Painting Pads:

The Scourge of CAD-to-CAM Communication

by Karel Tavernier
UCAMCO

SUMMARY: Often called the "backbone of the electronics industry," the Gerber format is the easiest and most reliable image data transfer format available to PCB designers and engineers. Unfortunately, this format is often used incorrectly, including one practice that PCB designers should cease immediately: painting pads and areas.

Over decades, Gerber has evolved into the bare board industry's CAD-to-CAM data transfer standard, capable of describing a PCB image to within an astoundingly accurate 0.1 nm, in a clear-cut language that is fast and easy to use. Designs described in the RS-274X Gerber format are hassle-free, reliable and accurate, and can be implemented quickly, easily and cost-effectively.

This article will help the PCB design community get the very best out of this clean but sometimes poorly used format. We will show how old, outdated habits are creating problems for their manufacturing partners as well as compromising the quality of the final products.

The CAM Process

Before going on to discuss the problems, it is worth taking a little time to explore what happens, and what does not happen, to Gerber data once it enters the PCB manufacturer's systems.

Many users of PCB CAD systems believe that the data files they send to their PCB manufacturers will drive the fabricators' production machines; they think that their Gerber image files are production tools that will be used directly on the PCB manufacturers' photoplotter, the Excellon drill files will go straight onto the manufacturers' drilling machines, and their IPC-356 electrical test netlist information will go right into electrical test machines. They don't.

Manufacturers never use the Gerber or Excellon files directly on their equipment—never. There are numerous reasons for this, but the simplest is panelisation: The designer's data describes a single PCB. However, PCBs are always manufactured on panels with borders for plating, test coupons, and so on. Furthermore, to minimize costs, the manufacturer will produce several PCBs at a time on a single, larger panel. Another reason is that manufacturing processes inevitably introduce deviations: For exam-
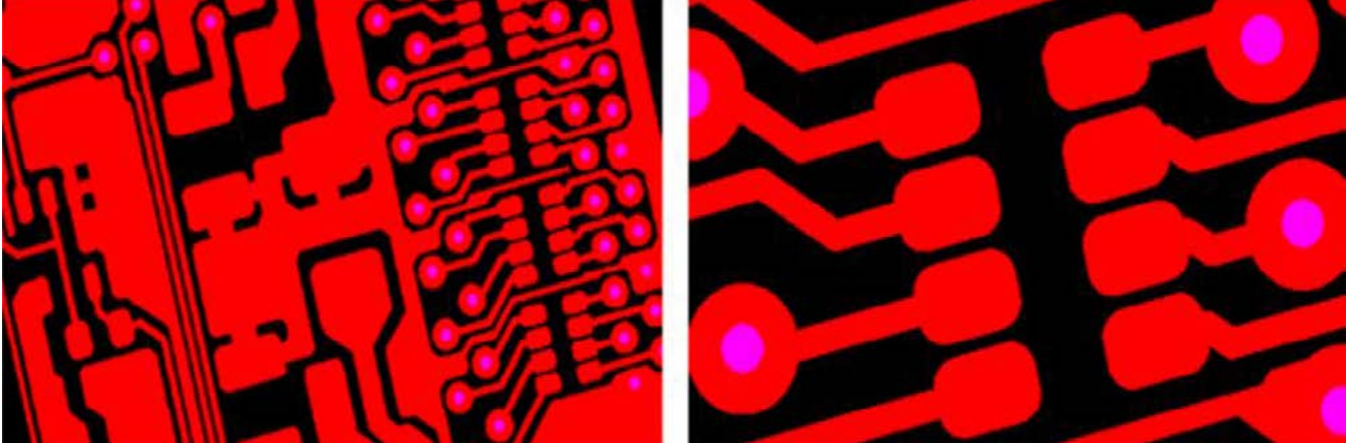
PAINTING PADS *continues*

Figure 1: Painted features are visually clean...

ple, layers are distorted during lamination, and etching reduces line widths. The manufacturer must therefore modify the data to pre-compensate for these deviations.

So incoming data is always read into the manufacturer's CAM system, verified and transformed into valid production tools before it ever gets anywhere close to the PCB manufacturing line. In other words, the manufacturer loads the designer's data into his CAM system and reconstructs the PCB design and then transforms it into something the PCB facility can use. It is not possible to manufacture PCBs without this step.

For this to happen, the elements comprising the PCB must be clearly recognised and understood by the CAM system. Data files must therefore be valid, i.e., clear, unequivocal, and in a recognisable format so that they are readable in a digital system.

