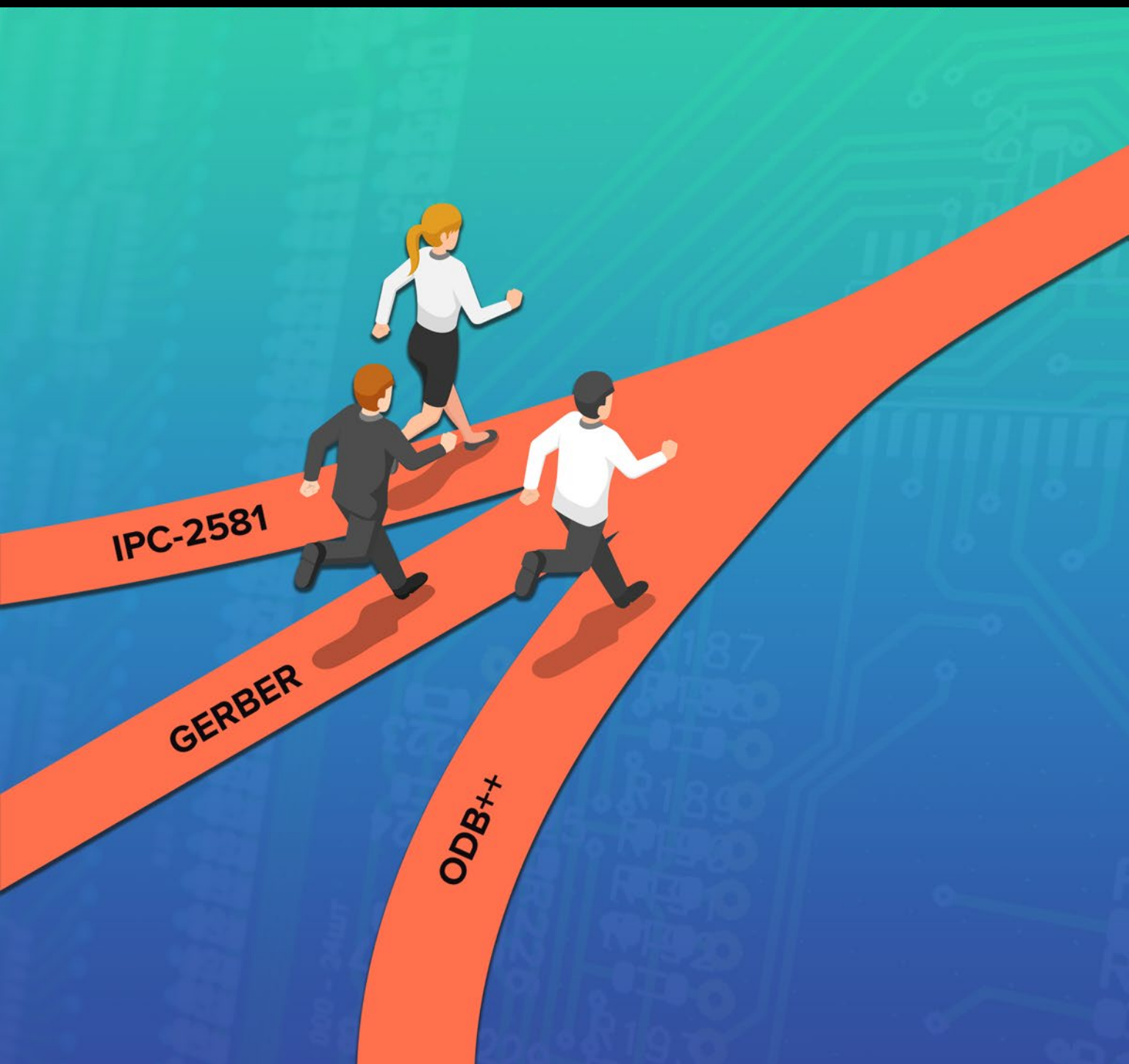
OCTOBER 2021

I-Connect007

DESIGN007 MAGAZINE



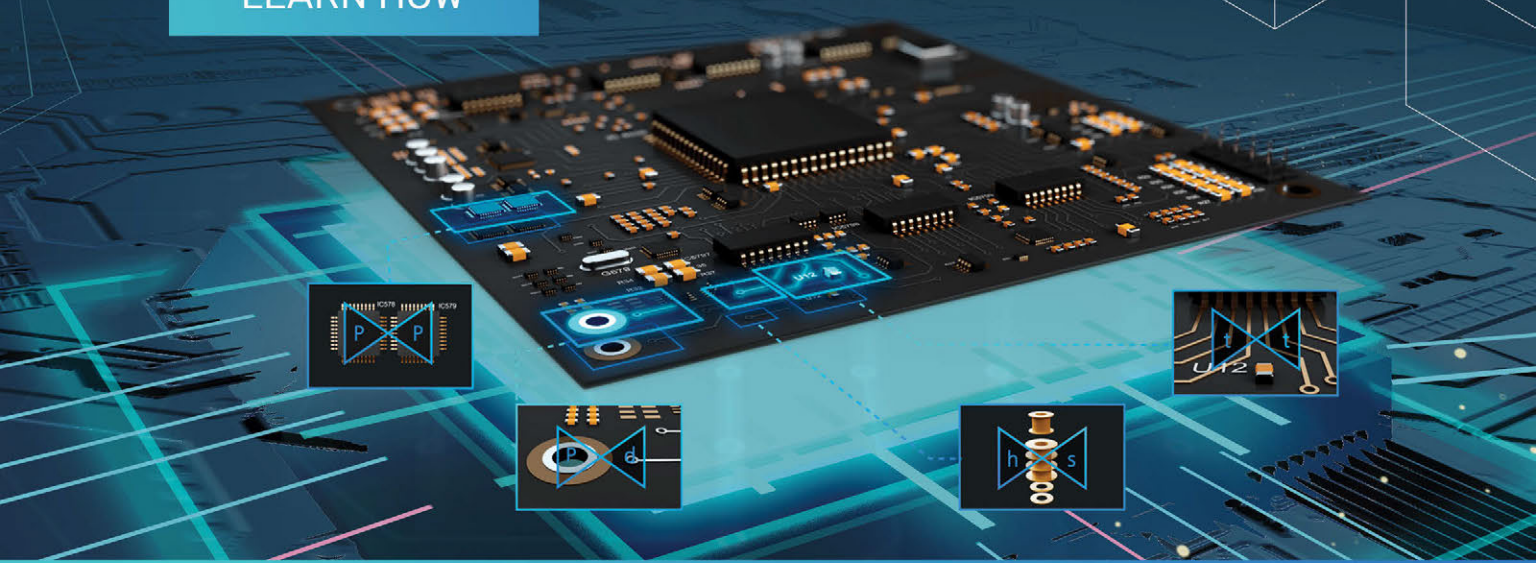
cadence®

GET EBOOK

Real-Time DFM with DesignTrue

Identify and correct DFM issues prior to releasing designs to manufacturing slashing turn time.

LEARN HOW



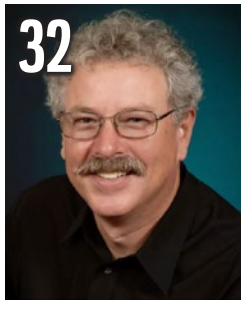
www.cadence.com

UNLEASH IMAGINTATION

DESIGN007 MAGAZINE

Design Data Formats

Gerber, ODB++, and IPC-2581 all have enthusiastic advocates, and all three are being used to produce circuit boards. Which brings us to these questions: Is there really a problem with having three perfectly good design data formats? Does the industry have to unite around one? Or do we all just like a good horse race? These are just a few of the questions that our contributors discuss in this issue.



PCBs are complex products which demand a significant amount of time, knowledge and effort to become reliable. As it should be, because they are used in products that we all rely on in our daily life. And we expect them to work. But how do they become reliable? And what determines reliability? Is it the copper thickness, or the IPC Class that decides?

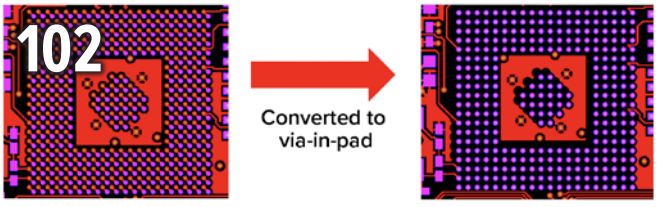
Every day we get questions like those. And we love it. We have more than 490 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

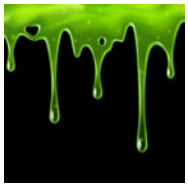
What's your PCB question?
www.ncabgroup.com/PCBquestion

Reliable answers. Reliable PCBs.

NCAB GROUP

DESIGN007 MAGAZINE

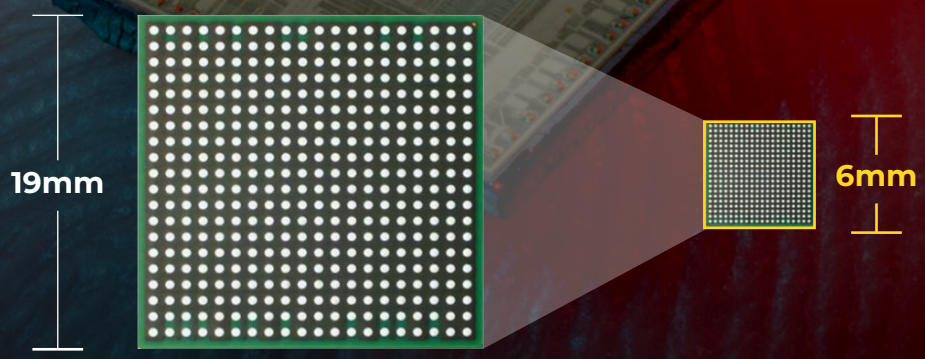




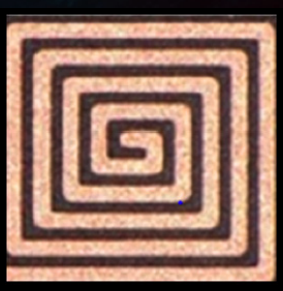
Smaller is better.

A-SAP™ Averatek semi-additive process offers a 90% reduction in size and weight over current state-of-the-art with 15 micron (.6 mm) trace and space

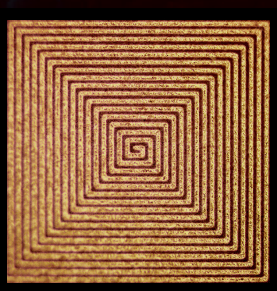
A-SAP™ enables ultra-high electronics density



A-SAP™ allows use of much smaller packages



Typical capability in the USA today: 75 microns



A-SAP™: 25 microns and below

More info on A-SAP™

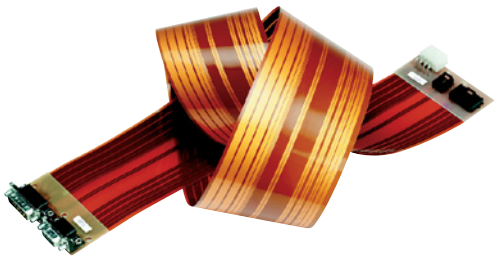
Averatek

www.averatek.com

FLEX007

This Month in Flex

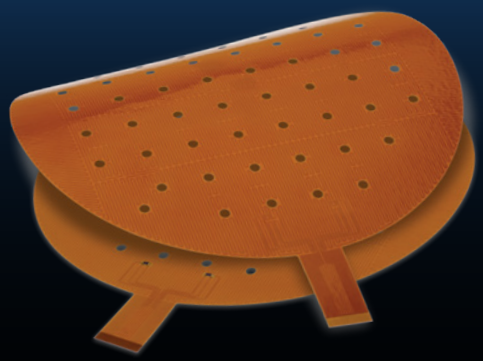
The typical challenges found in rigid board design pale in comparison to the issues designers face with flexible and rigid-flex circuits. This month, Pete Starkey discusses his RealTime with... interview with Anaya Vardya and David Lackey of American Standard Circuits, which focuses on flex and rigid-flex design and manufacturing. And columnist Joe Fjelstad explains why flexible circuits can be a "catalyst for technical evolution."



all flex

Flexible Circuits • Heaters • Assemblies

- Design application assistance
- Component assembly
- Fast response
- Quick delivery



All Flex Flexible Circuits & Heaters
1705 Cannon Lane
Northfield, MN 55057
(800) 959-0865
(507) 663-7162

allflexinc.com

Data Format: Designers Like What They Like

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

When we started planning this issue on design data transfer formats, we knew what we were getting into. While most of the topics we cover involve objective discussions with designers—i.e., "Here's the best way to do this"—any conversation about data formats is bound to be a subjective affair.

Like the trailer for the revenge movie says, "This time, it's personal." With data formats, as one veteran designer summed it up, "We like what we like."

Data formats have been a great source of disagreement and discussion for as long as I've been covering this industry.

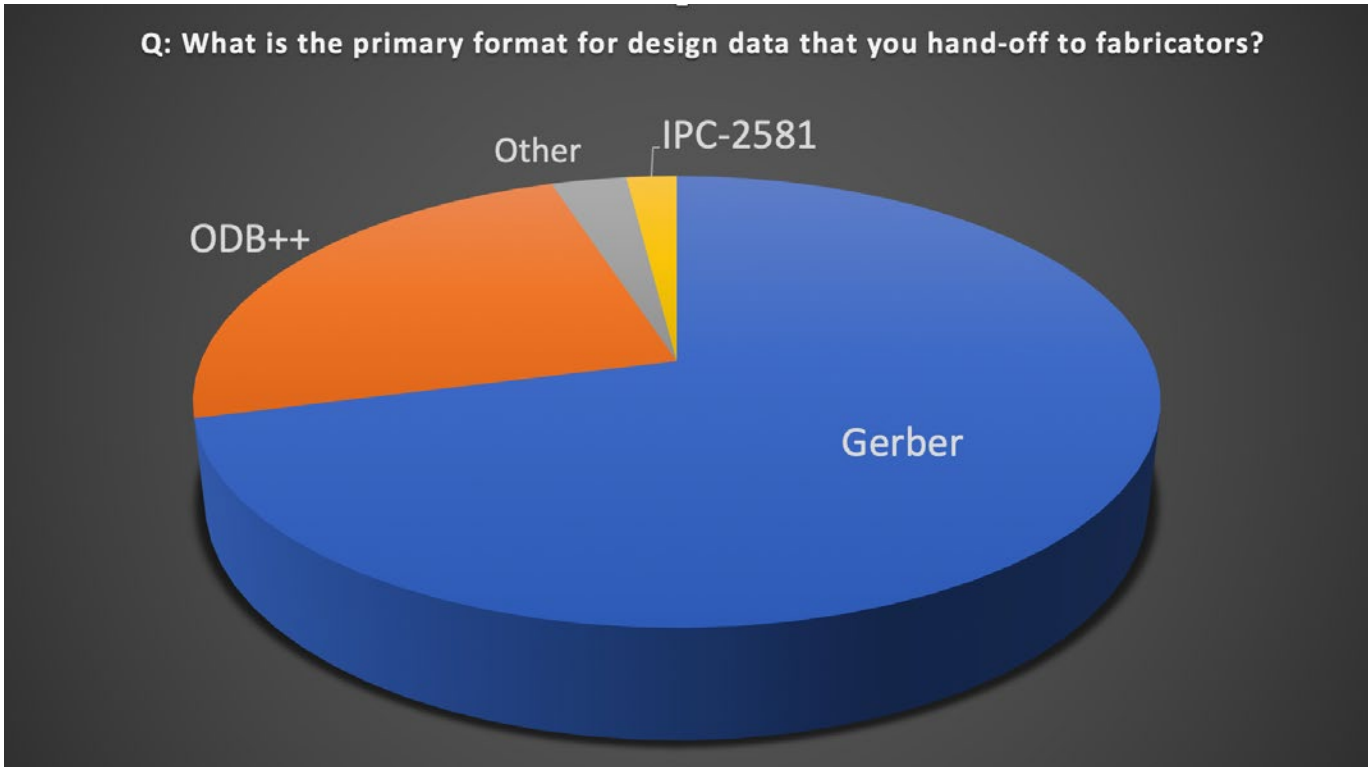


Figure 1: According to our designer survey, the overwhelming majority of designers use the Gerber file format.

Pro tip: If you're ever moderating a design panel at a trade show and the discussion is starting to lag, just mention Gerber, ODB++, and IPC-2581. You're guaranteed a half-hour of rambunctious conversation, complete with stories of how one format saved the day and that other guy's format almost sank a design project—nay, the entire company, possibly Western civilization.

In fact, the only really nasty emails I've ever received (not counting those from ex-wives) have been related to my reporting on data formats. In each case, I wasn't even championing one over the other; it's not my job to pick winners and losers. I was just passing on information. But some people come unglued when you say anything negative about their favorite data format, or anything positive about the other guy's favorite.

Without picking winners, I think we can agree that all three formats are perfectly capable of transferring your design intent to a fabricator and creating a PCB. So, we surveyed our designer readers about their chosen data format.

As you can see in Figure 1, it's Gerber's ball game all the way. That makes sense: The PCB design community is chock-full of people who have been designing boards for more than 35 years, and they've been using Gerber for their entire careers.

This group of designers is slowly aging out of the industry, and there's a chance that the next generation of designers will embrace the two "intelligent" formats, ODB++ and IPC-2581.

Each of these newer formats has its own set of advantages. ODB++, originally a Valor format, was designed to work with Valor CAM tools already in use by many fabricators and assembly providers. And IPC-2581 is now part of IPC's Connected Factory Exchange (CFX) flow, allowing design-through-assembly data transfer.

All three formats are currently being used to produce circuit boards. Which brings us to these questions: Is there really a problem with having three perfectly good design data formats? Does the industry have to unite around one? Or do we all just like a good horse race?

These are just a few of the questions that our contributors discuss in this issue. We start off with interviews with representatives of each format: Cadence's Ed Acheson of the IPC-2581 Consortium, Patrick McGoff of Siemens EDA for ODB++, and Ucamco's Karel Tavernier and Eurocircuits' Dirk Stans for Gerber. They all make their cases for their formats, with surprisingly little "smack talk" about the other formats. Matt Stevenson has a feature column on finding the value of Gerber files. Jen Kolar of Monsoon Solutions explains how her company decides which data format to select for each job. Hemant Shah and Patrick Davis delve into the advantages of IPC-2581, including being open and neutral. Gene Weiner discusses the role of tribal knowledge in the data format conversation. Sunny Patel of Candor Industries talks about the data formats that he sees each day, and we have another short piece by Matt Stevenson discussing the ins and outs of Gerber and ODB++.

We also have columns from regular contributors Barry Olney, Martyn Gaudion, Tara Dunn, John Coonrod, Kelly Dack, Beth Turner, and Joe Fjelstad. And this month, we're launching a new Siemens column, Digital Transformation, which will focus on the drive to digitize PCB design and manufacturing data.

It's almost time for SMTA International and PCB Carolina is right around the corner. I hope to see you all on the road. It's been too long. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 20 years. He can be reached by clicking here.

The Case for Gerber:

Interview with Karel Tavernier and Dirk Stans

Feature Interview by the I-Connect007 Editorial Team

There has been a lot of activity by the intelligent design data formats over the past year, and the newcomers (which are not really that new) are gaining users. But the overwhelming majority of PCB designs are still output in Gerber, a 60-year-old format that was never meant to convey PCB designs to manufacturers.

The I-Connect007 Editorial Team recently spoke with Karel Tavernier, managing director of Ucamco, which now takes care of the Gerber format; and Gerber advocate Dirk Stans, managing director of Eurocircuits. They discuss why Gerber continues to be the most popular format for PCB designers, the advantages it offers designers and fabricators, and what the future holds for this resilient format.

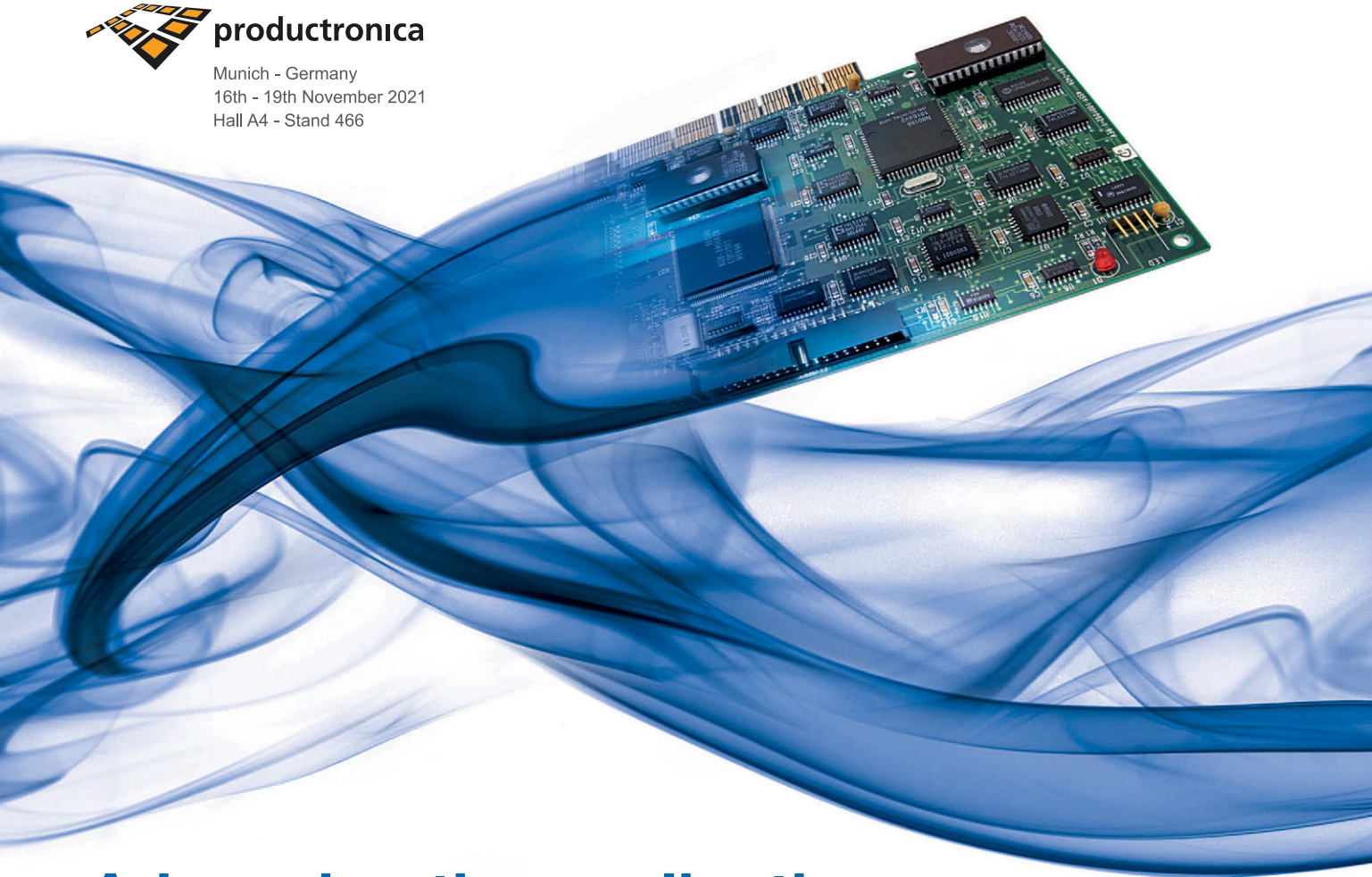
Andy Shaughnessy: Karel, would you walk us through the history of Gerber and the most recent revisions?

Karel Tavernier: Gerber goes back a long time. It was originally the input format of the Gerber line of photoplotters. It was a pure image format. There were a lot of photoplotter-specific variants. Gerber was very successful; many people adopted a Gerber format or variants of it as their image format. Ucamco took over Gerber Systems Corp. in 1998. At that time, we brought out Extended Gerber, which, essentially, was a unification and simplification of the Gerber family of image formats. A lot of machine-specific stuff was thrown out, and it became a simple and clear unified format. It was wildly successful, and everyone adopted it.

That's about the same time that ODB came out, around 1995. Extended Gerber came to be the dominant format—not that it was necessarily better or worse than ODB, but because it was simple to adopt and largely compatible with existing software. For a long time, nothing changed from the Gerber format. ODB added component information, and this was

productronica

Munich - Germany
16th - 19th November 2021
Hall A4 - Stand 466



Advancing the applications of conformal coatings

Our knowledge of the wider industry, combined with our ever advancing approaches to research and development, have created a range of Conformal Coatings with the necessary high performance and reliability required for the military, aerospace and commercial industries.

Although every electro-chemical solution is in our spectrum, our specialist approach to problems has established us as a key provider and developer to the technical electrical industry, both commercial and domestic.

Isn't it time you discovered out how Electrolube can serve you? Simply call, or visit our website.

Scan the code to discover our full spectrum of superior Conformal Coatings and electro-chemical solutions.

Tel: 800 474 1472

www.electrolube.com

ELECTROLUBE

MacDermid Alpha ELECTRONICS SOLUTIONS

Electronic & General Purpose Cleaning	Conformal Coatings	Encapsulation Resins	Thermal Management Solutions	Contact Lubricants	Maintenance & Service Aids

Karel Tavernier

a very positive thing. Around 2010, Ucamco again took up evolving the Gerber format.

In a first phase, we simplified it by throwing out everything that was very rarely or never used. Simplify before you extend. In a second phase, we added information that Gerber lacked as a CAD-to-CAM data transfer format. It was already a perfect image format but lacked design intent information; it did not contain the information about which drilled holes were vias or component holes, for example. Gerber X2 was brought out in 2014. Gerber X2 added extra information to the Gerber file so that from within the Gerber file you can see whether it's positive or negative, if it's a copper layer or a solder mask, which pads are component pads, and what's top or bottom, etc. This has been very widely adopted, and most Gerber files today are X2 files.

The reason for its quick adoption, again, is simplicity. The difficult part of output is the image; getting that right is difficult. For a CAD-vendor that already has an Extended Gerber output (and that is all of them), generating the extra X2 information is straightforward. If Gerber software outputs a top copper layer in Gerber, for example, chances are that software knows it's outputting the top copper layer; adding a line saying, "I am the top copper layer" is not rocket science.

Gerber X2 is compatible on input. The simplest is to ignore the X2 lines; then you have gained nothing, but it continues to work as before. Plenty of people receive Gerber X2 files without having updated software, and they do not even know it's an X2 file. They don't derive the benefit, but they have no disadvantage.

We had discussions with the group for IPC-2581 around that period. I had just read an article in *Design007 Magazine* about how IPC-2581 DPMX is great. However, it is difficult to adopt because you must implement a new image language, and that's very hard. This new image language brings nothing to the table. But DPMX had things that did not exist in Gerber at that time, and which were needed. I proposed that DPMX release a version that allows use of Gerber image files, but to add the new stuff in DPMX. Maybe somebody only wants to do stackup, describe finisher, etc. You must allow him to just implement that part. Alas, the DPMX core group only wanted to do full implementations, and that's very hard.

I think the non-existent acceptance of DPMX is mainly because the DPMX group has made it incredibly hard to adopt DPMX. Many companies that supply software in the industry are very small. When they must wrestle through a 1,000-page manual before they can start, they wonder, "When am I going to do that?" And they don't. So, this plan didn't work out. They chose a road that could not work. That's why we started the X2 project—in despair. I had hoped I wouldn't have to do it and that the whole IPC team would do it, but okay (laughs).

Later, there was the introduction of Gerber X3, and that is adding component information to Gerber files. Again, praise to the ODB people; they did this a long time ago, but in a rather complicated way. The neat thing about the implementation of components in Gerber is that it is so simple. It's syntactically and

semantically compatible with X2. Even if you have an old Extended Gerber reader, you'll still be able to read the components and derive some information. If you have an X2, you derive more information out of it. Of course, if you need component-specific functionality, you need to implement it. But for the reading and writing implementation, you do nothing. Anybody that has an X3 file can see it using the online viewer we have, and you see the component information. The component information was added with a lot of support and prodding from Eurocircuits.

The third step added was the Gerber job file, a separate file that contains the finishes, IPC-class, the thickness of the material stackup, etc. Its spec is a separate document, less than 20 pages. This job file is needed because the information, like finishes, is typically not even contained in the CAD system. It's somewhere in the ERP system written in Excel files or others. Adding this information deeply into a complicated image format makes no sense, because the people who write or read that information are ERP programmers, not CAD programmers. They can read files like the job file, and they are willing to read a 20-page spec. But if you are going to give them a 1,000-page spec pack describing all image files, etc., to wrestle through, they're not going to do it. I have people who have implemented and used Gerber job files internally because it's a convenient format to describe materials. This has been implemented by ERP programmers who have never seen a Gerber file in their lives and probably wish to die before they ever must.

So, in its current state, the Gerber format is complete. I think it contains everything that is needed for manufacturing, although not for design. Surely there will be things that aren't in the Gerber file that are in an ODB file, and vice versa. I mean, there will be a difference in the details, but all the major stuff is there. The benefit of Gerber is that it's done in a compatible manner. It's easy to understand. The Gerber file is an easy thing to implement.

Dirk Stans

You can make PCBs with Gerber and ODB, and you probably can make PCBs with DPMX, but that's not the point. The benefit of Gerber is simplicity and compatibility. A lot happened in the last 20 years.

Shaughnessy: Dirk, tell us about your point of view on Gerber as a fabricator.

Dirk Stans: We deal with a lot of prototypes and small series. Our job is making parts for many, many thousands of customers. We have about 12,000 active customers in Europe, and these customers have almost 20,000 active users using our system. These people, of course, use a big, big range of different CAD systems. We have a section about Gerber on our website explaining the history, through the latest format and all the benefits of it. It is that information that we take to designers who are using CAD systems.

In a CAD system, you think in electrical functionalities. On the other hand, the manufacturer is working with a CAM system, and he thinks from the mechanical side, "Can I make this?" There is a big discrepancy between those two worlds. You need to somehow bring

these two worlds together, not only in understanding what each of them is doing, but also making them talk the same language. If you have a fantastic CAD system with all colors, 3D dimensions, 3D views, and so on, then you need to convey this information to your manufacturer. There are hardly any means to do so. Most of these CAD systems have little to no knowledge whatsoever about decent DFM rules and doing design for manufacturability.

Most of these CAD systems have little to no knowledge whatsoever about decent DFM rules and doing design for manufacturability.

There is no real manufacturing experience in these systems. The idea that you can drive a printed circuit board manufacturing shop from a CAD system is ludicrous. I mean, there are people thinking that this is possible. I don't think that I will ever experience this in my life. It's because these people are living in a different world. A CAM system is needed. You need some translation from CAD to CAM, and my dream is to build a bridge between CAD and CAM and being able to cross this bridge in two directions. And why is that? Because in the prototype phase, you're not dealing with a purchaser. The purchaser has probably never seen a PCB board from a close range.

You're dealing with a designer, the guy who made the layout, and designers make mistakes. There's hardly any board where there is no mistake. Prototypes are also quick turnaround. We need them yesterday. Mistakes need to be cleared and corrected, and many times, the manufacturer corrects these mistakes. So, how do you get the corrections back to the designer? It would be very efficient if you could cross this bridge in two directions. The Gerber format is a nice tool for this.

For 30 years, we've made prototypes in PCB, and for the past four years in assembly; that's where it becomes even more complex. The world of PCB is very organized, but the world in assembly is a jungle. Nothing is organized. Not even formats are well specified. You can make your BOM in any way you like. As long as you understand what you're doing it's fine. As long as you stay in your CAD system, the world is wonderful. Then you output it, and nobody understands it anymore. We need standardization coming from within the CAD system.

For many years, I've been nagging Karel, saying, "Please make me X3," because at least then we have standardization about how to bring component information out of the CAD system. Some CAD systems have already adopted the component placement file (CPL); where components are concerned, that's dramatic. My idea is a big step forward. The only question is, "Do people implement it from the CAD side?" That is not an unimportant thing. If you look purely at our situation, we process over 100,000 PCB data sets for orders per year; about 10,000 of them are assembled. For every order there are at least five times the number of calculations and analyses made before that. People can do that freely on our website. The tools are available, free to use, for all Eurocircuits users. That means that, per day, at least some 2,000 data sets are processed by our tools. The underlying tools are Karel's software. We built our own platform on the Integrator.

Tavernier: An impressive platform, I would say.

Stans: Thank you. It's a PCB and PCBA visualizer, so you can visualize and virtually produce your PCB already, including the assembly, online, prior to ordering. It's free to use for Eurocircuits users. If I take this data per day, then the bulk that is supplied to us is Gerber X2, and some part in native EAGLE and native

EMC

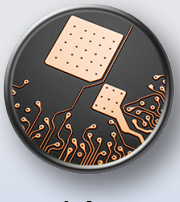
STRIVE FOR **EXCELLENCE**

www.emctw.com | EMC.INFS@mail.emctw.com

Enable 800GbE Ethernet Speed with EM-892K and EM-892K2 Materials

Industry's First Halogen-free Extreme Low Loss (ELL) Laminates

World's largest producer of environmentally-friendly advanced laminate materials



Key Material Features:
EM-892K @70%RC:
Dk: 2.84/ Df: 0.0017 @10GHz
EM-892K2 @70%RC:
Dk: 2.76/ Df: 0.0013 @10GHz
with Cavity Resonator Method

Lead-Free Soldering Test:
PASS LF-260°C/ 10X
with 32L 0.6mm hole pitch

Anti-CAF Test:
Pass 100V/ 85°C/ 85% RH
with 32L 0.6mm hole pitch
for 1K Hours

IST Test:
Pass Room temperature
to 150°C for 3K Cycles
with 32L 0.6mm hole pitch

TECHNICA, U.S.A.

North American Master Distributor
1-800-909-8697 • www.technica.com

Ki-CAD. Why native EAGLE and native Ki-CAD? Because they are also open, readable formats, and we have programmed our own input processor to transform this data automatically into the internal Ucamco format, the DPF format. We skip the Gerber translation in between and we go directly from EAGLE and Ki-CAD into the Ucamco format, which of course is an advantage, because then you don't have this translation step. But from these 2,000 files per day, if I look at ODB here in Europe, I think 10 to 20 times a month I get such a file, whereas from the other, I get 2,000 a day. It's this kind of relation.

Why is that? I think it's because many people we are dealing with, the bulk of the market, are not using the big CAD licenses. The big CAD companies are big in number of dollars, but they are not big in number of licenses they have in the market. The number of licenses in the market used by a huge number of small designers are smaller CAD systems, and they don't have these fancy output processors that Karel is describing; they just make Gerber, Gerber X1, and now Gerber X2. That's about it. That's the reason we get this. For EAGLE and Ki-CAD, we solved the matter altogether, so then we don't need it anymore. That's where we are with our prototypes and small series business in Europe.

Shaughnessy: You do accept ODB++, also.

Stans: Yes. But we would get ODB++, let's say, 10–20 times a month. Everyone knows that you can't misuse Gerber, because if you send us a Gerber data set, the data set is only for one board, for one version of the board, and that's it. You're never doubtful. If you start misusing the ODB container as a filing cabinet, and you keep version number one and version number two of the board, in finalized form and non-finalized form, if you send that to me, the first question I ask is, "Which of the stuff in there should I make for you?" It gets messy because people have the freedom to start fiddling around in it. But you don't have that freedom with a one-to-one clear output for Gerber. No discussion. It's simple for everybody. If you want me to make three different boards then you need three data sets. If you want me to make three versions of the same board, then send me three data sets. Don't send it all in one big ODB container where I must answer a riddle and find the weapon that killed the colonel (laughs).

Tavernier: Actually, currently, Gerber X3 is complete. I have a few attributes that I would like to add, but that will depend on somebody who is interested in implementing it. I don't like writing a spec that is not implemented. As Confucius said, "You only understand the spec if you have implemented it." And also, if it's not used, it just clutters the spec. There's nothing in the Gerber spec that is unused, and I want to keep it that way.

Stans: There is one thing missing, my friend.

Tavernier: What is it?

Stans: That's my long-time dream, to put a D behind it and make it X3D. You could include the STEP files, but that is up to the CAD systems. CAD libraries are usually poorly maintained, and you can only output what's in them. If there is no stuff in it, you can't output it. But that would be, really, a step forward. We are moving toward electronics manufacturing, meaning we'll do the PCB, the assembly, and by the end of the year, we will start building the enclosure, the foils, and do everything in-house. With X3D, we could then do complete virtual manufacturing at the beginning of the process, doing a full DFM of all these factors that will follow later in the value chain like enclosure, foil, assembly, everything.

That would be a big benefit, because a mistake that you correct at the beginning of the value chain is a very cheap mistake. A mistake that you need to correct at the end is a very expensive one. And that is the whole idea

behind it, doing things right the first time. I can only contribute for what I have the information for. But that really comes down to efficient CAD libraries.

Nolan Johnson: Dirk, you said something I want to expand on. We've already discussed CAD tools and whether they support the formats, but you touched on CAD libraries as an issue just now. Can you talk more about that?

Stans: It all starts with what the designer has in his CAD library. Where does his information come from? Does it come from a spec or from something that he downloaded on the internet? What is the quality of his information that he puts in his database? We see many mistakes. It's rare that we have a job for which we need to make the board and assemble it, that does not have a mistake in there. These mistakes are very, let's say, down-to-earth mistakes, such as non-matching footprints. The customer designs the board, and says, "For this manufacturing part number, that's the footprint," in his CAD system. Then we do the DFM. We countercheck this with this manufacturing part number in our database and then the footprint doesn't match. Subsequently you have an anomaly, and you need to find the problem.

In many cases, the customer's footprint is wrong. Why? Because in many cases, we've seen this footprint already 10 times before, and that's a daily practice. If you don't do the virtual manufacturing before you start manufacturing, then you get a situation in which the EMS companies order the part based on the customer's data, the board comes in, the component comes in, and then they say, "Hey, it doesn't fit." Now we have a problem.

Johnson: So, that's a problem with selection of the footprint in the CAD system? The file export, the Gerber file format itself, is accurately representing what the user put in their database.

Stans: Yes.

Johnson: It's not a file problem as much as it is a parts quality problem?

Stans: Exactly.

Tavernier: In the implementation of Gerbers, you see errors in the attributes. For instance, if the shape is a little bit more complicated than a rectangular shape, it is sometimes put in the parts library as different flashes that are overlapping, or you would get the same thing in Gerber output, and it's not even the fault of the Gerber format or the CAD software. It's just the way the part has been constructed. More errors come from database—user—errors than from weaknesses in the CAD software.

