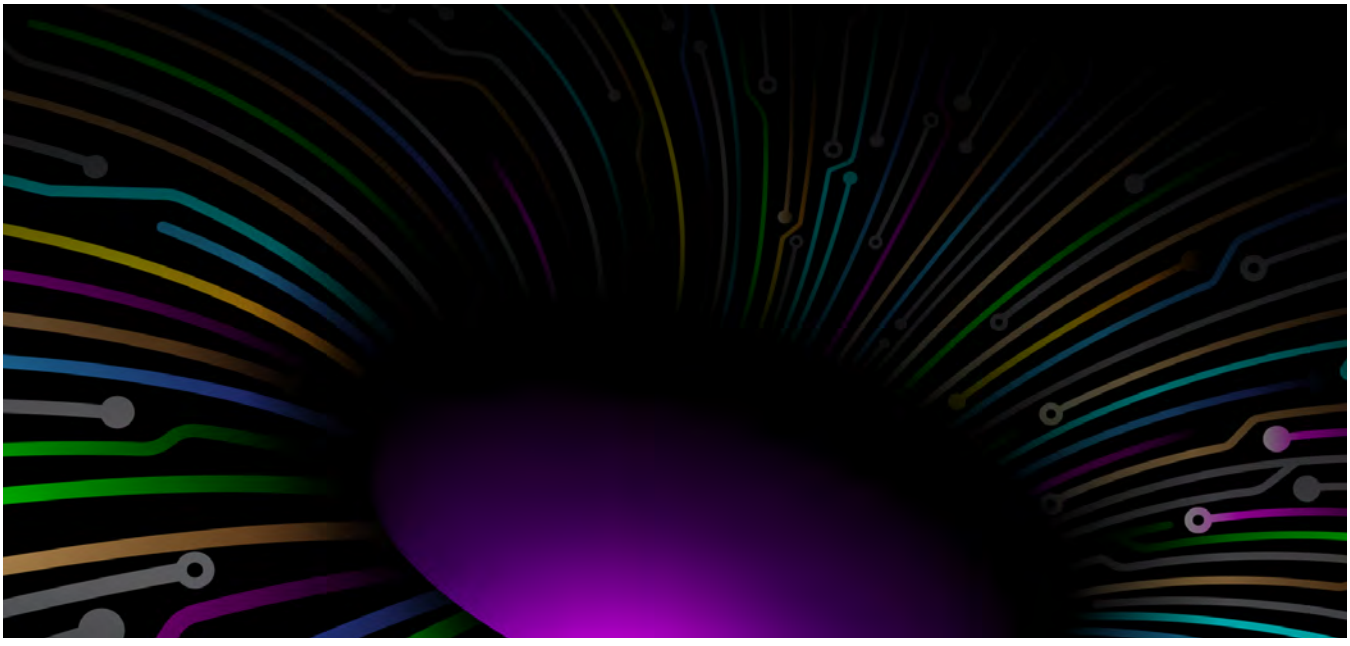

XNC Format: Gerber Takes Data Into the Future

Feature by Denis Morin, Karel Tavernier, Jean-Pierre Charras, and Marius Matioc
UCAMCO, KICAD AND PENTALOGIX

Gerber is the world's favourite data exchange format for PCB image data: it's easy to use, crystal clear, and gives designers and engineers an unequivocal language with which to communicate with each other. And this grand, old man of the PCB industry has remained at the forefront, powered by ongoing developments that add capability and functionality without ever compromising its characteristic simplicity and ease of use.

It's the ideal solution for transferring drill data too, as can be seen from scanning the specification. And many in the PCB industry use it for just this, but the majority are still transferring their routing and drilling coordinates using NC formats. These were never designed for data transfer, and more often than not create confusion and waste time.

Some argue that Gerber files, unlike NC files, can't be sent to a drilling machine. True enough, but PCB manufacturers never send their clients' incoming files to their machines anyway. Instead, the data goes through the CAM process and is then altered and output as is appropriate to the manufacturer's specific production line. For CAD, the question should not be which format is best for the machines, but rather which format is best for input into CAM. As we've said, this is undoubtedly Gerber.

