Gerber extends into assembly

A call for review and comment on new draft Gerber specification

*Ucamco's new **Gerber X3 format** includes essential information about components, extending Gerber's simple yet powerful capabilities beyond the bare board sector and into assembly. Ucamco welcomes comments on its proposed new specification, which can be seen at tiny.cc/ekqvfz or via https://www.ucamco.com/en/news/towards-gerber-x3*

Gent, Belgium – November 7, 2019 – Ucamco is delighted to announce its plans to further develop the PCB industry's premier data transfer format. Currently at draft stage, the new Gerber X3 specification adds component layers to the top and bottom of the Gerber file structure, enabling circuit designers to add assembly information and communicate as clearly with their assembly manufacturing partners as they do with PCB fabricators. The new layers are described using Gerber's trusted syntax and methodology, so they are a simple add-on for developers.

The new layers show component location, shape, fiducial locations and footprints, all data which fit naturally with the Gerber format's image files. This data is supported by a new set of attributes that are specific to the component layers and that provide non-geometric information about the component such as manufacturer part numbers.

Ucamco's Karel Tavernier explains this latest development: "While designs for bare boards are transferred using Gerber files, the associated component data is typically contained in separate, non-standardized drawings and pick & place and BOM files. The Gerber format lends itself perfectly to the transfer of component data as well as bare board data, and so it is a natural step for the format to be used to describe the board in its entirety". He adds that Gerber X3 remains fully compatible with existing workflows, installed base and legacy software: "The new component data is in dedicated files, separate from the bare board files, so if this data is not needed, these files can simply be ignored and the manufacturer proceeds as before".

The combination of bare board and component data in Gerber files provides an integrated overview of the entire board, enabling operators to:

- visualize component placement to check for errors and set up assembly
  generate manufacturing tools such as paste stencils and programme pick and place machines
- assist in component procurement

It is easy to read Gerber output CAD data into the assembly company's CAM system, which then analyzes the incoming data, allowing for visual inspection and the generation of programmes for assembly production line equipment.

Under Ucamco's unstinting stewardship, Gerber goes from strength to strength. The PCB industry's de facto standard for image data transmission, Gerber is a brilliantly simple, intelligent format that enables designers to convey their intentions clearly, unequivocally and easily to manufacturing partners all over the world, irrespective of language and manufacturing culture. Using simple machine-readable syntax that is also straightforward enough to be understood by people, Gerber has been putting designers and PCB fabricators on the same page for decades. Thanks to the new Gerber X3 extension, the world's assembly companies will soon be joining them.

This draft proposal was not developed by Ucamco alone, but in discussion with industry experts, who are acknowledged in the draft. Our special thanks go to *Jean-Pierre Charras*, who developed a

prototype X3 input/output for KiCad thus testing the format under real conditions and to *Wim De Greve*, head of software tools development at Eurocircuits and early adopter of the X3 format to receive complete CAD data sets from designer customers – they contributed so much to the draft, especially to keep it simple, practical and down-to-earth.

The new draft specification can be seen via https://www.ucamco.com/en/news/towards-gerber-x3
Comments are welcome at gerber@ucamco.com.

About Ucamco

Ucamco is the market leader in PCB CAM sand Pre-CAM oftwareLaser Photoplotters and Direct Imaging Systems, with a global network of sales and support centers. Headquartered in Gent, Belgium, Ucamco has over 25 years of ongoing experience in developing and supporting leading-edge photoplotters and front-end tooling solutions for the global PCB industry. Key to this success is the company's uncompromising pursuit of engineering excellence in all its products. Ucamco also owns the IP rights on the Gerber File Format through its acquisition of Gerber Systems Corp. (1998).

For more information on Gerber X3
please contact Ucamco:
Phone: **+32 (0)9 216 99 00**
Email: ***gerber@ucamco.com***
Web: **www.ucamco.com**