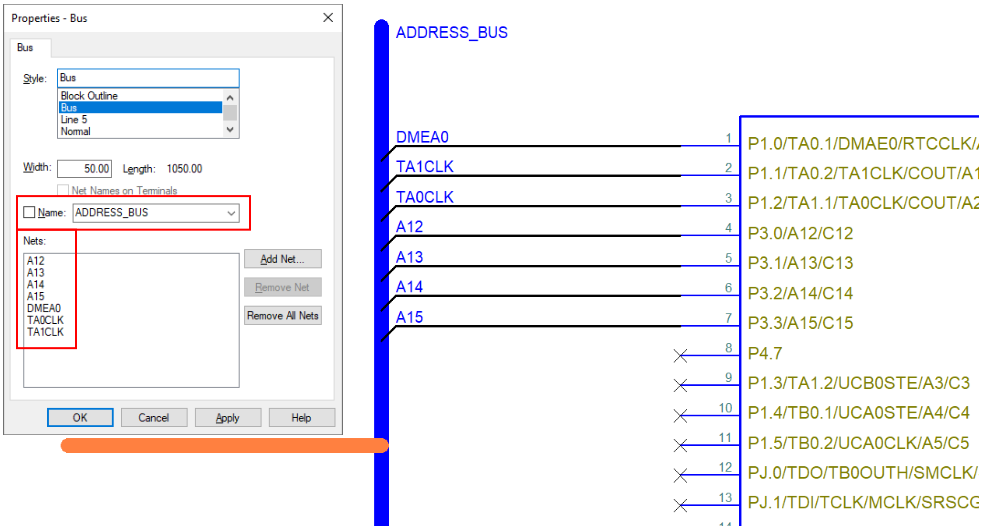
2. Tool and tips for good copper distribution: https://www.eurocircuits.com/blog/the-influence-of-copper-distribution-on-pcb-quality/

Acknowledgment and copyright to Multi Circuit Boards and Eurocircuits.

Inherited Net Name and Bus Name from Bus

Available in subscription package: Engineer

When starting or ending a **Bus** on the segment of an existing **Closed Bus**, it will now inherit the **Net Name** and **Bus Name**.



