

The Gerber Format v2 ("X2") FAQ

*K. Tavernier
October 2015*



FAQ Questions Overview

This FAQ answer the following questions:

- What is new in Gerber X2?
- What are the benefits of Gerber X2?
- Which software supports X2 today?
- Is Gerber X2 compatible with Gerber version 1?
- Is X2 a new format or is it still Gerber?
- Gerber version 1 is simple and human-readable.
What about version 2?
- Is it difficult to implement Gerber version 2?
- Will my software vendor support X2 in the future?
- What do the different names for the Gerber format mean?

FAQ Answers

What is new in Gerber X2?

In version 2 three new commands (TF, TA and TD) attach *attributes* to a Gerber file. Attributes are akin to labels that, when added to a Gerber file, provide meta-information about the image as a whole or individual objects in it. This is done with a flexible yet standardized syntax that is independent of the specific semantics or application.

The most important new attributes are:

- **File function:** Is the file the top solder mask or the bottom copper layer, etc.?
- **Part:** Does the file represent a single PCB, an array, a coupon, etc.?
- **Pad function:** Is the flash an SMD pad, a via pad, or a fiducial, etc.?

Attributes are superfluous when only the image is produced, but are invaluable when PCB data is transferred from design to fabrication. The reason is that the PCB fabricator needs more than the image alone: for example, he needs to know which pads are via pads to fabricate the solder mask. The attributes transfer this information in an unequivocal and standardized manner. *They convey the design intent from CAD to CAM.* This is sometimes rather grandly called “adding intelligence to the image”. Without attributes, the fabricator must reverse engineer the designer's intentions, which can be a time-consuming and error-prone process.

Attributes do not affect the image. A Gerber reader will generate the correct image if it ignores or does not recognize the attributes. Thus attributes can simply be ignored if just the image is needed.

What are the benefits of Gerber X2?

With Gerber version 1, layer and pad function is transferred informally with drawings or notes according to the designer's preferences or conventions. This must be deciphered by the manufacturer which often entails intensive manual CAM work and all its associated delays, costs and – above all – risk of error. With Gerber version 2, PCB design data is transferred in a formally defined standard, with machine-readable layer structure; this means that all the files are automatically placed in their proper order. Furthermore, by clearly identifying pad function, Gerber version 2 enables even greater precision and automation at CAM stage. This is best illustrated by the Gerber X2 intro movie at www.ucamco.com/gerber/intro. This shows why, if you value the secure, reliable transfer of your manufacturing data, you should use version 2.

Even if your Gerber input software does not yet support version two, you can still reap benefits from the attributes if there are questions about pads or the layer structure: you can directly look at the Gerber source – the attributes are pretty clear – or, better, read the data in GC-Prevue, which supports version 2 and have everything displayed unequivocally. Not so good as native support of version 2, but still better than version 1.

Which software supports X2 today?

The following software vendors have implemented or announced X2 support



Numerical Innovations announced support for the new Nested Step & Repeat feature in DFM Now! and FAB 3000.

If your software supports X2 but is not on the list, we apologize. If you are aware of X2 compliant software that is not on this list please contact us at gerber@ucamco.com.

Is Gerber X2 compatible with Gerber version 1?

Yes. Gerber version 2 is both *backward and forward compatible*.

- Backward: A compliant Gerber X2 ready reader will read a Gerber version 1 file perfectly.
- Forward: A compliant Gerber version 1 reader will read a Gerber X2 file and generate the correct image. The legacy reader will of course not take advantage of the new attributes. It may give a warning about unrecognized commands; the warnings can be ignored; if they are sufficiently clear they even reveal useful information such as the function of the file.

Version 2 adds three commands (TF, TA and TD) that carry meta-information about the image. These new commands do not affect the image in any way.

A simple script will convert an X2 file to an X1 file. This script can easily report the meta-information in a report.

The attributes are optional, not mandatory. Therefore valid version 1 file is a valid X2 file.

Is X2 a new format or is it still Gerber?

X2 is still Gerber. It is a concise way of saying "Gerber with attributes". An X2 file contains attributes. And X1 file does not.

Attributes were defined in Gerber version 2. A new version is not a new format. PDF evolved from version 1.1 to version 1.9; it never stopped being the same PDF format. If you support the PDF format, you implicitly support its latest version; if you support just v1.6 then you must make this clear. ODB++ is now at version 8; it's still ODB++, and support for ODB++ implies eventual support of version 8. In the same way, support of Gerber now means support of Gerber version 2, even though the legacy version and its supporting software continue to be compatible to allow sufficient time for the new version to percolate through the industry.

Gerber version 1 is simple and human-readable. What about version 2?

