

Release GC-Prevue v26.3 Notes

Table of Contents

NEW AND IMPROVED FEATURES.....	2
SMOOTHER TEXT APERTURES.....	2
ITEMS FIXED SINCE V18.2.....	3

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.

Items Fixed since v18.2

This list is customer reported issues fixed for this release.

#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.

#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.

#586 Fixed an issue causing a crash while importing a Gerber file into 32bit build of software.

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.