

Neutral, Open and Global Standard for Efficient PCB Design Data Exchange

For 60 years, printed circuit boards have been designed on the computer and data for production of PCBs has been transmitted electronically. Data formats for writing the instructions for photoplotters have changed over time. As technology of plotters and technical possibilities on printed circuit boards progressed, formats of Gerber and Extended Gerber also changed to open IPC Standard 2581.

Throwback

One of the first photoplotters was introduced in 1960 by Gerber Scientific Inc. A photoplotter is a special electronic, optical and mechanical device that can be used to create an exposure stencil on a monochromatic film with high contrast. Light is exposed onto the film through selected aperture openings, similar to analog photography. There was a small set of commands for

different functions such as light on or off, selection of one of the 8 or 16 apertures, and movement in X and Y direction. Commands were a subset of RS-274-D command set for controlling numerical (NC) machine tools.

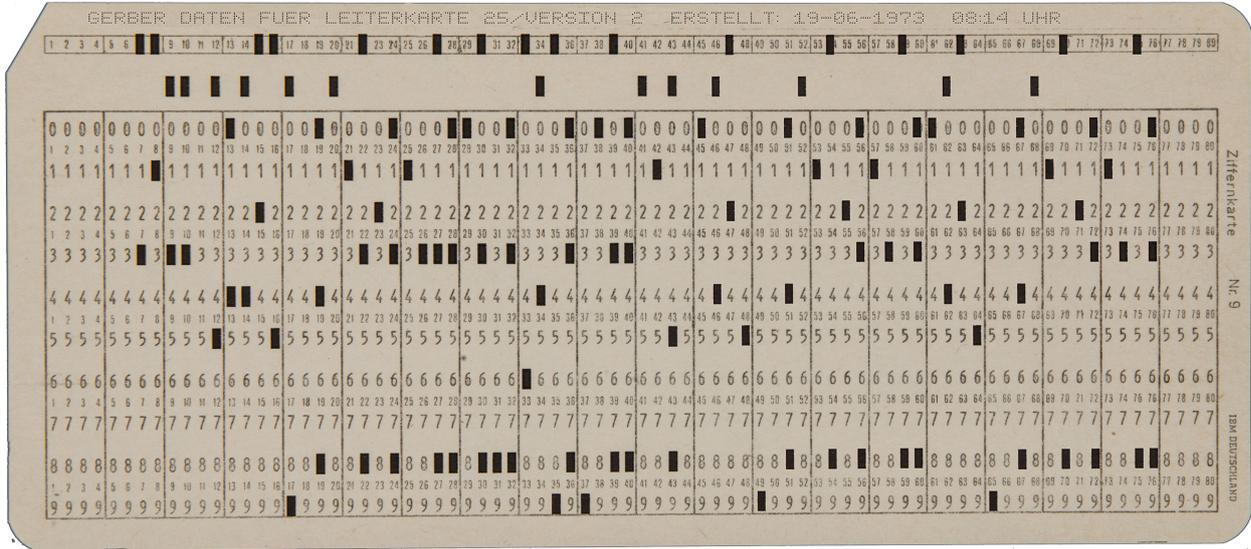


Image: Standard Gerber data for first photoplotters on punch cards

This instruction set, specially adapted to describe wires, copper areas, solder stop areas and legends, has been exported in Gerber format, also known as standard Gerber or Gerber RS-274-D. At that time, data was still transferred by punch card and amount of data had to be small. Since this format did not contain any information about apertures, there was another file with aperture information for each plotter. Due to the progress of technology in printed circuit boards as well as in photoplotters, Gerber format was no longer sufficient, became obsolete and was expanded in 1998.

Gerber Scientific Inc. was acquired by Barco ETS in 1998, which expanded the format to Extended Gerber in September 1998 and published RS-274X Format User Guide.

Extended Gerber

Extended Gerber or RS-274X is a human readable instruction set in ASCII format. It contains a set of commands from graphical objects with positive or negative properties. In Extended Gerber aperture information is included in the file. From these objects the image is then created in resolution of the plotter and currently exposed with laser beams. Ucamco (formerly Barco ETS) owns the rights to Gerber format.

In September 2014, Karel Tavernier, Managing Director of Ucamco, wrote that Standard Gerber is technically obsolete and should no longer be used. Although one could derive it from its name, Standard Gerber is not a defined standard for PCB data transfer: Units and aperture definitions are not determined by any recognizable standard, but are contained in an informal document whose interpretation is necessarily subjective. For this reason, Standard Gerber files cannot be read by machines in a standardized, reliable way. Standard Gerber requires aperture marking and copper plating, both are manual tasks in CAM, resulting in increased costs, delays

and risks in PCB manufacturing. Extended Gerber can be read by machines and consequently replaced Standard Gerber.