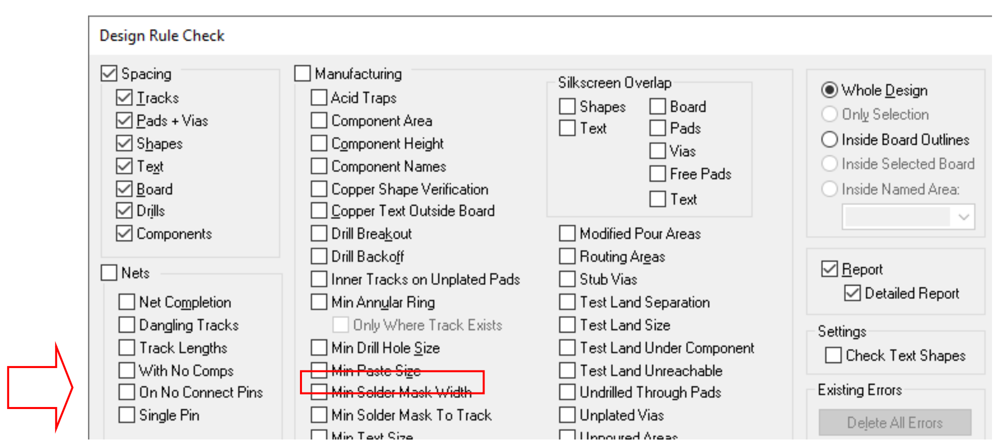
New DRC & DFM Checks

Available in subscription package: Engineer

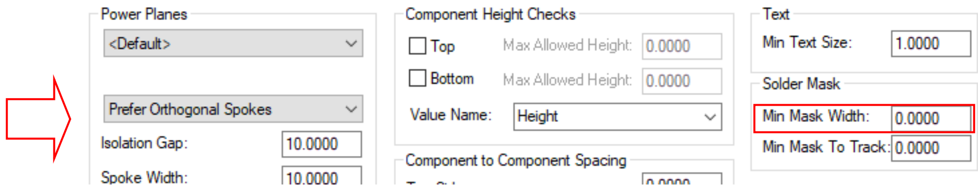
Additional checks have been included to enhance the Design Rules Checking for various design rules and Design For Manufacturing (DFM). When run, these checks will validate your design so that you have an additional level of confidence when sending it off for manufacture.

Min Solder Mask Width

A new check for **Min Solder Mask Width** has been added to the **Manufacturing** section in **DRC**.



The minimum mask to track value can be set in the **Design Technology** under **Rules** and **Solder Mask**.

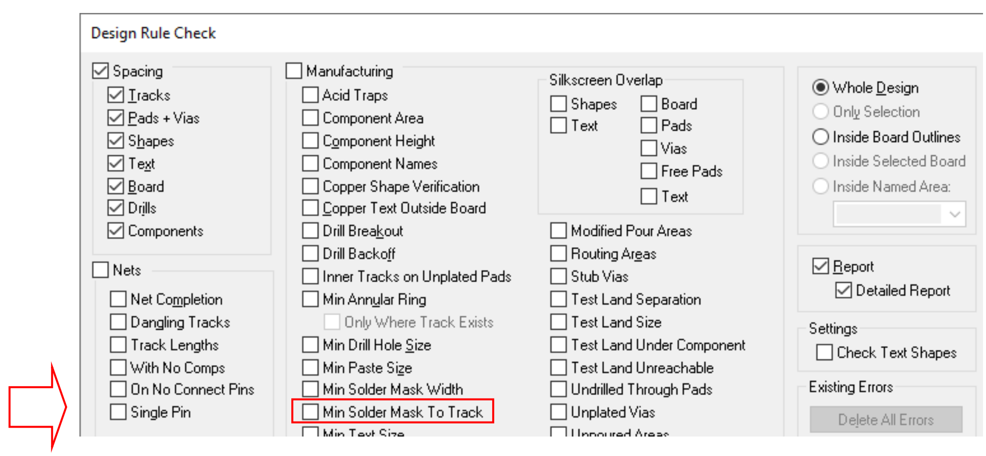


Long thin slivers of solder mask can potentially peel off during manufacturing, especially when under heat and should therefore be avoided.

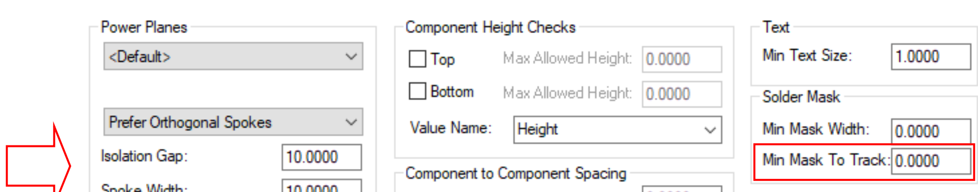
Minimum Solder Mask To Track

Available in subscription package: Engineer

A new check for **Min Solder Mask To Track** has been added to the **Manufacturing** section in **DRC**. The solder mask to track check will check any layer with a layer type that has the Min Mask To Track value defined (this value can only be defined on Non-Electrical layer classes).



The minimum mask to track value can be set in the **Design Technology** under **Rules** and **Solder Mask**.