Version 2 remains simple and man-readable. If you understand version 1 you will quickly learn version 2. See for yourself: Below is a small version 2 file with the new commands highlighted. Chances are you will understand most of them without even looking at the specification.

```
G04 Small example Gerber version 2 file*
%FSLAX35Y35*%
%MOMM*%
%TF.FileFunction,Copper,L4,Bot,Signal*%
%TF.Part,Single*%
%TA.AperFunction,Conductor,NotC*%
%ADD10C,0.15000*%
%TA.AperFunction,ViaPad*%
%ADD11C,0.75000*%
%TA.AperFunction,ComponentPad*%
%ADD12C,1.60000*%
%ADD13C,1.70000*%
%SRX1Y1I0.00000J0.00000*%
G75*
%LPD*%
D10*
X7664999Y3689998D02*
X8394995D01*
X8439999Y3734999D01*
X9369999D01*
D11*
X7664999Y3689998D03*
X8359999Y1874998D03*
X9882998Y3650498D03*
D14*
X4602988Y7841488D03*
D15*
X10729976Y2062988D03*
X10983976D03*
X11237976D03*
M02*
```

With this extension the Gerber file maintains its key benefit of being simple and human readable.

Is it difficult to implement Gerber version 2?

No, it is quite straightforward. The neat thing is that the complex part of PCB data exchange – the image data – remains unchanged. Furthermore, the attributes are not mandatory, one can simply choose not to use them or, when they are used, choose to ignore them. It is also OK to implement just, say, the easiest attributes. Of course, as the attributes convey important meta-information, the more complete their implementation, the better.

When outputting a PCB layer the software ‘knows’ which layer it is, so it is quite straightforward to add a line in the header that defines that layer. Implementing the pad attributes is more subtle but still not rocket science.

Input is rather simpler. As previously mentioned, even a version 1 reader will read the image correctly, throwing an annoying warning that can be safely ignored. A correct but minimal implementation will just detect the new commands and suppress the warning. This is all that is required of a software supplier to continue to support Gerber. Hardly a big task. Of course, then no benefits are reaped from the attributes. A full implementation of version 2 takes a little more work but will derive maximum advantage from the wealth of information conveyed by the attributes.

Will my software vendor support X2 in the future?

Well, we cannot say, you must ask your vendor. That said, it's not an onerous task to support Gerber version 2. For output can do nothing if they wish: valid X1 files are also valid X2 files. Of course it's better to add the attributes, so that the readers get the information in a standardized manner, but it's not mandatory. For input, the minimum requirement is to suppress the error messages on the new commands and further continue to ignore them. This is not very onerous. It is of course better to use the information conveyed by the attributes, but they can be ignored and one continues to work in the old way.

That said Gerber version 2 is today's Gerber. Over time, not supporting version 2 is not supporting Gerber, abandoning Gerber. Of course, vendors will a reasonable time to implement version 2. What is a reasonable time? We must leave this to your judgment. The draft specification for version 2 was published in 2013 Q3, the final specification in 2014 Q1. *The clock for reasonable time starts ticking from 2014 Q1.*

What do the different names for the Gerber format mean?

“**The Gerber Format**” describes the current specification. Today, this is Gerber version 2, or X2 for short. There is only one Gerber format. Its proper name is, well, the *Gerber format*.

The terms **Extended Gerber**, **Gerber X** and **RS-274X** are historic names that distinguished newer versions from good old Standard Gerber. They are no longer necessary or helpful and must be abandoned.

Good old **Standard Gerber** is now obsolete and should be consigned to history. The format is revoked. As it no longer conforms to the specification, it can no longer be called Gerber. It should be referred to using its full name (Standard Gerber).

Anyhow, **do not use Standard Gerber any longer!**

Copyright, Intellectual Property and Trade Name

© Copyright Ucamco NV, Gent, Belgium

All rights reserved. This material, information and instructions for use contained herein are the property of Ucamco. The material, information and instructions are provided on an AS IS basis without warranty of any kind. There are no warranties granted or extended by this document. Furthermore Ucamco does not warrant, guarantee or make any representations regarding the use, or the results of the use of the information contained herein. Ucamco shall not be liable for any direct, indirect, consequential or incidental damages arising out of the use or inability to use the information contained herein.

The information contained herein is subject to change without prior notice. Revisions may be issued from time to time to inform about changes and/or additions.

No part of this document may be reproduced, stored in a data base or retrieval system, or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from Ucamco.

This document supersedes all previous versions.

All product names cited are trademarks or registered trademarks of their respective owners.

Ucamco developed the Gerber Format and improves it from time to time with updates. The Gerber Format is Ucamco intellectual property. No derivative versions, modifications or extensions can be made without prior written approval by Ucamco. Developers of Gerber software must make all reasonable efforts to comply with the latest specification.

Gerber Format is an Ucamco trade name. Users of Gerber Format will not rename it, associate it with data that does not conform to the format or modify the graphical interpretation of the format.

Correspondence regarding this publication can be sent to:

gerber@ucamco.com

or

Ucamco NV
Bijenstraat 19,
B-9051 Gent,
Belgium

For more information see www.ucamco.com