

## Graphiccode's GC-Prevue v22.3 supports new Gerber X2 format

**Gent, Belgium - Snohomish, WA United States - July 29, 2014** Graphiccode is pioneering a wave of developments in PCB software that promises to revolutionise the PCB design to manufacture process: the latest version of its GC-Prevue software now fully supports Ucamco's groundbreaking second extension Gerber X2 format.



Introduced earlier this year, Gerber X2 delivers unprecedented clarity to the CAD-to-CAM data transmission process. It does this by supporting the tried and tested Gerber image format with an equally clear and easy-to-use standard for all the additional data that precisely and reliably describes the image components. This includes details on stackup, the functions of layers and individual features such as pads, and a host of other parameters. With Gerber X2, designers can now rest assured that their manufacturing partners have all the data – and clarity – needed for efficient, reliable manufacture.

All without disrupting existing procedures, as Ucamco's Managing Director Karel Tavernier emphasises: "While X2 represents a revolutionary step towards unequivocal communication, it maintains the trademark simplicity for which Gerber has always been known and used, requiring very little change to working practices, and certainly none that would require approval, testing and all the rest".

The new development centres on the use of standard attributes to define a number of key parameters. These attributes were carefully selected by Ucamco to maximise efficiency improvements in CAD-to-CAM communications by describing characteristics such as:

- Gerber file function: top copper layer, top solder mask, etc.
- Part: single PCB, customer panel etc.
- Object function: SMD pad, via pad etc
- Drill tolerances
- Locations of impedance-controlled tracks
- Filled vias
- An MD5 checksum for added security

As the creator of GC Prevue, the PCB industry's leading image format viewer and the first to be offered freely to the PCB industry, Graphiccode was a natural partner for Ucamco's development of the X2 format. Graphiccode's Paul Wells-Edwards explains: "Used by countless PCB professionals globally and throughout the PCB pre-production process, GC-Prevue is the industry's de facto standard Gerber viewer, so we have worked closely with Ucamco and Gerber for years. The beauty of Gerber is that it's simple, and very widely used, and Ucamco's use of attributes is a very clever and straightforward way to improve and build on it. By extending the format and making it far clearer, Ucamco has improved the CAM task no end. When the opportunity arose to participate in that improvement, we jumped at it." He adds that the implementation of X2 is simple and straightforward, given that it consists of just 3 new commands and a clear list of standard attributes: "It took us just a week and a half of coding to be able to add the necessary information, so it's a very small step for any CAM system to be able to handle it".

Other software companies are following suit, according to Tavernier: "Altium, global leader in Smart System Design Automation, and creator of Altium Designer, has also been quick to recognise the value of X2 and will support it in the upcoming version of Altium Designer. Furthermore, I was recently informed that DipTrace and Kicad will output X2 in the course of 2014, and LPKF will support it from Q1 2015. Eurocircuits and AT&S offered to beta test it, and it will be in real production by the end of 2014, less than 12 months after its introduction".

X2 will boost productivity most and bring maximum benefits when used with fully X2-compatible software, enabling the whole Gerber archive to be read automatically, with all layers in place, while identifying the function of each object. And it's easy to use, says Wells-Edwards: "Adding intelligence to the core Gerber format is much simpler than people imagine. The reward is immense: a far better way of transferring data".

But even if X2 is used with older, non-compatible software, it will still generate the correct image – as demonstrated by the sample PCB jobs that can be downloaded from [www.ucamco.com](http://www.ucamco.com). This shows the format's upward compatibility, as well as the beauty and simplicity of the X2 concept. Tavernier: “Nobody is forced to buy anything to adopt X2. All users can happily use it with no problems, deciding in their own time if, how and when to adopt new X2-ready software to take their processes to the next level”.

**Gerber X2 file format specification, sample X2 archives and background articles on X2 are available for download at [www.ucamco.com/downloads](http://www.ucamco.com/downloads)**

**GC Prevue v 22.3 with X2 support is available for download at <http://www.graphicode.com/GC-Prevue>**

#### **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 please contact Ucamco:



#### **About GraphiCode**

GraphiCode is a global provider of software innovations for electronics manufacturing. Our products and services facilitate the fabrication, test, inspection and assembly of Printed Circuit Boards. We supply specific programming solutions to equipment manufacturers. We also provide value-added services through an international distributor network. GraphiCode's commitment to providing exceptional support and services ensures successful integration of our products into the manufacturing process.

For more information on GC-Prevue please contact GraphiCode:

