Why Does the PCB Industry Still Use Gerber?

Feature by Karel Tavernier
UCAMCO

Every so often, I hear technologists ask why so many PCB designers still use Gerber. That is a fair question. Ucamco has over 35 years of experience in developing and supporting cutting-edge software and hardware solutions for the global PCB industry. Our customers—small, medium, and large PCB fabricators—include the electronics industry's leading companies, and many of them have been with us for over 30 years. We are dedicated to our industry and excellence in everything we do, which includes our custodianship of the Gerber format.

Data Intelligence

With us, the Gerber format has undergone a significant evolution in the past 10 years, and a near revolution in the past five years alone. Like the previous Extended Gerber (X1) format, today's Gerber X2 is simple, easy to use, and freely available to our industry, but simple and free doesn't mean dumb—far from it. Thanks to the use of cleverly designed attributes, Gerber X2 is an intelligent format that can do all of the things that some critics say it can't. Gerber X2 can differentiate between pins, vias, and traces, and anyone who cares to read section 5.6 of the Gerber X2 specification will see that it does so in more detail and with greater precision than the ODB++ and IPC-2581 formats.

The industry's professionals know this because we talk with our customers daily and listen very carefully to what they tell us about being at the electronics production coalface. What they tell us is confirmed by a quick glance at the industry's use of the different formats. IPC-2581 is used for a negligible fraction of the world's fabrication data sets, ODB++ is used for 5%, Gerber X2 is used for 10%, and the rest—the vast majority—uses the traditional Gerber X1 format.

Supporters of ODB++ and IPC-2581 point to the "intelligence" of these PCB data formats, but Gerber X2 is the global PCB industry's most popular intelligent PCB data format. Sure, these formats are more intelligent than the old Gerber X1, which can't differentiate between SMD and BGA pins. But when ODB++ was launched, it did not contain any component information either.

Figure 1 shows an apples-to-apples comparison of ODB++, IPC-2581, and Gerber X2. These

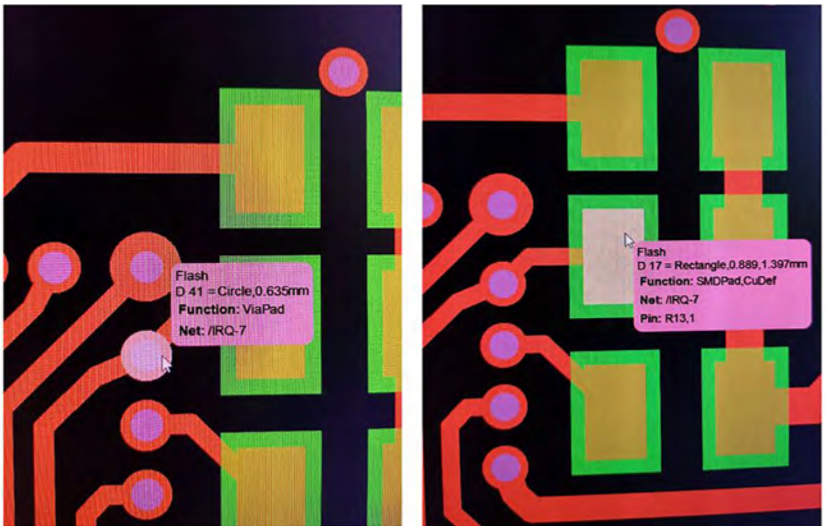

Figure 1: Output from a popular CAD system illustrates that Gerber X2 fully identifies pad types, sizes, nets, and pins.

screenshots from the Reference Gerber Viewer illustrate the level of information carried in a current Gerber X2 file. The pad selected on the left is identified as a via pad, and the pad selected on the right is identified as a copper-defined SMD pad, and as pin 1 of R13 with net /IRQ-7. With Gerber X2, a designer can easily distinguish between a pin, via, and trace.

As you can see, any claim that Gerber is "dumb" is wrong and misleading.

Directories and Files

Some detractors claim that Gerber's files and directories are awkward to use. But realistically, a zipped directory with ODB++ files is no different from a zipped directory of Gerber files. Furthermore, a typical ODB++ package is not even a single directory but a complex directory structure. To be fair, the IPC-2581 package is a single file, even before zipping it. This can be seen as an advantage, but it can be seen as a disadvantage too. Individual layers—such as AOI or imaging—must often be accessed during fabrication, so storing these layers in separate files (as in Gerber and ODB++) makes their extraction easier and cleaner.

DFM

Other naysayers claim that it is not possible to generate full and meaningful DFM reports from Gerber files. Figure 2 shows part of a QED report generated by Ucamco software from a Gerber job. The first block shows the rings categorized by vias, laser vias, component pads, and mechanical pads. The last block shows the clearances to outline split by pads, tracks, and copper pours. Clearly, meaningful DFM reports can be generated from Gerber files.

