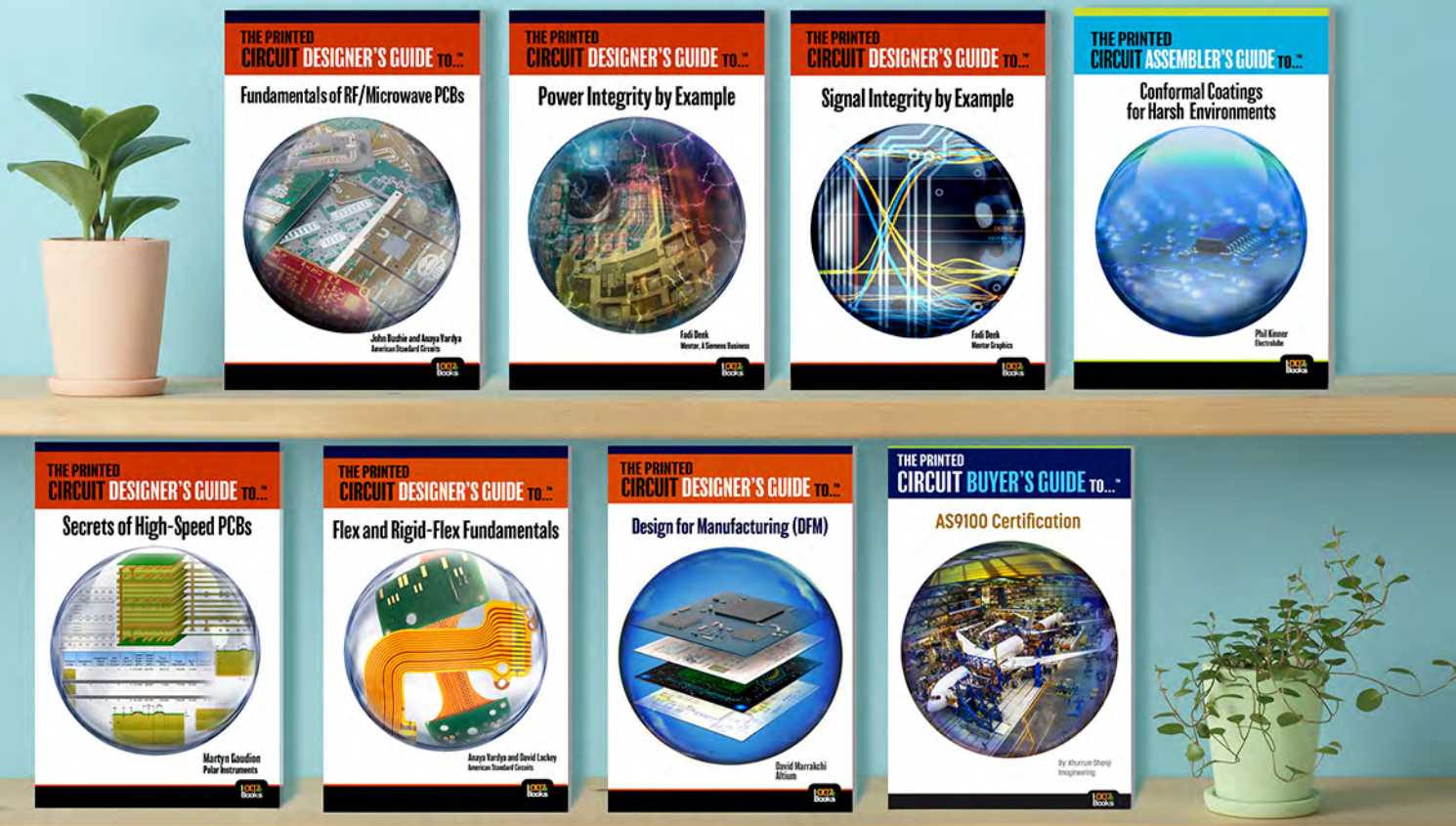
FEBRUARY 2018

I-Connect007

DESIGN007 MAGAZINE

Knowing Your Customers

Download free micro eBooks from our library.



Written by industry experts, each book contains valuable information on relevant topics in an easy-to-read format.

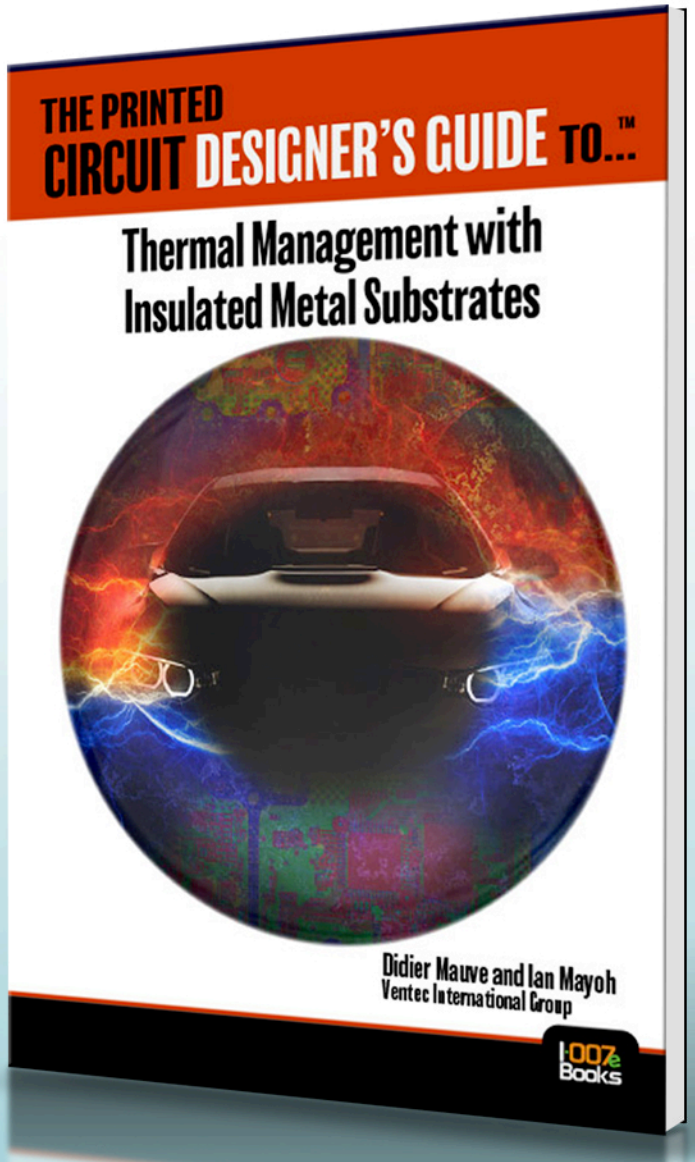
Log in or register to access all of our premium content at: myiconnect007.com

my I·Connect007
GOOD FOR THE INDUSTRY

Announcing Our Latest eBook

“A comprehensive tour through the world of thermal management. Required reading for designers and end users wishing to understand the science and practice of using insulated metal substrate materials.”

Alun Morgan
Chairman, EIPC



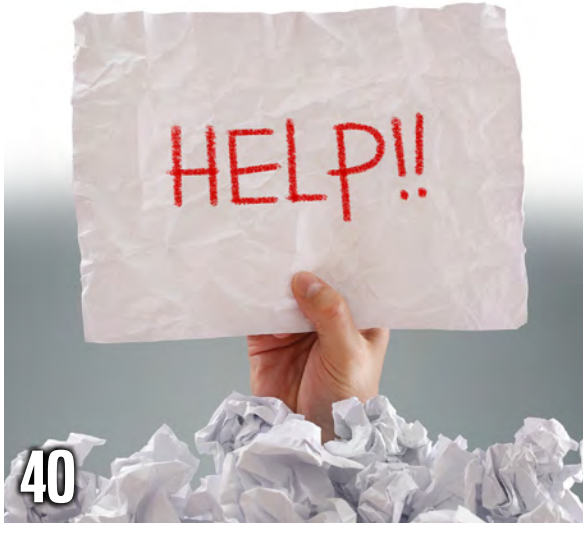
Download Now

Written by Didier Mauve and Ian Mayoh of Ventec International Group, this book highlights the need to dissipate heat from electronic devices.

I-007e Books

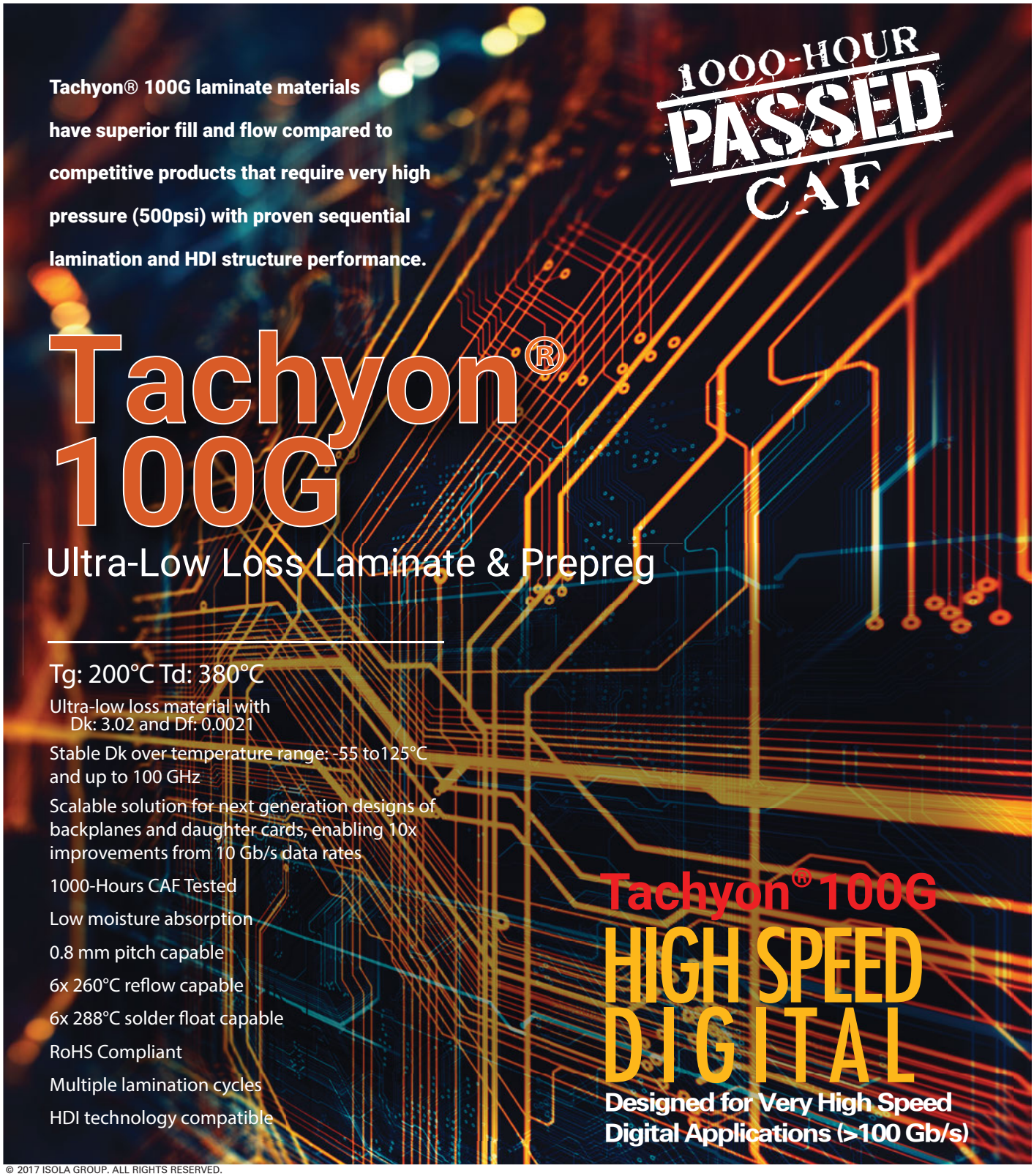
I-007ebooks.com

DESIGN007 MAGAZINE



Knowing Your Customers

How do you feel about your suppliers' customer service? How do your customers feel about your service? Every company in our industry claims to put the customer first, but what does that mean in a firm's day-to-day operation?



AEROSPACE & DEFENSE • COMPUTING • STORAGE • NETWORKS • COMMUNICATIONS

isola

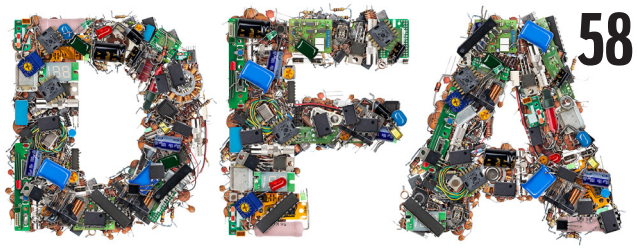
FOR MORE INFORMATION, CONTACT YOU YOUR LOCAL SALES REPRESENTATIVE.

Isola Group
3100 West Ray Road
Suite 301
Chandler, AZ 85226
Phone: 480-893-6527
info@isola-group.com

Isola Asia Pacific (Hong Kong) Ltd.
Unit 3512 - 3522, 35/F
No. 1 Hung To Road, Kwun Tong,
Kowloon, Hong Kong
Phone: 852-2418-1318
info.hkg@isola-group.com

Isola GmbH
Isola Strasse 2
D-52348 Düren, Germany
Phone: 49-2421-8080
info-dur@isola-group.com
Website: isola-group.com

DESIGN007 MAGAZINE



Your signal needs to get around the world fast... Why go halfway around the world to get your high-speed PCBs?

ACE: your high-speed experts right here at home, specializing in microwave and hybrid PCBs.

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

RoHS & WEEE compliant • Certified AS9100, ISO, ITAR, MIL-PRF-31032 & 55110

Customer Service Still a Big Issue

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

The customer comes first. We know the customer comes first because companies in this industry say so in the "About the company" section of their company websites. A visit to a handful of company sites reveals customer-oriented copy like this:

"We're here for our customers."

"Your success is our success."

"Customers are not an interruption of our work day; they are the reason for our existence."

"The customer is always right!"

"We're customer-centric."

Of course, everyone wants to satisfy their customers. But think for a minute: When was the last time you heard a colleague saying nice things about a supplier's customer service? When did you last experience great customer service from one of your suppliers?

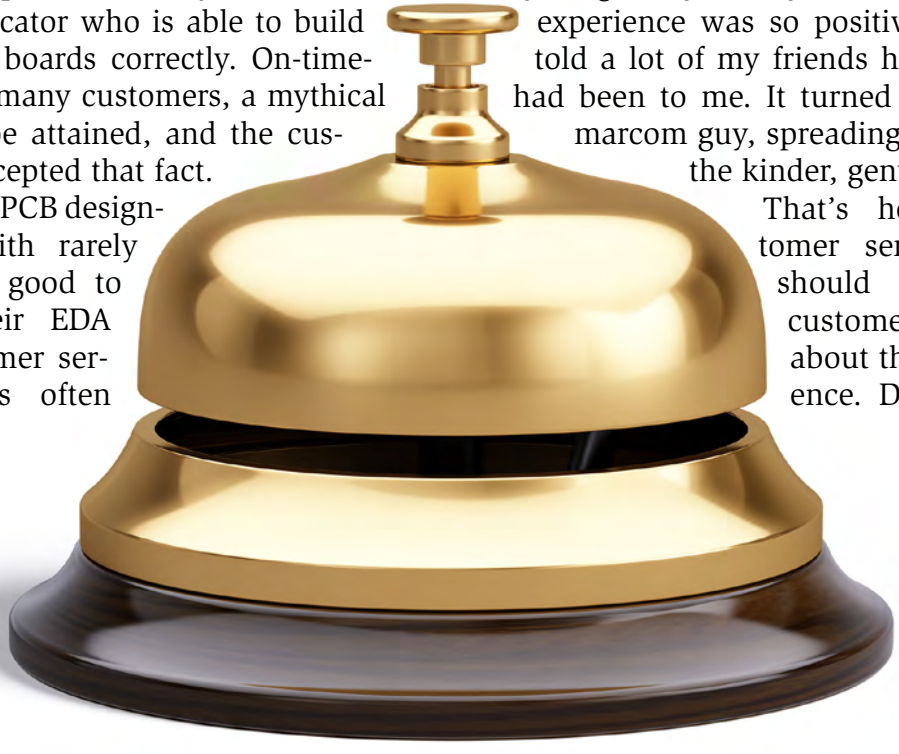
At trade shows and conferences, I hear stories about terrible customer service. Fabricators claim to offer on-time-delivery, but many OEMs just accept late delivery as the cost of using a fabricator who is able to build their high-tech boards correctly. On-time-delivery is, for many customers, a mythical goal never to be attained, and the customers have accepted that fact.

Similarly, the PCB designers I meet with rarely have anything good to say about their EDA vendor's customer service. Designers often say something along the lines of, "I really like this tool, but their customer service is awful. I'm still waiting for a call back, and it's a week later. But I love that router."

Even retail, which typically boasts higher customer service satisfaction rates than B2B, has its share of issues. For instance, take Comcast. I love Comcast's super-fast Internet, but if I have a technical problem, it's always a soul-destroying experience. I've moved three times since getting Comcast, and each time, the tech guy forgot about my set-up appointment, even after I called and said, "Don't forget this time!" Comcast knows they have terrible customer service, and they don't care.

The IRS, on the other hand, offered me excellent customer service. I owed them some money years ago, after the divorce, and I kept putting off the problem. I wound up working out a plan with some IRS folks who couldn't have been nicer. They'd say, "Don't give us your grocery money. You have to eat!" The experience was so positive for me that I told a lot of my friends how nice the IRS had been to me. It turned me into an IRS marcom guy, spreading the word about the kinder, gentler IRS.

That's how every customer service encounter should end: with the customer telling others about their great experience. Do your custom-



ers tell other people about their great interaction with your company?

So, for this month's issue, we asked a variety of industry experts to weigh in on the venerable customer. For our experts interview, Sunstone Circuits' Nolan Johnson and our columnist Dan Beaulieu had a rousing discussion about what it takes to satisfy a customer, and how far a company can—and should—go to please that customer. Next, we have an interview with Jay Gorajia of Mentor, who discusses the company's consulting services and the group's focus on the "digital twin." Then, we feature an interview with Ben Jordan of Altium, who explains how the latest tool upgrade was driven by customer demand, as well as the many methods Altium uses to acquire and measure customer satisfaction. And consultant Tim Haag, formerly in tech support for an EDA vendor, tells us what designers really want, along with a few horror stories.

We also have columns from our contributors Barry Olney, Martyn Gaudion, and Phil Kinner, and we'd like to welcome our newest columnist, John Talbot of Tramonto Circuits, as well. Lastly, we have an article by Hemant Shah and Ed Acheson of Cadence Design Systems on the latest IPC-2581 updates, and an article on NPI by Dora Yang of PCB Cart.

It's already February, and we're getting ready for IPC APEX EXPO in San Diego. If you're at the show, stop by our booth and say hi. And if you can't make it, don't worry. We'll have video coverage of the entire event from start to finish.

See you on the road! **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 18 years. He can be reached by clicking here.

First 3D Imaging of Excited Quantum Dots

Quantum dots are rapidly taking center stage in emerging applications and research developments, from enhanced LCD TVs and thin-film solar cells, to high-speed data transfer and fluorescent labeling in biomedical applications.

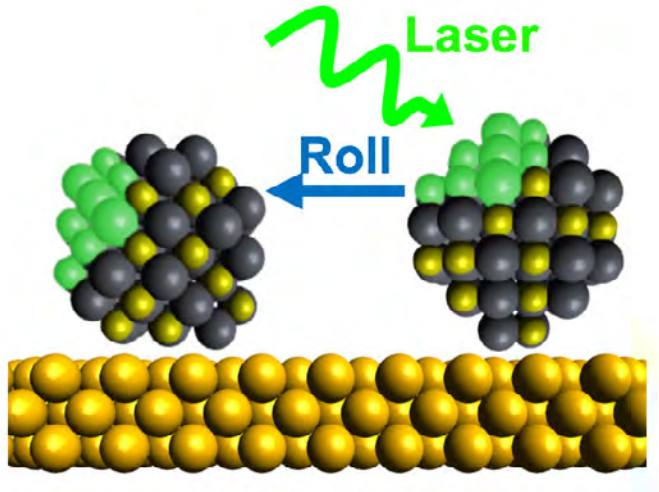
Researchers are still studying how to precisely control the growth of these nanoscale particles and their underlying quantum behavior. For instance, defects form during production of semiconductor materials, so identical dots can differ in composition from one another.

To learn more about these defects, a U.S. research team, from the University of Illinois and the University of Washington, has, for the first time, demonstrated imaging of an electronically excited quantum dot at multiple orientations. They report their findings this week in The Journal of Chemical Physics.

While the research in this article was limited to lead sulfide and cadmium selenide/zinc sulfide quantum dots, the technique can potentially be expanded to other compositions. Furthermore, SMA-STM can also be used to explore other nanostructures, such as carbon nanotubes and photocatalytic metal clusters.

Researchers are now working to advance SMA-STM into a single-particle tomography technique. But, before SMA-STM becomes a "true single-particle tomography approach," they still have to ensure that the scanning and rolling does not damage the nanoparticle while it is being reoriented.

Single-particle tomography would provide a clearer picture than conventional tomography by singling out defects in individual nanoparticles rather than re-creating an averaged 3-D image that combines the measurements of many particles.



Feature Article by Andy Shaughnessy, Patty Goldman, and Stephen Las Marias
I-CONNECT007

For the upcoming issue of our I-Connect007 magazines, we interviewed Nolan Johnson of Sunstone Circuits, and Dan Beaulieu of DB Management—our regular columnist—on the topics of knowing your customers, the challenges in dealing with customers, and providing excellent customer satisfaction.

Johnson has been with Sunstone for about 12 years now. His background is in computer science and then in capital equipment and display technologies, as well as PCB manufacturing. Currently, he is a project marketing manager at Sunstone. He's also on special assignment to their in-house sales team and doing some special projects around for that.

Beaulieu, meanwhile, has been in the consulting business for 20 years now. He works with PCB companies and contract manufacturers, helping them with their strategies, and sales and marketing, primarily for growth.

Patty Goldman: One of the things we hear in our ongoing expert meetings is that there is not enough communication between the different parts of the supply chain—the supplier and customer. There is a lack of communication; working together is not what it should be.

Beaulieu: It's very interesting because what's going on lately is that even my customer's customers are starting to come to me. I've had a few calls where a long-time buyer at Draper Labs, which is one of the highest technology buyers in the country, told me he has such a problem with board shops. And I told him it's because he doesn't communicate with them as well as he should. Going back to the old days when our customers, the Martin-Mariettas and the Raytheons, used to literally move into a board shop and work side-by-side with us on products that "no one" could build. And that's kind of gone by the wayside as we get into the no-touch stuff, which I picture as a kind of counter communication, if you will. And it's not the fault of the people that offer the no-touch service, it's kind of the fault of the corporations.

This is the way I envision it: it's almost like down in the basement of one of the large companies are designers and engineers who don't want to go upstairs to the traditional


Mentor®
A Siemens Business
Speed up your
PCB design verification
by up to 90%
Get HyperLynx DRC for
FREE for 12 months
COMPATIBLE WITH ANY PCB LAYOUT TOOL!
www.mentor.com/get-drc

buying channels and go through all the bureaucracy of buying boards. So, they simply whip out their credit cards and buy boards online or use design services online. And there's nothing wrong with that. But what happens is as that board is elevated and goes upstairs to the traditional buying systems, the people who end up building the boards, the more traditional board shops, have not gone through the development phase with the customer. That's where the communication breaks down. That's an aspect I'm working on a lot right now. When you start dealing with companies that are literally building products that the world has never seen, including circuit boards with technology that the world has never seen, it's time that somebody talks to somebody.

Nolan Johnson, Sunstone Circuits

Johnson: Well, this gets to be kind of fun because I do live and breathe in the area that Dan is calling the no-touch. We tended to find that this area is where the prototypers are doing their work, and Dan touches on something that's particularly important. If you're a prototyper, you're looking for a shop that can get you your prototype in a couple of days, quickly, in small quantities, and can be nimble and make changes alongside you to keep your design team moving forward until you get your prototype ready to go and to optimize it for production. Then once you've optimized it for production, maybe you've taken it from a 4-layer prototype down to a 2-layer prototype where you changed the dimensions or cleaned up the DFM warnings and you are ready to go into production and get some good deals out of your overseas production shop or your major production shop in the U.S.

Once you're there, it does change and there's information that we have established as the prototype partner that needs to be transferred over to the production shop to keep everything flowing properly. That's something that we see with our customers on a regular basis. We see it from the other way. We start handing things off, moving things over, trying to help the customers get their designs moved over into production and have things fall down on their face for the first couple of runs while they're getting up and going. So Dan's point is exactly right. How do we get that key step from helping with the prototype, all that knowledge we've built up because we're spinning that board with the customer and then getting it over into production where it can stay put and be stable with high yield and high profit for a long time for this customer? That's a key thought.

We've been working to develop some relationships with some production houses in order to be able to create a communication channel to do that. It's interesting from our perspective that it's difficult to get the attention of the production houses to do that. We're in a place where we're working with a lot of prototypers on a lot of different levels. Everything from the breakers that Dan mentioned, and down to university teams, individual entrepreneurs, and hobbyists and makers. Many of these projects are turning into production products at some level, maybe small, maybe huge, but that transfer over into production is an area where we're struggling to get that information passed over consistently and heard. I think there's some room for some protocols around that.

Stephen Las Marias: How is it in the contract manufacturer or EMS space, Dan?

Beaulieu: There is more conversation there by far. But for the most part, it's a longer quote cycle with parts and putting the package together, and there is much more conversation going on there. You just have to be closer. I work with one right now, for example, where I watch the project managers go back and forth for literally two or three days with the customer. Especially as you've got the quote cycle, which needs the customer to be at the table; and once wherever

it will be placed, then a great deal more conversation takes place.

Two things are key on a contract manufacturing level. They actually measure their sales people on the number of NDAs they bring in to get signed—that's number one. And then number two is plant tours. Plant tours to contract manufacturers are far more important. They're important to board shops, but they're far more important to contract manufacturers—because on the board side, you're literally building one part of a product, whereas the contract manufacturer is going to build the entire product. So, it is a much closer relationship. Several of my friends own quick-turn CM shops and I know they do a great job. They do things faster. They're much more streamlined but there are things like understood parts—part substitutions that are understood—because of speed. It's like they'll settle, they'll build the first pieces, but they don't always stay there. After the prototype is done, they might go to a part that's harder to get, that has a longer lead time but is a better part for it; but they wanted to see that the first builds work. But I find there's less of this breakdown in communications, if that's the shorter answer.

Goldman: What we're all hearing here is that regardless of who your customer is—and customer can be defined rather broadly, internal, external down the stream a bit—but the big thing is communication. Would that be true?

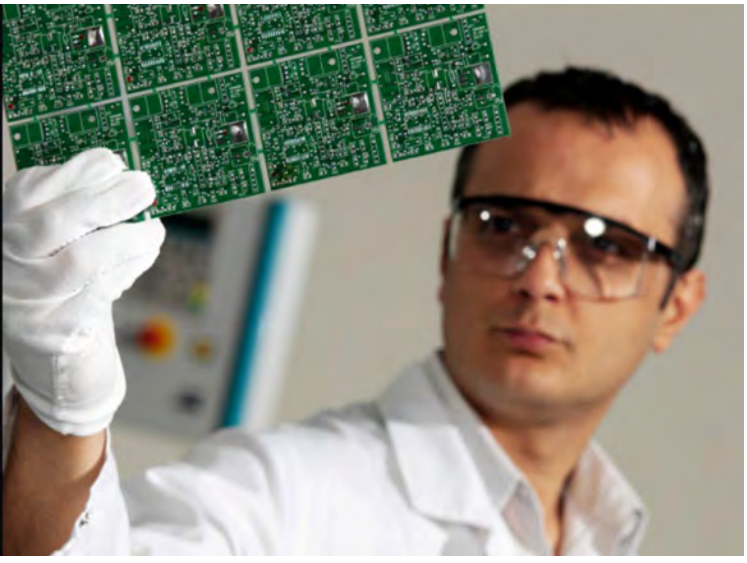
Beaulieu: Absolutely. And also, in what Nolan was talking about, I understand the need for that type of enclosed no-touch business. I've talked to one of the presidents of our country's largest no-touch and he told me that a lot of his business comes, believe it or not, on Christmas Eve, and on Christmas Day. Because that's what a lot of designers do. Not at the higher level, but at the NPI and hobbyist level; they really do not want to communicate, you know? I worked with one company—that we all know as a traditional company—that's got a call center and also has no-touch. I was running its sales force. I asked why my sales guys couldn't have the list of the no-touch customers. And the boss said, "Are you kidding? Those people would go berserk if somebody called them up. They don't want to be called up." It was very interesting.

I managed designers for years and I know the sales guy that I am. One of my friends advised me to tone it down when talking to designers because they're just a much more methodical group who doesn't want some backslapping sales guy talking to them.

Johnson: I think you're right Dan. We have plenty of customers who just don't even want to be contacted, and then we have other customers who are perfectly willing to start having that conversation. For me, what I'm discovering is that if they're thinking production, they're more likely to want to talk to us and keep the information flowing. There's a reason that Sunstone also keeps a 24/7/365 customer support line going. We have our customers who are placing orders on holidays. You know, introvert type designers hiding out from their family on Thanksgiving; while the turkey is being cooked, they are making their design order. We see this all the time, which is a key part of how this business operates. But it's interesting, what I was hearing from this whole communications bit, the further downstream you are, closer to production, the more likely communication is to happen.

There's a reason that Sunstone also keeps a 24/7/365 customer support line going.

We're in a unique spot. We're at the very front of that whole manufacturing chain where there are a lot of things being sorted out. We do learn a lot of things, but the next step down doesn't necessarily mean that everybody's ready to hear from upstream. I think that's the point I'm trying to make there. The



further downstream you get, the more likely the communication is to settle in. We're living up in the spot where the BOM is changing, where the design is changing. There is a lot of flux going on, as you settle it out. And maybe there's too much noise there. But at the point that we're finished and handing it down, there should be enough signals to make it useful to the next step.

