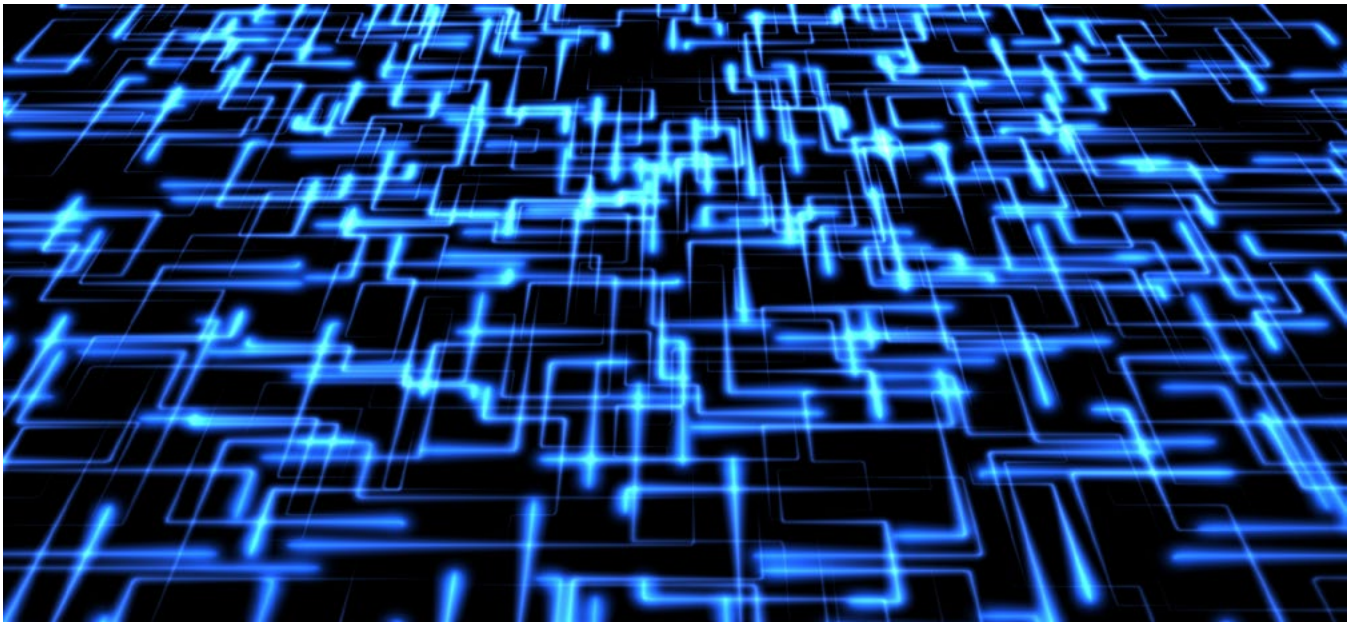
IPC-2581B Eases Stackup Development

by Amit Bahl
SIERRA CIRCUITS

PCB manufacturers often find that important details are unclear or missing from the designs that customers submit, and such projects can't move forward until the designers are reached for clarification. In many of those cases, it is the way design data are organized and conveyed that is ultimately to blame.

Whatever the EDA platforms that were used to create them, most PCB designs are output and sent to manufacturers as a collection of Gerber files that graphically define the layers, a drill file, a netlist, a board drawing including the stackup, and a readme text of notes and instructions. Turnkey orders for combined fabrication and assembly also include a BOM file and coordinate data for pick-and-place operations. There is no common data format among all the files: They're simply bundled together and forwarded to the manufacturer to download and interpret. And that disparity among formats can lead to omissions on the design side and miscommunication with manufacturers.

Now consider what occurs (or should occur) long before any complex design is completed and sent for fabrication. The initial design step leading to a successful PCB layout for any complex, high-speed circuit is close consultation with your manufacturer to determine the optimum stackup. The process is iterative, a back-and-forth collaboration to select the right materials; determine line widths, spacings, and layer thicknesses to meet impedance values; and minimize the number of layers and nail down the via set within budget. Here again, there's no common protocol for how that's accomplished. The process involves telephone calls, drawings, and emails until consensus. On one hand, there's stackup information the designer will use as the foundation for the layout. On the other hand, there's a detailed stackup from a manufacturing perspective, which describes how each layer will be constructed, and if the design will involve sequential laminations, how the layers will be grouped for fabrication.



IPC-2581 Advances

I am a strong advocate of the IPC-2581B unified file format, because it will simplify the transfer of PCB designs from CAD tools to board manufacturers, incorporate rich attributes that help thoroughly explain exactly what designers intend manufacturers to build, and ease stackup development. The standard aggregates all the elements of a design, every aspect from the stackup to assembly operations, in a single common format for transmission from the CAD platform to the manufacturer.

In my November 2013 column in The PCB Design Magazine, I described what my company experienced as the manufacturer of a test vehicle, a 12-layer network line card designed by Fujitsu Network Communications, which was sent to us as an IPC-2581A file (version A preceded the current version of the standard). We traced some minor anomalies to evaluation software supplied by our CAM vendor, which I understand have been corrected, but we proved the integrity of the IPC-2581A design file. Under the auspices of the IPC-2581 Consortium, which now numbers 60 member companies (see www.ipc2581.com), there have since been major enhancements to the standard and there is work underway to explore how it could integrate other elements to augment conventional PCB description and manufacture, such as device firmware.

The most significant new provision in the B version, from my vantage point as a manufacturer, is the ability to develop a stackup interactively with a designer. Gary Carter, one of the consortium founders and senior manager for CAD engineering at Fujitsu Network Communications, put this in perspective during a recent conversation.

"Material stackup is now supported in RevB. That's a big win in the early part of design, when designers spend a lot of time talking with their fabricator about the choice of materials to meet the design requirements they specified," Carter explained. "Today, it's the passing of napkins, telephone calls, and maybe at the end of the day a JPEG or PDF file from the fabricator following the dialog. Then they [the designers] have to hand-enter the particulars back into their CAD system."

> ***"Material stackup is now supported in RevB. That's a big win in the early part of design, when designers spend a lot of time talking with their fabricator about the choice of materials to meet the design requirements they specified."***

Carter continued, "With RevB, we've demonstrated the capability to move information from a CAD system into a SI tool that has some knowledge of materials and from there, back over to a fabricator, who can respond with suggestions. We can then go back into the same tool and verify that the attributes of the suggested materials do in fact yield the correct results, and then [seamlessly] go back into the CAD system to finalize the stackup, and off we go."

There is no paper involved, and the bidirectional exchanges between the designer and the manufacturer during the stackup development take place via the CAD tool that will render the design. The ability to develop and communicate design information in a consistent format in the same electronic medium from the very inception of a project at the stackup stage all the way through to the CAM equipment used for production ensures that no detail will be misplaced or misinterpreted. This is indeed a big win. **PCBDESIGN**

Amit Bahl directs sales and marketing at Sierra Circuits, a PCB manufacturer in Sunnyvale, California. He can be reached by clicking here.