Improving CAD to CAM Data Transfer A Practical Approach

Karel Tavernier
Ucamco

1. Introduction

The designer's CAD system has a perfect digital representation of the board but in transferring the product data to the manufacturer's CAM system much is "lost in translation". Not all useful information is transferred. Much of what is transferred is contained in free format text files or drawings, which must be retyped or redrawn manually in the CAM system. This results in wasted labor, longer cycle times and worse, production errors.

The state of the data transfer CAD to CAM is dismal, unworthy of a great technological industry. What is worse is that it barely improving; the quality of data is scarcely better than it was in the 1990's.

This article analyzes why progress is so slow and suggests a way forward.

2. Computational geometry is difficult

Experience shows that new geometric applications are initially plagued by tricky bugs. This is not because the programmers implementing them are particularly incompetent or sloppy. It is because this type of programming is very difficult.

This difficulty is recognized in the literature. "The Algorithm Design Manual" [1] states: Implementing basic geometric primitives is a task fraught with peril... There are two different issues at work here: geometric degeneracy and numerical instability... Geometric applications can be made robust by writing special code to handle each of the special cases... Expect to expend a lot of effort if you are determined to do it right... The difficulties associated with producing robust geometric software are still under attack by researchers. Or from Computational Geometry in C [2]: There is no easy solution to the fundamental problems faced here, an instance of what has become known as robust computation. There are several coping strategies...

Each new image format will initially be plagued with bugs. The PCB industry knows this very well. A bug in the CAD output or the CAM input of an image file will often not be noticeable in CAM and result in scrap. This explains why the industry is reluctant to take on new image formats. This reluctance is well illustrated by the following example. Some datasets contain the data both in Gerber and in the newer ODB++ format. These datasets often contain a readme.txt file stating:

```
BARE BOARDS MUST BE FABRICATED WITH GERBER,
DRILL AND IPC-356 NETLIST PROVIDED. BOARDS ARE
NOT TO BE FABRICATED FROM ODB++ FILE.
```

This does not indicate that there is anything intrinsically wrong with the ODB++ format. On the contrary, the ODB++ is included because it may contain useful information. It does indicate, however, a concern about the reliability of the newer ODB++ software.

3. RS-274X as a 2D image format

The RS-274X format is simple, compact and unequivocal. It is easy to interpret. It describes an image with very high precision. It is complete: one single file describes each production layer. It is portable and easy to debug as it uses printable 7-

bit ASCII characters. The mix of D codes, G codes and parameters has developed historically and may not be very elegant, but it has all the necessary features: areas, definable apertures, pos/neg exposure etc [3].

A well-constructed RS-274X file precisely defines the PCB image data and the functions of the different image elements. The image and drill layers of the most complex design are being transferred flawlessly in RS-274X. RS-274X is an excellent 2D image format for PCB applications.

Most RS-274X software is very robust and very reliable. This is the very rational basis for the popularity of the format.

4. The issues with the current RS-274X datasets

However, there are definitely issues with the current datasets. We will discuss the different issues below.

4.1. Usage of RS-274X format

A few implementations are wrong due to a misunderstanding of the format specification. To address this issue we at Ucamco have brought out a new version of the specification document [3] in which frequent misunderstanding are clarified.

A more significant issue is bad practice. In these cases the file is valid but the it is are poorly constructed. Poorly constructed files take longer to process, require more manual work and increase the risk of errors. Typical bad practices are

- Low numerical resolution. This may have been useful in the 1980's to save bytes, but nowadays it makes no sense. Low resolution results in an inaccurate image, bad registration of features and increases the risk problems due to rounding errors (invalid arcs, invalid outlines etc).
- Painted or stroked pads and areas. This is especially troublesome. The CAM operator has to replace all the painted pads by flashes and the painted areas by outlines, a time-consuming, tedious and error-prone process. Painting is a hang-over from the vector plotters of the 1960's and 1970's. There is no valid reason why it is still used.
- Presence of junk. Junk are all image elements in the file that are not part of the PCB, such as registration marks and comments in drawing. They should not be there. Junk interferes with regular CAM operations and has to be deleted by the operator. It may contain useful information, but that information should not be hidden in the PCB image layers, but put in a separate file, typically a readme text file.

