

# Release Notes GC-PrevuePlus v21.1

## Table of Contents

- NEW AND IMPROVED FEATURES ..... 2**
  - COMPONENTS LOADED FOR GERBERX3 ..... 2
  - COMPONENT PIN INFORMATION LOADED IN IPC-356..... 2
  - ADDED PLUGIN FUNCTIONS..... 2
- ITEMS FIXED SINCE V20.3..... 3**

## **New and Improved Features**

### **Components loaded for GerberX3**

The newly released GerberX3 format includes information about components placed on the circuit board. This information is now interpreted and all available information is created on a Top or Bottom Parts layer along with the information about the component extents which are created on a component outline layer.

Copper pad sizes are not assigned to the component layer as the Gerber layer does not describe this specifically. Pin 1 for each component is displayed as a diamond shape and the other pins as small round pads. This information as loaded will allow for the Component Placement List to be generated as well as identifying components through the Jump to Component function.

If pad sizes and shapes are required to be present in the output then a new function called Merge Footprints will merge a Pad layer (the Footprints) with the available component information in order to update the Part layer with the pad size and shape for all available pin locations. This Merge Footprints feature is only available in GC-PowerPlace and the Assembly Programming option for GC-PrevuePlus.

### **Component Pin information loaded in IPC-356**

IPC-356 files can contain part / pin information associated with a point. Previously this information was ignored on import. The information is now assigned to the pin as an attribute that can be viewed in the Properties page. This attribute information is also accessible through the Plugin so that it can be utilized further. The attribute name is .P with a value of Part,Pin (for example C5,2) which also matches the Gerber X3 attribute descriptor.

### **Added 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 v20.3

This list is customer reported issues fixed for this release.

#684 Enhanced the Auto Convert Sketched Pads algorithm to reduce the number of apertures generated during the conversion process. Previously a larger than necessary number of custom apertures were generated.

#681 Updated the ODB++ input process to handle layers containing both components AND pad/trace data. This addresses the previous restriction of buried resistor being handled.

#679 Updated the old FTIP2 script so that it works with the current build of software.

#678 A common issue where dialogs were stored off-screen has been resolved. Moving from two screens to one screen had occasionally resulted in the software appearing to freeze because a dialog was displayed off screen. If the dialog is detected to be out of the visual range of the current screen it will be displayed in the default location.

#676 Gerber X2 file contained an attribute that contained previous RS-274X command characters. The code was updated to ignore command characters within attribute definitions.

#674 Defined D-Code within a file, but embedded within the XY co-ordinates was incorrectly specified as a new aperture and then subsequently used. This produced an incorrect image. The D-Code is now ignored unless a G54 command is defined.