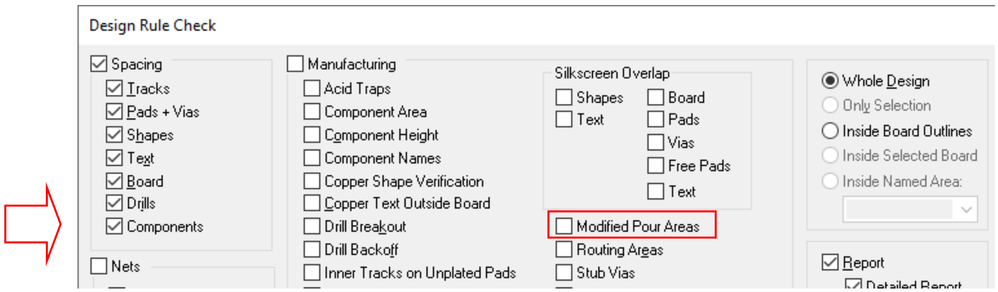


This check ensures there is a minimum distance between a track and a solder mask opening (not including the opening for the tracks start and end nodes if they exist).

Modified Pour Areas

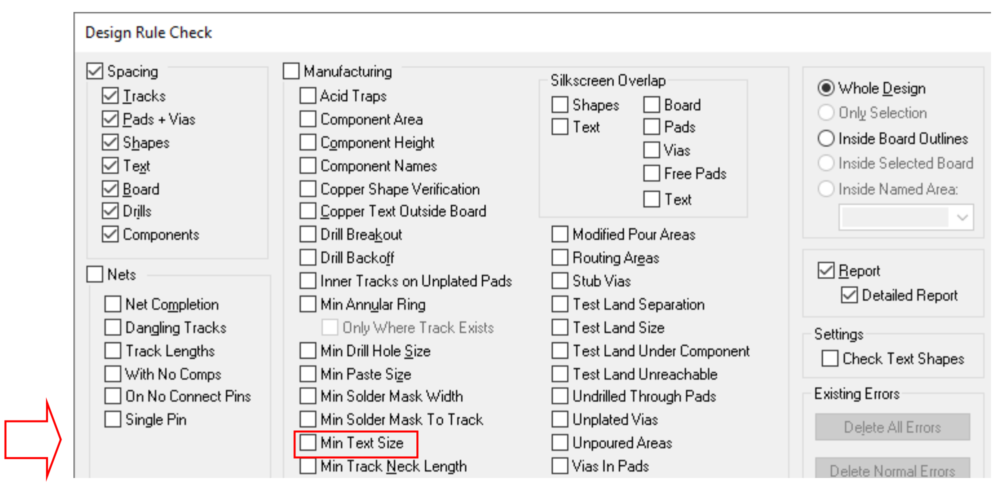
Available in subscription package: Engineer

A new check for **Modified Pour Areas** has been added to the **Manufacturing** section in **DRC**. When run, this check will give an error if a area has been modified (i.e. the area shape has changed, items inside area have changed or moved etc.) but the area has not been repoured. It requires the area to be poured first though. In addition, also run the additional existing DRC check - Unpoured Areas.

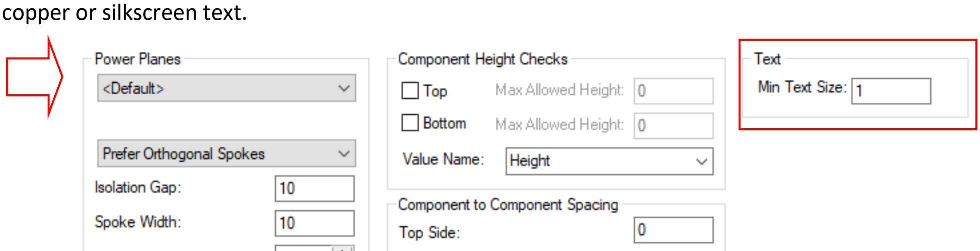


Minimum Text Size

A new check for **Min Text Size** has been added to the **Manufacturing** section in **DRC**.



The min text size is defined in the **Design Technology**, **Rules** dialog on the **Settings** menu. Defining and running this check will ensure that all text added to your design is manufacturable, for example, copper or silkscreen text.