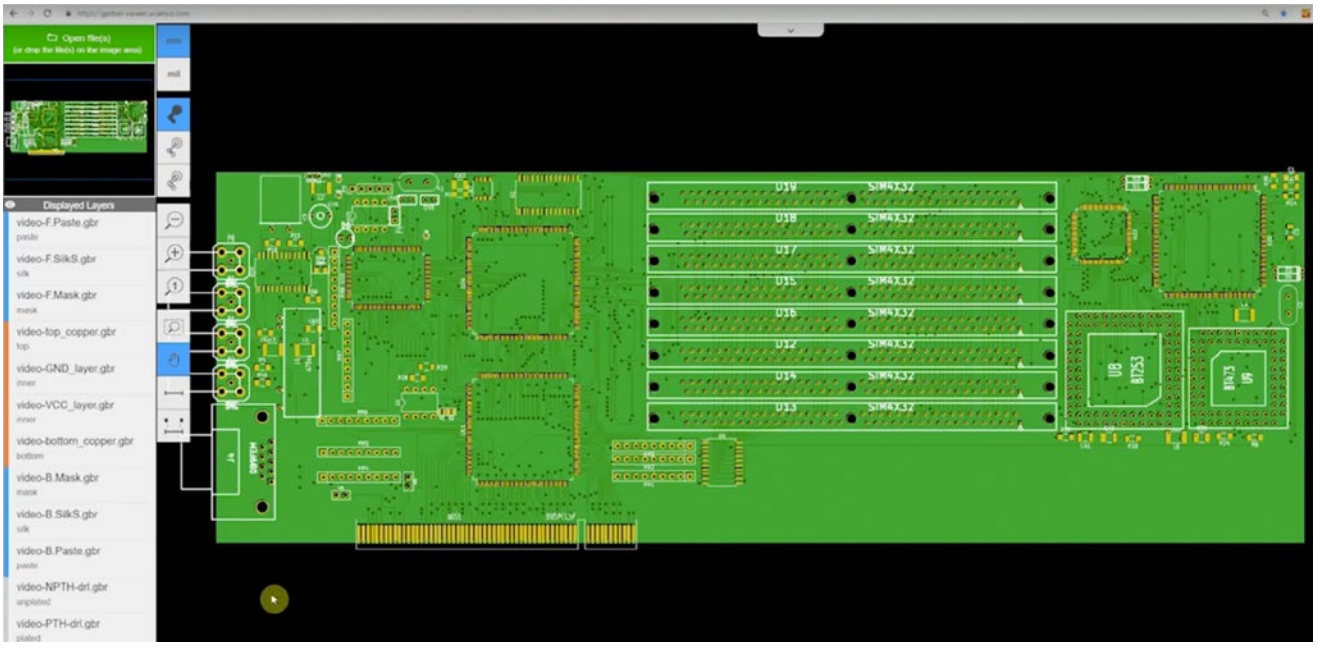
More errors come from database—user—errors than from weaknesses in the CAD software.

Stans: For assembly, the real drama is that the IPC, which has existed for a long time, never succeeded to convince the component manufacturers to create one joint worldwide accessible database with all the components in it. The whole world is forced to do this work repeatedly, and that means mistakes are made repeatedly. That is a huge part of it.

Johnson: Using an example, it's as if the word processors that we all use refused to supply fonts and made all of us make our own.

Tavernier: That's a good analogy.

Stans: It's exactly that. It's a scandal that component manufacturers are not willing to supply the data component information in one standardized, unified database, accessible by everybody. It's a scandal. But it is what it is.



Ucamco provides a free online reference Gerber viewer.

Shaughnessy: We're seeing more young people come into the industry now, fortunately, after about 20 years of not very many new people. Do you think these new people will be more inclined to use any of the more supposedly hipper formats? ODB or 2581? They say they're intelligent, bi-directional, digital from the start. Do you think it's a generational thing? Do you think that Gerber might fall off over time? What do you think?

Tavernier: I think it won't fall off, because the other formats don't bring material benefits, in a sense; there is nothing in the ODB file for fabrication or assembly that is not in Gerber. Furthermore, you can put more in an ODB file, as Dirk mentioned. ODB is the Genesis CAM format, and it's not really designed for transferring data between CAD and CAM. So, you have a lot of stuff in there, but in data transfer, it gets in the way. For data transfer between CAD and CAM, there's nothing in an ODB file that you won't find in a recent generation Gerber file. Why would you abandon Gerber, except for political reasons? ODB is now owned by Siemens, which is a formidable company. This may play a role if there is a fight between ODB and Gerber. But in the end, it is the software developer community that decides and that is a whole host of small companies. You give a programmer a DPMX spec pack and a Gerber spec pack and he'll hesitate 10 milliseconds for what's he going to use. I don't think Gerber will ever go away. I think it's to the contrary.

Stans: If you look back 30 years, you had specialization. You had a designer making schematics and then the next one you had a components specialist; the other one was a layout specialist, and he came closer to the manufacturing side as well, and then you had an embedded programmer. But now, these young electronics guys need to do everything. Deep specialization goes away, and they would rather tell us, "Tell me what to do and we will do it. Tell me what material we need to use for this and this and we will use it. Give me a standard and I will use the standard."

Today's designers need to do too many things. We see these same questions every day in our chat rooms. People are asking the same stupid questions they did 20 years ago. There is no difference.

Tavernier: I can concur. We are always, with the Gerber format, very open in trying to engage

with people and get input. You send an email, and you get a response. But if you discuss this with CAD programmers, you may say, "We could do it this way, and we could do it that way." But they say, "I don't care, just tell me what I have to do." They often don't want to discuss how. They must implement something, and to do that they must write the Gerber file, and they don't want to spend time on discussing how it the spec should be; they just want to implement.

Stans: Yeah, and they don't have to make their money with a fancy output; they make their money with fancy new tools of calculating impedance or whatever they need to have in that CAD system. It's the same with Altium. They talked about it for years, and now they have this department especially oriented to creating a decent output, for which I'm very grateful. Finally. Because that's what you need to make your CAD system more performant toward manufacturing. But it took a very, very long time before people saw the light in this. It's not sexy. You don't sell any CAD systems with this.

Tavernier: Gerber is the PDF of the electronics industry. Those PDF files, do you really care what's inside of them? When you send the PDF file, the other side should be able to read it, and you don't bother whether it's this or that, you only vaguely bother with which version it is.

Shaughnessy: That's a great way to wrap it up, gentlemen. Thanks for you input on this.

Tavernier: Thank you, Andy. Always a pleasure.

DESIGN007

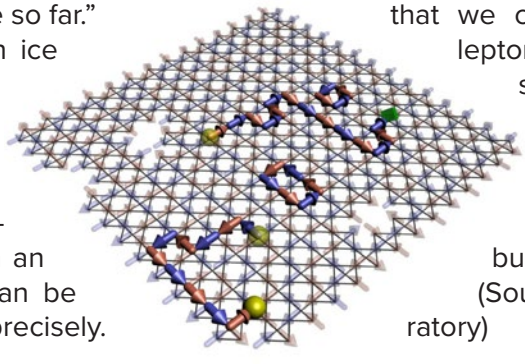
Karel Tavernier is managing director at Ucamco.
Dirk Stans is manager partner at Eurocircuits N.V.

Emergent Magnetic Monopoles Isolated Using Quantum-Annealing Computer

Using a D-Wave quantum-annealing computer as a testbed, scientists at Los Alamos National Laboratory have shown that it is possible to isolate so-called emergent magnetic monopoles, a class of quasiparticles, creating a new approach to developing "materials by design."

"We wanted to study emergent magnetic monopoles by exploiting the collective dynamics of qubits," said Cristiano Nisoli, a lead Los Alamos author of the study. "Magnetic monopoles, as elementary particles with only one magnetic pole, have been hypothesized by many, and famously by Dirac, but have proved elusive so far."

They realized an artificial spin ice by using the superconducting qubits of the quantum machine as a magnetic building block. This work demonstrates unambiguously that magnetic monopoles not only can emerge from an underlying spin structure, but can be controlled, isolated and studied precisely.



"These results also have technological consequences particularly relevant to DOE and Los Alamos, specifically in the idea of materials-by-design, to produce future nanomagnets that might show advanced and desirable functionality for sensing and computation," noted Alejandro Lopez-Bezanilla of Los Alamos, who works on the D-Wave processor and assembled the team.

Nisoli, moreover, suggests that beside fruitful applications, these results could perhaps also provide food for thought to fundamental physics.

"Could even 'real' particles and interactions that we consider fundamental, such as leptons and quarks, instead be construed as an emergent, higher-level description of a more complex lower-level binary substratum, much like our monopoles emerging from a bunch of qubits?"

(Source: Los Alamos National Laboratory)

The Case for ODB++:

Interview with Pat McGoff and Max Clark

Feature Interview by the I-Connect007 Editorial Team

To many, ODB++ still seems like a new format, but it has been around in one form or another for decades. ODB++ is gaining users every year, and has clinched a strong second place behind Gerber—first among intelligent formats.

To learn more about ODB++, we spoke with Pat McGoff and Max Clark of Siemens Digital Industries Software, two developers who have been working on the format for years. In this interview, they explain how ODB++ works, why they believe designers and manufacturers should switch to this format, and what's next for ODB++.

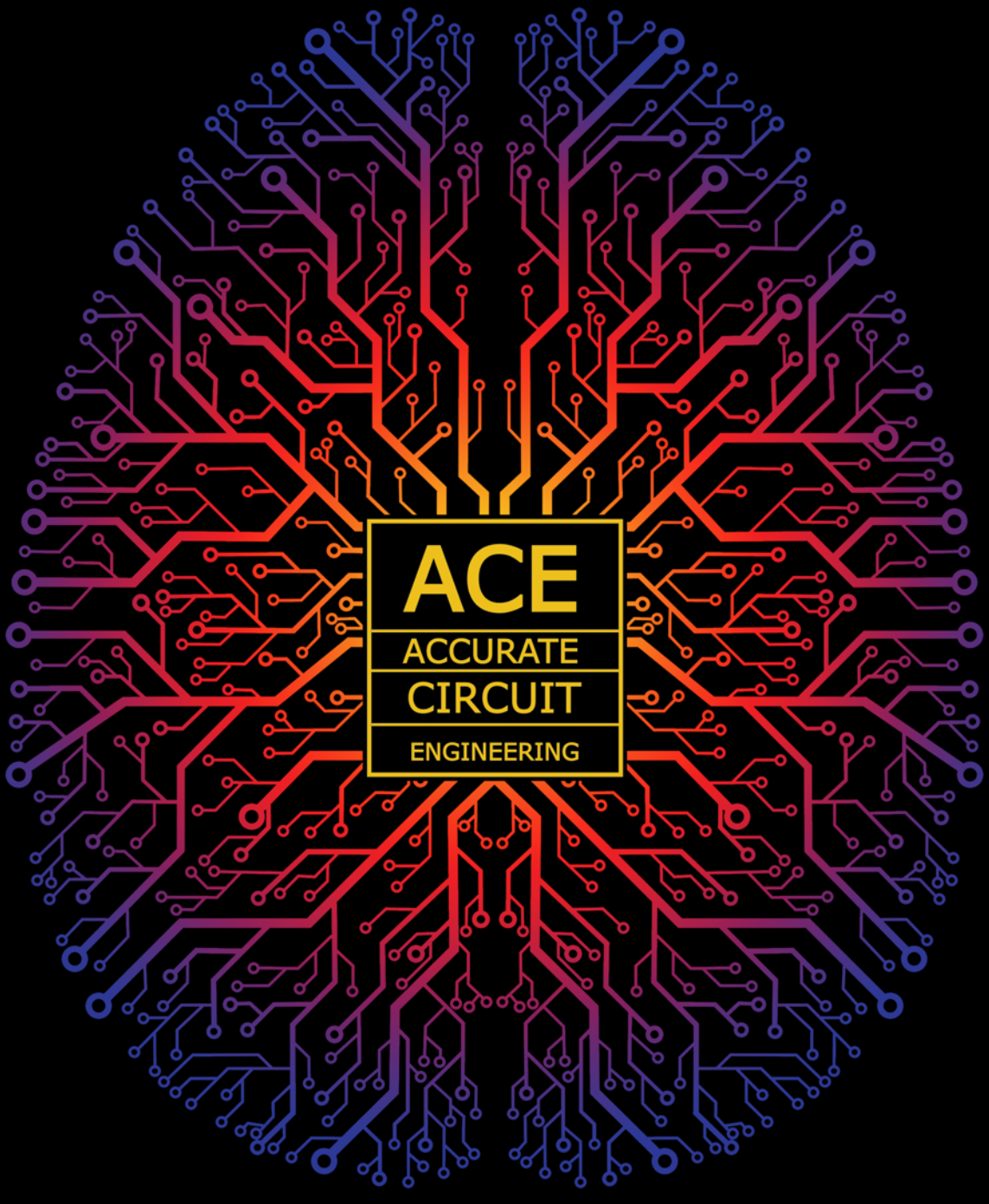
Andy Shaughnessy: Pat, would you give us a brief background about ODB++ and how it got to this point?

Pat McGoff: ODB++ originated in 1995, so the industry has been using it for 26 years. I think that is noteworthy. It's not the new kid on the block; it's not unproven. It absolutely is proven. The important thing is that the industry has determined its value over the course of 26 years. The customers decide what tools and formats are best for them. Nothing can be dictated or pushed on them, and they'll decide over time if this is going to serve them best. Then, it's like water flowing downhill.

ODB++ today is not the same as what you thought of ODB++ in the past. Before, ODB++ was a singular format; today, it's part of a family of formats. A year ago, we announced the family of ODB++. We've broken it down into ODB++ Design, ODB++ Process, and ODB++ Manufacturing. Just to give you a simple explanation, ODB++ Design was formerly known as just ODB++. That's the format that used to communicate the full manufacturing product model from design to manufacturing.

ODB++ Process is a format to provide common communications for driving PCB assembly equipment, to take data from a CAM system and drive the SMT equipment directly. The third one is ODB++ Manufacturing, and that's the protocol for the collection of data

Don't Overthink Your Prototype Builds



Go with the team that can make your ideas a reality...

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS

ACE
ACCURATE
CIRCUIT
ENGINEERING

Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:
James Hofer
(714) 227-9540

www.ace-pcb.com


Pat McGoff

from the shop floor, so you've got a common standard of how to collect it and aggregate it from the wide variety of both automated machines and manual processes.

Dan Feinberg: ODB++ as it is today combines the ability to view both process and manufacturing, correct?

McGoff: Yes. I'm trying to be clear and call it a family. There are three different formats within the family, so I think that's the gist of what you're interested in, but I want everybody to know that the family of formats, of ODB++, has expanded beyond just that one.

Nolan Johnson: If you don't mind, can I attach some categories to this? Design is going to be the design data. Process is going to capture more of the assembly and manufacturer instructions—how to put it together. Manufacturing, then, is basically capturing the as-built data.

McGoff: That's a very good way to put it, Nolan. With that said, obviously, the format has evolved. It started with the origins of Valor. Max was employee number three for Valor here in the U.S., and I was employee number nine. For the past 14 years now, we both have been directly responsible for the format. Clearly, Max is on the technical side, and me on the marketing and customer support side, but we consider ourselves stewards of the format. It's something that Valor originated, Mentor took ownership of it when they acquired us in 2010, and Siemens took ownership of it when they acquired Mentor in 2017. It's perpetuated. As a matter of fact, you can go to the ODB++ website and see that we currently have more than 73,000 users registered on the ODB++ website. That's a very significant number. It clearly is the highest number of users of any intelligent format you're going to see in the industry, so that shows how broadly it's being used.

Equally important is we have 80 software companies signed up on our partners page, committed by contract to support it. That's important, as being software companies, they are the tool suppliers—the people who make the ecosystem work. I emphasize this because of these 73,000 users, maybe that represents 10,000 or 20,000 companies. What's important for any format to be successful and have value for the industry is that the tools in the ecosystem must support the format because if a format runs into a dead end anywhere in the process of design through manufacturing, then it's just that, a dead end. It's not serving the flow completely. Then, you would have to complement that format with additional data in a different format to complete your process. You're not going to find any other formats out there in the industry that are supported by as many software companies in the ecosystem as ODB++ has.

Max Clark: I could add a little bit on the technical side to that. On a regular basis, we receive inquiries about the format from partners who are trying to understand how something is to be supported. I wanted you guys to know that we do get those types of questions.

McGoff: We manage and track this. Obviously, it's a responsibility, and we don't take it lightly. I have a staff, my technical marketing engineers are responsible for supporting it, and of course our web team maintains the content and keeps it current. But we track it to make sure that we're getting the visibility, as well as providing the support that the customers are requiring. On average, we get approximately 2,500 visits per month to the website, and more than 600 new users per month. On top of that, which dovetails with what I just said, we get more than 600 downloads of the ODB++ Viewer each month. These are good, solid numbers.

Feinberg: I would think those are significant numbers. I think you're being modest, but is that a trend that has increased or decreased over the last few months or years?

McGoff: Actually, the trendline is almost linear. It's not exponential, but it is upward in a slow, steady rate.

Feinberg: Which is what you'd want, really.

McGoff: Yes, but we would like a faster one. We've been doing this for quite a few years. We have new versions of the Viewer, and we have new versions of the ODB++ Inside application. When we announced those, we got some spikes, but if you did the run rate on it, it's been a nice steady increase for about 10 years.

I just mentioned the ODB++ Inside application. This is the application that we write for the Cadence Allegro users. It's something that Valor committed to do for Cadence back in 1995, and we've been doing it ever since. It allows the Allegro users to convert their designs into ODB++ with correct format support and current version support for Allegro. On that one, we get more than 200 downloads per month, so again, not a bad number. We have 200 Cadence customers downloading ODB++ every month.

I think it's safe to say that ODB++ is the most widely used intelligent data exchange format in the industry. Now, we're doing all this and have always done all this free of charge for the industry's benefit. The ODB++ specification can be downloaded for free, the ODB++ Viewer is free, the ODB++ Inside application is free, the technical support for software developers that Max spoke about is free, and the technical support even for all these 73,000 users of the

Viewer and the ODB++ Inside is free, handled by my team of TME's that I mentioned. There's nothing about the ODB++ format that costs anybody any money.

Feinberg: That's an important point, Pat, because that isn't necessarily the understanding throughout the industry.

McGoff: Yes. You know how misunderstandings go. Anybody who has a question about it, I'm always free to discuss it directly. I can't stop people from having misunderstandings, but I can make sure that when we have the chance to communicate, that they know things clearly.

I can't stop people from having misunderstandings, but I can make sure that when we have the chance to communicate, that they know things clearly.

Clark: Yes. Even the use of the document is drastically different than in some other cases where developers are able to share the document internally and even copy and paste directly out of the ODB++ format specification. That's not the same in all cases, so we're very developer friendly as well when it comes to supporting ODB++.

McGoff: Furthermore, we try to be as open as we can for ideas. Max is brilliant and he knows a lot about where the industry needs to go and almost all the requests come from our customers, but these software partners, all 80 of them, by registering as partners, have a special portal that allows them to submit ideas for enhancing the format. They might have a test software application, and they might even be a direct competitor to Siemens, but they can submit a request for enhancement of format that might benefit their application and their customers by doing so. We take that into consideration. Customers can submit enhancement requests as well. We get them regularly.

Shaughnessy: You're serving two different masters here, because your tools have to work with any format.

McGoff: Yes, that's correct. We're a tools vendor that happen to be stewards of a format.

Shaughnessy: I bet it must get interesting sometimes.

McGoff: Everyone has opinions and thoughts and we're not going to stop them, but all businesses are driving to be efficient. Their ecosystem extends well beyond their own four walls. They look at it and say, "In this globalized, outsourced economy, with all the dynamics that have been in play, especially in the past 18 months, how do we assure that we have the most efficient processes in place and our supply chain is streamlined?" For that, we want to say, "Great. My common denominator is the most widely used and the most comprehensive data exchange format, and it has the least issues within my supply chain."

Shaughnessy: Why should somebody consider switching to ODB++? What are the biggest advantages?

McGoff: Here's a story that will help explain why. Before joining Valor, I spent 10 years at Gerber Scientific and worked indirectly for Mr. Gerber. I have strong familiarity with the Gerber format, its origins, and when 274X came about. A little quick side trivia for you: Do you know who invented 274X?

Shaughnessy: I have no idea.

McGoff: Two individuals. Ed O'Reilly was the

software development manager at Gerber, and Randy Allen was the CAM front-end manager at AT&T, Richmond, Virginia.

Feinberg: I'm familiar with that facility. I spent an awful lot of time there. It's now a shopping mall.

McGoff: Yes. Randy Allen approached Gerber in the late 1980s and said, "Is there any way we can embed an aperture wheel and polarity indicators for the layers into the Gerber format?" He and Ed O'Reilly got together in South Windsor, Connecticut (because there wasn't email back then), and they exchanged ideas. They had a two-day meeting, and they designed 274X.

More trivia for you. 274X was never adopted as an industry standard. There was no industry committee or board that ever approved it. It was just those two individuals who launched it because of the popularity and respect, or credit given to Gerber, and it was adopted. It's still today the most widely used format of all even though it's not intelligent.

When you were talking about the legacy formats, we all shortcut our words to be fast, and we say we send the Gerber files. Well, the Gerber files phrase means about six different formats, six different deliverables. You've got the Gerber files for the layers, the photoplotter files; you've got the Excellon drill files, test files, netlist files, centroid data, and then you have drawings and documents and the bill of materials for the job as well. This package that's delivered constitutes multiple content in different formats, file structures; some of it's human-readable only, like a drawing file or instructions. All must then be sent to the suppliers, who reverse-engineers it to make it a

product model going back the other way, so they can then do their value-add, which is the tooling for manufacturing.

Here's my Excel analogy. Imagine that all of us on this call were working on a project, and it required a spreadsheet. I start it and I create my spreadsheet with its multiple tabs. I've got formatting and formulas and illustrations in it, font, and calculations, and whatever you want to add to it. After I'm done with my part of it, I would say, "Andy, your turn," but instead of sending you the Excel file, I send you a Word doc or a text file that instructs you how to recreate it yourself, and cut and paste and create your Excel file from what I send you in that text file. How happy would you be? That's the same thing the industry is doing by sending legacy data to their suppliers.

The suppliers are always receptive to do whatever they need to do to get a customer's business, so try to picture any salesperson in any industry that says "no" quickly. They don't.

They're handcuffing them, saying, "I'm making you be inefficient." Now, the suppliers are always receptive to do whatever they need to do to get a customer's business, so try to picture any salesperson in any industry that says "no" quickly. They don't. It's not their job and you would fire them if they did say "no" quickly. The status quo remains in effect because of the commercial impact. Whether they charge a different NRE for it is a different story, but they're always going to say yes.

Feinberg: This is true.

McGoff: Thanks. We consider it our mission to help customers see the light and prove it out, and we do this constantly. I'm not allowed to say the name of the U.S. chip company we did this with, but we engaged in the process of showing them the inefficiencies. They went through a six-month process with their suppliers, getting them onboard, exchanging some pilot data and doing it in parallel with the legacy data package. Then, they saw no downside after all that was fleshed out, and for eight years now they've exclusively used ODB++.

Shaughnessy: Are you noticing that younger people coming into the industry are a little bit more interested in embracing ODB than the "graybeards" with 40 years under their belt? Are you noticing any trends like that?

McGoff: Andy, we are not collecting any kind of age records when people register for the Viewer or anything else. I can't say this empirically, but when we had our initial meeting with that IC company that we worked with to convert to ODB++, I brought along two different props for show and tell. I brought an eye loupe, for those of you that remember eye loupes, and a Gerber aperture wheel. In the conference room at the customer's site, I think the average age was 30.

I went around the room and let them pass the props around, and then I asked them if they knew what they were. None of them could tell me what either of the props were. I told them that you may be doing leading-edge technology, but you're limited by this aperture wheel you're holding. They asked me to explain, which I did. I said, "Everything is based on draws, flashes, and floods on this format. There's data that has no bearing on the ICs you're producing today, and all the handicaps that go with that." We've got new people on our staff that helped us invigorate in ways we weren't thinking about, so maybe that will bleed through on the format discussions, too.

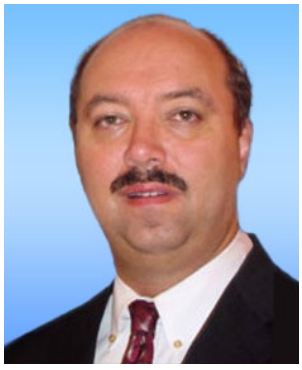
Shaughnessy: I was wondering about ODB++ Manufacturing. Is this more or less comparable to the DPMX CFX flow?

McGoff: Yes. Of course, we would suggest that it's more comprehensive, and it's coordinated and easier to keep in sync. We've got customers using all three elements, so it's widely used. Again, this is something that our customers have asked for, and we've delivered. It's not the only game in town on any of these, but when you create it and customers adopt it, you owe it to them to continue and enhance it and develop it for their needs.

Shaughnessy: Yeah. Well, what's next for ODB? Where do you go from here?

Clark: The ODB++ format is driven based upon what our users come to us with, as well as what technology is coming out. There are needs that we've recently addressed, and you might say, "Wow, I'm surprised these haven't been addressed years ago." Some things like intentional shorts being transmitted from design into manufacturing. Intentional shorts to this day are documented on a document layer, and then no one finds that. Next the manufacturer calls and say, "Hey, do you know that signal one and signal two are shorted together?" "Yeah, we do. It says it on the document." "Oh, we didn't see it." That's the kind of messaging you get there.

We added the intentional shorts, and obviously, the idea is that everything you add will be utilized by the software products on each end or throughout the flow. That goes regardless of whatever we add, so you have the intentional shorts we've included in stackup information. We included stackup zones for rigid-


Max Clark

flex designs. The different stackup zones indicate where there are different stackups. Then you also have, for each of those layers, different shapes, so you have different outlines or profiles for each of the individual layers. There is a lot of design-to-manufacturing content to make use of in ODB++ today.

Those are probably the most recent enhancements that we put inside the application, so the intentional shorts, complete stackup, and rigid and rigid-flex designs can all now be passed through ODB++. Previously, we added things like backdrills. I know it's been around for a while now, but it's still relatively new technology, so we added the ability to transmit backdrills.

We added characteristics like the plated rout and the IPC via types. On pads there is an indication pertaining to whether a solder mask opening is defining the pad. This is important because you don't want to alter the size of the solder mask opening during fabrication, the fabricator wants to know that this pad's size is being defined by that solder mask without question. Wire bonding information, basic dimensioning we've added, drill directions, and whatnot. Those are, within the last one to two years, the types of content that we've added based upon our customers' feedback to us.

We are now working with customers to add content that has become more technology-based, especially in embedded components. When we start to investigate the embedded components closely, we see components now coming out with leads on both sides. They have leads on the top, leads on the bottom, and you come down to them and kind of sandwich the component in between the layers. We're in the process of defining the support for double-sided components. Then we're also going to update the format because it doesn't support edge components where you have a component on the top and on the bottom and it slides over the edge of the component. We need to support this.

We support screen components right now, but not to the degree that I'd like to see us do it. It's a little bit older technology, but for embedded resistors and embedded capacitors I believe we should improve upon how we're conveying that information as well as for partially embedded components—a component that is on layer, say, three, but exposed through the top of the board. Then, of course, you have stacked components after that. How do you convey it if the components are stacked? We can already do this with the format, but I want to clarify it a little bit more in the next release of ODB++. I believe you can see we always consider how we could intelligently and effectively convey manufacturing requirements using ODB++.

Shaughnessy: Yes, that's good.

Clark: In everything that we do, this typically comes from our customer base. We're not just sitting around trying to decide what our customers would want. We look at what the industry trends are, we take their input, and we work to implement based upon that need.

McGoff: To that point, let me just make a comment. I think one of the reasons ODB++ has been as successful as it has been, is because of how quickly we deliver capabilities in the format that our customers and the market requires. You're speaking to the two people responsible, and again, Max carrying the heavy part of the load here, when we see and hear our customers, especially the big ones that we're all familiar with, say, "Can you add this? Can you do this?" And when we understand why, we're able to make the decisions pretty quickly, yay or nay, and how quickly we can work it in.

They appreciate that speed in getting solutions that the industry needs.

Clark: Another reason why ODB++ is widely adopted, from a developer's viewpoint, is in the way the ODB++ format is managed. Anytime there is a minor or update release of ODB++, it has to remain backward compatible to the originating major release. A software product, if the developer developed their software wisely—we can't guarantee that, but wisely—and we add new functionality, new structure to the format, if they simply ignore that new content and structure, their products will continue to function.

That's very important, otherwise, every time there is a format update you have to be concerned and wait for full adoption. The ODB++ version 8 now has been out for about four or five years, and I can take an ODB++ file that's generated on our current version of application or out of another ODB++ supported application and read it back into older versions of our Valor NPI and it would just work because we guarantee that backward compatibility at the format level. Once I go to, say, ODB++ version 9, then all gloves are off. That's when we make our most drastic changes, and then my life of having to live with that format and maintain its backward compatibility continues. We try to keep that kind of a change limited because we know that it's time-consuming and costly for the industry to turn on a dime just because there is a new revision of a format.

Feinberg: That's so true, and I wish a lot of people in the computer industry and the software industry would take that to heart.

Clark: Right. It always surprises people when I take a file that I just generated in today's application, I start up a version of our product that might be five years old, and it just goes in. That's what we strive for. If we find something that's broken, then we have to try to figure out what went wrong. We consider that a failure.

It's something that we view as very important and when we define the format, we must take that into very careful consideration. As Pat said, when we look at something for a customer, whether it's something that we could do, it's not always just that we can't do it. It's just that we can't do it without breaking the format, so we have to wait. I could give you an example: components with leads on both sides. The ODB++ format doesn't support that today. As soon as we put component leads at the top, all the generations of software out there will break. All software providers will need to address having leads on both sides of the board or at least support that when ODB++ is encountered. That type of introduction needs to be carefully considered with the industry at large.

In Valor products, we will always be able to support current and one-down revision of ODB++ that we can continue to work with what we currently have as the format and don't break the downward exchange of ODB++ because that would be devastating. That's why we always support at least one version of ODB++ downward because we know that people are going to run into this.

McGoff: Current version minus one.

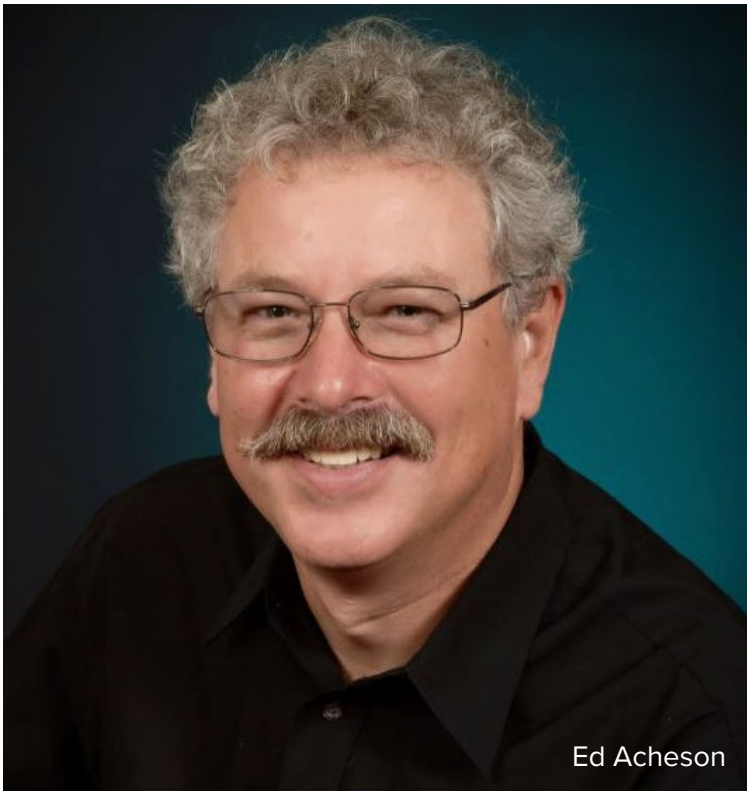
Clark: Right, that's our standard.

Shaughnessy: This has been very insightful.

McGoff: Thank you, Andy. DESIGN007

Patrick McGoff is market development manager, and **Max Clark** is business unit manager, at Siemens Digital Industries Software.

The Case for IPC-2581: Interview With Ed Acheson


Ed Acheson

Feature Interview by the I-Connect007 Editorial Team

Over the last few years, IPC-2581 has hit several milestones: Revision C was released in late 2020 and it now includes complete build intent for rigid-flex circuits. It is also integrated with IPC's Connected Factory Exchange assembly format.

To learn more, the I-Connect007 Editorial Team spoke with Ed Acheson, a senior principal product engineer with Cadence Design Systems. Ed is also one of the developers behind the IPC-2581 design data transfer format. He walks us through the ins and outs of IPC-2581, and explains why he believes this open-source format could be just what PCB designers and fabricators need today and tomorrow.

Andy Shaughnessy: Ed, would you give us a basic background on IPC-2581 and how it developed out of the previous format efforts?

Ed Acheson: Back in the late 1990s, there was an effort to form a standard format for exchanging data from electronic CAD systems into manufacturing. At the time, there were a couple of formats that were fairly common. One was Gerber, and the other was ODB. The idea of having an IPC-recognized standard was to form something that everybody could work with. There would be nothing proprietary about it, and the idea was to make it more intelligent than the original format. People started thinking, "I could use this to archive data for an entire design." As they were going through it, they said, "Okay, where do we start?" Well, you start with the GenCAM format and try to merge a couple other formats into it.

They spoke to the people at Valor and worked with them to get ODBX, which is the XLM version of ODB and the base that was the starting point for having an XML formatted file. The nice part about XML, they realized, is its being an extensible language that you can build on. You start with basic elements, and as new features and new technology changes, it's very easy to adopt it into that format. They released the first version of it, version A, but it wasn't too popular. Nobody could really see an advan-

Our capabilities are simply out of this world.

We manufacture printed circuit boards for a wide range of advanced applications using just about any type of substrate currently available.

EXPLORE OUR CAPABILITIES

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com | (630) 639-5444

tage to using and adopting it as much as they wanted to make changes to it. With the lack of interest, it went by the wayside and was lost.

Then, in 2010, Valor was acquired by Mentor Graphics, which raised concern among non-Mentor users. They were concerned that ODB, a proprietary format for Valor, and which was considered independent, would now be controlled by Mentor Graphics. They were worried that Mentor would focus on its own customer base more than it would anyone else.

So, an effort was put forth to create the IPC-2581 Consortium, which is celebrating its 10th year in 2021. Driven by Hemant Shah, the 2581 Consortium began with about 10 members, and now is well over 100 members. When I became involved in 2011, we asked, "What can we do to 2581 to make it more acceptable in the industry? What data is missing, and what has been done to validate the data that was output at that time?" We spent a great amount of effort between various tool providers like Cadence, Downstream, Wise, Adiva, and Zuken. We all created 2581 output and compared the data outputs against each other. We learned that in formats, if you don't explicitly state something, it's like a Rorschach inkblot test where everyone sees something different. With that, we realized that we needed to have a consensus on how things should be and standardize everything.

We saw the benefit of the XML format and its ability to add more and more intelligent data, so that not only am I creating output like outlines, drill holes, and artwork layers, but now I can start embedding some BOM information. I can put in my company part number for a part and link it to manufacturers' part numbers. I can add attributes to objects (which we call specs), where I can designate something specific in the format that must have particular attention paid to it—something like a chamfered edge, or a plated edge. We began to build and add more data into it until we got to what we see today.

Shaughnessy: I know IPC-2581 has been growing. What are the advantages? Why should someone be motivated to switch to 2581?

Acheson: Number one, it's an open format. I don't have to buy a license to use it. I can actually create software around it so I can embed or extract certain pieces of information from it. That's a big advantage, especially if I'm an enterprise type of company that has the technology to do this. It's also a one-file database; I can output everything I need in one file. I don't have to worry about synchronizing multiple files and trying to merge them together to understand an end result. It's all there in one file. It's not binary. It's XML.

Shaughnessy: We did a survey recently asking what would motivate any of the designer readers to switch to another data format, and the number one thing they said was, "Simplifying the process." They said they don't care about cost.

Dan Feinberg: They lie (laughs).

Shaughnessy: Designers wouldn't lie (laughs).

Acheson: For users of a lot of the tools, and I can speak on Cadence Allegro and OrCAD, it is a simple process. You still go through the typical steps of saying, "Here are my layer structures, this is what makes the top layer, and this is what makes the bottom layer." But when I go to output the data, I simply select where I want the file to be saved, what function mode I want, I hit my generate button, and the file is done.

Feinberg: Ed, I have a question about IPC-2581 being an open format. You can use it without paying IPC anything, but what happens with the software? Let's say you create something that's really unique and valuable. Do you own it, or is it then open for anyone else?

Acheson: If you create the software, it's up to you if you want to give it away or sell it.

Feinberg: Do you have unique rights to it?

Acheson: Yes.

Feinberg: Anything you've got rights to, you can assign the right to others or open it up. But I just wanted to know that if you came up with something great for it that works with an open-source standard, you own it. That's important, I think.

Acheson: Yes.

Happy Holden: In your opinion, why does everyone continue to use Gerber? Is it because they've always done it that way, or they don't want to learn about 2581? Do they have false impressions that it's too complicated or costly?

Acheson: I think the most conservative people in the world are board designers. If you're going to make a change, what the heck's going to happen if I make this change downstream of my process? If it's not broken, why should I fix it? The realization of where problems may lie is here: If I create an artwork and my origin for my artwork is a lower-left corner of the board, now I create an NC drill file. The origin for that is the tooling hole in the upper right corner of the board. I bring that into a CAM system and now I must align those files. There's a good chance for human error if someone is not paying attention to the database accuracies and so forth. So, you can create some issues down that process.

There's a good chance for human error if someone is not paying attention to the database accuracies and so forth.

What does that mean to have a single source file with all that data contained within it? It means you've got the same origin source, the same database units, and the same level of accuracy. The other part of being slow to adopt is the fabricators. They're limited to the number of tools that they have, and the time to read the 2581 data. They're charged for the adapters to do that, so there's a cost factor for the fabricators.

For a lot of them, they've written decades of scripts around their hand tools, and to introduce a new format to them means more work without realizing that it's really not a lot more work. So, they hesitate. It's easier for them to say, "No, we don't support it." What has been and must happen is that larger corporations are now understanding both how adaptive and reliable IPC-2581 is to their environment. They are now telling fabricators, "By such and such date, the only information you're going to get from us is 2581. If you don't adapt, we're going to go somewhere else." Now we're starting to see a trend in the fabrication side of adopting it, so they can maintain their customer database.

Shaughnessy: I've been wondering why fabricators don't just say, "We're only accepting one of the intelligent formats from now on."

Acheson: If you look at Axiom, they created a policy that is unique in the industry: If you send us intelligent data, it's this price; if you send us unintelligent data, aka Gerber, it's going to cost you more. They're seeing a huge increase in people sending them intelligent data.

Nolan Johnson: That's an interesting situation because when we surveyed the designers, their response was, "I want to send what's easy, and I send what my manufacturing guy tells me to send." I don't want to sound like using Gerber is a problem, but the adoption of intelligent data has a lot to do with what seems to be inertia at the fab.

Acheson: Gerber is a good format. It's tried and true, and failure is very rare. But as data, on the unintelligent side, I need a bunch of different pieces to get intelligence out of it. That's where 2581 comes in; all that intelligence is there at once.

Johnson: Ed, if I can make this analogy, you'll get it, and Dan will too. Gerber is basically structured like Unix from the '60s.

Shaughnessy: Right. If you liked Unix, you didn't want to switch to Windows.

Acheson: Yes, that's true.

