

Design data transfer to Manufacturing using IPC-2581

DIRK MULLER

The independent, neutral and global design data transfer standard - IPC-2581 - provides PCB design and manufacturing supply chain companies an intelligent format which includes all necessary information in one single file.

PCBs have changed significantly over the past three decades. New technologies like surface mount components, JTAG test circuits, BGAs, blind and buried vias, microvias, buildup layers, embedded components (discrete and active), and embedded waveguides are making PCBs very complex electrical circuit carrier today. However surprisingly, we are still using the data transfer formats from the last millennium to transfer PCB design data to manufacturing. The over 30 years' old data formats were originally conceived to drive the emerging numerically controlled machines. As these early formats such as Gerber RS274 alone can no longer meet the new PCB design and manufacturing needs, more and more formats were created to provide supplemental information in separated files for manufacturing.

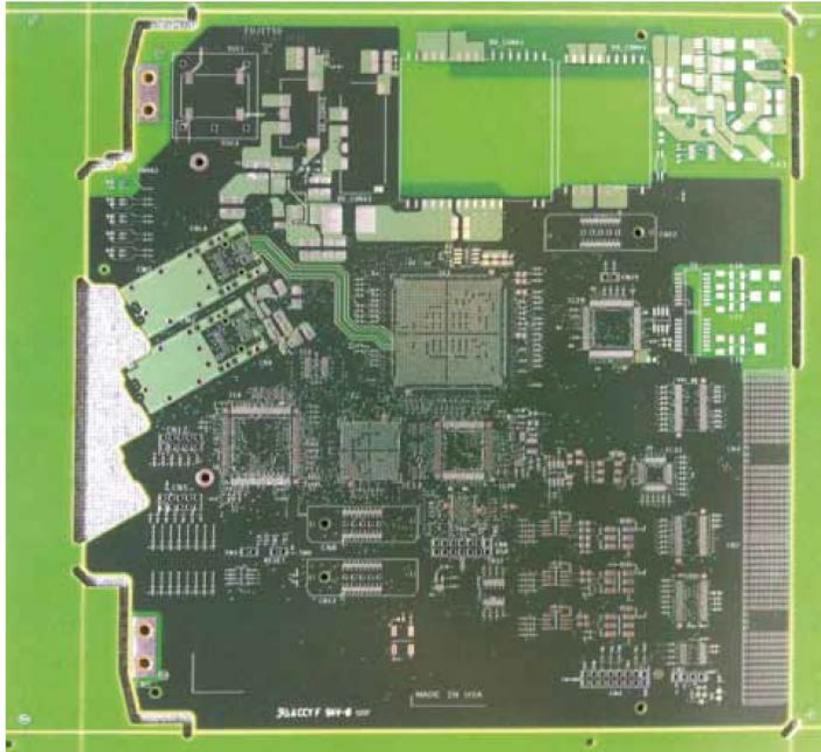


Image 1: First PCB fabricated using IPC-2581 – a 12 layer board – a board that is an active product being built by Fujitsu Network Communications, reducing the overall upfront fabrication time by 30%

The complexity of multiple formats and documents in transferring PCB design data raised the needs of a single and globally accepted data format, generated from EDA tools by the industry. Meanwhile, today's electronic industry is facing the challenges of miniaturization of electronic components and various manufacturing technologies with faster time to the market. A reformation of PCB design data transfer formats is critical to move the whole industry forward.

Many formats existing today were created by manufactures themselves to control equipment including photoplotters, drill/mill machines, assembly (insertion/pick –and –place), test, etc. However, these formats are only capable to meet the technology needs of 1980s. For example, the Gerber Scientific Corp., a photoplotter hardware manufacturer, created one of the industry's best-known de facto standards, Gerber RS-274-D. This format is still in use today, however todays photoplotter are no longer exposing analog film with moving aperture plates. Also, with compiled data enter into the modern digital imaging system, errors and deviations will be hard to eliminate from the data output from PCB tools, as well as from data explanation and translation of CAM tools. And any error or small deviation can lead to huge malfunctions in today's GHz range high speed PCB designs.

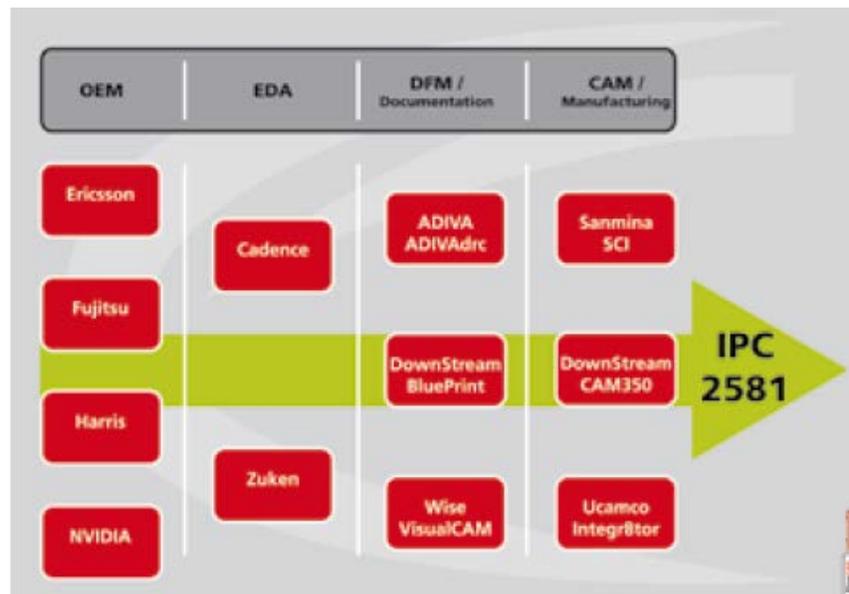


Image 2: Under the leadership of Cadence, a group of 40 companies take active part in promoting IPC-2581 in the whole industry. The consortium is driven by major OEM companies. Members also include PCB software vendors who provide solutions for PCB design and manufacturing.

There are still more information needed for the complete documentation of a PCB design, such as the layer structure, netlist, drilling and milling information. These data are not standardized and designers and manufactures exchange this information through agreed formats. So if for any reason the supplier changed, the existing data transfer format would be invalid. For normal PCB, it might work with the traditional way like Gerber and other format for design data transfer. However, if a new technology or application comes, traditional way will not fit the new needs. Thus, detailed explanation through drawings or text descriptions will be necessary, thus misinterpretation will be inevitable. So one of the goals of IPC-2581 is to eliminate the uncertainty and undocumented of multiple formats as much as possible, and make it a single, transparent global standard.

It always happens that the manufacturing data are inconsistent because of different design data transfer formats. One reason is that the manufacturing information is not connected to the design data and rely on the manual settings of EDA tools. In order to address the increasing

demand of more functional and reliable products, PCB designers are required to develop smaller, faster, higher capacity designs with lower cost, and fully leverage existing manufacturing technologies. Therefore, there is a desire from the industry of a data format which can include all the descriptions needed to fabricate and assemble a printed circuit board in one file. This one file should include information from Gerber Plot-Files, which are used as the template for etching copper structures on various layers, information about layer structure, sequence of the construction (cores), drilling, netlist name of test point and bills of materials, and furthermore, to include information to be distributed across corporate and national borders. All this information should be in one file.

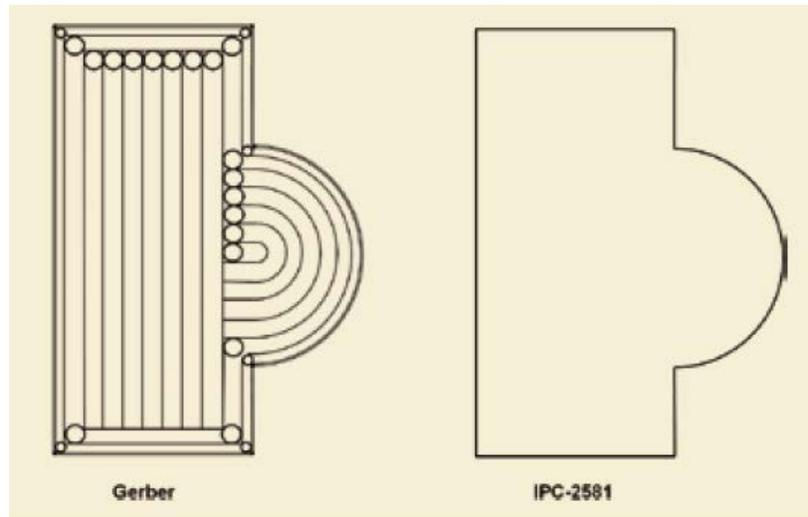


Image 3: Comparison of Printed Circuit Board preparation data between Gerber and IPC-2581

