

Call for constructive comment

an open invitation to the global PCB design, CAM and electronics assembly community

Proposal to include CAD Netlist in Gerber

Gent, Belgium - August 10th 2016 - 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 – our industry's designers, engineers, fabricators and assembly professionals – 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 CAD Netlist in Gerber. Our principal aim in doing so is to make life easier for you as electronics design, manufacturing and assembly professionals by facilitating and empowering clear, unequivocal communications prior to production.

We have taken great care to make this proposed new capability complete, and yet in true Gerber style, we have kept its implementation and adoption simple, straightforward, non-mandatory and fully compatible with existing and legacy files and applications.

Before this proposed new draft proposal goes live, we ask you to read it [here](#) and let us know what you think of it by sending your comments and constructive criticism to Ucamco at gerber@ucamco.com

Thank you.

Karel Tavernier,
Managing Director,
Ucamco