



Gerber Format X2 FAQ

Gerber X2 格式常见问题

*K. Tavernier
September 2016*



FAQ Questions Overview 常见问题概览

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?

常见问题如下：

- Gerber X2 有什么新特性?
- Gerber X2 有什么优点?
- Gerber X2 和 Gerber X1 格式兼容吗?
- Gerber X2 是否和 Gerber X1 一样简单易读?
- 应该选择 X1 还是 X2 格式的资料档案?
- 现在有哪些软件支持 Gerber X2?
- 实施 Gerber X2 格式困难吗?
- 我的软件供应商是否支持 X2?
- X2 是一个全新的格式，还是 Gerber 格式的一种?
- Gerber 格式的不同名称有什么不同的含义?

FAQ Answers 问题解答

What is new in Gerber X2? Gerber X2 有什么新特性?

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.

Gerber X2 引进了三个新指令 (TF, TA 和 TD) , 给 Gerber 文件增加了新的属性。属性类似于标签 , 当添加到 Gerber 文件中 , 可以提供关于物件和图形的元信息。其灵活和标准化的句法可独立于特定的语义或应用。

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.?

其中最重要的三个属性为 :

- **文件属性 :** 此文件是顶部阻焊层还是底部线路层等?
- **组成部分 :** 此文件是单片 , 连片还是阻抗条资料等?
- **盘属性 :** 这是 SMD 盘 , 通孔盘还是定位盘等?

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.

当只用到图像信息时 , 属性信息对于我们是不必要的。但在实际中将 PCB 的设计信息转换成生产信息时 , 属性信息非常重要。PCB 厂商需要知道图像信息以外更多的信息。例如 , 对于阻焊层他们需要知道那些是通孔 , 那些是零件盘。属性功能可以标准的方式传达这些信息。他们将设计者的意图从 CAD 传达到 CAM。我们称之为赋予图像以智慧。如果没有属性信息 , PCB 厂商将不得不花费大量时间还原设计者的用意 , 且容易出错。

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.

属性信息并不影响图像。无论 Gerber 阅读软件是否可以识别属性信息，都可以正确地导入图像信息。在只需要图像的时候可以选择忽略属性信息。

What are the benefits of Gerber X2? Gerber X2 有什么优点?

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.

为了方便，我们将之前的 Gerber 格式版本称为 X1。Gerber X1 中各层和盘的功能属性信息需通过机械图或者说明文档传达设计者的用意或者完全没有附带此类信息。PCB 厂商需借助 CAM 软件人工去提取这类信息，过程费时费力并且有出错的风险。如果使用 Gerber X2 格式，PCB 设计信息，连同机器可读的层结构，以标准化的定义方式传达到 PCB 厂商。这就意味着所有层文件将自动地进行排序。此外，Gerber X2 中清晰定义的盘属性可为 CAM 作业带来更高的精确度和自动化。请在此网址 www.ucamco.com/gerber/intro 观看相关的演示视频。

如果你希望你的生产信息更安全可靠，请使用 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.

即使你的 Gerber 输入软件不支持 X2 格式，你仍然能受益于增加的属性。你可以在 Gerber 源文件查看各项属性。或者你可以通过 GC-Prevue 软件读入资料，GC-Prevue 已支持 Gerber X2 格式，可以清楚显示各项属性。

Is Gerber X2 compatible with Gerber X1? Gerber X2 和 Gerber X1 格式兼容吗?

Yes. Gerber X2 is both *backward and forward compatible*. 是的。Gerber X2 格式是上下兼容的。

- 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.
- 向上兼容：兼容的 Gerber X2 阅读器可完美读入 Gerber X1 文件。
- 向下兼容：兼容的 Gerber X1 阅读器可读入 Gerber X2 文件并生成正确的图像。阅读器软件可能会出现关于无法识别的指令的警告信息，但可以放心忽略，也有可能显示例如像文件属性的元信息。当然老的 Gerber X1 阅读软件是不能识别读取新增的属性信息。

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.

属性信息是可选的而不是强制性的。因此一个有效的 X1 文件可视为一个有效的 X2 文件。如果一款软件不能读入 Gerber X2，那么它也不能正确地处理 Gerber 格式。我们也可以通过简单的脚本从 X2 文件中轻松提取元信息。

Gerber X1 is simple and human-readable.

What about X2? Gerber X2 是否和 Gerber X1 一样简单易读?

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.

