

Version 22 Software Update Details

Problem Fixes in Version 22.0.4 (30-Jan-2019)

Drawing

- Text items and Value Positions were not being drawn in the lowlight colour when Fixed.

Gerber

- When plotting a panel, pads using an offset shape were plotted in the wrong position if the PCB instance on the panel was mirrored.

N.C. Drill

- When writing the format specification for leading and trailing zeroes (LZ and TZ) to the header of an Excellon drill file, the value being written out was the opposite of the correct value.

Plotting

- When using the Auto Mask feature to auto-generate paste and resist masks for copper shapes in footprints, the auto-generated shapes were being created on too many layers.
- In certain situations, plotting silk screen with the 'break shapes' option enabled on the Layer Type could cause the application to quit.

Problem Fixes in Version 22.0.3 (21-Dec-2018)

Component Edit

- It was not possible to name a PCB pin using the same number as was already 'reserved' for an unmapped (unassigned) PCB symbol pin. Such unassigned pins are now prefixed with "#" to allow those pin numbers to be used as required for 'real' pins.

DXF Output

- Slots were not being included in the DXF output file.

Plotting

- When auto-generating the set of plots for a PCB design, no plots were created for resist or paste if the oversize (or undersize) value on the Layer Type was set to zero.

Pour Copper

- Attempting to pour an area that is not attached to a net would cause the application to quit if that area ended up having 'no connected copper'.

Problem Fixes in Version 22.0.2 (23-Nov-2018)

Add Inserted File

- Attempting to insert an empty text file was causing the application to quit.

Associated Parts

- Adding an Associated Part to a Component in the library was not copying across the Description and Values.
- The Help page for Update Components did not include a description of the option to 'Keep Associated Parts'.
- When adding Associated Parts using the Properties dialog for a Component, clicking Apply then OK would add the parts twice.

Cross Probe

- The Help page for Cross Probe did not mention how to cancel out of the cross-probing mode.

Databook

- The coordinates and sizes displayed for Symbols and their pads/terminals were always displayed in thou/mil and did not use the units of the chosen Technology file.

File Open/Save

- Occasionally the standard Windows file dialog would stop working, not browse folders, ignore the Open/Save button, and other odd behaviour.

Folders

- For Report Templates, enabling the setting to 'Include Project Folder' had no effect.

Gerber Import

- When using the Add button to add more files to be imported, the coordinate precision was not always being read from the Gerber file.
- The Intelligent Gerber Import dialog would allow the Technology file selection to be "[None]" which was not helpful as you need the layers defined in a Technology file in order to have somewhere to place the data.

Go To

- When typing in the name of an item (such as a Component) in the edit box below the category name when working in a Schematic design, the keyboard focus was being lost as soon as the first match was made with an item in the list, preventing you from typing the rest of the name.
- The list of styles shown for Pad Style included styles that were not actually used in the design because they had been over-riden by 'Pad Style Exceptions' on the Component pads.

Groups

- Duplicating a selection containing groups was not also duplicating the group state to the new items.

Library Loader

- Invoking a new instance of the application from Library Loader would sometimes cause the dockable bars to misbehave.

N.C. Drill

- The 'zero suppression' setting (leading or trailing) was not being included in the header of the drill file unless the option to include Precision Format was also enabled.
- Setting the option to include the decimal point in the output data had no effect.

PDF Output

- Any shape that was no longer marked as filled but had in the past been set as filled and hatched would still be output to PDF as a filled shape.

Plotting

- Depending on the settings for the folders for plot output, sometimes additional unnecessary sub-folders would be created.
- If you add a plot to generate IDF data and also choose to Zip the output files, the IDF files would not be added to the Zip file.
- On the Compose dialog used to build up a template for file/folder names, the button for a 'hyphen' did nothing.
- The Compose dialog would not allow you to enter some characters that are not allowed in file names, thus preventing the ability to set folder names such as "..\Plots".
- Clicking on Copy Plot to make a copy of the currently selected plot, both original and new plots ended up as selected in the list, instead of just the new one.
- When saving a Plot Job file, some of the settings for a layer of type 'Special Effect' were not being saved.
- In some situations, generating plots for silk screen with the option to 'break around pads' enabled would cause the application to quit.
- The Settings tab for Drill Ident plots included a spurious extra checkbox.
- When plotting a pad where the drill hole was larger than the pad land, a round item the size of the drill hole was being plotted, instead of simply plotting the pad land as defined.
- The Plot Preview for a Drill plot was not displaying the holes for vias, only those for pads.
- When plotting a negative plane with the option to include board outlines enabled, the board outline was not being included in the plot output.
- When adding DXF as an output on the Plotting dialog, the DXF settings dialog would display -1 as the default value for the coordinate precision, and would not always remember the setting for 'Use Design Units'.

Pour Copper

- It was not possible to get pour copper to merge copper at a Component pad marked as a Star Point.

Problem Fixes in Version 22.0.1 (26-Jul-2018)

BOM Composer

- Clicking on the first list box entry under the Sections heading was causing the application to quit.

Component Edit

- It was only possible to type a small selection of the full character set into some of the edit boxes in the Component Editor.

Design Rule Check

- Calculations for the Acid Trap check did not take account of the existence of some items (such as the connect points between tracks and copper) that do not have any pad style, and would thus cause the application to quit.

Dimensions

- The new separate visibility setting for Dimensions was being initialised to off (hidden) where it should have been on (visible), giving the impression that dimensions added to designs in earlier releases were being lost when in fact they were only hidden.

Display

- The new command True Width Off was leaving true width enabled for poured shapes.

Duplicate

- The message displayed when being asked about merging nets during duplicate contained the wrong text, telling you instead that there were no layers available for adding shapes. The dialog would still work, but the message no longer told you what the buttons did.

Grids

- On checking the box to enable display of the Secondary grid, the associated controls for colour and lines did not become enabled until you had been out of the dialog and back in again.

Plotting

- On entry to the plotting dialog in a PCB design, the only plot being automatically created was the drill output, all the normal Gerber plots were missing and had to be generated using the New Job button.
- The plot preview was failing to resize the picture to show the entire plot area.
- When running a plot that generated a BOM Composer report, if the option to put output files into a Zip was enabled then this report was not moved into the Zip.

Properties

- On the component properties tab, it was possible to remove the component name entirely without getting any error message displayed, thus leading to the presence of unnamed components that could cause problems for some functions including auto-routing, as all components are expected to be named.

Setup

- On some systems, setup stopped with "unable to load Keylock.dll" message until Visual Studio pre-requisites were installed by hand.

Copyright © 1997-2018 WestDev Ltd

Number One Systems and Easy-PC are trademarks of WestDev Ltd

All trademarks acknowledged to their rightful owners. E&OE