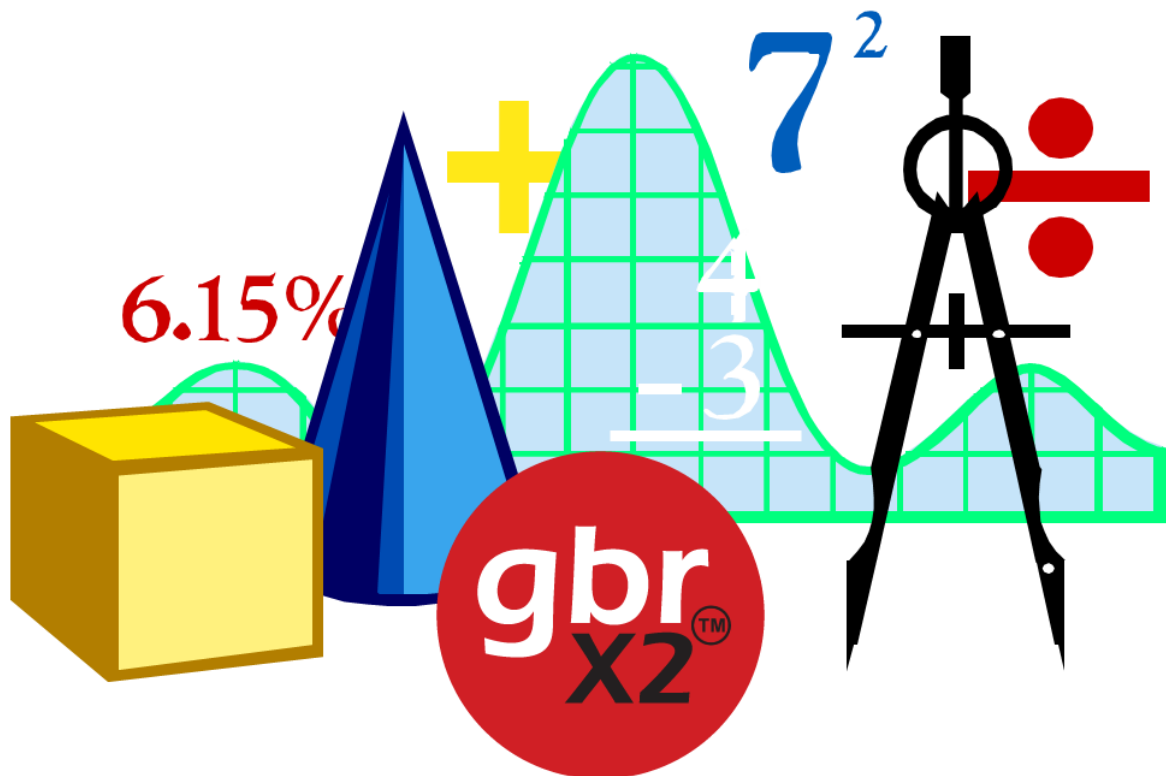


# The Gerber Format v2 ("X2") FAQ

---

*K. Tavernier  
December 2014*





# FAQ Questions Overview

---

This FAQ supplies an answer to the following questions:

- What is new in Gerber X2?
- What are the benefits of Gerber X2?
- 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?
- Which software vendors are supporting X2?
- Will my software vendor support X2 in the future?
- What do the different names for the Gerber format mean?



o

## FAQ Answers

---

### What is new in Gerber X2?

In version 2 three new commands (TF, TA and TD) attach *attributes* to individual features in a Gerber file or to the file as a whole. Attributes are akin to labels that, when added to a Gerber file, provide meta-information to the image in the file. 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](http://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.

## Is Gerber X2 compatible with Gerber version 1?

Yes. The only difference between Gerber version 1 and version 2 is that version 2 adds three commands (TF, TA and TD) that carry meta-information *about* the image. These new do not affect the image in any way. Therefore:

- A Gerber-version-2-ready reader will automatically read and handle the image and the meta-information in a Gerber version 2 PCB job.
- A conforming legacy reader will read a Gerber version 2 PCB job, but as it does not recognize the new commands, it will skip them, continue processing and generate the correct image. The only downside is a superfluous warning about unrecognized commands that users will quickly learn to ignore. If necessary, a simple script will remove the three new commands and thereby output a valid version 1 file. Even with a legacy reader you can still reap benefits from version 2 - see “What are the benefits of Gerber X2?”
- A Gerber version 1 PCB job will be read perfectly by a Gerber-version-2-ready reader. Of course, as there are no attributes their benefits will not be reaped, but the correct image will safely be generated.

## Is X2 a new format or is it still Gerber?

“Gerber version 2” is quite a mouthful, so it is often convenient to call it just “X2”. Thus Gerber version 2 and X2 is the same, as distinguished from Gerber version 1, which could also be called Gerber X1.

X2 is still Gerber: 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.

## Which software vendors are supporting X2?

The following software vendors have implemented or announced X2 support

- Altium - [Altium Designer 15](#)
- DipTrace - [PCB Layout v2.4](#)
- Graphiccode - [GC Prevue v22.3](#)
- Kicad-pcb - [KiCad](#)



- LPKF - [CircuitPro](#)
- Macaos – [PCB Factory](#)
- Pulsonix - [Pulsonix 8.5](#)
- Ucamco - [UcamX 2.1](#) and [Integr8tor 9.1](#)
- ZofzPCB - [3D Gerber Viewer](#)

If your software supports X2 but is not on the list, we apologize. If you are aware of such cases please contact us at [gerber@ucamco.com](mailto:gerber@ucamco.com) and we will rectify this without delay.

## 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. Vendors can, if they wish, do nothing for output: valid X1 files are also valid X2 files. Of course it's better if the vendors tweak their output so that the attributes can be used, but it's not mandatory. For input, the minimum requirement is that the new commands are ignored. Better to use them, of course, but they can be ignored without problems.

That said Gerber version 2 is today's Gerber. Over time, not supporting version 2 is abandoning support Gerber. Of course, vendors will need a reasonable time for implementation. What is a reasonable time to support version 2? This is not for us to say, we leave this to your judgment. We can only mention that the version 2 draft specification was published in 2013 Q3, the final specification in 2104 Q1. The clock for reasonable time starts ticking from then.

## 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. 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](mailto:gerber@ucamco.com)

or

Ucamco NV  
Bijenstraat 19,  
B-9051 Gent,  
Belgium

For more information see [www.ucamco.com](http://www.ucamco.com)