Valor ODB++

In PCB manufacturing not only films for layers are created, there was always a lot of descriptive information about production or assembly besides (extended) Gerber files. Valor Computerized Systems Ltd. developed ODB++ format, which allowed more different information to be stored in a zipped data container from various files. Files include e.g. netlist, component information as parts list or texts. Valor was taken over by Mentor Graphics in 2010, Mentor has since been "swallowed" by Siemens. Goal of ODB++ is to output and transfer all design data for manufacturing, assembly and testing together in a single file. ODB++ contains a complete layer table that describes original layer designations, layer type and order of layers. Unlike Gerber data, ODB++ usually makes it clear what a pad and what a trace is. An attribute system can be used to define register marks and test points.

IPC-2581

Technological leaps in printed circuit board technology clearly showed the problem when so-called standards for data transmission belong to individual companies and are defined by these companies. Extended Gerber belongs to a photoplotter manufacturer and ODB++ to a PCB layout provider. Especially when innovative technologies are developed, new parameters have to be transferred from development tool to CAM software, exposure units or automatic placement machines.

The two administrators of the standard did not meet industry's wishes to extend the standard to new technologies that they themselves did not support in their products. Therefore, many companies joined forces and agreed to establish a neutral, open standard for PCB development that would meet diverse requirements. The IPC - Association Connecting Electronics Industries was chosen as parent organization. IPC is a worldwide trade and standardization organization based in Illinois, USA, which deals with concerns of electronics manufacturing. Its activities include publication of industry standards and criteria for evaluating various goods in electrical and electronics industry.

```

1 <?xml version = "1.0" encoding = "UTF-8"?>
2 <IPC-2581 revision="B" xmlns="http://webstds.ipc.org/2581" xmlns:xsi="http://www.w3.org/2001/XMLSchema-i
3 <Content roleRef="Owner-FlowCAD-Demo">
4 <FunctionMode mode="ASSEMBLY" level="3"/>
5 <StepRef name="module1_uvia"/>
6 <LayerRef name="DRILL_1-8"/>
7 <BomRef name="BOM_module1_uvia"/>
8 <DictionaryColor>
9 <EntryColor id="COLOR_DRILL_1-8">
10 <Color r="251" g="216" b="181"/>
11 </EntryColor>
12 </DictionaryColor>
13 <DictionaryLineDesc units="MILLIMETER">
14 <EntryLineDesc id="ROUND_0">
15 <LineDesc lineEnd="ROUND" lineWidth="0.000000"/>
16 </EntryLineDesc>
17 </DictionaryLineDesc>
18 <DictionaryFillDesc units="MILLIMETER">
19 <EntryFillDesc id="SOLID_FILL">
20 <FillDesc fillProperty="FILL"/>
21 </EntryFillDesc>
22 <EntryFillDesc id="HOLLOW">
23 <FillDesc fillProperty="HOLLOW"/>
24 </EntryFillDesc>
25 </DictionaryFillDesc>

```

Bild: Readable IPC-2581 information in XML format

IPC-2581 is not only being developed by two individual companies as before, but IPC-2581 consortium comprises more than 100 members. Participating companies come from EDA-CAD,

CAM, assembly (EMS), PCB manufacturing and testing, as well as users from all over the world. They all have a common goal: a modern data exchange standard for all, from all industries such as automotive, aerospace, medical, telecom, industrial and consumer.

IPC-2581 is free of licenses as well as usage agreements and is supported by all major market participants. Various companies in the USA and Asia have already fully converted to IPC-2581 standard and transmit their data exclusively in this format.

The reason is not only that all data can be transferred in one data container. Interesting applications have already been established.

Advantages of IPC-2581

Main differences between ODB++ and IPC-2581 are that IPC format is not defined and maintained by one company, but by a consortium of over a hundred companies. This ensures that it is an open and manufacturer-independent standard and that requirements easily can be adapted to new technologies. There are no commercial advantages or licensing rights with this standard.

Every software manufacturer can program and maintain his IPC interface license-free himself and is not dependent on the help of others. This provides security for CAD flows.

All data is contained in a readable XML file, so there can be no confusion. File can be filled differently depending on the user, so that only information that the recipient is allowed to see is exchanged.

Layer stack-up with IPC-2581

If circuit boards needs to be technically demanding, particularly reliable or inexpensive, a precise specification of layer structure is required. For this purpose, developers request a layer structure from PCB manufacturers and are then get layer structure for requested specification. Materials, layer thicknesses and other parameters are defined here.

For impedance controlled printed circuit boards, PCB manufacturer adapts materials, e.g. by system-specific undercuts so that they can guarantee an impedance of 50 ohms between two layers with specified conductor track widths. Many PCB manufacturers use software from Polar and can output layer stack-up as IPC-2581.

