

# A proposal to include the CAD netlist in Gerber

## Rev 2016.07

---

**Draft for review only**

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

The proposal was developed by Karel Tavernier

# 1 Preface

---

The new X2 capabilities in this proposal allows to include the CAD netlist in Gerber fabrication data. New object attributes allow to

- attach a row in the CAD netlist (the component reference designator, pin number and net name) to the component pads in the Gerber file.
- attaches the netlist name to any conducting object.
- attach the component name to any object. For example, this identifies to which component the legend objects belong.

The design goal of the Gerber netlist is to facilitate upfront communication, between the different parties involved in design, assembly and automation. For example:

- The CAD netlist attached to pads defines the position and orientation of external components which is needed in assembly and for complete board display.
- The netlist names allow lightweight viewers to display netlists without the need to develop algorithms to compute connectivity.
- The netlist provides a powerful redundancy check during CAM input, whose presence reduces the probability of image errors to virtually zero.

Great care has been taken to make this complete, but simple, straightforward to implement and to adopt. No needless bells and whistles. Simplicity remains the Gerber hallmark.

Compatibility is maintained. The new attributes do not affect the image. When processing a Gerber file you can simply skip the attributes if you do not need the CAD netlist information. Legacy applications and files remain valid: when generating a Gerber file it is not mandatory to add these attributes; it is not mandatory to spend the development effort to include them if they bring no benefit in your application; however, for fabrication data they are extremely useful.

There is overlap with the venerable IPC-D-356 format but the design goal is completely different. The design goal of 356 is data input for bare board electrical test. The design goal of the Gerber netlist is to facilitate upfront communication, before proper manufacturing, and not drive testers. Indeed, it does not contain the adjacency list, fiducials, and other information needed for electrical test. It is this limitation that is the benefit of Gerber netlist: it is easier to understand and implement than 356. As Goethe famously said: *In der Beschränkung zeigt sich erst der Meister*.

We explicitly thank Jean-Pierre Charras for his insightful comments which contributed so much to this draft.

Ucamco publishes this draft specification to allow the Gerber user community to review and comment on the new feature before it is cast in concrete. Please send your comments and criticism to [gerber@ucamco.com](mailto:gerber@ucamco.com).

Karel Tavernier,  
Managing Director,  
Ucamco

## 2 Object Attributes

---

There are three types of attributes by the *domain* they attach to:

- ❑ File attributes attaching metadata to the file as a whole.
- ❑ Aperture attributes attaching metadata to an aperture or a region.
- ❑ Object attributes attaching metadata to graphics objects

During the processing of a Gerber file an *attribute dictionary* is maintained. Dictionary entries consist of the attribute name its domain and its value. The attribute name is the key of the entry; it must consequently be unique.

The current aperture dictionary is defined after each command in the file according to the following rules:

- ❑ Initially the attribute dictionary is empty
- ❑ File attributes are added or updated with the TF command
- ❑ Aperture attributes are added or updated with the TA command
- ❑ Object attributes are added or updated with the TO command
- ❑ Aperture attributes are deleted from it with the TD command

When an aperture or a graphics object is created all attributes with the proper domain in the dictionary at that moment are attached to it. They remain fixed and cannot be changed.

An *object attribute* is attached to graphics objects. When a D01, D03 or G36/G37 creates an object all object attributes in the attribute dictionary at that moment are attached to it. They remain fixed and cannot be changed.

The TO command adds an aperture attribute into the attributes dictionary. It has the same syntax as the TF or TAc command:

**<TO command> = %TO<AttributeName>[,<AttributeValue>]\*%**



**<AttributeValue> = <Field>{,<Field>}**

The attribute name must follow the naming syntax in section **Error! Reference source not found.** This name cannot be used for any other attribute. The fields composing the attribute value must follow the string syntax in section **Error! Reference source not found.** with the additional restriction: a field must not contain commas.

The value of an aperture attribute can be modified by a new TO command with the same attribute name.



### Example:

```
%TO.N, Clk13*%
```

This command adds the .N attribute with value Clk12 to the dictionary.

The TD command deletes an attribute from the attributes dictionary. Note that the attribute remains attached to apertures and objects to which it was attached before it was deleted.

**<TD command> = %TD[<AttributeName>]\*%**

The <AttributeName> is the name of the attribute to delete. If omitted, all attributes are deleted.

## 3 Standard Object Attributes

### 3.1 CAD Netlist (.CN)

The .CN object attribute attaches a row in the CAD netlist (the component reference designator, pin number and net name) to the component pads in the Gerber file. It has the following syntax:

**<Attribute .CN> = .CN,<component>,<pin>,<netname>**

.CN fields	Usage
<component>	The component reference descriptor. It can take any value conforming to the field syntax.
<pin>	The pin number. It can take any value conforming to the field syntax. This means it can actually be a name, not only a number.
<netname>	The CAD net name. It can take any value conforming to the field syntax.



#### Example:

D13*	Select aperture for the pads for R5
%TO.CN,R5,1,Clk3*%	
X5600000Y1200000D03*	The pad for R5, pin 1, net Clk3
%TO.CN,R5,2,3V*%	
X5600000Y1202500D03*	The pad for R5, pin 2, net 3V

The .CN attribute can be attached to *flashed pads only*. It can therefore not be used on painted pads.

A single component pad can consist of different flashes. Each flash then carries the same reference descriptor, pin number and net. These pads are deemed electrically connected.

It is technically possible to create a single object representing multiple pads. For example, a single macro aperture can describe a complete component footprint. This is possible, but *not allowed*. (It would be a neat way to make life miserable for CAM engineers though.)

The .CN attribute can only be used in copper layers. Solder mask openings and paste flashes derive their component information from the underlying copper pads. Only copper layers with components carry the .CN attribute. Typically, these are the outer layers. However, embedded or etched components can be on inner layers and then these inner can carry the .CN attribute.

For convenience, single pad nets can be given the name N/C instead of unique names. Each N/C net represents a single pad net, not connected to any other pad. The net name N/C is reserved for that purpose. Except the reserved N/C all net names must be unique.

If the .CN attribute is present then the *complete* CAD netlist must be present, including all edge connectors test points and etched components. In other words, all the end points of the nets

must be included, all the pads, *not only* the pads of the physical components that are part of the BOM.

### 3.1.1 Etched components

Etched components are embedded inductors, transformers and capacitors which are etched into the PCB copper. The following illustration shows two etched inductors.



They are identified by the value of the .AperFunction attribute attached to the aperture used to create them. See <Cross-reference>.

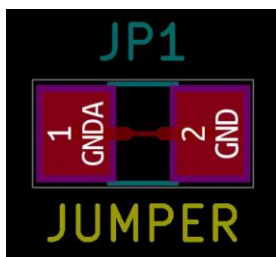
For the CAD netlist these are components like others: the net names are different on both sides. (However, for bare-board electrical test they are just conducting copper: they connect the net on both sides.)



#### Example:

Here is an example of how etched components interact with the CAD netlist. The example is provided by Jean-Pierre Charras.

The etched component in the following illustration is a fuse connecting the nets GND and GND.



This results in the following Gerber file. The corresponding attributes are highlighted.

```
TF.GenerationSoftware,KiCad,Pcbnew,(2016-07-19 BZR 6986)-product*%
%TF.CreationDate,2016-07-20T09:39:42+02:00*%
%TF.ProjectId,microwave,6D6963726F776176652E6B696361645F,rev?*%
%TF.FileFunction,Copper,L1,Top,Signal*%
%FSLAX46Y46*%
G04 Gerber Fmt 4.6, Leading zero omitted, Abs format (unit mm)*
G04 Created by KiCad (PCBNEW (2016-07-19 BZR 6986)-product) date 07/20/16
09:39:42*
%MOMM*%
%LPD*%
```

```

G01*
G04 APERTURE LIST*
%ADD10C,0.100000*%
%ADD11C,0.400000*%
%ADD12C,0.150000*%
%ADD13R,2.000000X2.500000*%
G04 APERTURE END LIST*
D10*
D11*
G04 etched component To be Defined
X161000000Y-83000000D02*
X162500000Y-83000000D01*
D12*
G04 etched component To be Defined
X162500000Y-83000000D02*
X163600000Y-83000000D01*
D11*
G04 etched component To be Defined
X163600000Y-83000000D02*
X165000000Y-83000000D01*
D13*
%TO.CN,JP1,1,GNDA*%
X161000000Y-83000000D03*
%TO.CN,JP1,2,GND*%
X165000000Y-83000000D03*
M02*

```

### 3.1.2 Benefits of the Gerber CAD netlist.

For the assembly process the location and orientation of each component must be known. This can easily be extracted from the Gerber file.

The netlist and component names facilitate the communication between the parties involved in design and fabrication. Viewers show more complete PCB information.

More importantly the netlist information dramatically increases the security of the design to fabrication data transfer. If the image CAM reads from a Gerber file is significantly different than the image intended by CAD, due to bugs, operator errors or transmission errors, the inevitable result is scrap. Such a difference in image results in a difference in netlist, which can be detected with the CAD netlist. The CAD netlist therefore provides a very powerful redundancy check against image errors. To be precise, the following assert must be valid:

- Replace all N/C's with unique names
- Flashes with the same reference descriptor and pad are deemed connected
- Etched components are removed before connection is calculated
- Assert 1: pads with the same net name must be connected
- Assert 2: pads with different net names must be isolated.





## 3.2 Net (.N)

The .N object attribute attaches the CAD netlist name to any conducting object. The attribute can be attached to objects on any copper layer.

**<Attribute .N> = .N<netname>**

.N field	Usage
<netname>	The CAD net name. It can take any value conforming to the field syntax. It must be the same names as used in the .CN attribute, if present

Note that a single object can be split in several parts by clear (LPC) objects. Each part can belong to a different net. An object can therefore belong to different nets. This situation can of course be avoided by using single net objects only – an image consisting of several disconnected parts always be created as multiple simple objects, with for example regions.



### Example:

D21*	Select aperture 13, used for tracks
%TO.N, Clk3*%	Select net Clk3
X5600000Y1200000D02*	Move to the start of a track
X5600000Y1202500D01*	Draw the tracks
X5605000Y1205000D01*	
X5605000Y1220000D01*	
...	

Net Clk13 is attached to all tracks.

The .N attribute is especially important for quick visualization of nets.

### 3.3 Component (.C)

The .C object attribute attaches the component reference descriptor to an object. The attribute can be attached to objects on legends, assembly drawing and in fact in any extra layer were objects belong to a component as a whole and not individual pads.

**<Attribute .C> = .C<component>**

.C field	Usage
<component>	The component reference descriptor. It can take any value conforming to the field syntax. It must be the same names as used in the .CN attribute, if present

Note that a single object can be split in several parts by clear (LPC) objects. Each part can belong to a different net. An object can therefore belong to different nets. This situation can of course be avoided by using single net objects only – an image consisting of several disconnected parts always be created as multiple simple objects, with for example regions.



#### Example in a legend layer:

D21*	Select aperture 13, used for component symbols
%TO.C,R2*%	Select reference descriptor R2
X5600000Y1200000D02*	Move to the start of the symbol
X5600000Y1202500D01*	Draw the symbol
X5605000Y1205000D01*	
X5605000Y1220000D01*	
...	

The attribute .C,R2 is attached to all tracks drawing the resistor symbol.

## 4 Revisions

---

Rev 2016.07 Initially created by Karel Tavernier on a long and boring flight. Jean-Pierre Charras provided very important insight and input to this draft. Remco Poelstra suggested to include the information carried by the .C attribute.

# 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®, 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®. The Gerber Format®, this document and all intellectual property contained in it are solely owned by Ucamco. Gerber Format® is a Ucamco registered trade mark. 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® based software.

By using this document, developing software interfaces based on this format or using the name Gerber Format®, users agree not to (i) rename the Gerber Format®; (ii) associate the Gerber Format® 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® 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.