

# Towards Gerber X3

## A proposal to include assembly information in Gerber data Revision 2019.10

---

**Draft for review only**

**Please send your comments to [gerber@ucamco.com](mailto:gerber@ucamco.com)**

The proposal was developed by Karel Tavernier, in extensive discussion with Wim De Greve, Jean-Pierre Charras, Thiadmer Riemersma and Rafal Powierski.

# 1 Preface

---

The purpose of this draft Gerber standard is to transfer the component information held in CAD which is needed in manufacturing to:

- visualize the component placement to check for errors and set up the assembly,
- generate the manufacturing tools, such as the paste stencils and the pick and place machine programs,
- assist in the procurement of the components.

The bare board design is transferred as Gerber files, while component information is currently transferred as separate, non-standardized drawings, pick & place and BOM files. This works fine for the bare board, but not for the components, neither location nor identification. The standard will include component data in the Gerber data. Component location is geometrical data: the centroid, outline, fiducial locations and the footprints; geometric data fits naturally in a Gerber image file. Combining bare board and component data in Gerber files allows a holistic review of the final board.

The intended workflow is that the CAD Gerber output data is read into an assembly CAM system, which analyzes the incoming data, allows for visual inspection and generates the outputs tuned to the specific assembly equipment.

The scope of this specification is intentionally limited to *technical* product information of a single physical PCB. It is not mixed with commercial information such as order quantities - the same physical PCB can be ordered or sold under different commercial terms, and this must not affect the product model.

It is natural to put the component in two new Gerber files: the top and bottom component file. By placing the component data in separate files, full compatibility is maintained: if one does not like the new files, simply ignore them. The new standard is therefore compatible with existing workflows. Legacy software handles the CAD data with the new assembly information without change, of course also without benefiting from it. Great care is taken to minimize the development effort by keeping the existing fabrication outputs in place and sticking to an existing syntax. Benefits:

- Compatible with installed base
- Minimal implementation work.
- Any decent legacy viewer displays the new component layers.

## 2 Component Layers

---

This proposal adds new component layers to a Gerber data set, typically a top and bottom component layer. A set of new attributes dedicated to the component layers specifies positional and non-geometric information about the component, for example the manufacturer part number. Except the .P and .C all attribute and attribute values in this document are new and can only be used in the new component layers.

### 2.1 Identifying the new layer type

The new layer type is identified by a new .FileFunction value

.FileFunction value	Usage
Component, L<p>, (Top Bot)	<p>A component layer.</p> <p>L&lt;p&gt; (p is the copper layer number to which the components described in this file are attached) specifies the physical copper layer number. (Top Bot) indicates if the components are on top, upwards, or on the bottom, downward, of the layer to which they are attached. This syntax caters for embedded components.</p> <p>For jobs without embedded components there is an intentional redundancy. This syntax caters for jobs with embedded components.</p>

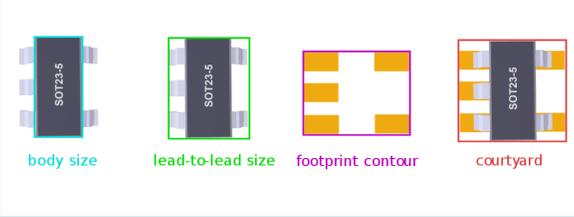
An example, the component layers of a four-layer board.

```
%TF.FileFunction,Component,L1,Top*%  
%TF.FileFunction,Component,L4,Bot*%
```

The coordinate system of the component layers as for all layers: a right-handed system when the PCB is viewed from the top.

## 2.2 The overall component properties

These are identified by new .AperFunction values and object attributes. These can only be used in the component layers.

.AperFunction value	Usage
ComponentMain	<p>This aperture is flashed at the centroid of a component. The flash carries the object attributes with the main characteristics of the component.</p> <p>The following aperture must be used:</p> <pre>%ADD10C,0.300*% (mm) %ADD10C,0.012*% (in)</pre>
<p>ComponentOutline, &lt;type&gt;            &lt;type&gt;= (Body Lead2Lead Footprint Courtyard)</p>	<p>This attribute is used to draw the outline of the component. An outline is a sequence of connected draws and arcs. They are said to connect only if they are defined consecutively, with the second starting where the first one ends. Thus, the order in which they are defined is significant. A contour is closed: the end point of the last draw/arc must coincide with the start point of the first. Outlines cannot self-intersect.</p> <p>Four different types of outlines are defined. See drawing, courtesy Thiadmer Riemersma:</p>  <p>Outlines of different types on the same component are allowed.</p> <p>The following aperture must be used:</p> <pre>%ADD11C,0.100*% (mm) %ADD11C,0.004*% (in)</pre>

Object attributes	Usage
.C, <refdes> <refdes>=<field>	This is an already existing attribute. See section 5.6.15 in the main specification for more information. It identifies the component reference descriptor.
.CRot, <decimal>	The rotation angle of the component.  The rotation angle is consistent with the one for graphics objects. Positive rotation is counterclockwise viewed from the top side, even if the component is on the bottom side. The zero-rotation orientation of a top side component as in IPC-7351. The base orientation of a bottom side component is the one on the top side, mirrored around the X axis.  The rotation is around the flash point.
.CMfr, <field>	Manufacturer
.CMPN, <field>	Manufacturer part number
.CVal, <field>	E.g. 220nF
.CMnt, (TH SMD BGA Other)	Mount type
.CFtp, <field>	Footprint name. It is strongly recommended to comply with the IPC-7351 footprint names and pin numbering for all standard components.
.CPgN, <field>	Package name. It is strongly recommended to comply with the JEDEC JEP95 standard.
.CPgD, <field>	Package description
.CHgt, <decimal>	Height, in the unit of the file.
.CLbN, <field>	Library name
.CLbD, <field>	Library description
.CSup, <SN>, <SPN>, {<SN>, <SPN>}	<SN> is a field with the supplier name. <SPN> is a field with a supplier part name

## 2.3 The pin locations

The pin locations are included in the component layer to unequivocally determine location and rotation of the components. This is done with a dedicated aperture function and a re-use of the .P attribute which already exists for pads on the top and bottom copper layers

.AperFunction value	Usage
<p>ComponentPin</p>	<p>An aperture whose flash point indicates the location of the component pins (leads). The .P object attribute must be attached to each flash to identify the reference descriptor and pin.</p> <p>For the key pin, typically pin "1" or "A1", the following diamond shape aperture must be used:            %ADD12P,0.360X4X0.0*% (mm)            %ADD12P,0.017X4X0.0*% (in)</p> <p>The key pin is then visible in the image.</p> <p>For all other pins the following zero size aperture must be used:            %ADD13C,0*%... (both mm and in)</p> <p>These pins are not visible which avoids cluttering the image.</p>
Object attribute	Usage
<p>.P,&lt;refdes&gt;,&lt;number&gt;[,&lt;function&gt;]            ]            &lt;refdes&gt;=&lt;field&gt;            &lt;number&gt;=&lt;field&gt;            &lt;function&gt;=&lt;field&gt;</p>	<p>This is an already existing attribute. See section 5.6.14 in the main specification for more information. It identifies the pin reference descriptor, pin number or name, and optionally the pin function.</p> <p>It is strongly recommended to include <i>both</i> pin number and pin function. The pin function is less prone to error than the pin number, and the redundancy provides extra security This is especially important for diodes and transistors.</p>

# 3 Annotated Example File

This is an example of a top component layer. It assumes a general knowledge of that Gerber format as only the new, component related, commands are annotated.

Commands	Annotation
%TF.GenerationSoftware,KiCad,Pcbnew,(5.99.0-190-g0fd48dd4f-dirty)*%	
%TF.CreationDate,2019-10-02T18:52:55+02:00*%	
%TF.ProjectId,kit-dev-coldfire-xilinx_5213,6b69742d-6465-4762-9d63-6f6c64666972,2*%	
%TF.SameCoordinates,PX3e22018PY8d89728*%	
%TF.FileFunction,Component,L1,Top*%	<b>This file is the top component layer</b>
%TF.FilePolarity,Positive*%	
%FSLAX46Y46*%	
%MOMM*%	
%LPD*%	
G04 Aperture begin list*	
%TA.AperFunction,ComponentMain*%	
%ADD10C,0.3*%	<b>The aperture for the flash with the main component information</b>
%TA.AperFunction,ComponentOutline,Courtyard*%	
%ADD11C,0.1*%	<b>The aperture to draw the outline</b>
%TA.AperFunction,ComponentPin*%	
%ADD12P,0.36X4X0.0*%	<b>The aperture to flash the pin 1 location</b>
%ADD13C,0*%	<b>The aperture for the other pin locations</b>
%TD*%	
G04 Aperture end list*	
G04 Begin component info*	
D10*	<b>Select main component aperture</b>
%TO.C,R301*%	<b>Attach reference descriptor R301</b>
%TO.CFtp,R_0805_2012Metric*%	<b>Attach footprint</b>
%TO.CVal,4K7*%	<b>Attach value</b>
%TO.CMnt,SMD*%	<b>Attach mount type</b>
%TO.CRot,-90*%	<b>Attach rotation</b>
X218250000Y-73000000D03*	<b>Flash at reference point</b>
D11*	<b>Select outline aperture</b>

