Release Notes GC-PowerStation v18.3

Table of Contents

New and Improved Features

Smoother Text Apertures

The descriptions of many Text Fonts included in the software contained Bezier curves which are not allowed within the Gerber specification and therefore would cause problems on output if retained. Text characters were therefore vectorized in order to be compatible with other features. This vectorization was at a fairly low resolution and the resulting text characters did not appear smooth at high zoom levels. The vectorization process has been improved to allow smoother vectorization of Bezier (and non-Bezier curves) and the resulting Text characters are much smoother at high resolutions.

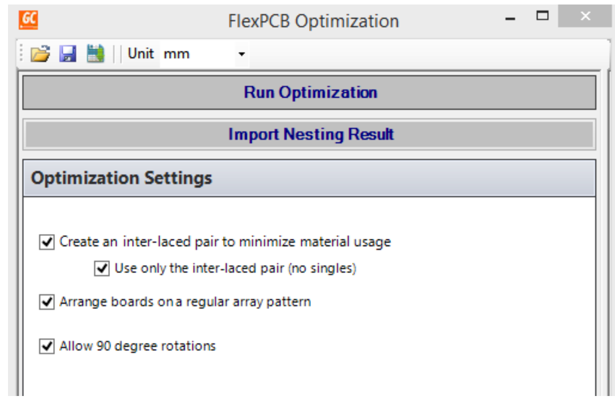
Before:

After:

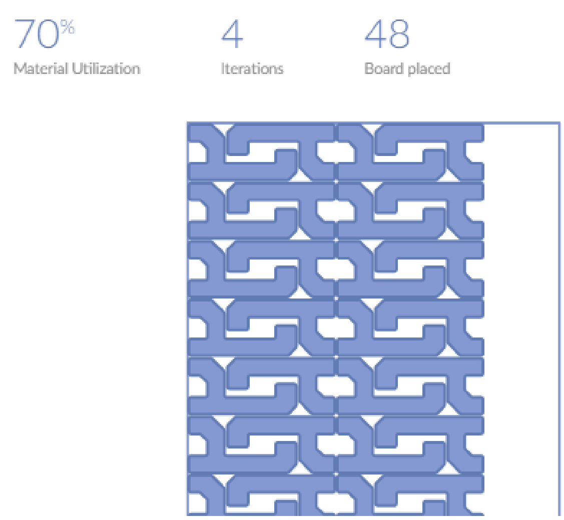
Note: Text generated using the original method will still display with the original vectorized appearance when loaded from a GWK into the v18.3 software. This is to retain image integrity within the GWK. Refreshing the font within the new version will update the text to the new vectorized appearance.

Flex PCB Panelize

A new function has been added to better step and repeat odd-shaped PCBs. The function allows a nested pair of boards to be created that optimizes material usage based on the PCB outline and then can step and repeat that pair across a panel. The function also allows for additional single boards to be added to the panel if it is possible to further improve material usage.
There is also the ability to pass the complex step and repeat image generated back to the GWK so that the panel can easily be generated using the Apply Panelize Layers function.



The algorithm to place the boards can be run as many times as needed to see the effect of changing input variables.



Added numerous Plugin functions

A number of plugin functions have been added to enhance the customization of the product. Details can be found within the Intellisense for the plugin.

Items Fixed since v18.2

This list is customer reported issues fixed for this release.

#603 An issue with non-symmetrical apertures being incorrectly rotated within a rotated ODB++ STEP has been fixed.
#602 The Remove Material function added to the Plugin Extension.
#600 Updated interpretation of illegally drawn apertures in Gerber. If a custom aperture is defined as a trace but the length of the trace is zero, then a pad is now created using the Customer aperture and a warning is displayed to the user. The improvement is to better display the intended image. All non-zero length traces are drawn with a Round Aperture and an error is produced upon import.
#598 The new implemented fonts (#552) for UniCode characters caused text apertures to be graphically different in v18.2 as compared to older versions. We now gracefully handle this situation by creating a scaled version of the Text aperture so that the result is graphically identical to whichever version the GWK was created.
#597 Poor Gerber construction from new version of EAGLE CAD resulted in multiple layers unnecessarily being created.
#596 Updated the DPF output to no longer generate contours that had breaks in their path. This was done for Gerber output but due to the limited use of DPF format, this had unfortunately been overlooked.
#595 Changed the behavior of the drawing engine when loading ODB++ database files. The setting for the Negative Layer Elements is now read from the registry settings and no longer defaulted to the 'Color' option.
#593 Checking the Package Tab of the Properties page after opening a PADS ASCII database results in a software crash. Problem has been fixed.
#591 New functions have been to the Plugin for Drawing Negative Layers in Color or Black, Creating Step and repeat patterns and Drawing Panels in Full or Quick mode.
#589 Improved the smoothness of Text apertures (see above).
#588 Eagle CAD Excellon Drill output containing comment lines with a '%' character included were causing a false End of M Code action. This resulted in the file failing to load. This has been fixed.
#587 Fixed a bug that caused the C-Pad creation function to fail in v18.2.
#586 Fixed an issue causing a crash while importing a Gerber file into 32bit build of software.
#585 ODB++ output modified to better represent soldermask images. The soldermask in ODB++ is defined using positive apertures so matching the GraphiCode positive / negative interactions on a soldermask was incorrectly representing the GraphiCode image in the ODB++.
#583 Updated the situation of a missing aperture definition to an Error rather than a Warning. The image is still drawn using a dummy sized aperture.