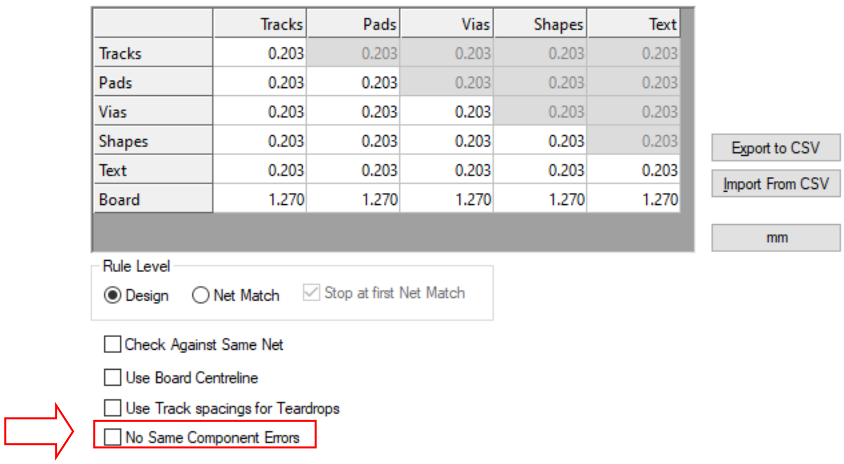


Ignore Same Component Errors

Available in subscription package: Engineer

There is a new setting in **Design Technology** and **Spacings** for **No Same Component Errors**.

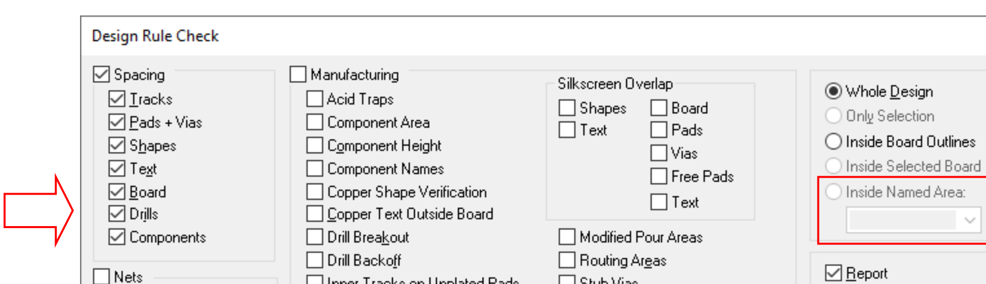
When set, DRC will ignore errors if they are in the same component, for example pad to pad spacing, or Silkscreen overlap checks.



DRC only Check in Named Area

Available in subscription package: Engineer

From within the **DRC** option, there is now a check box to only check for errors inside a specific **Named Area**.



Named areas are added using **Add Component Area** and then named using **Properties** and setting the name there.