Commands	Annotation
X219250000Y-71310000D02*	<b>Draw outline</b>
X217250000Y-71310000D01*	<b>Draw outline</b>
X217250000Y-74690000D01*	<b>Draw outline</b>
X219250000Y-74690000D01*	<b>Draw outline</b>
X219250000Y-71310000D01*	<b>Draw outline</b>
D12*	<b>Select key pin aperture</b>
%TO.P,R301,1*%	<b>Attach ref. desc and pin number</b>
X218250000Y-72045000D03*	<b>Flash at key pin location</b>
D13*	<b>Select subsequent pin aperture</b>
%TO.P,R301,2*%	<b>Attach pin 2 ident</b>
X218250000Y-73955000D03*	<b>Flash at key pin location</b>
%TD*%	<b>Clear attributes before next component</b>
G04 Next component*	
...	

# 4 Revisions

---

**Rev 2019.10** . Added "ComponentPin" attribute, which specifies the locations of all pins. Added attributes for library, library name, suppliers and package description as suggested by Wim De Greve. Apertures and D codes now have mandatory values. Added annotated example Text improvements a.o. suggested by Radim Halír, Rik Breemeersch, Rafal Powierski and Nicholas Meeker. This is the first published draft.

**Rev 2019.09.23** Text improvements. Added the .CFtp (footprint) object attribute. Clarified usage of the pre-existing .P (pad) attribute. Circulated privately.

**Rev 2019.09** Convergence after extensive discussion among Wim De Greve, Jean-Pierre Charras, Thiadmer Riemersma, Rafal Powierski. Circulated privately.

**Rev 2019.07** Processed input from Thiadmer Riemersma: split value and MPN, only centroid as reference point, clarify that outline includes pins, add TH|SMD|BGA field, etc. Circulated privately.

**Rev 2019.01** Simplified. Put all information in top and bottom component layers. Removed BOM from Gerber Job File and put subset in the component layers. Circulated privately.

**Rev 2018.12** Prototype draft specification. Circulated privately.

# 5 Copyright

---

© Copyright Ucamco NV, Gent, Belgium

All rights reserved. No part of this document or its content may be re-distributed, reproduced or published, modified or not, in any form or in any way, electronically, mechanically, by print or any other means without prior written permission from Ucamco.

The information contained herein is subject to change without prior notice. Revisions may be issued from time to time. This document supersedes all previous versions. Users of the Gerber Format<sup>®</sup>, especially software developers, must consult [www.ucamco.com](http://www.ucamco.com) to determine whether any changes have been made.

Ucamco developed the Gerber Format<sup>®</sup>. The Gerber Format<sup>®</sup>, this document and all intellectual property contained in it are solely owned by Ucamco. Gerber Format<sup>®</sup> is a Ucamco registered trademark. By publishing this document Ucamco does not grant a license to the intellectual property contained in it. Ucamco encourages users to apply for a license to develop Gerber Format<sup>®</sup> based software.

By using this document, developing software interfaces based on this format or using the name Gerber Format<sup>®</sup>, users agree not to (i) rename the Gerber Format<sup>®</sup>; (ii) associate the Gerber Format<sup>®</sup> with data that does not conform to the Gerber file format specification; (iii) develop derivative versions, modifications or extensions without prior written approval by Ucamco; (iv) make alternative interpretations of the data; (v) communicate that the Gerber Format<sup>®</sup> is not owned by Ucamco or owned by anyone other than Ucamco. Developers of software interfaces based on this format specification commit to make all reasonable efforts to comply with the latest specification.

The material, information and instructions are provided AS IS without warranty of any kind. There are no warranties granted or extended by this document. 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. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Ucamco. All product names cited are trademarks or registered trademarks of their respective owners.