Gerber Netlist goes live with KiCad

KiCad is the first CAD tool to implement the Gerber netlist and component attributes.

Gent, Belgium – November 8, 2016 – Earlier this year, a draft specification for netlist and component attributes in Gerber X2 was published on the Ucamco website by its Managing Director Karel Tavernier, and made available to the Gerber community for review and comment. Following a number of revisions, the final specification was published on on October 2.

KiCad's founding creator Jean-Pierre Charras contributed substantially to the specification and its review process, and also worked closely with Gerber's developer Ucamco to ensure the successful implementation of netlist and component attributes in KiCad's own Gerber X2 output. The output has been fully verified by Ucamco as conforming perfectly to the specification and a sample file is available on the Ucamco download page. The new output, says Charras, will be released in KiCad's PCBnew in nightly builds at the beginning of November 2016. "In Gerber X2, Ucamco has given us a format that provides greater transparency and information in the output stage of the design process. By adding the netlist data in this format, we demonstrate our commitment to providing the best possible manufacturing outputs in our PCB layout program and to keeping our Gerber output at the highest standard: netlist and component information is essential for assembly and for the communication between all parties in the PCB design and manufacturing process. It's always good practice to verify Gerber files produced by board editors, and here, a Gerber viewer like the KiCad GerbView tool, which can highlight nets and graphic elements from a given component, is really very useful. "

Ucamco's Managing Director Karel Tavernier states: "As it is compatible with current workflows, X2 is the safest, most cost-effective and practical way to take PCB fabrication data to the next level. Now, with CAD netlist, X2 provides a powerful checksum of the image data and makes data transfer even more secure. We are honoured that a well-respected software such as KiCad is implementing the Gerber netlist and component output, and that they should do this within weeks of its release illustrates our point that X2 is straightforward to develop. By supporting X2 and adding the CAD netlist KiCad is showing what it means to be a good PCB citizen, helping not just its own users, but the industry in general, to move PCB fabrication along the most practical path forward".

About KiCad

KiCad is a mature ECAD tool (Free Open Source Software) for board design. It contains KiCad, a Project Manager, Eeschema, a schematic editor that allows simple and complex hierarchies with analog simulation, and Pcbnew, a board editor (32 copper layers, a push and shove router and a 3D viewer). It also includes a number of helper tools including the GerbView Gerber viewer that enables the verification of Gerber and drill files.

For more information on KiCad please visit
http://kicad-pcb.org/

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 Gerber please contact Ucamco:

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