

Ucamco Releases Specification for Fabrication Documentation in Gerber

The specification describes the new **Gerber Job File** which transfers fabrication documentation in a machine-readable manner as part of Gerber fabrication data.

Gent, Belgium – 23 April, 2018 – Since decades Gerber is the standard for describing the images – copper layers, solder mask, drills – in PCB fabrication data. Attributes introduced with Gerber X2 ‘add intelligence to the image’ by identifying layers, vias, SMD pad, component pin numbers and reference designator.

But PCB fabrication data is not just about images - it includes fabrication documentation such as finish, overall thickness and materials. It is essential for the quoting, planning, engineering, CAM and fabrication of the bare and assembled board. It is often transferred informally, in drawings and texts, and handled manually, wasting time and risking errors. The Gerber Job File transfers this information in a machine-readable manner as part of Gerber fabrication data.

The extension is compatible with both Gerber X1 and X2. Existing image input/output code does not need to be changed when implementing the Gerber job file.

The Gerber hallmark of simplicity and human-readability is maintained.

The Gerber Job File is straightforward to implement. Partial implementations are allowed, even encouraged. Better a partial job file than no job file at all. Developers can pick and choose from the spec what suits them. Several software suppliers have already committed to support the new Gerber Job File.

The initial draft was published on the Ucamco website for public review by the Gerber community in March 2017. There was a lively discussion, the draft went through seven public revisions before being finalized early April 2018. It became much more practical in the process.

The specification and example files are available at the Gerber page at www.ucamco.com.

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).



*For more information on Integr8tor,
please contact Ucamco:*

Phone: +32 (0)9 216 99 00

Email: info@ucamco.com

Web: www.ucamco.com