Gent, September 2014

Offener Brief an die Gerber-Benutzer-Community

Bitte benutzen Sie **Extended Gerber** für Ihre gesamten Arbeitsabläufe.
Standard Gerber ist technisch veraltet. Wenn Sie es weiterhin benutzen, setzen Sie Ihr Geschäft und das Ihrer Kunden und Geschäftspartner einem unnötigen Risiko aus, ohne davon irgendwelche Vorteile zu haben.

Ucamco entwickelt und pflegt das Gerber-Format und möchte daher die folgenden wichtigen Informationen zum Standard Gerber-Format mitteilen.

Das Standard Gerber-Format ist jetzt technisch veraltet.

- **Obwohl man es von seinem Namen ableiten könnte, ist Standard Gerber nicht ein definierter Standard für die PCB-Datenübertragung**: Einheiten und Apertur-Definitionen werden nicht von einem erkennbaren Standard bestimmt, sondern befinden sich in einem informellen Dokument, dessen Interpretation zwangsläufig subjektiv ist. Aus diesem Grund können Standard Gerber-Dateien von Maschinen nicht auf standardisierte, zuverlässige Weise gelesen werden.
- Standard Gerber erfordert Apertur-Beschriftung und Kupferbeschichtungen, die beide manuelle Arbeit beim CAM erzeugen und so zu mehr Kosten, Verzögerungen und Risiken beim Leiterplatten-Herstellungsprozess führen.
- Das Standard Gerber unterstützt diese Attribute nicht.

Extended Gerber-Dateien KÖNNEN von der Maschine gelesen werden, benötigen *keine* Beschriftung und *unterstützen* Attribute. Praktisch jede Software kann Extended Gerber lesen und viele neue Implementierungen unterstützen das Standard Gerber-Format nicht mehr. Es gibt nicht einen einzigen guten Grund, Standard Gerber weiterhin zu benutzen. Die Benutzung des Standard anstatt des Extended Gerber-Formats ist ein selbst erzeugter Wettbewerbsnachteil.

Extended Gerber ersetzt das Standard Gerber-Format vollständig**. Extended Gerber ist das aktuelle Gerber-Format. Standard Gerber-Dateien entsprechen daher nicht der Gerber-Spezifikation.**

Ucamcos Position zum Gerber-Format ist daher wie folgt: **Jeder, der das Standard anstatt des vollständig standardisierten Extended Gerber-Formats benutzt, ist für alle Probleme, die durch die Benutzung resultieren können, selbst verantwortlich.**

Vielen Dank.

Karel Tavernier,
Managing Director,
Ucamco

Dieser offene Brief mit weiteren Informationen befindet sich auch als Download auf der Website www.ucamco.com

Gent, May 2014, Karel Tavernier

Standard Gerber

What is Standard Gerber?