Beaulieu: Yes, I think you're right. The other extreme I see is this: a more traditional PCB prototype company that has a barrel full of testimonials from customers who praise them for calling them up and saying this is wrong and things like that. By the same token, they have an equal barrel full of customers who are furious, who say "just shut up and build a board, quit bothering me." The same group of salespeople went to one of the major fabricators located in New Mexico, and they had a group of designers there that said when they were trained, they were told never listen to the board shops, they don't know what they're doing, designers are what you want. This PCB company ended up doing a lunch and learn. Another PCB company I work with also ended up doing a lunch and learn, and they were very, very well-received. They both came up with rooms full of people anxious to understand what goes on in a circuit board shop. And don't forget, most of us around this table grew up when people visited circuit board shops. But times have changed. When I managed designers for ASI, I had 30 designers. ASI had a board shop, and just three of them had ever been in that board shop. They were all 20-year people. That's the kind of communication that I'm really struggling to make happen—going to a PCB shop and understanding how a board is made. It's not a plastic card, you know?

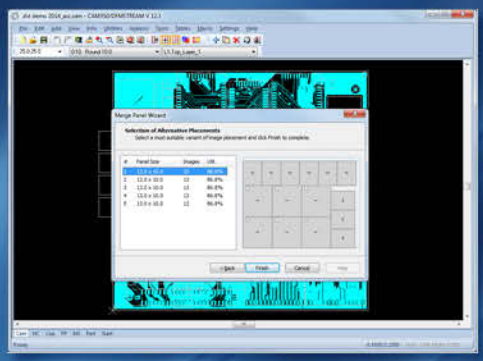
Johnson: We have plenty of customers who are angry at us because we're calling them back to ask and not just making the part, and then we have other customers who are trying to figure out why we aren't talking to them more often. That's never going to go away for us. It's always about how to walk that line. And you're right, Dan. I have a customer that I've been working with here for the past year or so. Sunstone's done a case study on them; the company is Eagle Harbor Technologies, out of Seattle. When I first started talking with them, they were giving us designs that were effectively unmanufacturable, and I was digging into them about why. It took talking to the customer. What I learned from them was that they're a startup. They're a team of physicists. None of them are EEs. They're all high-end physicists and they're building very fast switching, high voltage power supplies. This is some really cutting-edge stuff that they're doing, and the boards that they're turning in look like they should be automated test equipment probe cards. They're circular, nothing's on an angle. They're all over the place as far as that goes, and the DFM rules are really stressed when you're checking on them to see if they're manufacturable.

And yet we talked it through. Helped them understand exactly what the chemistry is going on inside the facility. Got them so that they had a real-world vision of what's happening once they finish their design. It helped change their perspective. It's not just, "Well, if I can define it in the CAD tool, it's got to be manufacturable, right? The tolerances and precision on everything is perfectly infinite." No, it's not. And as they understood that more, they started changing their designs. As they changed their

Solutions to Ensure PCB Manufacturing Success!

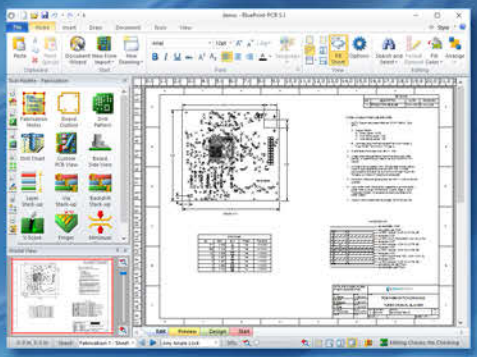
CAM350

Verify and Optimize PCB Designs for Successful Manufacturing.



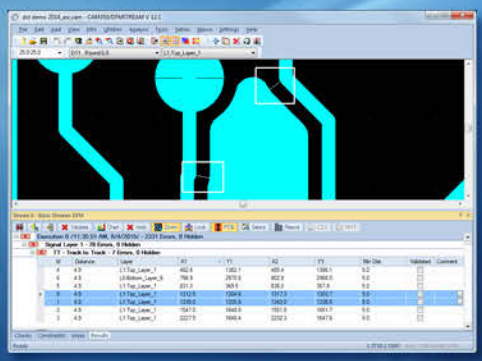
BluePrint·PCB

Create comprehensive documentation to drive PCB fabrication, assembly and inspection.



DFMStream

Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.



At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB design post processing needs.

For more information please visit downstreamtech.com or call 800-535-3226.

DownStream Technologies

designs, we were able to get them working prototypes with fewer turns and they were quicker to market. So, that part of the communication and education to our customers is huge. It is amazing how few designers even understand what's going on once they send an ODB++ file over to the manufacturer.

Beaulieu: I think there's a need for it, without a doubt. Based on the columns I did recently about customers. My friend Bob St. Pierre, who's a longtime buyer and now a consultant at Draper Labs, has agreed to work with me and put together some give-and-take columns on it because he was one of those people saying that.


Dan Beaulieu, DB Management

So, in the spirit of getting at least the high-tech customers together with board shops, we're going to start conversations that hopefully you guys will publish and take it from there. Maybe even do a traveling road show with it. Because I think it's needed. Especially with the high-end guys like Lincoln Labs, and Draper, the guys who are committed to having to be in the United States, by the way. They can't go offshore with the stuff they build. So there's a desire there for us to get this solved and to get the vendor and the customer closer together.

Andy Shaughnessy: What are some of the lengths you guys would go to for a customer?

Beaulieu: I can talk to that on a personal basis. I'm just open day and night. That's my business. I had a family Christmas party and I was on the phone with two customers for an hour upstairs in my sister's house. That's just that personal advisor, trusted advisor part of my business. I worked with this supplier for a long time, I still do. One of the things that came out of it, and I think it applies to all vendors of frankly, the American circuit board industry, is that right now, the American circuit board industry is very nervous of their vendor base. It's not a big secret, because you can sell 80 laser drills in China, whereas you might sell only three in the U.S. So who are you going to service?

One of the bases that I used with that company was that you're not just selling them a solder mask, you're selling them a complete service. Because the board houses just don't have the engineering base they used to have. Our suppliers in the circuit board industry have to engineer and they have to help us. All the good suppliers do. And we're going to need even more of that as time goes on because the vendors are our engineers, particularly in the $5 – 10–15 million board shops. They rely on their vendors to support them and that means doing everything. If you're selling solder mask you need to be their solder mask expert. If you're selling drills, whether they be mechanical or whatever, you have to be the expert. You have to do that service for them. It goes way beyond the selling. They are intimately involved now. All suppliers should be intimately involved with their customers. You could apply the exact same thing to people who are selling assembly equipment. It's really the ones that stay and service that do well. Not only service, but make that correlation, that synergy between putting this piece of equipment into the whole production line and making sure it all fits above and below where it lies in the production line.

Johnson: Yeah, at Sunstone we call that Lean. We call that 5S. And we actually have been doing a lot of work to do exactly that. Moving that expertise from the vendors into our shop to help out and make us more expert overall has been a side effect of lean and 5S.

Shaughnessy: What are some of the craziest things that people have asked you to do at Sunstone, Nolan? What are some of the horror stories?

Johnson: Being where we're at in the process, our challenge has a lot to do with how familiar is our customer with the actual jobs we do. I've shared this story before. I grew up here in the Portland, Oregon, area in the '60s and early '70s, and that's when Tektronics employed 30,000 people and they manufactured everything in the Beaverton campus. Nuts and bolts, the ceramics to make the displays, they did everything on campus. And if you wanted to understand what was going on with a particular project that was under development, it was my grandma's house for Sunday night dinner. Because my dad worked in the warehouse and then moved over to Electro Cam. My mom worked as a secretary watching what was going through procurement. My grandmother was an assembler. My uncle was an engineer. My aunt was another assembler. I mean, we had family members spread out through the departments of the company.

And the status report on the product went around the table with the mashed potatoes. If we didn't have an answer there, just walk down the street because some neighbor actually worked in the department you were looking for. If you were a designer and you wanted to know if this board was going to be manufacturable, you walked across the street into the fab clan and talked to the guys there. That was how you did it. That's how you had all that knowledge inside your company so you could do that.

As we streamlined and moved into the world we're in now, designers don't have that touch. They don't have that understanding of what's going on. They don't see it. They don't have the opportunity to figure out what goes on with that and work with the technology. So they end up sending stuff in that becomes unrealistic on the shop floor. And to answer your question Andy, the example I gave you with Eagle Harbor was one of the craziest examples of that. Because we were ready to lose that customer. They thought that we just couldn't build what they were doing at all. What it became was an ongoing conversation to help them understand what they were doing that was going to be a problem for anybody, and to help bring up their sense of knowledge. Helping a team of theorists become engineers is really what was going on there. And it was an interesting conversation for all of us. Education is a big part of this. Helping people understand, in this world, where the designers just do not know what's going on with the chemistry and the dynamics of that.

One of the projects I've been working on the past couple of months is putting together a series of short videos that spends some time on each individual manufacturing step of the process at Sunstone. These are something we want to use to help do exactly what we learned with Eagle Harbor. We want to help all our customers be able to plug into the manufacturing steps. See it in action. Get a little information about what the context is, why the particular stuff is used, what we're doing, what that means to your board, and give some specs and some tolerances, and do all of that in 45 seconds.

We're spending quite a bit of time to get this boiled down to the essence of it, so our customers can spend 10–11 minutes, watch the videos and at least get a sense of it.

We're spending quite a bit of time to get this boiled down to the essence of it, so our customers can spend 10 – 11 minutes, watch the videos and at least get a sense of it. And then this goes right back to the comment that was made early in the whole conversation. I think getting a customer tour of your facility is an underrated milestone in the sales process, which is of course a challenge for those of us who specialize in no-touch.

Beaulieu: I agree with that. Even more than a planned tour, I mean go back to the way things were done before. I know a number of

programs that were worked on in my career, the lantern project for Marietta. Those people came from Florida and literally lived at ASI, which was the company I worked with years ago. Because nobody could build those boards, their engineers and our people worked side by side on the board.

Even going back further than that, Rockwell and Motorola worked on the Viking project with the same thing. I mean, I can go back so far that I was a kid when I watched them measure impedance for the first time. We were doing the 16-layer board and I would deliver them to these two guys. One was from Burroughs in Pennsylvania, the other was at Maine Electronics. And they would look at the boards, they'd check the boards and they'd say, "Nope, throw these away. They're no good." I didn't even know what they were doing. And it was the first time I heard of impedance and that was like early in the '80s.

But all these stories go back to one thing, and that's where the customer was in the shop with the people building the boards and had a complete understanding. And keep in mind, years ago our customers all had board shops. All the OEMs had their own board shops. So the people we dealt with knew something about boards. They built their own. And the people who were used in the support groups for buying boards were people who had built boards. Those are all gone. How long has it been since there's been a captive board shop? The newer generation had never been in a board shop. They've never worked with a company that had a board shop, and that's what causes a disconnect.

Our technology stabilized for a long time, but in the last few years, it started taking off again and really taking off to the point where we can't get away with this gap any longer. The customer has to come to the shop. The engineer has to come to the shop, and I'm seeing a time when those customers are going to have to invest in the shop. I see that happening, where a lot of the companies, the smaller shops, just do not have the bandwidth financially to be buying the equipment that they're going to need to build the boards for these companies building products of the future.

So, I see right now a crossroads, a time where we're going to have to break through this thing. We're going to have to teach designers and we're going to have to invite designers into the board shops to give plant tours. Also, that's a two-sided thing. It's not all about that direction. There's also the direction of the board guys are going to have to open up their minds and listen to the designers and find out what they're trying to do. What the end product is and what they're trying to accomplish. We're going to have to do that as well. So that there's real give and take between the two. A true partnership that'll help the whole electronics industry move forward.

Johnson: Dan, I agree with you. I see that there are increasingly two communities emerging. There are the customers that are, just as you're speaking about, needing to get closer to the board shops and figure out how to do the designs they're doing. There is a definite statistical increase we're seeing in our customers for wanting to do HDI technologies, and at that point there's a lot more interactive discussion with the board shop to make that happen.

There is still though, a very large community of people who will back off that cutting edge who are doing very conservative work with basic SMT, or even still through-hole technology. Those customers tend to also be the ones

who don't want to talk so much. So, depending on where they fall in the technology scale, you could be in a place where they need to reach out, we need to reach out; there needs to be plenty of conversations and plenty of work behind there that is basically commodity work. Not everybody necessarily needs to go work directly with their board shop on every project but that is on the increase, I will agree with that.

Beaulieu: I'll tell a story here I've run across. I'm helping a company out of Brooklyn, New YorK, NYU guys, who are building boards with 3D printers. And the interesting thing is I asked them why they're doing this. These are NYU graduate students who won a technology contest by building a neat device, which is controlling a keyboard with your eyes, if you can believe that. I asked, "Okay, you went off in that direction, why did you get involved in 3D printing of circuit boards?" And they said because while they were building their project, it was so darn hard to find circuit boards, and so hard to get good circuit boards, they decided there's got to be a better way. This, as you know, is a long way from 28-layer blind and buried via boards, but on the same token that was the perception of smart graduate students coming into an industry where they would make boards. Believe it or not, their educational gap was such that they're graduate students. I asked them what their biggest problem is, and they said the ink. They were using stationery ink and I introduced them to Taiyo. They didn't even know about Taiyo. And I talked to my friend John Fix at Taiyo and he said he'd be there tomorrow. Because they're looking for that too, and you've got to realize, nothing against anybody here, it's just that people are working in a vacuum, in terms of circuit boards. I have a friend who's an instructor at British Columbia Institute of Technology and he uses Bob Tarzwell's PCB 101 handbook for his students, saying since he has a whole two-year course, these students might spend a day on circuit boards and then move on. So we've got to all work on that.

These are NYU graduate students who won a technology contest by building a neat device, which is controlling a keyboard with your eyes, if you can believe that.

Las Marias: In some of our conversations with contract manufacturers, they're also saying that designers should also consider speaking with them because sometimes, what these designers are designing are just not manufacturable, and quite difficult to adapt to whatever SMT system they have set up in place. So, they're saying that it's also important for them to talk to the assemblers. In fact, when we ask some people, they're saying that in their 12 – 20 years, they've had only one designer that came to them and talk to them about the design that they're going to do. Just one in that many years. But it turned out that the final output of the product is quite good. So they're saying that's also important.

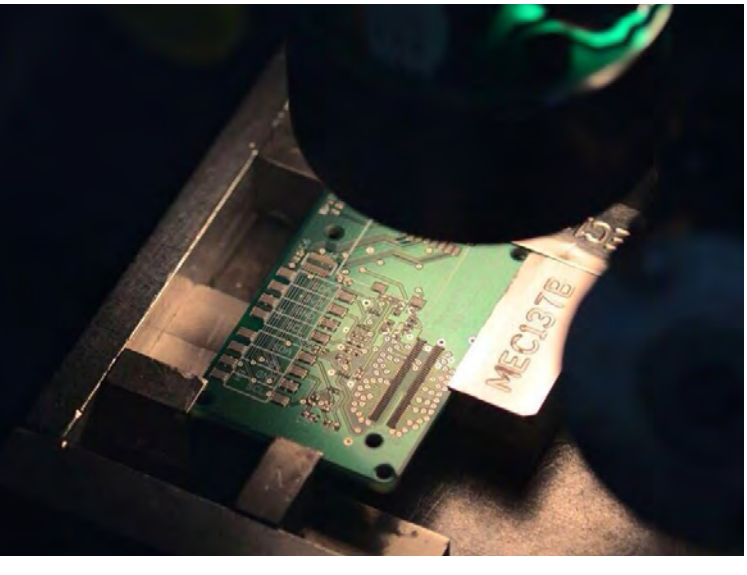
Beaulieu: Absolutely. I was at one last week and it was the same thing. Everything from basically ease of production to one of the important ones, which is component selection. Because there are certain parts that have longer lead times, that are more expensive, and they can be substituted with another part. And that drives the price up, it drives the lead time up, and it's much better when they coordinate. When I said they talk to each other, that's one of the things they're talking about.

If you look at my friends who have quick-turn production manufacturing shops, that's what they're doing. They get trusted for component substitution, even have little matrices that point out this part can do the same as this part and they figure it out that way. But if you

think about it, when you're building a box or a product, there are things that happen when you're actually building it, such as if this screw was moved one eighth of an inch, they would be able to fit this in. That kind of stuff, that's important too.

I worked on a project years ago where it was a board part, but it was a part assembly with a copper core with two polyamide skins over it. The core was live, so the boards had to connect through the core. Those boards cost almost $3,000. The designers called us in and asked why these boards cost so much, and what can we do. My partner simply went up to the boards, drew a picture of the core, drew a little slot in it, said "You put a connector over the top here and it won't hit the dimension of the board. And now these boards will cost like $600 and you're getting everything you need." I've seen contract manufacturers doing a lot of that. That one necessarily is on boards, but they're doing a lot of that kind of insight to help the customer who's fighting for nickels and dimes, lead time, and time to market. It does apply as well to contract manufacturers. Sometimes even more.

Johnson: That's a great point, Dan. That's why you're using contract manufacturer nowadays, because they are doing that sort of work. They have that experience. They have a lot of other jobs from a lot of other customers. They've done all of this before. They can often have the practical experience that can tell you how to do it better.



From my perspective. I'll just speak from my perspective on this one. I represent PCB123, at Sunstone as well, so that's the PCB schematic capture, physical layout, has an autoplace, autorouter in it. We just came out with a new version that connects to a startup called SnapEDA. And part of the whole part substitution issue is understanding if the part is a new part, or it has just been given a warning that it's going obsolete, or it is on a long lead time to get supply from the manufacturer. All the statuses for this and how it is going to affect your project. It's one thing to be having the conversation with the assembly houses and sitting there with your bare board, and they're telling you that the part is on a 12-week delay; and the other one is actually knowing that information when you're designing your board. If you can get some of that information into the CAD tool, you can make adjustments to supply issues. If you can make the designers aware of that, then they can make some adjustment decisions while it's still cheap, while they're laying out the board, before they've committed to something.

And that's exactly what we've been trying to do at PCB123. I saw this years back when I started my career at Mentor Graphics, and we were working with PCB123. We have the bill of materials that we've built up as you're laying out your design and, right now, you've got one source. You can query DigiKey to get a sense of the pricing and availability of the parts in your bill of materials, right inside the CAD tool as you're designing. Just go over and click a button and it gives you some information about that. If you've got a serious issue, you could figure it out. One of the side effects for our customers, is when they come back to a design they did a year or two ago, it's a one click process to find out what things of their bill of materials have gone away. Sent to them in seconds. In September, we introduced support for SnapEDA, a great product in the sense that the SnapEDA team is out there building a cloud-based library of parts definitions for schematics, physical layouts, 3D, working with

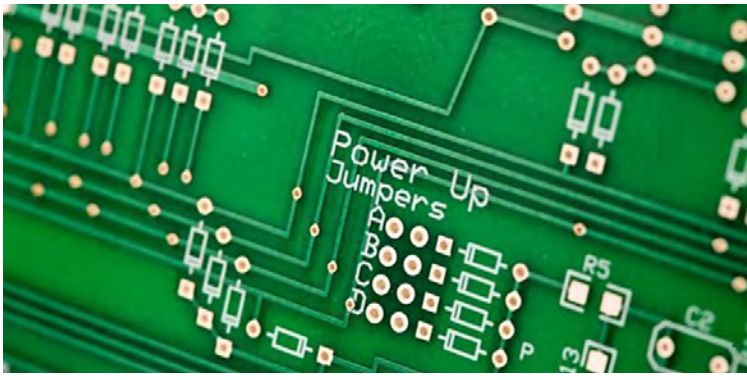
the part manufacturers to supply their parts directly to the library, and crowd sourcing with engineers to get new parts in the library. We've got millions of parts and it's a couple of clicks to download them out of the library, out of SnapEDA, into your CAD tool. They're supporting most of the major CAD tools including PCB123. And that process means that it's like using iTunes to get your parts definitions and know that you've got good ones that have been certified and validated as correct. Giving me, the designer, up-to-date information as far as how available that part is and the peace of mind that you didn't just spend two days defining a brand-new footprint for your library that is wrong.

So those are some of the places where I think that's a little bit overlooked through the manufacturing supply chains. The more we can get those decisions to be made well in the CAD tool, the fewer conversations we have to have further down the street. At least that's one way that we've been approaching it with PCB123.

Las Marias: Dan, what is your advice for our readers when it comes to ensuring 100% customer satisfaction? Maybe you can talk from the perspective of PCB fabrication and the contract manufacturer's side.

Beaulieu: It's an understanding and it's listening. I like to tell my guys to make sure they not only understand the product they're building for, whether it be a box build or it be the board itself, but also understand the customer. What does the customer need for success? What is their market? Whether it is medical, defense or security, or commercial where it's very competitive. Get the characteristics of the customer and apply those to the product in the end and it makes a lot more sense. I also like to encourage people, say it's a circuit board for example, to think what that circuit board is seeing when it enters the customer's facility. Even to the point of the receiving, because as you know, documentation has become more vital now than ever.

Put yourself in the customer's shoes. Seeing what the customer faces, what their challenges



are, and what it takes for them to be successful in their marketplace. Because if you do that, you become completely valuable to your customer. And if you do that enough, after a while, that customer is going to give you leeway on pricing. I used to say when that hotshot accountant shows up and says that your solder mask is 20% more per kilo than the other guy, you'll want the guys in solder mask and engineering to tell him to shut up and talk about all the value they're getting with that product. And it's the same thing with circuit boards. Supercede the pricing with value.

Johnson: I think Dan's right on the money there. One of the things that I talk about with our team often is just pulling one of Steven Covey's seven highly effective methods, and that's "if you wish to be understood, seek first to understand." There needs to be a dialogue that we create in both directions. Not just us understanding our customers, but also helping our customers stop and step back and seek to understand what happens as soon as they've handed their design over. That dynamic, once you have that going on in your relationship with your customer, then everything that Dan was discussing starts to happen pretty organically.

Las Marias: Gentlemen, thank you very much for your time and insights. We greatly appreciate you speaking with us.

Beaulieu: Thank you.

Johnson: Thank you very much. DESIGN007

Worldwide Systems Consulting: Tools, Process, People

Feature Interview by Andy Shaughnessy and Barry Matties, I-CONNECT007

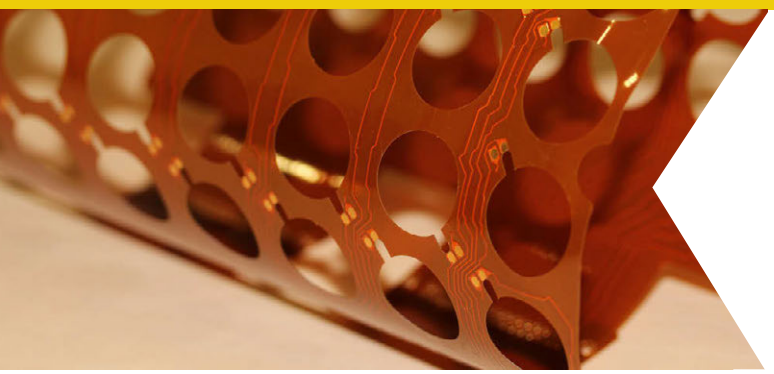
In Munich for productronica in November, Editor Andy Shaughnessy and Publisher Barry Matties sat down with Mentor's Jay Gorajia, director of worldwide systems consulting. Gorajia discussed Mentor's systems consulting business, their focus on the "digital twin," and how their acquisition by Siemens is benefitting Mentor and their customers.

Andy Shaughnessy: Jay, please give us a background on what worldwide systems consulting entails for Mentor.

Jay Gorajia: First, a little about consulting in general. Mentor focuses on many different areas, although our strength has been in the EDA market, and more specifically, the IC design, simulation, verification and validation flow, and PCB design, simulation, verification and validation flow. We are organized into four main consulting groups, two focusing on IC design, simulation, verification and DFT, one focusing on cabling design, wire harness design and services around products for design and manufacturing of wires and wire harnesses, and the group I manage, focusing on PCB design, simulation, verification and validation as well as serving electronic manufacturing business. The electronic manufacturing business came into Mentor about six years ago, with the acquisition of Valor. My group specifically focuses on PCB design, simulation, validation and enterprise data management, component engineering consulting, as well as the electronics manufacturing industry to help customers with methodology services, and services around either automating, integrating, or

We deliver Advanced PCB Solutions!

Our unique processes enable us to create and deliver Advanced PCB Solutions that meet the needs of the demanding market at competitive prices!



Impedance Tolerance <5%!

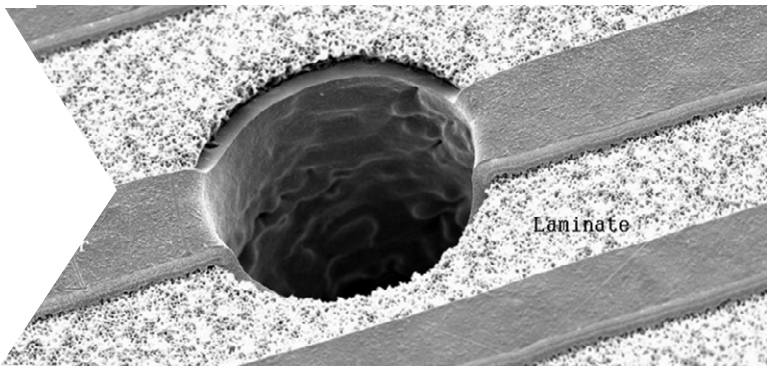
With our panel plating process combined with continuous foil lamination, our impedance tolerances are second to none!

CLICK TO LEARN MORE

Landless Via Technology

We're the only manufacturer in the world to have perfected a liquid photo-imageable resist (PiP) in electrolytic form, used for all our production.

CLICK TO LEARN MORE



Tensile Strength achieved with Candors Panel Plating

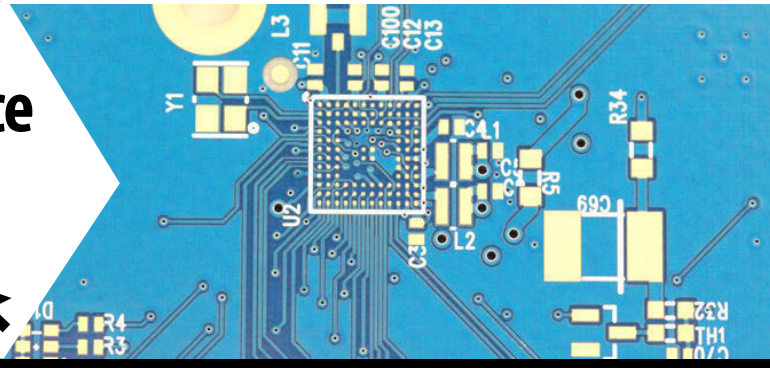
Practically perfect plating exceeds class 3A elongation, purity, and tensile strength

CLICK TO LEARN MORE

Fine line PCB, 2 mil line /space

We have developed a unique set of processes based on electro-depositable liquid photoresist.

CLICK TO LEARN MORE



CANDOR INDUSTRIES INCORPORATED

ADVANCED PCB SOLUTIONS

PHONE: (416) 736-6306

WEB: www.candorind.com

SALES: sales@candorind.com

further leveraging the Mentor suite of tools. In short, we help customers achieve value realization from either software or equipment they've purchased.

On the PCB side, in addition to the full breadth of PCB design, component management, analysis and simulation tools, we also have a team focused on integration of PLM and integration with various other ecosystems to optimize that value chain.

We've embarked on a mission of the digital enterprise. The digital enterprise as a concept means we're moving to help organizations virtually ideate, simulate, and emulate, as much as possible before product designs get into the real world of manufacturing where costs are applied for real materials, for real assets, for real production and production time consumed. Since being acquired by Siemens, we are core to the strategy for a digital twin, when it comes to the electronics flow, as a key part of the digital enterprise strategy.

Shaughnessy: Tell me more about the "digital twin."

Gorajia: A digital twin is literally a virtual version of a product and the physics-based models around it and everything one needs to do physically. A virtual product model, in which you would, in the design stages, perform all the simulation, validation, and analysis whether it's thermal characteristics, signal integrity characteristics or design for manufacturing analysis. Then, as we move into manufacturing, how can we simulate what the manufacturing environments would be? How do we design for manufacturing? How do we simulate the process engineering and manufacturing engineering for the product? What equipment is best to be run for this particular product at what time and how? What are the parameters of those and what are the constraints (cost optimization, material availability, asset availability)? Do as much of that product engineering offline as possible without tying up physical assets.

When we move into manufacturing execution we have all the tools for that as well. We have tools to virtually simulate before you commit resources, labor, and assets in terms of material. Strengthening the digital twin was a key piece in the Siemens strategy. Siemens has all those tools for the mechanical design through manufacturing and related tools to enable a digital thread through that flow. We now bring to the table the electrical and the electronics flow. We're filling in that gap, because almost everything today—cars included—have not only mechanical pieces and electromechanical pieces, but have electronics, wires and in many cases wire harnesses. All that needs to be designed, modeled and simulated and validated before it goes into manufacturing. From then manufacturing needs to be optimized and managed as well.

That's the direction we're headed, is to be able to take and optimize as much as possible, all the business processes, all the physical processes, manufacturing processes, for example, the design process to make sure that at the end of the day you're creating a product, first time, at quality, at margins, at cost, on time. That's the goal.

As mentioned earlier, one part of my team focuses on manufacturing consulting. We'll go into factories and analyze how a company is operating today, what their inventory looks like, what's their material flow, and how well it all works. Based on the initial assessment, we put together a blueprint of recommendations to

optimize further. Siemens has a digital enterprise maturity index or assessment that we are incorporating into the electronics manufacturing business. Industry 4.0 is an example of a digital enterprise implementation. Industry 4.0 is focused on virtualizing and connecting design and manufacturing, and automating those processes to have a highly flexible, automated operation. It includes the supply chain; it includes decisions made at the level in which you're still ideating. Virtualization starts at ideation and continues all the way through the product lifecycle.

