

## Simple, streamlined CAD-to-CAM communications

Thanks to Ucamco's new Gerber Job File Editor, designers can create and edit job files that provide all essential PCB production data, including essential non-image-based information, in a standard format that can be put into an automatic production flow. Ucamco's principal aim in doing so is to make life easier for electronics design and manufacturing professionals by facilitating clear, unequivocal communications prior to production.

**Ghent, Belgium – June 26, 2019** – At Ucamco, we are dedicated to excellence in everything that we do. That includes our decades-long stewardship of Gerber, the de facto standard image format for PCB design: our ongoing work on Gerber ensures that this free, open format keeps abreast of, and indeed drives, beneficial developments in PCB design and engineering. As part of this, we are delighted to announce that our Gerber Job Editor is now available free of charge to the PCB design and manufacturing community. This standalone application allows designers to create and edit PCB job files complete with everything including non-image data, that is essential for quoting, planning, engineering, CAM and fabrication.

For decades, Gerber has been the de facto standard for describing 2D images – copper layers, solder mask, drills – in PCB fabrication data. Gerber X2 added intelligence to that image data by providing a standard for defining the layer structure and providing information such as via and SMD identification, pin numbers and reference descriptions.

So far so good, but PCB fabrication data must also include essential information about IPC class, finish, soldermask colour, thicknesses and materials and so on, that cannot be conveyed using images. This is typically transferred manually using annotated drawings, PDF files and texts, which takes time, is prone to potential errors, can be limited by language and terminology barriers, and is not designed for automatic workflows.

Ucamco's Gerber Job File specification changes this by providing designers with a standardised format with which the non-image-based elements in their PCB designs can be transferred to the PCB manufacturer in an unequivocal, machine-readable manner. Compatible with Gerber X1, Gerber X2 and ODB++, this is a simple add for CAD suppliers and indeed many of the leading CAD systems support it and can output complete Gerber job files. Where this is not the case, Ucamco's new standalone Gerber Job File Editor allows PCB designers to create job files that fully describe their PCBs' non-image characteristics, no matter which CAD system they use. It's simple to download and simple to use. The designer chooses which characteristics are important to each specific PCB and ignores the rest, and even if the fabricator cannot yet handle the Job File automatically, the fabrication data is clear and presented in an unambiguous, standard, human-readable manner.

Gerber Job File Editor can be downloaded free of charge from

[https://www.ucamco.com/files/downloads/Gerber\\_Job\\_Editor\\_Installer.zip](https://www.ucamco.com/files/downloads/Gerber_Job_Editor_Installer.zip).

Download your copy today and discover just how simple it is to make your life a whole lot easier.

### 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 35 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](mailto:info@ucamco.com)

**Web:** [www.ucamco.com](http://www.ucamco.com)