Shaughnessy: One of the respondents in our survey said, "We output in all three formats and hope for the best." Really? I assume that's because they don't know where it's going to be fabricated.

Acheson: Early on in our surveys, when the consortium was promoting the adoption of IPC-2581, we did an audience poll and asked, "How many of you in the group use strictly Gerber?" A good percentage, I'd say 60 or 70%, raised their hands. "How many of you send ODB++?" The remaining 30% raised their hand. They said, "Of those of you who send ODB, how many of you send Gerber with it?" Of that 30%, 95% raised their hands. Whenever there was a conflict, guess what the answer was? Use Gerber.

Shaughnessy: Does it seem to be a generational thing? Do you think the young people, these digital natives coming into the industry, are going to drive the adoption of "intelligent" data formats like 2581 and ODB++?

Acheson: Yes, I think they will. We know the board designer generation is getting older, and the old "I don't like change" people are slowly stepping out of the scene. What we're seeing in our consortium is the younger people are more adamant about using it because of what they can have in it. The depth of understanding and the education comes through the different pushes that we do at the trade shows, the different things we do in articles to promote it, and why it is better for the industry. We're seeing a lot of traction in moving toward adoption of it.

Holden: Our columnist Dana Korf is a big supporter. But Michael Ford, who's a big name in assembly, is pushing assembly-level IPC software that is going to be based on IPC-2581. My hope is that it pushes it over the hill because you're going to need to use 2581.

Acheson: Yes, there are a couple things that will push the adoption of 2581. One, as you mentioned, is CFX, the Connected Factory Exchange. CFX uses 2581 as the descriptor of that model. In fact, 2581 is being looked at as a digital twin of the physical board. So, the data for physical boards is based on what's in 2581. On top of the ability to push the assembly data, and the details needed for the CFX, in Revision C we're introducing a standard

format to communicate between the fabricator and design center.

If I send a board to a fabricator, they will send back a series of technical queries (TQs). They may have questions about something in the data. "We found an error on the board. They sent it to me in one format. If I go to another company to fabricate my board, they send that information in a different format." So now, one's a PDF, one's in Excel, and you have to go through all this and figure out what's what.

In 2581, we're standardizing that format, so why send a TQ? Not only do I have it from all my vendors who are using 2581 so I have one reader to look at all this stuff, but I'll be able to have links that go directly to the database and point to the exact location.

2581 is being looked at as a digital twin of the physical board. So, the data for physical boards is based on what's in 2581.

Shaughnessy: I think getting 2581 integrating with CFX has been a really big step forward. How does that work?

Acheson: As I mentioned earlier, there's this thing called a function mode in 2581. One of the concerns of our customers is, "How do I protect my IP? I don't want to send my whole database to the fabricator in one area, my assembly to someone else, and they get all the data." They can use one of our function modes that says, "I'm going to a fabricator." You select bare board manufacturing and it outputs just the data for that. It eliminates the building materials for components and so forth. One of those pieces for the assembly side could be extracted or will be extracted for the CFX portion. Now they know all the information related to assembly.

They have the bill of materials with them, they have the component placement, the orientation, any of the other key factors that may be attached to it, whether it has multiple manufacturing part numbers or what have you. Then, it has some basic description of the component itself where the pins and components lie, and the height. Then, they can extract that information for the machine work and they can create their CFX packets and everything about each board based on the data contained within the original database.

Let's go from start to finish. I develop my design, output the data in 2581, and I can target it to whatever functionality I want it to. Then, as it goes through the process, and questions and problems occur, I've got that 2581 format that I can send it back to. I can review the issue rather quickly, send that information back with all the answers, and they can continue with the process. I'll have a history of it through the whole cycle.

Johnson: That's a pretty powerful thing, but I also recall you saying earlier that this is an XML format. How do you secure the data?

Acheson: It's a lot like how you would protect data today with anything else. It's between you and the fabricator, or the contract manufacturer, with a secure link. We bounced around putting security built into XML, but when you say, "I'm going to do this," because it's an open format, you must publicize it. There goes your security because everybody in the world knows how to do it. We've consciously made the decision not to embed that, but it becomes an agreement between you, whoever you're contracting with, and how that encryption would occur.

Johnson: Your security is not in the file; your security is in the channel?

Acheson: Correct.

Holden: Have they discussed whether 2581 will be at the IPC APEX EXPO in January? Or will it all be CFX?

Acheson: Well, IPC-2581 has been at APEX EXPO since our consortium was launched. We've always had a booth. We've had different ways of promoting it. We've had opportunities for people to watch demonstrations of moving data from one system to the next, and how each tool uses 2581 in the process. We were trying to get a board for CFX at the last APEX EXPO, but we didn't get it done in time for the show.

So, that's something that's still on the table. It's just a matter of getting people to buy in and be part of that. I know Michael Ford has been in favor of it; Hemant Shah and the consortium have been fully in favor. We got a lot of people backing us in creating the design and the database for it. We're ready to go, we have the data, we just need the CFX group to consume and produce it.

Shaughnessy: What is the biggest hurdle to getting it adopted? Is it just changing people's minds?

Acheson: It's education and understanding. We talked about this a little—the manufacturers hesitating because they have to buy new software and they want to justify it. Customers are waiting to see who's first, the chicken or the egg: "I don't want to do it unless somebody else has done it and proved it." Well, we know it's been proven. We do have a customer who has produced every single board since January 2017 with 2581. We're seeing several of our major customers moving forward. Fujitsu USA was the first ones to produce a board. When they went around a second time with it, they output nothing but 2581 for three fabricators. Each fabricator had the same three questions and when all was said and done, they had three sets of boards built by three different fabricators using only 2581.

We have a few other announcements coming along the way. I can't say much about it until they make the announcement, but we are seeing a big drive and big adoption.

Shaughnessy: How many members do you have in the IPC-2581 Consortium?

Acheson: If you look at the consortium website, we're seeing a lot of individuals joining the consortium because they all believe in it. I don't know the total numbers of associate members, but I know that it's in the hundreds. These are individual users who understand the format and are working, as much as they can within their corporations, to move toward that format.

Holden: Ed, I can't remember my history, but was Dieter Bergman around when you first sent out 2581?


Dieter Bergman

Acheson: Yes, he was. Here's a story about Dieter and 2581. Back when we were first doing this, we were working on moving toward Revision B of 2581. We were in San Diego in a conference room during IPC APEX EXPO with Dieter and Karen McConnell. We were talking about how we wanted to include more data into the format, and how we wanted to address it. You could see Dieter was a bit frustrated. He said, "When are you guys going to actually start producing this stuff? We've been talking about it for years, but nobody's producing it. I can't see this ever moving forward if people aren't producing it."

I said, "Dieter, we are producing it. There are a couple of us who are working behind the scenes exchanging data and looking at how it works." He said, "Well, I haven't seen anything." I said, "Would you like to?"


Karen McConnell

I thought he was going to tear the connector out of the back of his laptop trying to get the projector so that I could plug it into my machine and show him how it worked. I uploaded my 2581 data in my Allegro database, imported that 2581 into a Wise Viewer and showed him all the data and the detail in that format. He sat down in awe, and said, "I never thought I'd see this day."

Karen McConnell was also elated to see somebody actually produced it. What made my week was walking the show floor with Gary Carter, who was very active with Fujitsu on this. Dieter came walking up with his wife and his entourage. He saw us, stopped, introduced us to his wife, and said, "These are the guys that made my dream a realization. They're my heroes." I mean, he couldn't have said anything better. I really miss the guy.

Holden:: Nobody had the vision longer than he had. I only go back to '77 or so when I first met him but getting off Gerber onto an intelligent and interchangeable format was always his vision.

Acheson: Well, that's one of the things about 2581; it was perceived as an achievable database that could be opened in any format. There was an agreement amongst us software developers at the early part that for the time being, in a gentleman's agreement, we would not import 2581 from each other to gain a competitive advantage. That worked well until a few years back, and I won't mention who it was, but someone started importing 2581 data and then we said, "Well, okay. All bets are off."

Holden: That reminds me, Ed, since you're with Cadence, why are the semiconductor guys talking about Gerber files now? I was really blown apart by the fact that we have 2581, but the semiconductor guys are talking about Gerber now.

Acheson: We've met with several semiconductor companies, even some from Japan who were looking for a standard format that they could use to build their substrates and all the different packaging types that they do. IPC-2581 does support quite a bit of it. In fact, we've included definitions about wire bonds, for example. We don't specify the wire bond path and structure itself, but an envelope of where that wire bond would be limited to.

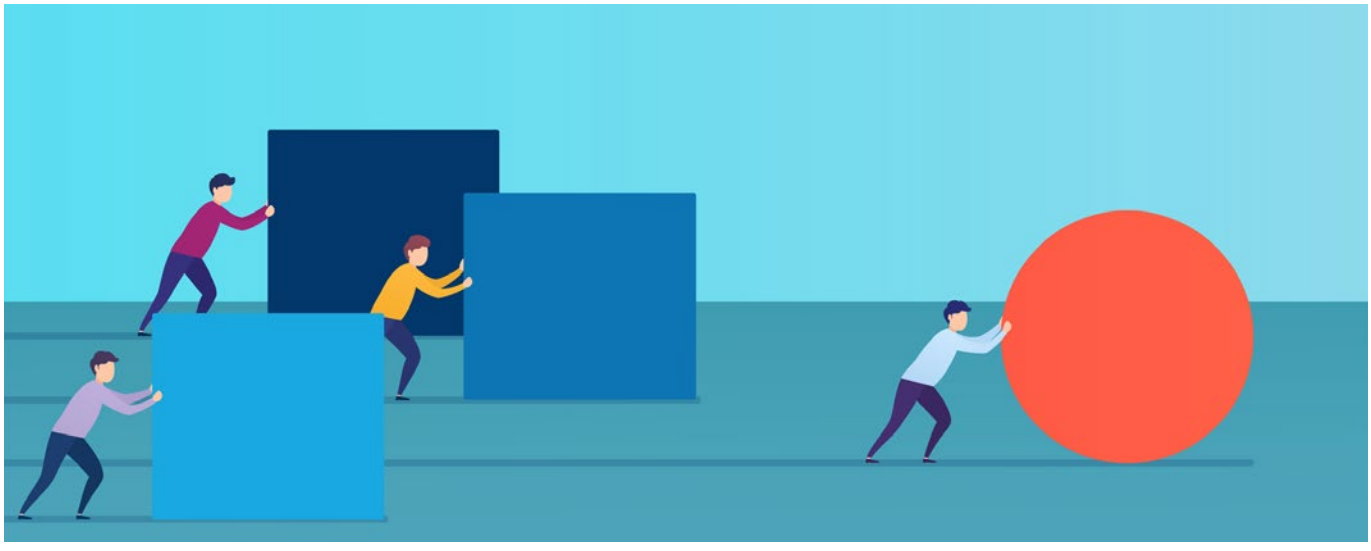
We can pass along any of the material's data for how the substrate is structured, whether it's glass, ceramic, or whatever material they're using. We've even addressed the ability to dictate where a bond pad starts on a component vs. a bond pad in an amphitheater part of the board. So, that data is there. Along with that, because of its XML structure you can actually build a subassembly of a subassembly in a single file. If I have an interposer stack of components on top of it, I can specify that in 2581. The trick is getting the offering tools to output that data. So, the standard supports it.

Shaughnessy: Is there anything you would like to add?

Acheson: I think if we keep educating people and allow them to understand and see the advantages that IPC-2581 provides, they'll be convinced—even us old-timers. I was convinced.

Shaughnessy: Thanks for your time, Ed.

Acheson: Thank you, Andy. DESIGN007



IPC-2581: An Open, Neutral, Efficient Data Transfer Format

Feature Article by Hemant Shah and Patrick Davis
CADENCE DESIGN SYSTEMS

Introduction

PCB design and manufacture has become exponentially more complex as modern electronics providers strive to meet consumer demand for greater connectivity and performance. In the current process, two major pain points need to be overcome to achieve an efficient design and manufacturing process that results in first-time-right products that go to market quickly.

The first pain point is time to manufacture—the flow from end of design to start of manufacturing is dependent on multiple communications and shared files between the design house and the manufacturing partner. The industry-standard Gerber file format-based package contains a set of files with data in different formats, along with documentation/notes/instructions, forcing the manufacturer to go back and forth with the design house customer to ensure that their interpretation is correct. An additional problem is that the data in Gerber files doesn't match what is in the netlist or in the instructions, leaving the manufacturer to determine which one is correct. This back and forth is done in written form so that the manufacturer can document what the overriding instructions are.

The second pain point is the design for manufacturing (DFM) analysis that all manufacturers run on the data to see if the design violates any of their manufacturing rules/guidelines. They can bend the rules to some extent in some instances, but if there are many violations, the customer must be told that either the design cannot be manufactured, or if manufactured, the yield will not be high, and therefore the cost to the customer will be high. The feedback from DFM analysis tools from the manufacturer is in electronic paper, such as Excel spreadsheets, HTML code, PowerPoint slides, Word documents, and email. This electronic paper is then read and interpreted by the design engineer/PCB designer and correlated into elements in the design. The DFM feedback and this second round of back-and-forth communication

APCT

Passion | Commitment | Trust

The Intelligence Inside

APCT Has Supported The Defense & Aerospace Industry Throughout The Years.
We Can Assist You As Well!

TECHNOLOGY

- Rigid Technology up to 42-Layers+
- HDI Technology up to 8-Lam Cycles
- Flex Technology up to 12-Layers+
- Rigid Flex Technology up to 26-Layers+ including “Bookbinder” Technology

CERTIFICATIONS

- ISO 9001 Certified
- AS9100D Certified
- MIL-PRF-31032 Certified
- IPC 6012 Class 3 & 3A
- ITAR Registered

PROUDLY SERVING

BOEING Raytheon LOCKHEED MARTIN
NORTHROP GRUMMAN KAMAN

Advanced Technology | Cycle Time Speed | Knowledgeable People

512.548.7435 | APCT.com

Printed Circuit Board Solutions

APCT.com

APCT Santa Clara HQ	APCT Anaheim	APCT Orange County	APCT Wallingford	APCT Global
408.727.6442	714.921.0860	714.993.0270	203.269.3311	203.284.1215

is a time-consuming and error-prone manual process in which both sides must keep track/document what they discussed, what was overridden, and what was decided.

This article examines the various PCB manufacturing handoff formats, such as Gerber, ODB++, and IPC-2581, and highlights why IPC-2581, the only standard that solves both the design-to-manufacturing handoff and the DFM issues, is becoming widely adopted.

Design-to-Manufacturing Handoff Challenges

There has been much debate in the PCB industry recently about the need for more modern, automated file formats for sending board designs to fabricators. The popular Gerber-based package, invented in the 1980s, is still the backbone of the industry because it is familiar and workarounds for its shortcomings, such as EDA library "hacking," have been devised in-house to produce Gerber-based handoff packages.

For engineers, the problem with using Gerber-based packages is that there is no efficient way for the design house to receive feedback from the manufacturing partner. At every stage that the data is handled by the manufacturing partner, feedback is sent via electronic paper, and because the feedback is shared in multiple pieces, those on the manufacturing side must reverse-engineer the design, hope they make the right decisions, and spend additional time asking questions to verify the accuracy of the data. It is time consuming and risky for design divisions to produce and manufacturing divisions to consume, because so many files are being produced that it is difficult to ensure they are all in sync when changes are made. There are too many formats, the data is split into too many incoherent pieces that are not easy to digitalize, and it isn't possible to do smart processing. If an error is made during handoff, the design house is responsible for the cost, and, conversely, if an error is made during manufacture, the manufacturer is responsible for the cost (Figure 1).

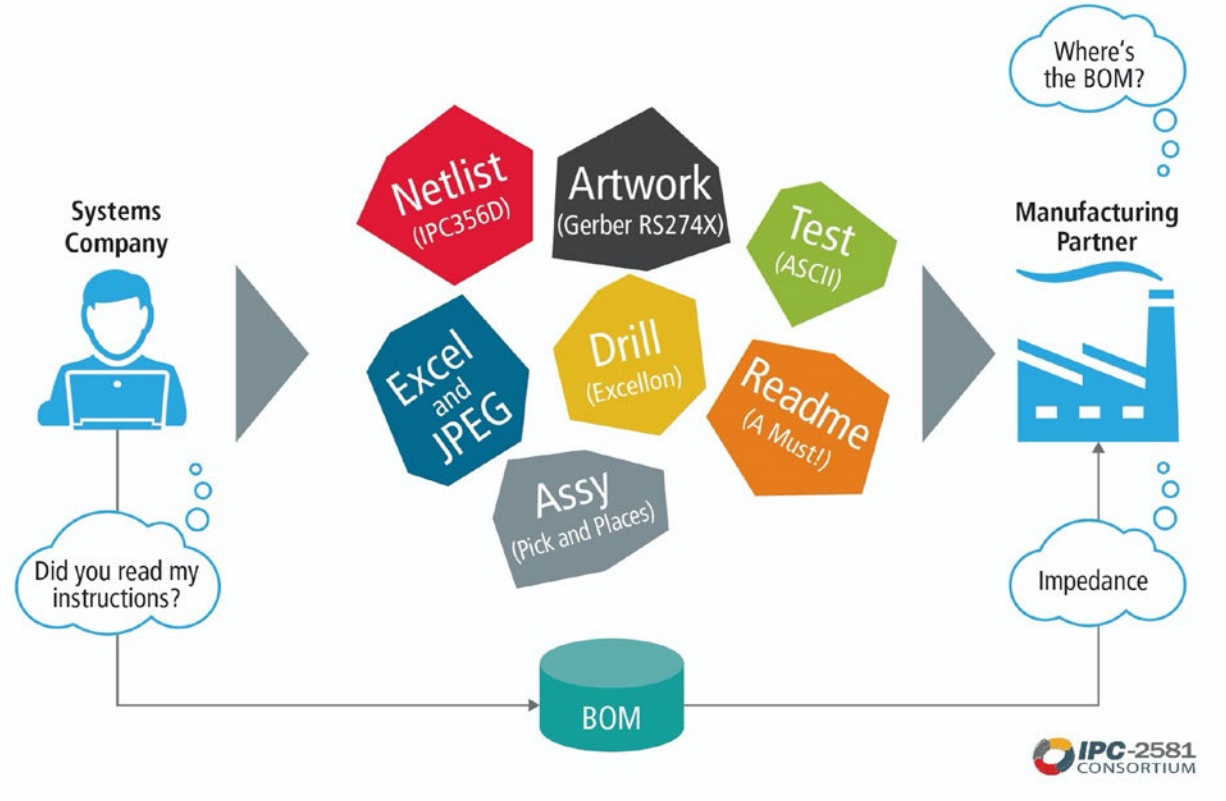


Figure 1: Smart processing is impossible with the many files needed using the Gerber format.

The graphic-based Gerber file package is unintelligent and impossible to process automatically. A few of the serious limitations of Gerber are that a separate physical file is needed for each layer of PCB information (such as copper images, fabrication notes, assembly drawing layer etc.), the stackup exchange is via electronic paper, and it does not include drill information, which must be sent in a separate file and is often mistakenly generated to a different scale or offset than Gerber. In addition, Gerber-based packages contain no electrical connectivity information, so a separate netlist file must be sent for electrical bare board testing that may not match the Gerber package. Finally, Gerber-based packages contain no component placement or bill of materials (BOM) information, so separate procurement files must be generated for those as well.

Using legacy Gerber-based packages for manufacturing handoffs affects both the product development division and the manufacturing division. For product development, experts are needed to ensure a successful handoff. On both the design and manufacturing sides, there are long process preparation cycles for DFM, design for assembly (DFA), and design for testability (DFT) review, prototyping, trials runs, etc., which increase communication issues between design and manufacturing teams. The Gerber files and netlists don't always match. The problem has been compared to the subtle differences in languages such as Spanish and Portuguese, which are almost the same, but not quite. In addition, the mechanical dimensions of the board in the fab document don't always match what is in the Gerber file.

This unreliable process results in long processing times and inaccuracies that affect product quality and time to market. The processes are not repeatable because every company has its own way of handing off that is not consistent.

ODB++ and Other Formats

Other formats used for the DFM handoff include GenCAD/FATF, an obsolete standard no longer supported or maintained. The Centroid XY data format, used by some for its assembly information, contains efficient content but can be out of sync with other data, which slows the handoff process. The ODB++ format is used by some designers, but it has compatibility risks over time, uses the same notes/instructions process as Gerber-based packages, doesn't provide stackup exchange, and, because it is a proprietary standard, users who are not customers or owners of the standard may not get the support they need. Figure 2 compares these formats and their business impact.

Format	Description	Business Impact
Gerber	Graphic picture, unintelligent • Impossible for automatic processing	• Takes very long time to process • Business risk for stencil preparation • Difficult for digitalization
GenCAD/FATF	Obsolete standard No longer maintained	• Business risk • No support
ODB++	Proprietary standard Compatibility risk over time	• Business risk if you are not a customer of owner of standard
Centroid XY data (placement data)	Efficient content Can be out of sync with other data	• Takes time • Business risk for placement programming

Figure 2: The business impact of various manufacturing handoff file formats.

Gerber-Based Package Example

To illustrate the low efficiency of a solder paste stencil design/preparation using Gerber data, three cases are presented. In Case A, a stencil issue leads to a quality event where the line is stopped during production to find the root cause, prepare a new stencil, and restart production, resulting in a 10-day delivery delay. In Case B, there is a long requirements preparation cycle in which the manufacturing process engineer screenshots with CAM350, then prepares the requirements with an Excel spreadsheet, which takes seven hours for the requirements preparation and a total stencil preparation cycle of 2.5 days. In Case C, repeated communication efforts are required in which, upon receiving the design from the stencil house, the manufacturing process engineer must confirm the design, which takes four verification and explanation cycles, resulting in eight hours for communication/validation and six hours for requirements preparation, for a total stencil preparation cycle of three days. Figure 3 summarizes the low efficiency of these cases using Gerber-based packages.

Manufacturing Feedback Challenges

All manufacturers run DFM analysis on designs to ensure manufacturability under their particular processes. DFM analysis also has issues to the disparate electronic paper files. The design house must correlate the files, interpret the data, respond to the queries, and approve or reject the request for change. This second phase of communications and feedback results in additional manual effort and time. The same thing happens with fab and test, and assembly and test as well. If the design violates too many of the manufacturer's guidelines, it must either be rejected as unmanufacturable, or the yield will be so low that manufacturing will not be cost effective.

IPC-2581: Intelligent PCB Design Handoff to Manufacturing

IPC-2581, an IPC standard, has existed since 2004. At that time, iNEMI brokered an agreement between IPC and Valor (the company that owned the ODB++ format at the time). With that agreement, IPC's GENCAM and ODB++ were merged to create IPC-2581. With IPC-2581, the handoff process can be performed much more efficiently and cost-effectively without the need to "dumb down" the data, segregate it into multiple pieces, and reverse-engineer it. Using IPC-2581, designers can create a single file that has everything the manufacturing partner needs to fabricate, assemble, and test the PCB, eliminating the struggle with multiple files and back-and-forth communication between the design and manufacturing sides.

Table 1 highlights, from the manufacturer's view, how deep in the manufacturing process the IPC-2581 format goes vs. Gerber- and ODB++-based packages. It delivers greater value by eliminating much of the manual intervention. Duplicate data, such as PCB outline dimensions, is also cut and an external

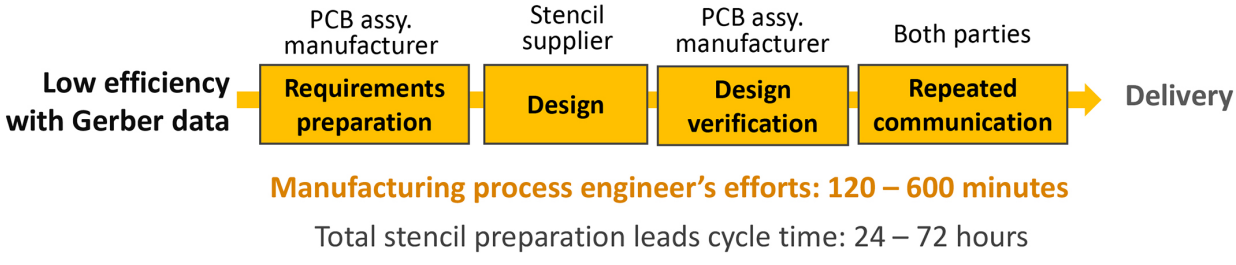


Figure 3: The results of using low-efficiency Gerber data.

	Graphical Features	Intelligent Features	Stackup with Impedance	Intelligent Notes	Intelligent Specifications
Gerber	√	Manually Added	ePaper Manual Intervention	ePaper Manual Intervention	ePaper Manual Intervention
ODB++	√	√	√	ePaper Manual Intervention	ePaper Manual Intervention
IPC-2581	√	√	√	√	Available 2021

Table 1: Manufacturing data comparison.

IPC-356 netlist is not required to validate data transfer quality.

The IPC-2581 format's digital product model uses electronic exchange, eliminating electronic paper by providing a single machine-readable file containing all the correlated data and enabling intelligent design data handoffs (Figure 4).

Features/Benefits of IPC-2851

What is IPC-2851?

IPC-2581 is the only open, global, neutral XML-based standard for efficient PCB design data transfer that can be influenced by the designer because it can be easily imported, augmented, and exported by any business sys-

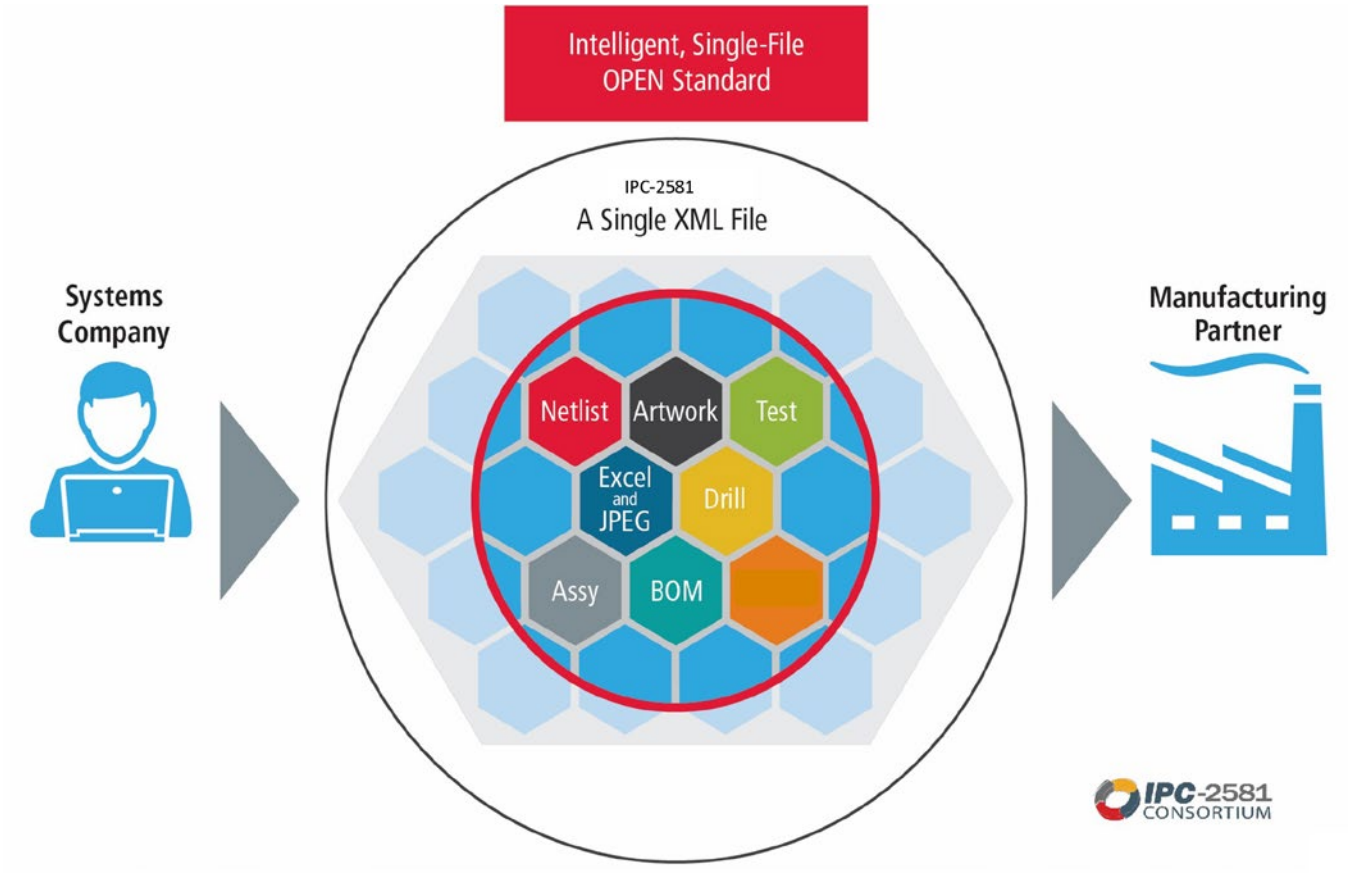


Figure 4: IPC-2581 format provides a single, intelligent file.

tem throughout the value chain, eliminating dependency on third parties. Because it is an open standard, no royalty payments or permissions are required. It is a complete digital PCB product model that allows the manufacturer to fabricate, assemble, and test the product. Most importantly, it is a simple, single file that includes everything from the design to the local BOM data and variants—all the information needed to be ready for direct process engineering tasks.

Everything Needed to Build and Test a Product

IPC-2581 contains everything needed to build and test the product (Figure 5).

- Part geometry and placement information, including centroid and component pin locations
- Physical and logical netlists and test points
- Completely support for rigid-flex data
- Variant data
- Intentional netshort data
- Full impedance specification for critical signals
- Tolerance information on stackup, drills, holes, etc.
- Complete support for embedded component information
- Multiple ways of placing the active components on inner layers

Intelligent, Efficient Handoff

The IPC-2851 standard provides rich, intelligent digital content for fabrication, assembly, and test. The single file includes a hierarchical BOM with complete component attributes for values, tolerances, units of measure, part rank, part class, and more. Unlike the Gerber and ODB++ formats, IPC-2851 includes a stackup exchange as part of the single file. Having a stackup that is accurate and precise going into the design helps in three aspects: It helps the designer design the board to have the imped-

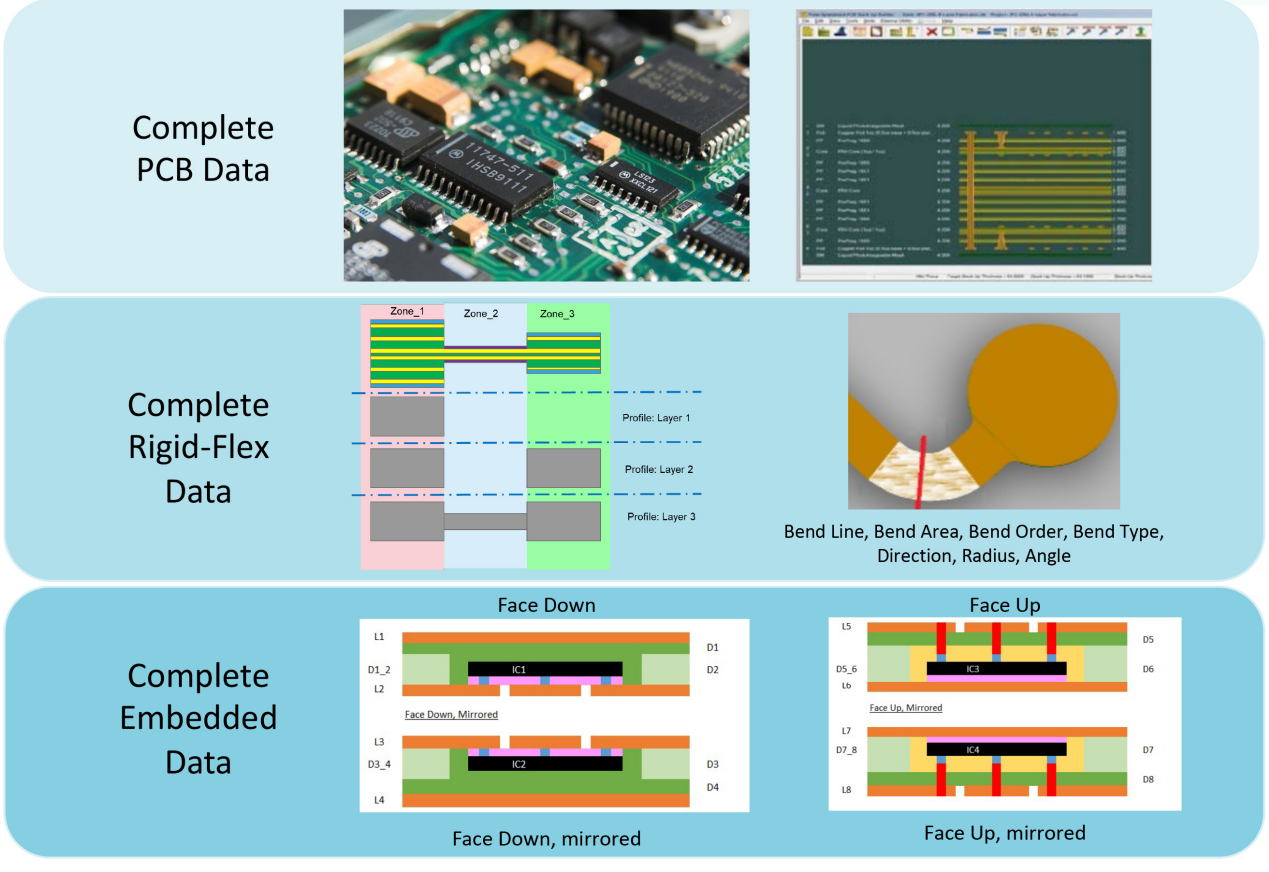


Figure 5: IPC-2851 provides complete data in a single file.

ances, the signal integrity and power integrity; designers don't have to question the correct materials; and everything is in one digital file so if the materials need to be changed there are no ambiguities and no wasted time.

In addition, the designer can include approved suppliers, corporate part numbers, and qualified manufacturer part numbers. The standard provides the material properties of the conductors, dielectrics, masks, surface finishes, etc. as well as tooling support for drilled/milled content, including blind/buried vias, back-drill, V-groove, slots, cavities, and more. Also included are the job notes and industry and corporate specifications and compliances.

The Digital Thread

Used with the IPC Connected Factory eXchange (CFX), IPC-2581 enables the creation of smart factory automation like Industry 4.0, a new industrial phase encompassing internet of things (IoT) and smart manufacturing that relies on interconnectivity, automation, machine learning, and real-time data. IPC-2851 is also the only standard that is part of the IPC-2551 Digital Twin standard that defines the design being built. This is beneficial to both the design and manufacturing sides because manufacturing issues are identified before they surface in real time, enabling engineers to make adjustments to the design to avoid them. This improves product quality and yield, and therefore reduces cost for the end customer and accelerates new product introduction.

A digital thread is possible only with IPC-2581. It provides everything the designer needs to communicate the build intent to the manufacturer and, for the manufacturer, everything is received in one file correlated and generated from an intelligent PCB design database to an intelligent IPC-2581 file that is an open XML-based format. This eliminates the time-consuming and unreliable back-and-forth communication common with legacy formats.

Electronically Executable DFM Feedback

IPC-2581 also provides electronically executable DFM feedback. While other formats require feedback data to be shared via electronic paper, IPC-2581 delivers that data in an XML format that can be executed by the design tools and allows the engineer to correlate the feedback with specific items in the design—the tools do the work rather than the designer intervening manually. This can be applied to fabrication DFM analysis feedback as well as assembly DFM analysis feedback. IPC-2581 is a win/win situation. (Figure 6).

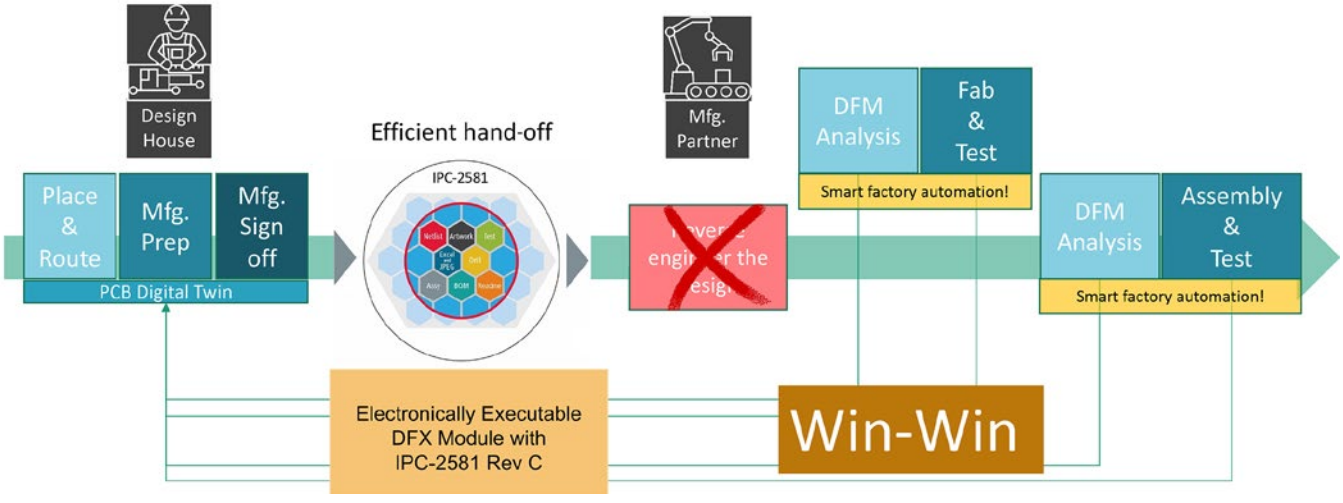


Figure 6: The single XML file enables the tools to do the work instead of the designer.

IPC Digital Product Model (IPC-2581) Consortium

A key advantage of IPC-2581 is that it is promoted and fully supported by the IPC Digital Product Model Consortium, whose sole purpose is to drive the adoption of this standard. It is an active group of PCB design and supply chain companies whose goal is to enable, facilitate, and drive the adoption of IPC-2581. The general business and technical committees meet twice a month with a mission to establish design best practices, provide guidance for adoption, and track tool support and usage. Thus, technical development is ongoing and will never become obsolete.