Too often, this is not the case, yet many of the designers and CAD vendors who could make a difference are unaware of this.

Painting

One of the most troublesome practices used by designers today is one in which pads, and sometimes other features, are "painted." This is otherwise known as "stroking," "paint-fill," "stroke-fill," and "vector-fill." This practice, born decades ago in the heyday of the Standard Gerber format, was then a necessary design step. The vector photoplotters used at that time were

driven by computers that were nowhere near as sophisticated as those we use today, so the shapes they could plot were very limited indeed. Thus designers would build pads using separate elements: curves or circles for the rounded corners, lines, or strokes, for the edges, and more strokes to fill the outline.

Things have, of course, moved on since then. In today's CAD systems the pad shape is described by a geometric primitive or its outline, but instead of outputting this shape properly, too many systems are still "improving" the output by filling the outline with numerous filling strokes. The result, to the naked eye, is a clear, nicely-generated pad, and a correct image (Figure 1). But to a CAM system, it's chaos. Rather than reading a single pad, the system sees the digital reality: a hodgepodge of disconnected curves and straight lines, which it simply cannot recognise as anything particularly meaningful (Figure 2). Consider this happening not just once but maybe hundreds of times, and in different sizes, and across a densely populated PCB. It quickly becomes obvious that painted features are a CAM engineer's worst nightmare.

Of course, the image is correct, so one may think there is no big problem, just the nuisance that the files are bigger than they could be. However, the manufacturer needs more than a correct image. As we will demonstrate below, he needs to know the exact location and shape of all pads, areas and tracks.

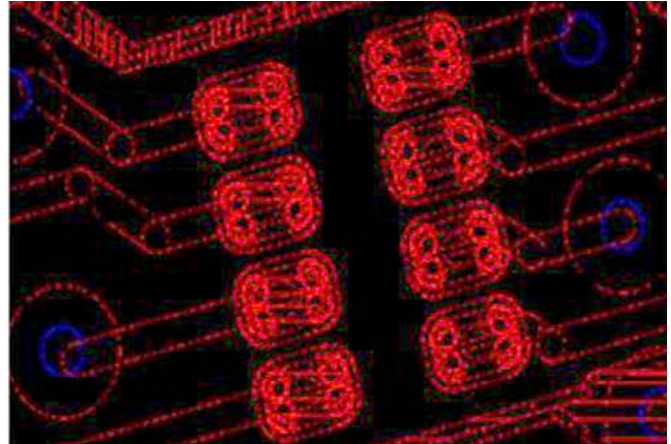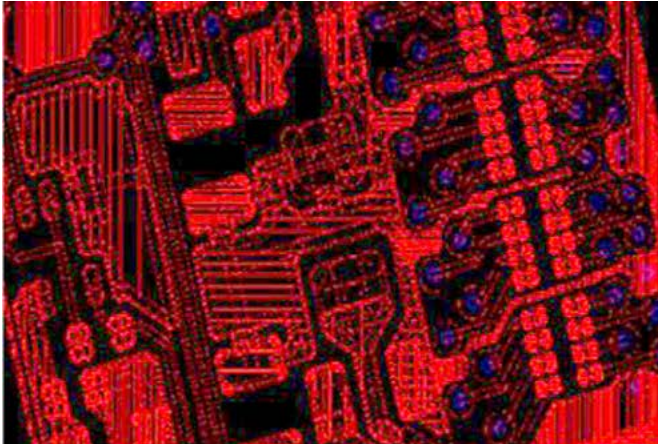Of course, CAM systems have tools that aid

Figure 2: ...but they are also digitally chaotic.

the search for painted features and their conversion into a CAM-legible structure, but this is still incredibly labour-intensive. Everything has to be verified manually, slowing the CAM phase considerably, potentially creating delays in delivery, or worse, giving rise to PCB errors further down the line. It also adds unnecessary complexity to the Gerber file, and as each feature is made up of a large number of objects, the files are humongous and slow.

Pads

PCB manufacturers need to know exactly where every single SMD, component and via pad is on the board. This information is used for netlist creation and electrical test, to ensure that the soldermask is applied precisely where necessary, for via plugging, and to ensure that all clearances are within spec. And, of course, the assembler needs to know where pads are in order to apply paste.

Instant recognition and selection of all pads is also important where feature dimensions must be modified to comply with designers' specs, for example, and as I mentioned, to compensate for the fabricator's specific etching process parameters.

