

Karel Tavernier, June 17th, 2011.

## Gerber Format Application Note Painting Considered Harmful

### **Summary**

- Always use flashed pads. With special apertures you can create any shape without painting. Never paint pads.
- Always define areas with the G36/G37 outline (polygon) fill. Make holes by adding a clear layer with %LPC. Areas of any shape can be created without painting. Never use painting to fill areas.
- Always define tracks by drawing with a single aperture. Never paint tracks.

Painting offers no benefit whatsoever, but it has significant disadvantages. Although painting can produce the correct image, painted files are very hard to use by the manufacturer. It increases the risk of errors, scrap, delays and extra costs.

### **Introduction**

This note supposes a basic understanding of the Gerber format.

There are three types of painting.

- 1) Painted pads
- 2) Painted areas
- 3) Painted tracks

Each type is discussed in a separate section.

Painting is also called stoking, paint-fill, stroke-fill, vector fill. We will consistently use the term painting.

### **Painting pads is harmful**

Apertures of any shape can be defined with the %AD and %AM parameters in RS-274X Gerber format. An aperture can be defined for any pad shape. The

aperture is then flashed at each pad instance. This is the simplest and only proper way to generate pads.

However, sometimes pads are not flashed, but painted. In each pad instance, the proper shape is constructed with a number of draws with a circular or rectangular draws.

This way the desired image can indeed be created. However, painting pad is harmful.

First of all painting pads makes the image construction more complex. Painting creates a number of objects for each pad instead of a single pad. The file becomes bigger. More importantly, such a file is slow and cumbersome to work with in a CAD/CAM system.

But there is a deeper reason why painting pads are harmful. The PCB manufacturer not only needs the correct image, he also needs to know where the pads are. He needs to know where the pads are to test the board electrically. He needs pads to make the solder mask right. The assembler needs pads to apply paste. The way to tell the user of the Gerber file that something is a pad is to flash it and to use flashes exclusively for pads. Then life is simple. A flash is a pad, and a pad is a flash. Painted pads completely mess up this simple rule. To know where the pads are, the manufacture has to scan the whole image, *guess* where the pads are and replace all painted pads by a proper flashed pad before he can even start to work with the file. This is a painful, slow and error-prone process step. It is solely there because the flashes were painted.

Of course, the designer could think this is not his problem. Not so. He sets the manufacturer up to fail. An error in the manufactured board will result in missed deadlines, angry customers and costly repairs. This is very damaging whatever compensation the poor PCB manufacturer pays.

### ***Painting areas is harmful***

Many PCB layers contain not only tracks and pads but also copper area's. The proper way to construct area is with the G36/G37 commands, creating what is called in Polyline in Gerber parlance. With these you describe the outline or contour of the area. This is concise and clear. Any outline shape can be defined. Sometimes the areas are not simple, but they have holes in them, e.g. antipads. The antipads are then clearly and explicitly defined in the file, and the manufactures knows where they are. The best way to make these

holes is to use the %LN parameter and make a negative layer with all the holes. Holes can also be made without a negative layer, using cut-ins. (There are examples of cut-ins in the RS- 274X Gerber format specification.) This is somewhat more complex, and the antipads are only indirectly defined.

Sometimes these areas are defined by covering – painting - them with a large number of draws, much like a child would make an area black with a pencil. To keep the holes open the painting goes around them. To construct the border or outline of an area precisely, a line with a small aperture is drawn along the outline.

The problems are similar as with painted pads. The files become very large, and the data very complex. The manufacturer needs to know what are tracks, e.g. to perform track width compensation or edits. This is simple if all tracks are represented by draws. However, when areas are filled with a humongous number of draws, it is painful to find out which draws are tracks, and which not. The best way for the manufacturer is to replace the painted area by a properly constructed one. However, this is complicated and error-prone manual work.

Often there are embedded pads in a copper area. Embedded pads do not affect the image, but they provide vital information to the manufacturer: the manufacturer needs to know where the pads – or component feet – are inside the area. It becomes a complete nightmare of both the pads and the copper areas are painted.

### ***Painting tracks is harmful***

The proper way to construct a track is to draw it with an aperture with the width of the track. Sometimes tracks are painted: they are made from different parallel draws with a smaller aperture instead of a single draw with the correct aperture.

Painted tracks are even more work to recover from and restore a proper job. Painted tracks are beyond contempt.

### ***Why is painting used?***

Painting is a relic from the vector photoplotters used in the 1970's and 1980's. The shapes these devices could image were very limited. The only way to produce SMD pads or areas was to painting them. The computers controlling them were very feeble and could not perform the painting themselves but had

to be fed files where the painting was already performed. Painting was then a necessary evil.

Vector plotters are now utterly obsolete. They are no longer in use since 25 years.

The RS-274X Gerber format allows to easily define objects directly and concisely, without painting.

Today, there is no valid reason to use painting. There are many valid reasons not to use painting.

Painting is as obsolete, as obsolete as papertape or ½" magtape.

Painting should never be used. Never.

With kind regards,

Karel Tavernier,  
Managing Director,  
Ucamco

© Copyright 2011 Ucamco NV, Gent, Belgium. All rights reserved.

All material, information, software and instructions for use contained herein are the property of Ucamco NV.

Except as permitted by the applicable license, no part of this document may be displayed, copied, stored in a data base or retrieval system, reproduced, distributed, downloaded, posted or transmitted in any form or by any means, including, but not limited to, electronic, mechanical, photocopying, recording, or otherwise, translated in any other language, published or included in software. Except as permitted by the applicable license, no right is granted to use the know-how implied and/or embedded into the document.

The material, information, software and instructions for use contained herein are subject to change without notice.

The material, information, software and instructions for use contained herein are provided on an AS IS basis. Nor Ucamco NV nor its licensors provide any warranty of any kind and expressly disclaim any and all warranties of merchantability, fitness for a particular purpose and non-infringement of third party intellectual property rights. Ucamco NV assumes no liability for any errors or inaccuracies.

Ucamco NV shall not be liable for any direct, indirect, consequential or incidental damages arising out of the use or the inability to use of material, information, software and instructions for use contained herein.