In addition to the business process optimization, we also have teams that will go on site and make process optimization real for an organization. If the organization is going to solve some of their business challenges using Mentor technologies, the team would be involved with implementation the Valor manufacturing factory automation systems. In addition, most systems would need to be stitched in to an ecosystem or other systems. That may be a warehouse management system, ERP system, PLM system, remote and/or local automated material handlers and perhaps an internal analytics infrastructure. My team plans, manages, and execute that as well.

Shaughnessy: It sounds like you're not doing so much putting out fires as you are helping somebody expand or add a new module or add a new step along the way.

Gorajia: The key here is to help organizations, at least those that want the help, along the journey of further automating, optimizing their business process, and optimizing their manufacturing processes to better realize the best that they can be.

Shaughnessy: Tell me, what are some typical examples of something where you or your team get called into a facility?

Gorajia: One common example of a business problem starts out with, "We've got a problem with our balance sheet. Too much material in the factory; can you help us with this?" Usually that entails us walking around and finding that there's large amounts of stored or buffered material. Areas where they're not moving material fast enough. There are parts in their stock room inventory that have been sitting there depreciating.

We would actually build a workflow, a material workflow, and optimize how it's used. There should be no reason why there's material sitting on a factory floor. How do you automate it so that there's just-in-time queueing, so the right materials are in the right place for the right production order at the right time, based on customer demand and agreed timelines? How do you make that a reality? Usually, we see 40-45% improvement on just the material inventory once we get that flow in place. That's a combination of technology, automation, and business processes. One can't always throw technology at a problem. There are processes and people considerations that have to come along with addressing a business challenge like that.

There should be no reason why there's material sitting on a factory floor.

Shaughnessy: Do you use the lean process?

Gorajia: Yes. Lean methodology is key to the analysis parts of an assessment, and the resulting process definition.

Barry Matties: When the owners of companies walk through and they see all this work in queues, they feel like, "Oh, we're doing great. We're busy."

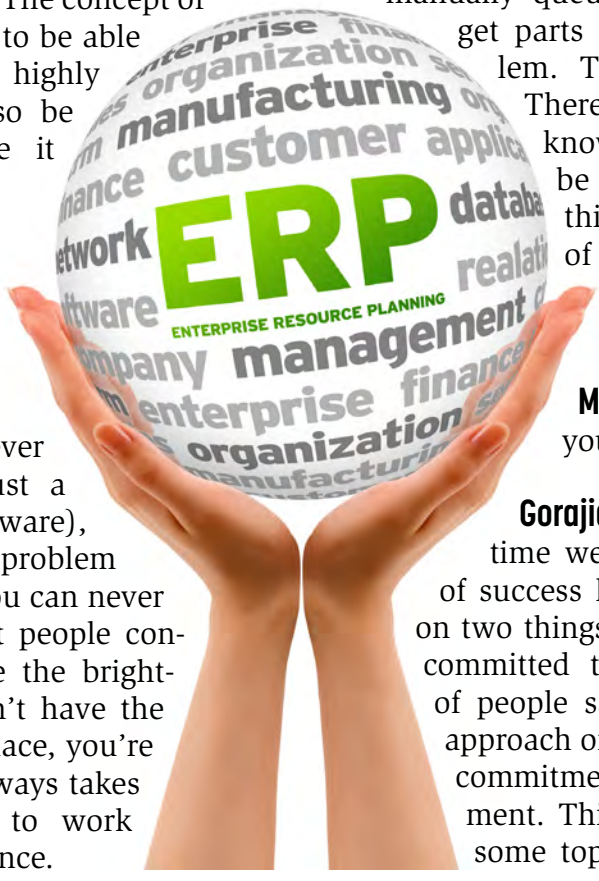
Gorajia: Right, but it could be exactly the opposite, actually.

Matties: The mentality is such that it's really hard to shed. What resistance are you meeting in that regard?

Gorajia: Usually, it's not that much resistance when you explain the problem in the terms of the financials or what's in it for them. I'll give you an example. Everyone has a feel for how much inventory they have, right? Kind of a rough number of X amount of millions of dollars of inventory. When you say that 30% of that can go away, then they start doing the calculations in their head: "Wait a second, that's 30% to the bottom line," Not all of it, probably a good 15%, is instant margins. As soon as you say that, it at least perks up the interest of, "Okay, how?" Then you can start the conversation. Although we may start the conversation that way to rattle their paradigm, it isn't far from the truth of what we can achieve in many cases.

Matties: What you were talking about though, is some of the process blocks and the bottlenecks, if you will. That's what they really have to attack to make this work, right? It's really coming back down to the mentality of lot size of one. We build one panel at a time.

Gorajia: That's the theory. I mean, a lot size of one is high customization. The concept of lot size of one, is, "I want to be able to manufacture in a very highly configurable way, but also be profitable." That's where it started.



Matties: Manufacturing is process-neutral. Does it care what you're putting in it?

Gorajia: It does. You can never solve a problem with just a technology (or tools/software), you can never solve a problem with just a process, and you can never solve a problem with just people considerations. You can have the brightest minds, but if you don't have the tools and the process in place, you're not going to make it. It always takes those three. They have to work together in a specific instance.

That's a key aspect of change that needs to be conveyed, because it's very easy for senior management to say, "This guy did this amazing work at this other factory. I'm going to bring him over here and miracles will happen." Also, the guys on the factory floor have been doing good work, for a long time. They are not interested in changing the way they do business. There needs to be something in it for them. Usually that's the point in which we move forward by saying, "Guys, here are 15 margin points you can take off right off the bat." "Well, how long will it take us to get there?" "Well, that depends on you, but here's your playbook." Sometimes we will leverage our digital enterprise maturity model and assessment to create this playbook for them, depending on the end-goal. That end-goal may not be margin points. It may be optimization of data flows to ensure synchronization with ERP, or meeting customer delivery dates more often, or even improving quality, while improving velocity.

There are technology pieces that have to be at play, because if today an organization is manually queuing inventory in order to get parts to the line, that's a problem. That could be automated. There's technology to be able to know this work order should be scheduled for this shift, on this line, due to availability of specific feeder inventory based on parts and parts size and parts availability.

Matties: How successful have you been in this endeavor?

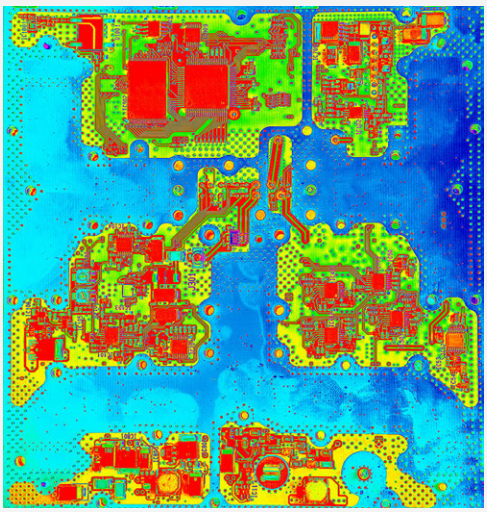
Gorajia: I think 80-90% of the time we're successful. The degree of success has a huge variance based on two things, and the first is how fully committed the organization is. A lot of people say we need the top-down approach on changes because we need commitment from the senior management. This is true. It needs to have some top-down buy-ins to get pro-

Technology leader for high performance thermal management materials

ventec
INTERNATIONAL GROUP
騰輝電子

Ventec VT-5A2 - Designed for the world's most thermally demanding PCB applications

VT-5A2 is designed for the world's most thermally demanding PCB applications such as power converters,hybrid multilayers and high power electronics with heavy copper designs providing enhanced signal stability in harsh environments. With VT-5A2, Ventec has created its highest performance thermally conductive laminate material so far, with a polymer matrix that is fully compatible with other Ventec laminates, epoxy- or polyimide based materials including our IMS tec-thermal® range & tec-speed®. With VT-5A2 total compatibility in the manufacture of hybrid multilayer boards is guaranteed.



VT-5A2 best-in-class performance:

- thermal conductivity of 2.2 W/m.K (8 times that of FR4)
- in plane thermal conductivity of 3.4 W/m.K
- high Tg of 190°C
- best in class thermal performance (T260 >60 Minutes, T288 >30 Minutes and T300 >15 Minutes)
- M.O.T. up to 150°C
- lead-free assembly compatible
- fulfills RoHS and WEEE requirements
- UL94 V0 compliant
- IMS materials compatibility in hybrid constructions
- choice of thin core and prepreg variants

The VT-5A2 thermally conductive laminate is manufactured by Ventec using strict quality controlled processes that are certified to AS9100 Revision C, ISO/TS16949 and ISO 9001:2015, and, like all Ventec products, is backed by a fully controlled and managed global supply chain, sales- and support-network.

Ventec - Your strategic partner for safety-critical supply chains

Ventec International Group
21 Water Street, Amesbury
Massachusetts, MA 01913, US
T: 978-521-9700
E: saleseast@ventec-usa.com
@VentecLaminates

www.**venteclaminates**.com

cesses and the resources for the actions needed. However, it is the guys on the factory floor, in the design room, or in any of these places that have to execute. You have to get their buy-in as well. Number one is how committed are they and how much do they want it to work.

The second factor also has a huge effect on the degree of success; the "final mile." I threw out 80-90% success, but not all of them have reached their final goal, let's say, the 40th percentile. Because what normally happens is they start seeing success, like in our earlier example; they start seeing all these materials on the factory floor disappearing. There is a common problem with patience these days. They see it and they're like, "Cool! We're done. You guys said we would get rid of this material." No, we're not done because this was step one. Step two, we need to stabilize that process so it's consistent and reliable, and then take it to the next level. Are you guys ready? There's a huge percentage that say, "Well, you met my goal for now. We're good with that." You may have the commitment, but only to a degree. They don't finish the final mile.

Shaughnessy: Because even if you get the tools and the processes right, you still have people as a variable.

Gorajia: Yeah, true. And they feel that, "Okay, I've got some success. Let's take the success and run." You kind of block yourself from, let's say, if it's a theoretical 40% improvement, at that 20-25% mark they see enough gains that they're happy. Then, usually it stalls after that because they've moved on. The focus, resources, and urgency is gone.

Shaughnessy: You said you're over on the design side too. How does that fit in and how does that typically go?

Gorajia: We're called 'systems consulting' at Mentor because about two years ago, we merged the PCB design consulting group with the manufacturing group because there are too many touch points. Designing, validation, simulation and analysis all have a huge influence on manufacturing. Many design organizations throw designs over the wall, and the manufacturers catch them and have to deal with what they have. If they're an OEM, vertically integrated organization, sometimes they can push back. An EMS company most likely won't. They'll just take it, and deal with it. They'll take the hit on the margins because of crap data they get.

What are synergies we can build? Going back to the original digital enterprise concept, how can we—at the design stage—start simulating as much as possible up front in this digital twin environment? Not only the design pieces, not only simulating for signal integrity, power integrity and thermal flows and all that, but also how do we start simulating manufacturability? How do we simulate what the capabilities of a factory are up in the design stage where you can get to a point where you run some process on a design and it gives you some sort of a producibility assessment? That was a key focus of creating our group. It's a lot of my focus as well, as we build out the consulting organization such that systems are modeled, simulated, and optimized more completely.

Matties: And that's based on the manufacturer that you're selecting? That's pretty cool.

Gorajia: We want to tie these together. First, you want to simulate as much up front, first-time-right design. The second is feeding back; that's the big thing that's missing in today's world.

Shaughnessy: Because the manufacturer will fix the crap data and just update the board and never even tell the designer.

Gorajia: The designers don't even know what's happening.

Matties: So many people say that designers really need to understand the manufacturing process. To a degree, yes, but there's the other side that says the designers just need feedback. The lag time between design and completed board, they're on to other designs by then and it's not a natural feedback loop. How do you improve that feedback loop so it's more automated?

Gorajia: First of all, in order to solve the problem, it needs to be a systematic or systems-based feedback, not a people-based feedback. People obviously will be the authors of feedback. But the system should drive the communication. As you said, it might not be what either side is working on right then and there…they've moved on to something else. For example, if a design organization is going to have an EMS partner run design reviews before production, is there a platform in which they can collaborate and capture that feedback such that both organizations can then monitor those recommendations that were done or not done, accepted, or waived? Is it a repeated violation? Does it show up two revisions later? Does the same design group have a consistent DFM violation compared to others?

We need tools, process, people. We need a business process that says, "The last five designs that we've done with these guys, they've recommended these changes. Can we then take it and add a process that updates our design characteristics, DRCs, the design rules, the DFM rules, the signal integrity rules, in order to compensate for all of these recurring problems in manufacturing?" That's step two. Then step three, how can we pull real quality data back into the design so that DFM is based on actuals? I'll give you an example of that. Most of the time, the bill of materials on the design side are chosen by an engineer who's familiar with the product and has worked with various components for some functional benefit. Usually, in every bill of materials there are three or four options of which part to use in manufacturing, many times called alternate parts. Which one is the best? That is usually determined in manufacturing by what best cost they get. If they have a choice of three, and one is already in inventory, that's usually the first one they choose. If it's not in inventory, then which one has the lead time with the target cost?

In manufacturing, that makes sense. In design, when they're picking parts based on form, fit, and function, where's cost? What ends up happening is there isn't any systematic way of understanding what the risks are when you select a part. A lot of designers now are moving towards integrating their upfront library management, component management process, with various suppliers. They're doing that part great. What they're missing, though, is with my manufacturing partner, whether it's an EMS A, B, C, or whether it's their integrated manufacturing line, what if we can feed back defect rates for a specific component or component vendor or distributor? That data is still missing. If you're integrating with IHS or Silicon Expert or any 3rd party component aggregator, and you can get all the parametric data, cool. You can get cost data, cool. But for the EMS partner, if a component has a higher defect rate compared to another, it can really

affect quality, cost, and for the manufacturer, margins.

Matties: How do we determine what the most important feedback is for a designer? Who decides?

Gorajia: Identifying the top-running manufacturability issues over the last three years would be a target place to start. Start building trends, start building correlations based on that. There are analytics tools all over the place now. Then, start honing on what those are and analyze if there is a systemic problem or a people problem. This is what we help organizations do.

Matties: Because we're in a period where data is overwhelming; you can get paralyzed in data. Mentor has been working a lot on Industry 4.0 total factory communication. How does that all play in to what we're discussing here?

Gorajia: Well, all this feedback can't happen unless we have it organized and structured in such a way where we're gleaning information out of all of this data. We know what parts and what defect rates there are in the previous example, right? We can tie that into a correlation that says, "Okay, this particular design organization, out of the five we work with, has a higher defect rate than these other ones. Why? What's the root cause of it? How do we whittle it down?" Now, with a lot of the algorithms out there, advanced neural networks, clustering and everything, we can actually start predicting trends and saying, "Okay, well in the last five years of data, this is how we've done. Now, let's see what course are we projecting and how do we correct the course?"

Okay, this particular design organization, out of the five we work with, has a higher defect rate than these other ones. Why? What's the root cause of it? How do we whittle it down?

Matties: is there a standard language for all the processes to talk to each other?

Gorajia: We would consult that problem, too.

Matties: There seems to be a battle for that stuff.

Gorajia: Yeah, but standards, as a topic, is fine to discuss. But standards don't solve business problems. A business problem says, "How much money am I going to save? How much more money am I going to make?" The typical factory doesn't make a dollar unless there's a product coming out the other end. You can have the discussion about standards, and I'm not against that, but those are problems that can be solved. Communication standards is a methodology. It consists of data that should be meaningful, and a communication transport that all the constituent systems can understand and glean what they need to. As an industry, PCB assembly manufacturing is probably the worst at adopting standards. The Semiconductor industry standardized. Many of the CNC machines now speak OPC UA (Unified Architecture) and the like.

Matties: Those are just arguments I hear and I agree with what you're saying. I think it's really insightful.

Gorajia: You get down in these proverbial rat holes on discussions, but we're here to make business. Manufacturers need to produce and designers need to produce. Companies are here to make money by pushing specific products. These types of discussions shouldn't stop it. All of this is solvable. I'm sure in time we will have a standard. Mentor has published standards as well.

Shaughnessy: It sounds like you really see design as the first step of manufacturing. A lot of people don't really understand that.

STRIVING FOR EXCELLENCE IN EVERYTHING WE SAY AND DO.

- Prototype to Production
- Time Critical Production
- Commitment to Customer Service
- Best Quality & Delivery Performance
- Fair and Competitive Pricing/Costs
- Rapid Response to Quote Requests
- Flexibility Scheduling
- Stock/Consigned Inventory Programs

Review Our Advanced Capabilities Now

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

The Absolute Best Value in High Technology Circuit Boards

847.891.5800 • www.eagle-elec.com

Gorajia: Right, and we're uniquely positioned as a company, because we have both. As an organization, Mentor has the full design suite of tools and processes. We have a full suite of manufacturing tools and processes. We're in a unique situation where we can have a big impact in the industry and for sure with organizations looking to streamline the full digital enterprise. We know where things tend to stitch together to optimize. Most organizations just want us to come in and help them and solve this or that problem.

Matties: So how does the consulting work? Do they hire you, or is it part of a package?

Gorajia: Both. I mean, it's kind of flexible. A lot of times I'll start the conversation with an organization because they want to either pull some experience, knowledge or ideas from us or take our guidance on which direction to take. Then, whether that conversation turns into a paid engagement, part of a package with Mentor tools or whether we give them enough information for them to go do their own research, that's up to them. Either way it's fine.

Matties: What is your background?

Gorajia: I have a bachelor of science in electrical engineering with an MBA in technology management. I've been with the Valor technology for 22 years. I ran the operations in Asia for Valor for almost eight years, which fully indoctrinated me in the manufacturing environment…even though electronic design was my degree. I was responsible for integrating the design consulting group and the manufacturing consulting group together two years ago. That enabled us to start creating these synergies I discussed earlier.

Matties: Now, as I was standing at your booth, I noticed people saying, "Oh, a Siemens business." How's that working out?

Gorajia: It's actually fantastic. Siemens has tons of tools around the mechanical, electromechanical, design, and manufacturing process. We fit in with all the electrical and electronics. When you build a car, you have all these ECUs and all the wire harness and all that that still needs to be modeled, simulated, designed, and manufactured. We kind of filled that out in their digital enterprise strategy.

Matties: Is simulation something that designers are using on a regular basis in an effective way? I hear mixed reviews about whether they use it to begin with, and why wouldn't they? Secondly, how are they interpreting what they see? Maybe that's just knowledge, because the tools aren't talking about best practice, are they? The tools are just talking about design rules.

You always have the SI expert, the PI expert, and the DFM expert. Many times, they're the bottleneck in the process.

Gorajia: Yeah, to some degree. I think one of the places that, at least in my group, we've embarked on is taking simulation and moving it from being—I hesitate to say it—but moving it from being an art to becoming a systemic science. You always have the SI expert, the PI expert, and the DFM expert. Many times, they're the bottleneck in the process. Organizations may have 20, 30, 40 designers, whether they're at the schematic capture or the layout side pumping designs out. Then it hits the experts and it comes to a grinding halt due to bandwidth. If we can take that knowledge, put them in a system that says, "Okay, for everything that looks like this, these are the things that need to run. Go." Then we can start moving it into the designers' hands, because the designers probably need that feedback much sooner than they're getting it today.

Shaughnessy: They all have signal integrity problems, just about.

Gorajia: I mean, especially for those things that are high-speed. Designers need feedback sooner and they would be happy to run it if it was automated and easy to use. We're running a lot of projects like that right now where we're encapsulating what's in the experts' head in terms of DFM and signal integrity rules for particular products and shoving it into a black box, so to speak. Then the design system can go in there, run it, build a report, and automate that whole process, whether it's DFM, manufacturing, signal integrity, or power integrity.

Matties: In terms of time, how much savings, on average, does that produce for a company?

Gorajia: I'll break down the question into two. I think it doesn't save any time on a designer's point of view. I think what it does do is the design process time is shortened dramatically, because you eliminate at least one or two loops internally. It's not just a single person. You'll have loops with a signal integrity expert. You'll have loops with the DFM expert. I'm not saying they will be eliminated, because you will always have those design and product challenges that one needs an SI expert for and especially for new products coming down the line; somebody needs to create the rules.

The expert is that guy who is pivotal to the success of an organization automating simulation and validation. You'll always need them. What we're trying to do is say, "Let's take a percentage of what they can do that's repeatable and bring it to the designer so they can run it quicker, faster, in an automated way." This also enables analytics around designs, to understand how often a violation occurs, where it occurs, whether it's all design or only some, all design group or only some, etc. We spend time with organizations on analytics of designs and the design quality as well.

Matties: Thank you for your time and insights.

Gorajia: Thank you. DESIGN007

The Advent of Flying Cars and the Road Ahead

Increasing traffic congestion across mega cities and large urban centers, coupled with the resulting loss to the economy, is driving the need for more efficient modes of urban transportation. Flying cars are being explored as an alternative form of future mobility, making use of the underutilized domestic airspace.

Flying cars are set to disrupt the personal mobility space of the future with at least ten early entrants expected to launch various versions of flying cars by 2022. OEMs and other major industry players are expected to follow suit with the introduction of prototypes during the subsequent five years.

Joe Praveen Vijayakumar, industry analyst at Frost & Sullivan, observed, "This space has been witnessing bustling activity, with new players from various industries entering the race to build flying cars. We have also seen a surge in funding as several companies have raised funds or been acquired by established players from the automotive industry."

The United States and United Arab Emirates are key markets for flying cars, driven by factors such as high per capita income, purchasing power, regulatory quality and technology capabilities.

Altium Loads the Bases, Customer Service Brings Them Home

Feature Interview by Andy Shaughnessy
I-CONNECT007

In this interview, Ben Jordan, director of product and persona marketing, discusses the latest iteration of Altium Designer, and the customer input that helped drive much of the update. He also explains the company's focus on user satisfaction, and why it's so important to get feedback directly from the PCB designer and design engineer.

Andy Shaughnessy: Ben please give us a rundown on what's new in Altium Designer 18, and then we'll discuss how you all approach working with your customers.

Ben Jordan: Sure. Altium Designer 18 is perhaps one of the biggest, if not the biggest release, in quite a while for Altium. We have to be very customer-focused in our industry. I've often said, in almost a complaining sort of way, that in the EDA industry, the people doing PCB design and PCB CAD software are like dogs fighting over the bread crumbs that fall of the table.

If you consider the size of the electronics industry, and the electronics manufacturing industry, it's trillions of dollars globally. Engineers have to have tools to design all this stuff. So, you would think that the design tools are worth a lot because they're contributing to an industry that has a lot of money flowing into it and even more all the time, as more and more devices become intelligent and have to have electronics embedded within them. So it's kind of crazy that you look at the size of the market for PCB design tools, and it's absolutely minuscule by comparison. That means customers are very hard won.

I think one of the reasons why Altium Designer became a very popular tool is not just because it was lower in cost than perhaps its nearest competitors, functionality wise. But really, because it was more focused on what the user was experiencing—the user experience and the user interface. We recognized that professional PCB designers and engineers

Don't Let Your Temperatures Rise

Let ROGERS' Thermal Management Solutions *Keep You Cool*

COOLSPAN® TECA Film • 92ML™ StaCool™ Laminates • ML Series™ Laminates & Prepregs

Heat can be damaging, especially when it is not managed. That's why Rogers Corporation invested so much time and energy into creating an array of material-based thermal management solutions to keep heat rise to a minimum in printed circuits. From automotive circuits to LED modules to power supplies, ML Series laminates and prepregs effectively conduct heat away from the source, while COOLSPAN thermally & electrically conductive adhesive (TECA) materials enhance the thermal management of new and existing designs. And for that extra cooling edge, 92ML StaCool laminates feature a thermally conductive metal bottom plate to enhance the heat dissipation.

Product	Thickness	Thermal Conductivity (Z-Axis), W/mK	Thermal Impedance, C-cm2/W	Tg, C	CTE (Z-Axis), ppm/C		Dk, 1MHz	Df, 1MHz	Breakdown Voltage, kVAC	Flammability
					<Tg	>Tg				
92ML	8mils	2.0	0.52	160	22	175	5.2	0.013	>50	HF V-0

Don't let your temperatures rise. Use Rogers' thermal management solutions.

APEX EXPO IPC 2018
SAN DIEGO CONVENTION CENTER
FEBRUARY 27- MARCH 1
Visit us at Booth #500

ROGERS CORPORATION
Advanced Connectivity Solutions
www.rogerscorp.com/acs

USA - AZ, tel. +1 480-961-1382
EUROPE - BELGIUM, tel. +32 9 235 3611

are staring at this product, teasing out their thoughts and routing traces, hours and hours on end at the end of the day. It can actually be detrimental, even to their health and stress levels if the tools they're using are frustrating in any way or difficult to look at.

Ben Jordan, Altium

One of the major things we've done for Altium Designer 18 is to actually observe our customers. We noticed they stare at screens for long periods, and we had this epiphany that a dark color theme was a very good thing. Engineers do it in all their text editors, so why not make it available in the actual schematic and PCB editor as well, and make a departure from the traditional shades of gray Windows user interface colors? So, with Altium Designer 18, we've gone with a dark background theme and a high contrast menu system where the text is much easier to see and requires less effort to find what you need.

We've also done a lot of work to consolidate the menus and functions within Altium Designer, because like any mature product that is growing over time and has a lot of different elements, we did a lot of analysis both from analytics from the software's product improvement program, but also from our yearly customer surveys.

Shaughnessy: How often do you communicate with your users?

Jordan: We do survey customers every year, and our development team, the actual people writing the software, are involved on our customer forums. On the forums, users are brutally honest and vent their frustrations or sing praises depending on what they have encountered that day. Plus, we have the analytics engine. It's optional, and most people opt out of it, but a good chunk of people stay involved because they want to make the product better. Through all of that, we have been able to see areas in the software that could be improved in usability just by removing some things or by consolidating others.

Something else users will notice with Designer 18: There were dozens of modal dialogues for object properties, and each one of them was different. If you double-clicked on a pad, you'd get the pad properties dialogue and then you could edit all the different characteristics of that one pad that you'd selected, or you could double-click on a via or a track segment or a component. It was the same in schematic as well.

All of those dialogues were modal, meaning you couldn't do anything else anywhere else in the software until you'd close them or hit OK. They would obscure the work space view. Something else we noticed is more and more engineers and designers are just using a laptop, and we've built the software platform to work very well across multiple monitors. Having modal dialogues pop up in the way of the schematic or PCB editor all the time becomes frustrating when you just have one screen.

We've replaced all of those with a single, unified properties panel that allows you to stay in the editor workspace but make changes in one place. It's basically one common UI that's easy to learn for editing any kind of object properties or groups of objects, if you multiple select them or if you use a selection filter, and select a big group of items with the selection filter, which is also something we've added that we didn't have before. We had a philosophical battle over that for years but we finally did it because users want a selection filter. Think about it: P-CAD had it, but Altium Designer didn't.

The other thing, and this is pretty big news, is that Altium Designer 18 is the first version that we have removed all the FPGA design capabilities from. We had a really awesome idea and I think it was a brilliant product that was great for people like me who do a bit of everything. I'm a systems' engineer who happens to also

design PCBs, so I loved it. There were a lot of advantages to that, but only a very small percentage of customers ever did initial product development with FPGAs.

While they liked the idea of using our software and they liked the user interface and preferred it, it wasn't a realistic play for them. We've actually spent a couple of years winding that FPGA stuff down and maintaining it up to this point for the people who were using it.

Shaughnessy: You guys went through a few company names over the past few decades.

Jordan: When Nick Martin founded Altium as Protel back in 1985, he predicted that personal computers and IBM PC clones were cheap enough that the PC would become the default desktop design platform. He was a Pascal programmer at the time and he decided he could write schematic and PCB design tools that were on par, or even maybe better in some ways than the dedicated workstations that had been around up to that point, and written for the IBM PC it would be accessible. The idea was to put the technology into the hands of any professional designer or engineer.

That mission, really, has never changed. We did a lot of cool, different technology experiments at Altium. The core mission of putting technology for designing board level electronics into the hands of anybody; that's still the same with Altium Designer 18. That's why we made the decision internally to not withhold any kind of design specific technology. For example, if you want ActiveRoute, we're not going to say, "That's an extra licensing option. You have to pay for it."

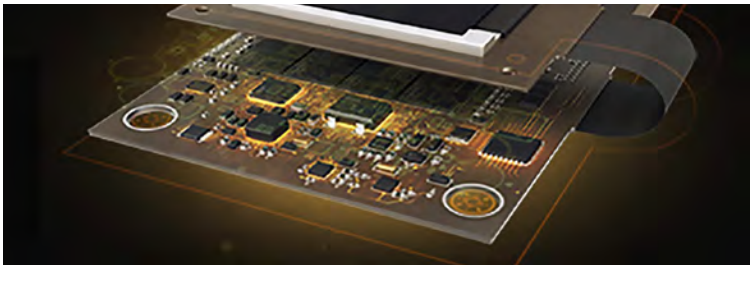
In Altium Designer 18, ActiveRoute has been enhanced further. It can now do length tuning of single-ended and differential pairs. It can do length and phase tuning and matching. It can do glossing. It has extra controls for meandering, so we're improving the routing technology for the PCB designer to be more accelerated in their design and to use the power of the PC for automation, and the graphics card, but still develop a board with user guided automation. It's human-guided automation.

The other big feature is multi-board assemblies. We actually did this back in 2006, years before anybody else. There is one other tool on the market with multi-board capability where you can manage the connections and you bring the PCBs into a 3D assembly and make sure it all fits in the enclosure. But the key difference is we're providing this to the mainstream user.

Again, we're not charging extra for this. It's just part of Altium Designer 18. Anybody on subscription or anybody getting Altium Designer will have multi-board capabilities. They can manage the connections. They can do electrical rule checking to make sure, from the schematic side, that the connector on board A is wired up exactly the same and matching to its mating counterpart on board B and that their pin numbers are correct. Everybody has been bitten by those kinds of problems when integrating multiple boards into one product. So you bring the boards into an assembly, you can manage the connections and you can synchronize pin swaps on connectors and all of that, but then you can bring it all into the 3D assembly and there's tools for aligning everything together. There's a collision checker in there as well.