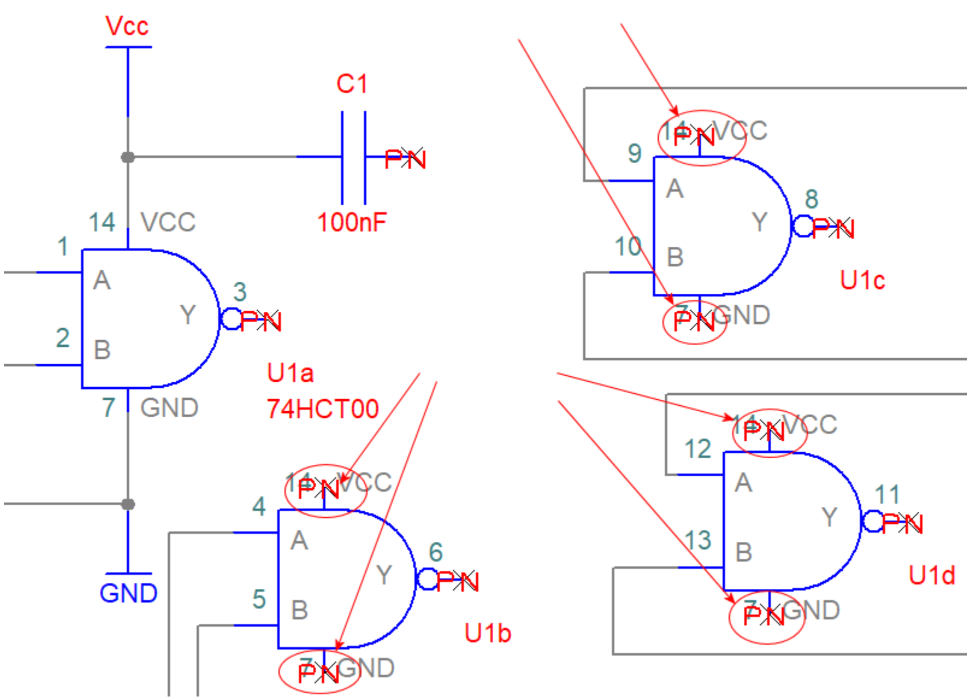
DRC in Schematics - Unconnected Gates

Available in subscription package: Engineer

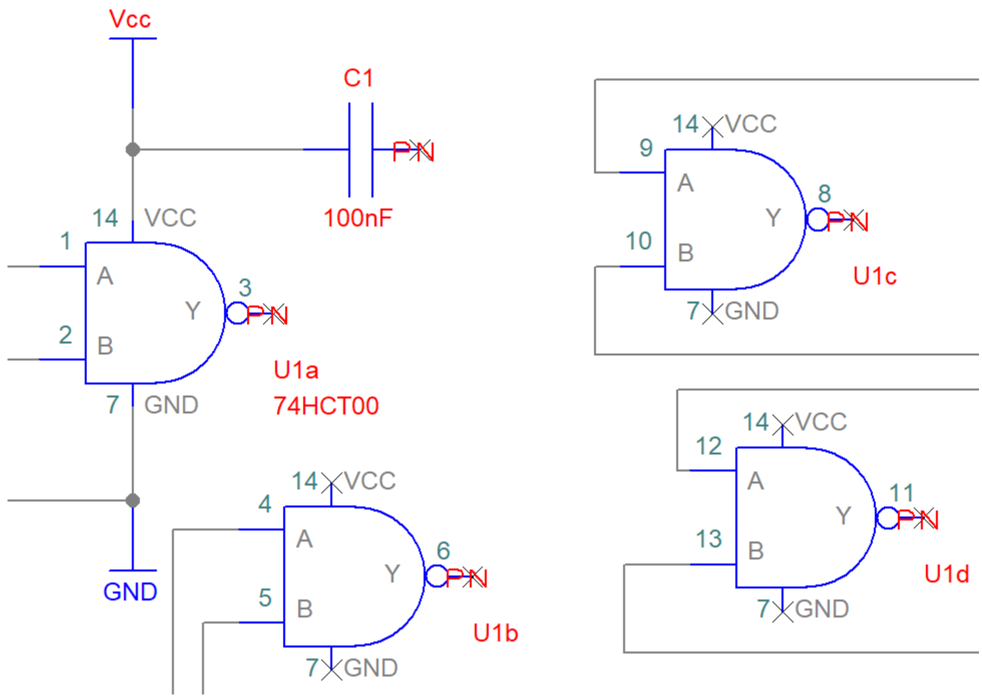
Within the **DRC** option in a **Schematic** design, reporting common power pins as **unconnected** on additional gates now no longer reports this as an issue.

Where gates are defined separately for heterogeneous devices, they may contain a power pin, especially if the same symbol has been used multiple times. Where the gate contains power pins for example, for the device to function, only one set of power pins must be connected on one of the gates. In this instance, this flagged an error in DRC. In previous versions, common pins 7 and 14 (for example) are connected on one gate but are flagged as "unconnected" on the other gates even though the device overall has the correct power pins connected.

