Easy-PC

V29.0 Supplement

Number One Systems
20 Miller Court
Severn Drive
Tewkesbury
Glos GL20 8DN
United Kingdom

Phone: 01684 296 501
Email: sales@numberone.com

Technical: 01480 382 538
Email: support@numberone.com

Web site: www.numberone.com

Number One Systems, a trading division of WestDev Ltd.

Manual date: 30/07/25

Contents

Chapter 1. Getting Started

Installation

Back-up your files

If you already have Easy-PC 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, 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 for the installation of Easy-PC would have been provided to you by email. Click on the link to download the executable named EasyPC.exe. This is the whole installation set and should be **saved and backed up for future use**. Any subsequent patches can be installed on top of this 'base' setup once installed.

Using Windows Explorer, find the executable in your *Downloads* folder and double-click it. You'll need to type (or copy/paste) the **password** provided to unpack this file. Once the unpack password has been successful, you will be allowed to continue with the installation. You will also need to have your **customer ID number** that will be in the download link and your **16-digit installation** code to fully install the product. See below for more details.

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 Easy-PC software

You cannot install Easy-PC into any folder that already has any contents. You must install into a new empty folder.

A new installation of Easy-PC 29 will present you with the default folder during installation. We recommend that you do not change this. If you do, please select an empty or non-existent folder to use.

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.

Uninstalling Existing Easy-PC Software

Uninstalling will still remove shared registry entries, so it is recommended that a configuration file be saved first using the **Configuration Files** option from the **Help** menu and **Support** option. This will provide a restore point for any settings which may be lost.

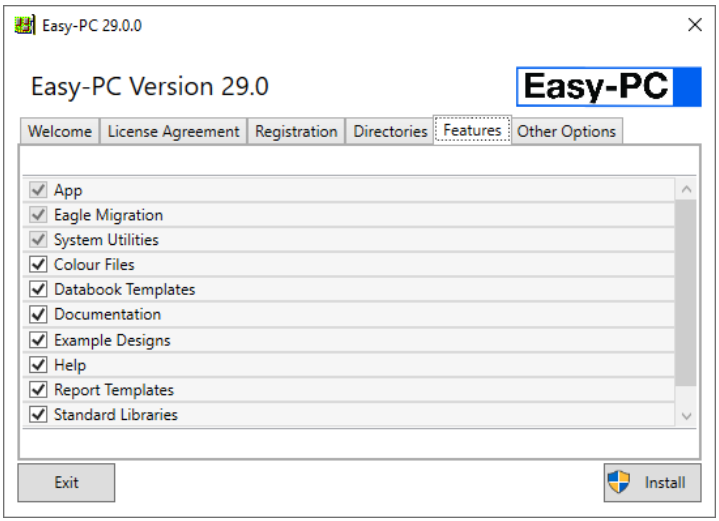
Data Files Location

The installer installs into your Documents folder if installing for current user. This would be a location such as C:\Users\You\Documents Easy-PC 29.0\ Under this folder you will see Library, Colour Files, Technology, Examples etc.

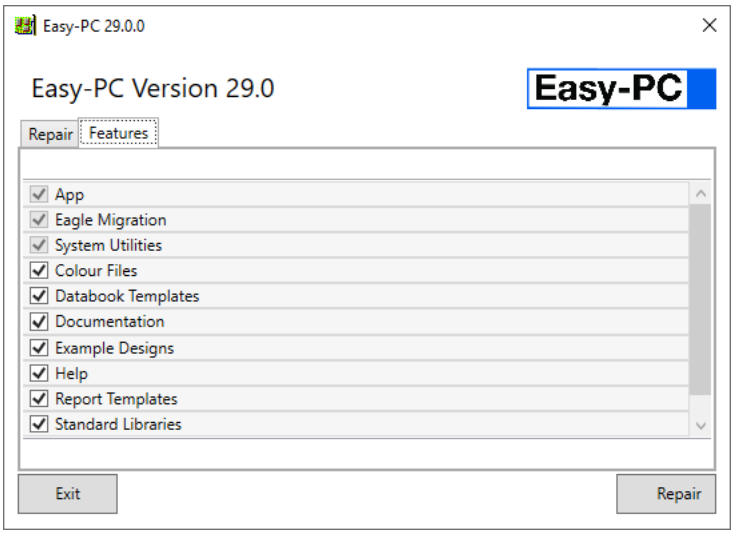
Customise Features During Installation

You can now customise the features that are installed with Easy-PC. Feature such as Libraries or Example files can be excluded from the installation.

There are three required features (that cannot be deselected), these are listed at the top of the list. The remainder of the features can be deselected (so they aren't installed).



This feature applies to first-time installation mode and **Repair** mode from the **Settings>** menu and **Apps**.



Running Easy-PC 29.0

Once installed, an Easy-PC shortcut icon will appear on your desktop. This is also available on the **Start** panel in the **Number One Systems** folder.

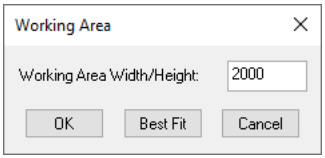
To start the program, double-click on the **Easy-PC** icon on your desktop.

Chapter 2. New Features in Easy-PC V29

Working Design Area Now 2m Square

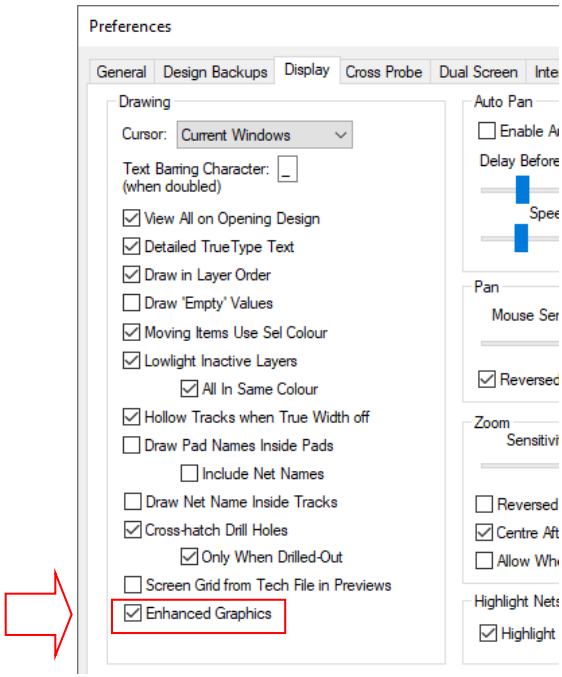
You can now create designs (Schematic and PCB) up to a maximum of 2 metres square (was originally 1 metre square).

The maximum size can be set in **Settings > Working Area…**



Enhanced Graphics with Antialiasing

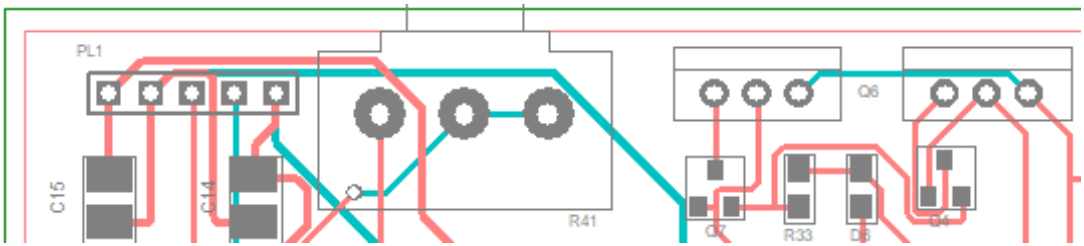
From the **Settings>** menu and **Preferences> Display**, there is a new switch named **Enhanced Graphics**.