To address these issues the latest revision of the RS-274X specification contains a section setting out recommended practices.

The issue here is not the RS-274X format itself. Bad practices can occur in format. In a new, more complex, less robust and less familiar format the same bad practices require even more manual work and increase the risk of errors even more.

These bad practices are the main problem in current datasets. To quote a manufacturer: "If we would only receive proper RS-274X files, it would be a perfect world."

4.2. Specifying the complete 2.5D PCB image

A PCB is not a 2D image nor is it fully 3D. It consists of both horizontal and vertical layers, horizontal copper and print layers and vertical drill and rout layers. Such a structure is often called 2.5D image.

For each image layer and f drill layer the function and position must be specified. The simplest

solution is to give each file a clear and unambiguous name. If there are several drill sequences (PTH, NPTH, blind and buried) the data can be split over several files, each clearly named. For example (the prefix can contain part name and revision):

```
<Prefix>_Solder_Mask_Top.ger
<Prefix>_Layer_Top.ger
<Prefix>_NPTH.ger
<Prefix>_Blind_1_2.ger
<Prefix>_Profile.ger
...
```

This specifies the PCB images completely and unequivocally. It is easy to load the job into CAM.

The next step should be for an international body like the IPC to standardize these names. With systematic and unequivocal names it becomes possible to input jobs fully automatically, as is demonstrated by Ucamco's automatic input and analysis software Integr8tor.

4.3. Non-image parameters of the PCB

These are the stackup, materials, finishes, thickness and so on. Each parameter is simple to specify. They can be put in a free format text file, or better in an XML file, which is easier to parse. The XML file can follow an ad-hoc scheme. Ideally, the XML file should be standardized, see below.

4.4. The netlist

The netlist should be included in every dataset as it provides a crucial cross-check on the accuracy of the Gerber data. This would benefit both the customer and the manufacturer. It is hard to understand why it is not included more often. It is definitely not a format issue: a netlist is a simple object, and there are widely implemented standards, e.g. IPC-356.

4.5. Use of RS-274-D

A few words must be said about RS-274-D or Standard Gerber. This format was developed to drive NC machine tools and was used for Gerber vector plotters in the 1960's and 1970's. An RS-274-D file by itself does not describe an image [4]. It needs a so-called wheel file, for which there are no standards. It is not an image description format.

The RS-274-D format specifies the X Y movements of machines that are no longer in use. It is amazing that RS-274-D files are still used. It is like using teletype paper tape to transfer text documents.

We call on industry experts and professional organizations to discourage the use of the obsolete RS-274-D format.

Summary of the issues

Issue	Comment
4.1 Usage of the format	This is not a RS-274X issue. The same bad practices can occur in any format.
4.2 Full 2.5 data	Not available in RS-274X. Overcome with straightforward naming conventions. Standardization useful.
4.3 Non-image data	Not available in RS-274X. Overcome with text table, preferably in XML. Standardization useful.
4.4 Netlist	This is not a RS-274X issue. Standard formats are available. The issue is that they are not used widely enough.
4.5 Use of RS-274-D	This is not a RS-274X issue. If D users do not switch to X, they will definitely not switch to another format.

5. Where we went wrong

For many years the PCB industry has an excellent image format available in RS-274X. It's the de facto standard for PCB image data transfer. Virtually every PCB design system outputs it and every PCB front-end engineering system inputs it. Implementations are thoroughly field-tested and debugged. Its widespread availability allows PCB professionals to exchange image, drill and rout securely and efficiently.

That does not mean there are no issues in the current datasets. These issues have erroneously been identified these issues as an image format problem. This is however not correct. Issues 1,4 and 5 are not caused by the RS-274X format. Another format would not solve these problems; it would probably make them worse. Issues 2 and 3 are data elements that cannot be expressed in RS-274X. However, this is not image data, it is far less complex and ban be specified straightforwardly in auxiliary files.

The solution proposed has been to introduce new formats. These unavoidably introduce dangerous bugs which are only ironed out over many years. The damage caused by bugs in image transfer is far worse than the problems caused by issues 2 and 3. Furthermore, a new format is an all or nothing solution: CAD and CAM software must fully and reliably support the new format or the new format cannot be used. Gradual improvement is not possible.