#	Name	Layer	Layer Function	Thickness		Layer ID	Material	Signal Integrity					
				Value	mil			Conductivity	Dielectric Constant	Width	Impedance	Loss	Shield
				mil			mho/cm		mil	Ohm	Tangent		
	SOLDERMASK_TOP	Surface	Solder Mask	6.984232			LIQUID PHOTODUMLABLE MASK	0	1			0	
1	TOP	Conductor	Conductor	1.41732	1	1	COPPER FOIL	599900	1	5.4	89.782	0.035	
		Dielectric	Dielectric Prepreg	2.37205			PREPREG 3113	0	4.2			0.035	
2	L2	Conductor	Conductor	2.88661	2	2	FR4 CORE	599900	1	5.0	46.526	0	
		Dielectric	Dielectric Core	2.93276			FR4 CORE	0	4.2			0.035	
3	L3	Plane	Plane	1.37795	3	3	FR4 CORE	599900	1			0	
		Dielectric	Dielectric Prepreg	2.71653			PREPREG 1080	0	4.2			0.035	
		Dielectric	Dielectric Prepreg	7.24809			PREPREG 2028	0	4.2			0.035	
4	L4	Conductor	Conductor	1.37795	4	4	FR4 CORE	599900	1	5.0	77.739	0	
		Dielectric	Dielectric Core	11.811			FR4 CORE	0	4.2			0.035	
5	L5	Conductor	Conductor	1.37795	5	5	FR4 CORE	599900	1	5.0	77.739	0	
		Dielectric	Dielectric Prepreg	2.71653			PREPREG 1080	0	4.2			0.035	
		Dielectric	Dielectric Prepreg	7.24809			PREPREG 2028	0	4.2			0.035	
		Dielectric	Dielectric Prepreg	2.71653			PREPREG 1080	0	4.2			0.035	
6	L6	Plane	Plane	1.37795	6	6	FR4 CORE	599900	1			0	
		Dielectric	Dielectric Core	2.93276			FR4 CORE	0	4.2			0.035	
7	L7	Conductor	Conductor	2.88661	7	7	FR4 CORE	599900	1	5.0	46.526	0	
		Dielectric	Dielectric Prepreg	2.37205			PREPREG 3113	0	4.2			0.035	
8	BOTTOM	Conductor	Conductor	1.41732	8	8	COPPER FOIL	599900	1	5.0	91.949	0	
	SOLDERMASK_BOTTOM	Mask	Solder Mask	6.984232			LIQUID PHOTODUMLABLE MASK	0	1			0.035	

Bild: Imported layer stack-up with materials and electrical properties

Layer stack-up is already defined when the designer reads the IPC-2581 file into OrCAD or Allegro. Routing on individual layers can now be optimally simulated and verified with 50 Ohm layer structure for signal integrity. Quick change of a layer stack also shows whether different suppliers with different materials have the same electrical properties and thus represent a second source.

Secure output for production, assembly and test

Production data output is a double-edged sword. On one hand, you want to transmit all information necessary for production. On the other hand, you want to retain know-how and only pass on most necessary information to external companies. Especially if PCBs are produced by different companies local and abroad, the amount of data should be controlled to protect the IP.

IPC-2581 format is a <XML> based data format that defines all possible values. When exporting to OrCAD or Allegro, output profiles can be defined, for example fabrication, assembly, test and internal. For each output profile it can be defined which data is required for this production step. Thus, PCB manufacturer does not receive any component information and assembly house does not receive any layered materials.

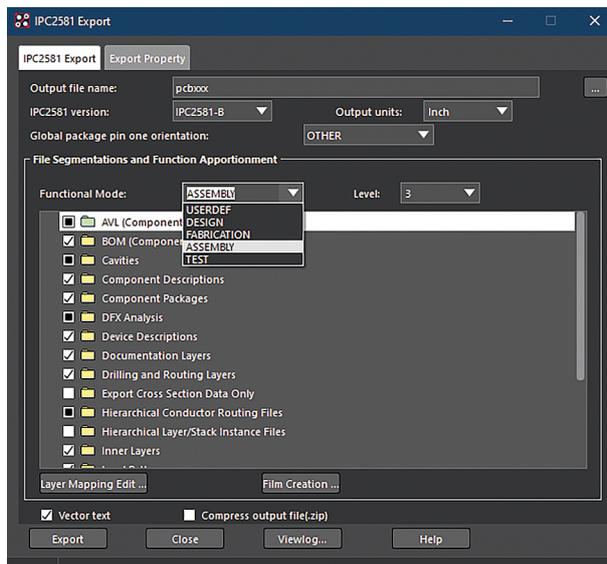


Bild: Export dialog with predefined configuration for different recipients

When exporting, four standardized files for different uses can be read out and stored in PLM system. Then purchasing department can send corresponding production data to suppliers with one file each, and each supplier has all the information it needs in one file. Confusion of individual Gerber films or time-consuming technical queries from the supplier to the designer are thus minimized. IPC-2581 format can also be compressed during output.

After 6 decades it is the right time to gain a market advantage through the advantages of IPC-2581. IPC-2581 embodies what electronics industry has always demanded. It is a

manufacturer-independent standard that is not restricted by restrictions of individuals and can be used by the entire electronics industry.

More Information

www.FlowCAD.com