In previous versions, it would be reported as an error like this on the additional three gates:



In this version, these errors are now longer reported when DRC is run:

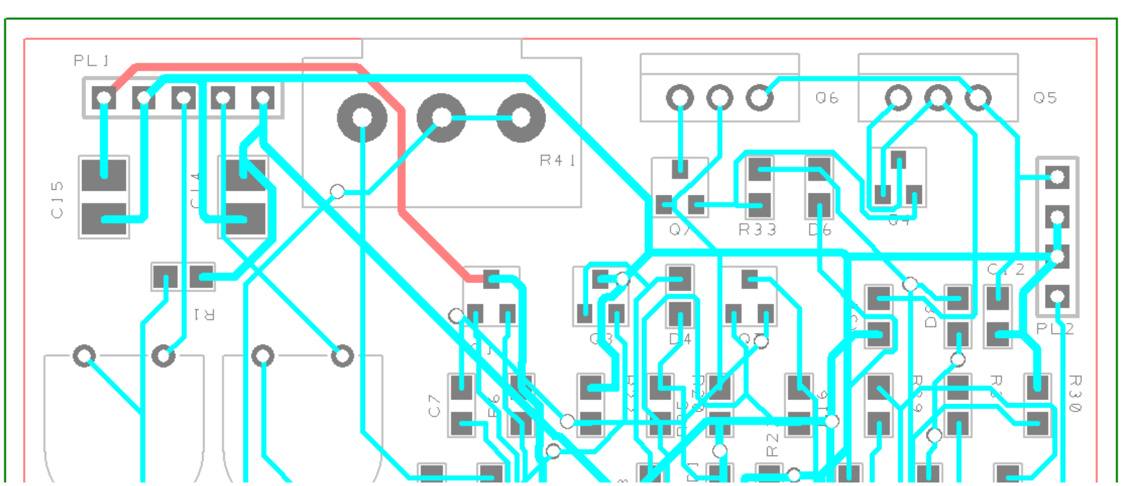


Invert Selection

Available in subscription package: Engineer

There is a new option on the context menu under **Select**, named **Invert Selection**. If you have items all of the same type selected, the selection will invert all items of that type. If you have different types selected, all unselected items will be selected.

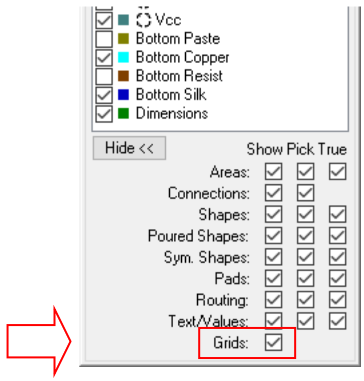
A practical example might include using **Shift-select** to pick one track path, **Invert Selection** to select all other tracks and then run the **Mitre Selected Tracks** or **Smooth Selected Tracks** options.



Grid Visibility Setting on Layers Bar

Available in subscription package: Engineer

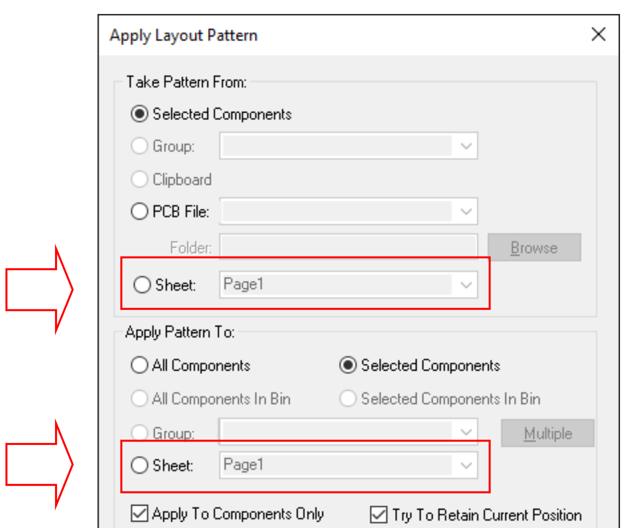
The **Layers Bar** has been updated to include the **Grids** check box on the bottom pane. Checking or unchecking this will change the display status of the Grids to either on or off.