Evolution

For some users, the fact that Gerber has—apparently—remained unchanged is another no-no. Apart from the fact that Gerber has indeed evolved over its lifetime and continues to do so, let's go with this. Let's take the automotive industry's steering wheel. This has been, well, a wheel, since 1894 when Alfred Vacheron took part in the Paris-Rouen race with a four-horsepower Panhard model that he fitted with a steering wheel instead of the then-popular tiller. It was such a great idea that within a decade, the vast majority of new cars had steering wheels. The round shape was a winner, and its position at the front of the car was a pretty good idea too.

That was 125 years ago, and guess what? Steering wheels are still round, in the front, and used to direct cars. Nobody would say that they haven't evolved because they're not square or attached to the roof of the car. Steering wheels have kept what works and have evolved with automotive steering and safety technology, integrating in-car entertainment and comfort features on the way. This is a bit like Gerber. Like the steering wheel, Gerber has stayed at the front, keeping what works, while the rest of this free, open format evolves to keep abreast of and drive beneficial developments in PCB design and engineering.

Why Use Gerber?

One common question I hear is, "Why do designers still use Gerber?" I believe that designers have very solid grounds for their reluctance to drop what works in favor of a new format. It is not that these new formats are bad—

Copper Layer Minima - Copper vs Drill - Original

File	Pos.	Ring					Copper to Plated Clr.	Copper to NPTH Clr.	Copper to Outline Clr.			
		Overall	Via	Laser Via	Comp.	Mech.			Overall	Pad to Outline	Track to Outline	Region to Outline
		mm	mm	mm	mm	mm	mm	mm	mm	mm	mm	mm
TOP_art	1	0.062	0.125	0.062	0.300	0.000	0.150	0.301	0.625	0.625	0.645	1.044
INTERNE1_art	2	0.062	0.124	0.062	0.275	0.250	0.138	0.087	0.540	0.600	0.867	0.540
INTERNE2_art	3	0.062	0.150	0.062	0.275	0.000	0.138	0.301	0.600	0.600	0.763	>1.600
INTERNE3_art	4	0.062	0.150	0.062	0.275	0.000	0.138	0.255	0.555	0.600	0.555	>1.600
INTERNE4_art	5	0.150	0.150		0.275	0.250	0.237	0.087	0.540	0.600	1.550	0.540
INTERNE5_art	6	0.000	0.000		0.300	0.250	0.237	0.087	1.050	>1.600	1.417	1.050
INTERNE6_art	7	0.000	0.000		>0.800	>0.800	0.237	0.087	1.100	1.145	1.550	1.100
INTERNE7_art	8	0.150	0.150		0.275	0.250	0.237	0.087	0.540	0.600	1.550	0.540
INTERNE8_art	9	0.062	0.150	0.062	0.275	0.000	0.172	0.202	0.600	0.600	0.748	1.100
INTERNE9_art	10	0.062	0.150	0.062	0.275	0.000	0.150	0.087	0.396	0.396	0.571	>1.600
INTERNE10_art	11	0.062	0.124	0.062	0.275	0.250	0.150	0.087	0.540	0.600	1.550	0.540
BOTTOM_art	12	0.062	0.125	0.062	0.275	0.000	0.162	0.301	0.625	0.625	0.904	1.094

Figure 2: A QED report generated from a Gerber job shows detailed DFM data.

they are not. But to gain access to the meta-information they contain, you must adopt them wholesale, and that includes a new image format. The problem is that image formats are notoriously hard to implement.

Much has been written about just how complicated geometric software is and how much effort it takes to get it right, not to mention the years it takes to debug. And that is a big consideration for anyone thinking about adopting a new format. Errors in image transfer are catastrophic as they inevitably lead to scrap. Gerber's image functionality has been thoroughly tried, tested, and debugged over many years, which is why the global PCB industry knows that its PCB data is safe with Gerber. Can the same be said of IPC-2581? How many jobs does this format have under its belt?

It's also worth noting here that a huge portion of PCB software is developed by tiny companies, often one-person outfits. For larger companies, implementing ODB++ (let alone IPC-2581) is already difficult enough, but for these smaller companies, it's a massive—and often prohibitive—undertaking. Compare this with the ease and simplicity of adding attributes to an existing Gerber implementation. Gerber X2 offers developers a straightforward, affordable, effortless way to add "intelligence" to their input/output, allowing companies of all sizes to participate in the growth and advancement of the global PCB industry. Similarly, it's very easy for designers and fabricators to implement Gerber X2 as it's compatible with X1, so if the software being used cannot handle X2's new meta-information, it can just skip it and continue to work as before. Easy.

So, why does the PCB industry still use Gerber? Because it is the very best PCB data format out there. Gerber offers our industry a standard with a great image specification that is trustworthy and easy to implement and use, and it has evolved over the years as intelligence has been added to the image data. Ucamco is dedicated to excellence—its own, that of its customers, and that of the industry. **DESIGN007**

References

1. Visit the Reference Gerber Viewer and download a sample job here.

Karel Tavernier is managing director of Ucamco.