Because pad data is so important, any fabricator that receives images with painted pads must scan the whole image, guess where the pads are and typically replace all painted pads by proper flashed pads (see Figure 3) prior to working with that file. This is a lengthy process

that can give rise to errors in the product, because the manufacturer ends up having to gauge the designer's intentions rather than dealing with clear data.

To obviate all of these problems, pads should be generated properly. This is easy with RS-274X Gerber, which has a number of built-in pad shapes, and a powerful and unique macro language that makes it easy to create any shape, easier in fact than with any other PCB image format. So there is really no need to use painting. Shapes are defined using the %AD and %AM parameters, and are then flashed wherever a pad should be: one flash, one pad.

Areas

Many PCB layers (e.g., power and ground layers) contain large copper areas, some of which can be extremely complex. The areas contain holes, or clearances that allow non-connecting vias to go through the planes. Theses are also known as antipads. Here too, some designers will paint these areas, carefully filling in around the antipads. The problems thus created are similar to those mentioned above, but whereas a pad is relatively small, an area can take up a significant part of a layer's surface area. The least of the CAM engineer's resulting problems is that the incoming data file is huge.

The real issue is that the inside of an area will accomodate so many draws that it becomes extremely difficult for the CAM system to differentiate between draws that should be within
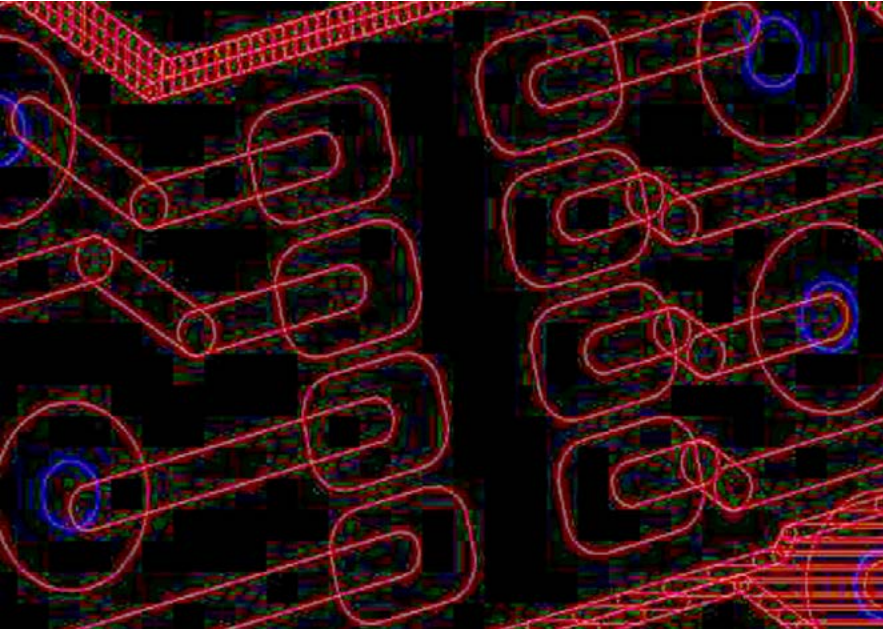
Figure 3: Painted pads are replaced manually with properly flashed structures, in this case using flashmaker.

the area, and draws that define neighbouring tracks. This creates all sorts of problems. For instance, when track width must be modified to compensate for etching parameters, it is crucially important to know which draws represent tracks and which represent painted pads or areas. The picture becomes even muddier when designers place embedded painted pads within painted areas.

Here too, in a lengthy, complicated, error-prone, manual operation, the manufacturer must replace the painted area with a properly constructed one, separating out the pads, tracks and areas from a messy jumble of draws.

This can all be avoided easily, as the CAD system will define the area by its outline, not by painting it in. The outlines can be directly stored in the Gerber file using the G36/G37 commands. This supports areas of any shape, size and complexity using concise, clear language, while antipads and their positions are defined precisely and efficiently by using the %LP parameter to make a negative layer containing all the holes. The areas are automatically filled later when the Gerber RS-274X file is created.

Tracks

Unfortunately, even tracks can be painted, the designer using multiple narrow draws to build tracks of the desired width. Here, the abovementioned problems are magnified hugely and recovering a proper job from them is a massive manual task. The only proper way to construct a track is to draw it with the correct aperture.