Apply Layout Pattern use Schematic Sheet

Available in subscription package: Engineer

On the **Apply Layout Pattern** dialog, there is now an option to select components from a Schematic **Sheet**. This is available for the Source pattern and the Target pattern.

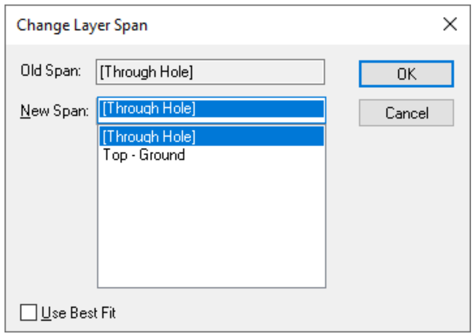


Select an existing Schematic design sheet from the **Project**. If the PCB was created without a project, only one sheet name will be available.

Updated Change Layer Span dialog

Available in subscription package: Engineer

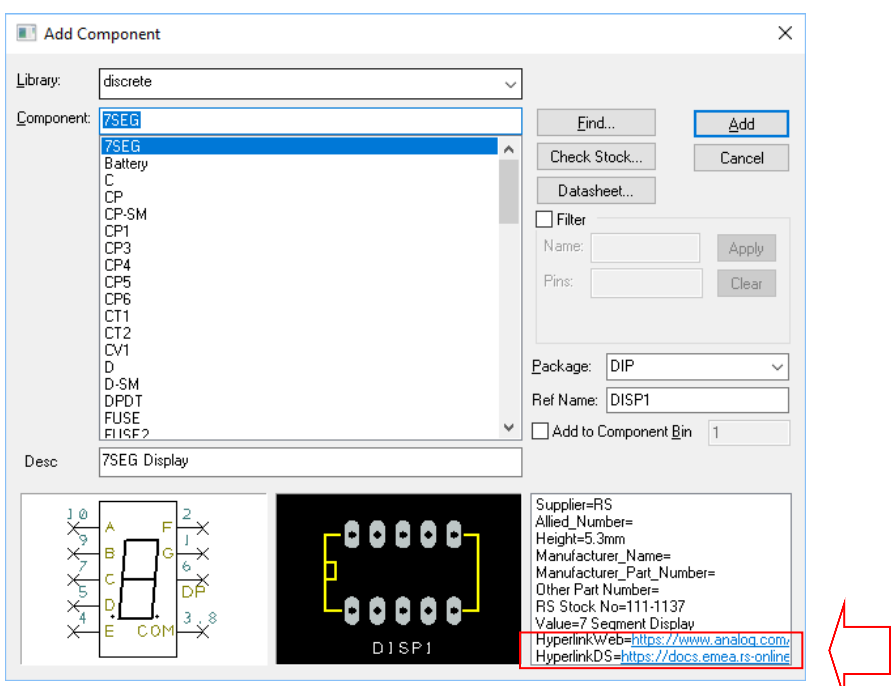
The **Change Layer Span** combo box is now a list and matches **Change Style,** it is a simple selection rather than a drop down, this will help save on mouse clicks.



Active Hyperlinks in Add Component and Add Component Bar

Available in subscription package: Engineer

In the **Add Component** dialog and **Add Component Bar**, Values that are hyperlinks will now be shown highlighted in blue. If you double click on the link, it will open it.



The **Add Component Bar** shows this in the Values pane: