



V30.0 Supplement

Copyright © 1998-2026 WestDev Ltd. All rights Reserved. E & O E

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

The product is licensed by WestDev Ltd to the original purchaser of the product for use only on the terms set forth in the WestDev End User Licence Agreement. Copying, duplicating, selling or using the product contrary to these terms is strictly forbidden.

This Manual Copyright to WestDev Ltd. All Rights reserved. No part of it may be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine-readable form without WestDev Ltd. prior written permission.

Information in this manual is subject to change without notice and does not represent a commitment on the part of the vendor.

Number One Systems, Easy-PC, Easy-Spice, Easy-Router, ProRouter, Pulsar, Layan, Analyser, MultiRouter, Filtech, Z-Match, StockIt and all variants thereof are Trademarks of WestDev Ltd.

All other trademarks acknowledged to their rightful owners.

Number One Systems, a trading division of WestDev Ltd.

While every care has been taken in the preparation of this manual WestDev Ltd cannot be held responsible for any errors or omissions within it. If informed we will add comments and features which you may like to see written which may help others using this manual. Please send your comments through the technical support desk.

Manual date: 12/08/26

Contents

CONTENTS	3
CHAPTER 1. GETTING STARTED	4
Installation.....	4
Back-up your files.....	4
Installation from a download link	4
Installing over existing Easy-PC software	4
Uninstalling Existing Easy-PC Software.....	4
Data Files Location	5
Customise Features During Installation	5
Running Easy-PC 30.0.....	6
CHAPTER 2. NEW FEATURES IN EASY-PC V30.....	7
Browse button for Report Folder in General Preferences.....	7
Comma as an Alternative Decimal Separator.....	7
‘Toggle Component Reference Name’ command.....	8
Goto – ‘Select All Find Items’ command.....	8
Execute Value Hyperlinks in Library Manager.....	8
Hyperlinks in Associated Part Values	9
Plotting and Printing	9
Unplotted Electrical Layers Warning.....	9
Plot Options - Default Job Folder set to a writable folder.....	10
Warning if Gerber plot is not 1:1	10
Print settings can now be optionally saved to registry	11
Customise Error reporting in the Copper Pour Report	11
Spoke Errors Reported for Pads Placed Partially Within Keepout Area	12
ERC - Common Pin Report now shows Pin Name and Nets	12
Support for QR Codes.....	13
Errors Bar Added	14
Colours –Star Point Visualisation	16
Last Visible Layers context menu option in Layers Bar	16
BOM Composer Enhancements	17
Open Preferences Button.....	17
Save Report Button	17
Report File Location	18
Open Folder	18
Explicit output to Plain Text	18
User Defined Report Title	19
File Already Open Warning	19
New Flashing Items settings	19
Improved Application Error Handling	20

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 30 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.

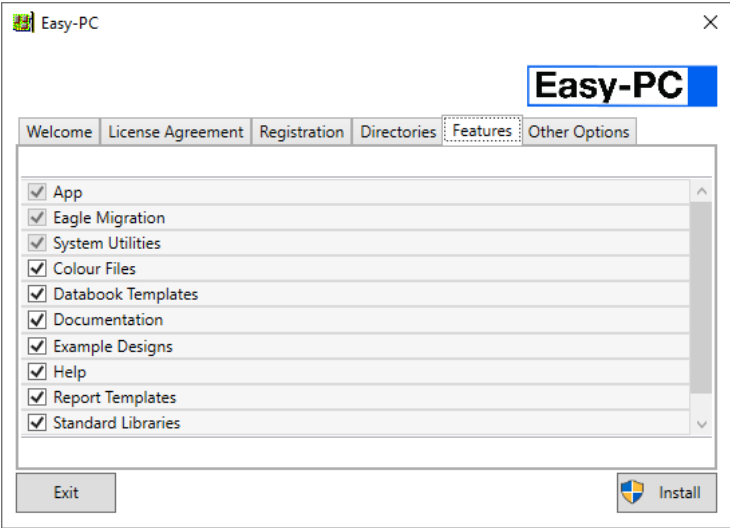
Data Files Location

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 30.0\ Under this folder you will see Library, Colour Files, Technology, Examples etc.

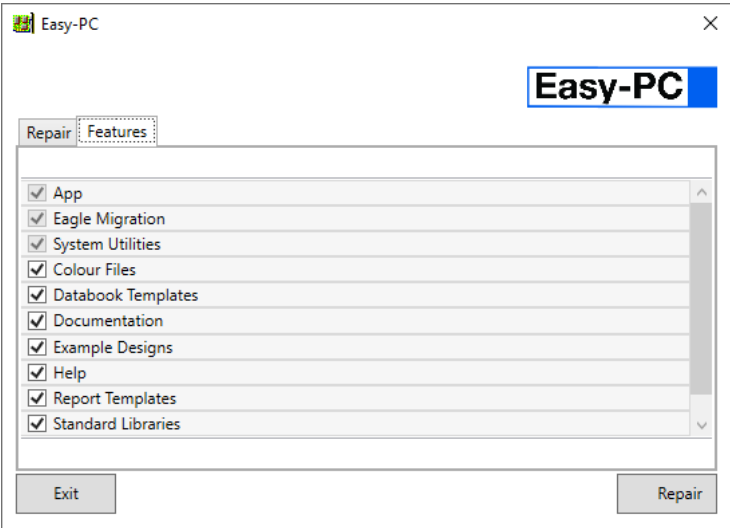
Customise Features During Installation

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).



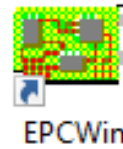
This feature applies to first-time installation mode and **Repair** mode from the **Settings>** menu and **Apps**.



Running Easy-PC 30.0

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.

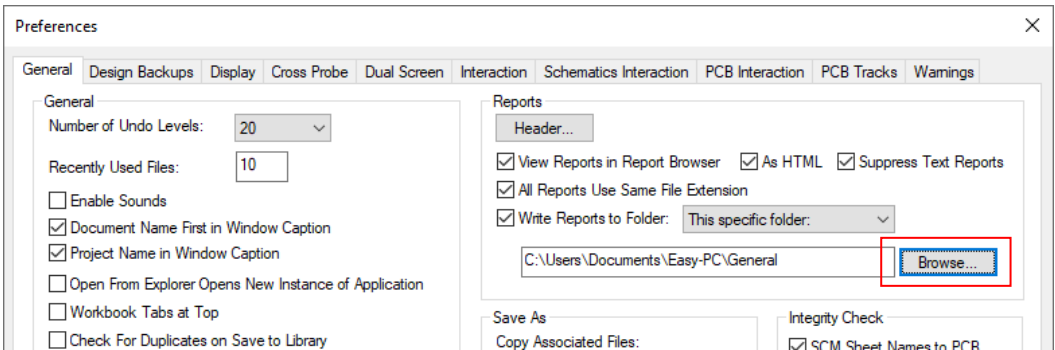


Chapter 2. New Features in Easy-PC V30

Browse button for Report Folder in General Preferences

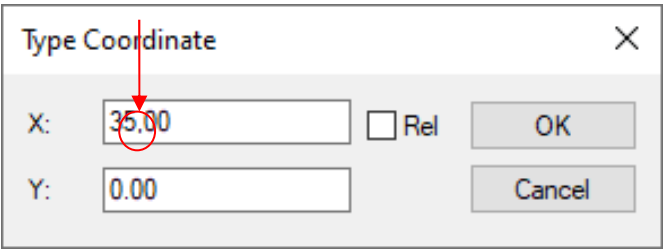
In **Preferences** dialog and **General** page, you now have a new **Browse** button under the **Reports** section. This button becomes enabled when the **Write Reports to Folder** check box is set to **This Folder**.

Clicking this button will open a dialog from where you can select the report output folder.

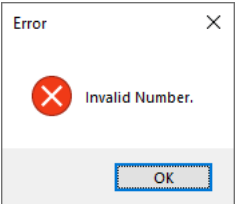


Comma as an Alternative Decimal Separator

Added support for commas as decimal separators wherever coordinates are entered. It now automatically recognises points and commas equally. The **Type Coordinates** option is a good example of where the coordinate can be entered as say 35,00 (using the comma as the separator).



Previously to Version 30, if a comma was used, the system would warn you of an invalid number:



‘Toggle Component Reference Name’ command

A new application command **Toggle Component Reference Name** is now available for PCB and Schematic designs. This command can be used to toggle the visibility of reference names for all selected Components. In Schematic designs, if a gate is selected, then all of the gates in that component instance will be affected.

This command can be assigned to a menu, toolbar or a shortcut key using the **Settings> Customise** option.

Goto – ‘Select All Find Items’ command

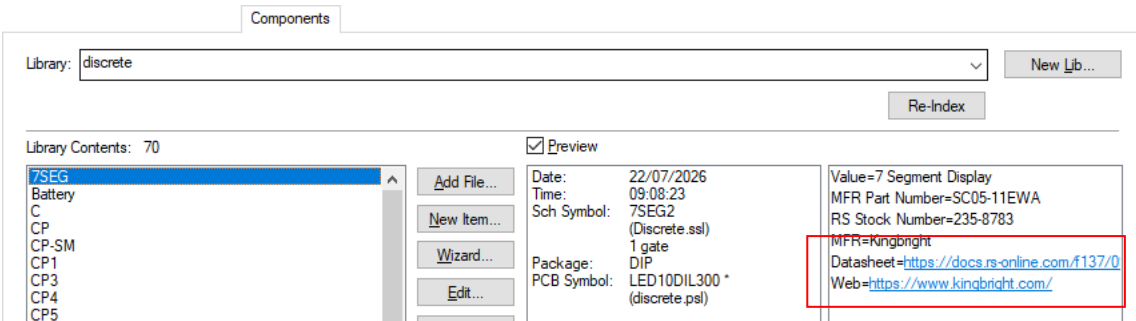
A new application command **Select All Find Items** is now available from the **Goto bar** context menu.

It can be added to a menu, toolbar or assigned to a keyboard shortcut using the customise dialog from the **Settings** menu.