The IPC-2851 consortium has over 110 corporate members, many of whom are already using it in production (it has been available since 2013). Those who have adopted it

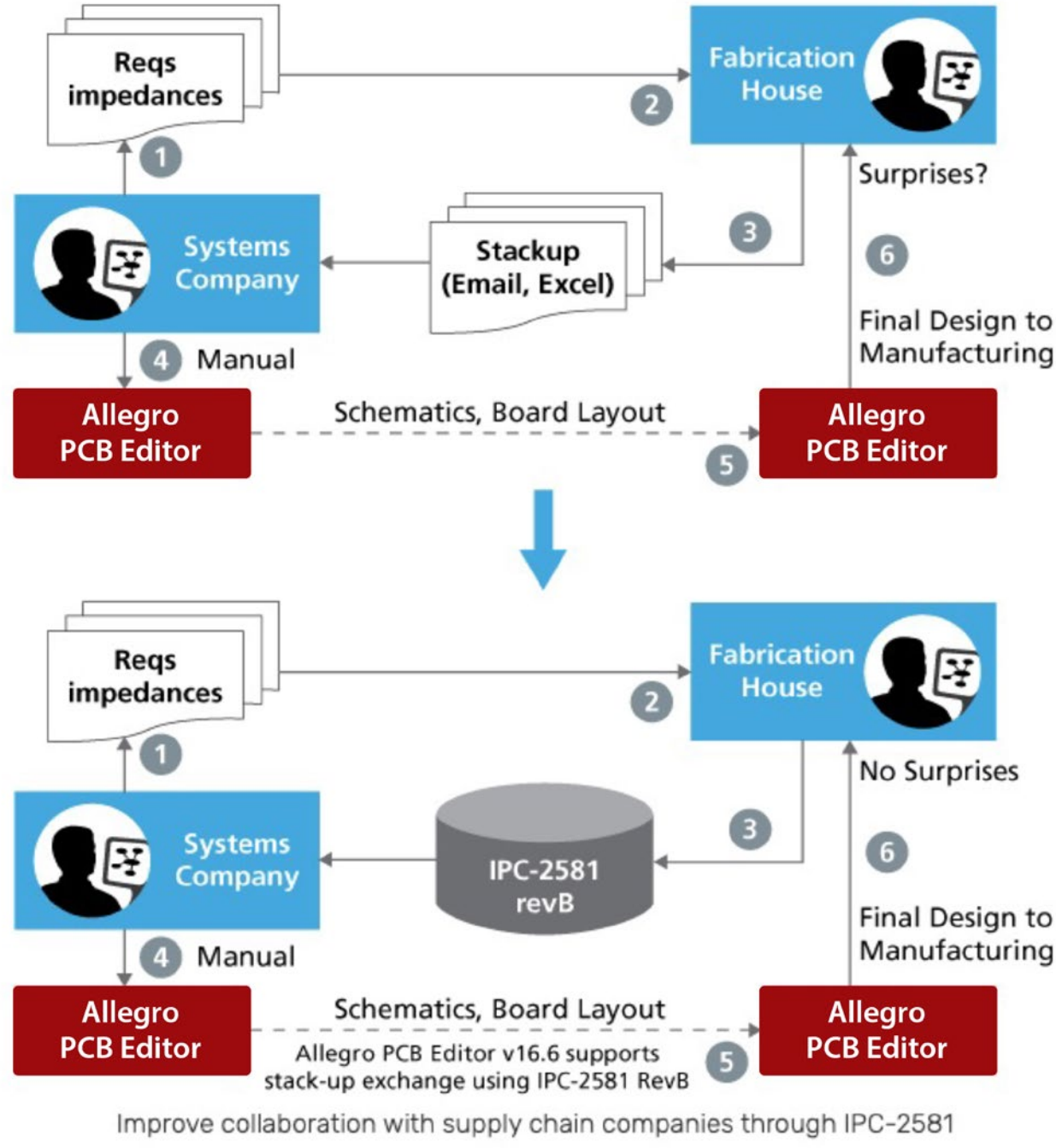


Figure 7: Cadence Allegro PCB Designer collaboration with IPC-2581.

have done so fully and use no other format. The complete, up-to-date list of consortium members, including manufacturers and software vendors who support the standard, plus details on the IPC-2581 format, are available at IPC-2581.com.

Cadence PCB Design Using IPC-2581

As a founding member of the IPC-2581 consortium, Cadence has built complete interoperability with the standard into its PCB design and verification tools, enabling users to efficiently and seamlessly hand off their complex designs to manufacturing. The Cadence design software enables efficient manufacturing collaboration through IPC-2581 (Figure 7).

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. The company is committed to developing and maintaining IPC-2581 import and export from its PCB design software and to staying current with the latest approved and published IPC-2581 specifications.

Conclusion

The PCB design and manufacturing industries are struggling with the need for more a modern, automated flow for sending board designs to fabricators. The popular but outdated Gerber format has many drawbacks, most importantly the many files and communications required during the handoff process that create numerous delays and mistakes, lengthening design and manufacturing cycles that result in production and time-to-market delays.

IPC-2581 is the only standard that resolves these issues that unnecessarily extend the time from design completion to manufacturing start. Because IPC-2851 is a single, comprehensive schema describing all relevant design, fabrication, and manufacturing assembly detail, it improves product quality and chances of first-pass success. With its ability to solve many current manufacturing handoff issues and ongoing development and support through the IPC-2853 consortium for cutting-edge technologies such as Industry 4.0, IPC-2581 is clearly the standard of the future. **DESIGN007**

Hemant Shah is the chair of the IPC-2581 Consortium and former product management group director for enterprise PCB products at Cadence Design Systems.

Patrick Davis is product management director for Allegro tools at Cadence Design Systems.

Finding Value in Gerber Files

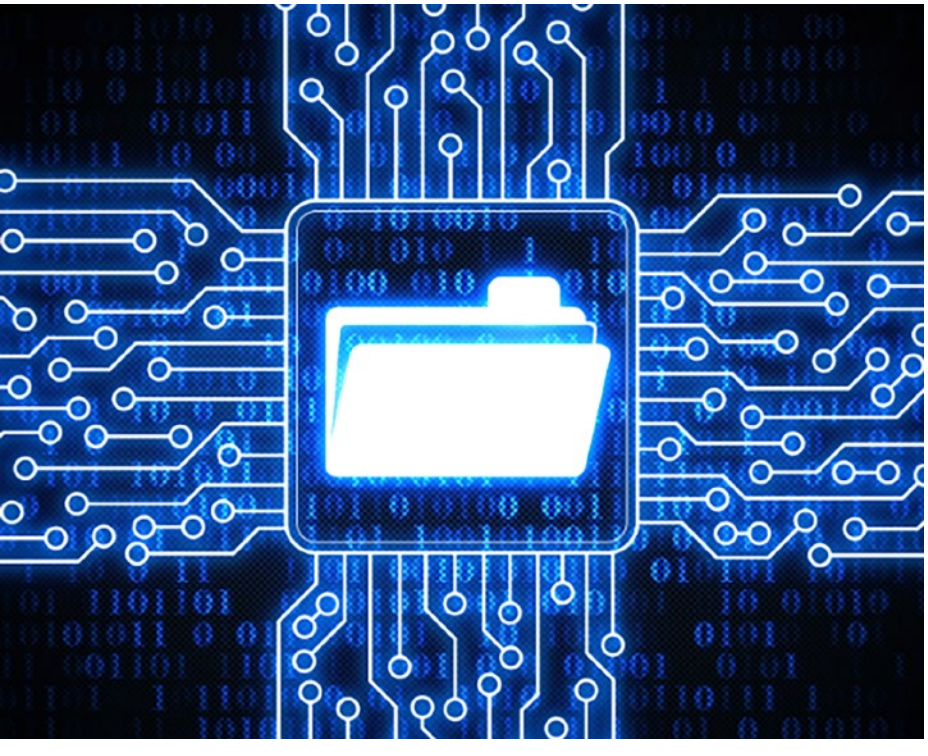
Connect the Dots

Feature Column by Matt Stevenson, SUNSTONE CIRCUITS

Quick turns are standard operating procedure in the PCB industry, but "measuring twice and cutting once" ahead of the quote process can get your boards into production faster and eliminate costly rework down the line. Gerber files remain a proven method for establishing an uncluttered view of the board design, one that illustrates each layer lined up together, and potentially reveals issues that can slow the quote process or even send designs back for rework.

What Is a Gerber File Anyway?

I am sure that everyone involved in PCB designing, manufacturing, and procurement has heard the term Gerber files at least 100 times. But what is a Gerber file and where did it come from?



The origin of the Gerber file can be traced to Gerber Systems Corp., and founder Joseph Gerber. The company first released *Gerber Format: A subset of EIA RS-274-D; Plot Data Format Reference Book* in 1981 to promote their vector photoplotters. Later in the 1980s, other plotter companies and CAM systems began to adopt what became the "Standard Gerber" format in the PCB manufacturing industry. Shortly thereafter, Gerber became the de facto data format.

As plotting technology continued to develop to more capable raster photoplotters, the "Standard Gerber" format was extended in 1991 to include polygons and other parameters. With this improvement, users were also able to create shapes of various sizes and shape and polygon area fills without having to draw them in; they could simply be defined with apertures.

In April 1998, Gerber Systems Corp. was acquired by Barco. In September 1998, Barco released the RS-274X format, also known as Extended Gerber Format, to consolidate and unify all the sub-versions that were present in the industry for specific plotter models. This new version quickly took over as the new de facto data format.

AltiumLive 2022

CONNECT

Connect with fellow engineering professionals, thought leaders, and industry experts from every aspect of the electronics development ecosystem.

January
26-28
2022

San Diego Convention Center
San Diego | California

Available Tickets:
3-Day All-Access
2-Day Summit Pass
Virtual All-Access

Learn More

HYBRID EVENT | In-person (San Diego Only) + Global Livestream

- → Five Unique Technical Tracks
- → Networking with Industry Peers, Experts, and Leading Electronics Companies
- → Four Exciting Keynote Presentations from Industry Leaders
- → In-person Full-access to the IPC APEX Expo

Join us at AltiumLive 2022: CONNECT, where you will **Learn, Connect, and Get Inspired** in a new immersive experience of technical tracks and convenient access to all that IPC Apex Expo has to offer.

Proudly Co-Located with

APEX EXPO IPC

Over the years, the 274X format has undergone revisions, improving the format and functions. The most current upgrade to the Gerber format has been the addition of the PCB metadata, giving the data more intelligence than the simple image representation. This current intelligent format is known as X2 and was developed in 2015 by Karel Tavernier, Ludek Bruckner, and Thomas Weyn of Ucamco, the PCB division of Barco.

Gerber may not be the only game in town for designers these days, but the format has been in widespread use for years and it is worthwhile to have an understanding of how Gerber files provide value.

Before You Convert to Gerber, Design Rule Check

You can prevent many issues by using the design rule check (DRC) tool in your CAD software. If you've properly configured the DRC tool, it answers questions that would often send a quote request back for more information. Are the layers all there? Are the spaces big enough? Do you have more drill files than copper layers? How are your tolerances?

These answers provide a good foundation for manufacturability, but just because your software will let you design a board in a certain way, that doesn't mean your fabricator can build it. Think about manufacturability in terms of fabricator capability as you perform your design rule check before converting to Gerber.

Check Your Gerber Files

Converting to a Gerber format provides a more holistic, simplified view of your design. Here are some categorized items you can more easily check for and make changes where needed when viewing Gerber files.

Minimum Feature Size

When you scrutinize minimum feature size and question what the DRC permits, you can:

- Find unintentional neck downs
- Locate places where traces just barely touch pins
- Discover breaks in traces or where ends just barely touch
- See where pads are too small
- Eliminate small traces or features that can needlessly drive up costs
- Validate minimum drill size, locating unnecessarily small drill holes that can also significantly increase costs

Feature Spacing

Solve clearance problems before they occur. Upon viewing the Gerber file, you'll have a sharper view of spacing types and be better able to recognize clearance problems for:

- Trace/trace
- Pad/pad
- Copper/copper
- Drill/drill
- Drill/copper
- Pad/pad

Look at Each Layer

This is really the only way to visually confirm each is correct. And by doing so, you accelerate the quote process and avoid cost overruns during production of the boards. Before submitting your quote request, consider:

- Does each layer import at correct scale and size?
- Is the number of copper layers correct?
- Are the silkscreen and solder mask layer counts accurate?
- Did the drill file read as intended, as a single file or multiple files? If not, why?
- Was the drill file updated at the same time as Gerber files?
- Do drill holes line up with artwork?
- Were all area fills refreshed just prior to generating Gerber files?
- Did you intend to have potentially expensive blind or buried vias?

Even one misstep in this category can create problems down the road. Though the list may seem long, the potential return on investment of a few minutes can be significant.

Check Solder Mask Layers

When you are double-checking each board layer, pay careful attention to solder mask layers. Oversights here not only can cause delays but result in boards that do not function properly or at all.

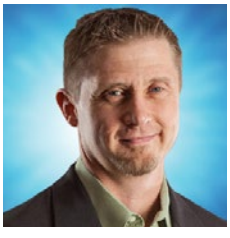
- Do all pins have mask openings?
- Vias under components should not have mask openings. Do they?
- Are solder mask openings slightly larger—at least 0.004"—than component pads?
- Are plated through-hole pads 0.015" larger than drill and therefore large enough to avoid breakout?

Leave Plenty of Breadcrumbs for Your Fabricator

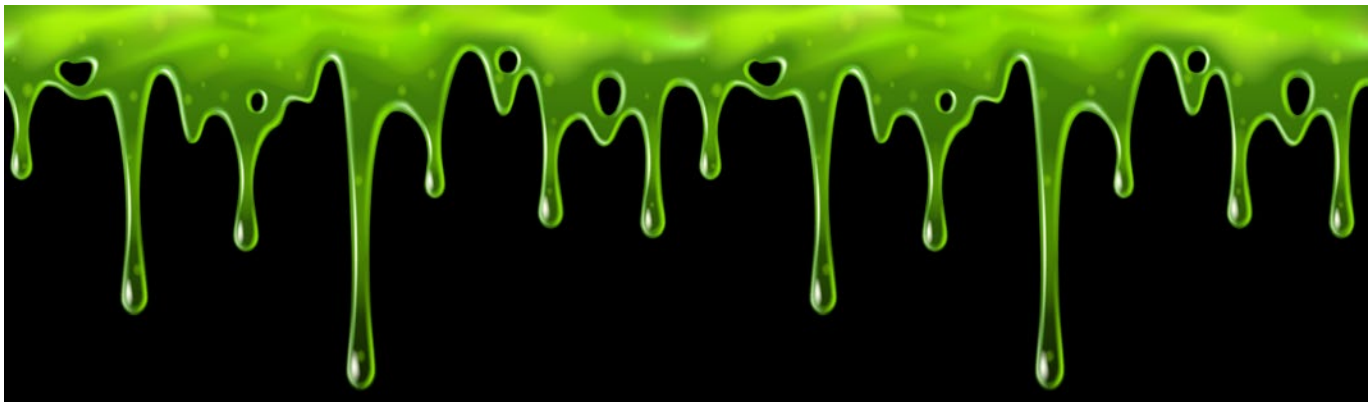
Fabricator notes can make the difference between a quick turn and a protracted delay. If file names are just a jumble of numbers or it is unclear about what services are required, the notes are where we can go for guidance.

- Will the fab notes conflict with your order form?
- Do notes call for unnecessary services?
- Do you reference documents that the fabricator does not have access to?
- Does the design service you selected include a technician checking your notes?

Not all quick turn prototyping services offer human intervention. So, if you chose to convert to Gerber, please invest a few minutes in your design and check your files before you engage your manufacturing partner. **DESIGN007**



Matt Stevenson is the VP of sales and marketing at Sunstone Circuits. To read past columns or contact Stevenson, click here.



The Impact of No-Clean Flux Residue on Signal Integrity

Beyond Design

by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

Most electronic products today are assembled using the no-clean soldering process. The need for no-clean solder pastes emerged in response to legislation against the use of ozone-depleting chemicals, and the appeal of removing the costly flux cleaning operations in the assembly of PCBs. However, the impact of no-clean flux residue on signal integrity is a concern for many high-frequency products such as 5G and other high-speed communications. No-clean flux residue can lower the characteristic impedance of an interconnect, create an alternative path to the signal, and degrade the signal integrity performance, especially at high frequency.

With the predominance of no-clean soldering processes and the ever-decreasing component standoff, the electronics industry has had to consider the reliability of what may be partially activated flux residues under component bodies. Similarly, questions have also arisen about the reliability of flux residues resulting from the reflow of no-clean solder pastes that are entrapped under RF shields, where the escape of the volatile ingredients of the flux is greatly hindered. The residue left behind is generally clear in appearance and tends to flow uniformly away from the joint. And due to the small quantity of solids, this residue is extremely thin in most places. This allows for a camouflage effect; upon visual inspection, the joint looks to be free of virtually all residues.

Little information exists about the effects of flux residues on electrical circuit functionality. But, with requirements for increased rise times and higher operating frequencies, there is particular concern about the magnitude of such effects on high-frequency, digital, microwave, and RF circuits. The effects of flux residues are measured in terms of their effective dielectric properties on microstrip structures. In this way, the change in microstrip circuit performance caused by the presence of flux residues can be deduced.

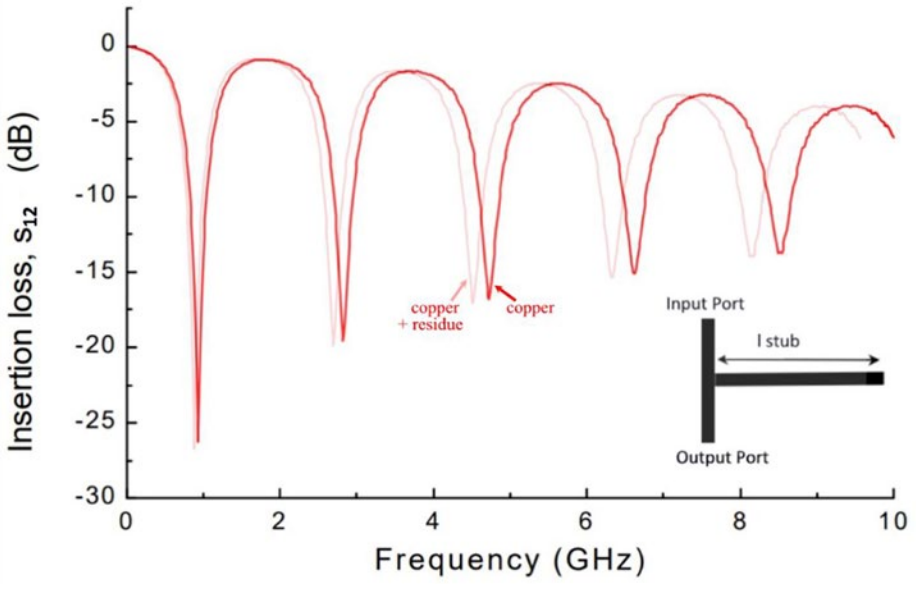


Figure 1: Typical T-resonator insertion loss (with/without residue)[1].

Microstrip resonator structures are widely used for determining the dielectric constant of substrate materials at high frequency. The results are applied to the design of controlled impedance interconnects, microwave, and RF circuit elements. By measuring the response of resonators that have been covered with and without no-clean flux, the change in effective dielectric constant (Dk) and dielectric dissipation (Df) caused by the presence of residues can be deduced and incorporated in the design.

The frequency response of a typical T-resonator consists of a series of resonant dips (Figure 1), measured at the input port S_{12}, the first of which occurs when the stub length (l stub) equals one-quarter wavelength (when the impedance of the stub tends towards zero). Subsequent resonant dips occur at the odd harmonics of the fundamental frequency. The resonant frequency of the T-resonator depends on the length of the stub (l stub) and the effective dielectric constant (ε_{eff})—where ε_{eff} is a combination of the material Dk and that of the flux (ε_{flux}). The presence of flux residue reduces the resonant frequency over the entire bandwidth but has its greatest impact at higher harmonics.

When the flux residue is deposited around a microstrip signal conductor, it also contributes to the effective dielectric constant of the structure. Figure 2 illustrates the results for different values of thickness and dielectric constant of the flux residue layers. It can be clearly seen that increased values of ε_{flux} also increase the effective dielectric constant.

The dielectric constant of a laminate is an important electrical property needed to accu-

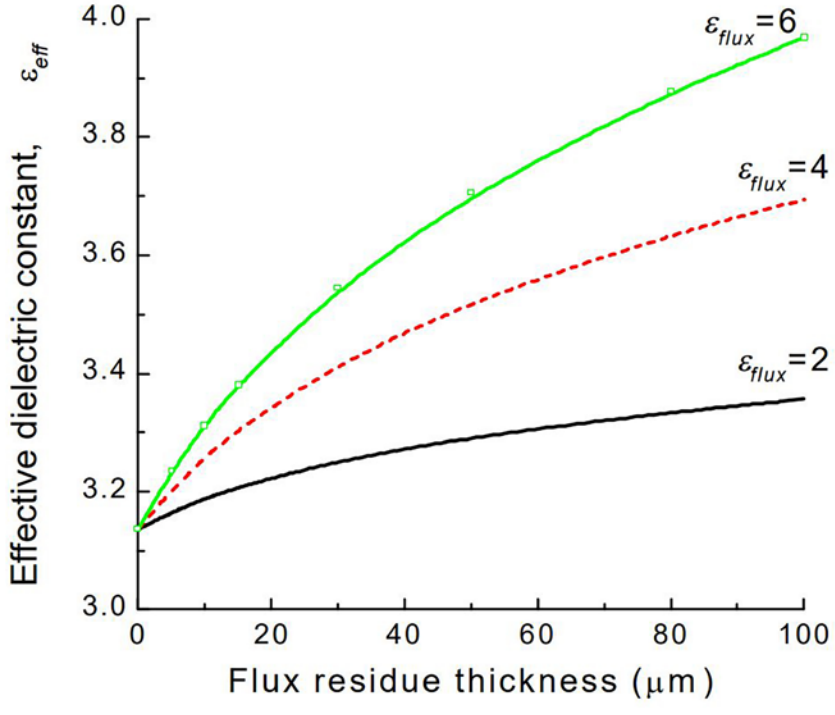


Figure 2: Effective dielectric constant vs. residue flux[1].

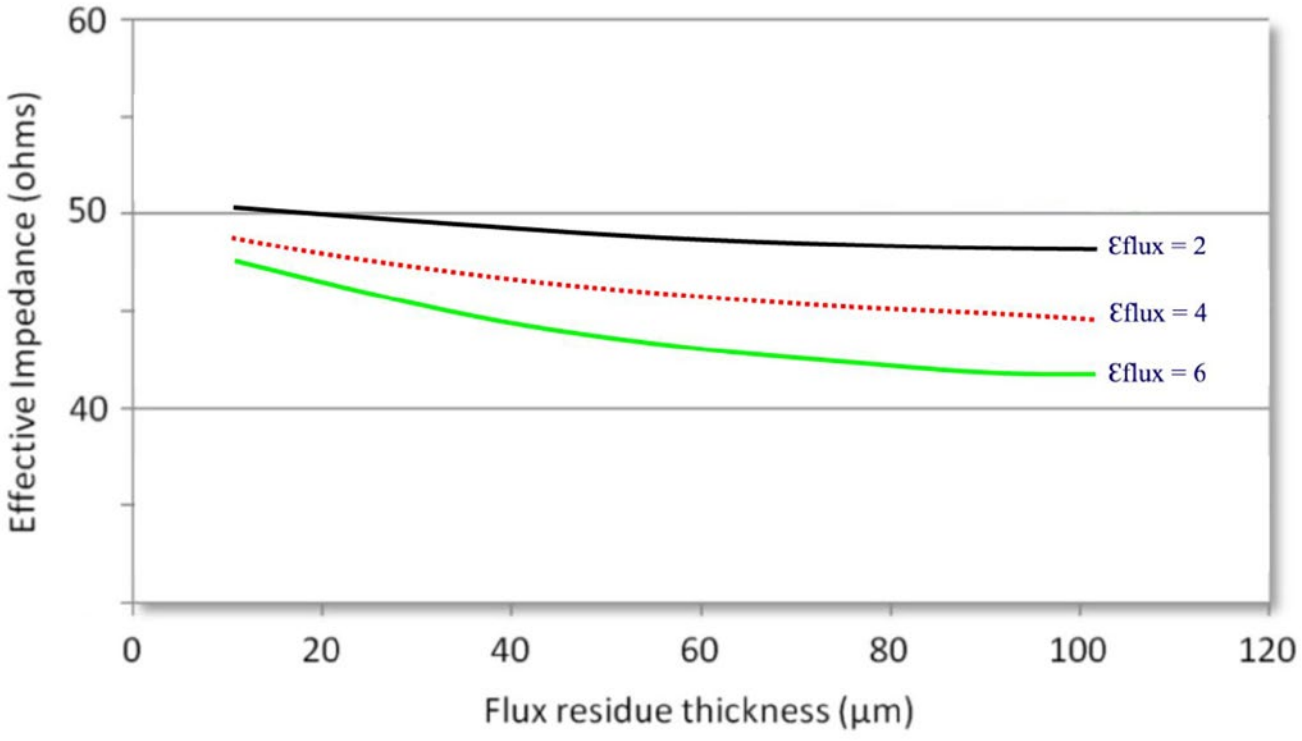


Figure 3: Residual flux vs. effective impedance (simulated by iCD Stackup Planner).

rately design a target characteristic impedance. It will affect the characteristic impedance of a transmission line and the propagation delay of the line. The strength and quality of the signal are augmented by a low dielectric constant of the materials. Low-loss and ultra low-loss materials, while being a bit more costly, can dramatically improve the propagation and quality of the signal at high frequencies. However, the presence of flux residue tends to increase the dielectric constant thereby reducing the signal quality.

Figure 3 shows how the change in flux residue affects the impedance of the microstrip trace. For a 15-µm residue, the impedance can drop by about 2.8 ohms, depending on the dielectric constant of the particular flux used. The impact of flux residue on microstrip impedance is not very large. However, the tolerance for controlled impedance boards is now regularly set at ±5%, making the use of no-clean fluxes borderline for high-density interconnects (HDI).

Changes in a PCB material's dielectric constant can adversely affect the performance of broadband high-frequency analog circuits as well as high-speed digital circuits, because it will change the impedances of transmission lines in unexpected ways. In particular, these undesirable changes in dielectric constant and impedance result in distortion to the higher-order harmonics making up a high-speed digital signal, with loss of digital signal integrity. In general, PCB materials with low and stable Dk values (with frequency and temperature) will support high-speed digital circuits with low distortion of the higher-order harmonic signal components[2].

Generally, we try to use low-Dk materials in the construction of HDI boards, which contributes to reduced interconnect delays and loss in the circuit. Consequently, an effective higher Dk caused by the no-clean flux is detrimental to high-speed design. Also, keep in mind that flux tends to congregate in the inner corners of right-angle bends which will add to this issue—another reason not to use right-angle bends when routing. RF and microwave signal interconnects are typically routed on the

outer microstrip layer. However, if critical digital signals are routed embedded between the planes, then this issue can be totally avoided.

Key Points

- The majority of electronic products today are assembled using the no-clean soldering process.
- No-clean flux residue can lower the characteristic impedance of an interconnect, creating an alternative path to the signal.
- One should consider the reliability of partially activated flux residues under component bodies and RF shields.
- The effects of flux residues are measured in terms of their effective dielectric properties on microstrip structures.
- Microstrip resonator structures are widely used for determining the dielectric constant of substrate materials at high frequency.
- The presence of flux residue reduces the resonant frequency over the entire bandwidth but has its greatest impact at higher harmonics.
- When the flux residue is deposited around a microstrip signal conductor it also contributes to the effective dielectric constant of the structure.
- The dielectric constant of a laminate affects the characteristic impedance of a transmission line and the propagation delay of the line.
- Low-loss materials can dramatically improve the propagation and quality of the signal at high frequencies.
- The presence of flux residue tends to increase the dielectric constant, reducing the signal quality.
- Unexpected changes in dielectric constant and impedance result in distortion to the higher-order harmonics making up a high-speed digital signal, with loss of digital signal integrity. **DESIGN007**

References

1. RF Characterization of No-clean Solder Flux Residues, Maeve Duffy, et al.
2. Selecting PCB Materials for High-Speed Digital Circuits, John Coonrod, Rogers Corp., *Microwave Journal.*

Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns or contact Olney, click here.

PCB007 Highlights

Nano Dimension, Fraunhofer Institute Team up to Develop Next-gen 3D Printing Systems ▸

Nano Dimension Ltd. and printed electronics manufacturing system provider, announced a collaboration with the Fraunhofer Institute for Manufacturing Engineering and Automation IPA, one of Fraunhofer-Gesellschaft's largest institutes.

Taiyo America Hires European Sales Manager ▸

Taiyo America, Inc is proud to welcome Stuart Down as the newest addition to their sales team staff. Stuart will take over the duties as European sales manager and will be based in the UK.

LPKF Accelerates Material Processing with Direct Laser Technology ▸

Research on flexible biomedical sensors, implant prototypes made of biocompatible materials, or combinations of microfluidics and electronics for lab-on-chip applications is extremely challenging.

Ucamco Releases Updates for UcamX and Integr8tor ▸

Ucamco has released UcamX and Integr8tor v2021.04. Both are major updates for your software suite and for the PCB sector at large.

IEC Partners atg and Eternal Technology Receive Award from TTM Technologies ▸

IEC is pleased to congratulate two of their supply partners, atg Luther & Maelzer and Eternal Technology, for winning a Global Supplier Award this year from TTM Technologies.

PCB Technologies Enhances Manufacturing Capacity ▸

PCB Technologies is enhancing its capacity with a new, state of the art, CNC routing machine. The machine, equipped with CCD cameras, ensures a high level of milling accuracy.

The Plating Forum: An Overview of Surface Finishes ▸

Surface finishes' research and development departments on the supplier side have been very busy coming up with new finishes to meet the everchanging demands of the electronics industry. Today, designers have wide variety of finishes to choose from. Columnist George Milad breaks it down.

Prototron Circuits Upgrades Via Fill, Planarization Capabilities ▸

Prototron Circuits has expanded its via fill and planarization capabilities by adding an ITC THP32 via fill machine and a Pola e Massa PLZ 3000 planarizer to its existing via fill equipment room.

Insulectro Hires Supply/Demand Expert Montserrat Barcelo as Director of Supply Chain ▸

Insulectro, distributor of materials for use in manufacture of printed circuit boards and printed electronics, has hired industry veteran Montserrat Barcelo as director of supply chain, replacing Jason Shuppert who was recently promoted to vice president of operations.

While others struggle just to keep pace, we stay ahead of the curve by constantly investing in the equipment, people and resources needed to be the best total PCB solutions provider in the industry.

VIEW OUR CAPABILITIES

EAGLE electronics Inc.

www.eagle-elec.com

Who Selects the Design Data Format, and Why?

Feature Interview by Andy Shaughnessy
I-CONNECT007

For this issue on design data formats, I wanted some feedback from a PCB design bureau, so I spoke with Jen Kolar, VP of engineering at Monsoon Solutions in Bellevue, Washington. I asked Jen who decides on the design data format at Monsoon, and why.

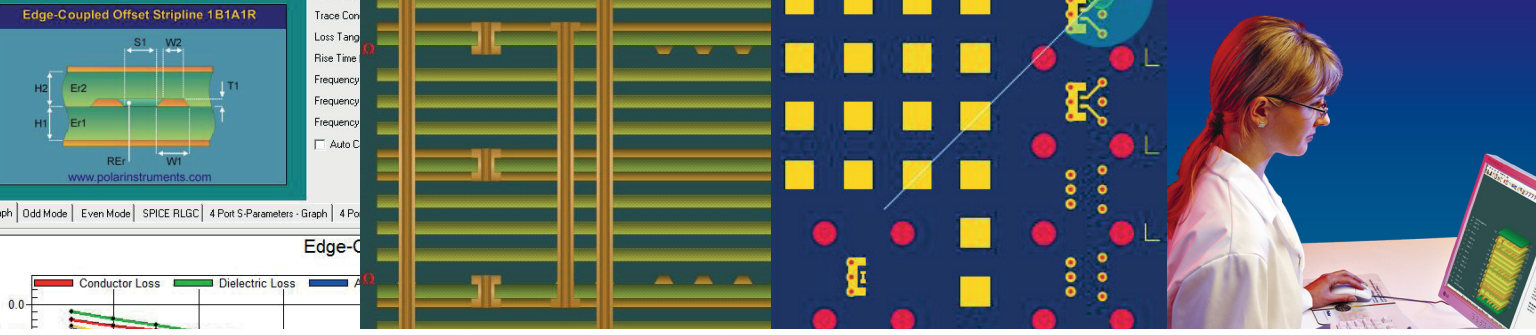
Andy Shaughnessy: Jen, who makes the decision at Monsoon about which design data format to use for a particular job?

Jen Kolar: It varies. We were just talking about this as a design team. At Monsoon, our standard (which is documented for our designers to follow), unless stated otherwise by the customer, is to output both Gerbers and ODB++ (as a TGZ file). We are considering dropping Gerbers from our standard format because when both are sent to the fab vendor, it always results in a question about which one to use. It is also more prone to having a file accidentally left out.

That said, when doing a really hot job, we might release copper first followed by silk later, and in that case, I expect most assemblers would prefer Gerbers and getting only the changed file vs. having to "trust" nothing else changed in the ODB++ package.

Interestingly, I find many fab vendors default to Gerbers vs. ODB++ if they have both. I've also seen issues with ODB 8 vs. ODB 7 with many vendors, and that might be another reason they may prefer Gerbers. We have some customers who mandate what they want. Some customers want only ODB++ and will refuse to work with any fab or assembly vendor who won't support it. This is not so much of an issue now, but it was several years ago, particularly with some assemblers.

We have others who insist on Gerbers only, who also like to review them as PDFs. Many customers don't really care, as long as they have something the fab vendor and assembler can use. We've never had anyone request IPC-2581 and we have never output it. Is that answer long-winded enough for you? (She asks with a laugh.)

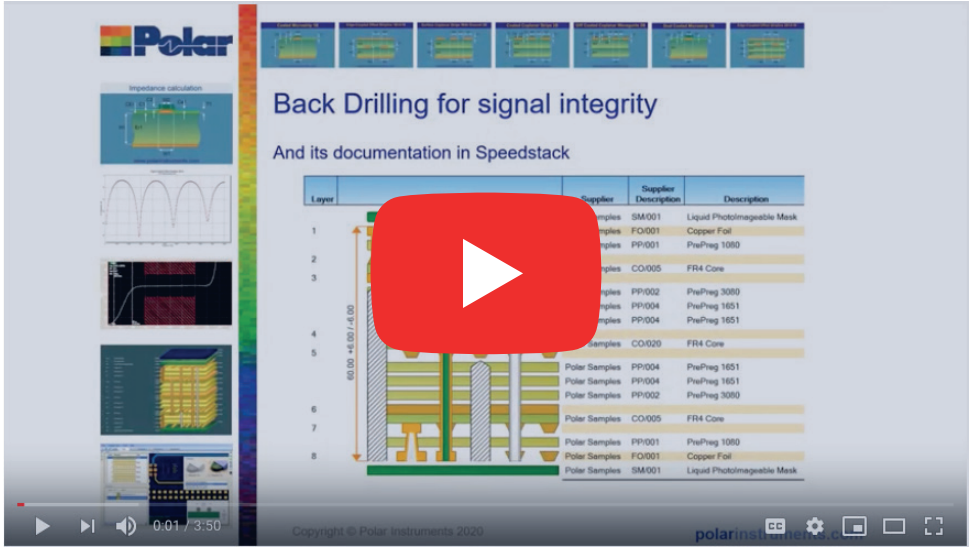


SUBSCRIBE

Subscribe to the Polarinstruments YouTube channel for helpful impedance and stackup videos

PCB Signal integrity tools for design & fabrication

- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics



polarinstruments.com

Jen Kolar

Shaughnessy: That will work fine. So, what is the title of the person who usually makes this decision?

Kolar: At Monsoon, it was once our COO, as he ran design. We now have one of our principal designers in a role to direct and document all design processes. It is all done with input from the whole team including me, as well as our director of manufacturing services, if there is a question of the impact on manufacturing.

Shaughnessy: Sometimes I wonder if this whole "horse race" angle regarding Gerber vs. ODB++ vs. IPC-2581 is a little overblown. The systems seems to be working fine with three different formats. Are there any inherent downsides to having more than one format for design data transfer in the industry?

Kolar: I don't see an issue with having more than one format, other than the risk of translation errors. But, hey, it's not a CAD tool if it doesn't have errors. The convenience of ODB++ is awesome. But for some companies without ODB++ viewers, or for people who are not familiar with them, Gerbers are just simpler.

Shaughnessy: This really helpful. Thanks for your time, Jen.

Kolar: Sure. Thank you, Andy. DESIGN007

New Tech Prevents Li-Ion Battery Fires

Materials scientists from NTU Singapore have found a way to prevent internal short-circuits, the main cause of fires in lithium (Li)-ion batteries.

Billions of Li-ion batteries are produced annually for use in mobile phones, laptops, personal mobile devices, and the huge battery packs of electric vehicles and aircraft.

This global battery demand is set to grow, with electric vehicles alone requiring up to 2,700 GWh worth of Li-ion batteries a year by 2030, equivalent to some 225 billion mobile phone batteries.

Even with an estimated failure rate of less than one in a million, in 2020 there were 26 power-assisted bicycle (PAB) fires and 42 cases of personal mobility device fires in Singapore.

In most Li-ion battery fires, the cause is due to a build-up of lithium deposits known as dendrites (tiny wire-like tendrils) that cross the separator between the positive (cathode) and negative (anode) electrodes of the battery when it is being charged, causing a short-circuit leading to an uncontrolled chemical fire.

To prevent such battery fires, NTU scientists invented a patent-pending "anti-short layer" that can be easily added inside a Li-ion battery, preventing any future short-circuits from occurring during the charging process.

This concept is akin to adding a slice of cheese to a hamburger's meat patty in between the buns, thus the new "anti-short layer" can be rapidly adopted in current battery manufacturing.

(Source: NTU Singapore)

IPC BUILD ELECTRONICS BETTER®

PCB DESIGN COURSES

DRIVE QUALITY AND INNOVATION

Introduction to PCB Design II	Military & Aerospace Applications	Rigid-Flex Boards
Upcoming class:	Upcoming class:	Upcoming class:
8/31–10/21	8/30–10/6	8/30–10/6

Look for these additional courses!

- PCB Design for Manufacturability
- PCB Design Embedded Components
- Introduction to PCB Design I
- PCB Design for Advanced Packaging
- PCB Design for Extreme Environments
- PCB Design for Printed & Wearable Devices

Learn more about our PCB Design Courses.

Tribal Knowledge and Design Data Formats

Feature Article
by Gene Weiner
WEINER ASSOCIATES

After more than six decades in the specialty chemical, electronic interconnect, and electronic packaging industries, I remain puzzled by the resistance to change. Not even higher costs, poorer quality, or impending failure appears sufficient to motivate many to act, even for their own betterment.

One may ask why I am writing about change today. We have seen dramatic changes in our lifetime. A half-century ago, Alvin Toffler reminded us of the inevitably of change in his classic book, Future Shock.

I recently began to investigate "tribal knowledge" in PCB and PCBA operations. My goal was to determine whether anything of value would be lost if it were not recorded or preserved for future industry technologists. In the face of modern material science and industry standards, the descriptive title "tribal knowledge" suggests that one must determine what is fact, what is fiction, and what may just be plain embellishment, coincidence, or even magic. In many cases, tribal knowledge seems to be composed of memories of problems solved due to trial and error, coupled with observations of resulting causes and effects.

During my search for more information, I read the August 2021 issue of *PCB007 Magazine* that focused on DFM from the perspective of the fabricator. Several of the articles discussed the challenges involved in DFM and design data transfer. What seemed to leap off the pages was that much of today's design and data transfer utilizes a 60-plus-year-old format named Gerber.

I remember visiting Gerber Systems Corp. (no longer in existence) in Connecticut many years ago. The system is now supported with continuing developments by an entirely different entity. This system does not capture and retain data as the modern intelligent systems do. Important DFM notes are not retained or transferred by Gerber. These notes are lost with the passing of those who have solved manufacturability issues. This important (tribal) knowledge may or may not have been recorded.

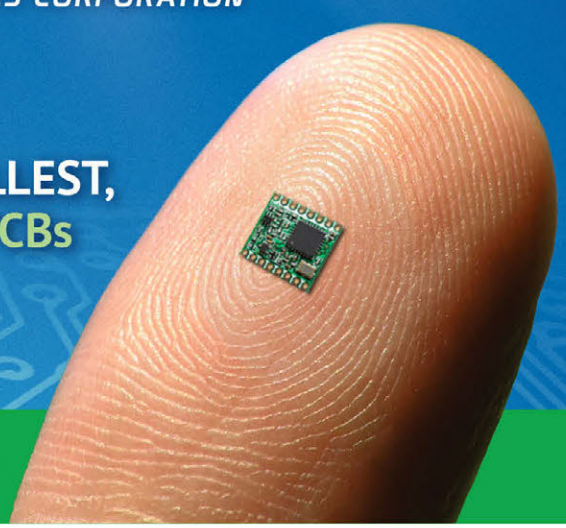
ENABLING NEXT-GEN SYSTEMS

DOMESTIC MANUFACTURING READY

Advanced additive manufacturing down to 15 µm features.
IC substrates and highly integrated HDI PCB solutions.
Phased array beamforming technologies.
100% American owned. 100% American made.
America's only HUBZone PCB manufacturer.

Calumet
ELECTRONICS CORPORATION

AMERICA'S SMALLEST,
LIGHTEST, FASTEST PCBs



Leaders in Averatek A-SAP™

NADCAP | HUBZONE | AS9100D | IPC-1791 | TRUST ITAR
ISO9001 | MIL-PRF-31032 | MIL-PRF-55110 | CAGE CODE | UL

That brought back memories of my time as a student technician at MIT Lincoln Labs in the mid-1950s. Designers would lay out a pattern on a large sheet of plastic. We would then apply black tape to the pattern, trimming with an X-Acto knife when needed. This was then placed into a large vacuum frame, photographed with a large Brown camera on a track, and photo-reduced tenfold to produce the artwork (negatives) used to image our circuit patterns with KPR (Kodak photoresist). In 1957, my supervisor, E.A. (Al) Guditz addressed this tortuous procedure and created what I firmly believe was the predecessor of today's imaging systems. He converted a programmable-head milling machine to directly write, in a "dark room," a pattern onto Kodak film using a xenon point light source collimated through a hypodermic needle to expose the film.

A lot has changed since then. More modern intelligent systems such as ODB++ and IPC-2581 record and transfer everything. They can help users to build databases. The latter can compile unique features. They can be used as archival tools for pricing. Their AI systems check comparable test specifications. They retain everything—even the designer's name—in a single file.

Today's technology is moving fast. Lines and spaces continue to shrink. Special features and properties, as well as manufacturing procedures and materials, are advancing rapidly. A simple unintelligent data transfer system is not sufficiently capable for relatively trouble-free use when DFM becomes even more important.

One more caveat is worth noting. Some design specialists and instructors state that it takes twice as long to produce a circuit with Gerber (with errors or problems to fix) as it does to generate one with an intelligent system.

Yet, most PCB designers and fabricators still use CAD layouts that are converted to Gerber and CAM as their primary design tool and fabrication systems. The fabrication shops appear to depend upon some level of tribal knowledge by experienced operators to solve the issues that arise, especially with mixed technologies, different substrates, and RF circuits.

All this points out the need for significant training and continual upgrades of designers (and their tools), especially in today's world of miniscule electronic packages and their shrinking substrates. Education and upskilling of all workers are necessary as part of continuing improvement. Designers should not be omitted from this. Management has often given short shrift to this idea.

Education and upskilling of all workers are necessary as part of continuing improvement. Designers should not be omitted from this.

We must also remember, as one fabricator noted in *PCB007 Magazine,* that automation will reduce errors and costs but not totally eliminate them as long as humans are involved. And humans will be part of the equation for some time to come as providers of CAM software are not likely to update their products (to be automatically uploaded, including design rules) in the near future.

How do we move forward? One must determine the causes of the apparent deterrent to advances in PCB design, and then translate this to a working device. It appears evident that many designers are working in a vacuum, in virtual isolation as far as understanding the total manufacturing process of an electronic device. Most are unaware of the materials and processes used to manufacture their devices. They are unaware of the physical and chemical environments endured as well as some of the electrical properties of various substrates

and coatings used. Part of the solution would be to have every engineer and technician that designs circuits visit a fabricator and an assembly facility and observe every step used to produce their parts.

I would also recommend certification and periodic upgrading of design skills with courses such as the Printed Circuit Engineering Professional, a comprehensive curriculum specifically for the layout of printed circuit boards offered by the Printed Circuit Engineering Association (PCEA) as part of its PCE-EDU professional development program. This curriculum was developed by instructors Mike Creeden, Gary Ferrari, Susy Webb, Rick Hartley, and Steph Chavez—professional development instructors with a combined experience of over 225 years.

Management must be made aware of the true costs of continuing to depend on obsolete technologies to produce the products of today and tomorrow. They must endorse the upskilling of their design engineers as well as all other employees to continue to grow and succeed in this rapidly changing digital world. Until then, corporations may have to continue to depend on the tribal knowledge of their PCB vendors. **DESIGN007**

Gene Weiner is president and CEO of Weiner International Associates and long-time technologist in the PCB industry.

COLUMN EXCERPT: **ELEMENTARY, MR. WATSON**

The Danger of Rogue Libraries

by John Watson

When we think of a library, the first thing that comes to mind is a building that holds materials of some kind. In April 1800, when the seat of the government moved from Philadelphia to Washington, one of the first acts of President Monroe was to allocate $5,000 to purchase books for Congress to use. With the establishment of the Library of Congress, it held 740 books and three maps.

It now contains 32 million cataloged books and other print materials in 470 languages, and more than 61 million manuscripts. It is the most extensive rare book collection in North America, and by far, the most extensive library in the entire world. For anyone who has ever visited the Library of Congress, it is incredible when you consider the tremendous amount of knowledge and materials in those walls. However, what I find even more astounding is how all that material is kept organized enough to find something quickly.

Library Defined

For PCB designers, it is not a building but rather a collection of information or data used to build a PCB design. The most common part of the library is the collection of components used in the PCB design process. But, I have seen some libraries have other information, including a resource area—a group of documents, standards, and articles. Basically, it can have anything you want.

It is essential to make sure the library is structured so that it is easy to expand and grow with the company. Unfortunately, some organize their library for their present needs with no consideration of the future.

To continue reading, click here.

John Watson, CID, is a customer success manager at Altium.


Matt Stevenson

Sunstone Circuits:

Use the Design Data Format That Gives You Best Results

Feature Interview by Andy Shaughnessy
I-CONNECT007

To get some feedback from a PCB prototype manufacturer about design data formats, we asked Matt Stevenson, Sunstone's VP of sales and marketing, to weigh in. As Matt explains, his company is equally at home using Gerber or ODB++, but it's not the fabricator's job to convince customers to use one data format over another. In the end, you should use the format that gives you the best results.

Andy Shaughnessy: I know that Sunstone Circuits deals with more than one type of design data format. What's your overall take on design data formats?

Matt Stevenson: Yes. Sunstone is a PCB manufacturer, so our view on data formats is mainly from the manufacturing and CAM processing standpoint. Our primary goal is to receive files that are accurate and well-defined. Most of our CAM processes were set up to automate Gerber files. ODB++ files are showing that they can be easily adapted to our process. We are not seeing much in terms of the IPC-2581 format and have not been able to evaluate their effectiveness and adaptability to our process.

Shaughnessy: What features do you value the most about ODB++?

Stevenson: When loading into CAM software, there is a single ODB++ file as opposed to multiple files in a zip file, with potentially hundreds of different naming conventions. When importing, all the layers come in predefined, the stackup is already present, and the drill files aligned, so it makes it easier in CAM and eliminates the guesswork.

Shaughnessy: What benefits do you get from utilizing ODB++?

Stevenson: From a CAM standpoint there is really no difference between this data format and Gerber once it's loaded, but it does make it easier to get all the data identified, aligned, and ready to begin the CAM tooling process.

PegaClad Series

for Sub-6 GHz Antenna (Dk 3.38, 3.45, 3.65)

PegaClad 1&2

for mmWave

Developed for Sub-6 GHz antenna applications, TUC's PegaClad 338, PegaClad 345, and PegaClad 365 - with respective Dks of 3.38, 3.45, and 3.65 - are advanced materials designed to minimize passive intermodulation and meet designers' needs. PegaClad 338, PegaClad 345, and PegaClad 365 can be used in multilayer and double-sided circuit board designs requiring excellent thermal reliability.

TUC's PegaClad 1 and 2 are designed for high frequency, very / super low loss applications, including mmWave printed circuit board designs. Their electrical properties are equivalent to PTFE laminates, yet they perform in multilayer circuit boards requiring excellent thermal reliability in harsh environments.

TUC's PegaClad Series and PegaClad 1 and 2 laminates also exhibit excellent moisture resistance, improved CTE, superior chemical resistance, and thermal stability, and are compatible with modified FR-4 processing.

tuc
Taiwan Union Technology Corporation
www.tuc.com.tw

TUC Asia
Raymond Foo
TEL : +886 3 5551103 x 129
E-mail :
raymond.foo@tuc.com.tw

TUC USA
Dana Korf
TEL : +1 408 643 1144
E-mail :
dana.korf@tuc.com.tw

TUC Europe
Marko Holappa
TEL : +358 40 770 9618
E-mail :
marko.holappa@tuc.com.tw

Shaughnessy: What is ODB's value-add for your customers?

Stevenson: As long as it's configured correctly at the design end, ODB++ contains needed assembly drawings that aid in the assembly process.

Shaughnessy: How would you convince reluctant customers to use ODB++?

Stevenson: I don't feel its CAM's job to try to convince a customer into changing what they are comfortable with. I would point out there are benefits to the assembly house in using ODB++, but if they are comfortable with what they are using, and we have no manufacturing problems with the designs, it works just as well for both of us if they continue with what they are currently using, as long as our CAM software is able to support it. If they are using something we can't support, then I would definitely recommend they give ODB++ a try for the reasons I've stated.

Shaughnessy: Do you believe that ODB++ simplifies your process, as compared to other formats?

Stevenson: From a CAM standpoint, once the data is loaded into our program there is no difference between the formats, so it doesn't affect the process in CAM. Both Gerber and ODB++ work equally well for our work here. It does help with there being only a single file to import instead of sometimes multiple raw board files, zips, and other data files—making it difficult to determine which of the many data sets we have been supplied we should use in CAM. However there are times when customers include multiple ODB++ files as well so that benefit is reduced in those cases.

Shaughnessy: What are the cons of the other two formats, in your opinion?

Stevenson: IPC-2581 is not currently supported by either of the two CAM programs we use here, so that is a major con for that one. The only con for the Gerber is there is more work at the CAM end in some instances if the files and drill format are not clearly identified. This causes us to spend more time trying to figure out which layers are which and getting the proper drill import settings, so all the layers are properly aligned.

Shaughnessy: Do you have any final thoughts or advice on design data formats?

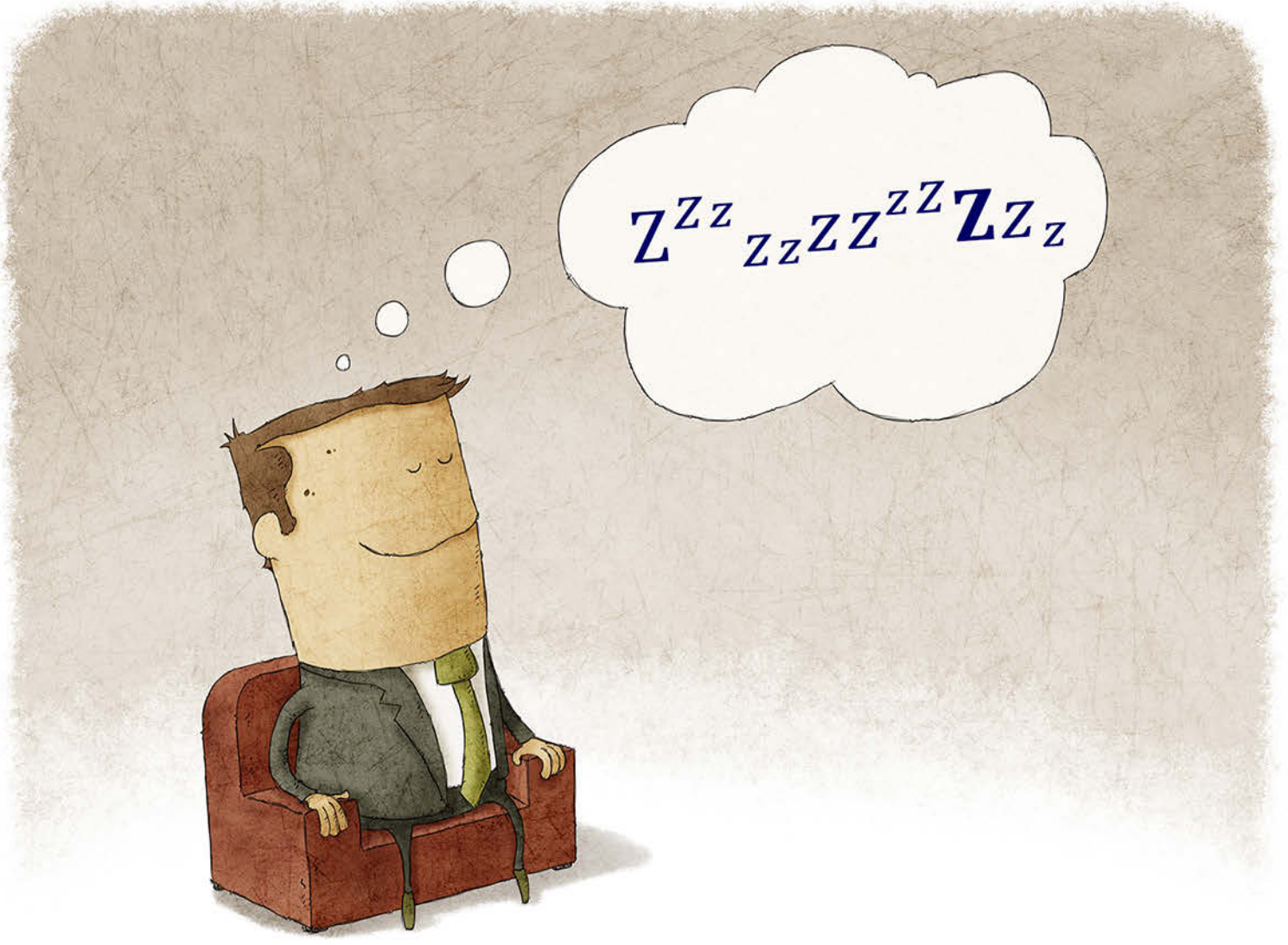
Stevenson: As noted earlier, in CAM the outputs and processes we go through are basically the same regardless of what file type we received from customer. Part of our job is to standardize the output from hundreds of different design packages. While there are definitely some things we really like about ODB++, we will continue to support either file type the customer feels comfortable supplying to us. ODB++ is just as problematic as Gerber for CAM if the correct settings and design outputs are not selected by the designer. If customers are more familiar with one or the other and output a quality data set that gives them what they need, then it makes more sense to have them continue supplying the data type they have been using.

From our partnerships with assembly manufacturers, we see that there are many advantages to the ODB++ format within the total supply chain from design through assembly. This "smart" data can flow more seamlessly into their manufacturing process and require much less manual intervention in the set up and processing.

Shaughnessy: Thanks for your input, Matt.

Stevenson: Thank you, Andy. DESIGN007

We DREAM Impedance!



Did you know that two seemingly unrelated concepts are the foundation of a product's performance and reliability?

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

iCD software quickly and accurately analyzes impedance so you can sleep at night.

iCD Design Integrity: Intuitive software for high-speed PCB design.

"iCD Design Integrity software features a myriad of functionality specifically developed for PCB designers."
– Barry Olney

iCD
In-Circuit Design Pty Ltd

www.icd.com.au

Overview of Test Methods to Determine Material Dk and Df

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

There are many different test methods which can be used to determine the Dk (dielectric constant or ε_r) and Df (dissipation factor, Tan Delta, loss tangent) for circuit materials. IPC has 12 different test methods to determine Dk for materials. Additionally, there are material Dk test methods defined by other industry organizations, universities, and many companies. I have a book on microwave material characterization which has over 80 different test methods to evaluate circuit materials for Dk and Df. The bottom line is, there is no perfect test method, and an engineer should use the test method that most closely emulates their end-use product. In this column, I'll give an overview of test method concepts and discuss common test methods which are used to determine Dk and Df.

Test Methods Concepts

There are two general categories for test methods used for high frequency circuit materials: material test methods and circuit test methods. The material test methods typically use fixtures to evaluate the raw dielectric material. This type of testing is evaluating the raw material only and without circuit fabrication variables. The other category is using circuit test vehicles to extract the Dk (and sometimes Df) based on circuit performance. Since the accuracy of the material test methods are dependent on fixture variations and the circuit test methods are dependent on circuit fabrication variations, the Dk/Df values extracted from these two different types of test methods should not be expected to be the same.

Candor
Circuit Boards

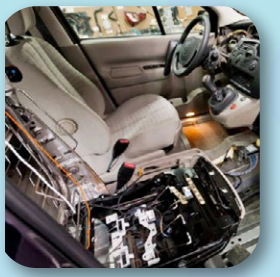
Innovative solutions for complex circuit board manufacturing

Click to see what we bring to each sector of the industry:

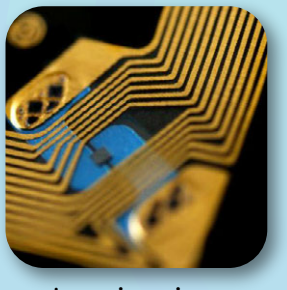
RF/Microwave PCBs

Power Electronics

High-Speed Data



Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

www.candorind.com | (416) 736-6306

Another issue is test methods will extract the Dk/Df values based on the orientation of the electric fields, in relation to the material being evaluated. Generally, some test methods will test the Z-axis (thickness axis) and other test methods will evaluate the X-Y plane of the material. Since most circuit materials used in the PCB industry are anisotropic, Dk is different on each axis of the material. It is possible to test the same piece of material with two different test methods and achieve two different Dk values, and both values are correct. If the material is anisotropic, one test method is evaluating the Z-axis of the material and the other test method is evaluating the X-Y plane; we should achieve different Dk values when testing the same material.

A few other considerations for test method concepts are material dispersion, copper surface roughness, and the use of resonant or transmission/reflection techniques. All materials have dispersion and that means the Dk will vary with a change in frequency. So, if the same material is tested twice using the same test method, but tested at different frequencies, there should be a difference reported in the Dk value. Basically, as frequency increases, the Dk value will slightly decrease.

All materials have dispersion and that means the Dk will vary with a change in frequency.

Copper surface roughness can slow the propagating wave and a slower wave will be perceived as a higher Dk value, regardless of the material Dk value. Some test methods are sensitive to the copper surface roughness and other test methods are not.

Lastly, it is well accepted that a test method which uses resonance is typically more accurate than a test method which uses transmission/reflection. The resonator test methods are usually more accurate, but they typically give Dk results at a discrete frequency or multiples of a discrete frequency. Many of the transmission/reflection techniques will give Dk results vs. frequency over a wide band of frequencies.

Following is an overview of several common test methods used in the PCB industry.

X-band Clamped Stripline Resonator Test per IPC-TM-650 2.5.5.5C

After the laminate is made, all copper is etched off and samples of the raw material are put inside a clamping fixture. The fixture has a very thin resonator circuit image in the middle, a ground plane on both sides of the resonator, and the material under test (MUT) is put between the resonator and the ground planes. When clamped together, the fixture has the RF structure of a stripline which is ground-signal-ground; more specific to this test method is the clamped structure, ground-MUT-signal-MUT-ground. This test method evaluates the Z-axis of the material for Dk and Df. It can be used at increments of 2.5 GHz and up to about 12.5 GHz. Typically, this test method is used at 10 GHz and is a relatively accurate test method.

One drawback to this test method is the reported Dk value will sometimes be lower than the Dk of the material, due to the natural issue of a clamped fixture having entrapped air (air has a Dk of about 1). Another potential issue for this test method is that when testing material with high anisotropy (Dk is very different on all three axes), the resonant peak can be altered in such a way to be less accurate for the Dk extraction. This is typically not a concern, except for some materials with high nominal Dk values, such as having a Dk of 6 or more. Overall, this is an excellent test method for a high-volume circuit material manufacturer to use for ensuring consistent Dk/Df properties of their material.

Split Cylinder Resonator Test per IPC-TM-650 2.5.5.13

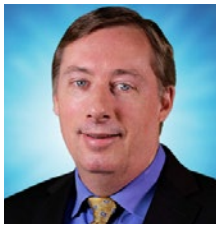
This test method is a cylinder resonator and as the name implies, it is split and can be opened and closed. After the laminate is made and all copper is etched off, the material to be tested (MUT) is put between the split cylinder and it is closed. The resonator will have several different resonant peaks for the user to choose to evaluate the Dk and Df. These different resonant peaks are at different frequencies. This test method will evaluate the X-Y plane of the material and not the Z-axis. Due to this difference and the fact this test method can also operate at the same frequency as the clamped stripline test method (Z-axis method), a comparison of data when evaluating the same material in these two tests will give information regarding the Dk anisotropy of the MUT. Additionally, if the material is anisotropic, it should be expected to obtain a different Dk value in the split cylinder test when evaluating the same material with the clamped stripline test.

Microstrip Ring Resonator Test Method

This method is a circuit test method, and the ring resonator circuit pattern is the test vehicle which is built on the material to be evaluated. The ring resonator typically has open-ended 50-ohm transmission lines (feedlines), which bring the RF energy to both sides of a ring circuit pattern (ring pattern looks like a very thin donut). The gaps between the two feedlines and the ring structure are critical and variations in the gap areas can cause the Dk extraction to be inaccurate. Also, if the ring circuit structure is plated with thicker copper in the PCB process, as compared to another ring resonator built on the exact same material but having thinner copper plating, the gap area will have more fields in air for the thick copper-plated circuit and the resonant peak will shift to be different than the ring resonator with the thinner copper plating thickness. Because of the difference in copper plating, the extracted Dk value will be different and is not correct when trying to evaluate the Dk of the material only. The gap and the copper plating thickness variation is a normal circuit variable; a circuit test will include this, but most material tests do not. Copper plating thickness has a natural variation in the PCB fabrication process and that thickness difference can cause inaccurate Dk results when using a ring resonator. Assuming the engineer is aware of the copper thickness issue and accounts for it in the extraction process, the correct Dk can be found. Additionally, this test method is affected by copper surface roughness, whereas the previous two test methods are not impacted by roughness effects. The ring resonator evaluates the Z-axis of the material.

Understanding the differences between test methods can be very important for a design engineer and especially when the engineer is comparing Dk and Df values on data sheets. If a comparison of data sheet Dk values is being done, the test method which generated the Dk values needs to be understood. Ideally, it would be best to compare Dk data using the same test method and at the same frequencies, but that is not feasible sometimes. However, if different test methods are used for Dk values on data sheets being compared, and both data sheets are reporting the Z-axis Dk at approximately the same frequency, it should be a good comparison.

When comparing datasheets or investigating material properties for a new design, it is important to consult the material manufacturer to understand how the critical data on the datasheet was obtained. **DESIGN007**



John Coonrod is technical marketing manager at Rogers Corporation. To read past columns or contact Coonrod, click here.

Sunny Patel On Design Data Formats


Sunny Patel

Feature Interview by the I-Connect007 Editorial Team

For this issue on design data file formats, we wanted to speak with someone who has experience using Gerber, IPC-2581, and ODB++. Sunny Patel of Candor Industries has used all three of these formats over the years, so we asked him to share his opinion of each format. He also points out why it might be more important to focus on flexibility than any one specific format.

Barry Matties: Sunny, this month we're looking at design data file formats: Gerber, IPC-2581, and ODB++. Which of these are the most common files coming into your shop?

Sunny Patel: Gerber and ODB++.

Matties: One of the things we hear is that IPC-2581 has a lot of benefits, but everyone is most familiar with Gerber, and they're saying, "Well, if a fabricator specifies that you have to provide 2581, they run the risk of losing the business." Does it matter to you what file type comes in?

Patel: No, to be honest, we're always trying to capture more customers. So there's rarely any pushback to a customer, even material- or technology-wise. It's really hard to push back on anything at this point, even with the extra business. The customer is in the driver's seat for sure.

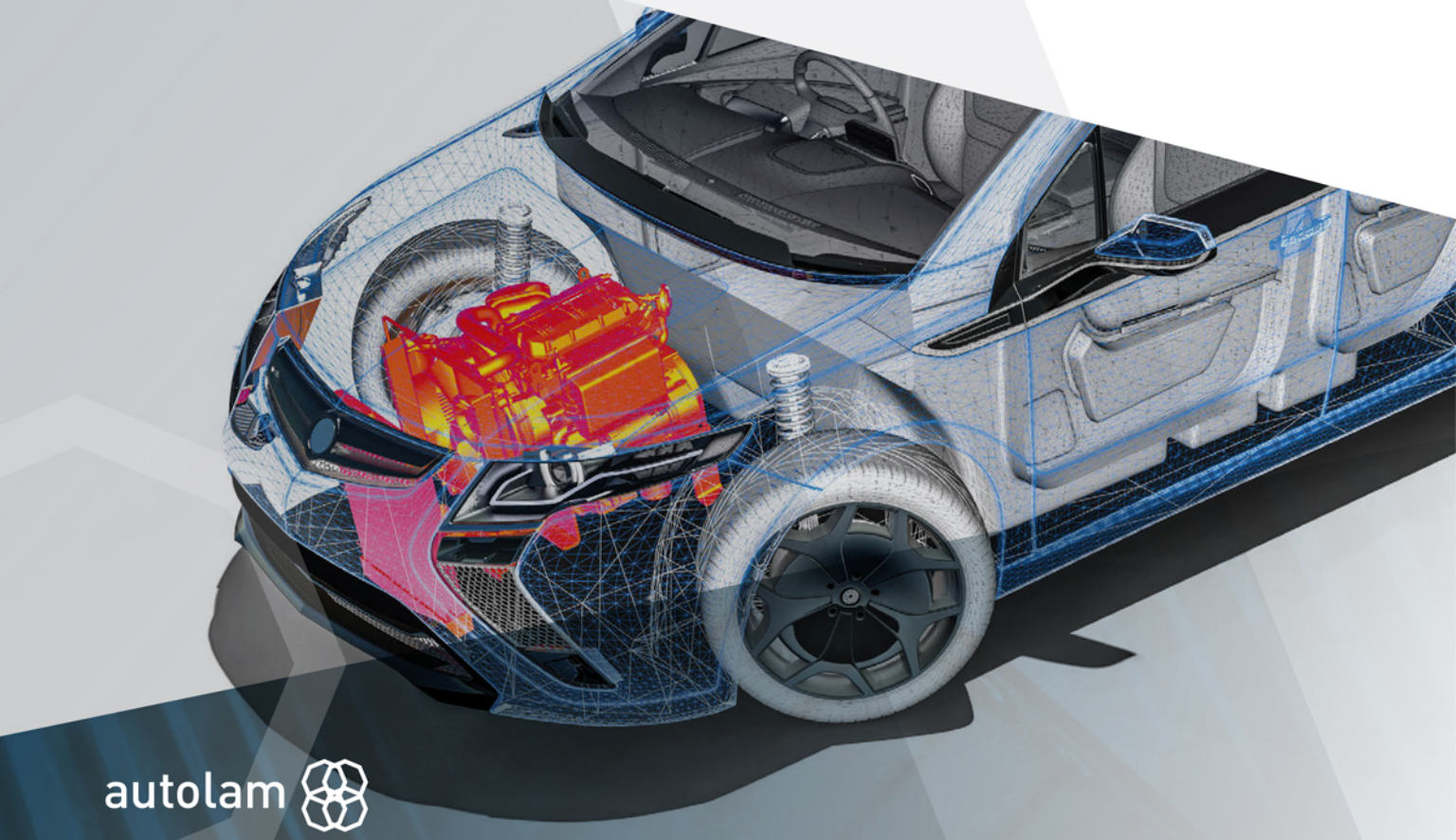
Matties: Is there any benefit that you see for one format over the others that the industry should be aware of?

Patel: I know that with Gerbers there's a lot more opportunity to put different types of data into our pre-CAM software. That could be just us, but we haven't had any issues so far, knock on wood, with any of the file formats. I just feel like there's more opportunity to edit and make micro-adjustments on the HDI side with Gerber. But let's see what happens. Everyone is always improving their file formats.

Matties: Right. It's interesting because there are a lot of people advocating for each of these. Obviously, everyone has a horse in the race somewhere. So, we're trying to understand

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

Automotive electronics technologies are evolving at an increasing rate. Paying attention to the properties of materials at the substrate level is the first step towards achieving the most stringent performance targets of today's automotive manufacturers. autolam offers the solutions demanded by the diverse and unique requirements of automotive applications today and in the future.

IATF 16949:2016 CERTIFICATION

venteclaminates.com

why Gerber is sticking around, despite the fact that the newer formats may offer some efficiency gains for the users.

Patel: Yes, ODB++ is definitely easier for most customers in terms of output, and it's just what it is. Whatever they output is what they're going to get. But I believe there's some risk of missing data errors if it's ODB++, but you know what, I've never run into any issues, especially with customers frequently sending IPC net list files to compare data. So, who knows?

Matties: But from your point of view, it's one or all, it doesn't matter.

Patel: So far, so good. Either way. Flexibility, to me, seems more important. We should be ready for anything, rather than asking for something specific.

Nolan Johnson: Sunny, I think I heard you say that you're using the Ucamco software in your CAM department?

Patel: Yes.

Johnson: And Ucamco is also the owner of the Gerber file format.

Patel: Exactly. So I may be a little biased there.

Johnson: Well, there's that, but really the gist of my question is: How well does that software handle the other formats like ODB++ and 2581?

Patel: There are no issues with ODB++. I haven't handled enough 2581 files to really make a decision.

Johnson: When your CAM department is processing the different files, is there a preparation efficiency discrepancy between the different file formats?

Patel: I haven't seen anything like that; I guess our pre-CAM software helps with all those things. So, I can't tell you if the software is working harder or not, but we seem to be doing alright.

Matties: Your input is really valuable on this topic. I can't tell you how much we appreciate your experience from actually using these formats. We hear a lot of anecdotal information along the way, but to have your voice, using all of these, is really helpful for us.

Andy Shaughnessy: Sunny, would you have any advice or any thoughts about the file formats?

Patel: Everyone likes their own way of doing things at the end of the day.

Shaughnessy: That seems to be the thing. We talked to people who just absolutely love Gerber or ODB or 2581. Is it really a problem to have these multiple formats? Does there have to be a "winner" out of all of this?

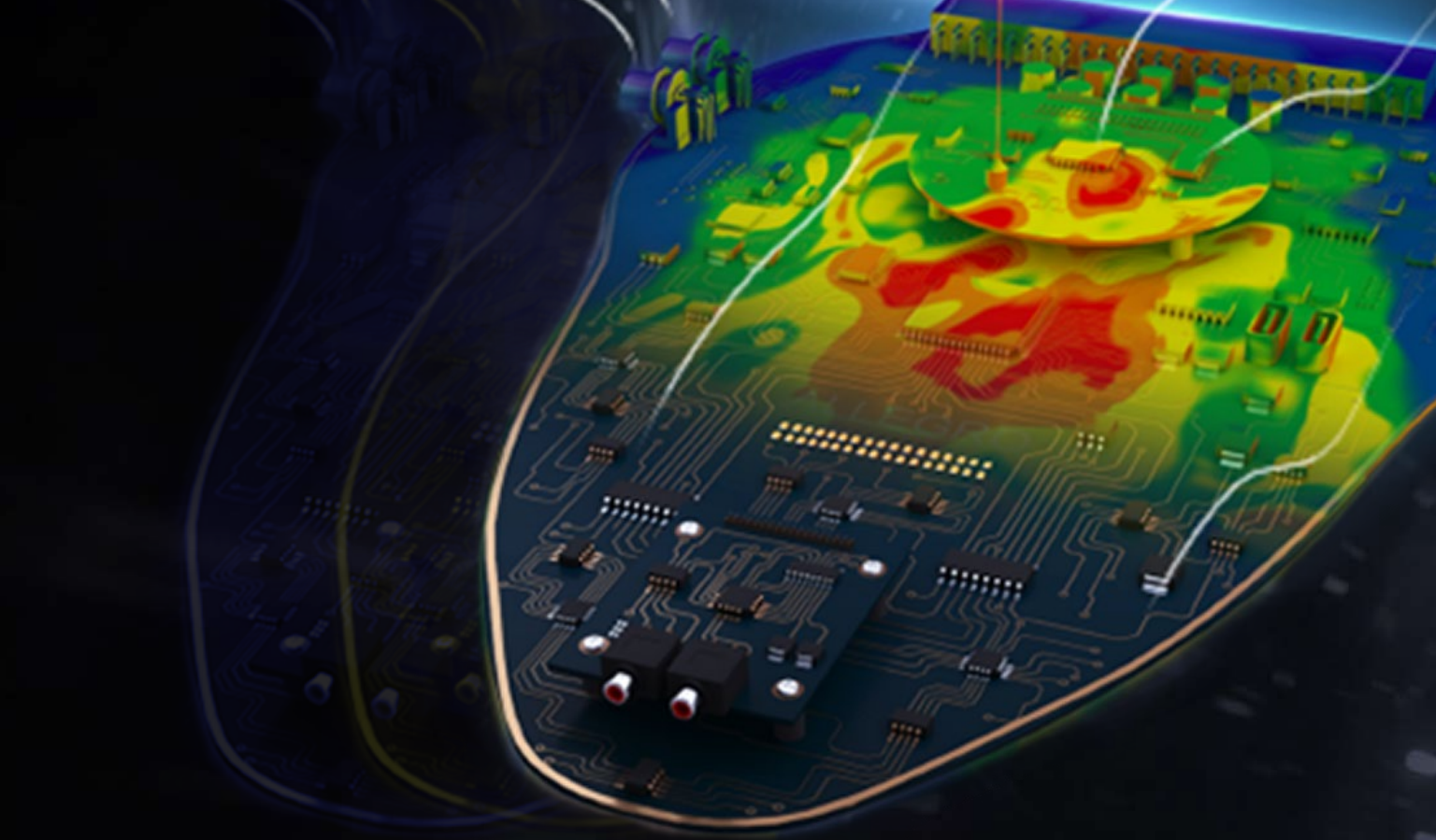
Patel: I think it's like Apple and Android, or USB 3.0 and Thunderbolt. Everyone is going to talk about it until something changes or something new actually unifies everything, I suppose.

Matties: Well, it's interesting because one of the things that we hear from some designers is that they're not going to change unless they're forced to change, or they find that it makes their life easier.

Patel: A lot easier. I don't think any file format right now has an extreme advantage. If it did, then it would be changing right now.

Johnson: I think we've covered everything. Sunny, that was awesome. Thank you.

Patel: Thank you all. DESIGN007



One Environment
Endless Possibilities

Solve all SI, PI, & EMC design challenges across the board.

Easily determine optimal design configurations, validate performance, and signoff on critical interconnects BEFORE prototype. Use Sigrity and be confident knowing your products are successful on the first pass with unified electrical analysis from Cadence.

Take Control Of Your Design Analysis »

EMA Design Automation®

cadence®

Fake Fudged Facts—
Using Software Tools to Get the Right High-speed Answer

The Pulse

by Martyn Gaudion, POLAR INSTRUMENTS

In the science of high-speed signalling, the signals obey the laws of physics, so when a design won't work or meet a specification, no amount of psychological persuasion will smooth the signal's path from source to load. Wouldn't life be different if speaking nicely (or shouting) at an underperforming circuit would spring it to life? (Though the application of warm air to drive out moisture can help, as long as it's dry, hot air.)

Feedback Facts

I'm not referring to electrical feedback, but process feedback. PCB fabricators will look at process measurements and adjust process parameters to optimise yields; this saves cost and time, and produces a better product for lower material cost over time. Trace width dielectric separation, glass-to-resin ratios, datasheet values, and measurements can all go into the mix when feeding back production measurement values to tailor transmission line characteristics to get better yields on the next build.

Fudge

Why the mention of this sugary sweet yet addictive substance? Sometimes the feedback from measurement doesn't correlate as expected; there is always a good reason for this, but until you know the root cause it can be tempting to "what if?" a hard-to-measure parameter from the datasheet that wasn't correct. Then the temptation is to "fudge" the results of the correlation by adjusting said tricky-to-measure parameter. Perhaps a more

www.prototron.com

Serious About Speed

For over 30 years Prototron Circuits has led the pack when it comes to providing quality circuit boards FAST.

Be it Class 3, AS9100, ITAR or MIL-PRF-31032, Prototron has the speed you need.

Prototron Circuits

America's **Board Source**

professional term would be to apply "goal seeking" or "empirical" correction. But this comes with a warning, as often noted by Polar's signal integrity product manager Neil Chamberlain, "Occasionally a little fudge tastes very good, but too much fudge can make you feel quite ill." The same is true with measurement correlation. Goal seeking—"fudging"—the results can actually be a very powerful tool, provided you have a good enough knowledge of the physics and the range over which goal seeking is reasonable.

How Much is Reasonable?

Sometimes a process engineer with an impedance correlation problem will microsection and find an inexplicable variation between the modelled and measured value of characteristic impedance. A microsection or two is all it takes to confirm geometries of the line and the dielectric separation of the planes. So, the temptation to enforce correlation is to take the only electrical characteristic that's not easy to measure and "goal seek it" to the correct value; that mystery characteristic is the dielectric constant.

Our technical staff has uncovered situations where the "goal seeking" of dielectric constant, commonly called Er, has gone way beyond the realistic limits of Er for a given material. Why should that be? Well, Er value could be part of the correlation but more often there are several parameters, each having leverage over the gap between measured and modelled. If a PCB fabricator goal-seeks outside of these limits, their customer may well raise eyebrows that the figure landed on is unachievable.

I have some personal experience with fabricators who are maybe new to the fabrication of transmission lines, goal seeking an Er of less than 2.0 on an FR-4 stripline. Given that the resin has an Er of 3.0 or thereabouts, and the glass of 6.0, then no amount of excess resin or resin starvation in FR-4 will get anywhere near 2, which is the Er of a pure PTFE material. It is also worth recalling that Z_O varies as 1/sq. root Er, so relatively large changes are needed in Er to effect small changes in modelled impedance. Line width and dielectric separation have a much larger effect.

Whilst on the subject of dielectric constant, many PCB engineers who may not have an electrical background obsess over Er at frequency. It's a bit trickier to measure Er than Z_O, but once you have a suitable coupon and the correct tools (in this case, a short line, long

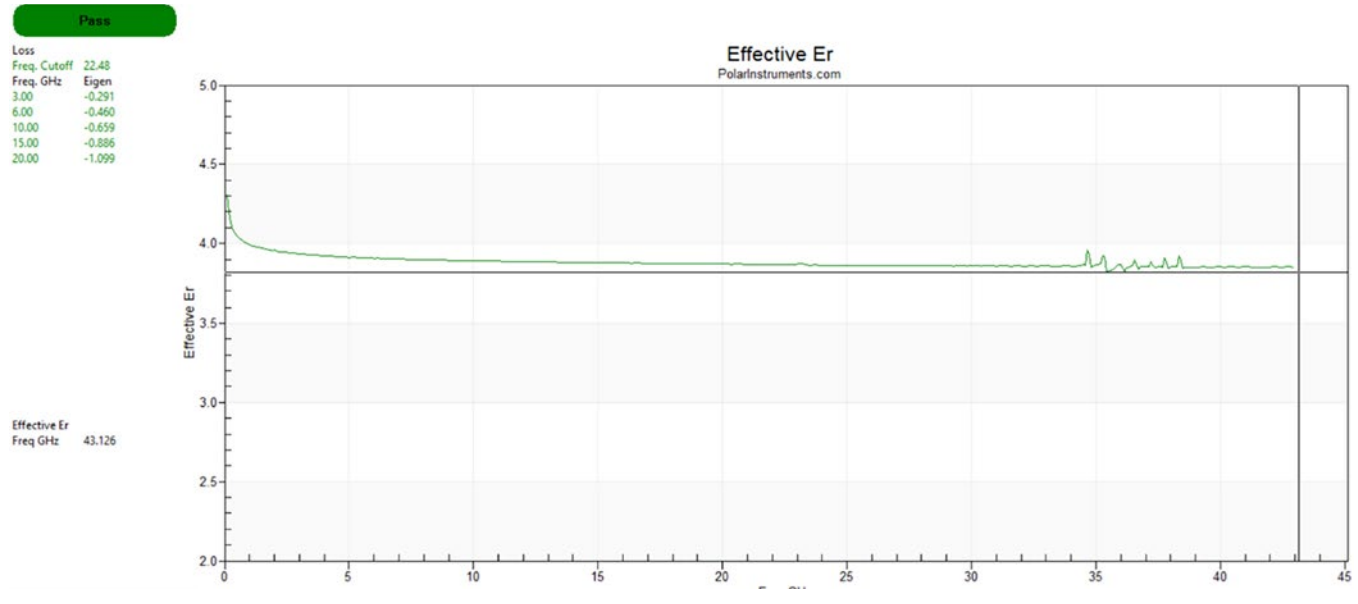


Figure 1: Effective Er from 100MHz to 43GHz. (Source: Polar Atlas for Anritsu VectorStar)

line coupon with Polar Atlas software driving an Anritsu VectorStar VNA), you can use the phase delay per unit length to get a very accurate measurement of transmission line effective Er vs. frequency. In Figure 1, you can see that above 1 GHz or so, the variation is absolutely minimal. Higher-loss substrates will see more variation, but surprisingly less than you expect.

Historically, with wider trace transmission lines, TDR measurement of impedance was a simple exercise of reading off the impedance from the TDR reflection, and the reflection over the whole coupon was flat. However, with fine lines, the traces exhibit an upward drift primarily from series resistance of the narrow trace. This upward drift must be removed by DC resistance compensation or launch point extrapolation before attempting correlation. Without correcting the Z_0 measurement for resistive effects, correlation will be poor, and if Er is used as a variable to correct the correlation then a false value for Er will be the result.

Conclusion

Whatever you are measuring or modelling it is important that you learn enough about the limits within the modelling tools' operation, and the physical limits of the "invisible" properties of the materials in the sample under test. Without background knowledge and intuition based on the knowledge and the careful application of measurement and modelling it can be easy to be seduced into correcting the wrong variable. **DESIGN007**

Martyn Gaudion is managing director of Polar Instruments Ltd. To read past columns or contact Gaudion, click here.

COLUMN EXCERPT: **All Systems Go!**

Comprehensive Thermal Analysis of a System Design

by Suketu Desai

In recent years, driven by the demand for smarter electronics, device designers have witnessed enormous scaling of large and hyperscale integrated circuits (ICs) and embraced development directions toward high density and reliability. These devices have increasingly higher thermal performance requirements—both transient and steady-state—and meeting them is becoming increasingly complex and time consuming.

Thermal Integrity and Challenges

Today, ensuring thermal integrity of an electronic device is a multi-dimensional problem. It straddles both the active (transistors and ICs) and passive (resistors, capacitors, and inductors) components, as well as the associated mechanical connectors and thermal dissipation elements. With all these mounted on one or more PCBs, ensuring thermal integrity across this full spectrum of components and related materials is a major and growing challenge. Not addressing thermal issues leads to thermal stress and early failure. As a designer, the typical challenges you face are:

- Performance is driving the need for accuracy, and hence the demand, for 3D thermal modeling of the system including all the components—active, passive, and mechanical
- Inability to accurately model the conduction, convection, and radiation of heat throughout the system
- Simulating and analyzing the thermal behavior of the system, both transient and steady state is becoming impossible with the proven legacy solutions because they are unable to scale to absorb the growing density and complexity of today's system designs
- The lack of accurate electrothermal co-simulation often leads to field failures
- Extensive prototyping is a major drag on cost and time to market.

To continue reading this column, click here.

Are Encapsulation Resins Suitable for EV Applications?

Sensible Design

by Beth Turner, ELECTROLUBE

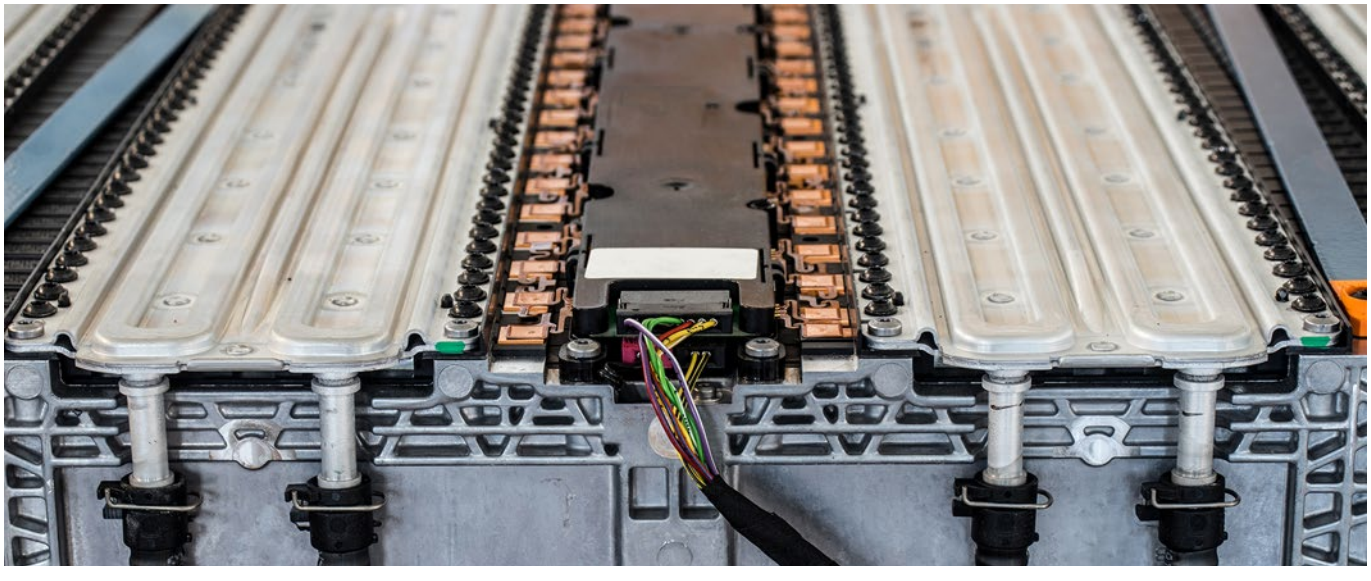
In this month's column, I will be exploring how resin chemistries can be incorporated by design engineers facing ever increasing new challenges from e-mobility applications—for instance, motors and generators offering higher levels of power output, noise reduction, and reliability. One of the biggest areas of concern for designers is the issue of heat impact on battery performance, particularly as drivers crossing over from traditional fuel vehicles to electric will expect no compromise in performance from their new electric/hybrid vehicle. Let's take a closer look at how resin systems can be implemented to improve performance, safety, reliability, and ultimately, lifetime.

Key Benefits of Using Resins in EV Applications

Protecting the electronics is vital to ensuring long-term performance and reliability. Encapsulation resins benefit e-vehicles in numerous ways, including improving thermal conductivity to dissipate heat away from hot spots, improved thermal endurance and stability in chemicals, and fast gap filling to provide resistance towards crack propagation. Encapsulation resins can also prevent costly maintenance and repair.

Why Encapsulate Electric Motors?

Historically, varnishing of the rotor and stator windings was common to achieve a minimum level of insulation; such varnishes were solvent-based and contained a high percentage of volatile organic compounds (VOCs). However, with increasing environmental awareness, legislation restricts the use of VOCs due to their emissions into the environment. Such varnishes do not fulfil the higher requirements of e-motors in the automotive industry where



Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

Use manufacturing data to generate a 3D facsimile of the finished product.

Verify

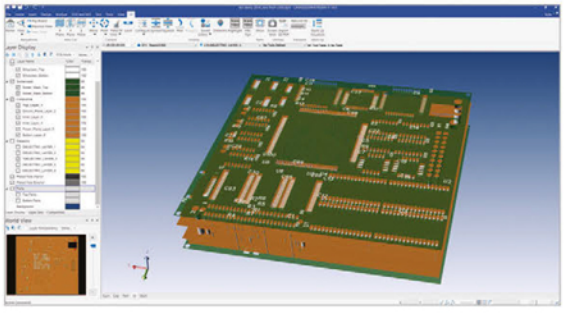
Ensure that manufacturing data is accurate for PCB construciton.

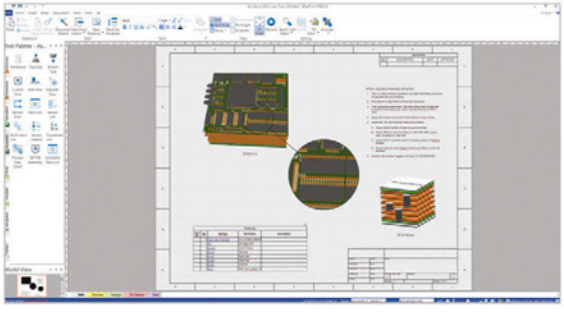
Panelize

Minimize costs in both PCB fabrication and assembly through maxiumum panel usage.

Document

Quickly and easily specify the PCBs form, fit and function.





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

DownStream Technologies, offers an integrated manufacturing data preparation solution for PCB post processing including manufacturing data visualization, stack-up definition, PCB panel visualization and support for 3D PCB documentation. This environment allows PCB designers and engineers to visualize, verify and document a printed circuit board before manufacturing.

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D

the challenges are higher power density caused by size reduction, increased power output, reliability, and endurance in harsh environments. This means there are now even more reasons to use encapsulation resins and they deliver plenty of value-added benefits:

- **All-around protection:** Acting as a barrier to the external environment, encapsulation resins provide physical, chemical, and thermal protection. They can physically dampen the impact of shock vibrations or dampen noise produced by the motor. Some encapsulation resins also benefit from flame retardant properties.
- **Electrical insulation:** Air and voids can create partial discharges between ground and primary insulation. Encapsulation resins can be used to completely fill all internal spaces, often helped by vacuum-potting methods to fill all the tiny spaces to guarantee electrical insulation.
- **Thermal dissipation:** Some encapsulation resins have high thermal conductivity and are extremely effective at dissipating heat away from hot spots and improve the overall thermal endurance.
- **Noise, vibration, and harshness (NVH):** Although e-motors are quieter than traditional internal combustion engines, and have fewer moving parts, electric vehicles create higher frequency noise that can be annoying over a long period of time. Resins are used to handle the NVH and add perceived value for any vehicle owner.

What Types of Resin Solutions Are Available?

Unlike aerospace applications, where two or three layers of redundancy may be built into systems, automotive designs typically must work the first time, every time, throughout the life of the product. There are a variety of expansive resin solutions that improve performance and reliability. By way of example, resin solutions for EV applications would include black epoxy resin, which is frequently chosen as the resin material to electrically insulate the coil ends of stator windings found in electric vehicles. This resin is highly resistant toward immersion in chemicals found in electric and hybrid vehicles, including cooling fluids, and has extremely wide operating temperature ranges. Its low viscosity also allows for easy potting of all the tiny spaces between the winding coils.

EV batteries produce vast amounts of heat during operation and whilst charging. To achieve prolonged battery life and higher efficiency, a potting material with a highly thermally conductive resin with low viscosity is ideally suited for potting cells within electric vehicle batteries. It also assists with securing cells in place whilst dissipating the heat away to the surroundings. The resin effectively creates a protective shield around the battery, enables adhesion, high temperature resistance, high thermal conductivity, and retention of characteristics throughout the thermal cycling process.

Are Changing Customer Requirements in Line?

There is clearly a change in customer requirements in line with the trend for EVs, such as improved range, faster charging times, and wider availability of charging stations. With each new iteration, a next generation of design typically means components get smaller in size and their power output increases. This normally means that the heat generated from higher density power output components increases. In turn, both short- and long-term operating temperatures of resins need to increase to perform sufficiently. Since resins are used in many areas on a vehicle, from sensors and displays to the motors themselves, a lot of development by Electrolube is focused on pushing the limits of thermal performance

and thermal conductivity to remain as the solutions provider of choice for the ever-changing automotive industry.

E-mobility is an evolving market, where the various performance criteria for vehicle components is constantly changing to keep up with the latest developments in technology. Encapsulation resins capable of dissipating heat away from hard working components clearly add value to the automotive electronics industry. I hope this has been of interest to those of you involved in EVs and helps make life a bit easier for those who are responsible for making the decisions on protecting components and circuitry. In my next column I will be exploring resin systems in more detail. DESIGN007

Beth Turner is head of encapsulation resins at Electrolube. To read past columns from Electrolube, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments,* and watch the micro webinar series "Coatings Uncoated!"

Excerpt: *The System Designer's Guide to... System Analysis*

Chapter 3: Limitations of Today's Electronic System Design

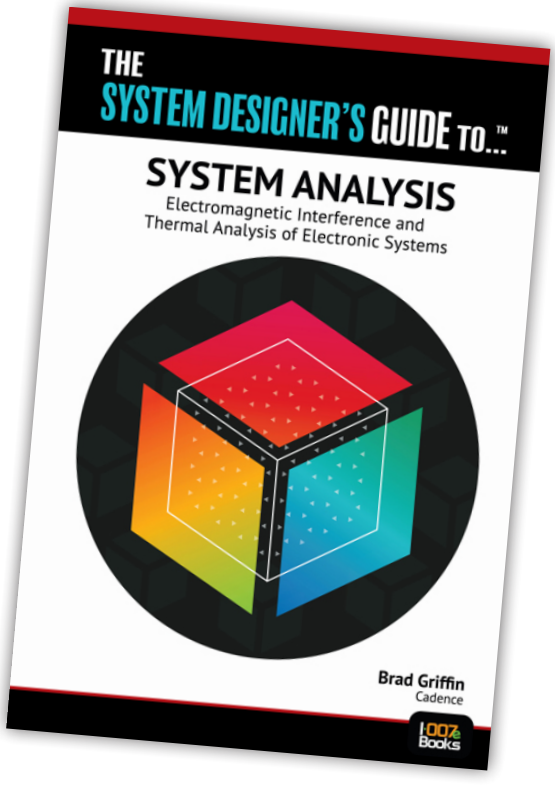
The rate of growth of transistors packed into an IC is slowing every year, marking the limits of physics. The economics of semiconductor logic scaling are also disappearing, and the cost per transistor has increased drastically. Advanced node chip design has become difficult, specialized, and expensive. SoCs are reaching the reticle limit. These limitations have made the industry look for alternatives beyond Moore's law.

The volume of data being analyzed is increasing in complexity and has created the requirement for better design methodologies. New application-based architectures push the limits of current design tools, and the requirements for PPA targets are much more aggressive than before. Advanced-node processes provide designers the opportunity to improve performance, reduce power, and meet area requirements through new device and manufacturing innovations. It's more than just doing synthesis and placing and routing better, instead it's a race to achieve computation excellence and high scalability. The cost of late market entry is another major challenge daunting the EDA industry. The traditional tools are not capable of providing a single impulse response to accurately simulate the system before manufacturing.

The transition from single monolithic to multi-chiplet architectures has introduced a plethora of issues to designers—like how to plan, manage, and optimize their top-level design and connectivity—and has led to the need for a new, system-level design management solution capable of aggregating data from the integrated circuit (IC) designer, the package designer, and the board designer, for system-level optimization and providing the top-level netlist for signoff connectivity verification. There is a need for methodologies and tools to handle the end-to-end flow from chip design to system analysis.

To download this free I-007eBook, click here.

To view the entire I-Connect007 library, click here.



Electronics Data Format Collaboration—Help Wanted

The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

Introduction

This month, I share some critical staffing requirements which need to be filled within the PCEA. Our Chairman, Steph Chavez, then expresses his excitement regarding the PCEA's involvement in recent electronics industry trade shows. I wrap up by providing our readers with a list of upcoming events.

PCEA Updates

This month's *Design007 Magazine* topic examining data formats aligns perfectly with the goals of the PCEA. The three data formats—Gerber, IPC-2581, and ODB++—are similar, yet very different regarding how they are leveraged by the differing stakeholder requirements of the PCB fabricator, assembler, and designer. The topic content is sure to raise as many questions as differing perspectives. Though, in the end, the data formats will never be expected to merge into one common industry standard solution providing all data needs to all stakeholders. A miraculous event such as this, if it ever could happen, might be called collaboration. Unfortunately, though, in our industry it seems as though collaboration rarely occurs without a profit motive.

As a nonprofit, PCEA's goal is to provide a place for all printed circuit engineering stakeholders to collaborate, educate, and inspire one another. We've been at it for over a year now, during some very tough times within our industry, and there has come a time for us to put out a call for help. While the PCEA has representation from some very heavy hitters in many disciplines of the printed circuit engi-

neering industry, we are inherently made up of leadership and membership which stems from the design portion of our industry. A few years ago, we consciously focused on the name of our organization to include the word engineering as opposed to design in order to include all the other important stakeholders—manufacturers, assemblers, test engineers, suppliers and others—along with designers, to collaborate, educate, and inspire each other. We are the Printed Circuit Engineering Association—not a design association—and if you are a stakeholder working on the manufacturing, test, assembly, procurement, or inspection of printed circuits, we need your help.

We are asking for your help in stepping up and proactively representing your respective colleagues in our organization.

We have an immediate need for executive level representation from the many complementing stakeholder areas mentioned above. We are asking for your help in stepping up and proactively representing your respective colleagues in our organization. We are looking for individuals who can unite with other representative stakeholders within our organizational chapters. We need your alter-design stakeholder perspective to homogenize our thinking as we move forward to serve this great industry. Are you willing to step in and serve? The pay is non-existent but the rewards of service in the PCEA can change your career trajectory overnight. If you are interested, contact PCEA Chairman Steph Chavez through our website contact page (pce-a.org/contact/) and let him know you believe that the PCEA should be more than design and design education; it needs to collaborate, be educated, and inspired by your alternatively important subject matter expertise.

Message from the Chairman


Stephen Chavez

As we start October, respective workloads and industry events are in full swing. PCEA is coming off its first official booth on a convention hall floor at a major industry "in person" conference. PCEA's booth at DesignCon 2021 was a great success. The marketing and events team did an amazing job bringing this together and showcasing polished professionals. They sure hit a home run on this one. Kudos to the PCEA team and to the support team of DesignCon for another successful event, especially considering today's circumstances and resurgence of the pandemic and a new variant. Even with this, PCEA did its part in collaborating with other associations or groups for the betterment of the industry.

We continued that same positive momentum going into another local PCEA chapter (Orange County) event on September 1. Although this was another successful virtual webinar, with great design and manufacturing content, the word on the street, per my follow-up conversation with local chapter Chairman Scott McCurdy, is chapter members can't wait to get back to their previous in-person meetings. I couldn't agree with them more. I was very excited to see over 100 attendees sign up and attend; a very strong industry engagement indeed. I love the continued collaboration between PCEA and American Standard Circuits, a PCEA sponsor and host of the event, and I want to personally thank American Standard Circuits for its continued support of PCEA. I thought the two topics presented, "Next Gen Line/Space Capability for PCB Designs" and "An Overview of Via Fill—Choosing the Right Type for Your Design Application," were on point for today's complex PWB designs needs. If you were not able to attend this event, no

worries. Look for the recording of it soon to be listed on our website.

With DesignCon in our rearview mirror, the next major event is PCB West, October 5–8. As I was not available to attend DesignCon, I will be attending PCB West. It will be my first industry event since the pandemic started. I am so excited to travel and attend this event but would be lying if didn't say I am a bit cautious and nervous with the travel logistics aspect related to the pandemic. I'm sure I'm not the only one feeling this way, but I am so looking forward to finally getting back into my element of in-person engagement, integration, and collaboration. I hope to see many of you there. I have no doubt that it will be another great industry conference.

After PCB West, there are two more outstanding scheduled conferences to attend here in the U.S.: SMTA International 2021 and PCB Carolina 2021. Both are highly recommended conferences to attend. Productronica is an overseas conference November 16–19 in Munich, Germany and is a great one to attend as well. So, there are lot of industry activities and opportunities taking place in these last remaining months of 2021. I hope you'll be able to take advantage of at least one of them.

As for PCEA membership and local chapter growth, we're continuing our positive trend upward. PCEA prides itself being an association for everyone in the industry. This means that we highly value every individual member, not just large corporations. PCEA is a common collective for everyone in the industry. As the saying goes, "There is strength in numbers." The collective of PCEA is only as good as its individual members. Therefore, as of today, our collective strength is extremely strong, and our positive synergy is very high indeed. We are now up to roughly 17 current local PCEA chapters with about six more in the infancy stage. It's amazing to see people who have that same knack tend to come together and flourish as a group. Check out our Events page for more.

As always, if you have anything to do with printed circuit engineering, I highly recommend you get involved with and join the PCEA collective, if you have not already. By joining the PCEA collective, your percentage of long-term professional development increases significantly.

Stay up to date with the up-and-coming industry events. There are so many free webinars, so please take advantage of these opportunities. To join the PCEA collective, please visit our website, pce-a.org.

I continue to wish you and your families health and safety. Best of success as 2021 comes into its final months.

—Stephen V. Chavez MIT, CPCD, CID+, Chairman

Next Month

As the industry marches on, I plan to interview new recruits to the PCEA from PCB fabrication, assembly, testing, and procurement. We'll discuss their plans to help attract more diversity into the PCEA and help foster collaboration, education, and inspiration.

Upcoming Events

Here is our list of upcoming events. Hope to see you there.

SMTA International 2021
Nov. 1–4, 2021
Minneapolis, MN

productronica
Nov. 16–19, 2021
Munich, Germany

PCB Carolina 2021
Nov. 10, 2021
Raleigh, NC

Altium Live 2022
Jan 26–28, 2022
San Diego Convention Center
San Diego, CA

IPC APEX EXPO
Jan 22–27, 2022
San Diego Convention Center
San Diego, CA

PCB East 2022
April 11–13, 2022
Marlborough, MA

Spread the word. If you have a significant electronics industry event that you would like to announce, please send me the details at kelly.dack.pcea@gmail.com, and we will consider adding it to the list.

Conclusion

It is often said there is strength in diversity. There is power in multiple points of view.

As a newer organization (and like many others), the PCEA has experienced the folly of groupthink and being over-represented by any one industry discipline. Likewise, we've also experienced the challenges presented by inviting open expression representing diversity of thought and multiple points of view. As we move forward, we are thankful believers that success in an organization and an industry as a whole is far better off after considering the perspectives of an entire body—not just the arm or leg.

As a nonprofit, we don't ask for much. But this month we are asking you to help us in our quest for diversity. We look forward to hearing from any "ankles, knees, hips, and heads" out there who wish to make a difference. We love the diverse perspectives shared in Design007 Magazine. We can't wait to work together within our chapters to pour these perspectives into the collaborative blender of our organization. Hopefully, someday, we can become inspired to shake hands, learn, collaborate, and pour out a smoothie of PCB data—or anything for that matter—which would be healthy for our entire industrial body. We hope you agree and can help us. Press pulse twice, blend, and drink up.

See you next month or sooner! **DESIGN007**

Kelly Dack, CIT, CID+, is the communication officer for the Printed Circuit Engineering Association (PCEA). To read past columns or contact Dack, click here.

It's Happening Everywhere—Be Ready

Digital Transformation

by AJ Incorvaia, SIEMENS DIGITAL INDUSTRIES SOFTWARE

As I speak to customers around the world, I keep hearing the same question: "How can your company help us achieve our digital transformation goals?" I'm surprised how frequently this question is asked, especially by many of my customers who are preparing by appointing executives to lead entire digital transformation teams. In fact, these efforts often reach beyond the domain of IT and into the heart of business strategies that include end-consumer use-models, supply chains, design, manufacturing, and field maintenance. Analyst firm IDC projects[1] that by 2022, nearly $2 trillion will be spent on solutions and services to fuel digital transformation.

Before we go any further, it probably makes sense to define "digital transformation." According to Deloitte, "digital transformation is about becoming a digital enterprise—an organization that uses technology to continuously evolve all aspects of its business models (what it offers, how it interacts with customers, and how it operates)."

A great example of a customer who is driving digital transformation into every aspect of its business is Polar, a leader in wearable training solutions for elite athletes. Polar's goal is to integrate every aspect of design, mechanical, electronics and materials with user diagnostics in an effort to not just drive better product design, but to be more responsive to market needs in general.

Today, according to Lifecycle Insights[2], a full 58% of all development projects either incur additional costs or fail to meet their delivery commitments. This amounts to enormous opportunity loss due to missed market windows. The reason? Insufficient readiness due to increasing product complexity, organizational complexity, and process complex-


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
ITAR

ity. These challenges are only growing more extreme with the advent of 5G, AI, AR, autonomous systems, and IoT everywhere.

So, digital transformation strategies need to be formulated today, but this it isn't easy, and I believe that there's quite a lot of work ahead. The good news is that critical enabling technologies are available to close the digital transformation gap within electronics systems design. The challenge is establishing a programmatic approach to both methodology and technology adoption in order to get ready for a digital future. So, where do we focus?

1. **Cross-domain collaboration.** It's amazing to me the degree to which silos still exist across so many different organizations, and this is after years of non-stop business-speak about the importance of tearing down silos. Look closely throughout the electronics industry and you'll see these barriers everywhere, separating teams and groups whose collaboration is essential to success—mechanical/electronic co-design, FPGA/PCB, and even design and manufacturing. Silos still exist and they are costly. But they can be dismantled more easily than ever using digital enterprise strategies and technologies that are available today.
2. **More verification, earlier in the cycle.** Even with today's complex designs, higher speeds, and advanced IC technologies, it remains quite common to see verification addressed in silos or within discrete groups. But, today it's possible to digitalize a large portion of verification and sign-off processes, from the schematic to manufacturing and in between. And since digital technologies often abstract complexity and optimize efficiencies via automation, many of these checks can now be completed with little-to-no domain expertise.
3. **Multi-board systems design.** Many in our industry are still primarily designing systems by starting with a discrete PCB and then proceeding to more complex multi-board systems, instead of starting at the system architecture and decomposing down into individual PCB logic and design. This approach not only fails to leverage the opportunity to make better decisions around trade-off and perform virtual simulation matter, but it also increases the risk of error associated with connectivity and signaling. Again, this can be handled today with readily available digital technologies.
4. **Design data management.** According to analyst firm Aberdeen[3], almost 30% of a PCB engineering teams' time is spent researching data—from component/libraries to work-in-process, to certified re-use of blocks. But establishing a formalized data management strategy is clearly the superior approach. In fact, it represents nothing less than the foundation of all digital transformation. And again, it can be achieved with technology that's here today.

Digital transformation is a journey, and like any journey, preparation is critical. I've presented four areas where I believe the PCB systems design community can take steps—today—to increase the success of their digital transformation programs.

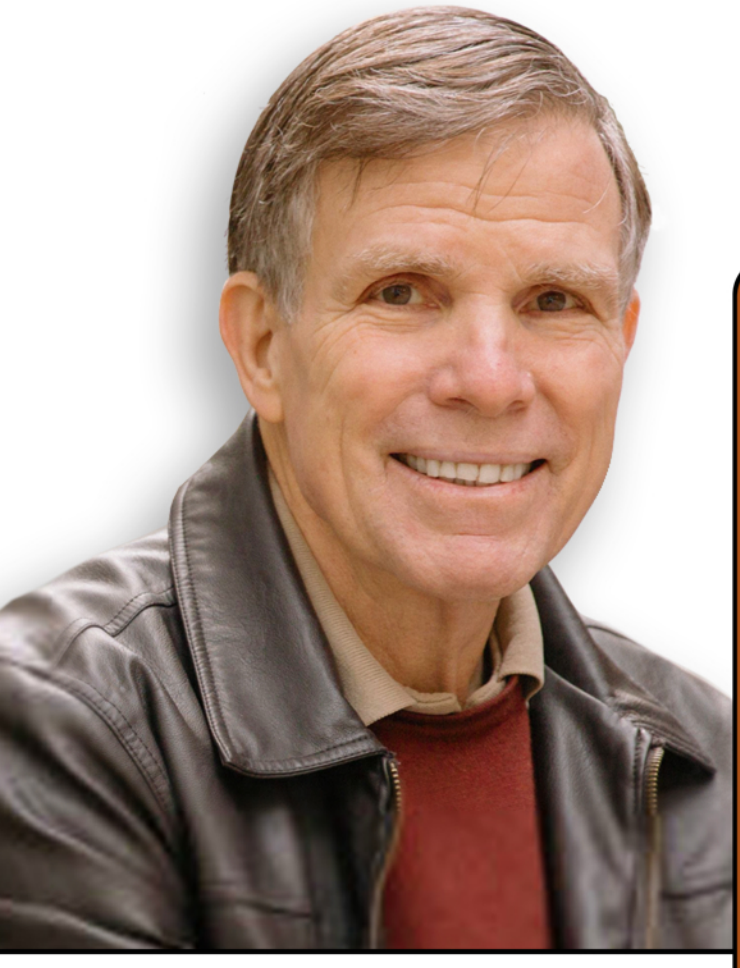
In future columns, we'll dig more deeply into ideas, case studies, observations and other content intended to help you on your digital journey, whether you're starting from the beginning, or already well on your way. **DESIGN007**

References

1. Worldwide Semiannual Digital Transformation Spending Guide, IDC, November 2018.
2. Simulation and Verification for Board Development, Lifecycle Insights, October 2019.
3. PCB Design Data Management: Smaller Windows and Higher Complexity, Aberdeen, May 2016.

AJ Incorvaia is senior vice president, Electronic Board Systems Division, Siemens Digital Industries Software.

I-007e WORKSHOP



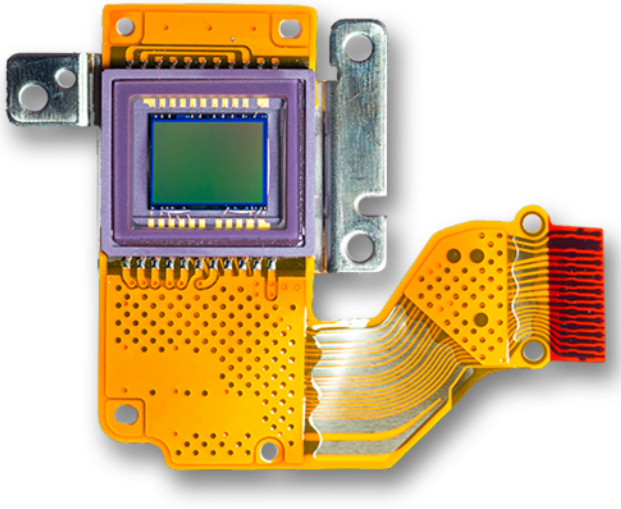
Online Training Workshop Series:

Flexible Circuit Technology

with Joe Fjelstad

This free on-demand workshop series is a comprehensive look into the structures, applications, materials, and manufacturing processes of flexible printed circuits.

START WATCHING



PRESENTED BY

I-Connect007
GOOD FOR THE INDUSTRY

SPONSORED BY

American Standard Circuits
Creative Innovations In Flex, Digital & Microwave Circuits

MilAero007 Highlights

The Government Circuit: A Summer Advocacy Update From IPC ▸

It's been a busy summer here at IPC, with policy debates heating up across the globe. Here in Washington D.C., the U.S. Congress adjourned for its August "District Work Period," but not before the U.S. Senate passed a major bipartisan infrastructure package, which we believe would positively affect our industry.

South Coast Circuits' Amanda Burgesser Invited to Join Chief's Private Network ▸

Chief's distinguished membership works to broaden female representation in senior positions within the business community.

Defense Speak Interpreted: What Does Convergence Mean to Defense? ▸

How can a simple term like "convergence" be confusing, even at the Department of Defense and the U.S. Army? Webster's dictionary defines convergence as "1. The act of converging and especially moving toward union or uniformity," and "4. The merging of distinct technologies, industries, or devices into a unified whole."

Libra Industries Dayton Facility Passes Audits for Aero/Defense, Manufacturing, Medical Certifications ▸

Libra Industries, a privately held systems integration and electronics manufacturing services provider, is pleased to announce that its Dayton facility has passed surveillance audits for its AS9100D (aerospace-defense), ISO 9001:2015 (manufacturing) and ISO 13485:2016 (medical) certifications.

Inovar Acquired by Spartronics ▸

Inovar Inc., an electronic manufacturing services provider to leading aerospace, defense, medical and commercial original equipment manufacturers companies, is pleased to announce its acquisition by Spartronics LLC.

3CEMS Group Prime Technology Receives Top Supplier 2021 Award from R&S ▸

3CEMS Group announced it has been recognized as a Top Supplier (2021) by Rohde & Schwarz, a company specializing in test and measurement, aerospace defense security, broadcast/media, and cybersecurity.

New Satellite Navigation Capabilities to Potentially Benefit Australia, New Zealand ▸

Thales Alenia Space, the joint venture between Thales (67%) and Leonardo (33%), announced that it has signed its first contract with the EU Agency for Space Programme (EUSPA), to provide new capabilities to Europe's EGNOS satellite navigation system.

Kodiak Assembly Solutions Now ITAR Compliant ▸

Contract electronic manufacturer Kodiak Assembly Solutions LP announces that it has successfully completed its ITAR registration.

BAE Systems Utilizes VJ Electronix's XQuik II ▸

VJ Electronix, Inc. is pleased to announce that BAE Systems Inc. has been using the XQuik II to solve an industry-wide problem with the industry-standard Waffle Pack design.

In times of challenging allocations, you need a supplier that will LEAD you through thick and thin
We accept the challenge! Utilizing our global supply network, we DELIVER an All-in-One solution!

WE DO IT ALL ...

We offer an All-in-One solution through our experience, integrity, and clear communication. Get on board for our joint journey into the future.

www.PCB-technologies.com

ALL IN ONE

PCB TECHNOLOGIES™
leading the way

Additive Electronics—
Are You One of the Curious?

PCB Talk

by Tara Dunn, AVERATEK

"Additive electronics" is a broad term in our industry. To many, this suggests 3D printing and the processes used to form circuit patterns with these additive methods. To others, this term conjures the image of newer PCB fabrication techniques that use semi-additive PCB fabrication processes to realize line width and space below the traditional 75-micron (3-mil) capabilities that are typically seen with subtractive etch processing. While both have interesting applications, the latter technology is the focus for this column. Recently, having the opportunity to attend two different sessions focused on additive electronics has driven home the fact that this technology is rapidly gaining momentum outside of the traditional high volume smartphone market, and is being adopted by several PCB fabricators offering low volume/high mix. Both events were very well attended, and these fabrication advancements are clearly catching the attention of the PCB design community.

For those new to these PCB technology advances, let's start with a few definitions:

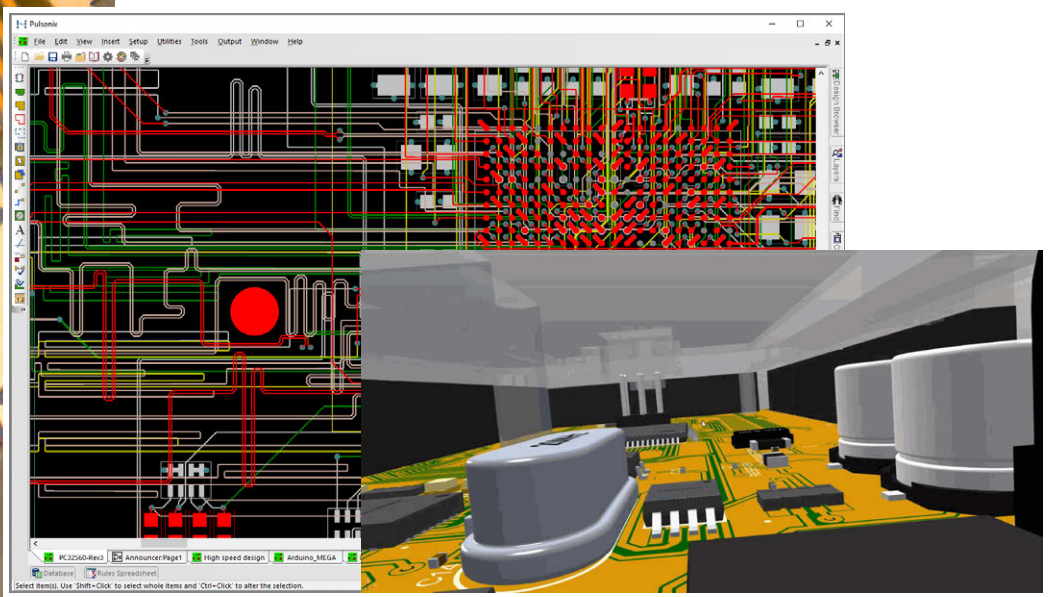
- **Subtractive etch process:** Traditional process used to fabricate printed circuit boards. This process begins with copper-clad laminate, which is masked and etched (copper is subtracted) to form traces
- **Additive PCB fabrication:** Starting with bare dielectric, this process utilizes additive process steps, rather than subtractive process steps to form traces
- **mSAP:** Modified, semi-additive process, adopted from IC fabrication practices

Pulsonix PCB Design Software

PCB design done right

New Version 11.0 Increases Speed up to 80%

Multi-threading and GPU processing in this new release create a gigantic leap in speed, dramatically reducing the time it takes to perform many PCB design operations, such as Design Rule Checking (DRC), Copper Pouring, and Gerber generation. That's PCB done right!