Shaughnessy: That's very cool. It sounds like you all are pretty responsive, whether the users are yelling on forums or not.

Jordan: Yes, our developers are actually very active on the forums and respond directly to customers who have complaints or issues. We have a tech support group and they're really fantastic. The technical support and sales people are great. People can call them for help with anything, but when you get on the forums, if you really encounter a serious issue with the software or you have a really good idea and you share it, we have an ideas area.



Our developers actually respond directly and ask for even more information or more details about what the customer might be thinking. It's very vibrant. One of the other cool things is, not all of the developers, but quite a decent number of them, more than 10%, actually tinker around with electronics themselves. They do designs themselves. That helps a bit too that they walk a mile in the shoes of the user. They're busy developing the tool most of the time, but quite a few of them are electronics hobbyists.

Shaughnessy: So, tell me: How do you measure satisfaction?

Jordan: Well, that's pretty simple and it's complex all at the same time. At the end of the day, it's measured by subscription renewals. It's actually hard numbers. If people love the software and love what we're doing, they stay with us. They become loyal and they don't just stay with a version. There have been software packages that I have acquired in the past, personally, and I've worked for companies that bought them. They bought it and it functioned well enough. There were things about it that were maybe irritating here or there, but generally it worked pretty well, and there was no incentive or motivation to upgrade or to pay maintenance other than to get technical support.

A lot of companies have different philosophies and decision-making processes around this, but the truth is, the real proof in the pudding is if you do the right thing by the user and if you listen to them, you implement things that make their life better. It may not be done in the way they think is the best way.

Either way, we have to advance the software. The user may have some small idea of an incremental improvement, but sometimes that triggers a whole re-evaluation of how something works. Either way, we respond. Besides actual feedback verbally and with comments and word of mouth and getting new business because someone moved from one company to a new company and they recommended our tool over anything else cause that's what they love. Apart from those ways, the hard numbers are the renewals and the loyalty to subscription.

Shaughnessy: What's the farthest you all have gone to accommodate a difficult customer? You know, in terms of technical or just on the admin side.

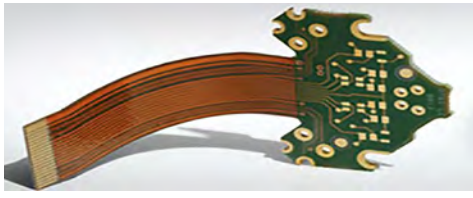
Jordan: I remember hearing from Dr. David Warren, who's on our board of directors, and he was co-founder of the company with Nick Martin. Dr. Warren shared a story with me a few months ago about this time when he was at a trade show in Silicon Valley. This is back in the Protel days.

A customer came in with an entourage, angry, red-faced, blaming Protel for a bug that caused him to produce tens of thousands of dollars' worth of scrapped boards. It cost him time and money and he was pretty upset and I don't think the fault was even with the software. If they were to dig into it, they could have found it was a user error. I've seen situations like this myself in technical support days in the past. I'd go through it with them and sure enough, it would turn out that they forgot to run the check again before they released their board.

So, Dr. Warren ended up giving him three licenses. It probably was the fault of the customer, but to make sure the situation turned around and to keep this person loyal, we were willing to give him what he needed to expand his business in terms of licenses. That's the nice thing about software. Sometimes you can make those exceptions and it doesn't hurt too much to do it.

On the other hand, just talking about supporting customers, there's a constant pull for us, and as a tool provider that largely caters to the mainstream, we're inclined to do things for large customers who have a very specific niche need. Then there's the enterprise busi-

ness model, where you consult with your top 10% of revenue generating customers, then give them exactly what they need to keep them happy. Certain competitors we have do this kind of thing and that's their business model. Nothing wrong with that, yet we have built our business on catering to smaller companies and individual users and small teams and providing what they need and doing what's best for the greater good, kind of thing. Doing something that's useful to everyone.



At the same time, we still have some larger customers who are important to us and do make unique demands from time to time. We do want to keep those people happy too. It's kind of a fine line in balancing on a knife edge for us sometimes. I think adding rigid-flex capability and multi-board are examples of that. We also know those are things that just about every designer has got to face at some point. It doesn't make sense to withhold it. Some might consider that a more extreme measure, and instead of having different tiers of licensing with different features, we instead, to address different market segments, introduce different product brands that have a different look and feel even. We introduced CircuitMaker for free for the whole maker community.

Shaughnessy: Are you seeing EEs moving into careers as designers?

Jordan: Yes, we've always had a fairly large proportion of our customer base as EEs. Our company was founded in Australia and, by and large in Australia, the electrical engineers are also PCB designers. It's such a small population in that country that the electrical engineer also laid out his own boards. The companies didn't have the budget or they didn't design enough things to have a dedicated PCB designer on full-time.

There were a few larger global companies, like the military contractors and so on that had more specific segregated workers and work flows, but by and large the market needed a tool that the electrical engineers would use. That's why we came out with the first unified design tool that kind of happened by default just because we were founded in Australia. As a result, most of our customers, for a long, long time, were EEs who also did their own PCB design. We still see that increasing, because PCB design has become so complicated with high-speed design that a lot of people eventually had the equivalent of an EE through industry training over many years of learning more about signal integrity. They would learn about fields, waves, RF, EMI and EMC. They'd end up with practically the equivalent of an EE degree anyway. But we are noticing that trend, especially with the younger designers.

Shaughnessy: I see Altium tools are in a lot of the colleges.

Jordan: Yeah, that's something Judy Warner, our director of community engagement, and Chris Potts, senior manager of marketing, sponsorship, and academia, are working on together: having a better, more formalized program for schools and for startups. We don't ever want price or cost to be a barrier for anybody who's got a bright idea and is willing to be an entrepreneur and launch a project on Indiegogo or Kickstarter or something like that. We have those startup programs to get the right tools into their hands right at the beginning so they can save time. We know they'll become good customers if they're successful anyway. We see ourselves as a partner in helping them get something off the ground. In the '80s, you could get OrCAD in every electrical engineering department across the country. We're still trying to catch up to that, but we're getting there.

Shaughnessy: Thanks for your time, Ben. I'll see you on the road, I'm sure.

Jordan: Thank you, Andy. **DESIGN007**

Customer Support: What Do PCB Designers Really Want?

Tim's Takeaways

Feature Column by Tim Haag, CONSULTANT

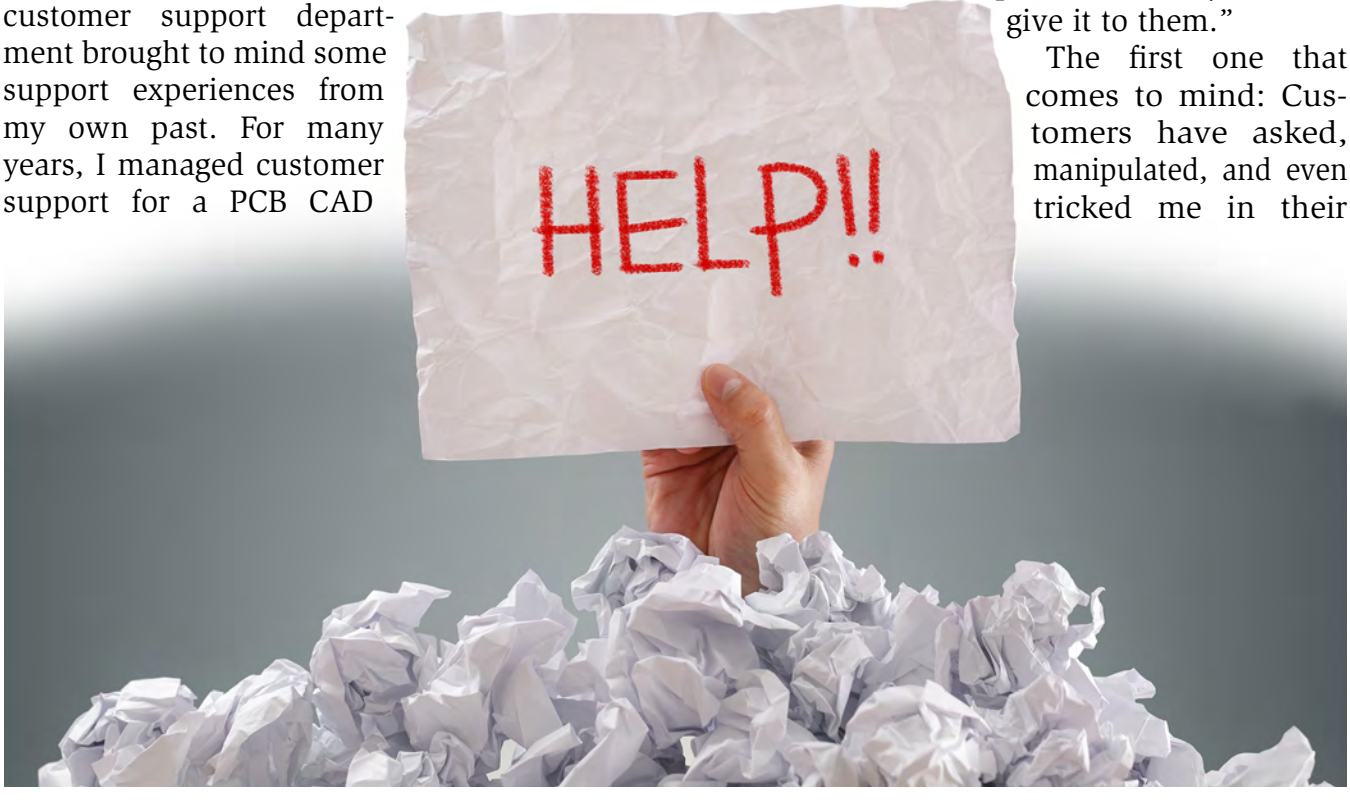
After a grueling month of research into the best medical insurance to invest in for 2018, we finally closed the deal back in December. I was therefore shocked to see new insurance cards arrive in the mail yesterday with a different plan name. As you can expect, I jumped on the phone and dialed my insurance company's customer support. Fortunately, the woman talking to me was a consummate customer support professional, and took care of my questions right away. As it turns out, someone had messed up and sent out incorrect cards to pre-existing customers for some unexplainable reason. Happily, this whole problem ended up being nothing that my paper shredder couldn't handle, and I was glad to emerge from this fresh hell so quickly.

My experience with the insurance company's customer support department brought to mind some support experiences from my own past. For many years, I managed customer support for a PCB CAD software company, and I have seen my fair share of different support scenarios. Although some of my time with customers could be rightfully categorized as a nightmare experience, it was far more common for me to work with customers who were a pleasure to spend time with. Through it all, I have seen what customers want and what issues they complain about most in their PCB design software. Here are some of those items; perhaps you can identify with a few of these.

The Two Meanings of "How can I help you?"

First, let's throw a leash around the elephant in the room and pull him out. That's my not-too-subtle way of saying, "Here are some things that designers want, but we in the support business just can't give it to them."

The first one that comes to mind: Customers have asked, manipulated, and even tricked me in their



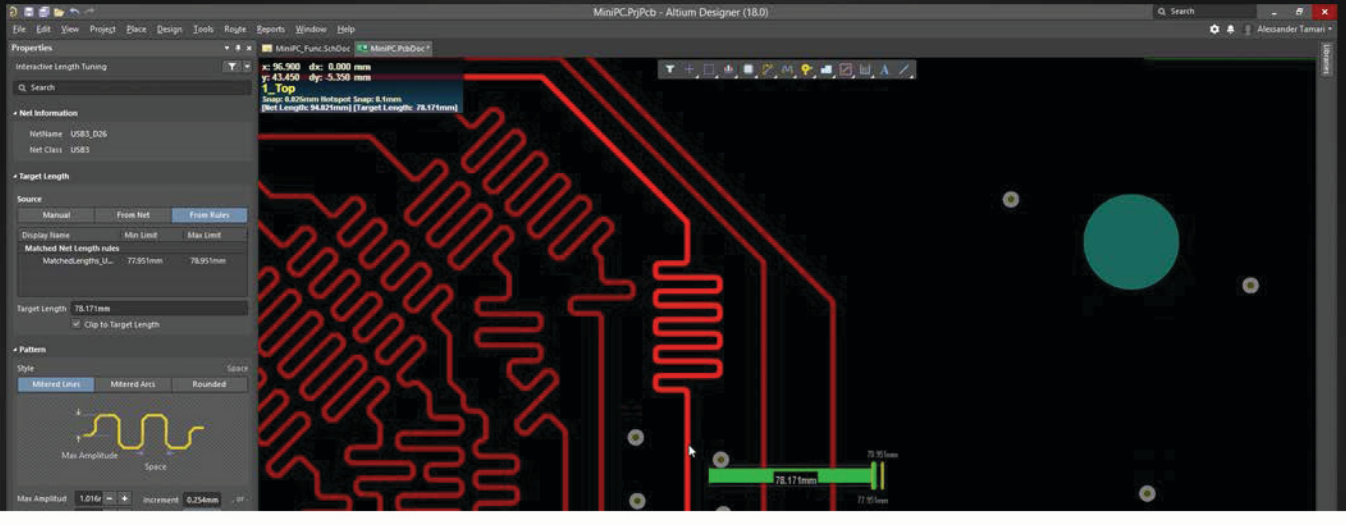
Altium®

High Performance Made Simple

ALTIUM DESIGNER18

The Easy, Powerful, Modern PCB Design Solution

AVAILABLE NOW



For more information please visit:

www.altium.com/altium-designer

attempts to get free software. Here's the deal: When you buy software, you are entitled to receive what you bought, but not more. That doesn't mean that the sales team might work out an arrangement with you, but as your support technician, I can't do that. I can only help you with the software that you are licensed for. It's sort of like buying a bunch of bananas while grocery shopping, and then insisting that the store should give you a couple of free apples too, because you are one of their best customers. It just isn't going to happen.

Another thing that I occasionally encountered were customers who would insist that their software should function more like Brand X software. Here's the 411 on that: The software that the customer is using isn't Brand X software and it never is going to be Brand X software. Although the tools may go through enhancements and changes to incorporate some of the functionality of Brand X software, the software is never going to completely change. The customer in this situation needs to learn to use the software the way it is, or switch over to Brand X software.

The last challenge for me as a customer support technician was to help a customer who wanted me to do their work for them.

The last challenge for me as a customer support technician was to help a customer who wanted me to do their work for them. Answering questions, filing bug reports, providing workarounds and even conducting limited training is all part of the normal operations of a CAD customer support technician. We must draw the line somewhere, though, and when a customer is expecting us to do their work for them we have to say no. There are some things we just can't do.

Now that the elephant has been tamed and led out of the room, let's talk about common customer support themes that I have seen. Although the specific support requests are extremely diverse, there are some common themes that wind through those requests. Here are the more popular ones that I have noticed over the years:

- Customers want the bells and whistles of other software: Although most customers realize that their software is not Brand X software, they still want the features that they were used to on other tools or that they have seen demos of. Here is where a strong internal relationship between support and marketing is very crucial to software companies, in order to hear users' requests and funnel them into product enhancements.
- Often customer questions are more design-related than software-support related: Sometimes customers need basic design help and are confusing that with tool support. Although I can't design their board for them, I have often offered ideas that have helped to get a customer moving again in the right direction.
- Sometimes the customer needs training: Although I have provided many instances of limited training during a support call, there are also times where I have had to suggest that the customer invest in some official training.
- Customers need to know that they have someone that they can go to: I can't tell you how many times I have fielded calls from customers who just needed a sounding board. There have been so many calls where I did nothing more helpful than to just say "hello" when the phone rang. These customers would talk themselves through their problem, and then identify what they needed to do. It may sound like a waste of time for a support technician, but I disagree. I believe that being there for my customers, even as a simple sounding board, was one of my principle responsibilities.

- Vulnerability, honesty, and transparency: Customers need support technicians who they can trust completely, so that they feel comfortable when asking for help. If you've ever gone to someone for help and instead got your trust abused by an obnoxious response, you would probably be hesitant to go to that person again. The same principle applies in customer support. As support technicians it is our responsibility to instill trust in our customers by being vulnerable, honest, and transparent so that they will trust us with their support problems.

Having worked in customer support for many years, I couldn't begin to tell you all the different, specific requests that users have asked for in their PCB design tools. Many of these have since been incorporated as standard functionality in most tools used today. There have been some though that have stood out from the rest; here are a few of those that I recall:

- Graphical user interface (GUI) enhancements: This has been a constant request over the years, and as evidenced by today's PCB design tools, many of these improvements have already been made. Customers will always want improvement in these areas, to match current operating systems and to support the ever-increasing amount of functionality.
- Menus and commands that aren't confusing: As a tool set grows, the amount of commands will grow with it. Sometimes these new enhancements are stuffed into menus or buttons that don't make sense to the users. A common request that I would get was to reduce the amount of different commands, and locate those commands in menus and pallets that were intuitively part of the work flow.
- Comprehensive and logical documentation: Users typically want to solve their own problems if they can, and to do so requires documentation that is written from the user's perspective. This includes step-by-step procedures to follow along with clear and concise images as well as appropriate links to other steps.
- A more comprehensive constraint manager: It has been a pleasure for me to watch all the different CAD tools expand and enhance their rules and constraint management over the years. This has evolved from a single drop-down menu with a few selections in it from many years ago, to advanced spreadsheet-type constraint managers today. Yet customers continue to ask for additional enhancements and improvements to make entering these rules and constraints more intuitive. CAD companies have responded with a variety of wizards and other enhancements for their constraint managers, and more will come as design rules become more complex.
- High-speed design enhancements: Along with enhanced constraints, users have also asked for continual improvements in their ability to work with high-speed designs. An example of this that I have seen is the request to work with high speed rules based in time, not length. Additionally, new routing capabilities have also been added to work with high speed rules and constraints.

The need for advanced PCB design capabilities will continue to grow to satisfy the requirements of new and emerging design technologies. And to meet these design needs, CAD tools will also have to continue to grow. This growth is a healthy part of our industry, and it is exciting to be part of it. Fortunately, PCB design CAD software companies are staffed with professional support personnel that are ready to work with you and listen to your requests for help and your recommendations. They're your support staff, make sure to use them so that they know what you really want. **DESIGN007**

Tim Haag is a consultant based in Portland, Oregon.

PCB007 Highlights

IPC-1401 Listed Amongst Top 10 Chinese Corporate Social Responsibility Standards ▶

IPC-1401, Supply Chain Social Responsibility Management System Guidance was listed as a top-10 standard in 2017, for Chinese Corporate Social Responsibility (CSR).

Punching Out! Survey on State of the North American PCB M&A Market ▶

Recently, my firm surveyed about 20 PCB manufacturers in North America with an estimated greater than $10 million in revenue. Quite a few replied, and we have spoken with many others throughout the year, which gives us a good view on the state of the PCB market.

The Right Approach: Culture Shift is Key to Quality Improvement ▶

Any major initiative, whether implementing ISO, lean manufacturing or introducing a new product, requires culture change. How this change is managed will be the difference between success or failure of the project. This column will offer some fundamental elements that will help navigate your next major implementation by shifting the culture.

All About Flex: Copper Grain Direction ▶

Many materials have different characteristics depending upon the orientation of the material. For example, woven textile materials have a warp and a weft direction. The warp direction is the longitude direction and the weft is the transverse.

CES 2018, Augmented Reality and Much More ▶

The actual CES show is spread across many locations in Las Vegas. The main exhibit halls are at or near to the Las Vegas Convention Center with three buildings, two floors each, all filled with hundreds of booths.

It's Only Common Sense: 6 Ways to Guarantee a Great 2018 ▶

Here we are again. The beginning of a brand-spanking new year—2018! Who would have thought we would make it this long? But we did, and the North American PCB business is still alive and kicking.

CES 2018 Showstoppers: LaunchIt and Press Event ▶

"ShowStoppers LaunchIt is about giving innovative entrepreneurs a shot at getting the attention of angel investors on the lookout for innovation and new ventures. It's also about gaining additional visibility with other industry influencers and dealmakers, as well as with the press who are always looking for the 'what's new' story at CES."

In Terms of Experience, a 10,000-foot View of China ▶

In the past 30+ years of PCB manufacturing in China, you would be hard-pressed to find someone more connected to the pulse of the Asian market than Gene Weiner. Barry Matties met with the industry veteran at HKPCA to get his take on the show, the current and future market conditions of China, and any effect the new U.S. administration might have on trade relations going forward.

It's a Long, Winding and Exciting Road for Automotive Electronics ▶

It was truly a delight to talk with Alun Morgan at productronica this year. He may be the most enthusiastic person in the field of electronics that I have ever met, as you will certainly understand as you read this interview.

Powerful Schematic Capture
and PCB Design Software

PULSONIX

Are you looking for cost effective **PCB Design Software?**

Look no further, **Pulsonix** is the answer.

- Fully Integrated Schematic Capture, PCB Design and 3D View
- Very easy to learn and use
- Seamlessly import design data from your existing CAD tool
- High-Speed board design with advanced constraint manager
- Reverse Engineer your designs from Gerber
- Technologies supported: blind/buried vias, Microvias (HDI)
- Support for Flexi-Rigid boards
- Optional Spice A/D mixed mode simulation

Download a **FREE** trial from

pulsonix.com

Contact us today and see how Pulsonix can work for you:

e: **sales@pulsonix.com**

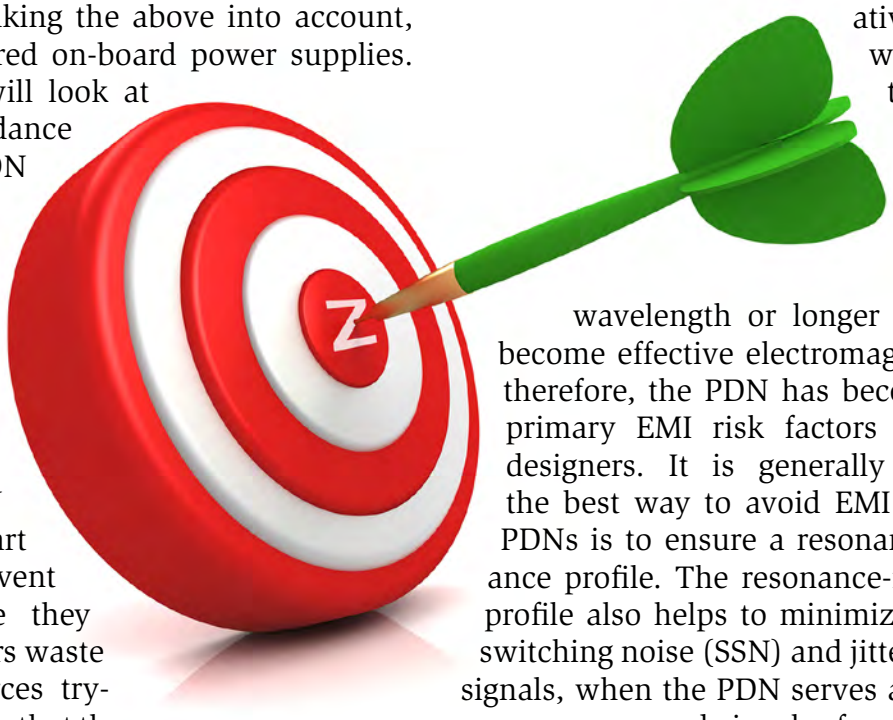
The Target Impedance Approach to PDN Design

Beyond Design

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

Today's high-performance processors employ low DC voltages with high transient currents and high clock frequencies to minimize the power consumption and hence the amount of heat dissipated. A typical high-speed design contains ten or more individual power supplies.

And unfortunately, the lower core voltages, higher currents and faster edge rates all impact on the power distribution network (PDN) design as well as signal integrity. The goal of robust PDN planning is to design a stable power source, taking the above into account, for all the required on-board power supplies. This month, I will look at the target impedance approach to PDN design.



Before you worry (or not) about post-layout PDN DC drop analysis, you first need to design an effective PDN pre-layout. Smart designers prevent problems before they arise, while others waste time and resources trying to fix the mess that they inadvertently created due to their lack of due diligence. Engineers and PCB designers need to visualize and understand how and where the currents flow. When I analyze a multilayer PCB, I immediately interpret the entire current loop including the return path by highlighting adjacent signal and associated plane layers. If you need to push a lot of current from one point to another, it is obvious where the hot spots will be. Increasing the plane copper thickness is a good solution, for DC and low frequency, but has little impact at high frequencies due to the skin effect. However, effective placement, the use of minimal antipads, and wide uninterrupted copper pours alleviates the issue before it arises.

The high clock frequencies and signal rates employed today push more of the physical board features into the red zone where their relative dimensions, with respect to the wavelength of the clock, approach or exceed the quarter-wave limit. Quarter-wavelength or longer structures may become effective electromagnetic radiators; therefore, the PDN has become one of the primary EMI risk factors for high-speed designers. It is generally accepted that the best way to avoid EMI radiation from PDNs is to ensure a resonance-free impedance profile. The resonance-free impedance profile also helps to minimize simultaneous switching noise (SSN) and jitter of high-speed signals, when the PDN serves as both a stable power source and signal reference.

In the early days of digital electronics, there was no such thing as PDN design. Instead, the focus was on the maximum number of 100nF (and the odd 47pF) capacitors that could be placed close to the ICs power pins. Connecting a capacitor by a thick trace to a 60 mil DIP power pad was easy—but try doing that with

COAST TO COAST CIRCUITS, INC.

TECHNOLOGY FROM COAST TO COAST

For over 40 years, we have been producing advanced solutions for our customers in many industries across the globe including Defense, Medical, RF/Microwave, Semiconductor and Down Hole Exploration among others. How can we add value to your next project?

COAST to COAST CIRCUITS

LEARN MORE | CHECK OUT OUR CAPABILITIES

Two Facilities. One Integrated Solution.

NEW YORK OPERATION
585.254.2980
sales@c3circuits.com

CALIFORNIA OPERATION
714.898.4901
sales@c3circuits.com

www.c3circuits.com

a dense BGA package. This approach was sufficient as long as the active devices did not generate significant transient noise spectrum above the series resonance frequency (SRF) of the decoupling capacitors. However, higher system speeds later created the need to design the PDN systematically, to meet tighter impedance requirements.

Designers often procrastinate about whether it is better to have a thick trace routed directly from the decoupling capacitor (decap) to the BGA power pad (Figure 1), which eliminates the power via pair and reduces the loop area. But does this reduce the inductance? This trace has to be 25 mils wide and 1 mil thick to match a via of 8 mils hole diameter with a 1 mil barrel plating thickness (πd). The top trace has an inductance of 460pH whereas the power via pair and plane combination has a total loop inductance of 324pH. Therefore, there is 30% less inductance, in this case, by directly connecting the decap to the planes (thermals should not be used on the via plane connections). Plus, it is difficult, if not impossible, to route a 25mil trace to a BGA pad–particularly the internal pads. 10 mil is more the norm, which would dramatically increase the trace inductance. BGA power/ground vias go directly to the plane and so should the decap power/ground vias.

Also with high layer count stackups, the decaps should be placed around the perimeter of the IC on the same side of the board and routed directly to the planes. Placing decaps directly under the IC is also an obsession of the past and may increase inductance and impedes fanout and signal routing. Please see my previous column "PDN – Decoupling Capacitor Placement" for further details [1].

Target impedance is the combination of the

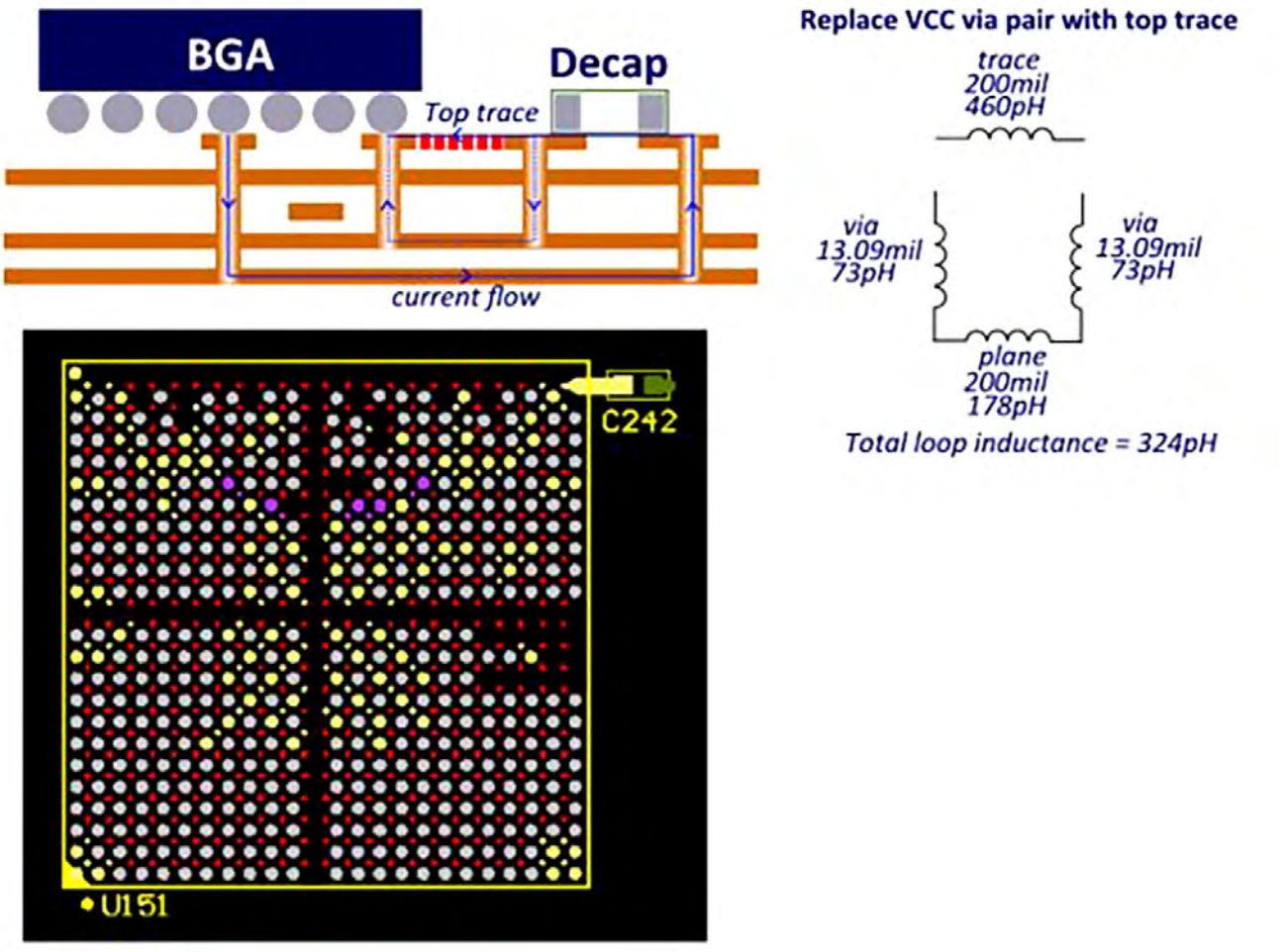


Figure 1: Trace on the top side eliminates the two vias but increases inductance.

