



Call for constructive comment
an open invitation to the global PCB design and CAM community
Proposal to include fabrication documentation in Gerber

Gent, march 22th 2017 - 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. Our work draws on our deep understanding of the needs of the PCB industry and its design and engineering functions, but also on the invaluable feedback that we receive from you as designers and engineers – because your day-to-day experience of what it means to use our hard- and software makes you the real experts.

Now, we welcome your input on our latest proposal, which is to include the fabrication documentation in Gerber. Our principal aim in doing so is to make life easier for electronics design, manufacturing and assembly professionals by facilitating clear, unequivocal communications prior to production. Our draft document proposes an evolutionary extension of Gerber that enables production parameters to be communicated in a standard format that can be put through an automatic workflow. Far from being a full and final draft, it is intended as a discussion document to solicit input and reach a consensus in the user community on how to handle these parameters. The proposal will probably go through several revisions based on this input.

We ask you to download the proposed new draft specification [here](#) and let us know what you think by sending your comments to us at gerber@ucamco.com

Further reading

Since time immemorial, Gerber has been the standard for describing the 2D images – copper layers, solder mask, drills – in PCB fabrication data. Now, attributes added with Gerber X2 provide a standard for defining the layer structure – a 'which layer is which' file – and to 'add intelligence' to the image by providing information such as via and SMD identification, pin numbers and reference descriptions.

But PCB fabrication data is not just about describing images. It must also include general information about the final PCB; production parameters such as solder mask color, finish, overall thickness, materials and the assembly array definition. These parameters cannot be conveyed effectively by an image and yet they are essential for the quoting, planning, engineering, CAM and fabrication of the bare board as well as for determining the optimal assembly array.

As there is currently no standard for this sort of data, it is typically transferred informally using drawings, PDF files and e-mail messages. None of these are designed for automatic workflow, so the information must necessarily be handled by people, which takes precious time and carries the risk of errors.

Our aim is to create – with your input – a simple, standard way of conveying this sort of data in a format that can be handled automatically. The standard must be easy to adopt and to implement, it must be compatible with existing workflows, and it must allow for partial implementations as some applications may only 'know' a few production parameters.

We invite you to look at the proposal and look forward to hearing from your input and constructive comment at gerber@ucamco.com

Thank you.

Karel Tavernier,
Managing Director,
Ucamco