Advanced PCB Design Features

- Intuitive placement and routing
- Dynamic copper pour
- Full rules engine for automatic DRC
- High-speed design rules
- Heads-up display length matching
- Rigid-Flex, embedded components, and chip-on-board
- Revision control and PLM integration

Try Pulsonix yourself at pulsonix.com/trial

PULSONIX

- **SAP:** Semi-additive process, adopted from IC fabrication practices
- **SLP:** Substrate-like PCB, a PCB using mSAP or SAP technology instead of subtractive etch technology

While semi-additive processes are commonly used in IC substrate fabrication, they are a new process to the PCB fabrication community. As this technology is adapted to and integrated into PCB manufacturing, this has the potential to fill a gap between IC fabrication and PCB fabrication capabilities. In an arguably overly simplistic light, IC manufacturing processes are limited in overall panel size, and subtractive etch PCB fabrication is limited in line width and spacing capabilities. Blending these two worlds brings finer feature sizes to the PCB design community on larger panels (and lower cost) than IC fabrication allows.

As this technology is adapted to and integrated into PCB manufacturing, this has the potential to fill a gap between IC fabrication and PCB fabrication capabilities.

To add a little color to the difference between SAP and mSAP in a PCB manufacturing environment, both SAP and mSAP processing start with the core dielectric and a thin layer of copper. A common differentiation between the two processes is the thickness of the seed copper layer. Generally, SAP processing begins with a thin electroless copper coating (less than 1.5 mm) and mSAP begins with a thin laminated copper foil (greater than 1.5 mm). There are multiple ways to approach this technology and decisions can be based on volume requirements, costs, capital investment needed, and process knowledge.

Two recent events highlighted this process: A webinar with PCEA's Orange County chapter where American Standard Circuits and Averatek presented on ASC's newly installed A-SAP process (Averatek's semi-additive process) and via fill options; and an SMTA-hosted session on additive electronics, which provided information from IDTechEx on the state of additive electronics and an open discussion polling audience members to structure the technical program for February's Additive Electronics TechXchange being hosted in San Jose. I am inspired by the discussion at both events and extremely interested in the questions that are being asked as people familiarize themselves with these new technologies and the impacts of finer feature sizes on PCB design.

It is intuitive to first look at the obvious advantages to increased circuit density: overall size reduction, potential layer count reduction, potential microvia layer reduction, and lamination cycle reduction, just to get started.

Beyond this, there were some common questions bubbling up from those curious attendees:

- What is the peel strength of additive copper compared to traditional laminate?
- What materials are these processes compatible with?
- Can these additive processes come from plated through-holes or only build-up construction?
- Can additive layers and subtractive layers be combined in the same stackup?
- Can additive processes and subtractive processes be used on the same layer?
- Who is currently implementing these processes in their PCB manufacturing locations?
- Can you use via-in-pad-plated-over technology with additive traces?

- What thickness of copper can be achieved with semi-additive PCB processes?
- What is the impact on impedance?
- What is the impact on insertion loss and crosstalk?
- Which EDA tools support this?
- What are the impacts to assembly?

As the title of this column suggests, are you also curious about the PCB design benefits and impacts associated with these new technology advancements? It is an exciting time as we all come together to better understand how to best apply this technology. I personally believe that we are just starting to scratch the surface in terms of applications and benefits, and I am excited to have the opportunity to dig into these questions in upcoming columns.

What questions do you have regarding additive electronics? Contact me with those questions and I will add them to my growing list of topics to explore. **DESIGN007**

Tara Dunn is the vice president of marketing and business development for Averatek. To read past columns or contact Dunn, click here.

24 Essential Skills for Engineers: The Story Behind the Book

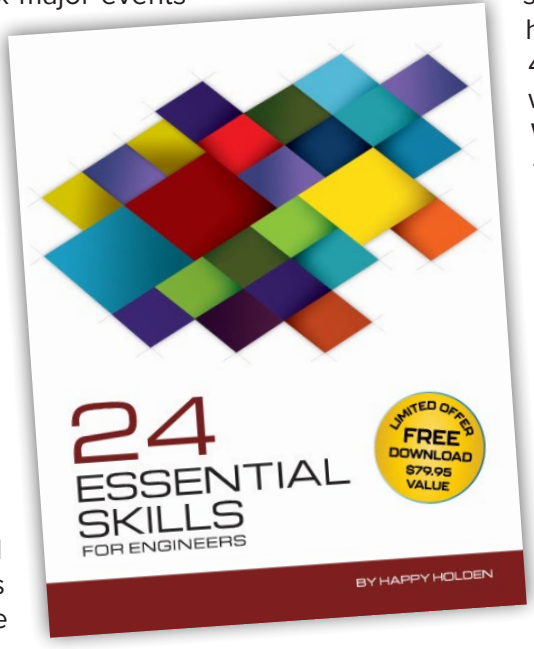
Interview by Nolan Johnson

In this interview with I-Connect007's own Happy Holden about his newest book, *24 Essential Skills for Engineers,* which he wrote over the span of his career, he highlights some particular moments from his time working at HP and as CTO of Foxconn which inspired many of the book's chapters.

Nolan Johnson: Happy how did you come to be inspired to write this book?

Happy Holden: There were six major events that set up the book. Although the book initially was written in 2011 and 2012, it was an accumulation of over 30 years of working and writing.

The first event was in 1966 at Oregon State University. Even as sophomores in chemical engineering, they had us writing simulations on the unit operation equipment, which, when I was interviewing with HP, they were surprised and said, "What? Only graduate students do that." I said, "Well, they forgot to tell us that you had to be a graduate student in order to write simulations." Nonetheless, in my sophomore year they had us writing complete digital twin simulations to add to the library of the system we called DISCOSA. The HP people were flabbergasted. One of the reasons they were interviewing me was because I had so much computer experience, which they said only graduate students and PhD students had.



Johnson: What other events precipitated this book?

Holden: The next big event was in 1974 with the success of the HP35 calculator. We had geared up to produce 300 or 400 a month. Instead, the orders were coming in at 3,000 an hour. We were totally unprepared for the fact that everybody in the world wanted one, as marketing had told us that at $400 each, we would only sell 300 to 400 each month.

To read the full interview with Happy Holden regarding *24 Essential Skills for Engineers,* sponsored by Calumet Electronics Corp., **click here.** The book is available at no charge for a limited time. To download the book, **click here.**

DFM 101: PCB Via Structures

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. This article is the latest in a series that will discuss these cost drivers (from the PCB manufacturer's perspective) and the design decisions that will impact product reliability.

DFM

Design for manufacturing (DFM) is defined as the practice of designing printed circuit boards that meet not only the capabilities of the customer's assembly manufacturing process, but also the capabilities of the board fabrication process at the lowest possible cost. While not a substitute for early design engagement with the PCB fabricator, these articles will provide guidelines that will help to "design for success."

Microvias

One of the most important technological advancements that made HDI viable was development of the microvia: a very small hole (typically 0.006" or smaller) that only connects certain layers either as "blind" or "buried" via holes. This represents a totally new way of making electrical connections between layers on a PCB. Traditional PCB technology has utilized "through-holes," which by definition, are drilled through the entire PCB connecting the two outside layers with all of the internal layers. The ability to strategically connect only certain pads on certain layers greatly reduces the real estate needed for a PCB design and allows a much greater density in a smaller footprint. Figure 1 shows through-holes and buried and blind vias.

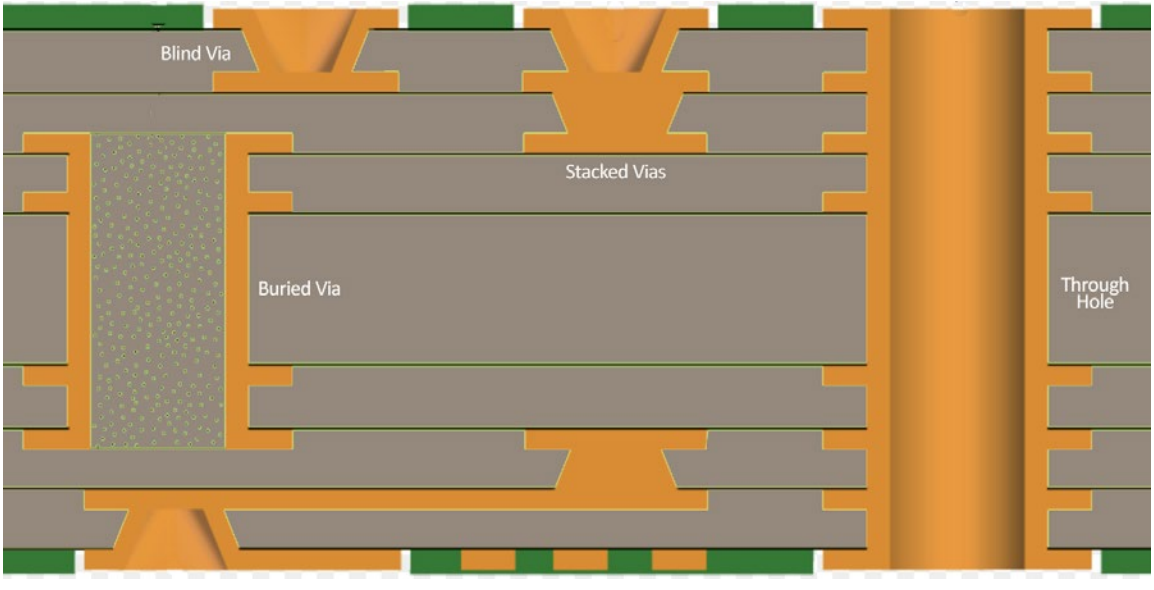


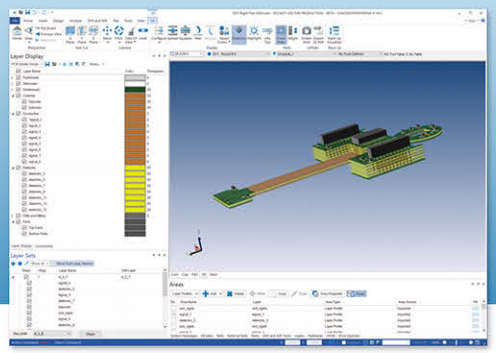
Figure 1: Microvias vs. through-hole vias.

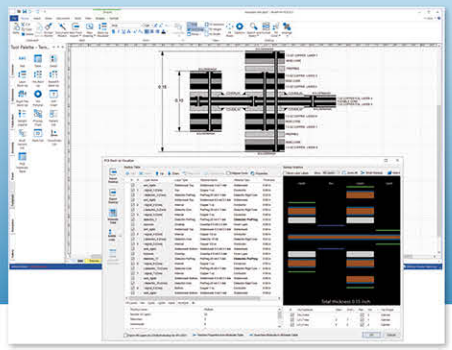
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

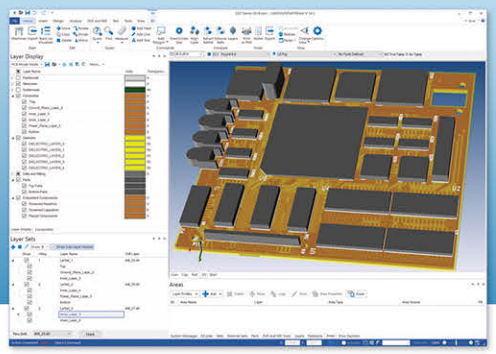
BluePrint-PCB®

CAM350®

- Import and Visualize Flex, Rigid-Flex and Embedded Component Designs
- 3D Visualization to Validate PCB Construction and Component Assembly
- Manage Variable Stackup Zones for Rigid-Flex Designs
- Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation







DownStream Technologies

For more information visit downstreamtech.com or call (508) 970-0670

Types of Microvias

- **Blind via:** Used to connect one surface layer with at least one internal layer
- **Buried via:** Used to create connections of internal layers with no contact to the surface layers
- **Via-in-pad:** A type of blind via in which the via hole is drilled in a surface mount pad, eliminating the need to run a trace and via pad from each SMT pad
- **Filled vias:** Completely filling the microvia with either a non-conductive or conductive paste. Conductive fill is typically used for heat dissipation and non-conductive fill is used to closely match the thermal expansion of the substrate.

Microvia Formation

Microvias can be formed through a number of methods, primarily mechanical drilling, laser drilling and sequential lamination.

- **Mechanical drilling:** Uses traditional drilling equipment to mechanically form holes, but typically limited to 0.006" diameter and dependent on the depth needed
- **Laser drilling:** Special drilling equipment that utilizes a laser to form the hole and can go down to 0.001" in diameter
- **Sequential lamination:** A process where the microvias are drilled all the way through a sub-panel of the layers that need to be connected by the via, which could require multiple lamination, plating, filling and planarization operations (Figure 2).

Stacked vs. Staggered Microvias

- **Stacked:** Microvias that are electrically connected and literally stacked vertically on top of each other through various layers of the PCB
- **Staggered:** Microvias that are electrically connected and offset to one another through various layers of the PCB (Figure 3)

Via-in-Pad Microvias

The via-in-pad production process allows you to place vias in the surface of the flat lands on your PCB by plating the via, filling it with

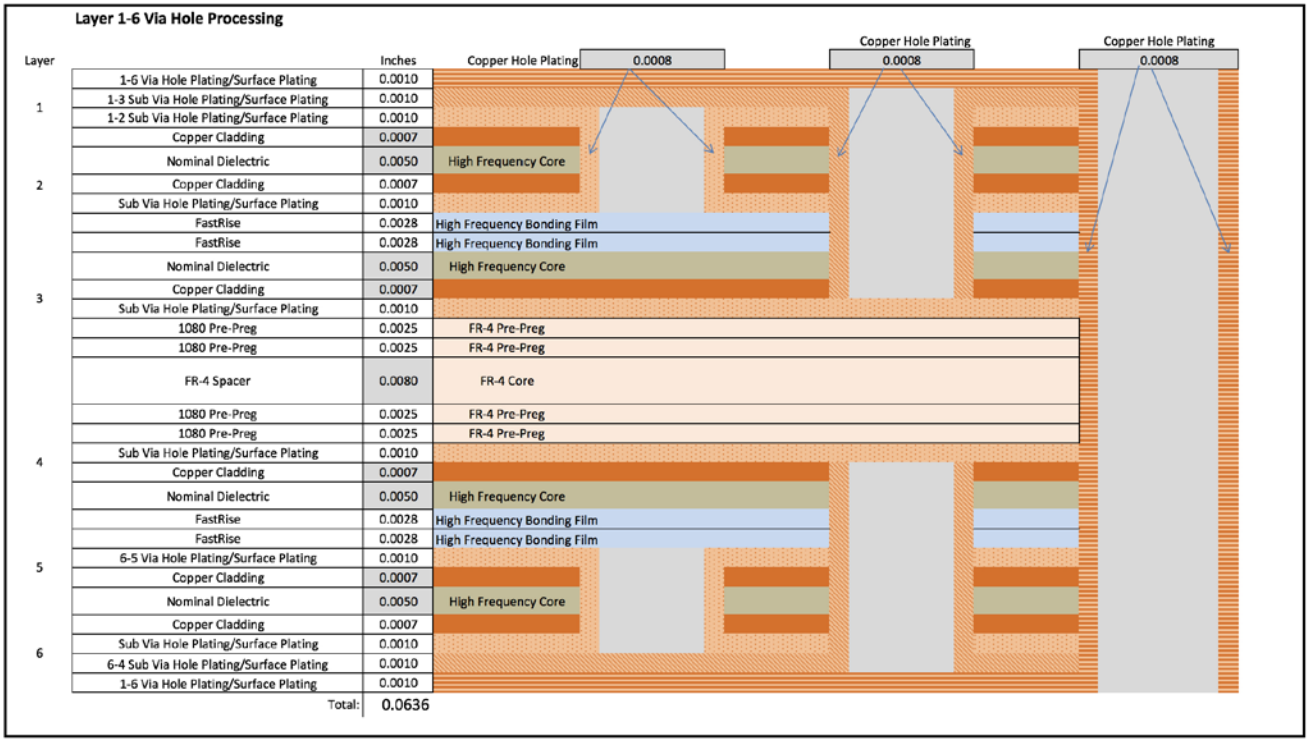


Figure 2: Sequential lamination.

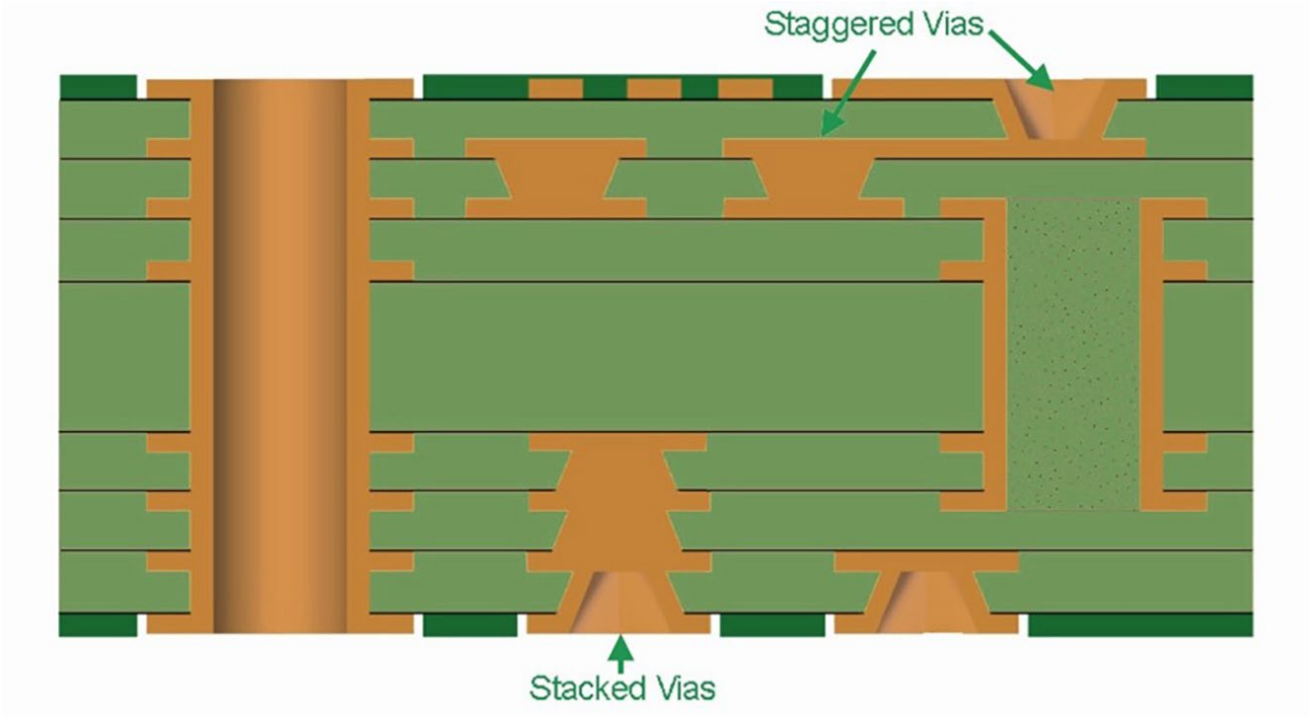


Figure 3: Staggered and stacked microvias.

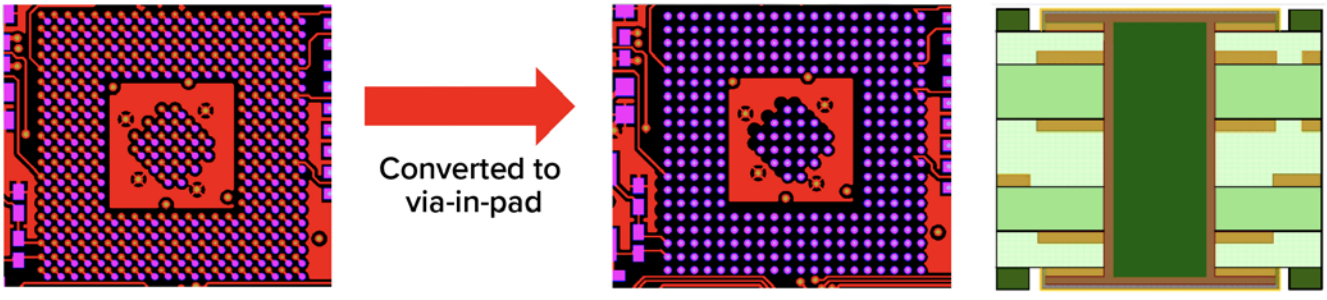


Figure 4: Via-in-pad.

one of the various fill types, capping it and, finally, plating over it. Via-in-pad is typically a 10- to 12-step process that requires specialized equipment and skilled technicians. Via-in-pad is often an optimum choice for HDI PCBs because it can simplify thermal management, reduce space requirements, and provide one of the shortest ways to bypass capacitors for high-frequency designs (Figure 4).

Understanding the cost drivers in PCB fabrication and early engagement between the designer and the fabricator are crucial elements that lead to cost-effective design success. Following your fabricator's DFM guidelines is the first place to start. **DESIGN007**

Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs* and *Flex and Rigid-Flex Fundamentals;* and author of *Thermal Management: A Fabricator's Perspective*. Visit I-007eBooks.com to download these and other free, educational titles. He also co-authored *"Fundamentals of Printed Circuit Board Technologies,"* and is an I-Connect007 columnist. To read past columns, or contact Vardya, click here.

INTERNATIONAL ELECTRONICS CIRCUIT EXHIBITION (SHENZHEN)
国际电子电路(深圳)展览会

HKPCA SHOW

5G

CONNECTING THE WORLD
万物互联

2021.12.8-10

深圳国际会展中心 (宝安)
Shenzhen World Exhibition & Convention Center (Bao'an)

www.HKPCAshow.org

主办单位
Organizers

HKPCA
Hong Kong Printed Circuit Association
香港綫路板協會

CPCA
中国电子电路行业协会

支持单位
Supporting Organization

CCPIT

承办单位
Event Manager

BAOBAB TREE 柏堡

Follow Us on WeChat
关注展会官方微信
获取最新资讯

Follow Us on LinkedIn
关注展会官方领英
获取最新资讯

FLEX007
A SPECIAL DESIGN007 MAGAZINE SECTION

RealTime with... American Standard Circuits: Approaches to Flex and Rigid-Flex PCBs

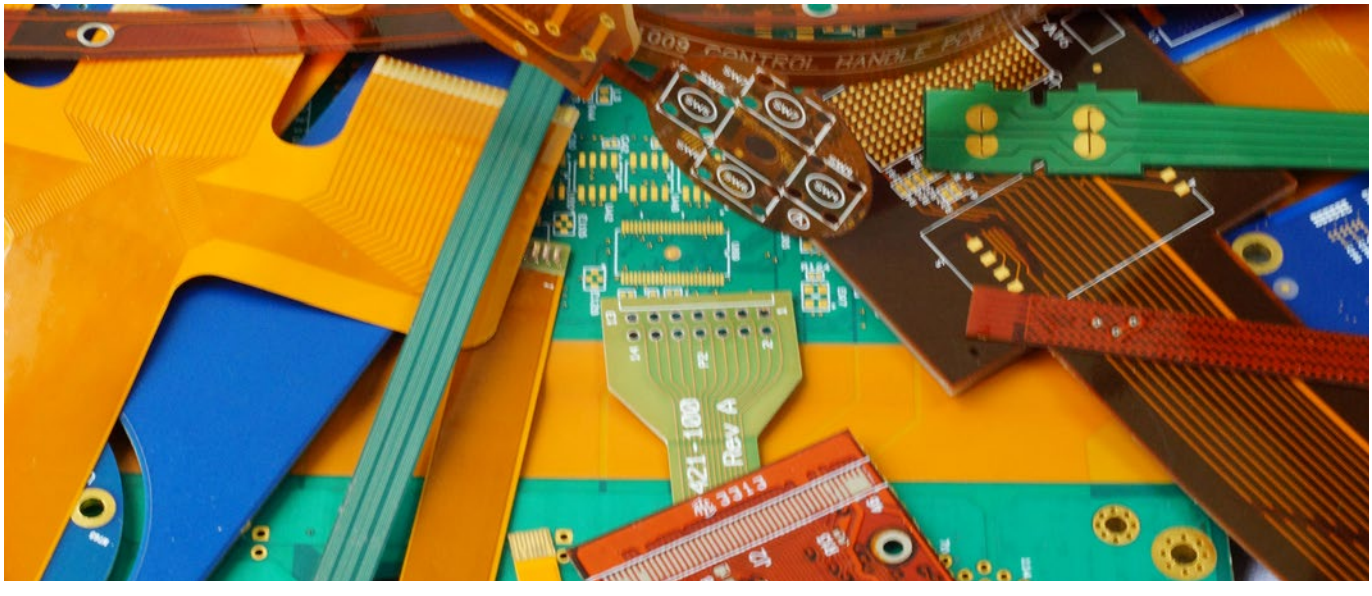
Flex007 Article By Pete Starkey
I-CONNECT007

In the first of a series of three RealTime with... interviews, I-Connect007 Managing Editor Nolan Johnson got some no-nonsense answers from Anaya Vardya and Dave Lackey on the topic of flex and rigid-flex PCBs. I was impressed by ASC's generosity in sharing knowledge and technology.

President and CEO Anaya Vardya introduced American Standard Circuits, a well-diversified PCB manufacturer, founded in 1988 and skilled in the fabrication of flex and rigid-flex, RF/microwave, digital boards, and a host of different thermal management solutions. As a total solutions provider, ASC focuses on working with designers up front in the process, minimising the number of churns, ensuring good manufacturability, and a high first-time success rate.

It was refreshing to have the opportunity to witness a down-to-earth Q&A session: intelligent questions from Nolan Johnson and knowledgeable answers from ASC's flex specialist Dave Lackey with an emphasis on in-depth, practical know-how and the importance of getting involved in the design process as early as possible.

Johnson remarked on the number of different approaches to flex design. Lackey used a



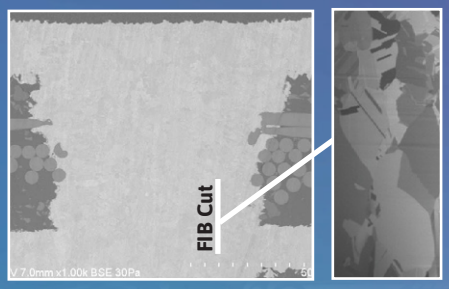
NEXT GENERATION

METALLIZATION TECHNOLOGIES

specifically designed to **SOLVE FLEXIBLE CIRCUITRY CHALLENGES**

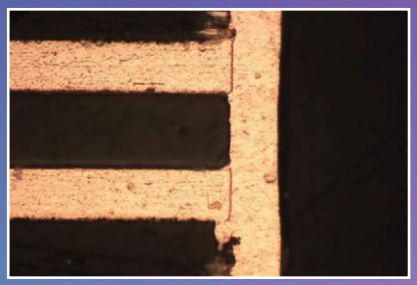
Blackhole LE / Eclipse LE

Low etch direct metallization processes with minimal etching that enable subtractive HDI and mSAP on a wide range of materials.



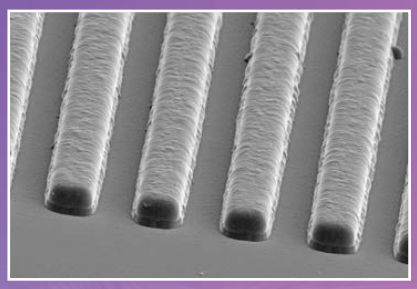
ViaDep 4550

Versatile electroless copper process that deposits copper with zero internal stress, enabling excellent adhesion on flex and rigid-flex designs.



Systek SAP & SAP Flex

Semi-additive processes for creation of initiation layers in package substrates and with excellent adhesion to flexible materials for ultra fine lines and spaces.



MacDermid Alpha ELECTRONICS SOLUTIONS

CONTACT US

series of clear photographs and schematics to describe three main categories:

1. Pure flex circuits, whether simple single-sided or complex multilayer, were constructed from all-flexible materials, copper-clad substrates, adhesives and bond-plies, with laminated polyimide coverlayers. In certain instances, liquid photoimageable solder mask could be used as an alternative.
2. Flex with stiffeners to add rigidity in certain areas. Different stiffening materials were used for different applications, temporary or permanent, in component-assembly areas or to reinforce connectors.
3. Rigid-flex, a combination of flexible and rigid materials, with flex layers incorporated within the rigid areas enabling the elimination of cables and connectors to achieve more compact packaging and higher reliability.

In response to Johnson's enquiry about typical applications, Lackey declared that there was an unlimited number, from simple connecting ribbons to complex devices with 90- and 180-degree turns enabling smaller-volume higher-reliability packaging.

Regarding criteria for successful design and reliability, Lackey drew a distinction between static and dynamic flex, the former being bent to fit, the latter being subject to continuous motion. It was essential to recognise the actual mechanical performance requirement in a particular application and to understand bend radius in relation to the thickness of the flexible materials and the geometry and orientation of the conductors.

The contribution of the mechanical designer was as fundamental as that of the electronics designer, and both could benefit enormously by involving the PCB fabricator as early as possible in the design cycle. The fabricator could advise on the choice and suitability of materials and constructions, as well as the benefits of details like cross-hatched ground planes and knowing how to properly model characteristic impedance. "A lot of factors to balance: here's what we can do to make this work." Lackey quoted case histories and examples of improving manufacturing yield and reliability while saving on overall cost.

His opinions were echoed in Vardya's closing words: "If the viewer takes nothing else away from this discussion, it is to really involve your PCB fabricator right from the word 'Go.' The most important thing any designer can do in designing a flex circuit is to work with the PCB fabricator. They will save you tons of time and you will be much better off for having done that."

The most important thing any designer can do in designing a flex circuit is to work with the PCB fabricator.

Finally, in answer to Johnson's question about critical things for me to keep in mind, Lackey advised giving careful consideration to what IPC classification was required, avoiding specifying name-brand material by referring to IPC slash-sheets and providing clear fabrication notes, particularly if quick turnaround was needed.

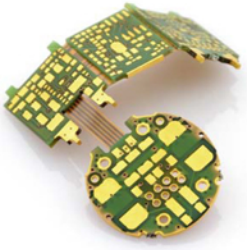
In my opinion, taking the time to view this RealTime with... video is 15 minutes well-spent; excellent information professionally presented, and the opportunity to submit further questions by email. Thank you ASC.

To view *RealTime with ... American Standard Circuits:* Flex and Rigid-Flex PCBs, click here.

FLEX007

Pete Starkey is an I-Connect007 technical editor.

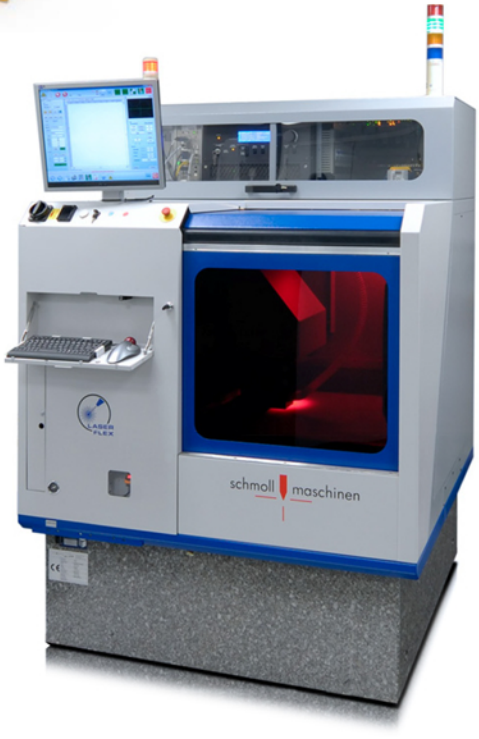
Focused on Flex



schmoll optronics

LaserμFlex series

The versatility and precision of laser processing makes it the ideal tool for machining a wide range of materials.



Typical applications:

- Polymer cover foil
- Covered polymer laminates
- Thin, rigid-flex materials
- Inner layers and prepregs
- Separation or decap from tracks
- Structuring cavities
- Micro-via and through-hole drilling

1991
2021

BURKLE
NORTH AMERICA, INC.

For more information call us: +1 (336) 660-2701

burkleamerica.com

Flex007 Highlights

Volunteers Honored for Contributions to IPC and the Electronics Industry ▸

IPC presented Committee Leadership, Special Recognition and Distinguished Committee Service Awards on August 30 at its SummerCom Standards Development Committee meetings in Milwaukee, Wisc.

Flex DFM: When All Things Must Be Considered ▸

One of the best ways to avoid flex failures is by communicating with your flex fabricator early and often. Here are a couple key reasons why involving your flex and rigid-flex supplier early in the design of your product will help you save time and money—and produce a more reliable flex circuit.

Additive Circuits Technologies Acquires Bench 2 Bench Technologies ▸

Additive Circuits Technologies LLC (ACT) announced it has acquired Bench 2 Bench Technologies, a Fullerton, Calif.-based high-performance manufacturer of flexible circuit boards for the medical device markets.

Tuning Flexible Circuits with Light ▸

Researchers use ultraviolet light to tune circuit performance with a photoreactive polymer based on controlled changes in the chemical structure, which may lead to the development of wearable electronics and medical sensors.

Taiflex Reports 19% Revenue Growth in Jan-Aug 2021 Sales ▸

Taiflex Scientific Co. Ltd, a Taiwan-based manufacturer of flexible printed circuit materials, has announced consolidated revenue of NT$876 million ($31.67 million at $1:NT$27.66) in August 2021, up by 1.8% year-on-year, but down by 7.5% from the previous month.

Flexium Reports 24% YoY Revenue Growth for Jan–Aug 2021 ▸

Taiwan-based flexible printed circuit (FPC) manufacturer Flexium Interconnect Inc. has posted sales of NT$3.065 billion ($110.78 million at $1:NT$27.67) in August, up by 20.4% year-on-year, and by 10% from the previous month.

Trackwise Reports 71% Revenue Growth for 1H 2021 ▸

Trackwise Designs, a provider of specialist products using printed circuit technology, has reported revenues of £4.1 million for the six months ended 30 June 2021, up by 71% from the £2.4 million revenues in the first half of last year.

Nan Ya PCB Reports 30% YoY Revenue Growth for August ▸

Nan Ya Printed Circuit Board Corp. has posted unaudited sales of NT$4.7 billion ($170.36 million at $1=NT$27.73) in August 2021, down by 3.2% from the previous month, but up by 30% year-on-year.

Matrix Announces Expanded Inventory for Flex Materials ▸

Matrix Electronics, a North American quick-turn supplier for raw materials to the printed circuit market, announced that they have completed an expansion of their inventory of flex circuit materials to support the growing flex and rigid-flex circuit business.

PREMIER GLOBAL SUPPLIER for **WEARABLES**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

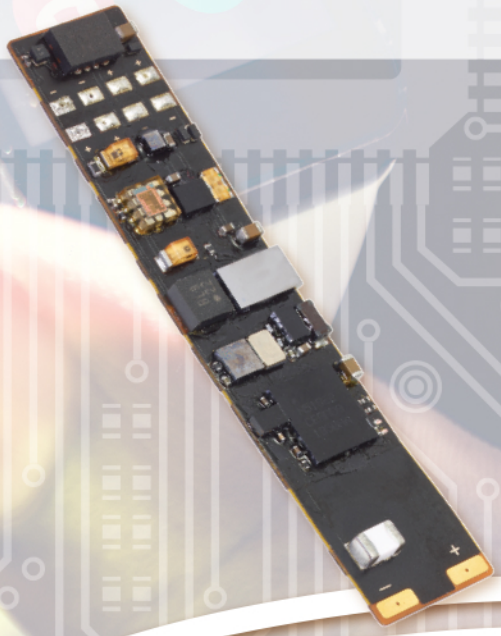
Industry leading Flex Design Support

Flexible Circuits, Rigid Flex, Membrane Switches, Plastic Moldings

Specialized EMS/Assembly Services

Product Module to Complete Product Box Builds

EXPLORE the POSSIBILITIES!



FLEXIBLE CIRCUIT® Technologies

Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

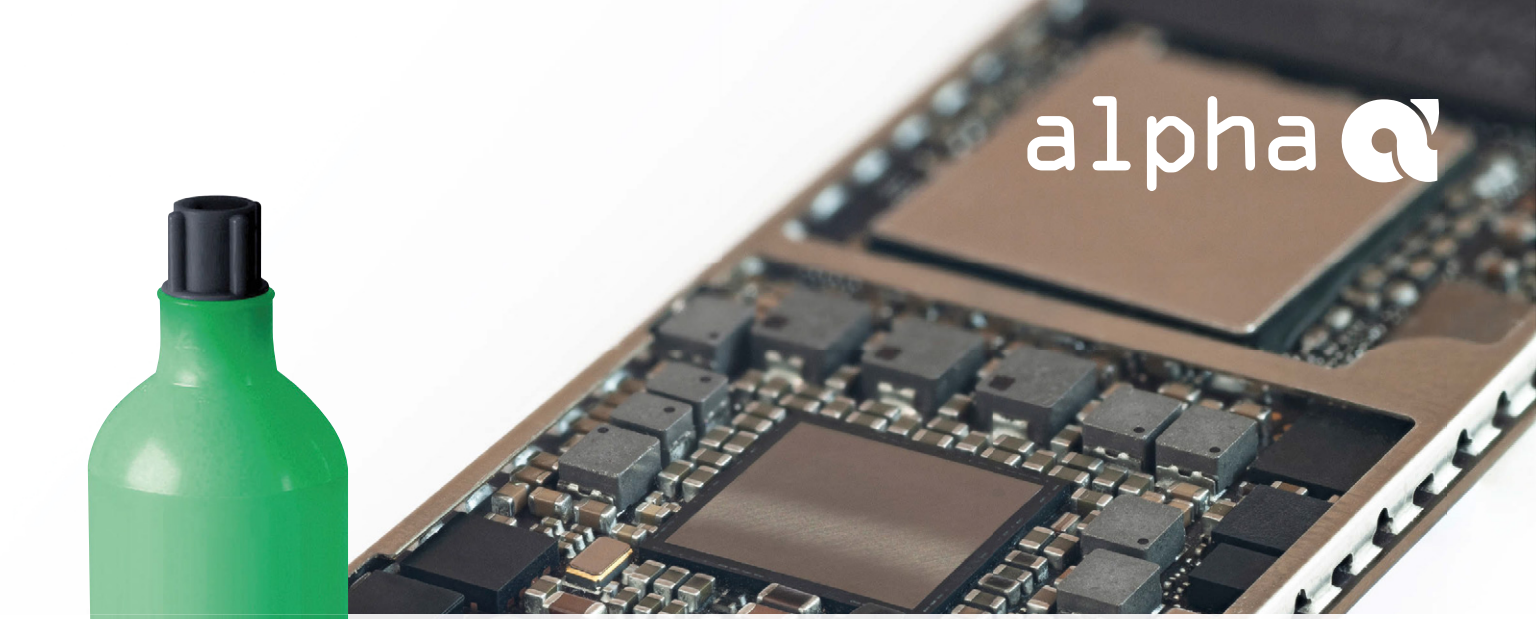
Flexible Circuits—A Catalyst for Technological Evolution

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Change has always been a hallmark of electronics technology. The driver of change has been that the industry is continuously being pressed to develop newer and better products with more functions and at lower cost. There is likely a tendency to think that change is the result of consumer demand, but as Steve Jobs observed many years ago, the consumer doesn't necessarily always know what they want until they see it and can sense or experience the value. The truth is more likely that the pressure for change comes as much, if not more, from the product developer than from the customer.

