Ucamco provides reference Gerber file viewer to the Gerber user community – free of charge.

Gent July 18th 2017 - Ucamco spared no effort to display all valid Gerber files correctly. The main goal of the viewer is to provide an easy way to see the correct interpretation of a file. Diagnostics tools help to analyze it. While it reverse engineers the intended image on an invalid file it will give a warning if it contains risky errors, thus helping to detect invalid files. It is positioned as an easy-to-use reference and a complement to specification. For this reason we called it the **Reference Gerber Viewer.** It will help to get and keep Gerber files on the straight and narrow path of full compliance to the specification. It is to the benefit of the whole Gerber user community to all sing from the same hymn book.

Note that this is a Gerber file viewer, not a PCB fabrication data set viewer. It does not do NC or IPC-D-356 files – as these are not Ucamco formats and it is not up to Ucamco to write a reference viewer for these.

The Reference Gerber Viewer is kept simple and easy to learn. As a cloud-based on-line web service there is no software to download, install and maintain – it is always up to date.

It is available at gerber.ucamco.com and its use is free of charge. It is also OK to forward the link to our online reference viewer (http://gerber.ucamco.com) or to integrate it in your website.

The Reference Gerber Viewer offers the following benefits:

- **For developers,** it provides an easy way to test their Gerber output and to answer questions about the interpretation of the specification.
- **For people who create Gerber output**, it provides an easy way to check the image of the file they are about to send.
- **For people who receive Gerber files**, it provides an easy way to validate their input and arbitrate in discussions about the valid interpretation of a file. Sharing the link to the reference with the file senders will avoid damaging misunderstandings.

About Ucamco

Ucamco (formerly Barco ETS) is a market leader in PCB CAM software, photoplotting and direct imaging systems, with a global network of sales and support centers. Headquartered in Ghent, 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) and continues to develop it.

For more information on Gerber X2 please contact Ucamco:

Phone: +32 (0)9 216 99 00
Email: info@ucamco.com
Web: www.ucamco.com