The Gerber Format and Painting

Some claim that painting is somehow intrinsic to the Gerber format. This is a fallacy.

But in the distant past, when RS-274-D was in use, there was some truth to this.

Using this format, the creation of non-standard shapes was so cumbersome that pad painting was a constant, and indeed painting in was the only way to create areas. So yes, D was guilty as charged for encouraging, even requiring, painting. But the D format is now obsolete, and these issues were solved more than 20 years ago with the introduction of the current Gerber format, RS-274X Extended Gerber. Why anybody in his right mind today would use D rather than X is a mystery to me.

So it is a fallacy to state that the current Gerber format requires or encourages painting. Areas can be created by contours; in fact, any pad shape can easily be created. Indeed, thanks to its aperture macros, Gerber offers designers the most powerful features available for the creation of arbitrary pad shapes.

The majority of Gerber files do not use painting, and there are plenty of files in ODB++, Barco DPF that do use painting. Which makes it very clear that painting is not a Gerber thing. Rather, it is due to a poor understanding of the CAM process, bad practices, bad setup, and sometimes poor implementation of file output software.

The Designer's Role

Of course, designers could say, and some do, that this whole issue is not their problem.

PAINTING PADS *continues*

Not so. By passing painted features to their fabrication partners, designers set up those partners to fail. The CAM engineer either resolves the painting issues pre-CAM, which is a time-consuming and error-prone process, or he passes the problems on, in which case the CAM process itself becomes time-consuming and error-prone. Caught between a rock and a hard place, the engineer's problems are made more acute when working to tight time constraints, and he may have to opt for speed rather than thoroughness.

So he's damned if he does and damned if he doesn't; he and the manufacturer get the blame for long turnarounds and missed deadlines, for quality problems in the product itself, and for potentially costly repairs and the wrath of his clients. It's an unenviable position and it's grossly unfair, but the problem is not just his to deal with. Poor data quality threatens the reputations and the businesses of everybody, from the manufacturer to the designer, the assembly company, and any OEM or EMS provider involved. And any compensation that may, unjustly, be exacted from the PCB manufacturer is poor recompense for all this damage.

The upshot is that it makes no sense whatsoever to use painting.

RS-274X does away with any need for this risky, outdated practice. I therefore urge any designers still using the obsolete RS-274-D Standard Gerber to move to the RS-274X format. I also strongly recommend that any designers using RS-274X, but who are still using painting, should make it a priority to review their working practices to eliminate it. The new, clarified RS-274X specification ensures that data is all-encompassing, clear and unambiguous. The unsatisfactory necessity of having to interpret data must become a thing of the past.

The CAD Vendor's Role

I would also like to put in a plea to CAD vendors to do their part in improving CAD-to-CAM data transfer. While it is understandable that they should retain painting as an option for compatibility with those rare systems that cannot handle proper Gerber, painting is at best a terrible waste of time, and often a quality risk. At worst, it is one of the most damaging CAD practices around. Painting should never be the default mode. The default mode should instead be flashed pads and the outlining of areas, with painting available as a legacy option.

Ucamco's Role

Ucamco does not make money out of the use of the Gerber format. On the contrary, caretaking Gerber is expensive and time-consuming. We do it because we are committed to the industry, and the industry is committed to Gerber; Gerber is used in possibly 95% of CAD-to-CAM data transfers globally. But it is also used improperly: An estimated 25% of designs created in RS-274X, for example, use painting. And then the format or the PCB fabricator are unjustly blamed. Designers seldom hear of the problems created and are often unaware that they are even using poor design practices, because fabricators feel unable to press for better data for fear of losing orders or clients.This is exacerbated by the fact that the supply chain is sometimes so long, complicated and successively outsourced that it is impossible to talk with anybody who is interested, or sufficiently versed, in design issues.

This article may have a somewhat revolutionary tone, but I hope that it at least provides food for thought. I hope the design community will take these points seriously enough to look at their design habits, to change them where necessary, and perhaps to find out where and how they can improve the lot of their PCB manufacturing partners.

Above all, I hope painting will soon become a thing of the past. It is a risky design practice that can impact every part of the electronics manfacturing chain, from the PCB manufacturer through the designer to the final customer. It is a throwback to days of old, as obsolete as paper tape or half-inch magtape. **PCBDESIGN**

Karel Tavernier is managing director of Ucamco. He has 30 years of experience with software and imaging equipment for the PCB and electronic packaging industry, including sales, service and R&D. He has been in his present role since 1995.