与 X1 格式一样，X2 格式简单易读。就如你看到的下面的例子，这是一个带有新指令行的 X2 格式小文件，新指令行以突出显示。很可能你不需要查看规格文件就能理解大部分的内容。

```
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.

增加了扩展指令后，Gerber 文件仍然保持了简单和易于阅读的优点。

Do I send out X1 or X2 files? 应该选择 x1 还是 x2 格式的资料档案？

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

如果你的软件支持附带属性的 Gerber X2 格式，请提供 X2 格式的资料档案。

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.

如果你的 PCB 制造商的软件可以利用属性功能，元信息将以标准化和机器可读的方式进行传递。这节约了时间和减少信息误读的风险，从而避免了浪费时间和成本。

If your partner has only legacy X1 software nothing seems gained but nothing lost either. Your partner still reads the image impeccably. 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.)

如果你的 PCB 制造商的软件只支持 x1 格式，似乎不能从 x2 格式中受益。但你的 PCB 制造商仍然能完美地读取图像。并且 Gerber 属性是可读易懂的，并提供了一种标准化的方式传递文件功能属性。尽管属性信息不能被旧的软件读取但至少是标准化的。你可以通过 GC-Prevue 导入资料。GC-Prevue 是一款免费的 Gerber 阅读器，它支持并可显示属性信息。或者，你可以借助简单的脚本从 x2 格式文件中提取文件功能和盘属性信息，生成报表。（如果读取 x2 文件出现问题，那是因为老的 x1 软件不兼容 Gerber 规格。如果一个应用软件不能读取 x2 文件，意味着它不能正确地读取 Gerber 格式文件。）

Which software supports X2 today? 现在有哪些软件支持 Gerber X2?

The following software vendors have implemented or announced X2 support.

下列软件厂商已经宣布支持或实施 x2 格式。



If your software supports X2 and you wish to add it to this list please contact us at gerber@ucamco.com.

如果你的软件支持 X2 格式而你希望加入上述名单，请通过 gerber@ucamco.com 联系我们。

Is it difficult to implement Gerber X2? 实施 X2 格式困难吗？

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.

不困难，相当简单。首先，在 X2 格式中，PCB 数据交换中最复杂的图像信息部分保持不变。此外，属性信息并不是强制性的。你可以在设计时选择加入它们或者在使用时忽略它们。你也可以选择实施最简单的属性信息。当然，属性信息实施越完整，他们传递的元信息越详细。

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.

当软件输出 PCB 层别时，它已知道其层别功能。所以只需在文件头加入一行定义该层的功能，非常简单。实施盘功能会复杂一点但并不困难。

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.

读入也非常简单。即使旧的 X1 格式阅读器也可以正确地读入 X2 格式文件的图像。也许会出现警告信息但可以放心忽略。简单的做法是发现新的指令后禁止警告信息。当然，这样就无法从属性信息里获益。完全实施 X2 格式需要更多的工作，但可以从层属性信息中获取最大的好处。

Will my software vendor support X2? 我的软件供应商是否支持 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.)

答案取决于你如何理解支持 X2。如果你指的是能够读写有效的 X2 文件，那么所有兼容 X1 格式的软件都支持 X2。（如果你的软件不能正确处理 X2 文件那么意味着它甚至不能与 X1 兼容。这是一个严重的问题。）

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.

一般而言，完全支持 X2 是指能够读写属性信息和利用其中丰富的信息。实施 X2 格式相当简单但需要一点时间。实施 X2 可以按以下不同阶段进行：第一步是添加最简单但非常重要的层文件功能属性，然后是光圈属性。你的软件何时支持属性信息将取决于你的软件厂商。

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.

Gerber X2 是最新的 Gerber。将来，不支持 X2 格式就等于放弃对 Gerber 格式的支持。当然，软件厂商需要一段时间去实施属性这一项新特性。何为合理的时间？这交由你去评判。Gerber X2 的规格草案发布于 2013 年的第三季度，最终的发行版是在 2014 年第一季度。所以所需的时间是从 2014 年第一季度开始计算。

Is X2 a new format or is it still Gerber? Gerber X2 是一种全新的格式还是 Gerber 格式的其中一种？

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

X2 是 Gerber 格式，准确的说法是带有属性信息的 Gerber 格式。X2 格式文件包含属性信息，而 X1 没有。

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.

Gerber X2 版本格式定义了属性信息。一个新的版本不是一种新的格式。PDF 从 V1.1 发展到 V1.9，其仍然是 PDF 格式。如果你说支持 PDF 格式，意味你支持它的

最新版本；如果只支持 V1.6，必须清楚说明。再比如，ODB++目前的最新版本是 8。它仍然是 ODB++ 格式，如果宣称支持 ODB++ 格式意味着最终对版本 8 的支持。同理支持 Gerber 格式意味着对 Gerber X2 版本的支持，而 Gerber 过去的版本和对应的支持软件将继续兼容新版本，以便 PCB 行业有足够时间去了解和应用新版本。

What do the different names for the Gerber format mean? Gerber 格式的不同名称有什么不同的含义？

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.

显然 Gerber 格式指的是当前最新的 Gerber 规格版本，而不是以前的旧版本。今天的 Gerber 格式指的是 X2。为了方便区分，X1 是指不带属性信息的 Gerber 格式，而 X2 是带有属性信息的格式。

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!**

标准 Gerber 格式已经被淘汰，成为历史。此格式已经被废除。因为它不符合 Gerber 的规范，所以不能被称为 Gerber 格式。当提及这个格式时，我们应该用它的全名标准 Gerber。

无论如何，请不要再使用标准 Gerber 格式。

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.

扩展 Gerber，Gerber X 和 RS-274X 这些历史曾用名是为了即将新版本的 Gerber 与标准 Gerber 区分开来。标准 Gerber 已经被淘汰而且极少使用，所以这些名称已经不再需要了。

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

现在只有一种 Gerber 格式。它的名称就是 Gerber 格式。

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