So, why are CAD developers and their users still stuck on NC formats? It's most likely a question of inertia or tradition. Drill information has been transferred for decades using NC formats, principally Excellon, that are similar to the 1985 IPC-NC349 specification. Also, there's still a lot of legacy software out there, so NC files will likely be with us for a while.

The Problem With Existing NC Specifications

The problem is that so many NC files are of deplorable quality because the NC format was never designed as a data transfer format. It has always been a machine driver and contains all sorts of information that a drilling machine needs, but that is irrelevant and confusing for data exchange. For example, CAD software will typically ask users to specify whether routing should be achieved using nibbling or slot creation and which drill feeds and speeds are to

be used. These are decisions that only the fabricator can make, and yet many CAD professionals will feel duty bound to give some sort of answer, which will inevitably be wrong.

Clearly, CAD developers will try to avoid such scenarios by going through their chosen NC format and picking what they believe is most appropriate for their software and the CAD to CAM data transfer process. This is not easy, as the language is dense, sometimes redundant, and IPC-NC349 and Excellon 2 contain legacy Excellon 1 code—most of which CAD developers don't need. So, for fear of leaving out parts of the format that could be of value, developers tend to include more than necessary, which just adds more confusion for the CAD user.

Despite the developers' best efforts, it's not always clear how to use parts of the NC formats or if they are even capable of transferring certain data. CAD users will simply leave these parts out of their drill files and express the relevant information as sidecar information in comments or in separate text files.

But the biggest problem with these NC specifications is that, thanks to an age-old space-saving convention, the drilling coordinates lack a decimal point. That's okay if there's a command or header in the file that indicates where the decimal point must be—the so-called fixed "point format." But in Excellon files, there is no such instruction or standard for saying where it should be, so designers are on their own. Similarly, there is no standard for expressing whether the designs are in imperial or metric measurements. The Excellon specification does mention defaults, but these are also confusing and can end up being used in different ways.

All of this places the final responsibility on the CAM engineer to try different possible variants until the drill files fit with the copper files. This is fine—if not great—when handling normal boards, but is not okay when handling RF boards, for example, where there are no clear pads to fit the holes.

An additional issue in the NC stakes is that the specification is no longer published, so the industry has relied for decades on copies of copies that may be infringing copyright protections. And the IPCNC-349 specification—which is dense, overcomplicated, and

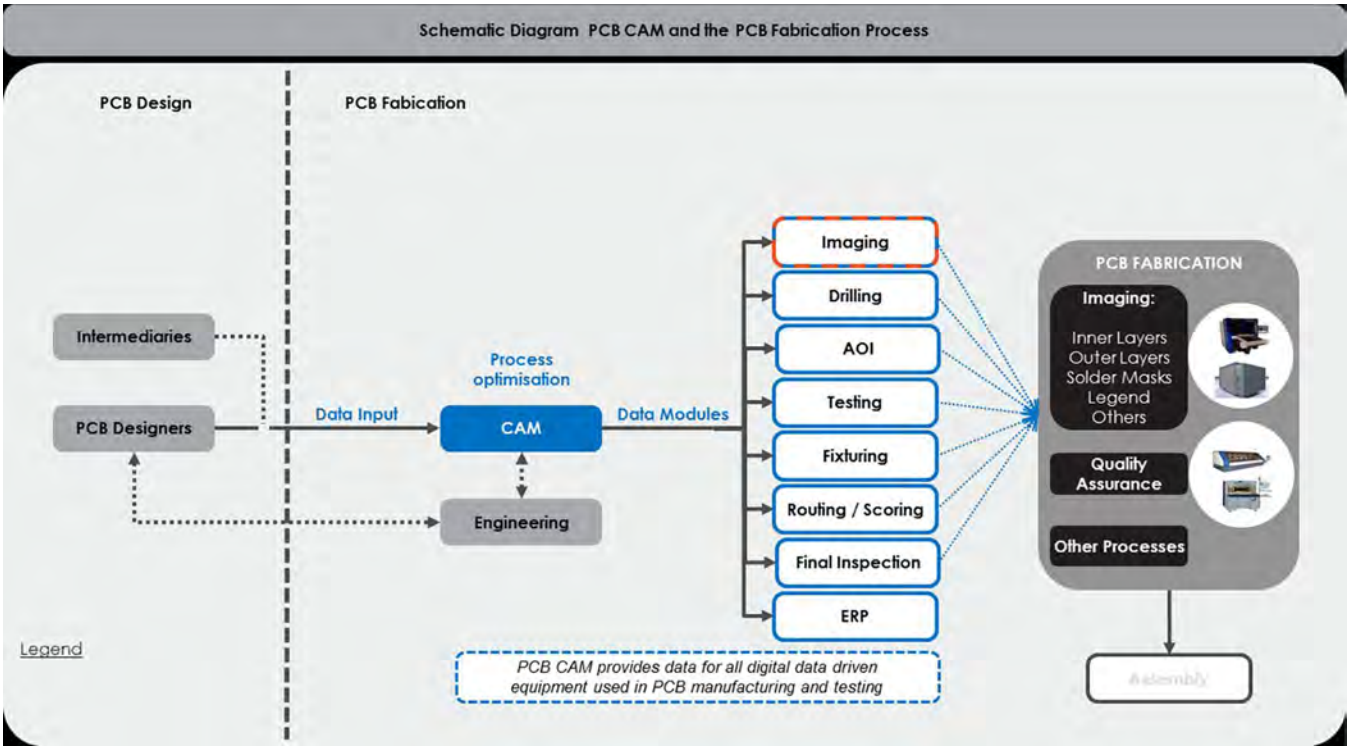

Figure 1: An example of the many steps of the process from PCB design to manufacturing.

full of overlapping methodologies—is hardly in circulation because it's expensive. So, ironically, the PCB industry is likely developing NC files based on non-existent, illegal, incomprehensible, or unaffordable specifications. And when it's not, CAD software is reverse engineered from the clutter of existing NC files.

XNC

So, where do we go now as an industry, given that we're not yet ready for Gerber drill files? Fortunately, there's a new format for PCB drill data that takes all the confusion out of CAD software development and the CAD-to-CAM drill data exchange process. Designed by Ucamco, KiCad, and PentaLogix with the support of Graphicode, Cuprum, and ZofzPCB, XNC is a strict subset of the widely used NC format.

We believe that the first step towards improving the NC drill chaos is to develop a simple, clear specification without embellishments—one based on an existing format that can be read by all decent PCB drill input software. To this end, we have taken great care to design the CAD/CAM Exchange NC (XNC) format—a complete, compact, and unequivocal subset of IPC-NC-349 that is capable of exchanging CAD/CAM drill information without the need for additional sidecar files.

And we've added to this the power of Gerber-type metadata or attributes. XNC attributes can be attached to the complete file, tools, or individual holes, describing characteristics in a standard, flexible way using similar syntax to that used for Gerber attributes. XNC files are compatible with Gerber X2 and can be added seamlessly to X2 data sets while also ensuring that the format is compatible with software that does not read attributes.

With XNC, CAD developers can create output software easily and quickly, using formats that are already well known but without the headache of having to wade through and choose from a bewildering array of possibilities and functionalities, or reverse engineer from multiple incomplete and confusing NC files. We guarantee that if CAD developers limit themselves to using just the XNC format, they will give their clients exactly what they need—a tight file format that will improve the CAD-CAM data transfer process overnight, and an NC reference towards which the industry can work so that NC files converge to a common standard.

Conclusion

Existing NC drill data CAD-CAM transfer processes are deeply flawed thanks to confusion at every level of the CAD and CAM information development process due to industry inertia and inappropriate data format specifications. A lot of time and resources are being wasted by CAD software developers trying to make sure that every eventuality has been accounted for in their software, CAD professionals delivering irrelevant and non-standard information about their designs, and CAM engineers interpreting the results. A new drill data exchange specification designed by some of the industry's leading PCB software providers and based on an existing and known NC format promises to alleviate many of the issues and pave the way to a new industry standard for NC formatting and files.

The first version of the specification of the XNC format can be found here. **DESIGN007**

Karel Tavernier is the managing director of Ucamco with 30 years of experience in software and imaging equipment.

Denis Morin is a software application engineer on Ucamco's European support team.

Jean-Pierre Charras is the founder of KiCad, researcher at Laboratoire des Images et des Signaux, and a professor in electronics engineering and image processing at IUT in France.

Marius Matioc is the creator of Lavenir CAM software and head of product development at PentaLogix.