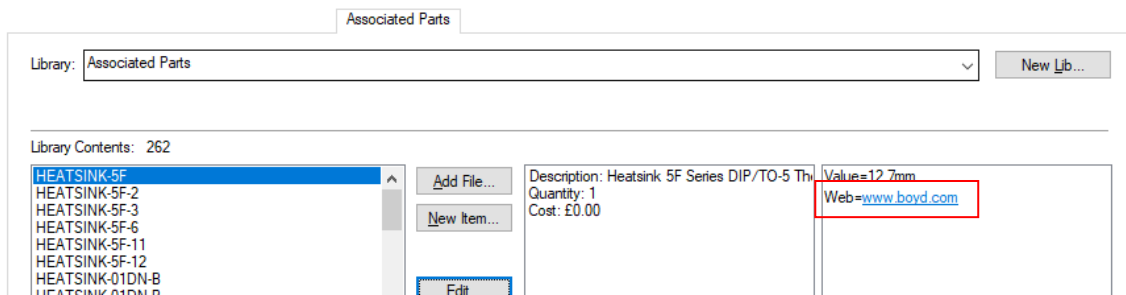
Using this command selects in the design all items shown in the Goto bar.

Execute Value Hyperlinks in Library Manager

In the **Component** page of the **Library Manager**, the list of **values** on the component is displayed to the right of the component details. If a value that has been set as a **hyperlink** is added to the list, double clicking on that item will now execute the hyperlink.



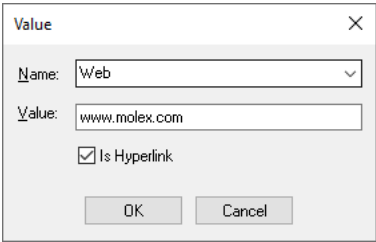
This new feature is also available on the **Associated Parts** page of the **Library Manager**:



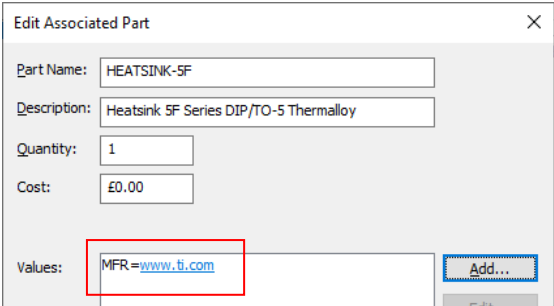
Hyperlinks in Associated Part Values

Associated Part values can now be marked as a **Hyperlink**.

This is available via the **Edit Value** dialog (accessed from the **Edit Associated Part** dialog. Check the **Is Hyperlink** box to activate this.



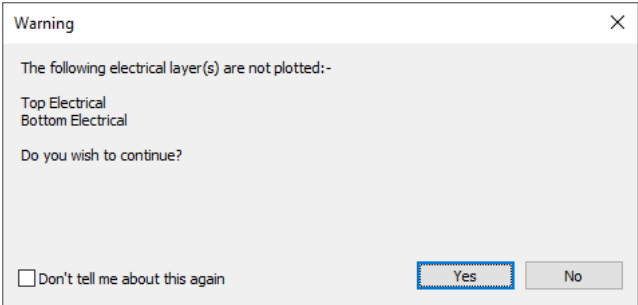
When values in the Associated Part are listed, double clicking the item marked as a Hyperlink in the list will attempt to execute that Hyperlink.



Plotting and Printing

Unplotted Electrical Layers Warning

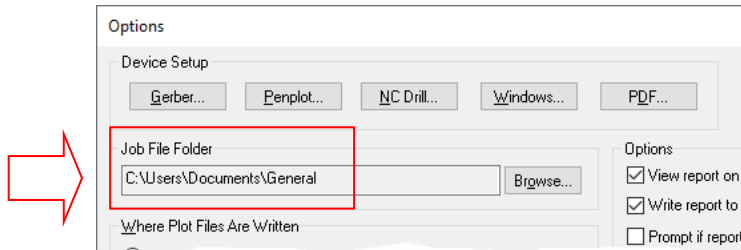
When the **Plotting and Printing** option is used, if there are any electrical layers not covered by the current set of plots, a warning message will now be displayed:



You can select **Yes**, to enter the plotting dialog and continue (and make any possible plot inclusion corrections), or press **No** to exit the option.

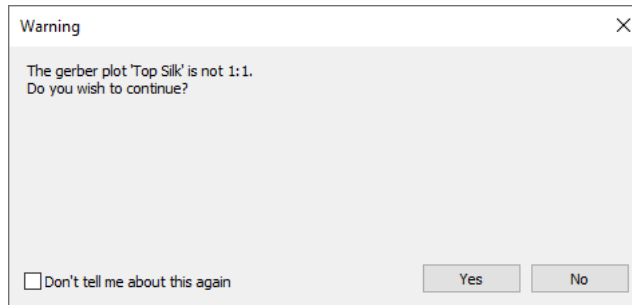
Plot Options - Default Job Folder set to a writable folder

The default Job folder in Options is now set to a writable folder, such as a folder under 'user documents'. Previously, it was set to the *Program files (x86)* location.



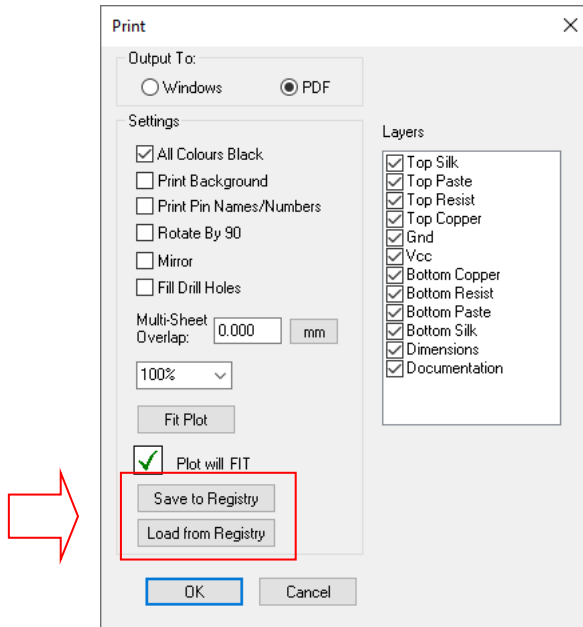
Warning if Gerber plot is not 1:1

In the **Plotting and Printing** dialog, if you start a Gerber plot, there is a new check to ensure that the Gerber plot has not been scaled (other than 1:1). If it has, a new warning will appear informing you that the plot is not 1:1. You can then select **Yes** to continue plotting or **No** to cancel. As this is a warning, you can switch it off at any time.



Print settings can now be optionally saved to registry

On the **Print** dialog, available from the **File** menu, there are two new buttons: **Save to Registry** and **Load from Registry**.



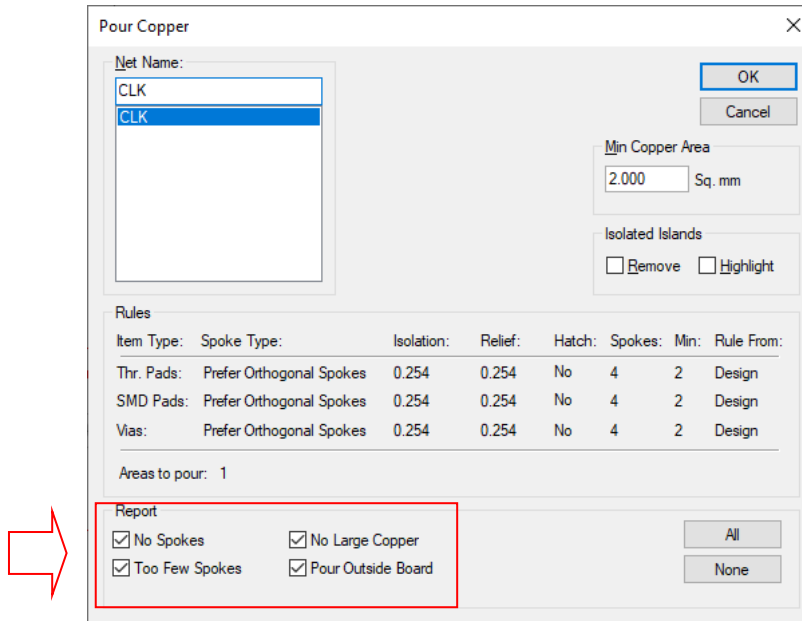
These buttons allow you to save and load print settings utilising your computer's registry rather than the current design. This enables you to load the same print settings across multiple designs.

Note that the **Layers** are not saved to the registry. Also note that the selected device is saved separately for Schematic and PCB contexts.

Customise Error reporting in the Copper Pour Report

You can now customise the errors that are displayed when running a **Copper Pour Report**.

From the **Pour Copper** dialog, there is a new **Report** section with a list of possible errors that can be reported when pouring copper. You can toggle whether you want such errors to be included in the report by selecting the relevant error type check box.



The errors that may be reported are:

No Spokes - The design technology requires spokes to be used and 0 spokes could be added for this pad.

Too Few Spokes - The design technology requires spokes to be used and the number of spokes that could be added for this pad is below the minimum.

No Large Copper - A piece of copper was not large enough for the pour to be retained.

Pour Outside Board - There is a copper pour area outside of the board extents.

Alternatively, you can use the **All** or **None** buttons to toggle all reported errors on or off.

Spoke Errors Reported for Pads Placed Partially Within Keepout Area

Spoke errors are now reported for pads that are partially within a copper keep out area that don't match the spoke requirements set on the design. Pads that are completely within the copper keep out area will continue to not report spoke errors.

The report will typically look like this:

U1.8 at 2500.00,2025.00 on layer Top Electrical has too few spokes (can fit 2, requires 4)

ERC - Common Pin Report now shows Pin Name and Nets

In a **Schematic** design, the **Electrical Rules Check** report for **Common Pins On Different Nets** has been improved to report more information. The report entry now includes the name of the pin and location with which the error is associated with and the name of the different nets.

The report will typically look like this:

Error : Common pin on different nets - Pad U1.14 is on nets 'CLK' and 'RST' at (19300, 23625)

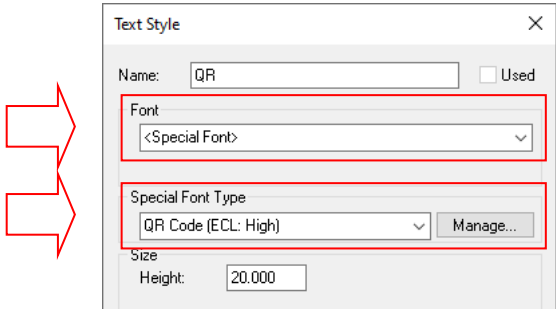
Support for QR Codes

You can now add **QR Codes** to your designs, this is in addition to the existing Barcodes.

To do this, firstly define a **QR Code Text Style** font in the **Design Technology** dialog.

Create a **Text Style** by selecting the **Font** type **<Special Font>** from the drop down list.

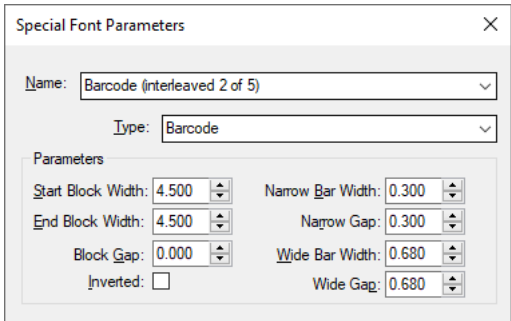
From the **Special Font Type**, you can select the font type required from the list. Predefined types to use include the usual **Barcode Interlaced 2 of 5** and various **QR codes** ranging in error handling ability.



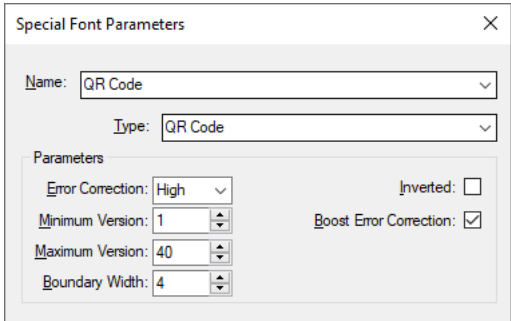
You can customise the font settings using the **Manage** button.

This will open the **Special Font Parameters** dialog from where you can enter your own values for a custom **Barcode** or **QR code**.

Special font Parameters for a Barcode:



Special font Parameters for a QR Code:



Adding a Barcode or QR Code to your design:

To add a Barcode or QR code to the design, from the **Add** menu, select **Text**. In the **Add Text** dialog, you can enter the value for the Barcode or QR code.

Barcodes are only suitable for numeric values whilst QR codes can handle alphanumeric values. Next select the text style that you defined earlier that is using the desired Special Font Type and click OK.

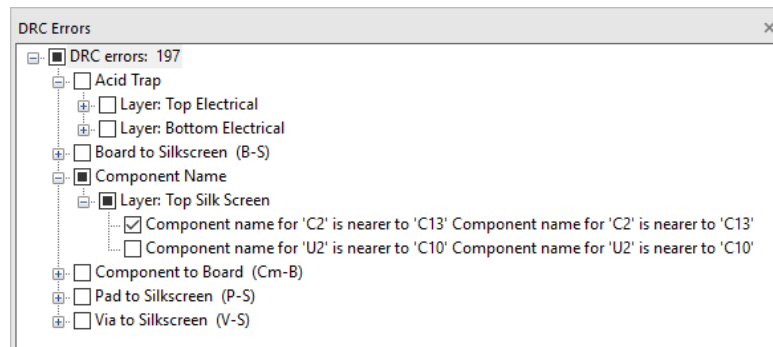
A QR code will appear in your design like this:



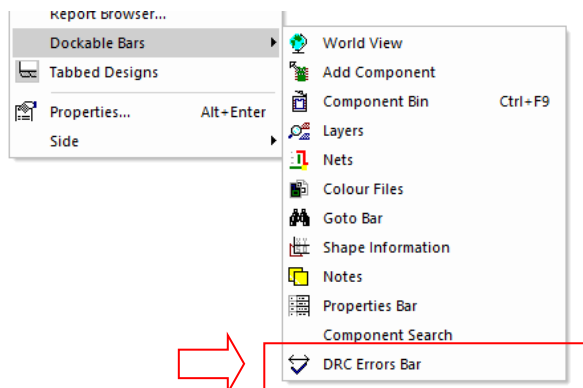
Errors Bar Added

There is now an **Errors Bar** that is available for both Schematics and PCB designs for **ERC Errors** and **DRC Errors** depending on what type of design you have open.

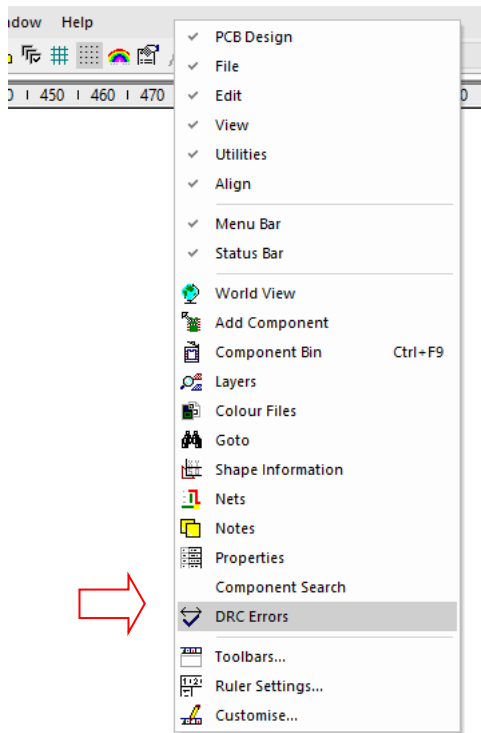
The dockable bar contains a tree which is populated with all the error makers in the design.



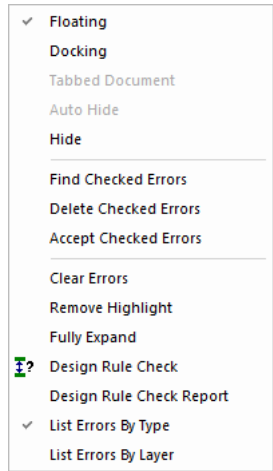
You can open the Errors bars by navigating to the **View** menu and **Dockable Bars> DRC or ERC Errors Bar**.



You can also access the Errors Bars by right clicking on the application framework and selecting it from the menu.



Right clicking on the bar reveals a context menu that provides you with options such as finding, deleting and in the DRC option, accepting errors.

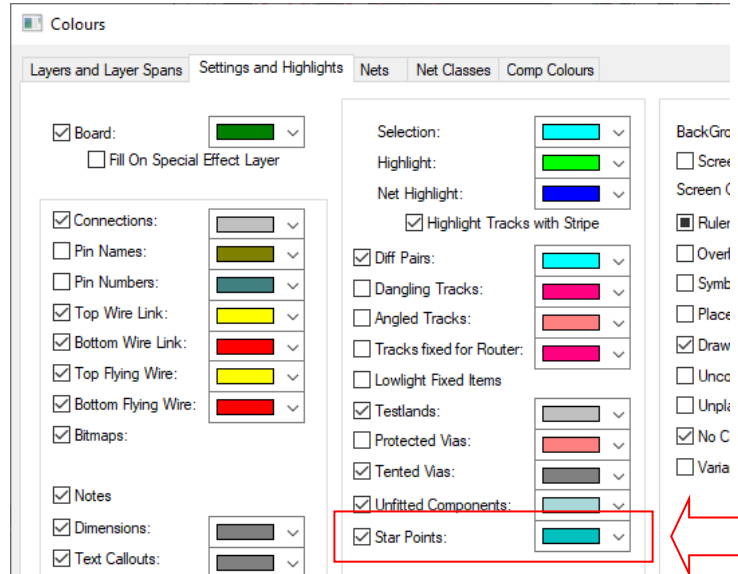


Double clicking on an error marker in the tree list will locate that error marker in the design.

Note, the Goto bar still also contains Errors. The new Errors Bar provides more detail and categorises the errors found.

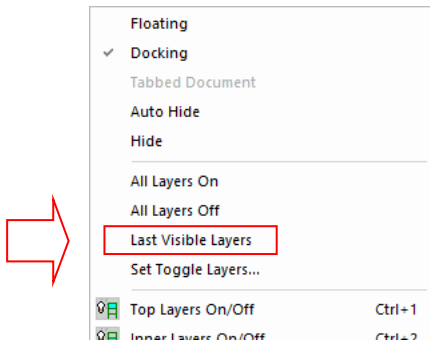
Colours –Star Point Visualisation

Within the **PCB Editor**, the **Colours – Setting and Highlights** dialog now has a **Star Points** option which, when enabled, will display all star point pads in the chosen colour.



Last Visible Layers context menu option in Layers Bar

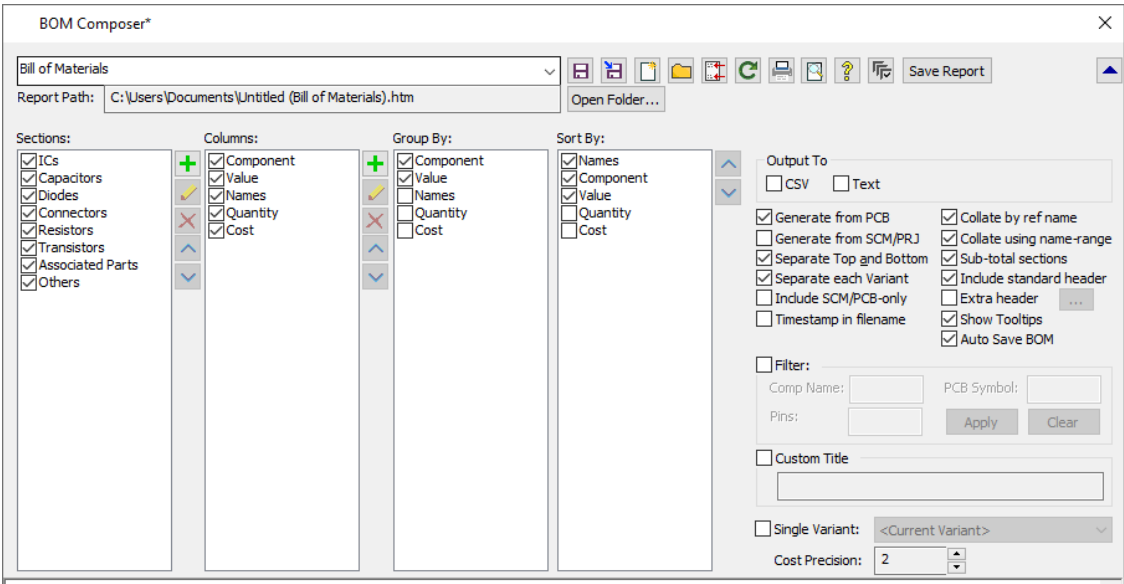
On the **Layers Bar** context menu, there is a new command to show **Last Visible Layers**.



This command allows you to toggle between the last set of visible layers and the current set. The last set of visible layers is updated each time a layers' visibility has been changed or when one of bulk layer visibility change commands has been selected, such as **Display All On**.

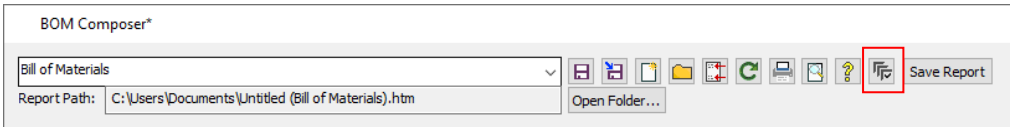
BOM Composer Enhancements

From the **Output** menu, **BOM Composer** option, there are new features added to the dialog:



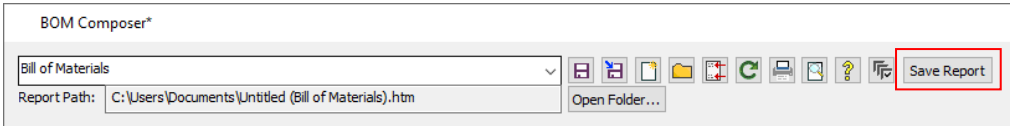
Open Preferences Button

There is a new icon named **Open Preferences**. When clicked, this will open the **Preferences** dialog. This is useful for quickly changing any of the **Report** settings without leaving the BOM Composer dialog.



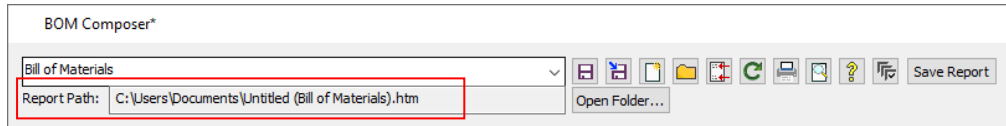
Save Report Button

The **Save Report** now icon has been changed to a button to make it more visible to promote its importance.



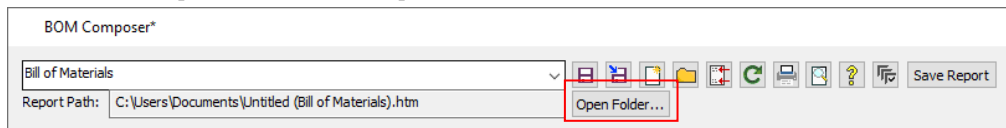
Report File Location

When selected, this new button displays the **Report Path** to the report file that is being generated.



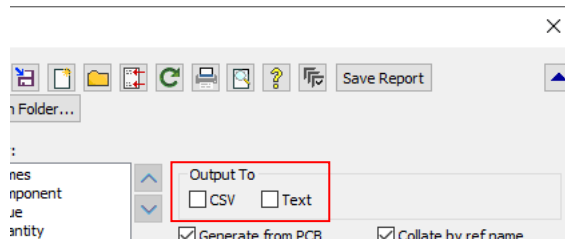
Open Folder

There is a new button to **Open Folder**. When selected, the folder where the output file is generated will be opened in a new File Explorer window.



Explicit output to Plain Text

There is a new section called **Output To**. Within this section are two check boxes **CSV** and **Text**.



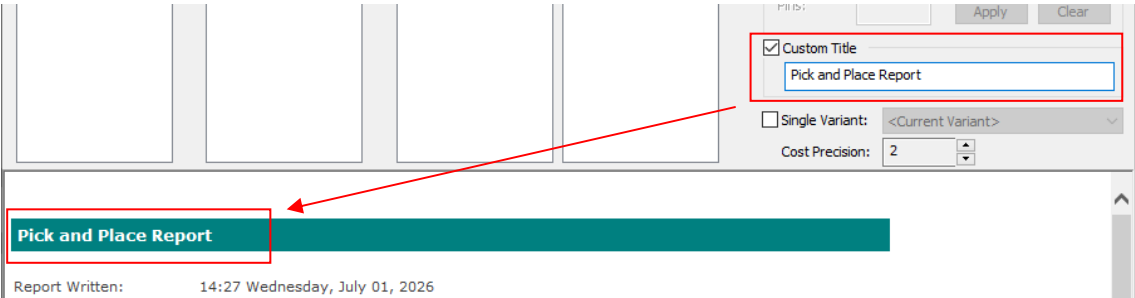
CSV performs the same action as the existing **Write CSV** check box.

Text is a new option where if ticked, an additional plain text copy of the report will be generated.

Please bear in-mind that the new **Text** option will ignore the setting of the **Suppress Text Reports** option in **Preferences – General**.

User Defined Report Title

On the BOM Composer dialog, there is a new section named **Custom Title**. If enabled, you can define the title of the report. This title will override the default **Bill of Materials** title inserted into the report.

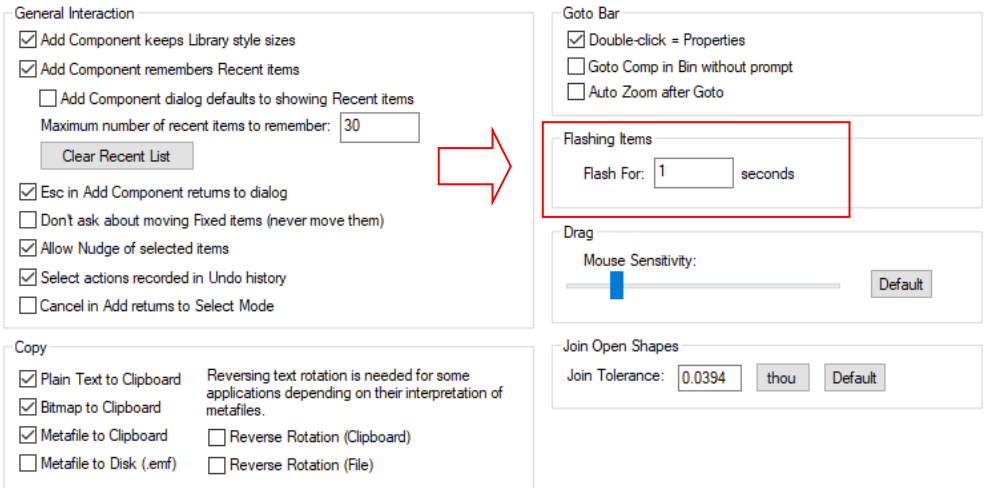


File Already Open Warning

When generating a report through the BOM Composer, you will now get a warning if the report file could not be opened for writing. This warning informs you of the file that is locked and the path of the temporary report that will be created instead.

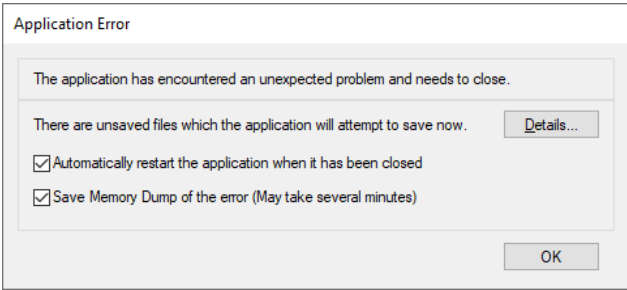
New Flashing Items settings

The **Preferences – Interaction** dialog now contains additional controls to set the length of time in seconds for which the flashing of found items will last. This includes items found in the **Goto Bar** functionality.

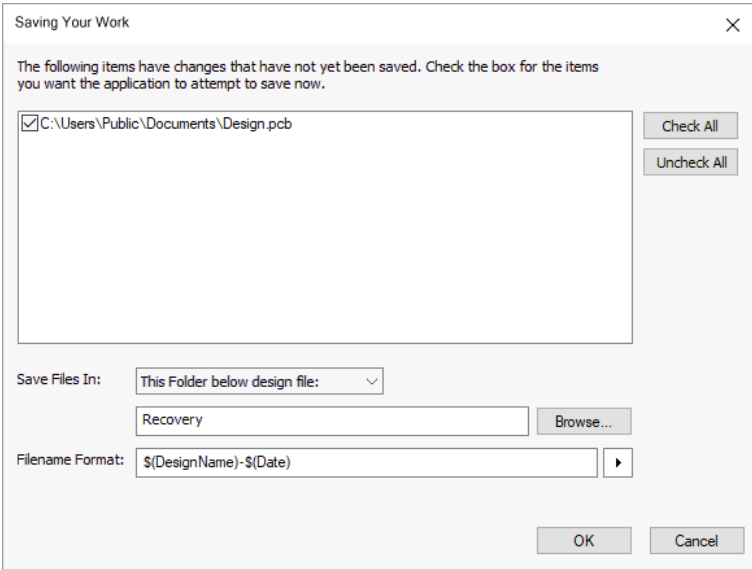


Improved Application Error Handling

When the application encounters a serious error that it cannot recover from, a new **Application Error** dialog will appear.:



Pressing **OK** will present you with the **Saving your Work** dialog:



This dialog allows you to optionally attempt to save any open modified designs or the Project. This will save a memory dump and support file, which can be sent to the support department. This will help the developers reproduce the issue and potentially provide a fix. It restarts the application once the existing instance has closed.

Note, if the source of the problem is underlying software, such as the operating system or start-up drivers, the application may not be able to generate this dialog.
