Ucamco
Former Barco ETS

Karel Tavernier, June 17th, 2011.

Gerber File Format Application Note
Painting Considered Harmful

Summary

- Always use flashed pads. Any shape aperture can create with %AM. 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 in this way. Never use painting to fill areas.
- Always define tracks by a single drawing or arc. Never paint tracks.

Painting offers no benefit whatsoever. It has significant disadvantages. Although painting can produce the correct image, it is confusing for the manufacturer. It increases the risk of errors, scrap, delays and extra costs.

Introduction

This note supposes a basic understanding of the Gerber file 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 the RS-274X Gerber file 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.

Ucamco - Bijenstraat 19 - 9051 Gent - Belgium
t +32 (0)9 216 99 00 f +32 (0)9 216 99 12 e info@ucamco.com

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. 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 %LP 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 file format

Ucamco - Bijenstraat 19 - 9051 Gent - Belgium
t +32 (0)9 216 99 00 f +32 (0)9 216 99 12 e info@ucamco.com

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. Te keeps 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 manufactures 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 manufactures 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 manufactures 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.

Ucamco - Bijenstraat 19 - 9051 Gent - Belgium
t **+32 (0)9 216 99 00** **f** **+32 (0)9 216 99 12** **e** **info@ucamco.com**

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

The RS-274X Gerber file 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

Ucamco - Bijenstraat 19 - 9051 Gent - Belgium

t +32 (0)9 216 99 00 f +32 (0)9 216 99 12 e info@ucamco.com

Ucamco developed the Gerber file format and improves it from time to time with updates. The Gerber file 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 file format will not rename it, associate it with data that does not conform to the format or modify the graphical interpretation of the format.

Ucamco - Bijenstraat 19 - 9051 Gent - Belgium

t **+32 (0)9 216 99 00** **f** **+32 (0)9 216 99 12** **e** **info@ucamco.com**