Ucamco
Former Barco ETS

Gerber Format X2 FAQ

K. Tavernier
March 2017

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?
- 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 intention, 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.

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 a 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*%
%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 can send 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.

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 the easiest attributes. Of course, as the attributes convey important meta-information, the more complete attributes, 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 subtler 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 since 2014 Q1.

Is X2 a new format or is it still Gerber?

X2 is Gerber. X2 is a concise way of saying "Gerber with attributes", or more specifically "Extended Gerber with attributes". An X2 file contains attributes. And X1 file does not. Both are extended Gerber files.

X2 is simply a new version of extended Gerber. It has also been called Gerber version 2. As X2 is compatible this does *not* make it a new format. PDF evolved from version 1.1 to version 1.7; it never stopped being PDF. ODB++ is now at version 8; it's still ODB+.

Supporting PDF means, today, supporting PDF 1.7. If you use legacy software that does only support PDF 1.5, your state you support PDF, but only till version 1.5, not unqualified PDF. Well, supporting Gerber today means supporting extended Gerber files with attributes, or X2 files. Point. That does not mean that you must use all the information conveyed by the attributes; it is OK simply to skip them; OK, but not ideal.

What do the different names for the Gerber format mean?

The **Gerber Format,** unqualified, obviously means what is described the *current* Gerber specification, not in older versions. Attributes are a new feature introduced in the specification in 2014 Q1. As attributes convey a new type of information, not image information, adding them was a qualitative leap in capability; it was called the second extension and people identified the new version as X2; quite handy. Therefore X2 files are Gerber files. So are X1 files.

Standard Gerber is now an obsolete and revoked format. Extended Gerber is not compatible with Standard Gerber. As Standard Gerber no longer conforms to the specification, it can no longer be called Gerber, unqualified. It should be referred to using its full name (Standard Gerber). Anyhow, **do not use Standard Gerber any longer**!

Extended Gerber, **Gerber X** and **RS-274X** are historic names that distinguished newer versions from Standard Gerber. As Standard Gerber is revoked and rarely used these names are no longer very helpful and are maybe better abandoned. If used, it includes both X1 and X2.

In other words:

- Standard Gerber is not Gerber
- Extended Gerber is the same as Gerber
- Gerber X1 is a subset of Extended Gerber
- Gerber X2 is a subset of Extended Gerber

Copyright, Intellectual Property and Trade Name

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