Change is often sparked as the result of some dissatisfaction with the status quo that the product designer/developer sees as an opportunity to improve the product. There is an almost biological aspect to the growth and evolution of electronic products. There is also, of course, a rational force driving the development and introduction of every new electronic product, but at a certain point, electronic products seem to take on a life of their own. Thus, in product development and growth (i.e., change), there is a Darwinian-like quality to the process. Electronic products that adapt quickest and most readily to the winds of change thrive; those that don't are pushed back to wither and die. Adaptation is obviously a key to survival and synergistic (or symbiotic, if you wish) linking of adaptive technologies



ALPHA® OM-372

High Electrochemical Reliability
Ultra-Fine Feature No-Clean Solder Paste

ALPHA OM-372 is designed for superior performance on assemblies with ultra-fine pitch components requiring excellent stencil transfer efficiency and high electrical reliability, such as those found in mobile and wearables, computing, and medical assemblies.

ALPHA OM-372 Key Features:

- Best-in-class electrochemical reliability on fine pitched low standoff packages.
- Ultra-fine feature printing and reflow capability down to 008004 components.
- Minimum post reflow residue provides high reliability performance for fine pitch, high density designs when flux is entrapped under devices
- Excellent HiP/NWO Performance on high I/O count packages.
- No-Clean, Zero-Halogen

MacDermid Alpha ELECTRONICS SOLUTIONS

www.macdermidalpha.com

offers obvious beneficial potential and a prospective path to securing such benefit. It is not difficult to assume that it is often preceded and influenced by some moment of inspiration that drives evolution.

With only a wee bit of prejudice, I would argue that flexible circuits are among the most adaptive and adaptable of all electronic interconnection technologies, and perhaps the most catalytic as well. Their adaptability has long been (and increasingly) seen and tapped by product developers, including everyone from assemblers to packagers to solving packaging and interconnection problems and challenges. Over the last quarter century, the range of applications for flexible circuits has grown at an impressive rate as the technology has been adapted to a host of new interconnection opportunities. The historical roles of flex circuits—wire harness replacement, 3D interconnection enabler, and dynamic interconnection scheme to connect parts of an electronic assembly designed to move relative to one another—will not go away. Instead, we will likely see an ever-expanding array of applications.

There are a lot of reasons for this but in recent years, there has been a push by consortia to help nurture the awareness and growth of flexible circuit technology. Perhaps the most notable has been NextFlex, a public/private partnership with a stated mission to advance U.S. manufacturing of what they refer to variously as "flexible hybrid electronics" (FHE) and printed electronics (PE). The terms, it appears, were created to fundamentally rebrand the time-honored subset of flexible circuits called polymer thick film technology, as near as can be told.

What is more important is that the NextFlex effort has shone a light on flexible circuit technology, writ large, as they seek to promote public/private partnerships which facilitate flexible circuit technology innovation and commercialization. An important, publicly-stated aspect of their effort has been to "accelerate manufacturing workforce development and promote a sustainable ecosystem for advanced manufacturing," which are vital to the continuing evolution, adaptation, and growth of the technology.

Inspiration, insight, or whatever else one wishes to call it, is vital in the execution of change. Over the last couple of decades, we have been watching the blurring of lines between what were formerly sharply divided and fundamental elements of every aspect of electronic manufacturing technology, especially in flexible circuit technology. The components, the substrates, and even assembly technologies are beginning to be used in a more coherent and cooperative way than ever before. One need only look at the recent advances in printed electronics to see the evidence of inspired evolution. The relative strengths of often very different technologies, all adaptive and adaptable, are enabling the electronics product industry to continue to get ever more value from our increasingly important and indispensable electronic devices. Ironically, and perhaps fittingly, the only thing that will never change is change. **FLEX007**

One need only look at the recent advances in printed electronics to see the evidence of inspired evolution.

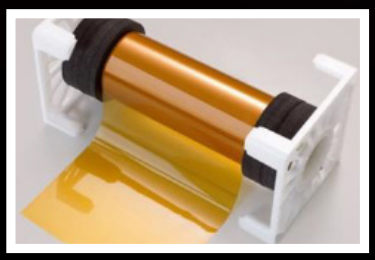
Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, click here. Download your free copy of Fjelstad's book *Flexible Circuit Technology, 4th Edition*, and watch his in-depth workshop series "Flexible Circuit Technology."

BENDING THE POSSIBILITIES

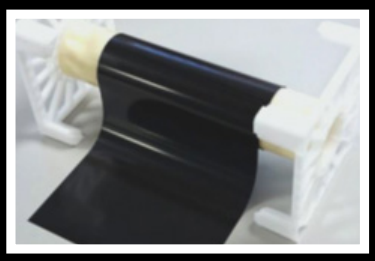
FLEXFINER

BY TAIYO

AVAILABE IN TWO COLORS



AMBER



BLACK

- Excellent bendability
- High Resolution by photolithography
- High Reliability
- Simplified process
- UL94 VTM-0 certified
- Low spring back force
- Resolution Line/Space=50/50μm
- Solder mask defined opening (30μm diameter)

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.

TAIYO AMERICA

UNMATCHED INNOVATION

Follow us on social media

TOP TEN

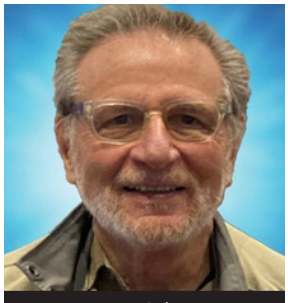
EDITOR'S PICKS

Patrick McGoff

How to Use Panelization Planning to Save Money and Resources

In this interview, Siemens' Patrick McGoff speaks with Nolan Johnson about strategies to utilize panelization planning to save money and resources.

Fein-Lines: PCEA Presentation—Latest PCB Fab Processes

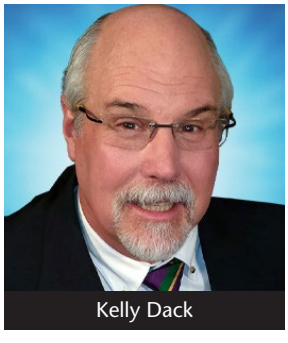


Dan Feinberg

As someone who started his career as a PCB fab process engineer in the early '60s I find it interesting that many of the processes for PCB interconnects are still in use today. Yes, we had eyelets then, but, hey, we did start using plated through-holes too.

I-Connect007 Launches *Real Time with... American Standard Circuits Event*

I-Connect007 is pleased to announce "*Real Time with... American Standard Circuits*," the first-of-its-kind event featuring three in-depth discussions and a virtual tour of American Standard Circuits' factory.



Kelly Dack

The Digital Layout: Educational Accolades for Our Chapter Liaison

In this month's column, I will give kudos to our PCEA Chapter Liaison who has been capturing the efforts of the PCEA to educate our members and our industry as a whole and then pass the mic to our PCEA Chairman, Steph Chavez, to provide some thoughts on the matter.

Altium's Nexar Platform Unites Design, Supply Chain, and Manufacturing, Part 1


Ted Pawela

I-Connect007 recently spoke with Ted Pawela, chief ecosystem officer and head of the Nexar business unit at Altium. Ted discussed Altium's new Nexar cloud platform which aims to build an ecosystem that connects designers, supply chain companies, and manufacturers.

Elementary, Mr. Watson: The Danger of Rogue Libraries


John Watson

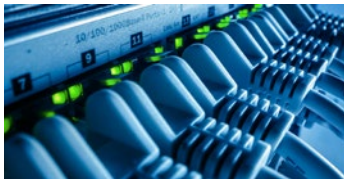
For PCB designers, the most common part of the library is the collection of components used in the PCB design process. But I have seen some libraries have other information, including a resource area, a group of documents, standards, and articles. So, basically, it can have anything you want.

From DesignCon: Rogers' Products at the Right Place and Time

Nolan Johnson recently spoke with Dean Marquart, product manager at Rogers Corporation, about his role in managing parts so that customers are getting what they need—and when they need it.


Dean Marquart

The Impact of Via Structures on Multi-Gigabit Signal Transmission



This article briefly introduces various via structures on the PCB for layer transition purposes. It also investigates the impact of these via structures on multi-gigabit transmission by analysing time domain reflectometry (TDR), differential insertion loss (Sdd21), and eye diagrams.

Collaboration: The Key to Streamlining Your Design Process

We recently spoke with Patrick Davis, product management director of Cadence Design Systems, about the benefits of collaboration between PCB designers and fabricators—and the drawbacks to throwing designs "over the wall."

The Power of Designer and Manufacturer Collaboration

Whether you are beginning board layout, building the first run of a prototype, or you're making a change to a mature product several years into its life span, there is no incorrect time to consult your PCB manufacturer. Circuit boards today are dense with features that push the limits of manufacturing technology, and as designers employ these technological advances it's easy to venture off the well-documented path of standard manufacturing.

For the latest news and information, visit **PCBDesign007.com**

Career Opportunities

Is your team growing?

Find industry-experienced candidates at I-Connect007.

For just $750, your 200-word, full-column ad will appear in the Career Opportunities section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, suppliers and the academic community.

In addition, your ad will:

- be featured in at least one of our newsletters
- appear on our jobConnect007.com board, which is promoted in every newsletter
- appear in our monthly Careers Guide, emailed to 26,000 potential candidates

Potential candidates can click on your ad and submit a resume directly to the email address you provide, or be directed to the URL of your choice.

No contract required. Just send over your copy and company logo and we'll do the rest!

Contact barb@iconnect007.com to get your ad posted today!

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

PERFECT POINT
PRECISION CARBIDE TOOLS

Field Application Engineer

Perfect Point Precision Carbide Tools, a leading supplier of circuit board cutting tools, is looking to add to our technical staff to enhance our customer support team at our Santa Ana repointing facility.

We offer competitive skills-based remuneration and additional performance-based compensation.

As field application engineer, you will be responsible for resolving technical issues and providing engineered solutions to our client base, as well as support sales of carbide cutting tools and repoint services.

Desired traits:

- A strategic thinker with strong management, analytical, and planning abilities
- Exceptional written and communication skills
- Self-motivated team player able to work collaboratively with our technical and sales teams

Send your resume to
shane@perfectpoint.us
for consideration.

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Customer Service Representative, UK

We are looking to expand our UK Customer Service/ Internal Sales team. As Customer Service Representative you will provide great sales and customer service support and respond to the needs of clients from industries including Aerospace, Defence, Automotive and Pharmaceutical. Duties include:

- Maintain & develop relationships with new and existing customers
- Make rapid, accurate cost calculations and provide quotations
- Accurately input customer orders through bespoke MRP System
- Liaise with colleagues at Chinese HQ and other Overseas Business Units to manage domestic and international requirements
- Assist sales team with reporting, sales analysis and other items at their request

Skills and abilities required for the role:

The ideal candidate is a proactive self-starter with a strong customer service background. Friendly, approachable, and confident, you should have a good phone mannerism and be computer literate.

- Previous experience in a Customer Service background, ideally management or supervisor role
- Experience with MRP Systems
- Good working knowledge of Microsoft Office Tools such as Outlook, Excel etc.

What's on Offer:

- Excellent salary & benefits commensurate with experience

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits.

Please forward your resume to HR@ventec-europe.com

apply now

Career Opportunities

Printed Circuits, a fast-growing printed circuit board fabricator, offers:

- Excellent opportunities for advancement and growth
- Dynamic manufacturing environment
- Excellent health, dental and other benefits
- Annual profit-sharing plan
- Signing bonus
- Additional incentives at the leadership level
- Clean facility with state-of-the-art manufacturing equipment
- Highly collaborative corporate and manufacturing culture that values employee contributions

Laminator Technician

Nature of Duties/Responsibilities

- Layup cover lay
- Layup rigid flex
- Layup multilayer/CU core boards
- Oxide treat/cobra treatment of all layers/CU cores
- Shear flex layer edges
- Rout of machine panel edges and buff
- Remove oxide/cobra treatment (strip panels)
- Serialize panels
- Pre-tac Kapton windows on flex layers (bikini process)
- Layup Kapton bonds
- Prep materials: B-stage, Kapton, release sheet
- Breakdown: flex layers, and caps
- Power scrub: boards, layers, and caps
- Laminate insulators, stiffeners, and heatsinks
- Plasma cleans and dry flex layers B-stage (Dry)
- Booking layers and materials, ready for lamination process
- Other duties as deemed necessary by supervisor

Education/Experience

- High school diploma or GED
- Must be a team player
- Must demonstrate the ability to read and write English and complete simple mathematical equations
- Must be able to follow strict policy and OSHA guidelines
- Must be able to lift 50 lbs
- Must have attention to detail

Wet Process/Plating Technician

Position is 3rd shift (11:00PM to 7:30AM, Sunday through Friday)

Purpose

To carry out departmental activities which result in producing quality product that conforms to customer requirements. To operate and maintain a safe working environment.

Nature of Duties/Responsibilities

- Load and unload electroplating equipment
- Fasten circuit boards to racks and cathode bars
- Immerse work pieces in series of cleaning, plating and rinsing tanks, following timed cycles manually or using hoists
- Carry work pieces between departments through electroplating processes
- Set temperature and maintains proper liquid levels in the plating tanks
- Remove work pieces from racks, and examine work pieces for plating defects, such as nodules, thin plating or burned plating
- Place work pieces on racks to be moved to next operation
- Check completed boards
- Drain solutions from and clean and refill tanks; fill anode baskets as needed
- Remove buildup of plating metal from racks using chemical bath

Education and Experience

- High school diploma or GED required
- Good organizational skills and the ability to follow instructions
- Ability to maintain a regular and reliable attendance record
- Must be able to work independently and learn quickly
- Organized, self-motivated, and action-oriented, with the ability to adapt quickly to new challenges/opportunities
- Prior plating experience a plus

Production Scheduler

Main Responsibilities

- Development and deployment of a level-loaded production plan
- Establish manufacturing plan which results in "best possible" use of resources to maximize asset utilization
- Analyze production capacity of manufacturing processes, equipment and human resource requirements needed to produce required products
- Plan operation manufacturing sequences in weekly time segments utilizing production labor standards
- Maintain, align, and communicate regularly with internal suppliers/customers and customer service on key order metrics as per their requirements
- Frequently compare current and anticipated orders with available inventory and creates replenishment plan
- Maintain master distribution schedule for the assigned facility, revise as needed and alert appropriate staff of schedule changes or delays
- Participate in periodic forecasting meetings
- Lead or participate in planning and status meetings with production, shipping, purchasing, customer service and/or other related departments
- Follow all good manufacturing practices (GMPs)
- Answer company communications, fax, copy and file paperwork

Education and Experience

- High school diploma or GED
- Experience in manufacturing preferred/3 years in scheduling
- Resourceful and good problem-solving skills
- Ability to make high pressure decisions
- Excellent written and verbal communication skills
- Strong computer skills including ERP, Excel, Word, MS Office
- Detailed and meticulous with good organizational skills
- Must be articulate, tactful and professional at all times
- Self-motivated

apply now

Career Opportunities

Fuji America Corporation is a rapidly growing electronics assembly equipment distributor. We support the factories of the future and smart factories globally. We offer an exciting and challenging career for a software support engineer and an applications engineer who want to join our growing company.

Software Support Engineer

As a software support engineer for Fuji America Corporation, you will be a customer-facing technical advisor with the opportunity to solve technically complex problems for our proprietary software. As a trusted advisor to our customers, you will have influence over a broad range of solutions that create business value. As a valued member on our team, the software support engineer will use advanced troubleshooting methods and tools to solve technically complex problems. These highly complex, escalated problems require broad and in-depth product knowledge, as well as exceptional troubleshooting skills.

- Field installation of proprietary software/automation equipment throughout North America
- Field troubleshoot, repair, training, and process support of proprietary software
- Provide remote and on-site technical support
- Troubleshoot Windows 10/Windows server installing, configuration, and support
- Networking experience—setting up and supporting networks.
- Exposure and/or experience with Oracle or Microsoft SQL server databases
- Strong verbal communication skills with both customer and other technical depts.
- Flexibility to travel and perform job assignments on short notice
- Strong aptitude with current computing applications and networking processes

Experience

- Bachelor of Science in computer science or related field preferred

Applications Engineer

As an applications engineer, you will be responsible for doing cycle time and studies in preparation to make recommendations of Fuji products for customers' applications. Support implementation of activities within the technical center such as customer visits, demonstrations, evaluations, testing, inspection of Fuji products, including peripheral equipment from other vendors.

- Assist sales representatives in technical aspects relating to machine and software functions and utilization.
- Assist sales representatives and customers with providing CTA (Cycle Time Analysis) to them for recommending Fuji products to customers' specific applications. This includes the sFAB machine as well as all other SMT machines.
- Schedule and perform product demonstrations on all available types of equipment and software to potential and existing customers.
- Test and evaluate existing as well as new technologies on equipment and software performance and reliability.
- Assist in the coordination of any new FAC projects by utilizing your full potential.
- Responsible for the setup of the equipment and its demonstration for various trade shows.
- Assist FAC staff in any technical issues which may require attention.
- Assist in the coordination of design and manufacture of customs tooling for placement equipment.
- Perform inventory checks every six months according to the schedule and manner regulated by the company, if applicable.

Experience

- Minimum five years programming/computer experience
- Bachelor's degree preferred

apply now

Career Opportunities

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB shop, is looking for sales representatives for all territories.

Reasons you should work with Prototron:

- Serving the PCB industry for over 30 years
- Solid reputation for on-time delivery (99% on-time)
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- AS9100
- MIL-PRF- 31032
- ITAR
- Global sourcing
- Engineering consultation
- Completely customer focused team

Interested? Let's have a talk.
Call Dan Beaulieu at
207-649-0879
or email to
danbbeaulieu@aol.com

apply now

ICAPE GROUP

PCB Field Engineer– North America Operations

ICAPE Group is a European leader for printed circuits boards and custom-made electro-mechanical parts. Headquartered in Paris, France, we have over 500 employees located in more than 70 countries serving our +2500 customers.

To support our growth in the American market, we are looking for a PCB Field Engineer.

You will work in our North America technical center, including our U.S. technical laboratory, and will be responsible for providing technical and quality support to our American sales team.

You will have direct customer contact during all phases of the sales process and provide follow-on support as required.

RESPONSIBILITIES INCLUDE

- Feasibility recommendations
- Fabricator questions and liaison
- Quality resolutions
- Technical explanation (for the customer) of proposals, laboratory analysis or technology challenges

REQUIREMENTS

- Engineering degree or equivalent industry experience
- 5 years' experience with PCB manufacturing (including CAM)
- Excellent technical understanding of PCBs
- Experience with quality tools (FAI, PPAP and 8-D)
- Good communication skills (written and oral)

Communication skills are essential to assist the customer with navigation of the complex process of matching the PCB to the application.

SALARY

Competitive, based on profile and experience. Position is full time in Indianapolis, Ind.

apply now

Career Opportunities

Application Engineer (m/f/d)—Fulltime, Germany

Our company is expanding its product portfolio into custom made products. This creates the need for an Application Engineer to provide technical support to our existing sales team and customer base.

Responsibilities:

- Analysis of incoming technical data and handling of engineering questions
- Technical consultation of customers (incl. new customer specifications and discuss with relevant technical and quality teams worldwide)
- Support and consultation for new projects
- Lead and/or participate in local, cross-location/global cross-department projects of various scale
- Develop and provide function-related trainings to existing and new staff in order to transfer and optimize know-how
- Provide technical solutions

Skills:

- Technical expertise in battery power solutions and technologies for Rechargeable and Primary cells and Battery Packs
- Mechanical background or knowledge to be able to discuss and manage other products, like custom made connectors, cable assemblies and keypad touch panels.
- Written and spoken English and German, any other European language a plus.
- Highly technical with a commercial flare.
- Self-motivated, ambitious, and eager to grow in a dynamic organization.

Interested? We are looking forward to your application!

Please send your application to **hr@cmit.support**. For any inquiries, please contact Mrs. Amélie Filler. For more information visit www.cml-globalsolutions.com

apply now

Sales Manager (m/f/d)—Worldwide Locations

CML Group is a leading provider of Printed Circuit Boards. We develop tailor-made sourcing concepts for our customers worldwide creating strong partnerships and reliable connections.

For the expansion of our target markets, we need you to generate new business, drive new projects from RFQ stage and manage the customer relationship.

Your Profile:

- Profound sales and technical expertise in printed circuit board industry
- Local market knowledge and ideally a customer base of contacts in one or more of the listed countries
- Have successful track records in developing new business opportunities
- Excellent command in spoken and written English and one additional local language
- Highly self-motivated, ambitious, eager to grow in a dynamic organization
- Able to work independently and have good communication skills and leadership skills
- Self-employed/contractor/commission-based agent also welcome

Your Target Markets:

- Europe: Spain, France, Germany, Netherlands, UK, Denmark, Sweden, Norway
- USA: New Jersey, Florida, Georgia, Michigan, San Jose, Bay area, Pacific Northwest and Canada
- Others: Singapore, Thailand, Malaysia, Australia, Brazil, Turkey, Russia, and South Africa

Interested? We are looking forward to your application!

Please send your application to **hr-china@cml-eurasia.hk**. For any inquiries, please contact Ms. Grace Feng. For more information visit www.cml-globalsolutions.com

apply now

Career Opportunities

The Test Connection, Inc.

Rewarding Careers

Take advantage of the opportunities we are offering for careers with a growing test engineering firm. We currently have several openings at every stage of our operation.

The Test Connection, Inc. is a test engineering firm. We are family owned and operated with solid growth goals and strategies. We have an established workforce with seasoned professionals who are committed to meeting the demands of high-quality, low-cost and fast delivery.

TTCI is an Equal Opportunity Employer. We offer careers that include skills-based compensation. We are always looking for talented, experienced test engineers, test technicians, quote technicians, electronics interns, and front office staff to further our customer-oriented mission.

Associate Electronics Technician/ Engineer (ATE-MD)

TTCI is adding electronics technician/engineer to our team for production test support.

- Candidates would operate the test systems and inspect circuit card assemblies (CCA) and will work under the direction of engineering staff, following established procedures to accomplish assigned tasks.
- Test, troubleshoot, repair, and modify developmental and production electronics.
- Working knowledge of theories of electronics, electrical circuitry, engineering mathematics, electronic and electrical testing desired.
- Advancement opportunities available.
- Must be a US citizen or resident.

apply now

Test Engineer (TE-MD)

In this role, you will specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly HP) and/or Teradyne (formerly GenRad) TestStation/228X test systems.

- Candidates must have at least three years of experience with in-circuit test equipment. A candidate would develop and debug our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Candidates would also help support production testing and implement Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks.
- Some travel required and these positions are available in the Hunt Valley, Md., office.

apply now

Sr. Test Engineer (STE-MD)

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/ GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

We proudly serve customers nationwide and around the world.

TTCI is an ITAR registered and JCP DD2345 certified company that is NIST 800-171 compliant.

apply now

Career Opportunities

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Maintenance Technician

Inspects work-related conditions to determine compliance with prescribed operating and safety standards. Operates power-driven machinery and uses equipment and tools commonly used to maintain facilities and equipment. Replace filters, belts, and additional parts for repairs and preventive maintenance. Moves objects weighing up to 150 lbs. using a hand truck or pulley. Cleans work area and equipment. Works with cleaning fluids, agents, chemicals, and paints using protective gear. Works at elevations greater than ten feet, climbing ladders, while repairing or maintaining building structures and equipment. Assists skilled maintenance technicians/workers in more complex tasks and possible after-hours emergency repairs. Must meet scheduling and attendance requirements.

apply now

Plating Operator

Plating operator for printed circuit boards. No experience necessary, will train. Must be able to work with chemicals, lift up to 50 pounds, and have good math skills. Minimum high school/GED or equivalent. All shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for steady overtime pay.

apply now

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Water Treatment Operator

Responsible for operating waste treatment plant, our operation that converts wastewater in drains and sewers into a form that's metal free to release into the environment.

Control equipment and monitor processes that remove metals from wastewater. Run tests to make sure that the processes are working correctly. Keep records of water quality and pH. Operate and maintain the pumps and motors that move water and wastewater through filtration systems. Read meters and gauges to make sure plant equipment is working properly. Take samples and run tests to determine the quality of the water being produced. Adjust the amount of chemicals being added to the water and keep records that document compliance.

apply now

Drilling Operator

Drilling operator for printed circuit boards. Minimum 2 years of experience. Minimum high school/GED or equivalent.

All Shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for overtime pay.

apply now

Career Opportunities

MivaTek Global

Product Manager

MivaTek Global is preparing for a major market and product offering expansion. Miva's new NG3 and DART technologies have been released to expand the capabilities of Miva's industry-leading LED DMD direct write systems in PCB and Microelectronics. MivaTek Global is looking for a technology leader that can be involved guiding this major development.

The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptions to Miva products
- **Design:** Be the voice of the customer for new product development
- **Quality:** Verify and standardize product performance testing and implementation
- **Training:** Conduct virtual and on-site training
- **Travel:** Product testing at customer and factory locations

Use your 8 plus years of experience in either the PCB or Microelectronic industry to make a difference with the leader in LED DMD direct imaging technology. Direct imaging, CAM, AOI, or drilling experience is a plus but not required.

For consideration, send your resume to N.Hogan@MivaTek.Global. For more information on the company see www.MivaTek.Global or www.Mivatec.com.

apply now

MivaTek Global

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now

Career Opportunities

ARLON ELECTRONIC MATERIALS

Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- **Engineering**
- **Quality**
- **Various Manufacturing**

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

Arlon is a major manufacturer of specialty high-performance laminate and prepreg materials for use in a wide variety of printed circuit board applications. Arlon specializes in thermoset resin technology, including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems. These resin systems are available on a variety of substrates, including woven glass and non-woven aramid. Typical applications for these materials include advanced commercial and military electronics such as avionics, semiconductor testing, heat sink bonding, High Density Interconnect (HDI) and microvia PCBs (i.e. in mobile communication products).

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information please visit our website at www.arlonemd.com

apply now

KOH YOUNG TECHNOLOGY

Logistics Assistant

Koh Young America is looking for a Logistics Assistant to assist and oversee our supply chain operations. Working alongside a Logistics Specialist, you will coordinate processes to ensure smooth operations using a variety of channels to maximize efficiency. You must be an excellent communicator and negotiator well-versed in supply chain management principles and practices. Also, you should be meticulous with a focus on customer satisfaction. These attributes are ideally complemented by a Bachelor's in Supply Chain Management or equivalent professional experience in the manufacturing industry.

This position is in our Duluth, Georgia, headquarters, where we serve our customers within North and South America. We offer health, dental, vision, and life Insurance with no employee premiums, including dependent coverage. Additionally, we provide a 401K retirement plan with company matching, plus a generous PTO policy with paid holidays.

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement and inspection technology used in the production of microelectronics assemblies. Using patented 3D technology, Koh Young provides best-in-class products in Solder Paste Inspection (SPI) and Automated Optical Inspection (AOI) for electronics manufacturers worldwide.

apply now

Career Opportunities

Manncorp™

SMT Operator
Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for a **surface-mount technology (SMT) operator** to join their growing team in Hatboro, PA!

The **SMT operator** will be part of a collaborative team and operate the latest Manncorp equipment in our brand-new demonstration center.

Duties and Responsibilities:
- Set up and operate automated SMT assembly equipment
- Prepare component kits for manufacturing
- Perform visual inspection of SMT assembly
- Participate in directing the expansion and further development of our SMT capabilities
- Some mechanical assembly of lighting fixtures
- Assist Manncorp sales with customer demos

Requirements and Qualifications:
- Prior experience with SMT equipment or equivalent technical degree preferred; will consider recent graduates or those new to the industry
- Windows computer knowledge required
- Strong mechanical and electrical troubleshooting skills
- Experience programming machinery or demonstrated willingness to learn
- Positive self-starter attitude with a good work ethic
- Ability to work with minimal supervision
- Ability to lift up to 50 lbs. repetitively

We Offer:
- Competitive pay
- Medical and dental insurance
- Retirement fund matching
- Continued training as the industry develops

apply now

Manncorp™

SMT Field Technician
Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for an additional SMT Field Technician to join our existing East Coast team and install and support our wide array of SMT equipment.

Duties and Responsibilities:
- Manage on-site equipment installation and customer training
- Provide post-installation service and support, including troubleshooting and diagnosing technical problems by phone, email, or on-site visit
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Participate in the ongoing development and improvement of both our machines and the customer experience we offer

Requirements and Qualifications:
- Prior experience with SMT equipment, or equivalent technical degree
- Proven strong mechanical and electrical troubleshooting skills
- Proficiency in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Travel and overnight stays
- Ability to arrange and schedule service trips

We Offer:
- Health and dental insurance
- Retirement fund matching
- Continuing training as the industry develops

apply now

Career Opportunities

SIEMENS

Siemens EDA
Sr. Applications Engineer

Support consultative sales efforts at world's leading semiconductor and electronic equipment manufacturers. You will be responsible for securing EM Analysis & Simulation technical wins with the industry-leading HyperLynx Analysis product family as part of the Xpedition Enterprise design flow.

Will deliver technical presentations, conduct product demonstrations and benchmarks, and participate in the development of account sales strategies leading to market share gains.

- PCB design competency required
- BEE, MSEE preferred
- Prior experience with Signal Integrity, Power Integrity, EM & SPICE circuit analysis tools
- Experience with HyperLynx, Ansys, Keysight and/or Sigrity
- A minimum of 5 years' hands-on experience with EM Analysis & Simulation, printed circuit board design, engineering technology or similar field
- Moderate domestic travel required
- Possess passion to learn and perform at the cutting edge of technology
- Desire to broaden exposure to the business aspects of the technical design world
- Possess a demonstrated ability to build strong rapport and credibility with customer organizations while maintaining an internal network of contacts
- Enjoy contributing to the success of a phenomenal team

***Qualified applicants will not require employer-sponsored work authorization now or in the future for employment in the United States. Qualified Applicants must be legally authorized for employment in the United States.*

apply now

U.S. CIRCUIT

Plating Supervisor

Escondido, California-based PCB fabricator U.S. Circuit is now hiring for the position of plating supervisor. Candidate must have a minimum of five years' experience working in a wet process environment. Must have good communication skills, bilingual is a plus. Must have working knowledge of a plating lab and hands-on experience running an electrolytic plating line. Responsibilities include, but are not limited to, scheduling work, enforcing safety rules, scheduling/maintaining equipment and maintenance of records.

Competitive benefits package. Pay will be commensurate with experience.

Mail to:
mfariba@uscircuit.com

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor

Longmont, CO; Phoenix, AZ; U.S.-based remote

Independent contractor, possible full-time employment

Job Description

This position is responsible for delivering effective electronics manufacturing training, including IPC Certification, to students from the electronics manufacturing industry. IPC instructors primarily train and certify operators, inspectors, engineers, and other trainers to one of six IPC Certification Programs: IPC-A-600, IPC-A-610, IPC/WHMA-A-620, IPC J-STD-001, IPC 7711/7721, and IPC-6012.

IPC instructors will conduct training at one of our public training centers or will travel directly to the customer's facility. A candidate's close proximity to Longmont, CO, or Phoenix, AZ, is a plus. Several IPC Certification Courses can be taught remotely and require no travel.

Qualifications

Candidates must have a minimum of five years of electronics manufacturing experience. This experience can include printed circuit board fabrication, circuit board assembly, and/or wire and cable harness assembly. Soldering experience of through-hole and/or surface-mount components is highly preferred.

Candidate must have IPC training experience, either currently or in the past. A current and valid certified IPC trainer certificate holder is highly preferred.

Applicants must have the ability to work with little to no supervision and make appropriate and professional decisions.

Send resumes to Sharon Montana-Beard at sharonm@blackfox.com.

apply now

American Standard Circuits
Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

The CAD/CAM engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creating manufacturing data, programs, and tools required for the manufacture of PCB.

Essential Duties and Responsibilities

- Import customer data into various CAM systems.
- Perform design rule checks and edit data to comply with manufacturing guidelines.
- Create array configurations, route, and test programs, penalization and output data for production use.
- Work with process engineers to evaluate and provide strategy for advanced processing as needed.
- Itemize and correspond to design issues with customers.
- Other duties as assigned.

Organizational Relationship

Reports to the engineering manager. Coordinates activities with all departments, especially manufacturing.

Qualifications

- A college degree or 5 years' experience is required. Good communication skills and the ability to work well with people is essential.
- Printed circuit board manufacturing knowledge.
- Experience using CAM tooling software, Orbotech GenFlex®.

Physical Demands

Ability to communicate verbally with management and coworkers is crucial. Regular use of the telephone and e-mail for communication is essential. Sitting for extended periods is common. Hearing and vision within normal ranges is helpful for normal conversations, to receive ordinary information and to prepare documents.

apply now

Career Opportunities

Now Hiring

Director of Process Engineering

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a director of process engineering.

Job Summary:

The director of process engineering leads all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering processes within the plant.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Provides guidance to process engineers in the development of process control plans and the application of advanced quality tools.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating them into the manufacturing operations.
- Strong communication skills to establish priorities, work schedules, allocate resources, complete required information to customers, support quality system, enforce company policies and procedures, and utilize resources to provide the greatest efficiency to meet production objectives.

Education and Experience:

- Master's degree in chemical engineering or engineering is preferred.
- 10+ years process engineering experience in an electronics manufacturing environment, including 5 years in the PCB or similar manufacturing environment.
- 7+ years of process engineering management experience, including 5 years of experience with direct responsibility for meeting production throughput and quality goals.

apply now

Now Hiring

Process Engineering Manager

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a process engineering manager.

Job Summary:

The process engineering manager coordinates all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering team and leading this team to meet product requirements in support of the production plan.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating into the manufacturing operations

Education and Experience:

- Bachelor's degree in chemical engineering or engineering is preferred.
- 7+ years process engineering experience in an electronics manufacturing environment, including 3 years in the PCB or similar manufacturing environment.
- 5+ years of process engineering management experience, including 3 years of experience with direct responsibility for meeting production throughput and quality goals.

apply now

Career Opportunities

INSULECTRO

Are You Our Next Superstar?!

Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers build better boards faster. Our nationwide network provides many opportunities for a rewarding career within our company.

We are looking for talent with solid background in the PCB or PE industry and proven sales experience with a drive and attitude that match our company culture. This is a great opportunity to join an industry leader in the PCB and PE world and work with a terrific team driven to be vital in the design and manufacture of future circuits.

View our opportunities at Insulectro Careers (jobvite.com)

apply now

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

Opportunities are available in Canada, New England, California, and Chicago. If you love teaching people, choosing the classes and times you want to work, and basically being your own boss, this may be the career for you. EPTAC Corporation is the leading provider of electronics training and IPC certification and we are looking for instructors that have a passion for working with people to develop their skills and knowledge. If you have a background in electronics manufacturing and enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Ability to travel required. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and skills

- A love of teaching and enthusiasm to help others learn
- Background in electronics manufacturing
- Soldering and/or electronics/cable assembly experience
- IPC certification a plus, but will certify the right candidate

Benefits

- Ability to operate from home. No required in-office schedule
- Flexible schedule. Control your own schedule
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now

Career Opportunities

APCT

Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

APCT, a leading manufacturer of printed circuit boards, has experienced rapid growth over the past year and has multiple opportunities for highly skilled individuals looking to join a progressive and growing company. APCT is always eager to speak with professionals who understand the value of hard work, quality craftsmanship, and being part of a culture that not only serves the customer but one another.

APCT currently has opportunities in Santa Clara, CA; Orange County, CA; Anaheim, CA; Wallingford, CT; and Austin, TX. Positions available range from manufacturing to quality control, sales, and finance.

We invite you to read about APCT at APCT.com and encourage you to understand our core values of passion, commitment, and trust. If you can embrace these principles and what they entail, then you may be a great match to join our team! Peruse the opportunities by clicking the link below.

Thank you, and we look forward to hearing from you soon.

apply now

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Pre-CAM Engineer

Illinois-based PCB fabricator Eagle Electronics is seeking a pre-CAM engineer specific to the printed circuit board manufacturing industry. The pre-CAM Engineer will facilitate creation of the job shop travelers used in the manufacturing process. Candidate will have a minimum of two years of pre-CAM experience and have a minimum education level of an associate degree. This is a first-shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to HR@eagle-elec.com indicating 'Pre-CAM Engineer' in the subject line.

apply now

Process Engineer

We are also seeking a process engineer with experience specific to the printed circuit board manufacturing industry. The process engineer will be assigned to specific processes within the manufacturing plant and be given ownership of those processes. The expectation is to make improvements, track and quantify process data, and add new capabilities where applicable. The right candidate will have a minimum of two years of process engineering experience, and a minimum education of bachelor's degree in an engineering field (chemical engineering preferred but not required). This is a first shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to HR@eagle-elec.com indicating 'Process Engineer' in the subject line.

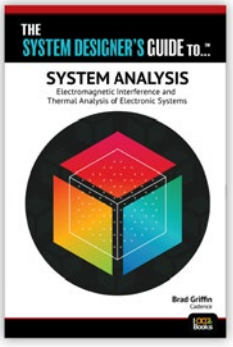
apply now

Introducing:

The System Designer's Guide to... System Analysis

Electromagnetic Interference and Thermal Analysis of Electronic Systems

In this latest title from I-007eBooks, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions. Get your copy now!

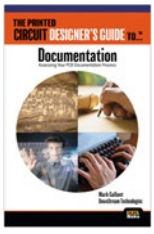


I-007eBooks The Printed Circuit Designer's Guide to...

Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

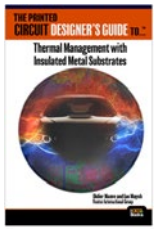
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



Documentation

by Mark Gallant, Downstream Technologies

When the PCB layout is finished, the designer is still not quite done. The designer's intent must still be communicated to the fabricator through accurate PCB documentation.



Thermal Management with Insulated Metal Substrates

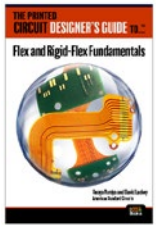
by Didier Mauve and Ian Mayoh, Ventec International Group

Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.

Fundamentals of RF/Microwave PCBs

by John Bushie and Anaya Vardya, American Standard Circuits

Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs. This micro eBook provides information needed to understand the unique challenges of RF PCBs.



Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

Flexible circuits are rapidly becoming a preferred interconnection technology for electronic products. By their intrinsic nature, FPCBs require a good deal more understanding and planning than their rigid PCB counterparts to be assured of first-pass success.

Our library is open 24/7/365. Visit us at: I-007eBooks.com

PUBLISHER: **BARRY MATTIES**
barry@iconnect007.com

MANAGING EDITOR: **ANDY SHAUGHNESSY**
(404) 806-0508; andy@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@iconnect007.com

EDITOR | COLUMNIST COORDINATOR: **MICHELLE TE**
michelle@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **DAN FEINBERG**
baer@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

PRODUCTION MANAGER: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **SHELLY STEIN, MIKE RADOGNA, TOBEY MARSICOVETERE**

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © JIW INGKA**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

October 2021, Volume 10, Number 10
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Problems solved!

SUBSCRIBE NOW