

# Release Notes GC-PrevuePlus v18.1

## Table of Contents

ITEMS FIXED SINCE V17.3.....	2
------------------------------	---

## Items Fixed since v17.3

This list is customer reported issues fixed for this release.

#559 Windows Update #1709 caused the 32 bit version of products to crash when importing data. Hot fix was issued and updated version released.

#558 Internal rules were tweaked to better address very small arc segments in Gerber data. If the length of the arc segments is less than one micron but greater than 1 nanometer a straight line segment is created and a warning is generated in the Import Results.

#557 Updated the NC Rout import to contain all drill locations specified in the file. Previously, certain holes were not being created when auto-identification of a file labelled the file as a Rout file.

#553 Bug reported when Merging two ODB++ files for comparison. The situation was that two versions of the same file were being merged into the same GWK. The BOM contained the same components and when the second version of the board was merged in, certain components has additional rotations applied. Fixed.

#552 Creating text with certain Unicode fonts resulted in incorrectly sized text characters being created on the layer. The issue has been fixed.

#549 Added Plugin support for Enhanced DXF / DWG export.

#548 Fixed a legacy issue within GC-Basic that caused a script crash when using the MoveDataLayerfromUnassigned function.

#547 Fixed a small bug that caused the DXF export to fail if a whitespace character existed as the first character in the filename of the layer to be exported.

#546 Fixed an issue that incorrectly positioned the PCB after an ODB++ import. The problem was caused by Steps within rotated Steps being incorrectly interpreted.

#542 Fixed an issue when importing step and Repeated data in the Gerber X2 format. The result has been verified.