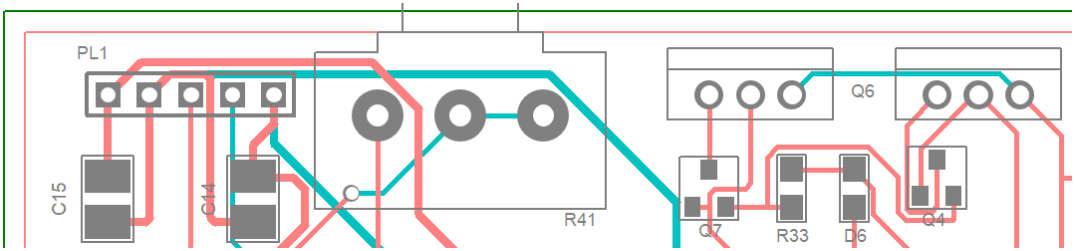
Check this option to take advantage of hardware accelerated, high quality graphic features such as anti-aliasing when they are available. The effect of this option may vary depending on the type of video adapter being used.

The effect of this option can be shown in the images below:

Without Enhanced Graphics enabled. Particular note the ‘jagged’ circles and ‘fuzzy’ text:

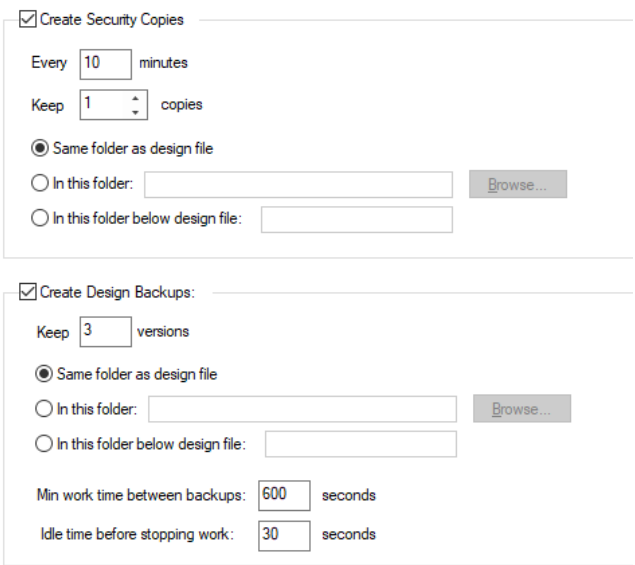


With Enhanced Graphics switched on displaying smooth circles and text:



Design Backups

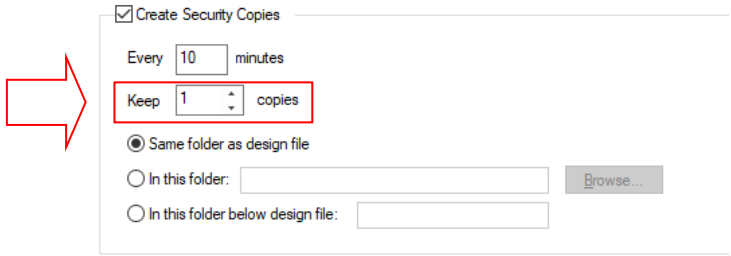
In the **Preferences>** dialog, there is a new **Design Backups** page. This allows you to specify backup and security copy settings.



Additional options have also been added to this page.

Changes to Security Copies Settings

You can now also specify how many security copies to create before overwriting the previous copies. This can be set in **Preferences> Design Backups** next to the **Keep** edit control in the **Security Copies** section.



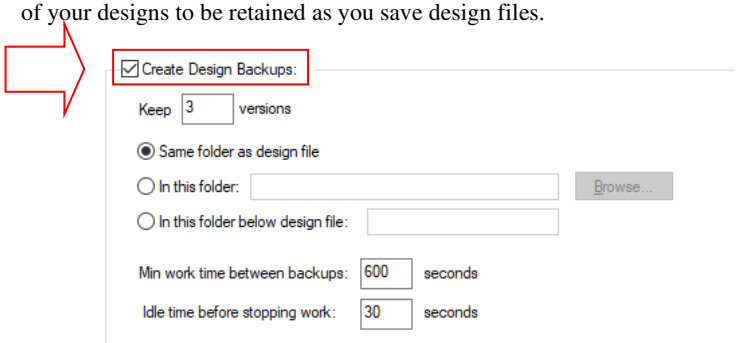
Each security copy is named ~filename, then incremented until the number specified in **Keep** has been exceeded, then it reverts back to the first name. For example, ~filename, then ~filename(1), ~filename(2), ~filename(3)…etc.

The **Location** button has been updated to now enable more refined file location information.

You to choose whether to write security copies to the **Same folder as the design file**, or a specified folder name using **In this folder** to enter one specific directory for all backups or In this folder **below design file**. You could create a folder called *Backups* for example, this may then be on a network path specified for daily backups.

Changes to Design Backup Settings

In **Preferences> Design Backups** check the **Create Design Backups** box to enable **backup copies** of your designs to be retained as you save design files.



Once enabled, you can enter the number of iterations (**Keep x Versions**) of your files to save.

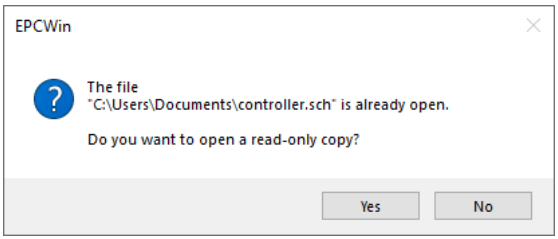
You to choose whether to write **Design Backup** to the **Same folder as the design file**, or a specified folder name using **In this folder** to enter one specific directory for all backups or In this folder **below design file**. You could create a folder called *Backups* for example, this may then be on a network path specified for daily backups.

Work Time - allows you to save Design Backups that are separated by a reasonable amount of work (regardless of how often you click Save) you can define settings for how much editing **work** (actions that record Undo steps) is done before a Design Backup will be created during Save (default is 600 seconds or 10 minutes), and how long it takes after design editing stops before the design is considered **idle** and thus the work time will not increment until the next editing action occurs (default

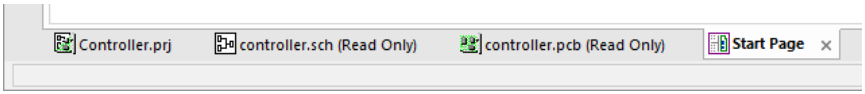
Idle time is 30 seconds). Setting **Work Time** to zero will disable the work time counter and Design Backups will always be saved every time you Save. Whenever one of the times expires, it resets the other.

Open Designs Read Only

If you attempt to open a design that has already been opened, you will now be presented with a warning message showing you that the design will be opened in 'Read only' mode:



The workbook mode tabs will display the Read Only status:



Package.txt and Def_Ref.txt Relocation

package.txt and def_ref.txt used to be stored in the hidden AppData folder. These have now been moved to the easier-to-access data folder location, by default, C:\Users\You\Documents\Easy-PC29.0 If you have specified a custom path for data files, that will be the path used instead.

Reminder of file uses

Package.txt is used when adding Packages in the **Component Editor** from the **Library Manager**. Any new names entered are added to the list saved in the file **Package.txt** in your data directory. This is the file that provides the drop-down list contents. Edit this file with a text editor if you wish to remove mistakes.

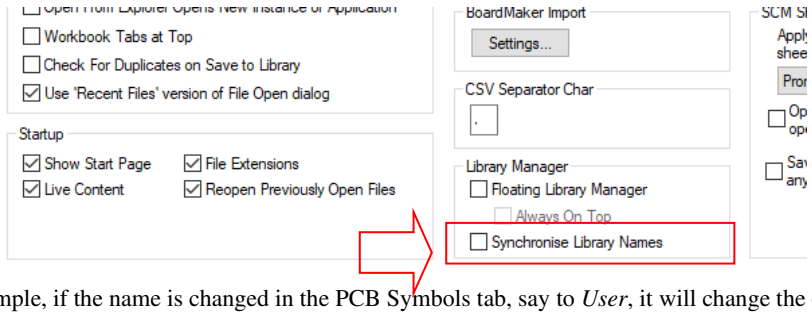
def_ref.txt is used to record name reference letters, such as U, IC, R, C etc., when adding **Properties** in the **Component Editor** from the **Library Manager**.

Any new references entered are added to the list saved in the file **def_ref.txt** in your data folder. This is the file that provides the drop-down list of contents. Edit this file with a text editor if you wish to remove or edit the letters.

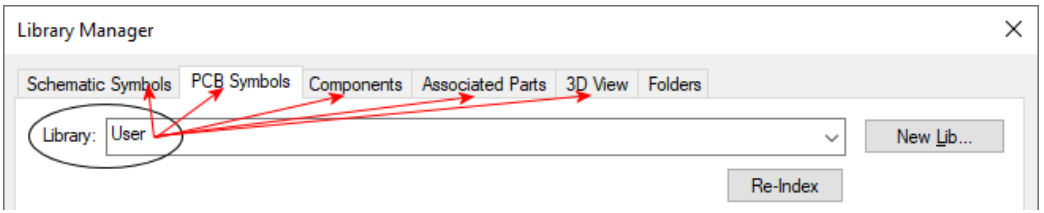
Synchronise Library Names

You now have the ability to toggle the synchronisation of library names. This means that with the switch on, changing a library name in one of the **Library Manager** tabs, such as **Components**, the library names will also change in the corresponding **Schematic Symbol** and **PCB Symbol** pages if the same named library exists.

In **Preferences> General**, there is a new option named **Synchronise Library Names**. When enabled, switching between library pages will attempt to pre-load the library with the same name.

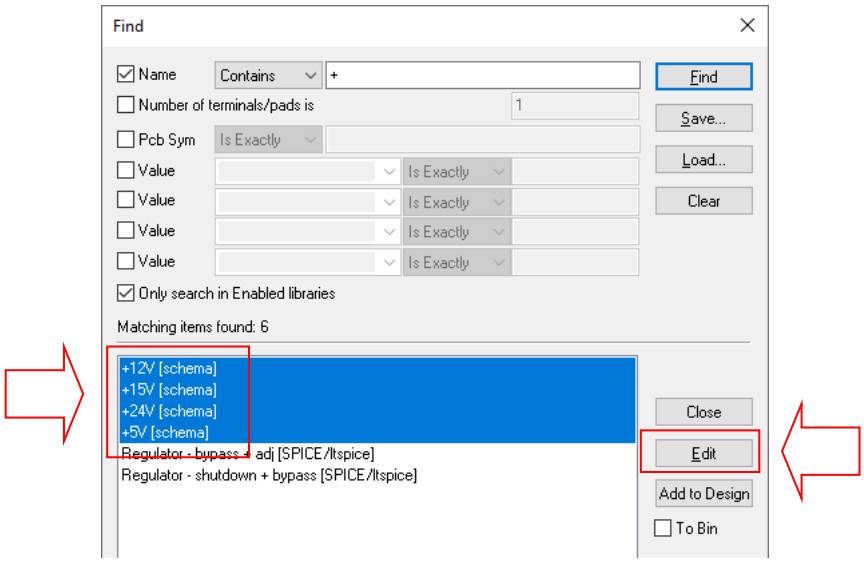


For example, if the name is changed in the PCB Symbols tab, say to *User*, it will change the other the library name in all other Library Manager tabs. This switch enables or disables this mode.



Select Multiple Found Library Items

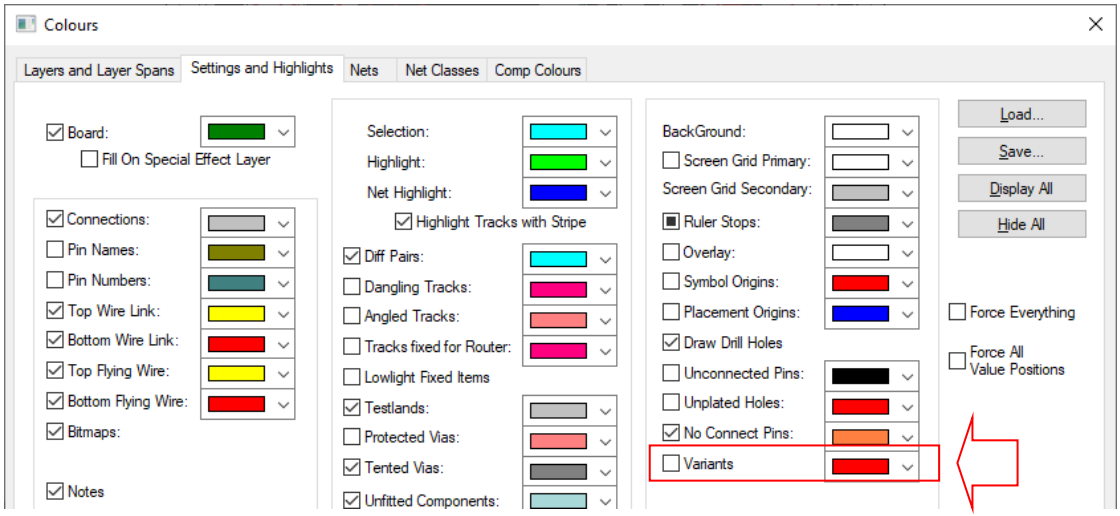
The **Library Find** dialog (available from **Library Manager> Find**) now allows you to select multiple found items. This enables you to edit more than one item in a single click.



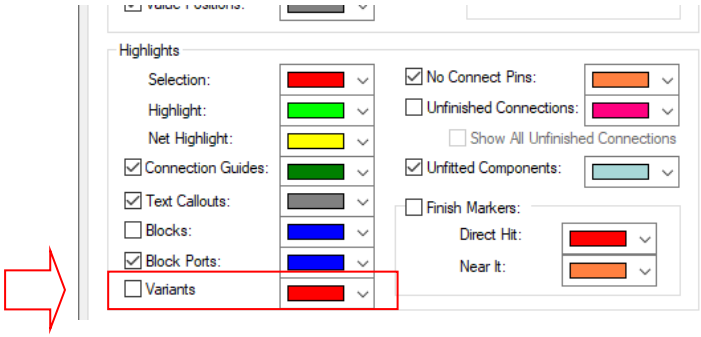
Highlight Variants

In the **Colours** dialog for PCB and Schematic designs, on the **Settings and Highlights** page, there is now a check box to show **Variants**. When enabled, any Component that is not fitted in any variant will be highlighted in the chosen colour. Only additional components will be represented. Components which are omitted in variants will not be marked.

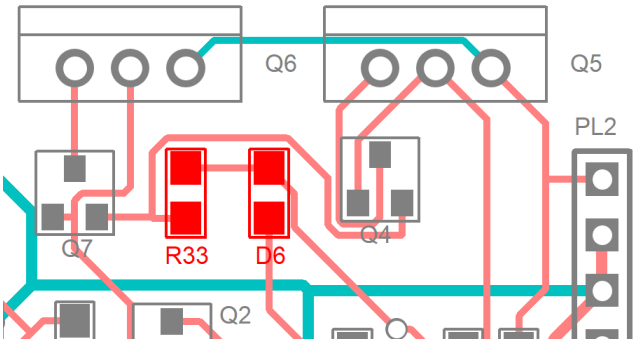
PCB Colours shows Variants here:



Schematic Colours shows Variants here:



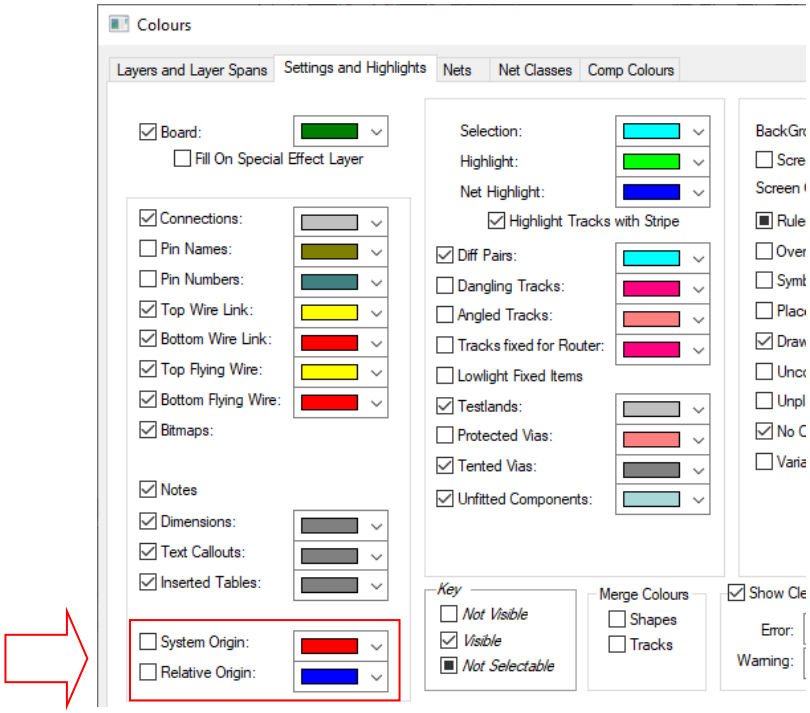
In the design, this means components defined as part of a variant will be displayed like this as demonstrated on R33 and D6:



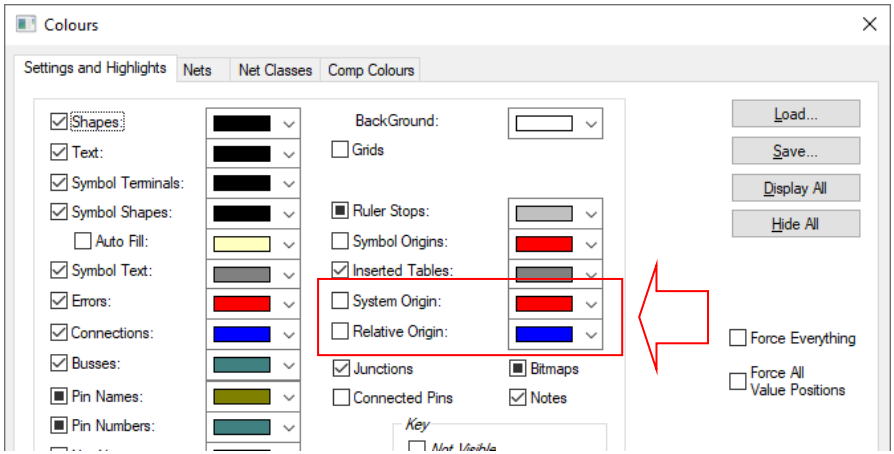
Set Colour of System and Relative Origins

In the **Colours** dialog for PCB and Schematic designs, on the **Settings and Highlights** page, there are two new check boxes to show **System Origin** and **Relative Origin**. When enabled, the colour specified will be used to colour the corresponding origin. When disabled, the system origin will be coloured in the **Primary Grid** colour and the relative origin will be coloured in the **Secondary Grid** colour.

PCB Colours dialog:



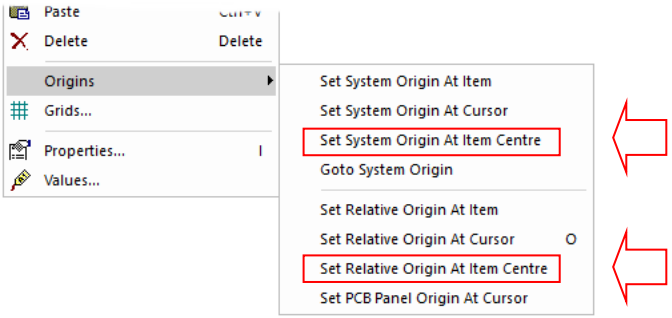
Schematic Colours dialog:



Set Origin At Item Centre

You can now set the absolute (**System**) or **Relative Origin** to the **centre of a selected item**.

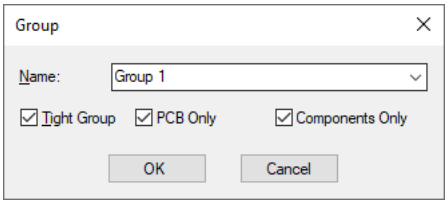
To do this, in **Select** mode, select any item (for example a Board outline) and from the context menu under Origin>, there are two new commands **Set System Origin At Item Centre** and **Set Relative Origin At Item Centre** which will set the corresponding origin at the selected item centre.



Group 'Only Components' Option

When creating a group, the **Group** dialog now gives you an option to select **Components Only**.

When selected, only the components in the current selection will be added to the group. If editing a group, the **Components Only** option will be selected if the current group only contains components.



Export PNG Image

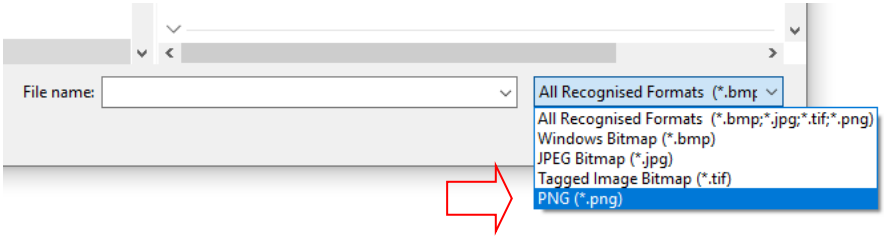
You can now export an image in .png format using the new command available under **File> Export> PNG Image...**.

When selected, a file save dialog will prompt you to choose a destination for the PNG file. Once a valid path has been selected, a PNG image will be created of your current design view. (This new feature works the same as the **Export Bitmap Image** option).

Add PNG Image

In addition to the existing supported formats (.bmp, .jpg, .tif), you can now add **png** images to a design.

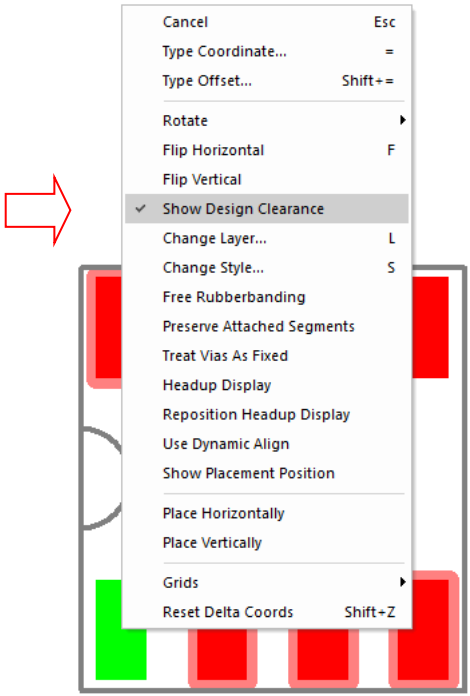
From the **Add>** menu, **Bitmap**, you now have the option to select a .png in the file browser.



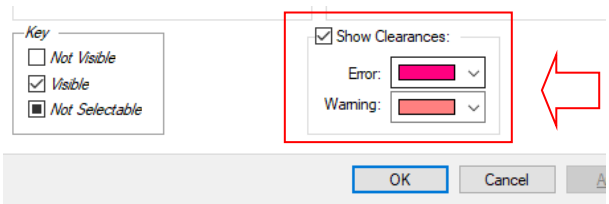
If a .png is selected, the image will be converted into a bitmap and added to the design.

PCB Symbol Design Clearances

In the **PCB Symbol editor**, you now have access to **Design Clearances**. To enable this, there is a new **Show Design Clearances** context menu option, available when moving an item. This is a design specific setting, not a global setting.



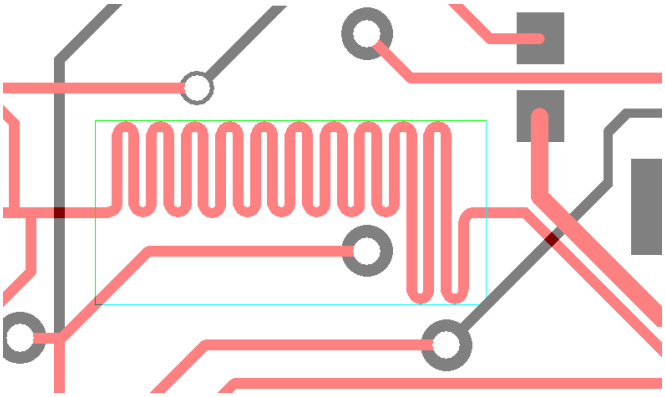
The clearance colours can be set on the **Settings and Highlights** page of the **Colours** dialog.



Serpentine Routing

Serpentine Routing is now available with two new modes under **Utilities> Serpentine Mode** and **Serpentine Selected Tracks**

Serpentines are used to increase the length of a track; this is required when signal timing is an issue. A serpentine route consists of a winding path of track within the routing.



You can also apply serpentine routing to one track of a Differential Pair by selecting either of the tracks. This will serpentine one side of the pair without disturbing the other track in the pair lengthening one side relative to the other.

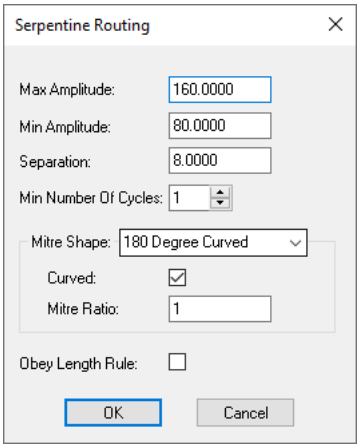
Serpentine Modes

Serpentine Mode is a dynamic interactive mode that creates the serpentine as you move the cursor with optional length-based rules on the values defined in the **Net Classes** dialog for **Min and Max Track Lengths**.

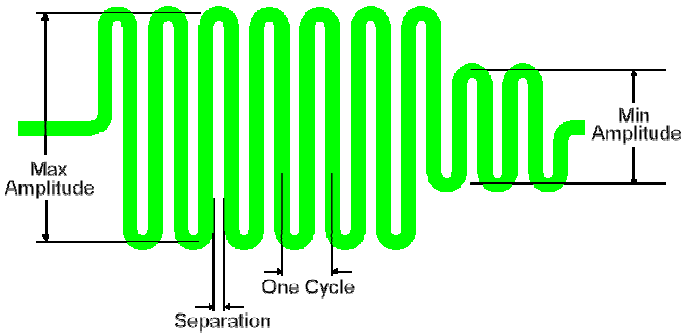
The second mode is a 'one shot' option on a selected track segment or multiple track segments to create the serpentine based on defined parameters.

Serpentine Parameters

The Serpentine Parameters dialog is available from the context menu during **Serpentine Mode**, and directly when you start **Serpentine Selected Tracks**. This dialog enables you to set various parameters and styles of serpentine routing.



The diagram below shows how these values apply to the serpentine:



Mitre Ratio and Amplitude

To fully define a Serpentine route, you need the **amplitude** and the **separation**.

The **amplitude** is the distance from the top to the bottom of the perpendicular segments, this includes the semi-circular sections, but does not include the width of the track.

The **separation** is the distance between successive perpendicular segments, excluding the track width.

Min and Max Amplitude

You can define two amplitudes, a maximum **Max Amplitude** which is the default, and a minimum **Min Amplitude** which is used if the maximum would cause **spacing** error. If the minimum would also cause an error, then no serpentine routing is added in that section.

Separation

The **separation** is the distance between successive perpendicular segments (the 'gap'), excluding the track width.

Min Number Of Cycles

You can also specify the **Min Number Of Cycles**, a cycle being a full up and down 'loop'. This minimum number of cycles defines the minimum size of a serpentine section and is also used as the minimum length of a straight section caused by an obstacle.

Mitre Shapes

There are four basic shapes for serpentine routing plus a user-defined shape which can be formed by customising one of the other four shapes. The four shapes available are - **180 Degree Curved**, **Octagonal**, **Sawtooth** and **Trombone**.

When adding serpentine routing, the ‘cycles’ will follow the min and max amplitude rules defined where it can, and where required. This may never exceed the max rule but will always not be less than the min rule.

180 Degree Curved - 'curved' corners.

Octagonal - same as 180 degrees curved but with Mitred corners.

Sawtooth - made up from 45-degree edges.

Trombone - this shape style extends the extra tracking length from one side of the selected track. Two styles are available for Trombone - **Octagonal** and **Curved**. These are selected using the **Curved** check box.

User Defined - shape can be formed by customising one of the other four shapes using the **Curve** and **Mitre Ratio** options on the Serpentine Parameters dialog.

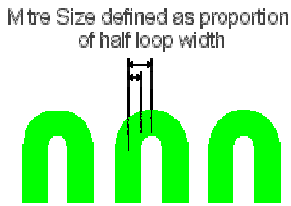
Curved

For each shape, you can also specify the **Curve** and **Mitre Ratio**, see below.

Mitre Ratio

You can control the shape of the top of each loop. The mitre around each turn can be curved or straight, and the size of the mitre is defined by the **Mitre Ratio**, which is the proportion of the 90-degree corner taken up by the mitre. A value of 1.0 gives a complete 180-degree curve (or *sawtooth* if straight) around the top of the loop; a value of 0.0 gives a squared off top to the loop; values in between give a loop of two curves or 45-degree angled lines with a flat top between. The default is **Curved** with a **Mitre Ratio** of 1.0, which results in 180-degree curve around the loop. A **Mitre Ratio** of 0.585786 and straight mitre gives an octagonal shape around the loop (angled mitres and flat top with the same length). The drop-down list, above the Curved and Mitre Ratio controls, allows you to set the parameters for the most common shapes.

The **Mitre Ratio** is not relevant if the amplitude is less than half the cycle length, because the size of the mitre is determined by the amplitude alone.



Obey Length Rules

When **Min and Max Track Length** values have been defined in the **Net Class** dialog, the serpentine lengthening follows these rules using the criteria defined below when the **Obey Length Rules** check box is selected:

If the Min length value is set, serpentine routing will ensure the lengthening is just over this value.

If the Max length value is set, serpentine routing will ensure the lengthening is just under the Max value.

If both length values are set, serpentine routing will ensure the lengthening is just under the Max value, but still conforms to the Min rule also.

If the **Maximum Track Length Difference** has been defined in the **Net Class**, this will also be taken into consideration during serpentine routing and the length will not be exceeded.

Adding Serpentine Routing to Tracks

Select a track or tracks and from the **Utilities** menu, **Serpentine>**, select the **Serpentine Select Tracks** option.

The **Serpentine Select Tracks** option is also available for a selected track on the context menu under **Adjust Track>**

The **Serpentine Routing** dialog will be displayed from where you can choose the parameters and serpentine routing style required. when you press **OK**, the selections will be applied to the track.

When **OK** is pressed, the styling and parameters will be applied to the track as a one-shot process.

If multiple tracks are selected, the serpentines will not be nested but will be implemented separately as space allows, as for differential pairs.

Adding Serpentine Routing Dynamically

From the **Utilities** menu, **Serpentine>**, select **Serpentine Mode**. You will be put in the dynamic Serpentine Mode, selecting a track will allow you to start routing. Dragging the cursor will be able to see the defining box which shows the limits of the serpentine. You can also start with a track pre-selected and it will automatically start when this mode is entered.

Whilst in this mode, the context menu gives you the option to change the **Serpentine Routing**. If you have **Online DRC** on, the loops will obey the design **Spacing** rules defined.

Press the Esc key to exit this mode.

Adding Serpentine Routing to Differential Pairs

Differential Pairs can have Serpentine routing applied to them. This can be for one (Single) track of the Diff Pair to provide Skew to the track. Both **Serpentine Mode** and **Serpentine Select Tracks** options can be used on the tracks of a Diff Pair.

When the serpentine is applied to a track, it will be applied 'outwards' away from the other paired track to ensure a DRC error does not occur.

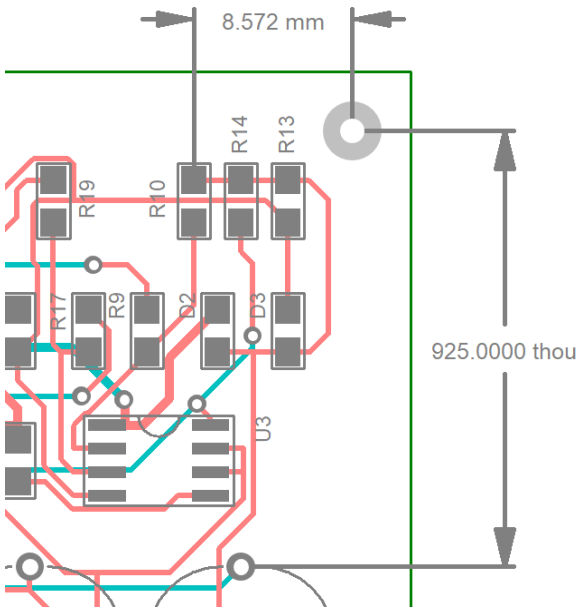
If the **Differential Pair Skew Length** has been defined in the **Net Class**, this will also be taken into consideration during serpentine routing.

Removing Serpentine Routing

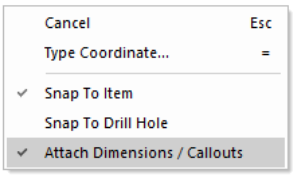
With the whole serpentine selected using **Frame Select** or by Ctrl-clicking each segment, right click and from the **context** menu, select **Delete Corner.** This will revert the serpentine to a straight track.

Attached Dimensions

Dimensions can now be attached to items so that when the item moves, the dimension moves with it and is dynamically updated.



The **Attach Dimensions** feature is enabled from the context menu when the **Add Dimension** option is first selected.



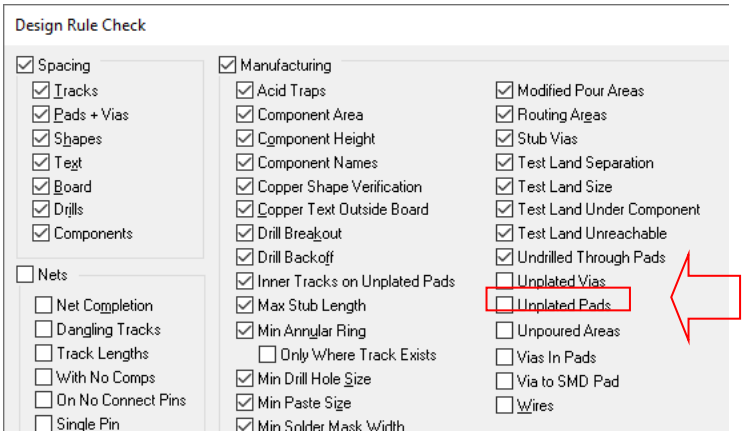
Attached Callouts

Callouts can now be attached to items so that when the item moves, the callout moves with it.

The **Attach Callouts** feature is enabled from the context menu when the **Add Callout** option is first selected.

Separate DRC Check for Unplated Pads

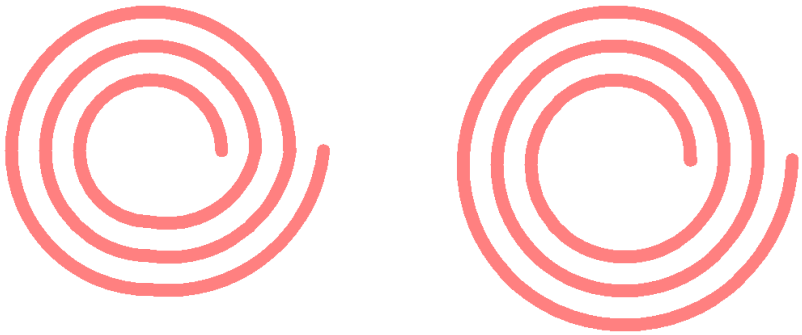
A new check has been added to the **DRC** dialog for **Unplated Pads**. When selected, this lists any pad without a plating status used in the design. These may of course be deliberate but it allows you to identify them.



Changes to Existing Spiral Shape Algorithm

The existing function for generating **Spirals** has been enhanced with a new algorithm for calculating the spiral. This means the spiral shape is far more accurate. This change affects the generation of spiral tracks and shapes.

The older algorithm generated spirals using approximated 'half' circles that offset each other. The use of the new Archimedean algorithm generates a more accurate and smoother spiral.



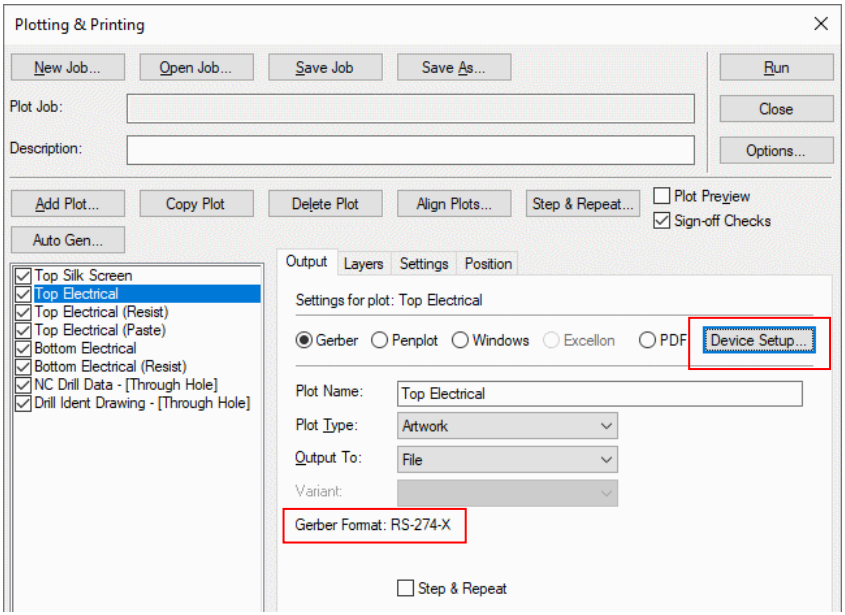
Old Spiral algorithm

New Spiral using Archimedean algorithm

If you wish to use this new algorithm, then you will need to remove then recreate the existing spiral shape as the shape is not regenerated automatically using this new algorithm.

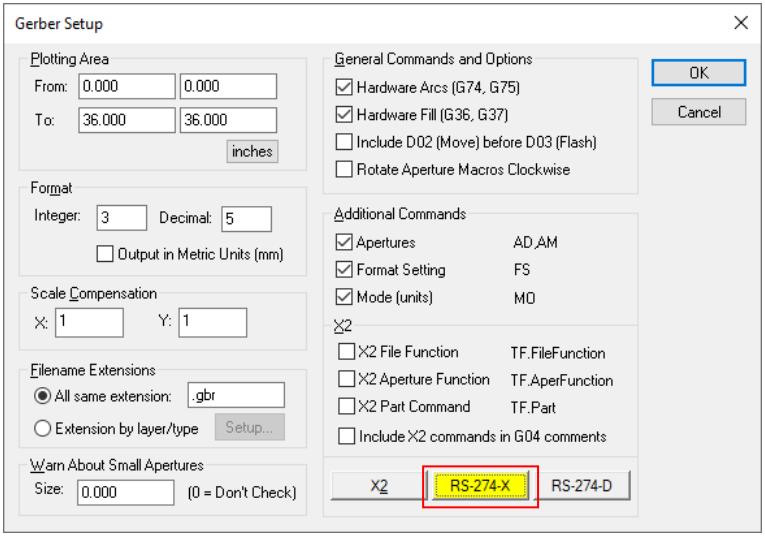
Report Gerber Format Used to Produce Gerber File

From the main **Plotting & Printing>** dialog, from the **Device Setup** button for **Gerber** plots, the dialog now highlights the button of the Gerber Format used.



For example, as shown below, if X2 is used, the X2 button will be highlighted.

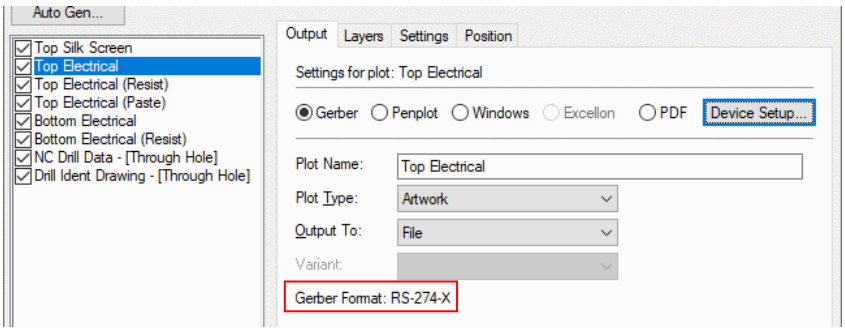
If you then disable one of the check boxes relevant to that particular format, the button will be unhighlighted, and the format is now considered to be a **Custom** Gerber format.



Plotting & Printing dialog highlights Gerber Format

Once a Gerber format has been selected, the main **Plotting & Printing** dialog now highlights the Gerber Format used. In the image below, X2 is shown.

As with the Device Setup page, if you disable one of the Gerber format check boxes relevant to a particular format, the dialog will display a **Custom Gerber format**.



Plotting Report shows Gerber Format used

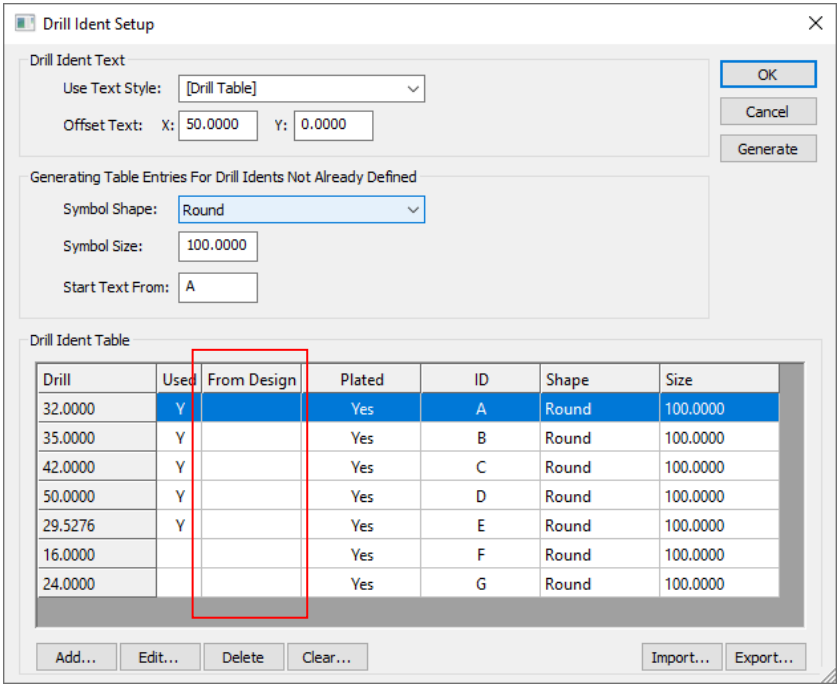
When the plot is generated, the plot report will now show you the Gerber Format used. As with the main dialog, if you disable one of the Gerber format check boxes relevant to a particular format, the dialog will display a **Custom Gerber format**.

```
Gerber Settings
===============

Gerber Format: RS-274-x
Leading zero suppression.
G01 assumed throughout.
Line termination <*> <CR> <LF>.
3.5 format absolute inches.

Format commands defined in Gerber file.
Aperture table defined in Gerber file.
Hardware arcs allowed.
```

Ability to Save Drill Ident Settings with the Design

Drill idents for any drill sizes in a design are now automatically included when a design is saved. When a design is opened, any drill idents not already present are automatically added to the drill ident table.

You can edit drill idents from the **Settings >** menu, **Drill Ident…,** this will open the **Drill Ident Setup** dialog. It can also be accessed from within the **Plotting & Printing** option, **Options** button and **Setup Sizes and Symbols**.



The Drill Ident Table now has an extra column **From Design** which displays whether the drill ident is saved in the design or not.

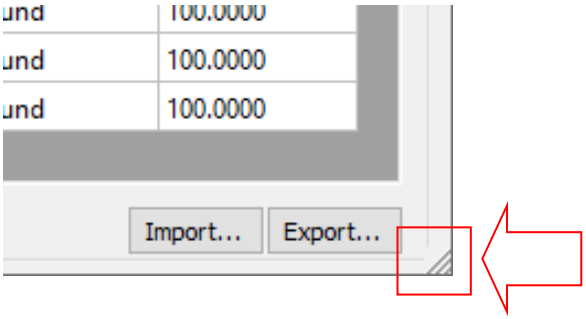
You can use the **Import** button to load in drill idents from a Drill Ident File (.did). When doing this, you will be asked to overwrite; Clicking **Yes** will remove the current drill idents and replace them with those in the file. Clicking **No** will only add drill idents that have not already been defined.

Note that adding a drill table to a design will also cause the drill sizes to be added to the drill ident table.

The **Export** button allows you to save the current drill idents to a Drill Ident File (.did).

Drill Ident Setup Dialog Resizable

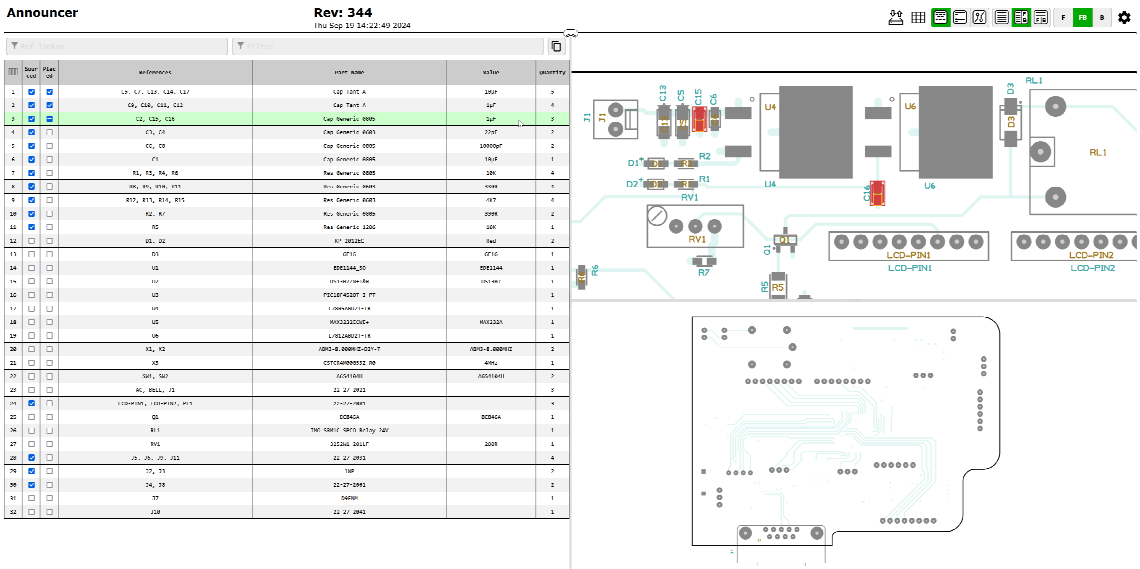
The **Drill Ident Setup** dialog can now also be resized using the small 'gripper' at the bottom right of the dialog.



Interactive HTML BOM Export

Support has been added for the **Interactive HTML BOM** option. This tool is a plugin option that exports limited Easy-PC design information into a HTML format that produces an interactive design and BOM listing that can be viewed and is searchable in a web browser. Only the Top and Bottom layers are exported along with BOM (Bill Of Materials) data. No inner layers or value details are exported.

This tool is particularly useful if you are producing boards in-house and hand soldering prototypes. It will guide to finding Parts with the same value and their locations. Items selected in the BOM list are highlighted in the design view.



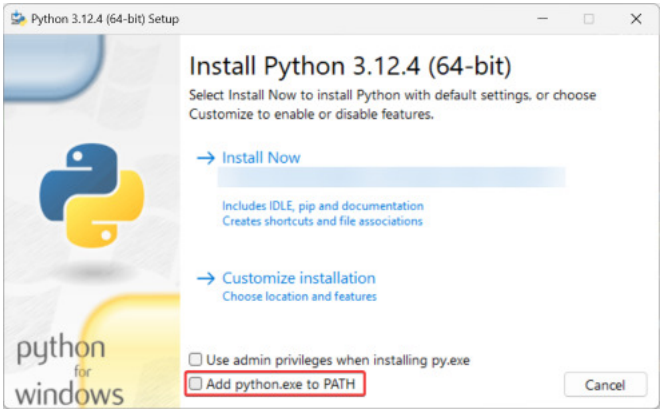
Easy-PC automatically generates an input file that is read into the browser ready for use. You must first have Python installed and configured for this to be usable

Python 3.8

The generation of the Interactive BOM requires **Python 3.8** (or later) to be installed.

You can download the Python installer here https://www.python.org/downloads/

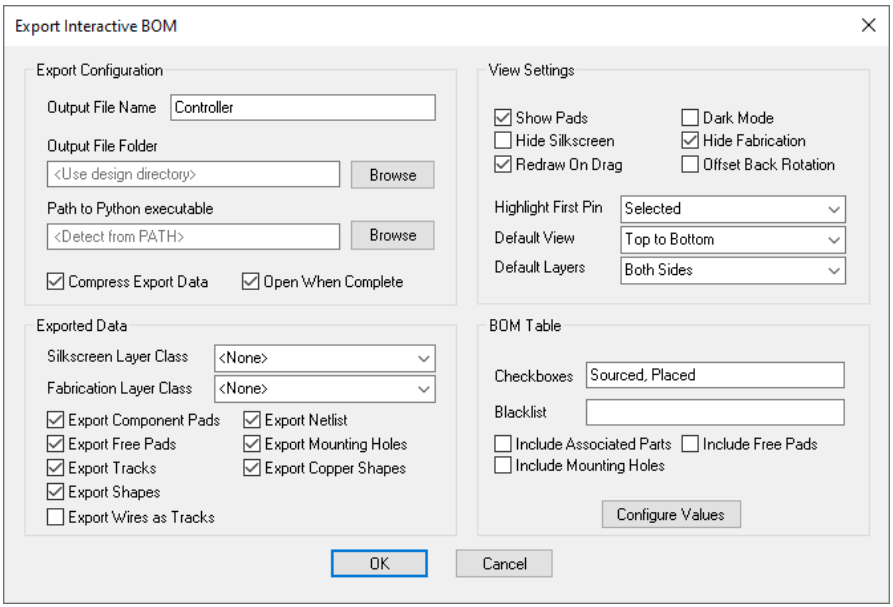
If you installing Python 3 for the first time, it is recommended that you select the **Add python.exe to PATH** option in the installer, this will allow the application to automatically locate your installation.



Once successfully installed, the option is ready to run.

Using the Interactive BOM option

From the **Output** menu, select the **Interactive IBOM** option. You are presented with a dialog.



Once configured, pressing **OK** on the dialog creates a HTML file that is then automatically sent to your web browser for viewing. You must have the **Open When Complete** switch selected on the dialog for this to happen. The HTML file saved is self-contained and can be used at a later time or given to another person on another machine (provided they have Python 3.8 or later installed).

Most of the check boxes are self-explanatory, with more detailed descriptions available in the online help under the **Index** entry **Interactive HTML BOM Export**. It is worth highlighting a few that should be brought to your attention:

Output File Name

If this is left blank, the design name will be used.

Blacklist

A **blacklist** of component names and component name prefix can be provided by typing into the entry box. Any components that match this blacklist will not be displayed on the BOM Table.

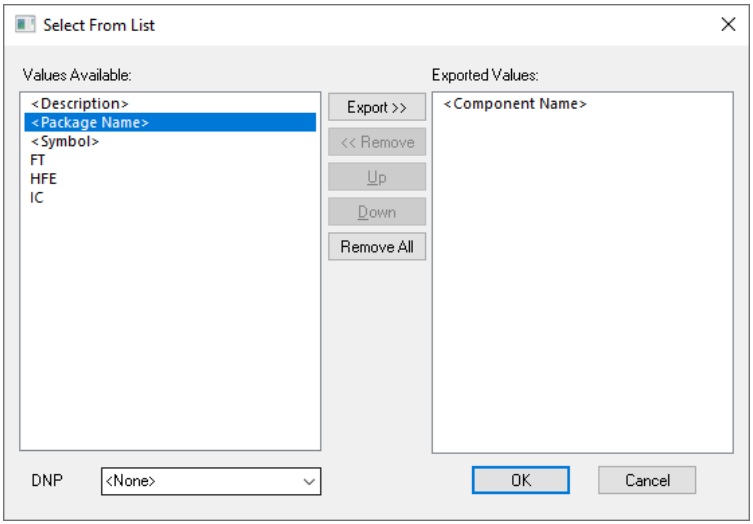
Name and prefix should be comma separated

Spaces after commas should be omitted unless they are part of the component name (e.g. use R1,R2 instead of R1, R2)

Prefix are marked with an asterisk symbol (e.g. R*,C*,LED-*)

Configure Values

Selecting the **Configure Values** dialog enables you to select the values required:



Pad Styles Exceptions Report Renamed

The standard report, **Pad Styles Exceptions** report under **Output> Reports**, has been renamed to **Pad Style Override** report to reflect its true purpose.

As a reminder, this report is used to report all Component pad styles in the PCB design that have been assigned a **Pad Style Override** (using Pad Properties) that changes the style from that of the original Footprint defined in the library.