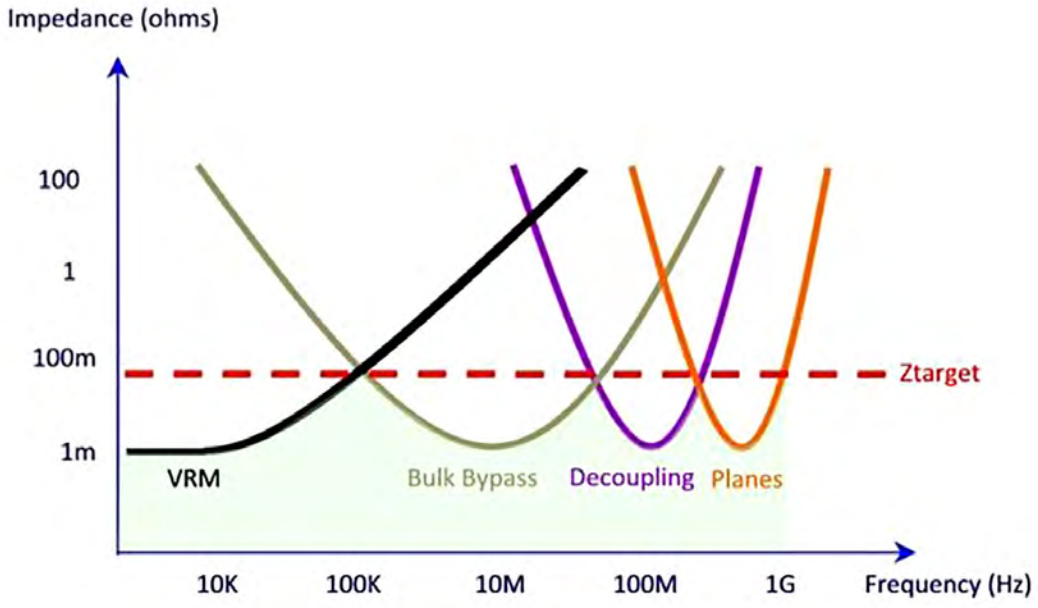


Figure 2: Fundamental target impedance, VRM, capacitors and plane profiles.

worst-case transient current and the voltage noise specification which act together to set the maximum allowable PDN impedance with assured performance. The target impedance (Ztarget) is determined based on the maximum voltage rail noise (ΔVnoise)–VDD by the ripple voltage–and the worst case transient current (Itransient)–maximum current by the duty cycle.

$$Ztarget = \frac{\Delta Vnoise}{Itransient} = \frac{VDD\ x\ 5\%}{Imax\ x\ 50\%}$$

Ideally, the effective impedance of the PDN should be kept below the target impedance up to the maximum required bandwidth as in Figure 2. However, if the impedance is too far below the target, then this implies that the PDN has been overdesigned which unnecessarily increases costs with little added benefit. If your company intends building hundreds of thousands of assemblies, then the potential cost saving can be quite significant. Analyzing the PDN ensures best performance at the most cost-effective price.

Target impedance is the most crucial metric when evaluating PDN performance. The further the PDN impedance is above the target impedance, the greater the risk of intermittent operation or even complete product failure.

In practice, accurately calculating the transient currents and the precise requirements for the target impedance can be challenging. Since we typically do not know the transient noise current excitation very accurately, it is customary instead to design the PDN to meet the required impedance profile. Also, it seems that the current portion of the target impedance equation varies from point-to-point, on the board, depending on a host of intricate relationships. One must always apply engineering judgment in translating the information available into the requirements for a cost-effective PDN design.

Fortunately, for double data rate (DDR) memory, the power supply is only utilized for the memory ICs and the memory drivers/receivers of the processor. Transient currents from circuits on different clock domains are statistically independent. Meaning there is no interaction between PDN current (except for any coupled noise) and therefore the DDR supply is isolated from the noise of other supplies assuming good design practice. Therefore, the target impedance calculated for the DDR PDN is sufficiently accurate.

However, within a single IC, there may be several circuit blocks that draw current from the same power rail. The same concept applies here: if the circuit blocks are independent

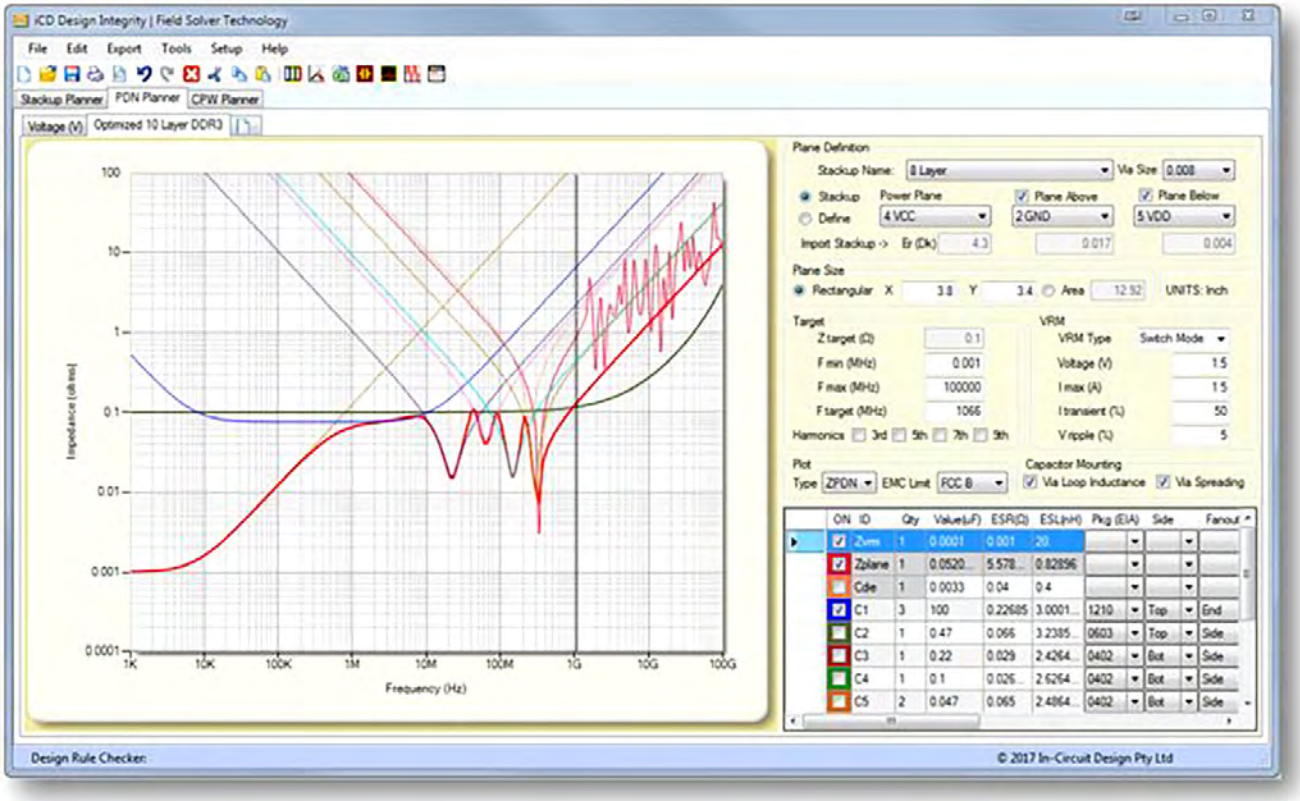


Figure 3: Optimized PDN profile (simulated in iCD Design Integrity).

(which is most likely the case if they are on different clock domains) then the RMS value of the transient currents can be combined. Using this concept, the target impedance for a single IC can be calculated from knowledge of the circuit blocks drawing current from the voltage rail in question. And at the PCB level, transient current drawn by several chips can be calculated statistically. This makes the target impedance a function of frequency. The circuit designer can combine the target impedances, for several chips on the same PCB power rail, by adding linearly at DC and low frequencies and statistically at higher frequencies.

Generally, all circuits on the PCB will draw DC current at the same time and so the voltage regulator module (VRM) must be designed to accommodate the potential surges. But as the frequency approaches the 1GHz band, the probability of drawing transient currents in phase and at the same frequency greatly diminishes. As frequency goes up, peak currents become spatially isolated. The target impedance at the different chip locations on the PCB should reflect this in different frequency bands.

The PDN profile should be optimized (Figure 3) such that the effective impedance, of the combined VRM, bypass and decoupling capacitors and plane resonance, falls just below the estimated target impedance, of the PDN, up to the maximum required bandwidth.

Key Points

- The goal of robust PDN Planning is to design a stable power source, taking into account the worst-case transient conditions.
- Smart designers prevent problems before they arise; others waste time and resources trying to fix problems.
- Engineers and PCB designers need to visualize and understand how and where the currents flow.
- Increasing the plane copper thickness is a good solution to current hot spots, for DC and low frequency, but has little impact at high frequencies due to the skin effect.
- Quarter-wavelength or longer structures may become effective electromagnetic radiators.

- The best way to avoid EMI radiation from PDNs is to ensure a resonance-free impedance profile.
- BGA power/ground vias go directly to the plane and so should the decap power/ground vias, which reduce loop inductance by ~30%.
- Decaps should be placed around the perimeter of the IC on the same side of the board and routed directly to the planes.
- The effective impedance of the PDN should be kept below the target impedance up to the maximum required bandwidth.
- Target impedance is the most crucial metric when evaluating PDN performance.
- The circuit designer can combine the target impedances for several chips on the same PCB power rail by adding linearly at DC and low frequencies and statistically at higher frequencies.
- As frequency goes up, peak currents become spatially isolated.

References

1. Beyond Design: PDN: Decoupling Capacitor Placement, Power Distribution Network Planning, Learning the Curve.

2. SI List forum: Steve Weir, Larry Smith.

3. Comparison of PDN Design Methods, Istvan Novak, DesignCon 2006.

4. Principles of Power Integrity for PDN Design, Larry Smith and Eric Bogatin.

5. High-Speed Digital Design, Howard Johnson.

6. Target image: Modified from Melanie Boylan, Stomp Social Media Training.

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 from www.icd.com.au. To contact Olney, or read past columns, click here.

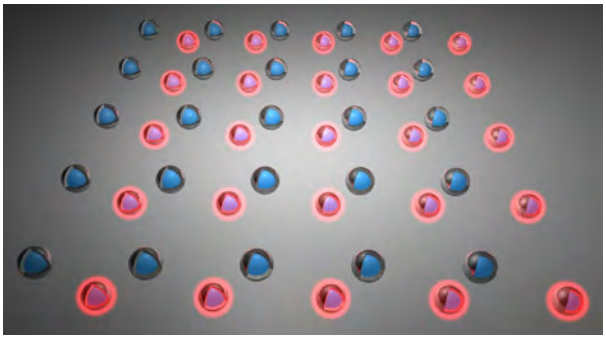
Manipulating Light with Nanoscale Objects

Scientists at The University of New Mexico studying the field of nanophotonics are developing new perspectives never seen before through their research. In turn, the understanding of these theoretical concepts is enabling physical scientists to create more efficient nanostructures.

The research, says Assistant Professor Alejandro Manjavacas, in the Department of Physics and Astronomy at The University of New Mexico in a paper titled "Hybridization of Lattice Resonances," investigates how periodic arrays of nanospheres or atoms interact with light. These systems are made by repeating a unit cell periodically, much like a chessboard is made by repeating two differently colored squares in a pattern.

As part of the research, Manjavacas and his team composed by Sebastian Baur, a visiting graduate student from Germany, and Stephen Sanders, a graduate student in Physics and Astronomy, investigated the optical properties of periodic arrays of plasmonic nanoparticles with multi-particle unit cells. Specifically, they sought to understand how the geometry of complex arrangements of plasmonic nanostructures can be harnessed to control their optical responses.

Manjavacas and his team also explored systems with three- and four-particle unit cells, like a chessboard with three or four different kinds of colored squares, and showed that they can be designed to support resonances with complex response patterns in which different groups of particles in the unit cell can be selectively excited.



The Rough Journey to Revelation

The Pulse

by Martyn Gaudion, POLAR INSTRUMENTS

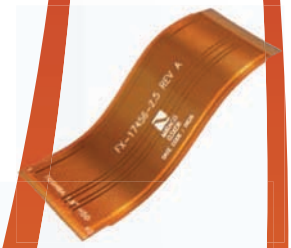
Signal integrity is something of a journey, and as you travel on the road to ever higher speeds, some characteristics of the PCB itself become more influential, and others less so. I am daily reminded of these changing circumstances, as Polar is based on a small island with a very large tidal range, about 33 feet (10 meters) from low to high tide. Seafaring types need to be aware that at high tide, there is a comfortable 25 feet or so beneath the keel, but at low tide, your hull may be damaged, if you're not aground on dry land. Some less experienced yachtsmen may even be concerned with delamination, but not in a PCB way.

Several years ago, an unsuspecting French yachtsman moored his yacht to the railings of the local harbour. For a very nervous full tide cycle, he awaited to see if the cleats would pull out of the glass fiber hull—a different type of delam, certainly. Fortunately, the glass held and the yacht sailed out on the next high tide.

As circumstances change, interesting solutions to problems are often revealed with the changing situation. A yachtsman at high tide isn't too worried about whether the seabed is rough or smooth, but at low tide, the concern about a sandy or rocky seabed is altogether different. With PCBs, the move to low-loss laminates exposes a similar situation.

In higher-loss laminate base materials in most popular applications, the bulk of the loss occurs in the laminate itself, and this is especially the case if the track geometries are on the generous side. So, you can choose to brush concerns about surface roughness to one side.

Engineering And Providing Balanced Interconnect Solutions



MIRACO

Miraco, Inc.
Manchester, NH
miracoinc.com
603-665-9449

...connecting technology

A Practical Example

I have taken a range of scenarios to show the interplay between loss drivers and how the key drivers of loss will gain and lose (relative influence) as you vary base material and geometries. While we consider the following figures, I would like you to focus on the yellow (copper loss with roughness) and the green (dielectric loss) series in the graphs and consider how each property's influence waxes and wanes, depending on the geometry and material selection. I have used generic "good enough" values to illustrate the point.

In Figure 1, the generous line width keeps the copper losses low, and even with a fairly rough surface, the copper loss and roughness loss combined are less than half of the contributors to total attenuation.

In Figure 2, the line width is slashed to 3 mils and the heights adjusted to maintain a 50-ohm environment. The effects of reduced copper are clear even with a high-loss basic FR-4 material; cutting the line width to 3 mils pushes the losses, with roughness included, to significantly more than the dielectric loss.

Figure 3 takes the same 3 mil 50-ohm line but substitutes a mid-loss material with a loss tangent of 0.007. Now copper losses are lofted to the vast majority of attenuation and when seeking better performance with this scenario it starts to pay to look at copper profiles. To keep this in perspective, the loss is much lower than with the high-loss material, but any variation is now increasingly likely to come from the copper rather than variations in the base material.

Taking this example to the extreme, which is a great way to highlight these kind of scenarios, I have inserted a very low-loss material into Figure 4 with a loss tangent of 0.0021. Here you can see the drivers of loss are entirely in the copper; there is no more gain to be made from better laminate, other than having smoother copper.

Let's think a little more about Figure 4. Say, for example, you purchase two identical PCBs from different suppliers, both made with the best base material, and the insertion loss of one board is significantly different from that of the other. Where do you look first? A glance at these graphs leads you to suspect that the

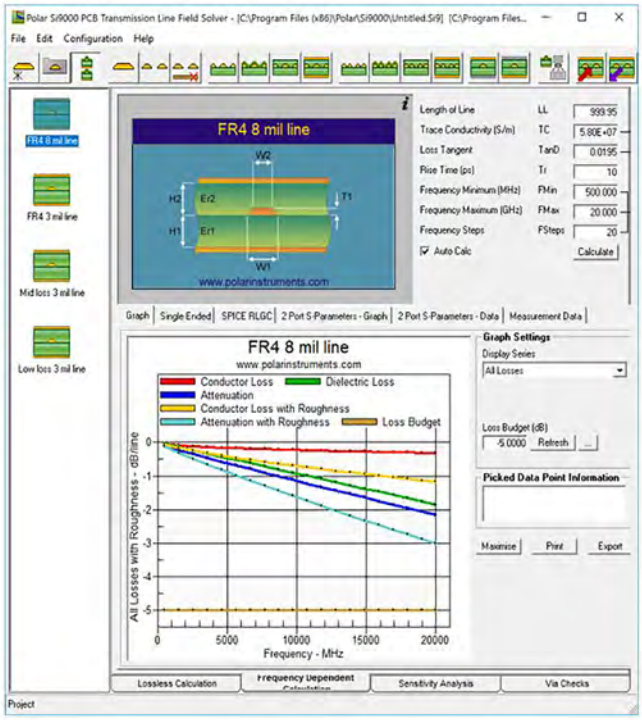


Figure 1: Generic FR-4 with 8 mil line width and heights set to achieve a 50-ohm line.

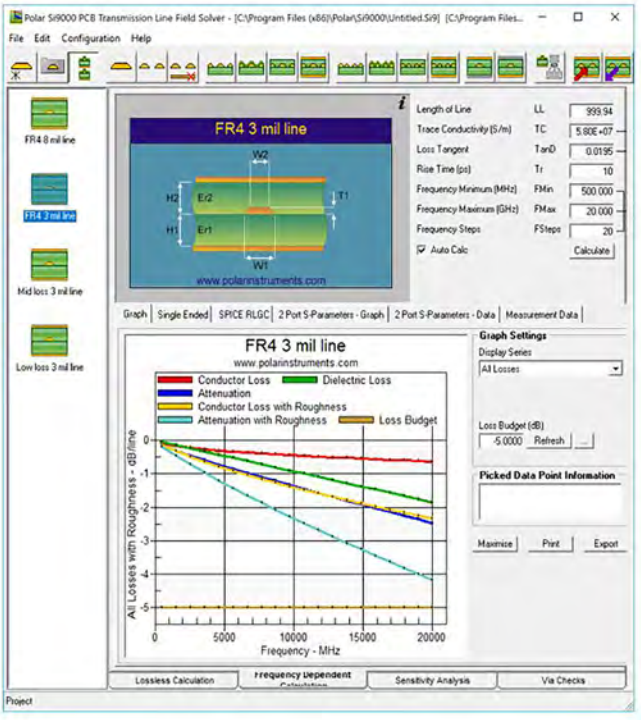


Figure 2: The same characteristic impedance, and same material, with line width reduced to 3 mils.

copper is the largest potential driver of loss. This leads me on to the mythical concept of the "perfect material library."

The Mythical Perfect Material Library

"If I could specify every material precisely and draw the material from a library, I would never have to rely on the fabricator not to mess with my designs," says the PCB designer. Whilst there is an element of truth to this, it overlooks the fact that PCB manufacturing is not an assembly process where raw materials are simply assembled. In order to make a robust, reliable, manufacturable PCB, the fabricator has to process the raw materials. Copper layers are an excellent example of how manufacturing processes get in the way of the designer's dream of "locking down the spec so tight this fab will have to make it my way."

Let's start with foils, which can be made with a variety of profiles. These will either be supplied to the fabricator or the base material supplier, who will laminate the foils to the base material to create a core. The laminate side foil stays much as received and has as little chemical treatment as possible to ensure bonding. What happens to the outside of the foil is partially in the hands of the fabricator and partially dictated by the stackup.

For a start, any outer layer which is drilled and requires plating will have plating copper applied to the whole surface; this is also true for blind or buried vias. So for HDI and sequentially laminated boards, designers should be aware that any layers with drill ends, laser or conventional, will have additional copper plating and the plating roughness will be dependent upon the fabricator's process. You need to consider this for low-loss designs—the copper foil you specify may not end up on the critical surface because of the stackup design.

This is not anyone's fault. It is simply an inherent necessity of today's PCB fabrication process. Likewise, the non-drilled end layers will also be treated to promote adhesion. The chemistry suppliers are well aware of the drive for smoother copper, and generally the trend is towards less aggressive pre-treatment for bonding, but bonding really smooth surfaces, whilst possible, is expensive and still in its early days. PCB fabricators have long memories and are

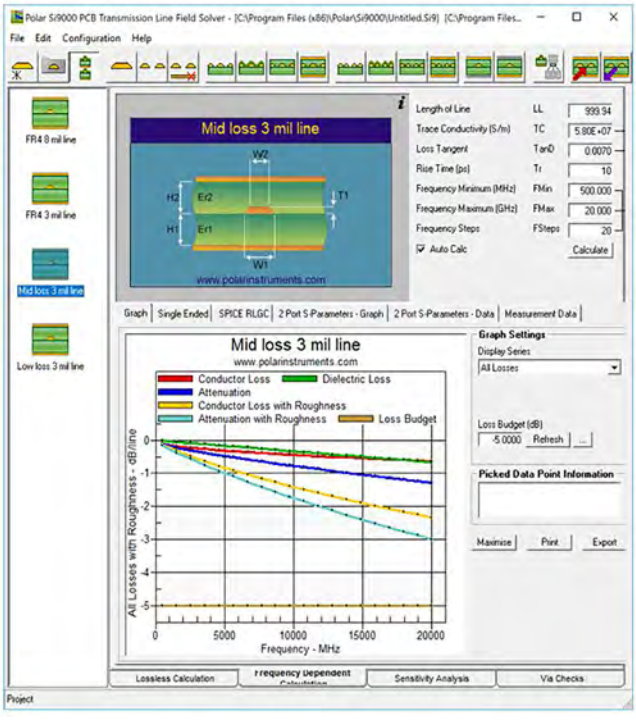


Figure 3: Showing 3 mil lines again, but with a mid-loss material.

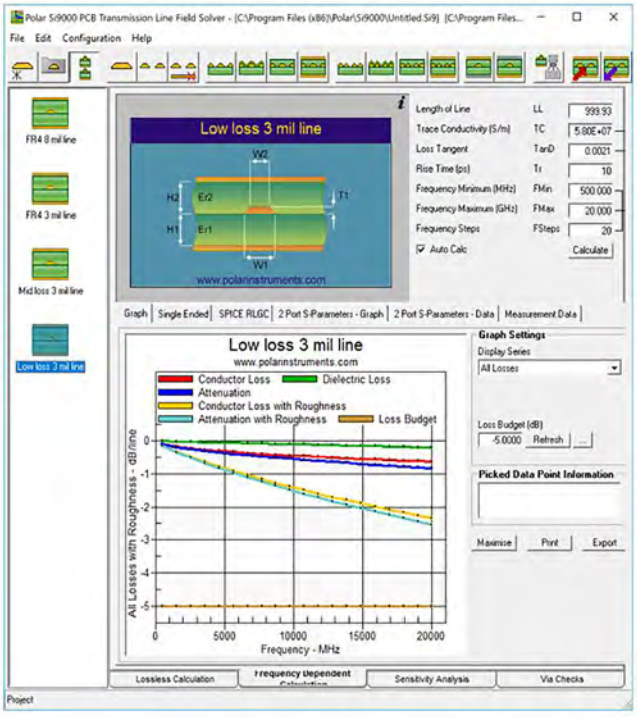


Figure 4: Taken to the extreme, with a low-loss material.

haunted by past nightmares of black-pad problems with oxidation treatments, and so are rightly cautious about adopting new processes for bonding without a concrete demand.

Key points: Smoother copper and lower loss laminates are critical to lower insertion loss. Where you choose to focus primarily depends on the geometries you are working with and the type of material you have chosen to start with. When working with smooth copper, bear in mind that the copper you specify does not appear on every layer, as the fabricator has to plate up the layers with drill ends to make the vias conductive! For now, PCB fabrication is still a subtractive process in the vast majority of cases.

As always, if you are moving into a new area of expertise as a designer, you should be comfortable in discussing the critical areas of your design with your fabricator, who will be happy to help and likely to save you both time and money. **DESIGN007**

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

Your Gadget's Next Power Supply? Your Body

Searching for a power outlet may soon become a thing of the past.

Instead, devices will receive electricity from a small metallic tab that, when attached to the body, is capable of generating electricity from bending a finger and other simple movements.

That's the idea behind a collaborative research project led by University at Buffalo and Institute of Semiconductors (IoP) at Chinese Academy of Science (CAS). The tab–a triboelectric nanogenerator–is described in a study published online January 31 in the journal Nano Energy.

"No one likes being tethered to a power outlet or lugging around a portable charger. The human body is an abundant source of energy. We thought: 'Why not harness it to produce our own power?'" says lead author Qiaoqiang Gan, PhD, associate professor of electrical engineering in UB's School of Engineering and Applied Sciences.

Researchers have proposed numerous nanogenerators that utilize the triboelectric effect; however, most are difficult to manufacture (requiring complex lithography) or are not cost effective. The tab that the UB and CAS team are developing addresses both of those concerns.

It consists of two thin layers of gold, with polydimethylsiloxane (PDMS) sandwiched in between.

The study describes a small tab (1.5 centimeters long, by 1 centimeter wide). It delivered a maximum voltage of 124 volts, a maximum current of 10 microamps and a maximum power density of 0.22 millwatts per square centimeter. That's not enough to quickly charge a smartphone; however, it lit 48 red LED lights simultaneously.

Co-authors of the study include Huamin Chen at IoP and CAS; and Nan Zhang, a PhD student at UB.

Because the tab is easily fabricated, Zhang is leading a team of UB undergraduates which is tasked with improving the tab's performance. The team plans to use larger pieces of gold, which when stretched and folded together are expected to deliver even more electricity.

Superior Quality Begins Here.

Since we first began in 1985, delivering the highest quality products possible has been our first priority. Our quality control systems are certified and we renew our commitment to quality each and every day.

Delivering quality products on time, combined with timely and accurate communication from our customer service team is how we create lasting relationships with our customers.

Take a look at our quality »

COMMERCIAL • MILITARY • MEDICAL • BAREBOARD

U.S. Circuit, Incorporated
2071 Wineridge Place
Escondido, CA 92029
+1 (760) 489-1413
www.uscircuit.com

Design for Manufacture or Design for Application?

Consider This

by John Talbot, TRAMONTO CIRCUITS

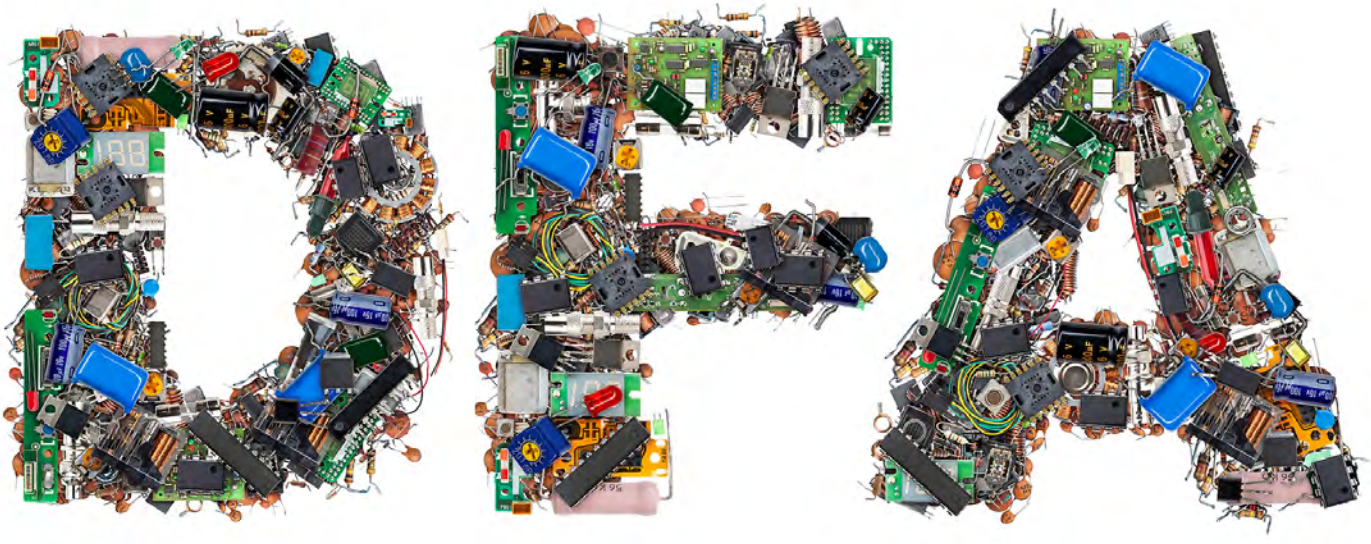
Please welcome our newest columnist, John Talbot. John is president of Tramonto Circuits, a manufacturer of rigid PCBs and flexible circuits.

Design for manufacture (DFM) has been a common topic for circuit designers for a long time. Before manufacturers started publishing their capabilities on websites, a trip would be scheduled to the supplier so that designers could talk with the operators at each fabrication station and the production engineers to better understand their supplier's capabilities and limitations. That, it was surmised, would make designs better from the customer and save time during the review process.