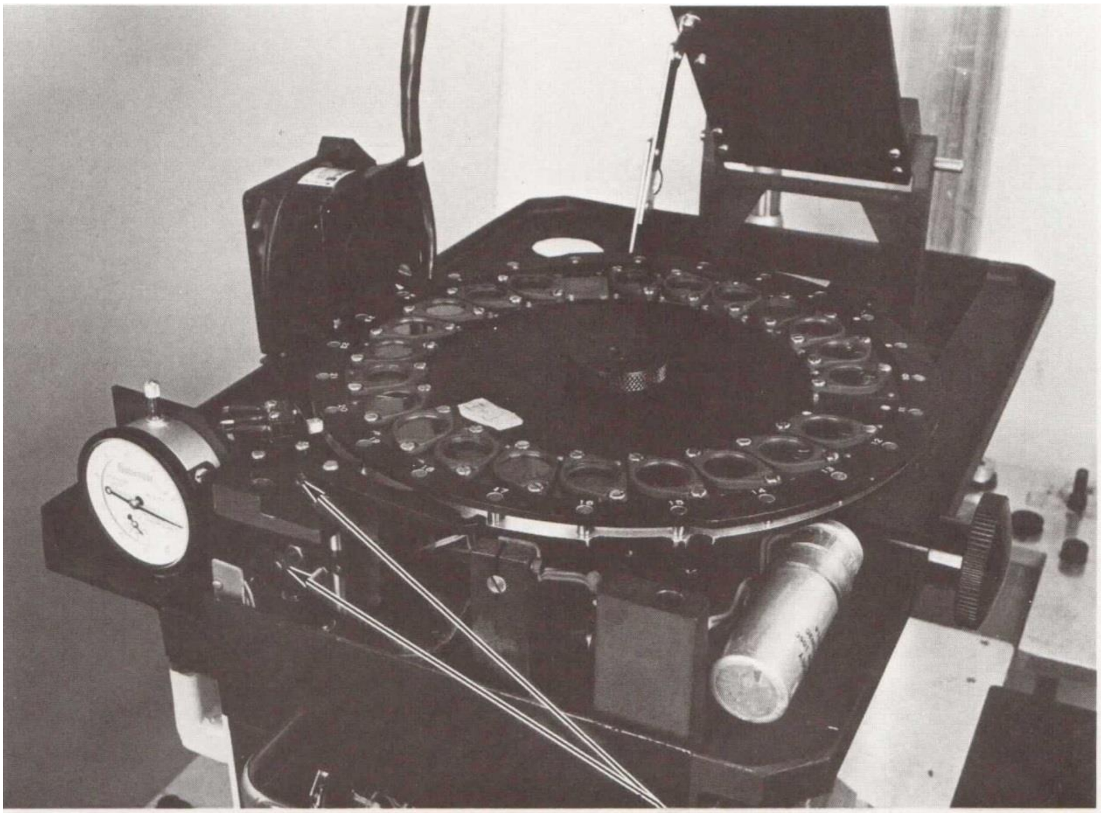
The current Gerber file format is also known as RS-274X or Extended Gerber. There is also a historic format called Standard Gerber or RS-274-D format. This differs from the current Gerber file format (RS-274X), in that it supports neither G36 or G37 codes, nor any parameter codes.

Consequently, Standard Gerber does not allow coordinate format and aperture shapes to be defined, and it lacks the imaging primitives needed to unequivocally transfer information from PCB CAD to CAM, making it incomplete as an image description format.

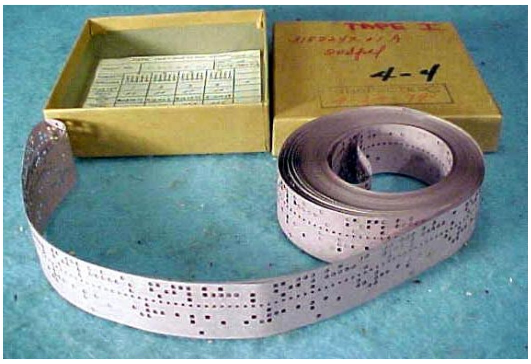
Despite its name, standard Gerber is not a standard image format - it is in fact a standard NC (Numerical Control) format, as explained below.

Origin and purpose of Standard Gerber

In the 1960s and 1970s, images were produced on lithographic film by a vector photoplotter; a precision optical NC machine. Images were produced by beaming light from the plotter's light source onto the film through an aperture on a wheel like that shown in the photograph below. This wheel was rotated to select the appropriate aperture, or it could be substituted by another aperture wheel if additional aperture sizes were needed.



The data for the exposure process was contained in a Standard Gerber file, which was typically recorded onto magnetic or paper tape (see pictures), which was in turn mounted onto the vector photoplotter by the operator.



The operator consulted the accompanying notes, typed the coordinate format on a machine console, mounted the appropriate aperture wheel, changed apertures if necessary, and started the plotter. The Standard Gerber file then drove the plotter through the required movements, controlled the aperture wheel and exposure light, and produced the desired image.

Standard Gerber was so well suited to this task that it became the industry standard.

That was decades ago. Vector photoplotters have not been used since, so Standard Gerber has lost its raison d'être. While it deserves a place of honor in the Computer History Museum, Standard Gerber has no place at all in the 21st century's electronics industry.

Standard Gerber is an NC format, not an image format

From the above, it is clear that Standard Gerber is an NC machine format, and not an image description format. It contains neither the coordinate format definition, so the meaning of coordinate data is undefined, nor aperture definitions, so the meaning of flashes and interpolations is undefined.

Thus if an image is to be defined using Standard Gerber, additional information is essential. This typically comes in the form of a so called “wheel file” consisting of notes in an informal text format, plus drawings that define the more complex apertures. The problem is that there is no standard for this extra information, creating enormous potential for error and misunderstandings. This puts the onus squarely on operators' shoulders to ensure that all of the information is assembled and checked on a workstation – manually and with the help of software tools – in order to be sure that all the necessary image data is present.