ODB++ was created by Valor Computerized Systems starting in the 1990s and promises to be an "intelligent" format that captures all information needed for assembly and fabrication. This format was then developed into two versions. One was becoming a proprietary format after Mentor's acquisition of Valor in 2010; the other version was developed by the industrial committee IPC to combine the best of ODB++X (no longer available) and GenCAM (IPC-2511) to a new standard IPC-2581. This new standard is an open, neutral, global standard to transfer PCB design data into manufacturing efficiently and comprehensively. For years there were many attempts to promote a new standard for efficient PCB data transfer. But most of these attempts have failed. The reason is because these data formats belong to individual companies, so they are not open and driven by the company's business needs. Thus, in order to promote the new standard in the whole industry, it is necessary the new standard is driven by a strong and neutral group.

The Consortium and the world first PCB fabricated by IPC-2581

Under the leadership of Cadence, a group of 40 companies take active part in promoting IPC-2581 in the whole industry. The consortium is driven by major OEM companies (Fujitsu, Intel, Ericsson, Harris, Lockheed Martin, Cisco, Velux, etc.) who are producing many complex PCBs in the world. They see the opportunities brought by IPC-2581 to improve efficiency and reduce

costs. Members also include PCB software vendors who provide solutions for PCB design and manufacturing.

On September 26, 2012, approximately one year after the establishment of the IPC-2581 consortium, the consortium built an optical plug-in module using IPC-2581 – a 12 layer board – a board that is an active product being built by Fujitsu Network Communications, reducing the overall upfront fabrication time by 30%.

Fujitsu exported the fabrication data contained within a single-file of the IPC-2581 format from the Cadence Allegro PCB Editor. The assembly pallet was constructed and the IPC-2581 data augmented and validated using VisualCAM from WISE Software. Finally, the bare board was fabricated by CC Electronics in the UK. This 12- layer bare board is a typical optical plug-in module consisting of BGAs, QFPs and SFPs, components rotated at odd angles, a series of complex milled cutouts, the use of split planes, positive and negative plane layers, high speed nets with controlled impedance, differential pair, and matched length constraints.

This industry's first PCB fabricated using IPC-2581 is a big milestone for the IPC-2581 Consortium. It allows data transfer smoothly in the CAD flow from design to manufacturing in one file. And finally this open, neutral standard will reach the goal of more efficiency, save cost and less errors in PCB manufacturing.

The data in IPC-2581 are stored in the modern XML structure. First of all, it allows rapid and targeted access to the desired information. Secondly, the format is flexible enough, so that it can be expanded according to future technologies needs. Meanwhile, the output of the information can be specified. It means PCB designers can choose which part of the information they want to transfer to their manufacturing partners and withhold the information they don't want to share. In this way, OEM companies can have a clear control of who get the information and what information they can get. And the file contains detailed production information including fabrication, assembly and test. The latest version of IPC-2581 is supported by Allegro PCB Designer 16.6 from Cadence Design Systems. There are also several free IPC-2581 viewers available to download provided by various software vendors. The format can describe following information: Copper structures each location, location construction, Build-up order, material information, Netlist, variant-specific assembly, Test points and other design intentions.

Cadence Design Systems and IPC-2581 Consortium

It is important for the industry to have an open, neutral, global standard for to transfer PCB design to manufacturing under the umbrella of IPC (Association Connecting Electronics Industries) as a recognized global working group association in the industry. The members from IPC-2581 consortium also represent the entire PCB Design and manufacturing supply chain (PCB Layout, DFM, CAM Aand test). Cadence commitment to IPC-2581 is also important to FlowCAD. And Cadence believes it is in the industry's best interest that an open, public, neutrally maintained standard be adopted by all segments of the PCB design, fabrication, assembly, and test supply-chain.