The customer would learn their suppliers' limitations on things like minimum plated hole sizes, minimum annular ring, minimum line width/space, and more. On the surface, this seems like a very valuable thing to do. It improved the communication between customer and supplier and theoretically improved the quality of the circuits and design cycle time as well. However, it was the customer who always compromised!

Of course, the circuit must be manufacturable. But is it best for the customer to compromise the design to meet their supplier's limitations? Remember, the customer will assemble and insert these circuits into their coolest new products with the intent of selling thousands or millions of them. With that fact in mind, it would make more sense for the supplier to compromise to meet the application's requirements to give it the best chance of success. That, of course, is difficult for a circuit manufacturer.

It is far easier for a circuit manufacturer to adhere to a strict set of capabilities and design limitations so that their factory runs



IT-988G SE / IT-988G

COME SEE US AT IPC BOOTH #133

IT-988G

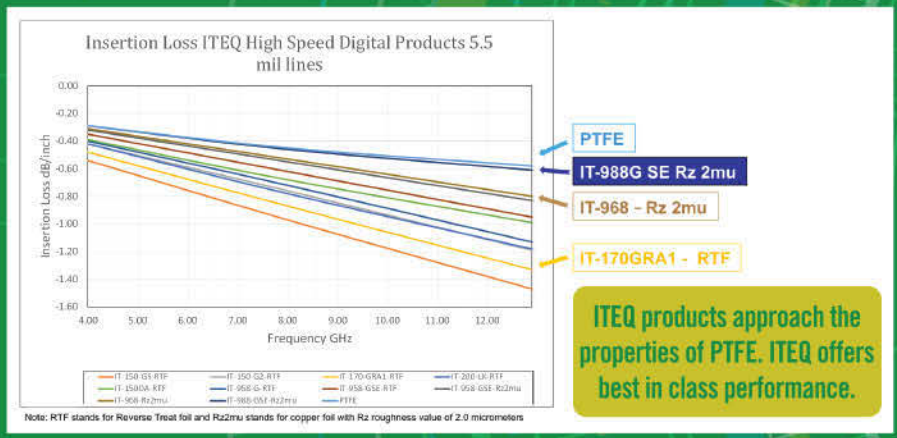
- Halogen Free
- Tg = 190° C
- Td = 405° C
- Dk = 3.46 @10GHz*
- Df = 0.0025 @10GHz*
- Available with low Dk Glass - IT-988G SE

IT-988G SE

- Halogen Free
- Tg = 190° C
- Td = 405° C
- Dk = 3.24 @10GHz*
- Df = 0.0014 @10GHz*

Ultra high speed > 56 Gbps per channel, NRZ (PAM2), PAM4 applications

Insertion Loss - Measured



ITEQ products approach the properties of PTFE. ITEQ offers best in class performance.

Note: RTF stands for Reverse Treat foil and Rz2mu stands for copper foil with Rz roughness value of 2.0 micrometers

IT-988G SE

- High Tg / Lead Free / Halogen Free
- Ultra Low Loss Laminate & Prepreg
- 100G / 400G Solution
- Lower Dk (<3.24 @ 10GHz)
- Ultra low Df (<0.0014 @ 10GHz)
- Very Stable Dk-Df across frequency
- HDPUG MRT-6 - LRC and HRC passed all thermal requirements
- HDPUG MRT-6 - LRC and HRC passed CAF requirements

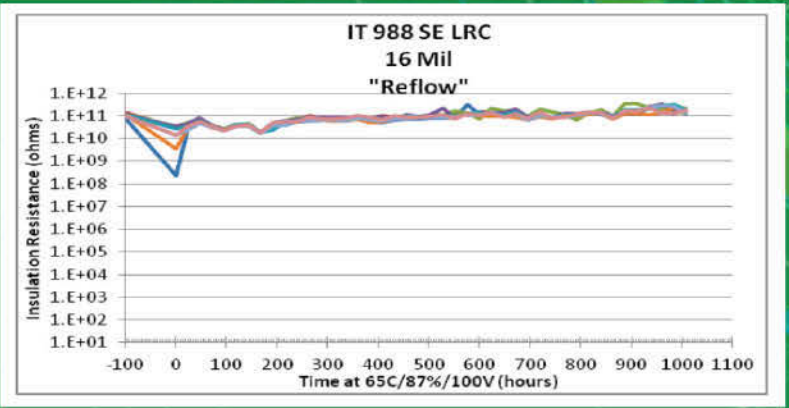
Sequential Lamination

7 Lamination cycle data

Lamination	DMA	DSC	TMA	T300 with CU	Solder Dip PCT: 1h @ 121°C	Td 2wt% / 5wt%
1	213	187 / 187	182	> 60	> 60	408 / 435
2	216	194 / 199	193	> 60	> 60	41 7/ 438
3	214	186 / 192	185	> 60	> 60	417 / 442
4	216	193 / 194	184	> 60	> 60	424 / 443
5	217	194 / 199	190	> 60	> 60	418 / 442
6	218	191 / 197	188	> 60	> 60	405 / 436
7	218	190 / 197	194	> 60	> 60	425 / 444

No themal degradation observable after 7 lamination cycles

HDPUG MRT-6 LRC CAF Data



ITEQ has the leading solutions for High Speed Digital applications with the extremely cost-effective IT-170GRA1 and IT-968 to the ultra high speed IT-988G and IT-988G SE for the highest end designs beyond 56 Gbps.

*53% resin content

ITEQ

as smoothly and efficiently as possible all the time. As well, all customers want the lowest price possible for their circuits and being efficient is an intuitive response. But what's more important to the customer? Is it the price of the circuit, or is it the best design for their new product? The customer might argue both, and it would be hard to disagree.

Is it the price of the circuit, or is it the best design for their new product?

Consider A New Way: DFA

Let's look at two theories in practice with a similar design. In the typical DFM model, a circuit would be designed with the supplier's specific capabilities in mind. For instance, the board size may be increased from the engineer's request to make room for the supplier's minimum trace width/space limitations, or the space between copper and the edge of the circuit. Or it could be to make room for larger diameter via holes because of the minimum hole size and annual ring requirements. These are not uncommon specifications defined as limitations from a circuit manufacturer. However, let's look at the outcome to see how the product may be affected. If the board size must increase then, in a typical situation, the entire product size must increase. In our current product environment of smaller is better, that would be a significant compromise for the customer to make simply to stay within their supplier's standards.

Let's consider an alternative theory: design for application, or DFA. If DFA is used in place of DFM in the same scenario, the board may be designed as intended and the circuit and product may remain at the original size it was imagined. The long-tenured supplier may choose not to make a circuit with these specifications. Or they may choose to look outside their standards and improve their processes to meet them. The advantages for the circuit manufacturer include the ability to continue to service a good customer, to expand their capabilities and challenge their limitations. This may cost the customer, initially, in the form of a higher priced circuit. However, they would have the opportunity to compare the increased circuit cost to the cost of a larger product profile and its acceptance in their industry. Product development engineers will appreciate the chance to weigh all the options intelligently, thereby landing upon the most valuable decision for the product and the company.

DFA has its challenges to be sure. The operators at the circuit manufacturer must be willing to try new processes and discuss all options. They must argue the pros and cons with equal enthusiasm to eliminate long-standing biases. And management must be able to sell the idea of change as being good for both the company and its employees. That's not a simple task. As well, the customer must understand that their current requirements fall outside of their supplier's standards and be patient and cooperative while they strive to improve.

This idea challenges decades of success of circuit manufacturers to educate their customers. But, it is just as exciting for the customer to educate the supplier. When they have suppliers that are willing to work with them and discuss available options as opposed to demanding the design be changed to suit their abilities, it should provide increased communication and maybe loyalty as well. The situation is a win for both parties. The customer gets the best possible solution to their challenges and the supplier improves and expands their capabilities. As time goes on, it would be reasonable to assume that the supplier's standards and limitations may be improved permanently and costs may be reduced for these improvements.

DFA. it's a new idea, but one worth exploring! **DESIGN007**

John Talbot is president of Tramonto Circuits, which designs and manufactures flexible circuits and printed circuit boards.

Ensure PCB Manufacturing Success!

CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing.

BluePrint-PCB®

Create comprehensive documentation to drive PCB fabrication, assembly and inspection.

DFMStream™

Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.

At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB design post processing needs.

Visit downstreamtech.com
or call 800-535-3226

DownStream Technologies

Top Coating Queries

Sensible Design

by Phil Kinner, ELECTROLUBE

My colleagues Alistair Little and Jade Bridges have been sharing their invaluable insights into encapsulation resins and thermal management products for the last few months. But this month I'm back on columnist duty. This is my first of many columns for 2018, and I have decided to share some top trending queries that concern many different applications and areas. LEDs are always a hot topic, as are volatile organic compounds (VOCs) and harsh environment concerns.

I have quite a few years of experience and knowledge of the coatings business, but of course, I'm still learning. Some queries tend to crop up more frequently than others, and I'm happy to provide some insight into these. Without further ado, here are Kinner's top queries.

Q So, I understand that historically the most widely used coatings are solvent-based. Can you explain the main benefits of solvent-free conformal coatings?

A These are very easy to summarise. Generally, solvent-free coatings tend to use less hazardous raw materials, have lower odour levels, and can be safer for the operator. Additionally, many solvent-free coatings are non-flammable, having a positive impact upon business insurance premiums. Environmental compliance is a growing trend and solvent-free alternatives certainly help to minimise the emissions of VOCs.

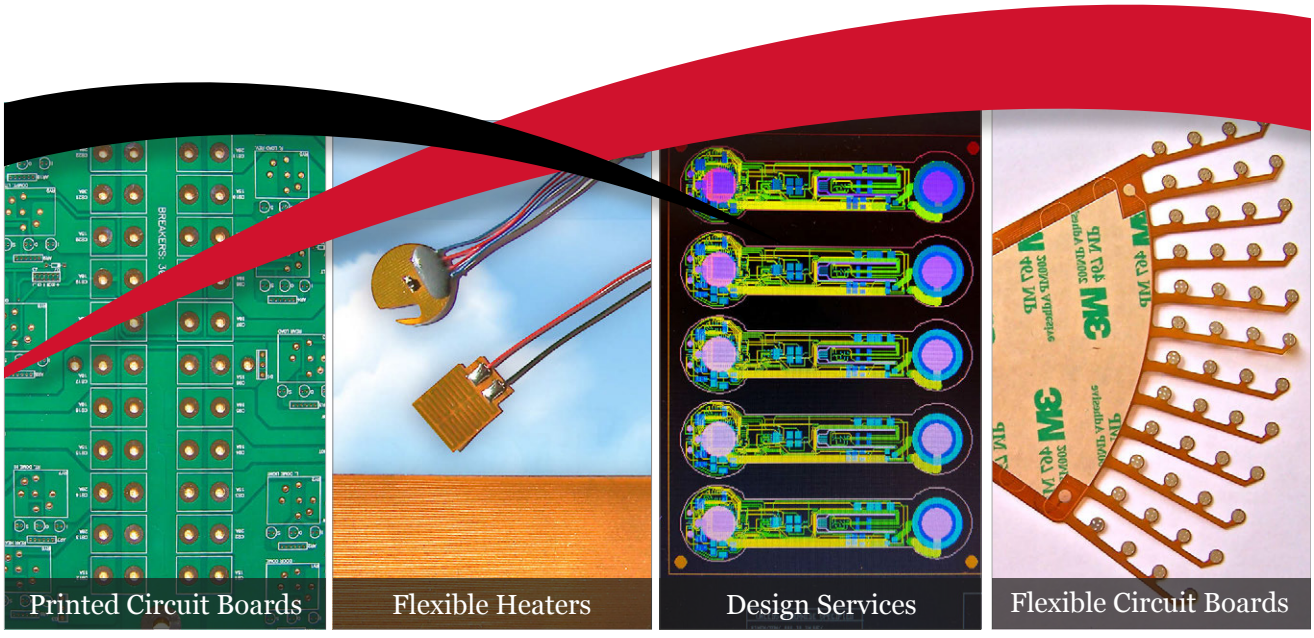
Q How has the trend of miniaturisation affected the development of coatings?

A The trend towards miniaturisation is most evident within consumer electronics, and has led to the development of ultra-thin coating materials (with thicknesses of less than 12 microns). In the case of mobile phones, for example, the reduced coating thickness in combination with internal gasketing and better case design has enabled the production of increasingly more water-resistant mobile devices.

Considering more traditional applications, such as aerospace and automotive, there is a double whammy effect of finer-pitch components (i.e., reduced space between the conductors) and a requirement to reduce weight, resulting in less protection being afforded by the board housing. Ultimately, the impact of this criteria means that coatings need to function more as a primary mitigation strategy against the environment, and the performance requirements of the materials are increasing dramatically. This is particularly evident in the case of condensation resistance.

TRAMONTO CIRCUITS™

Worldwide Provider of Flexible Circuits & Printed Circuit Boards



Unwavering Commitment

We put the customer first by listening to their needs, allowing our team to maximize accuracy and consistently deliver high quality products.

With our on time delivery record of 94.96%, you can expect your product to be delivered on time and to your standards, making us the best choice in the industry.

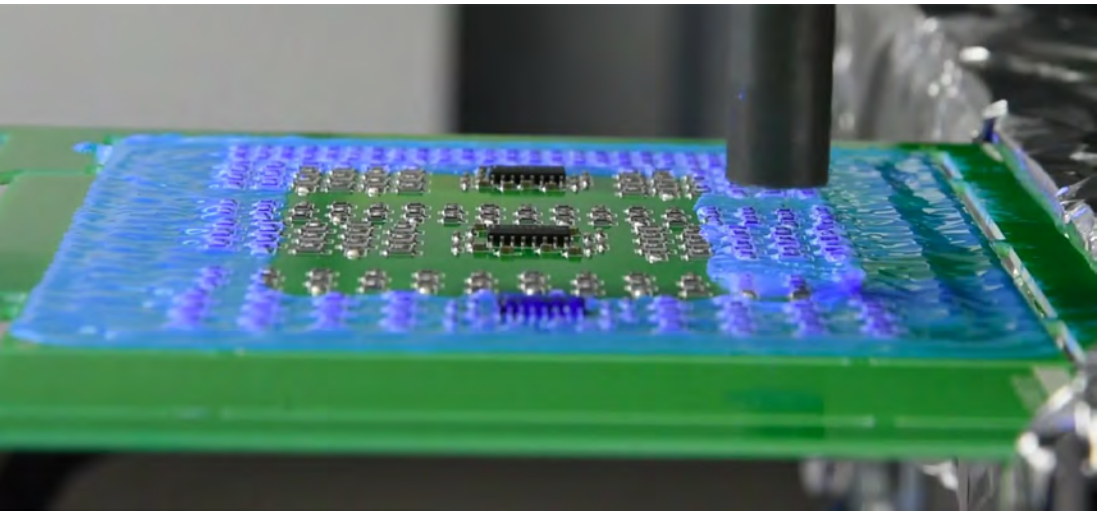
Contact us today for a consultation.

TRAMONTO CIRCUITS™

tramontocircuits.com

info@tramontocircuits.com

1.877.777.1977



Q The application method of coating materials is an important consideration. How can the best method be determined?

A To be completely honest, there is not necessarily a "best method." The application method selected for an assembly will be based on a combination of factors such as existing equipment/coating processes and the overall design of the assembly. This includes areas that should be coated and those which must not, such as switches and connectors, for example. The best application method would ensure that every board is coated and further receives coating coverage on all required metal surfaces at a sufficient thickness to afford protection against the environment. These requirements will change depending upon the specific board design and environment to protect against; there is no substitute for testing and verification.

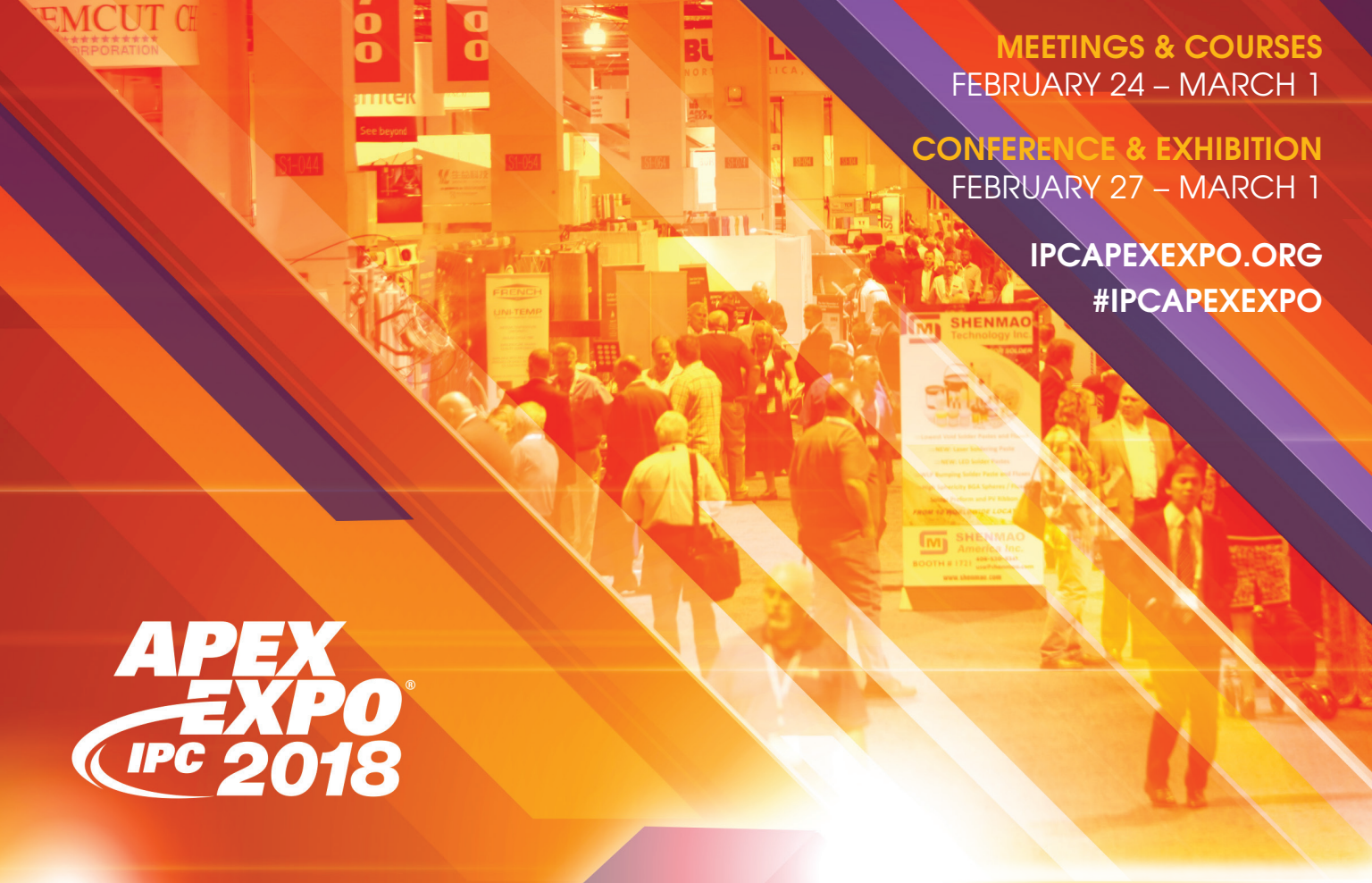
Q What typical combination of elements make up the harshest environments that coatings must protect against?

A For corrosion to occur on a metal surface, several factors must be present: namely, a potential difference, an electrolyte, and ionic impurities. If a coating is well applied, it prevents corrosion by preventing external ionic species and liquids such as water from reaching the metal surface. Any environment that can cause degradation of the coating prompting the conditions for corrosion can be potentially tough on the coating. An example of this could be an environment cycling between high and low temperatures thereby putting the coating under stress. Combined with the high temperature speeding up polymer degradation, cracking of the coating and thus failure points can occur. The cyclical nature of the thermal environment will usually lead to differences between the board and ambient conditions, prompting the formation of condensation and potential failures. Further, if this assembly were also subject to salt-spray, the potential for board failure would increase significantly.

Q Coatings are frequently utilised within LED applications, but just how do they work to extend the life of the luminaire?

A Modern LED luminaires consist of many discrete surface mount LED chips soldered to a support (either rigid or flexible). The role of a conformal coating in extending the lifetime of an LED luminaire is no different than with a regular circuit board—the coating must be able to resist the environment and prevent corrosion or signal leakage. In the case of LED applications, the coating must be sufficiently optically clear and remain thus to prevent colour or intensity changes in the LED output. **DESIGN007**

Phil Kinner is the global business/technical director for conformal coatings at Electrolube.



MEETINGS & COURSES
FEBRUARY 24 – MARCH 1

CONFERENCE & EXHIBITION
FEBRUARY 27 – MARCH 1

IPCAPEXEXPO.ORG
#IPCAPEXEXPO

APEX EXPO® IPC 2018

SUCCEED AT THE VELOCITY OF TECHNOLOGY

THE SHOW FLOOR WILL ACCELERATE YOUR INNOVATION

In 2018, your leading opportunity to stay on top of electronics industry innovation is on the IPC APEX EXPO show floor. Join thousands of your industry peers as you uncover new solutions that will move you in the right direction to capitalize on tomorrow's opportunities!

- >> Explore and compare equipment from more than 450 of the industry's top suppliers.
- >> Discover new processes to gain greater efficiency.
- >> Start your year with new suppliers that can save you both time and money.

VISIT IPCAPEXEXPO.ORG FOR MORE INFORMATION

NEW THIS YEAR!

WIN WITH OUR PASSPORT TO PRIZES GAME

Travel around the show floor collecting stickers from exhibitors, and enter for a chance to win an exciting prize!

SAN DIEGO
CONVENTION CENTER
CALIFORNIA > USA

MilAero Highlights

Standard of Excellence: 5 Things a Total Global Solutions Supplier Can Provide ▶

Our world now extends far beyond the traditional brick and mortar of the domestic board suppliers of yesteryear. Now our customers demand that PCB fabricators provide them with a complete and total printed circuit board solution, from domestically built ITAR-registered defense and aerospace boards to high-volume, offshore solutions.

Coast to Coast Circuits Adds CW Cernosek to Leadership Team ▶

Coast to Coast Circuits Inc. has added CW Cernosek to their leadership team as corporate business development and marketing manager.

2017: A Year of Progress and Poised for the Future ▶

Throughout 2017, NASA's Space Technology Mission Directorate (STMD) made noteworthy progress in maturing and demonstrating technologies to bolster America's space agenda, while setting the stage for vital advancements within the next several years.

APCT Holdings Acquires Cartel Electronics & Affiliate Cirtech ▶

APCT Holdings has acquired the Southern California based company Cartel Electronics and its affiliate company, Cirtech. With this acquisition, APCT now becomes one of the largest privately held printed circuit board manufacturers in North America.

European Aerospace Firm Acquires Nano Dimension DragonFly Pro 2020 3D Printer ▶

Nano Dimension Ltd. today announced that a leading European aerospace and defense supplier has purchased the company's DragonFly 2020 Pro 3D Printer for professional electronics manufacturing.

Ventec Passes AS9100 Revision D Transition Audit in China and UK ▶

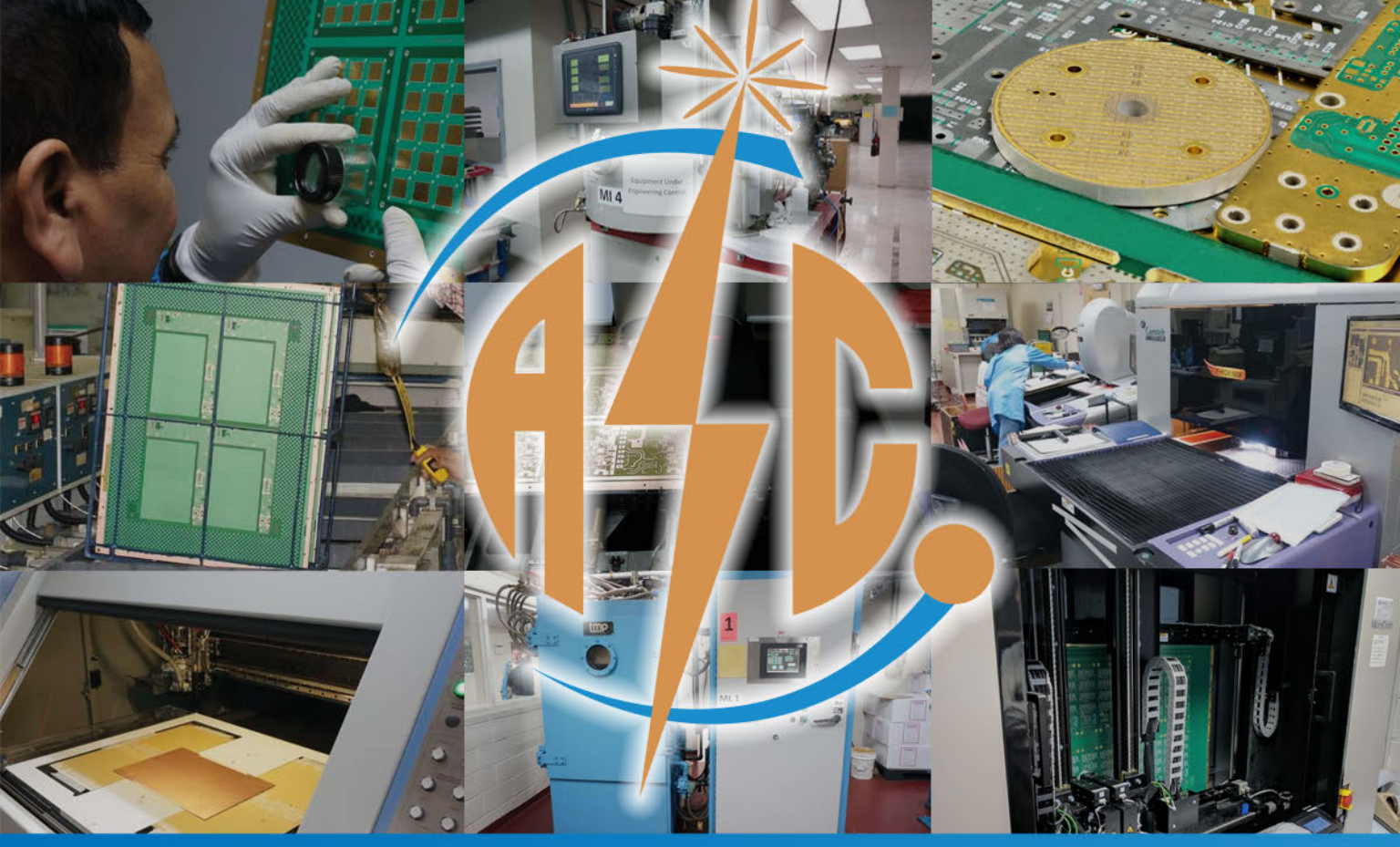
Ventec International Group is pleased to announce that the company headquarters in Suzhou, China and its European headquarters in the United Kingdom have both successfully passed the transition audit to AS9100 Revision D with zero non-conformances in accordance with the Aerospace Supplier Quality System Certification Scheme.

Global Electronic Warfare Market to Reach $17.7 Billion by 2027 ▶

The global electronic warfare market is expected to be valued at more than $13 billion in 2017, and will grow at a CAGR of more than 2.6%, to cross $17.5 billion by 2027.

Soft Materials Bring a Human Touch to Robotics ▶

Soft robotics has many promising applications. It brings a list of capabilities absent in most other robotics, including dexterity, flexibility and safe physical interaction with humans. Soft robotics refers to robotics that are comprised of either soft materials, soft joints or soft layers over hard armatures.



IT'S BEEN QUITE A YEAR AT ASC.

Sharing our Expertise

Everyone can tap into the decades of combined experience and expertise of our in-house experts on our recently launched **ASC Experts website**.

Meet the ASC Experts

Award-winning technology

We were recently presented with the **2017 Brilliance in Business Award for Outstanding Design and Innovation** by the City of West Chicago.

Learn More

Two E-Books

We followed our first eBook on Flex and Rigid-Flex with a guide to creating RF/Microwave PCBs which has been called the **"RF and microwave engineer's PCB fabrication bible".**

Download our eBooks

At ASC, we're not just building boards,
we're building the future. Let's build it together!

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

Now Available!

Still Using 1980s Formats for Design Data Handoff?

Hemant Shah and Ed Acheson
CADENCE DESIGN SYSTEMS

The IPC-2581 format was created in the early 2000s with the merger of two competing formats: ODB++ and GENCAM. The new format, the brainchild of the late Dieter Bergman, languished with no adoption until 2011, when a small group of companies created the IPC-2581 Consortium with the goal of getting this open, neutral and intelligent format adopted.

Today, the consortium consists of more than 90 corporate member companies with a single goal: migrate PCB design handoff from an assortment of unintelligent photoplot data to a single, intelligent file. Members from all electronics industry segments—design, manufacturing and supply chain—have collaborated to create an open format that also supports stackup data exchange between design houses and their fabrication partners.

Fabricators and assembly houses have reported that using a single file results in a 30% time savings compared to using multiple Gerber files and many other similar file formats. PCB designers and their partners are breathing a sigh of relief knowing that IPC-2581 eliminates the risks of mismatched data on both sides of the design process.

The consortium has been growing steadily in recent years. Its membership now includes more than 100 associate members in addition to its more than 90 corporate members, who represent a variety of PCB design and supply chain companies:

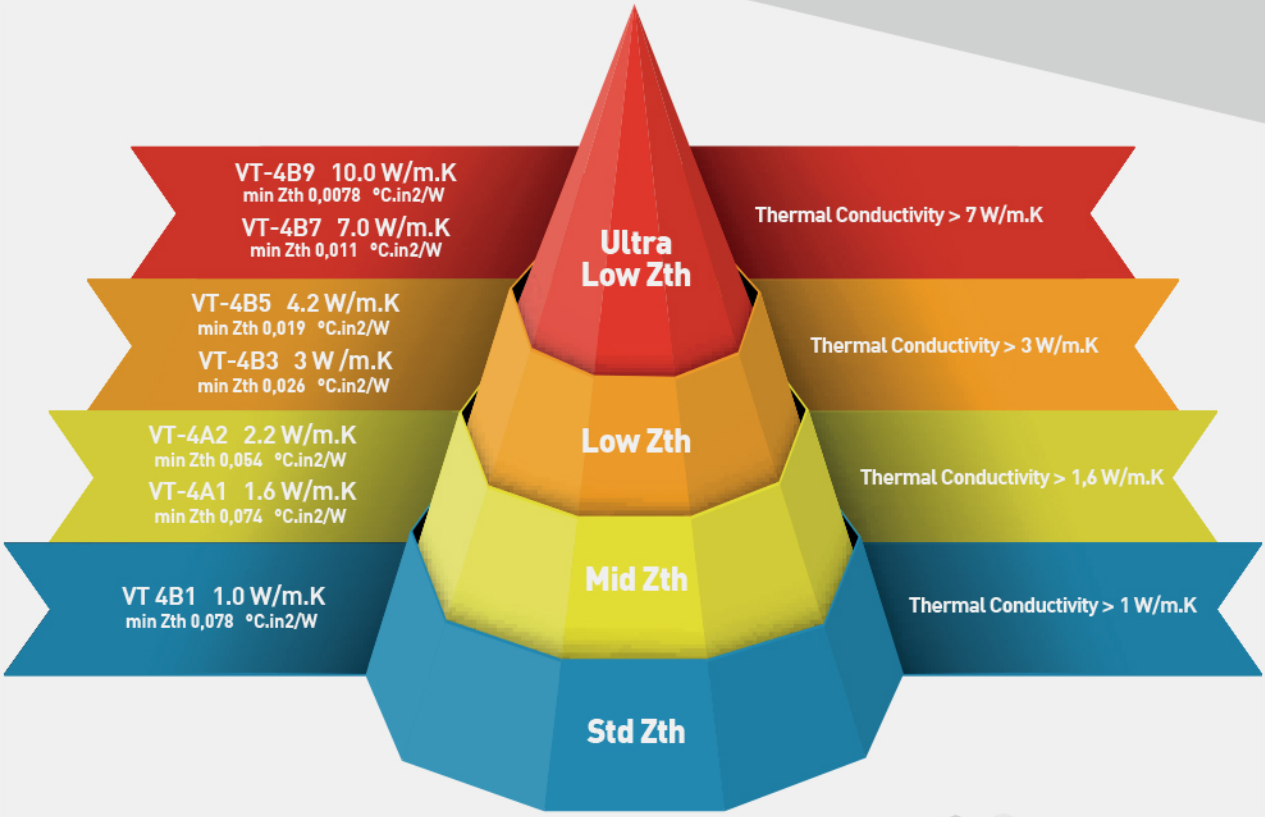
- PCB ECAD companies, including Altium, Cadence, Mentor, Zuken, ADIVA, DownStream Technologies, and WISE Software
- EMS companies, contract manufacturers, and fabrication companies
- Companies that provide software to EMS, fabrication and contract manufacturers, including AEGIS, Cimnet, Direct Logix, Easy Logix, GraphiCode, and Polar Instruments
- And, of course, IPC

What's Wrong with Current Approaches?

Today, the single-most commonly used format for fabricating the board is Gerber. This format was first introduced in the 1980s (almost four decades ago!). It was updated along the way—RS-274X in 1998 and X2 in 2014—but has not kept up with industry demands for reducing the risk of handing off inconsistent data to manufacturing partners. The GerberX2 format allows metadata to be added to the file, but doesn't convey all the necessary information to successfully manufacture and assemble a design. With the Gerber X2 format, multiple

Delivering an unprecedented level of thermal performance

ventec
INTERNATIONAL GROUP
騰輝電子



ventec-thermal

Technology leader for high performance thermal management materials

With the market need for ever-increasing thermal performance, reliability and quality, particularly for automotive and other LED lighting, power electronics and DC power conversion applications, the Ventec VT-4B family of thermally conductive IMS (Insulated Metal Substrates) deliver an unprecedented level of thermal performance through their established ceramic-filled halogen-free dielectric technology. Multilayered constructions are made possible through resin-coated foil and resin-coated film options. Contact us to discuss your requirements.

Ventec International Group
21 Water Street, Amesbury
Massachusetts, MA 01913, US
T: 978-521-9700
E: saleseast@ventec-usa.com
@VentecLaminates

www.**venteclaminates**.com

files in different formats are still required to describe stack-up information, drill information, test data, net list, bill of materials, and assembly data, all of which are included in a single IPC-2581 file.

The old Gerber-based approaches are error prone and require more designer effort to create these files (many of them) and make sure all the additional files that accompany the Gerber files are in sync. Guess who pays for the scrap if the files are not synchronized? The client.

The fabrication partner that receives the files must also do some work to convert this unintelligent data into an intelligent set of instructions for the fabrication machines. This requires more work by the fabricator. Guess who pays for that extra work? You guessed it, the client.

Why Use IPC-2581?

IPC-2581 is the only open, global, neutral format that is not controlled by any one company. With an open format, there are no licensing hassles and no contracts to sign.

As a neutral format, IPC-2581 is not owned by a for-profit organization. No revenue is gained by use of the format, and the format is driven by IPC members worldwide across all PCB product domains. These members define the necessary requirements for specific functions (fabrication, assembly, test, etc.) throughout the standard development process and help implement these requirements into the specification. Non-IPC members can have a voice during the definition process by working directly with the IPC-2581 Technical Committee.

IPC-2581 eliminates the risk of managing multiple files, as the single IPC-2581 file contains all the design data generated at the same time from the master design database. The following data, typically defined in multiple files, is now contained in a single IPC-2581 file with more in-depth attributes:

- Stack-up: Layer stack-up order, layer materials including all conductor, dielectric, mask, layer with thickness and other material characteristics

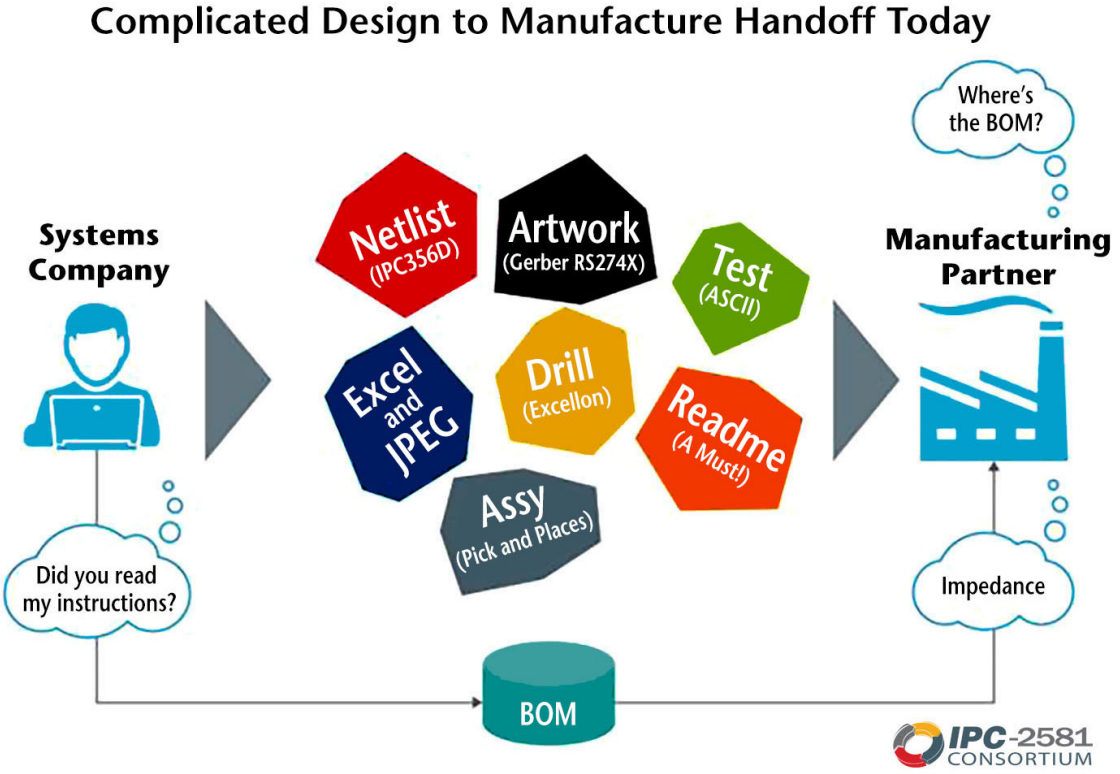


Figure 1: Intelligent board data is converted into disparate sets of data. The design house is responsible for sending the right set of correlated data.

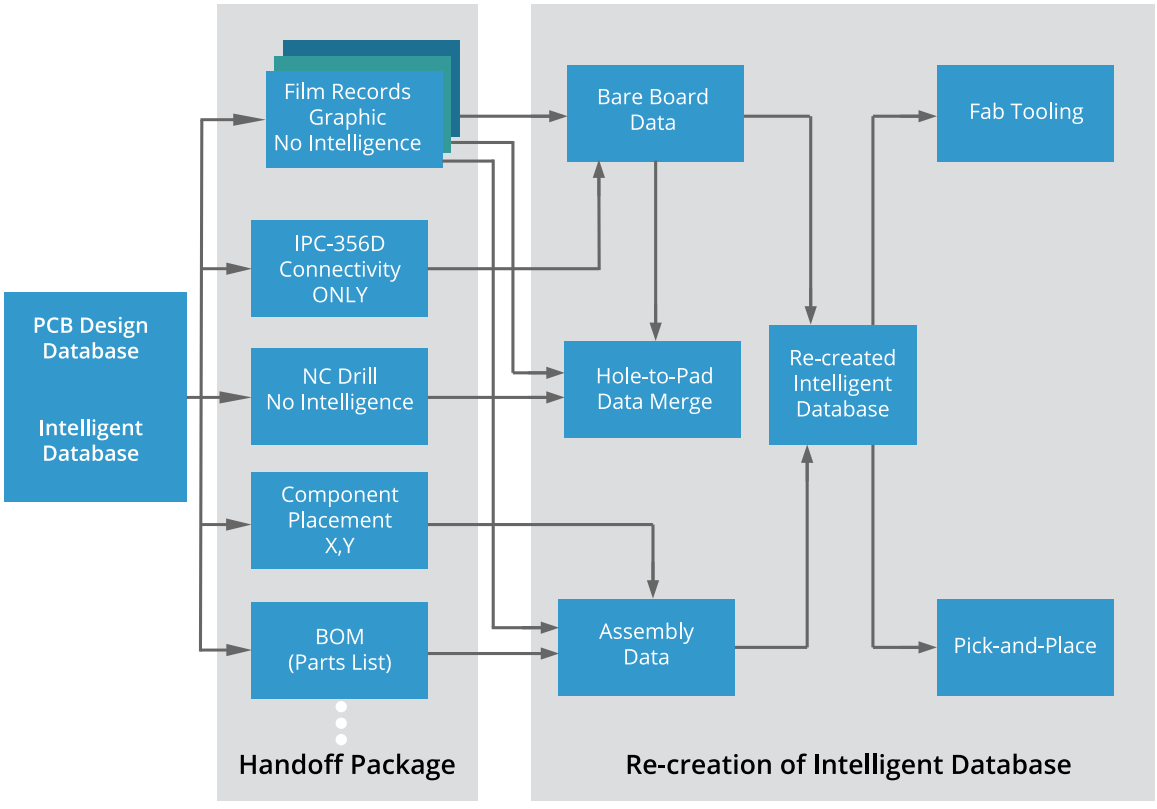


Figure 2: The fabrication house must convert disparate unintelligent data back into intelligent data. This is a waste of time and the client ends up paying for it indirectly.

- Drilling data: Through-hole, blind-buried holes and back drill
- Artwork: Images for all layer structures, plus conductor, masks, silkscreen and documents layer
- BOM: For board stack-up and assembly, also includes variant BOM
- Netlist: Both logical and physical netlist (like IPC-356D) for testing
- Component placement: XY location and top, bottom, or embedded
- Spec element: A special item in the format used to attach notes or other details to geometry, layers, nets (such as impedance) or components

This comprehensive information contained within the IPC-2581 data virtually eliminates the risk of interpretation errors.

IPC-2581 is designed to grow as technology grows and as processes improve. Driven by a cross-section of design centers, tool vendors and manufacturing representatives, the format continues to evolve while minimizing impact to older processes and data sets. Only an open format, like IPC-2581, can provide visibility into the data structure of the format and expansion capability.

Benefits of using IPC-2581 include:

- An intelligent data handoff
  - Faster for manufacturing partners and saves time at setup
  - Greatly reduces risks for both OEMs and manufacturers in data and version mismatches
- Driven by the industry, for the industry (no one company controls its development)
- A single source of truth (no room for manual errors), with the ability to send a subset of data to appropriate partners
  - Only bare board data to PCB fabricators in one file
  - Only assembly data to assembly houses in one file
- Also includes any combination or subset of data that is appropriate for the partner that is receiving the data

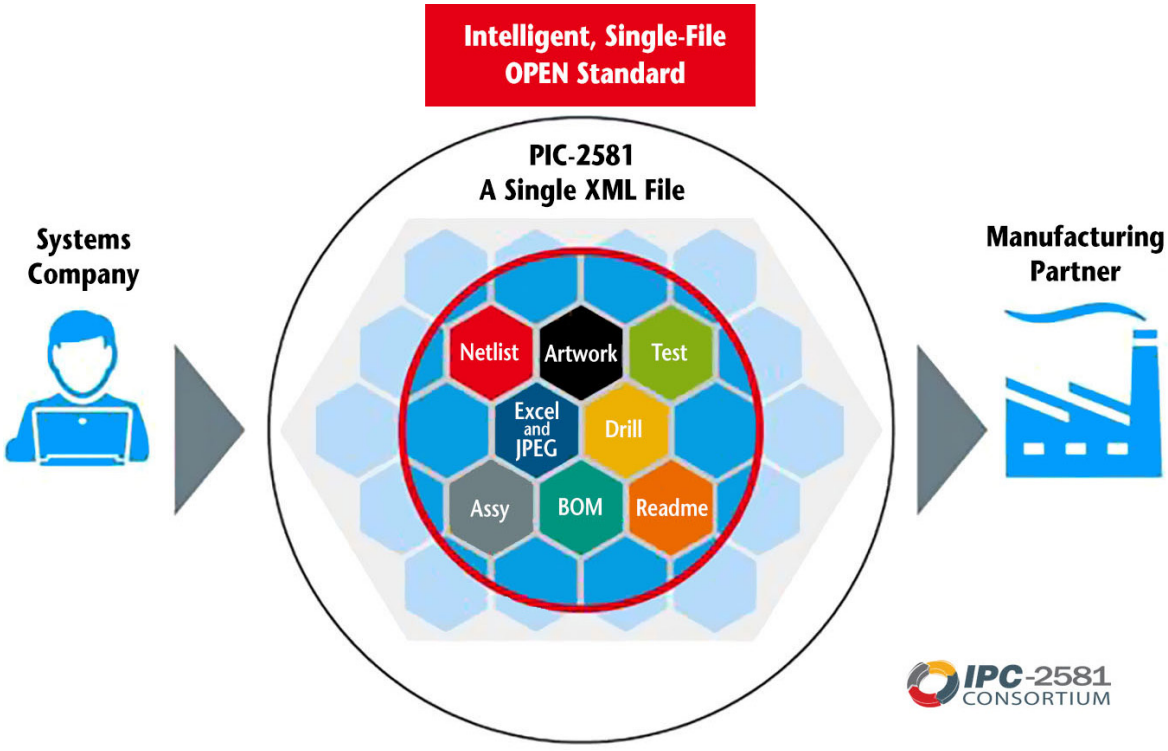


Figure 3: An intelligent data handoff to manufacturing is good for the design house and for the manufacturing partners.

- Efficiency: No need to check multiple files and verify their versions
- Easier for OEMs, better for manufacturers, and designers sleep better at night after handoff

Who Supports IPC-2581?

Thousands of designs from multiple design houses around the globe have been built using IPC-2581. Some companies have standardized on IPC-2581 as the only way to hand off data to manufacturing. Did you know that some leading manufacturers charge more money to accept Gerber designs?

Axiom Electronics encourages their customers to provide design data in the IPC-2581 format and has made continual advancements in their assembly process utilizing the data supplied. A PCB West 2017 presentation titled "IPC-2581 Digital Data Transfer Success Stories and Future" noted:

"Axiom has successfully integrated files supplied in IPC-2581 format into key areas of our manufacturing process to reduce time and to increase quality by adding the ability to visually see specific information on our SMT lines. Collaborative efforts with Wise Software and TTM along with key customers have enabled Axiom to be more efficient in building complex assemblies.

The ability to have machine operators verify BOM information quickly has increased our throughput, especially when it comes to DNI components. It's no longer the un-intelligent data we have been using for years, nor is it a

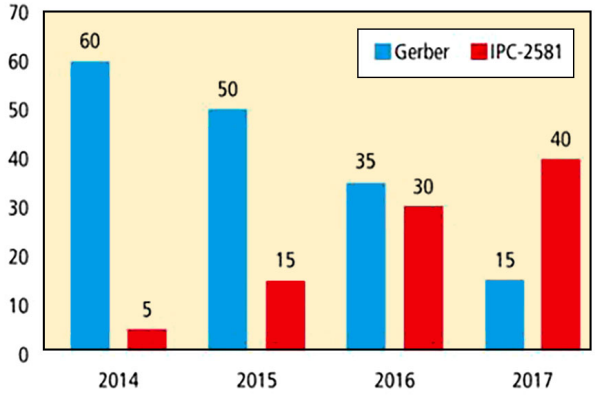


Figure 4: In 2017, the percentage of jobs sent to Axiom in IPC-2581 format surpassed the ones sent in Gerber format.

data file with multiple sub-folders. It's one IPC-2581 file, plain and simple."

Figure 5 is a list of manufacturing and software companies that support IPC-2581 today. This status changes over time. To see which software companies support IPC-2581, visit the support status page on the IPC-2581 Consortium website.

Using a single, intelligent file to hand off your PCB design data is better for both the design

Manufacturer Support Status

Company Name	Software Used	IPC-2581 Amend 1	IPC-2581 A	IPC-2581 B	Stack-Up Exchange
Accurate Circuit Engineering	Genesis, Insight, InPlan		✔		
Axiom	VisualCAM	✔	✔	✔	✔
CircuitCAM		✔	✔		
CC Electronics	VisualCAM	✔	✔	✔	✔
Electrostein	CAM350	✔	✔	✔	
Sanmina	Genesis, Insight, InCAM		✔		
Sierra Circuits	Genesis		✔		
TTM	Genesis		✔	✔	✔

Software Vendor Support Status

Company Name	Software Used	IPC-2581 Amend 1	IPC-2581 A	IPC-2581 B	Stack-Up Exchange
AEGIS	Genesis, Insight, InPlan	✔	✔		
ADIVA	ADIVAnet	✔	✔	✔	
	ADIVADRC	✔	✔		
	ADIVAview	✔	✔		
Altium	Altium Designer			✔	
Cadence	Allegro PCB Designer	✔	✔	✔	✔
	OrCAD PCB Designer	✔	✔	✔	
Downstream Technologies	CAM350		✔	✔	
	Blueprint PCB	✔	✔		
	DFMStream	✔	✔		
EasyLogix	PCB-Investigator	✔	✔		
In-Circuit Design	ICD Stackup Planner		✔	✔	✔
Numerical Innovations	FAB 3000 Version		✔	✔	
	ACE 3000 Version	✔	✔		
	PreflightPCB Version	✔	✔		
Polar Instruments	Speedstack			✔	✔
PTC	PTC Creo View ECAD		✔	✔	
Siemens	Test Expert		✔	✔	
	UniCam FX	✔	✔		
	UniDoc FX	✔	✔		
Vayo	VayoPro-DFM Expert		✔	✔	
	VayoPro-SMT Expert	✔	✔		
	VayoPro-Test Expert	✔	✔		
	VayoPro-Document Expert	✔	✔		
	VayoPro-View Expert	✔	✔		
WISE	VisualCAM	✔	✔	✔	✔
	GerbTool	✔	✔		
	WISE2581 Viewer	✔	✔	✔	
Zuken	CR-5000	✔			
	CR-8000		✔		

Figure 5: A list of industry companies that support IPC-2581.

houses and their manufacturing partners. The IPC-2581 Consortium continues to aid software vendors supporting the format as well as to design houses and manufacturing companies who are adopting the standard. The IPC-2581 Consortium offers technical and networking assistance to help you along the way. Through the consortium, you can easily learn from the experiences of others who have successfully adopted IPC-2581.

Join the Consortium

Be part of a unique movement in the electronics industry; we welcome new members at any time. There are two ways to join the consortium: become a corporate member or become an associate member. There is no fee, and there are no contracts required to join. Corporate members are invited to participate in bi-weekly business and technical meetings. Associate members receive regular updates on consortium activities via periodic emails. Visit www.ipc2581.com for up-to-date information on the format itself and the activities of the consortium, as well as support status and technical assistance. To join the consortium, click Join at the top of the page. **DESIGN007**

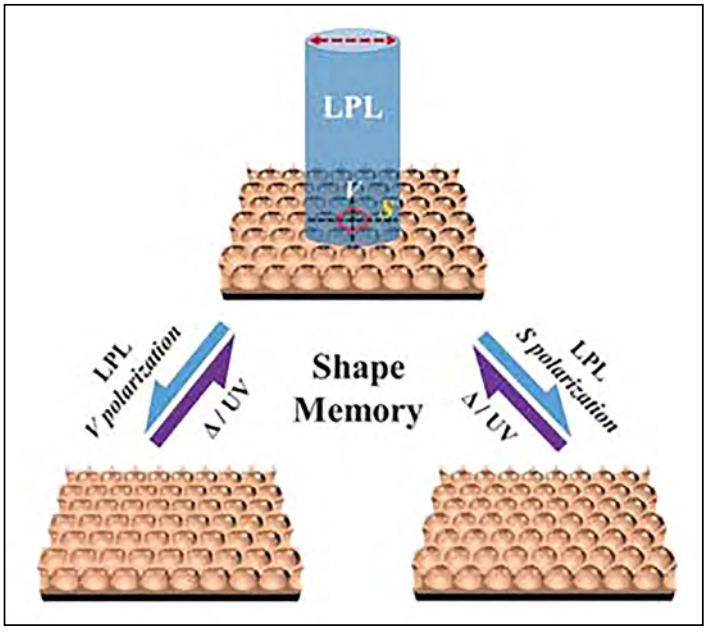
Hemant Shah is IPC-2581 Consortium chair and product management group director for Allegro PCB Products at Cadence Design Systems.

Ed Acheson is IPC-2581 Technical Committee chair and principal product engineer for Allegro PCB Products at Cadence Design Systems.

Pores With a Memory

Researchers from the East China University of Science and Technology in Shanghai, China; the Georgia Institute of Technology in Atlanta, USA; and Pusan National University in Pusan, South Korea have developed an unconventional strategy to prepare photoswitchable, porous films. Their success stems from a simple method based on condensation patterns known as breath figures. Breath figures are formed when water vapor condenses as a pattern of tiny droplets on a cold solid or liquid surface, like when you breathe onto a cold window.

One block of the polymer network has photoresponsive side-chains with azobenzene units that change their conformation (cis-trans isomerization) in reaction to light. The other block can be crosslinked to fix the configuration of the film. Irradiation with UV light or heating changes a part of the azobenzene units to the bent cis form, irradiation with visible light causes the groups to preferentially adopt the straight trans form. If the visible light is linearly polarized, the side chains arrange themselves in parallel. This rearrangement causes displacement of the material. Careful control of the direction of polarization allows the researchers to transform the originally round micropores into a variety of different shapes, such as rounded rectangles, or rounded rhombuses.



The researchers hope to use their new production method to make robust, switchable films for electronics, photonics, efficient separation and purification processes, and functional biomaterials for regenerative therapies.

Elements Ensuring First-Time Success in NPI

by Dora Yang
PCBCART

Successful launching of new products into market involves steps with challenges and complexities, each of which is closely associated with another. It's common to see, however, that a great idea fails to be smoothly transferred into a product solely due to insufficient focus on one or more steps. For example, a product concept may fail to be fully validated or even becomes degraded, or a validated product may not be optimized for cost-effective manufacturing and assembly.

New product introduction (NPI) starts with an exciting idea that will be converted into a final electronic product after going through delicate design, high-quality manufacturing and assembly, and rigorous inspection and testing. The new product should be fundamentally compatible with the following requirements:

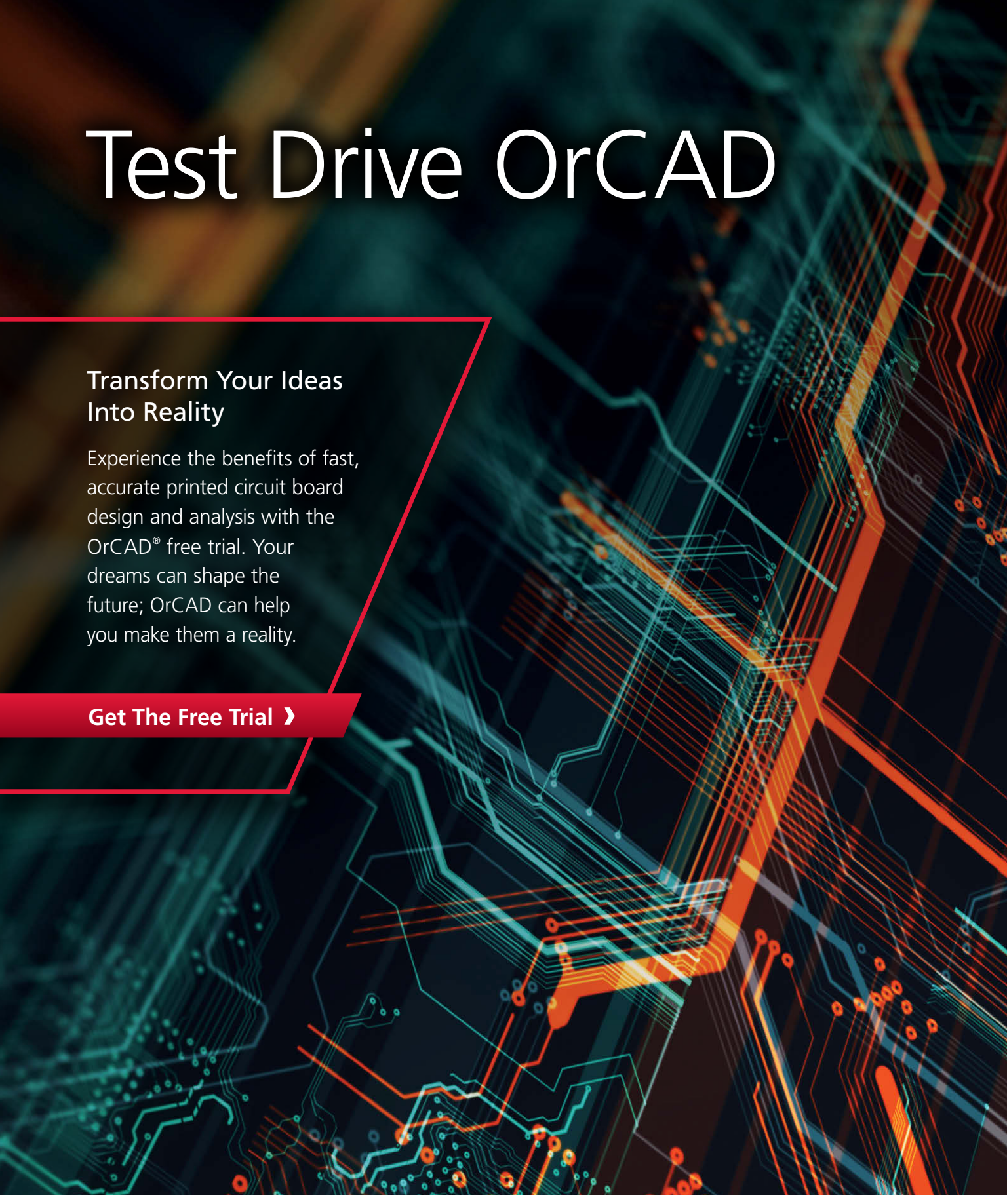
- It should be as functional as what is called for by the initial idea
- It should be available for the market quickly

The above two items are basic elements ensuring your success of new product, and both depend on the capability and reliability of your contract manufacturer (CM) and assembler. This article will uncover the leading elements that can guarantee your success in NPI, such as DFM/DFA checks, component availability, manufacturing/assembly capabilities, inspection and test, first article inspection and approval, and service quality.

DFM/DFA Check

Once your idea is converted into design files, you've kicked off your NPI process. The final product will be manufactured and assembled based on your generated file packages, including:

- Gerber files
- Schematics
- NC drill files
- Bill of Material (BOM)
- CAD drawings
- Assembly instructions, etc.



There is often quite a distance between the design engineer and the manufacturing/assembly engineer, which provides sufficient necessity for DFM/DFA check. An excellent design for manufacturing/assembly check helps identify potential issues prior to production, when changes are not as costly. An efficiency check must also be considered, since it determines a product's time to market. Improvement here derives from efforts in two aspects: your design file preparation and the manufacturer's DFM/DFA capabilities.

Your design files should be as complete as possible.

Your design files should be as complete as possible. For example, a DFM engineer once told me that she had been busy generating coordinate files all day, because their customers seldom provided them. Later, when customers were required to do so, her DFM check efficiency dramatically rises. Until now, DFM/DFA checks provided by most EMS providers were manually implemented. But now, some advanced EMS providers use automatic DFM/DFA systems, dramatically increasing efficiency. Our company utilizes the Valor DFM system.

Component Availability

During assembly, components play a key role in electrical functionality and connection implementation. Therefore, an assembler's ability to manage components directly affects the success of NPI.

Component management is primarily composed by three aspects: component procurement, inspection, and storage. Component procurement is always regarded as a critical standard of evaluating an assembler's capability. Long-standing and stable cooperation with authorized distributors and providers must be guaranteed so that component availability can be ensured, along with competitive prices. Moreover, the purchasing staff should fully respect their customers' selections when specific component distributors are designated by customers.