As if this were not enough, Standard Gerber also renders the informal description of complex apertures, SMD apertures and areas so difficult that designers give up, and opt instead to paint them. This in turn creates such chaos that there is a very real risk of losing valuable data in both CAD and CAM operations. Thus the CAM engineer must be extremely careful to recover, and piece together, the pads in the design.

All of which renders Standard Gerber totally unsuitable for current CAD to CAM data transfer. This format, from the days of paper tape, punched cards, teletypes and electrical typewriters, offers not one single advantage over Extended Gerber.

So Standard Gerber, despite its name, is not an image definition standard, as it must be supported by a whole lot of extra non-standardized information in order to define an image. That's why Ucamco has defined the new Extended Gerber format. This, unlike its predecessor, ***is*** a standard, as it standardizes the additional data needed, puts it in the file header, and adds some sorely needed extensions.

A fallacy

The following is sometimes said: "The only difference between Standard Gerber and Extended Gerber is that in Extended Gerber the wheel file is embedded in the file. As software was developed to extract data automatically from the wheel files, this is no big deal."

We beg to differ:

- It is *not* the only difference.
- This difference *is* a big deal.

Extended Gerber is a far richer format than Standard Gerber, and has all the constructs necessary for describing a PCB image efficiently. It has regions, positive/negative levels and powerful aperture macros. Planes and anti-pads can be described without painting, and pads are properly described as flashes, ensuring that no data is lost.

As we have said, this is a really big deal. While it is true that a lot of effort was spent on automating the input of accompanying notes, only a fraction of all data sets can in fact be read in automatically because they are often provided in a free format. While this freedom was perfectly adequate for the vector photoplotter operator of old, it flies in the face of standardization and automation, which consequently becomes a less reliable and higher maintenance process. And what happens if the notes arrive in another language – imagine, for example, automating the input of a wheel file in Japanese. Or of its supporting drawings, for which again, there are no format definitions. It becomes clear pretty quickly that it is not possible to fully and reliably automate the transfer of such informal data, so the operator must carefully check all results for errors. This is particularly important if we consider that a lack of standards can also mean lack of clarity about the intentions of the designer, as well as about where responsibility lies in case of errors.

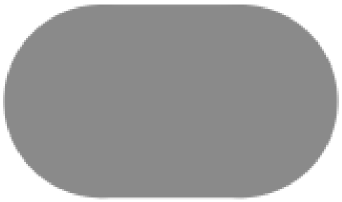
Unfortunately, these are not theoretical issues, as the following real-life example illustrates. A Standard Gerber file came with the following wheel file (abbreviated):

```
// Units: Inches
// Format: DCode, Shape, Width, Height, Hole diameter, Rotation
D10, Round, 0.007000, 0.007000, 0, 0
…
D51, ObLong, 0.024000, 0.070000, 0.000000, 0
…
```

Even though this is one of the better, more explicit wheel files, the manufacturer must nevertheless interpret the term ObLong. A Google search brings up a helpful Wikipedia entry:

- *Oblong, a rectangle with length greater than its width (i.e. not a square). The word is sometimes used of any shape longer than it is wide.*

This is clear, it is a rectangle – the designer than hardly have meant that he would accept any shape. The manufacturer supplies a board with rectangle shape. And then the wrath of the designer descends upon him, because the designer did not intend a rectangle, but an obround:



To be fair to the designer, when you search Google Images the following comes up:

So the designer has a point. But so does the manufacturer. Recriminations fly, discussions about who pays for the scrap become acrimonious, and in the meantime deadlines are missed.

If the job had been sent in Extended Gerber, the apertures would have been defined unequivocally, according to a public standard, and none of this would have happened. In Extended Gerber, the formal, standardized aperture definitions are clear, so reading them in is straightforward, with no need to pore over the results for errors. And as there is a standard, it is clear what was intended, who is responsible in case of a mistake and what to do to avoid the issue in the future.

So yes, this difference is a big deal. It is the difference between using a published standard format and each individual using his own unspecified format. It is the difference between painstaking, error-prone manual work and inspection, and reliable, automatic data transfer.

This is why there is a world of difference between Standard and Extended Gerber, and it's also why Extended Gerber is today's standard for CAD-to-CAM image data transfer.

Information about the Extended Gerber File Format, including the File Format Specification, can be found on the Download page of the Ucamco website.