Several attempts have been made to introduce new formats. Practical industry users have refused to follow. Wisely.

6. The road forward

The practical road forward is to stick to the RS-274X format for the image and address the issues directly.

Issue	Solution
4.1 Usage of the format	Use good practice.
4.2 Full 2.5 data	Use clear names. A standard for this specific item would be very helpful.
4.3 Non-image data	Provide the data in text or XML. A standard for this specific item would be very helpful.
4.4 Netlist	Always include a netlist, e.g. in IPC-356.
4.5 Use of RS-274-D	Do not use it.

Solving each of these issues, even partially, would improve the workflow. It would make it safer. **It would not introduce any new risk.** Each partial solution would help.

The end result would be a safe and seamless transfer of data from CAD to CAM. If after solving all the issues above, additional functions in the image format would be needed, they can be added to the RS-274X standard in an upwardly compatible way, without breaking existing implementations.

Ideally the data in item 2 and item 3 would be transferred following a standard XML format. An excellent XML scheme to describe materials, layer functions and other PCB parameters is found in the IPC-2581 format. However, IPC-2581 also contains a new image format which is the reason it is not used. A practical possibility would be to take the part of the IPC-2581 format that describes the stackup, but use it to refer to RS-274X files instead to a new-fangled image format.

7. Parallel with the graphic arts (or printing) industry

The workflow in the graphic arts (or printing) industry is similar to the one in the PCB industry. The printer receives a digital description of the

magazine, brochure or consumer package and produces the required number of copies. The core content of the description is image data. We may feel the production process in the printing industry is simpler than making PCBs, but it is not absolutely simple; just listen to an advertiser fussing over the precise hue in an ad, or a product manager worrying about the shape and color of a consumer product package.

In the 1980's the data transfer from customer to printer was partly digital and partly analog. It was even more dismal than it was in our industry. However, the graphic arts have evolved. The complete job is transferred securely in PDF, or strictly speaking in Certified PDF. The PDF job is read into the printer's CAM system without manual intervention. In fact, the data transfer is so reliable that workflows are largely automated. Many ads in magazines are printed in a completely automated workflow. No operator touches them or even looks at them. We can only dream of such automation.

The fully digigal data transfer in the graphic arts industry started in the 1980's with the gradual, and initially reluctant, use of PostScript, the so-called the "PostScript Revolution". PostScript is a page description language designed to drive black and white laser printers such as the AppleWriter. It had very severe limitations. It was succeeded by PostScript Level 2, PostScript Level 3, and later PDF. PDF [6] is an evolution of PostScript; they share the same imaging model. PDF itself evolved through a succession of versions and is now at version 1.7. Each version was upwards compatible from the previous one. In the course of this evolution the format added new capabilities, process color, spot color, overprint, transparency and security with Certified PDF. This tremendous success was achieved by gradually improving a functioning workflow, and not by attempting to overthrow it.

In the PCB industry we also had our revolution in the 1980's, when manufacturers started to take digital data rather than film. By analogy we could call it the "Gerber Revolution". However, since that time we are stuck.

I believe that we are stuck because as an industry we attempt to overthrow the current image format instead of improving the workflow. Nothing happens because a complete format replacement is too hard and too risky.

We must learn from the tremendous success achieved in the graphic arts, emulate it and firmly grasp the path of gradual improvement using upward compatible workflows.

8. Conclusion

There are serious issues in the transfer of data from CAD to CAM. Attempts to address these by introducing a new format have always failed, and for good reason. The best road forward for the industry is to stick to RS-274X and follow the path of gradual improvement: make better use of the format, include the netlist, stop using RS-274-D, and introduce industry standards for specifying the stackup and other product parameters.

References

[1] ***The Algorithm Design Manual*** by Steven S. Skiena. Springer-Verlag New York, Inc. 1998.

[2] ***Computational Geometry in C*** by Joseph O'Rourke. Cambridge University Press 2001.

[3] ***The RS-274X or Extended Gerber format. Revision G.*** Ucamco www.ucamco.com, 2010

[4] ***EIA Standard RS-274-D*** Electronic Industries Association, Engineering Department, 1979

[5] ***IPC-2581*** IPC www.ipc,org 2004

[6] ***PDF 1.7 Reference, sixth edition*** Adobe Systems Corporation, 2006.