

# Gerber Format X2 FAQ

---

*K. Tavernier  
January 2016*



# FAQ Questions Overview

---

This FAQ answer the following questions:

- What is new in Gerber X2?
- What are the benefits of Gerber X2?
- Is Gerber X2 compatible with Gerber X1?
- Gerber X1 is simple and human-readable. What about X2?
- Do I send out X1 or X2 files?
- Which software supports X2 today?
- Is it difficult to implement Gerber X2?
- Will my software vendor support X2 in the future?
- Is X2 a new format or is it still Gerber?
- What do the different names for the Gerber format mean?

## FAQ Answers

---

### What is new in Gerber X2?

In Gerber X2 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 objects such as flashes or about the image as a whole. The flexible yet standardized syntax 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 needed, but are invaluable when PCB data is transferred from design to fabrication. The PCB fabricator needs more than the image alone: for example, to fabricate the solder mask he needs to know what are vias and what are component pads. 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, 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?

For convenience we refer to previous version of Gerber as X1. In X1 layer and pad function is transferred informally with drawings or notes according the designer's preferences or conventions. Or not transferred at all. This information must be deciphered by the fabricator which often entails intensive manual CAM work and all its associated delays, costs and – above all – risk of error. With Gerber X2, PCB design data is transferred in a formally defined standard, with a machine-readable layer structure; this means that all the files are automatically placed in their proper order. Furthermore, by clearly identifying pad function, Gerber X2 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 X2.

Even if your Gerber input software does not yet support version two, you can still reap benefits from the attributes: you can directly look at the Gerber source – the attributes are pretty clear – or, better, read the data in GC-Prevue, which supports X2 and have everything displayed unequivocally. Not so good as native support of X2, but still better than X1.

## Is Gerber X2 compatible with Gerber X1?

Yes. Gerber X2 is both *backward and forward compatible*.

- Backward: A compliant Gerber X2 ready reader will read a Gerber X1 file perfectly.
- Forward: A compliant Gerber X1 reader will read a Gerber X2 file and generate the correct image. It may give a warning about unrecognized commands; the warnings can safely be ignored; they may even reveal the meta-information such as the function of the file. A legacy X1 reader will of course not take advantage of the new attributes.

The attributes are optional, not mandatory. Therefore valid X1 file is a valid X2 file.

If an application cannot read Gerber X2, it cannot read Gerber properly.

A simple script can easily report the meta-information from an X2 file.

## Gerber X1 is simple and human-readable.

### What about X2?

X2 remains simple and man-readable. If you understand X1 you will quickly learn X2.

See for yourself: Below is a small X2 file with the new commands highlighted. Chances are you will understand most of them without even looking at the specification.

```
G04 Small example Gerber X2 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.

## Do I send out X1 or X2 files?

If your software is capable of sending out X2 files – Gerber with attributes – then always send out X2 files.

If your partner's software is current and takes advantage of the attributes the meta-information is transferred in a standardized, machine-readable manner. This saves time and reduces the risk of misunderstandings, with the resulting costs and wasted time.

If your partner has only legacy X1 software nothing seems gained but nothing lost either. Your partner still reads the image impeccable. Actually, much is gained. Gerber attributes are human readable and provide a standardized human readable way to transfer the file functions. Not machine-read maybe but at least standardized. Or the job can be loaded in GC-Prevue, a free Gerber viewer which does support attributes, to display the attribute information. Alternatively, a simple script can create a report with the file functions and pad functions from an X2 file. (X2 can only lead to problems with legacy X1 software is it *not* compliant with the Gerber specification. If an application cannot read Gerber X2, it cannot read Gerber properly. Then the problem is much bigger than just handling X2 files and the risks serious, both in X1 and X2.)

## Which software supports X2 today?

The following software vendors have implemented or announced X2 support.



If your software supports X2 and you wish to add it to this list please contact us at [gerber@ucamco.com](mailto:gerber@ucamco.com).

## Is it difficult to implement Gerber X2?

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. Even a legacy X1 reader will read the image correctly, maybe throwing a warning that can safely be ignored. A minimal implementation just detects the new commands and suppresses the warnings. Hardly a big task. Of course, then no benefits are reaped from the attributes. A full implementation of X2 takes a more work but will derive maximum advantage from the wealth of information conveyed by the attributes.

## Will my software vendor support X2?

The answer depends on what you exactly mean by ‘support X2’. If it means being able to read and write valid X2 files, then any X1 compliant software supports X2. (If your software would not handle X2 files properly then it is not even compliant with X1 Gerber, a quite serious issue indeed.)

Properly supporting X2 of course means reading and writing attributes, and taking advantage of the rich information they contain. This is quite straightforward to implement but it takes some time. Stepwise implementation: first implement the file functions, the simplest and most valuable attribute, and later the aperture functions. When your software will support attributes is for your software vendor to state.

That said Gerber X2 is today's Gerber. Over time, not supporting X2 is abandoning Gerber support altogether. Of course, vendors need a reasonable time to implement the attributes. What is a reasonable time? We must leave this to your judgment. The draft specification for X2 was published in 2013 Q3, the final specification in 2014 Q1. The clock for reasonable time is ticking from 2014 Q1.

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

X2 is 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 X2. A new version is not a new format. PDF evolved from V1.1 to V1.9; it never stopped being the 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 X2, 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.

## What do the different names for the Gerber format mean?

The **Gerber Format** obviously means what is described the *current* Gerber specification, not in older versions. We are not in the history business. Today, this is Gerber X2. X1 is a convenient name for Gerber files without attributes. X2 is a convenient name for files with attributes.

**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!**

The terms **Extended Gerber**, **Gerber X** and **RS-274X** are historic names that distinguished newer versions from Standard Gerber. As Standard Gerber is revoked and very rarely used these names are no longer necessary or helpful and are better abandoned.

There is only one Gerber format. Its name is, well, remarkably, the *Gerber format*.

# 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)