

## Gerber's next step: Nested Step & Repeat

**Gent, Belgium - September 30th, 2015** - Ucamco proposes extending the Gerber format to make it more efficient in handling fabrication and assembly panels. The proposed new features will no doubt have other applications. A draft specification is published at [www.ucamco.com](http://www.ucamco.com) for review before it is cast in concrete. Gerber users are invited to comment at [gerber@ucamco.com](mailto:gerber@ucamco.com)

Printed circuit boards are fabricated in panels. The PCB is repeated a number of times on a production panel. The image file representing a panel must represent all instances of the PCB. One way to represent the PCB instances is with a so-called 'flat' file: the objects representing the PCB are simply copied n times in the file, each time at the appropriate place. While this defines the correct image it blows up the file size and slows down processing the image in CAM and on the production equipment. A more efficient way is to store the PCB objects only once, and add an instruction to step and repeat the PCB over the image. The current SR command in Gerber exactly does that.

However, the assemblers, where the bare boards are populated with components, more and more works in panels themselves, often called an 'array', 'biscuit' or 'assembly panel'. The PCB fabricator then ships *arrays* to the assembler, *not single PCBs*. What he repeats in his bare-board production panel are the arrays. The efficient way to represent this image is by a nested step and repeat: the single PCB is stepped into an array, and the array is stepped into the production panel. With a nested step and repeat the PCB data is only once in a file. The problem with the SR in Gerber is that it supports only one level, no nesting. So one has to flatten either the array or the working panel. The resulting big files can become a problem when a small but complex piece of electronics such as a smartphone is fabricated.

To address this issue Ucamco will extend the Gerber language with *nested* step and repeat. Tests performed together with Via Mechanics (former Hitachi Via mechanics) in Japan indeed demonstrated dramatic productivity increases in writing and reading the files as well as in processing them. To introduce this new feature in an easier and safer way Ucamco suggest not extend the capability of the existing SR command but created a *new command, called SN*. The reason is that legacy Gerber readers, that do not yet support nested step and repeat, might, when receiving an SR with nesting, not notice this and produce the wrong image, without warning. A new command is safer. Indeed, the conformance section in the Gerber format specification states: "To prepare for future extensions of the format, Gerber file readers must give a warning when encountering an unknown command, ...". When testing a file with the new SN on the well-respected Gerber viewer GC-Prevue an error message duly appeared.

Another issue is that the step and repeat only allows to repeat object blocks in a regular array. To allow more general repeats Ucamco introduces another new command AB that creates a block aperture that can be flashed in any location and orientation.

New attributes unequivocally identify the fabrication panel, assembly panel, and the single PCB.

Nested step and repeat and block apertures will make Gerber more efficient with panelized data. Further more, the block aperture is a powerful general constructs that will no doubt have many other applications. Together, they are a powerful extension of the Gerber format.

Ucamco want to allow the Gerber user community to review and comment on the new feature before it is cast in concrete and incorporated in the specification. To this end Ucamco publishes a draft specification and sample file for review on its website at [www.ucamco.com/gerber](http://www.ucamco.com/gerber). Your comments and criticism sent to [gerber@ucamco.com](mailto:gerber@ucamco.com) is welcomed.

### **About Via Mechanics (former Hitachi Via Mechanics)**

Via Mechanics is well known as a manufacturer of PCB processing equipment in Japan, More information can be found on the Via mechanics web site <http://en.viamechanics.com>

### **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:



**Phone:** +32 (0)9 216 99 00

**Email:** [info@ucamco.com](mailto:info@ucamco.com)

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