

Transferring Design Data to Manufacturing Using Open, Neutral, Global IPC-2581 Standard

Vijayakumar C Patil

Cadence Design Systems (India) Pvt Ltd,

2nd Floor, 3B Building, RMZ Ecospace, Sarjapur Outer Ring Road, Bangalore – 560103, India

vpatil@cadence.com

Abstract—Today, a surprisingly large percentage of PCB designs are sent to manufacturing through decades old approach of sending multiple files and formats. This approach can and does introduce many errors and ultimately adds extra cost through time spent on ensuring the data that is sent through multiple files and formats is consistent. In addition, there is cost incurred through scrap. Systems companies use this approach as they want to create internal processes that are not dependent on proprietary formats.

IPC-2581 provides the PCB industry with an opportunity to replace multiple files and formats with a neutral, open, globally supported standard and save millions of dollars caused by outdated numerous file formats used to drive PCB manufacturing. With the adoption of any standard, the supply chain has to adopt this standard from producers, consumers and supporters all using an open, neutral and efficient specification. It's the old chicken and egg scenario – which comes first? Solving this age old problem is the focus of a new consortium of PCB design and supply chain companies who have come together to enable, facilitate and drive the adoption and usage of IPC-2581.

This paper talks about the benefits of using IPC-2581 and why PCB design and supply chain companies should adopt it. It also discusses how the IPC2581 Consortium is accelerating adoption of this standard and how the consortium will help accelerate improvements to the format to encourage innovation, improve efficiency and reduce costs.

Keywords: IPC-2581, Manufacturing, formats, XML, pcb, supply chain, global

I. INTRODUCTION

We have been using printed circuit board (PCB) formats for manufacturing and assembly which were defined 30 years ago mainly for numerically controlled manufacturing units. With increased complexity of Printed Circuit Boards in terms of number of layers, number of pins / Vias, Miniaturization with ever decreasing form factor and sensitivity of traces due to Signal Integrity and timing aspects, the designer is in a nightmare situation between design and manufacturing stages of the design phase. A lot of efforts is required and the designers are worried until the physical board is working. This is an issue that the industry must resolve. Therefore Industry required an independent format that covered needs of today's and tomorrow's PCB designs. There emerged IPC-2581 in

March 2004 from the combination of GenCAM and ODB++. IPC-2581 is an independently developed and maintained format. IPC-2581 comes in XML representation. It contains PCB fabrication, PCB assembly and test data that manufacturer(s) would need for tooling, manufacturing, assembling.

II. DATA TO BE TRANSMITTED

Below Fig 1 indicates what a design engineer typically transfers to manufacturing

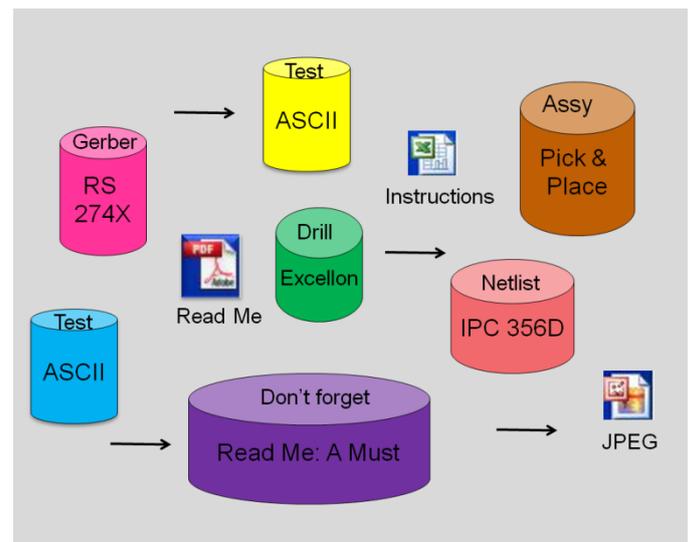


Fig 1. Typical PCB Manufacturing package

As you see in the Figure 1, so many different formatted files are used for different aspects to manufacture a PCB. While each of these formats like Excellon, Gerber etc have been proven for decades, manufacturing -“does require multiple inputs”- like fab-notes, drill details, design intent, and so on. Therefore we cannot manage with one or two inputs. There also there is no standard process to send all required data accurately in one go. When the designer has to deal with multiple files and formats, it becomes manual process to make sure no stone is un-turned. This adds up to design cycle and additional cost. Companies loose “time-to-

market window” and dollars. We all know in today’s competitive market being ahead and being competitive are critical to the success of the company.

III. IPC-2581

IPC-2581 is an Open, Neutral Industry Standard and a Global Industry Standard. It is also an “intelligent” data format. IPC-2581 is an amazing format because it embeds PCB assembly, manufacturing and test information in ONE SINGLE XML file.

Key advantages are: Ease of manufacturing data handling, No need to interpret data, no need to re-enter data from drawings, documents or emails. All companies in the design and manufacturing chain operate on the same set of data. Therefore collaborative communication can be leveraged between multiple companies in an organized manner. Lowered number of files being handled certainly increases operational efficiency and product quality. Companies using this format enable first pass success. Being a Neutral format there is no vendor dependency and thus risk free. Reduction in fabrication time can be achieved.

IV. FLEXIBILITY

The IPC-2581 provides five major “function modes” for data interface from the EDA tools to the various CAM tools. Every mode has further levels for granularity. Every level in every mode has well defined functions. In this manner a design data can be transmitted to manufacturing in a systematic manner covering all aspects in a single file.

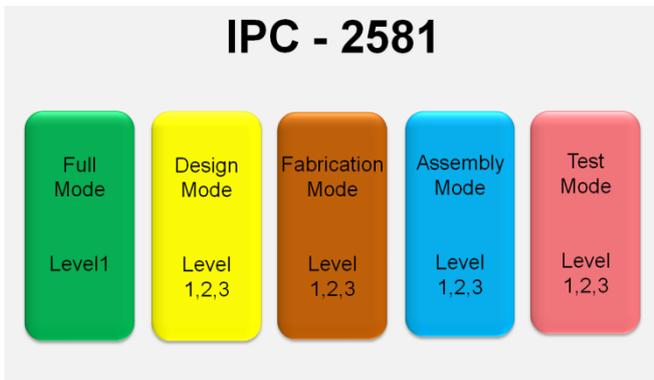


Fig. 2. Function Modes

FULL Mode is for complete information through 15 functions. DESIGN Mode is for details of design start to completion where manufacturing data is already produced, or design details where design is completed but manufacturing data is not yet produced. FABRICATION Mode - as the name indicates, it covers Fabrication related details. ASSEMBLY Mode is for Assembly related details. TEST Mode – contains bare board / assembled board (In-circuit) testing details /

Impedance information etc. In an EDA tool these Function modes can be accessed as shown in Fig 3.

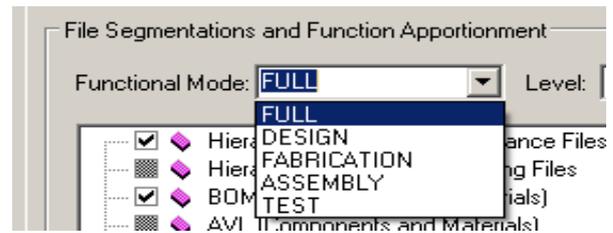


Fig. 3. EDA tool view for accessing Modes

Thus, classified Modes, Levels and detailed Functions enable designers to transfer data “as much as or as little as they need through selection process. In Fig 4. shows in an how designers can access these levels and functions and tick details to be transmitted in an EDA tool.

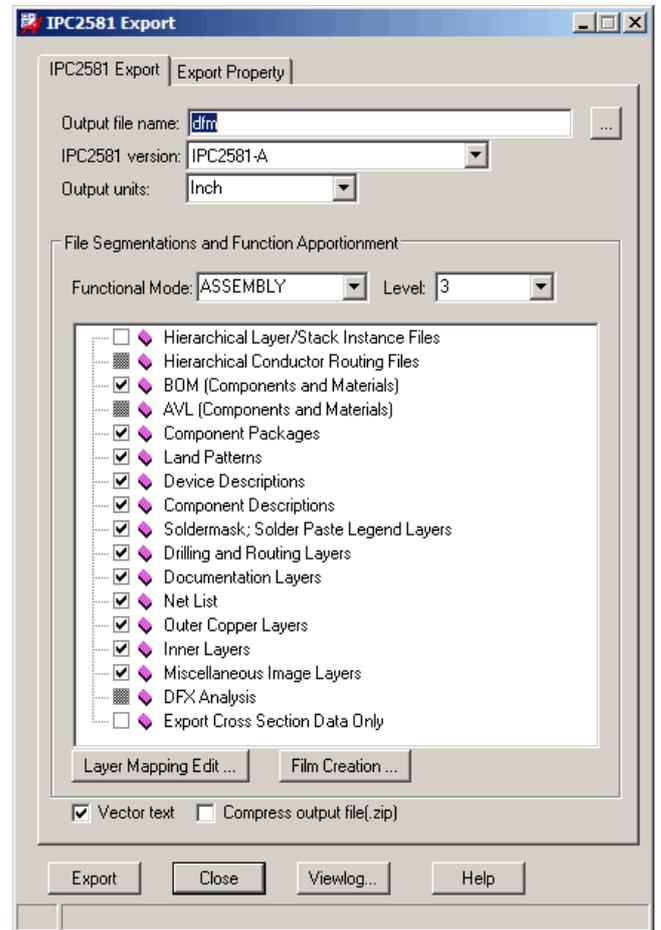


Fig 4. EDA tool view for choosing required functions

Now several EDA vendors have released software supporting IPC-2581. Free viewers are available

V. ABOUT IPC- 2581 CONSORTIUM

IPC-2581 Consortium consists of PCB design and supply chain companies. The consortium's goal is "to accelerate the adoption of IPC-2581 as an open, neutrally maintained global standard to encourage innovation, improve efficiency, and reduce costs."

VI. CONCLUSIONS

IPC-2581 replaces multi-file, multi-format design data delivery to manufacturing with a single XML file enabling "Do more with less" leading to better efficiency, improved quality, savings on design to manufacturing cycles and savings on costs.

ACKNOWLEDGMENT

The author would like to acknowledge Hemant Shah Cadence Design Systems Inc for his support while writing this paper.

REFERENCES

- [1] Keith Felton and Hemant Shah "Equipping the PCB Design and Supply Chain with 21st Century Data" pcd&f.com 28 July 2011
- [2] "IPC-2581 with Amendment 1" document from webstd.ipc.org
- [3] www.ipc2581.com



Vijayakumar C Patil received a Bachelor's degree in Engineering in Electronics and Communications from Gogte Institute of Technology, Karnataka, India. He is working with Cadence Design Systems (India) Pvt Ltd as Sales AE Director, India Technical Field Operations for Allegro®, Sigrity and OrCAD® technologies