Even components purchased from the most reliable sources should be carefully inspected prior to assembly. Tiny components require rigorous environment and storage conditions. Temperature and humidity are essential to the implementation of component performance and functionality. Sometimes, for the sake of low cost, many components are purchased, but not all are used, so the rest of the components need to be temporarily stored in the assembler's warehouse.

Fabrication and Assembly Capabilities

Fabrication:

When qualifying a company's PCB fabrication operations, the following items should be emphasized:

Quality Grade

The quality grade to conform to is a direct tool evaluating the quality of circuit boards because regulations compatible with concerning quality grade determine and affect all manufacturing details. Generally speaking, PCBs fabricated by reliable manufacturers will be built to IPC Class 2.

Layer Count

This aspect is often neglected because it tends to be taken for granted. A truly trustworthy manufacturer should be capable of producing boards whose layers range from one to many. Apart from single- or double-sided PCBs that are most widely applied, other layer numbers should be available as well, especially for those whose products call for HDI and miniaturization.

Substrate Material

Substrate material determines board's temperature and strength. Manufacturers should be able to deal with different substrate materials including standard FR4, Rogers, Teflon etc.

Copper Weight

Copper weight can be further categorized into outer layer copper weight and inner layer copper weight that should be respectively calculated and emphasized. Since copper weight is directly associated with board thickness, it has to be carefully measured.

Minimum Tracing/Spacing

Tracing/spacing is a delicate element of a manufacturer's capabilities, since it is associated with board size and density level. PCB designers should be fully aware of minimum tracing/spacing requirements so that they can optimize their PCB design files.

Solder Mask

Solder mask is more than just a variety of colors. Advanced solder mask, peelable solder mask, and carbon mask can be used to evaluate your partner's capabilities.

Surface Finish

Various surface finishes are selected for advantages and disadvantages on performance, cost and applications. Plus, RoHS compatibility matters for products that are required to be environmental-friendly.

Other Items

Other items are primarily associated with advanced capabilities such as blind/buried vias, via-in-pad, gold fingers, edge plating, countersink/counterbore hole, etc. As far as via or hole parameters are concerned, most attention should be paid to via diameter because it straightly reflects whether manufacturer's capability can meet your design demands.

Assembly:

When qualifying a company's PCB assembly operations, the following items should be emphasized:

Quality Grade

As with PCB fabrication, for advanced, high-quality assembly, the OEM should select an assembly provider that offers advanced capabilities.

Assembly Types

Two leading types of assembly are available: plated through-hole (PTH) and surface mount technology (SMT). In complicated conditions, both technologies are often simultaneously required. If assembly sequence is arranged improperly or an inappropriate soldering temperature is set, defects or failure may result. Thus, your contract assembler should be able to maintain high-level assembly capability.

Assembly Accuracy

Assembly accuracy is another key element evaluating assembler's manufacturing capabilities. Since miniaturization has become one of the key trends in electronics, assembly density and accuracy start playing an increasingly significant role that should be in your first batch of considerations. The facility's minimum component size, 01005, for example, should be known and confirmed in advance.

Component Packages

Various types of component packages are available, but not all assemblers can deal with all component packages, so it's necessary to get assured that your required component packages like QFN, BGA, and CSP can be assembled by your potential partner.

Inspection and Test

To be aware that your products have been manufactured in accordance with original design, inspection and test are vital. For PCB fabrication test, custom electrical test such as bed of nails or flying probe is often required. But for PCB assembly test, however, AOI or AXI is required. During assembly process, AOI usually comes in two formats: online and offline. For high-quality NPI, the OEM should know whether both types are available.

During assembly process, AOI usually comes in two formats: online and offline.

First Article Inspection

First article inspection is one of the crucial steps before mass production. The product available for first article inspection and approval can be totally regarded as a dress rehearsal before its formal performance on market stage.

To make the best of the functionality of first article inspection, you must ensure that your CM or assembler is able to provide first article inspection service and contains dedicated inspection lists to conform to. Then you should establish your own inspection lists and requirements to compensate for your specific demands, such as harsh environments or environmental-friendly concerns.

Speed and Response

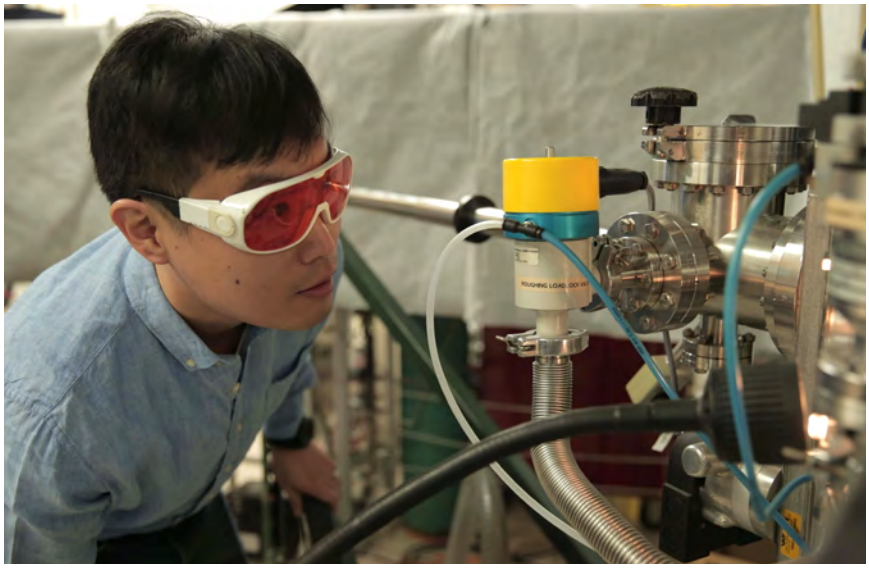
PCB fabrication and assembly are not so easy as building LEGOs. Deadlines and delivery dates must be confirmed between manufacturers and their customers. Furthermore, efficiency is vital for a new product entry into market. Smooth communication is the basis of long-term cooperation. As far as that is concerned, language does matter, which is especially true for offshore manufacturing. Nevertheless, the language barrier is being broken by the wide acceptance of English by electronics manufacturers around the globe.

All EMS provider swear that their focus is on customers. So, unless you have a lie detector handy, just submit your requirements and wait for their reply. Never take short cuts when qualifying a new manufacturing partner. **DESIGN007**

Dora Yang is a technical engineer with PCBCart, a China-based full turnkey PCB assembly services provider. If you have questions related to PCB design, manufacturing or assembly, reach her on Twitter @dorayang0227 or directly at www.pcbcart.com.

Beyond Silicon: Researchers Solve a Materials Mystery Key to Next-Generation Electronic Devices

Think of one half of any famous duo, and the other half likely comes to mind. The same is true in oxide electronics materials. Yet until now, a critical aspect has been missing–one that complements the function of electrons in oxide electronics. And a team led by University of Wisconsin-Madison materials scientist Chang-Beom Eom has directly observed that missing second half of the duo necessary to move oxide electronics materials forward.



While other researchers have made the material in a bi-layer structure, Eom designed a triple layer. He alternated layers of strontium oxide and titanium dioxide on the bottom, then layers of lanthanum oxide and aluminum oxide, then added additional layers of strontium oxide and titanium dioxide on the top.

As a result, the hole gas forms at the interface of the layers on the top, while the electron gas forms at the interface of the layers on the bottom – the first demonstration of a very powerful complementary pair.

iCD Design Integrity

Incorporates the iCD Stackup and PDN Planner software. Offers PCB Designers unprecedented simulation speed, ease of use & accuracy at an affordable price

Dielectric Materials Library
30,700 Rigid & Flex Materials to 100GHz

Termination Planner
Extracts IV Curves from IBIS Models
Calculates Series Terminator of the Distributed System Including Loads

iCD Stackup Planner
Field Solver Accuracy, Characteristic Impedance, Edge & Broadside Coupled Differential Impedance

Allegro, Altium, Excel, HyperLynx, OrCAD, PADS, IPC-2581B Interfaces

Extensive Capacitor Library
5,650 Decaps Derived from SPICE Models

Matched Delay Optimization
Relative Signal Layer Propagation
Ideal DDRx Trace Delay Matching

iCD PDN Planner
AC Impedance Analysis & Plane Resonance

Integrated Stackup Planner, VRM Definition, Plane Resonance, Planar Capacitance & EMI to 100GHz

Heads-up Impedance Plots

PDN EMI Plot with EMC limits (FCC, CISPR) to 100GHz

iCD Stackup Planner - Offers Engineers & PCB Designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- Industry Leading 2D (BEM) Field Solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Relative Signal Propagation with 'Matched Delay Optimization'–ideal for DDRx design
- Termination Planner - series termination based on IBIS models & distributed system
- Unique Field Solver computation of multiple differential technologies per signal layer
- Extensive Dielectric Materials Library –over 30,700 rigid & flexible materials up to 100GHz
- Interfaces–Allegro, Altium, Excel, HyperLynx, OrCAD, PADS, Zmetrix TDR, Zuken & IPC-2581B

iCD PDN Planner - Analyze multiple power supplies to maintain low AC impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance and projected EMI
- Definition of plane size, dielectric constant & plane separation
- Extraction of plane data from the integrated iCD Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- PDN EMI Plot with FCC, CISPR & VCCI Limits. Frequency range up to 100GHz
- Extensive Capacitor Library –over 5,650 capacitors derived from SPICE models

"iCD Design Integrity software features a myriad of functionality specifically developed for high-speed design."
- Barry Olney

In-Circuit Design Pty Ltd

www.icd.com.au

1 The PCB Design Magazine is Renamed Design007 Magazine; January 2018 Edition Now Available ▶

To ring in the new year, we've made a few changes to our magazines. You may have noticed that our name has changed from The PCB Design Magazine to Design007 Magazine. Not only did we update our magazine's name, but we also refreshed the interior pages of the publication.

2 Beyond Design: Signal Flight Time Variance in Multilayer PCBs ▶

Signals travel at the same speed, given the same medium. However, the microstrip (outer layer) traces are embedded in a mélange of dielectric material, solder mask (if required) and air. This lowers the effective dielectric constant and increases the propagation speed compared to that of stripline (inner layer) traces. This month, I will look at the disparity in signal propagation in multilayer PCBs.

3 Dr. Johannes Adam Gets Hot on Thermal Management ▶

Thermal management expert Johannes Adam, PhD, was kind enough to take the train from his home in Leimen, Germany to meet with me during productronica in Munich.

4 EMA Cloud Lined with Capability ▶

Editor Kelly Dack speaks with EMA Design Automation Marketing Manager Chris Banton about the company's new free cloud-based OrCAD schematic capture tool, OrCAD's design constraint manager, and the Sigrity electrical rule checker.

5 I-Connect007 Launches Fundamentals of RF/Microwave PCBs Micro eBook ▶

Written by John Bushie, director of technology at American Standard Circuits, and Anaya Vardya, president and CEO of ASC, this micro eBook provides information needed to understand the unique challenges of RF/microwave PCBs.

6 Rogers Continues Expansion into High-Speed Digital Materials ▶

Mahyar Vahabzadeh discusses a paper presented by his colleague Dr. Allen Horn at DesignCon. He also explains some of the different characteristics Rogers has discovered as they move from RF into high-speed digital materials.

7 Cadence Reports Q4 and Fiscal Year 2017 Financial Results ▶

Cadence reported 2017 revenue of $1.943 billion, compared to revenue of $1.816 billion for 2016. Cadence recognized net income of $204 million, compared to net income of $203 million for 2016. Revenue for the fourth quarter of 2017 totaled $502 million, compared to revenue of $469 million reported for the same period in 2016.

8 Fully Automated Schematic Verification ▶

The schematic is the controlling document for every PCB design. It captures the design intent and drives all downstream processes including simulation, analysis, layout, fabrication, and assembly. As such, it is critical that the schematic accurately reflects the product's electronic requirements and specifications. Mentor's Craig Armenti explains.

9 APCT's Cartel Acquisition Adds Capacity, Technology and Certifications ▶

APCT recently acquired Cartel Electronics and its affiliate, flex and rigid-flex maker Cirtech. During DesignCon 2018, I spoke briefly with APCT President and CEO Steve Robinson about what this acquisition means for the San Jose-based fabricator.

10 Mike Jouppi Discusses his Drive for Better Thermal Data ▶

Mike Jouppi is an engineer and founder of the Thermal Management LLC consulting firm. He spent years updating IPC's charts on current-carrying capacity, which had been unchanged since the 1950s. An advocate of a multi-physics approach to solving thermal issues, Mike is always on the cutting edge of electronics cooling. I recently caught up with Mike and asked him to give us his views on the state of thermal management, as well as the tools and standards related to thermal design.

Career Opportunities

Pssst! Are You Looking for Someone?

Place your notice in our Help Wanted section.

For just $500, your 200 word, full-column—or, for $250, your 100 word, half-column—ad will appear in the Help Wanted section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs and suppliers.

Potential candidates can click on your ad and submit a résumé directly to the email address you've provided. If you wish to continue beyond the first month, the price is the same per month. No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:
Barb Hockaday at barb@iconnect007.com or +1.916.608.0660 (-7 GMT)

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

APCT

Passion I Commitment I Trust

GRANTED!

The Acquisition of Cartel Electronics & Affiliate Cirtech Grants The APCT Wish For:

- Expanded Capacity
- Regional Support in Southern California
- Flex/Rigid-Flex Capabilities
- Additional Defense & Aerospace Certifications

A Leading Manufacturer of High Reliability Printed Circuit Boards

APCT has acquired the Southern California based company Cartel Electronics and its affiliate Cirtech. With this acquisition, APCT now becomes one of the largest privately held printed circuit board manufacturers in North America.

The acquisition of Cartel brings APCT much needed capacity and regional support, while the acquisition of Cirtech brings APCT Flex & Rigid Flex capabilities and the corresponding certifications of the Defense & Aerospace industries.

According to APCT President / CEO Steve Robinson, "As APCT continues to grow, we will not only share our expertise with those new to us; but will continue to nurture the culture of Passion, Commitment & Trust that has become the very fabric of our existence."

APCT Continues Its Mission Statement of:

Passion	*To Provide Ultimate Customer Satisfaction*
Commitment	*To Service and Execute with High Reliability*
Trust	*To Be Earned By Our Actions*

APCT Delivers

LEAD TIMES	World Class Cycle Times 2 - 10 Layers; 24 hrs 12 - 24 Layers; 48 hrs We can support Via in Pad 48-72 hrs We can support HDI 3-5 day turns
QUANTITIES	From Prototypes to Off-Shore solutions through APCT Global
PRICING	No tooling charges; No test charges
LAYER COUNT	Standard: 2 - 28 Layers Advanced: 30 - 38 Layers Development: 40+ Layers Flex/Rigid Flex: 2 - 24 Layers
CERTIFICATIONS	ISO 9001 Rev 2008 Certified AS9100D Certified NADCAP Certified MIL-P-55110 Certified MIL-PRF-31032 Certified ITAR Registered IPC 6012 Class III

APCT, "Where Our People Differentiate Us From The Competition."

APCTinc.com

APCT Santa Clara
Corporate HQ
408.727.6442

Cirtech
A Division of APCT
714.921.0860

APCT Orange County
714.993.0270

APCT Wallingford
203.269.3311

APCT Global
203.284.1215

Career Opportunities

Printed Circuits

THE FUTURE OF RIGID FLEX CIRCUITRY

Sales Administrator

Purpose:
To assist the Sales Department in entering and tracking customer orders, supporting sales and marketing functions, and growing Printed Circuits customer base and sales.

Nature of Duties/Responsibilities:
- Provide point of contact for customers' quotes and orders
- Enter purchase orders
- Check orders for accuracy and completion
- Resolve order errors and inaccuracies
- Handle customer emails and phone calls
- Track and expedite customer requests and inquiries
- Work with customers to resolve outstanding questions and/or issues
- Report on open orders
- Keep customer contact database current
- Work with Engineering and Quality Assurance to meet customer expectations
- Complete other sales and/or marketing tasks as required

Education and Experience:
- At least 2 years of previous customer service center experience
- Ability to work with Microsoft (MS) Office, with focus on demonstrated working knowledge of MS Excel and Word
- Ability to work well in time-sensitive situations where customer satisfaction is the goal
- Ability to apply creative problem-solving techniques to situations using sound business judgment
- Excellent verbal and written communication skills
- Ability to multi-task in an effective, timely and professional manner
- Proven ability to apply attention to detail, role-related accuracy and task follow-through
- Willingness to learn new software products such as ACT!
- Bachelor's degree a plus

apply now

saki

The Future in Focus

Field Application Engineer

Saki America Inc., headquartered in Fremont, CA, a leader in automated inspection equipment, seeks two full-time Field Application Engineers (FAE), one in the Fremont headquarters and the other for the Eastern and Southern United States.

The FAE will support the VP of Sales and Service for North America in equipment installation, training, maintenance, and other services at field locations. The FAE will provide technical/customer support and maintain positive relationships with existing and future customers.

Strong analytic abilities and problem-solving skills are a must in order to understand customer applications and troubleshoot issues. The FAE will perform demos and presentations for customers and agents as well as assisting in trade show activities. Candidate must have a minimum of a two-year technical degree, experience in AOI, SPI, and X-ray inspection, and strong verbal and written communication skills. The position requires the ability to travel about three weeks per month. Must be a US citizen and be able to lift up to 40 lbs.

apply now

Career Opportunities

eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor at EPTAC

Job Summary:
We are growing! EPTAC, a leading provider in the electronics training industry is looking for some great people to join our team. 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. 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 an enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Opportunities available across U.S. and Canada, especially in our growing markets of California, Chicago, Minnesota and New England. Some travel involved. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and Skills:
- A love of teaching and an enthusiasm to help others learn new concepts and skills
- Background in electronics manufacturing
- Previous soldering and/or electronics/cable assembly experience
- Previous 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 time, work as often as you like
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now

CHEMCUT
BOUNDLESS INNOVATION | UNBEATABLE PRECISION

Chemical Process Engineer

Chemcut, a leading manufacturer of wet-processing equipment for the manufacture of printed circuit boards for more than 60 years, is seeking a Chemical Process Engineer. This position is located at Chemcut's main facility in State College, Pennsylvania. Applicants should have an associate degree or trade school degree, or 4 years equivalent in chemical process engineering.

Job responsibilities include:
- Developing new industrial processes
- Providing process criteria for both new equipment and modifying existing equipment
- Testing new processes and equipment
- Collecting data required to make improvements and modifications
- Assisting in investigating and trouble-shooting customer process problems
- Ensuring that equipment works to its specification and to appropriate capacities
- Assessing safety and environmental issues
- Coordinating with installation/project engineers
- Ensuring safe working conditions and compliance with health and safety legislation

Key Skills:
- Aptitude for, and interest in chemistry, IT and numeracy
- Analytical thinking
- Commercial awareness
- Ability to perform under pressure
- Communication and teamwork
- Problem-solving

Experience with circuit board processes is a plus.

Contact Arlene at 814-272-2800 or by clicking below.

apply now

Career Opportunities

IPS INTEGRATED PROCESS SYSTEMS INCORPORATED

PCB Equipment Sales

World-class manufacturer of wet process equipment for the PCB and plating industries, Integrated Process Systems Inc. (IPS) is seeking qualified candidates to fill a position in equipment sales. Potential candidates should have:

- Process engineering knowledge in PCB manufacturing
- Outside sales background
- Residency on the West Coast to manage West Coast sales
- Knowledge of wet process equipment
- Sales experience with capital equipment (preferred)

Compensation will include a base salary plus commission, dependent upon experience.

more details

ACE ACCURATE CIRCUIT ENGINEERING

PCB Assembly Supervisor—full time
Accurate Circuit Engineering—Santa Ana, CA

Position Summary: Responsible for all assembly processes to ensure continued growth as directed by management.

Essential Job Functions:

- Create, implement, and supervise in-house manufacturing facility
- Recruit, hire, train, and supervise assembly floor personnel
- Extensive hands on experience with all aspects of PCB assembly
- Understanding of IPC-A-610 standards
- Research and acquire additional assembly resources
- Gather data on product shortages, lead times, price changes, etc.
- Coordinate the assembly activities with sales to ensure 100% on-time delivery
- Create, implement, and supervise daily quality processes to ensure 100% accuracy
- Document, monitor and review progress of the business unit
- Respond to internal and external customers in a timely manner
- Coordinate walk-through, site audits, etc.

Qualifications:

- Minimum 3 years as operations supervisor of electronics assembly house
- 5+ years' experience in the electronics industry
- Previous experience as a quality or operations supervisor preferred
- Ability to solve practical problems using pre-established guidelines
- Strong facility in Microsoft Office applications
- Excellent verbal and written communication skills
- Ability to work with people of diverse backgrounds
- Highly organized/excellent time management skills
- Ability to perform at the highest level in a fast-paced environment
- Valid California driver's license.

apply now

Career Opportunities

Electronics Expert Engineer

Orbotech is looking for an Electronics Expert Engineer to handle various hardware activities, including communication, data path processing, device interfaces and motion, as well as system supporting functions in a multi-disciplinary environment.

What Will Your Job Look Like?

- Providing cutting edge hardware solutions for challenging product line needs
- Developing board design and Logic in VHDL
- Defining and managing interfaces (software, algorithm, mechanics and electricity)
- Successfully integrating hardware with other product disciplines
- Supporting the product needs during and following release

What Do You Need to Succeed?

- BSc in electronics engineering
- At least 5 years of R&D experience in complex board design, mainly FPGA (communication interfaces, DDR controller, algorithm implementation)
- Experience in an Altera/Xilinx development environment
- Experience in ECAD design tools (DxDsigner, ModelSim) is an advantage
- Knowledge in laser interfaces, RF and analog is an advantage

Who We Are

Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

PCB Process Planner

Accurate Circuit Engineering (ACE) is an ISO 9001:2000 certified manufacturer of high-quality PCB prototypes and low-volume production for companies who demand the highest quality in the shortest time possible. ACE is seeking a skilled individual to join our team as a PCB process planner.

Responsibilities will include:

- Planning job travelers based on job release, customer purchasing order, drawings and data files and file upon completion
- Contacting customer for any discrepancies found in data during planning and CAM stage
- Consulting with director of engineering regarding technical difficulties raised by particular jobs
- Informing production manager of special material requirements and quick-turn scheduling
- Generating job material requirement slip and verify with shear clerk materials availability
- Maintaining and updating customer revisions of specifications, drawings, etc.
- Acting as point of contact for customer technical inquiries

Candidate should have knowledge of PCB specifications and fabrication techniques. They should also possess good communication and interpersonal skills for interfacing with customers. Math and technical skills are a must as well as the ability to use office equipment including computers, printers, scanners, etc.

This position requires 3 years of experience in PCB planning and a high school level or higher education.

apply now

Career Opportunities

Altium

Business Development Representative at Altium

New Logo Business Development representatives are highly motivated and hardworking with an upbeat can-do attitude. They work with our New Logo Sales Team to displace our competition in accounts by offering Altium's unified PCB development tools within a defined region.

The New Logo Developer's (NLD) main responsibilities will be qualifying leads and prospecting into competitive lists, searching the web, and utilizing internal sales tools (Inside View, LinkedIn, Marketo, SalesForce) to uncover and work with opportunities for the New Logo Closer to close. They are expected to meet or exceed monthly, quarterly & annual quota.

Responsibilities:

- Develop lead opportunities by collecting information that includes business pains/needs, timelines, authority and project teams, budget, competitive information, etc.
- Aggressively drive daily prospecting calls to build pipeline of prospective clients and occasionally closing smaller deals
- Develop relationships with key partners in their territory to identify new business opportunities
- Plan and prioritize personal sales activities in conjunction with the New Logo Closer, with the goal of achieving sales targets
- Work alongside inside sales teams on specialized projects such as call-out campaigns, promo drives and webinar fulfillment
- Once trained, maintain an in-depth knowledge of Altium products and technologies, competitive products, and industry trends.

apply now

CHEMCUT

BOUNDLESS INNOVATION | UNBEATABLE PRECISION

Field Service Technician

Chemcut, a leading manufacturer of wet-processing equipment for the manufacture of printed circuit boards for more than 60 years, is seeking a high-quality field service technician. This position will require extensive travel, including overseas.

Job responsibilities include:

- Installing and testing Chemcut equipment at the customer's location
- Training customers for proper operation and maintenance
- Providing technical support for problems by diagnosing and repairing mechanical and electrical malfunctions
- Filling out and submitting service call paperwork completely, accurately and in a timely fashion
- Preparing quotes to modify, rebuild, and/or repair Chemcut equipment

Requirements:

- Associates degree or trade school degree, or four years equivalent HVAC/industrial equipment technical experience
- Strong mechanical aptitude and electrical knowledge, along with the ability to troubleshoot PLC control
- Experience with single and three-phase power, low-voltage control circuits and knowledge of AC and DC drives are desirable extra skills

To apply for this position, please apply to Mike Burke, or call 814-272-2800.

apply now

Career Opportunities

Electronics Team Leader

Orbotech is seeking an Electronics Team Leader to join our electronics team, which develops multi-disciplinary systems, including vision/laser, image processing, and control and automation missions.

What Will Your Job Look Like?

- Lead a team of electronics engineers in a multi-disciplinary environment
- Lead electronic activities from requirement phase to development, integration and transfer, to production
- Be the focal point for other disciplines and projects managers
- Maintain and improve existing electronics platforms

What Do You Need to Succeed?

- BSc/MSc in electronic engineering/ computer science from a well-recognized university
- 5+ years' experience in digital board design, high-speed links, computing embedded systems, and HW/SW integration
- 2-3 years' experience in leading a team of engineers
- Solid skills in complex FPGA design with multi-modules
- Solid skills in high-speed board design, DDR3/4, PCIE, USB, IO, and optic links
- Ability to design and execute end-to-end solutions

Who We Are

Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting-edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

Ventec Seeking U.S. Product Manager for tec-speed

Want to work for a globally successful and growing company and help drive that success? As a U.S.-based member of the product and sales team, your focus will be on Ventec's signal integrity materials, tec-speed, one of the most comprehensive range of products in high-speed/low-loss PCB material technology for high reliability and high-speed computing and storage applications. Combining your strong technical PCB manufacturing and design knowledge with commercial acumen, you will offer North American customers (OEMs, buyers, designers, reliability engineers and the people that liaise directly with the PCB manufacturers) advice and solutions for optimum performance, quality and cost.

Skills and abilities required:

- Technical background in PCB manufacturing/ design
- Solid understanding of signal integrity solutions
- Direct sales knowledge and skills
- Excellent oral and written communication skills in English
- Experience in making compelling presentations to small and large audiences
- Proven relationship building skills with partners and virtual teams

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

Please forward your resume to jpattie@ventec-usa.com and mention "U.S. Sales Manager—tec-speed" in the subject line.

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Master Instructor

This position is responsible for IPC and skill-based instruction and certification at the training center as well as training events as assigned by company's sales/operations VP. This position may be part-time, full-time, and/or an independent contractor, depending upon the demand and the individual's situation. Must have the ability to work with little or no supervision and make appropriate and professional decisions. Candidate must have the ability to collaborate with the client managers to continually enhance the training program. Position is responsible for validating the program value and its overall success. Candidate will be trained/certified and recognized by IPC as a Master Instructor. Position requires the input and management of the training records. Will require some travel to client's facilities and other training centers.

For more information, click below.

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Technical Sales Engineer

Positions available in the Chicago area and California

Do you want to advance your career by joining a globally successful and growing world class CCL manufacturer and help drive that success? As a California-based member of the technical sales team, your focus will be on Ventec's core market segments: mil/aero, automotive and medical, offering a full range of high-reliability materials including polyimide, IMS and thermal management products.

Skills and abilities required:

- Drive & Tenacity!
- 7 to 10 years of experience in the PCB industry in engineering and/or manufacturing
- Detail-oriented approach to tasks
- Ability to manage tasks and set goals independently and as part of a team
- Knowledge of MS office products

Full product training will be provided. This is a fantastic opportunity to become part of a successful brand and a leading team with excellent benefits.

Please forward your resume to:

jpattie@ventec-usa.com and mention "Technical Sales Engineer - California Based or Chicago area" in the subject line.

apply now

Career Opportunities

Altium

Application Engineer

The application engineer is the first contact for our customers who have technical questions or issues with our product. We value our customers and wish to provide them with highest quality of technical support.

Key Responsibilities:

- Support customer base through a variety of mediums
- Log, troubleshoot, and provide overall escalation management and technical solutions
- Create various types of topic based content, such as online help, online user guides, video tutorials, knowledge base articles, quick start guides and more
- Distill complex technical information into actionable knowledge that users can understand and apply
- Continually develop and maintain product knowledge

Requirements:

- Understanding of EDA electronic design software, schematic capture and PCB layout software
- Bachelor's degree in electronics engineering or equivalent experience
- Sales engineering and/or support engineering experience
- Circuit simulation and/or signal integrity experience
- Understanding of ECAD/ MCAD market segments
- Understanding of micro controllers, SoC architecture and embedded systems market
- Database experience preferred (i.e., MySQL, PostgreSQL, Microsoft Access, SQL, Server, FileMaker, Oracle, Sybase, dBASE, Clipper, FoxPro) etc.
- Experience with PLM/PDM/MRP/ERP software (Program Lifecycle Management) preferred
- Salesforce experience a plus

Salary based upon experience. Comprehensive benefits package and 401k plan. Openings in USA, UK, and Germany.

For more information, contact Altium.

apply now

MacDermid PERFORMANCE SOLUTIONS

Do you have what it takes?

MacDermid Performance Solutions, a Platform Specialty Products Company, and daughter companies manufacture a broad range of specialty chemicals and materials which are used in multi-step technological processes that enhance the products people use every day. Our innovative materials and processes are creating more opportunities and efficiencies for companies across key industries – including electronics, graphic arts, metal & plastic plating, and offshore oil production. Driving sustainable success for companies around the world, and at every step of the supply chain, takes talent. Strategic thinking. Collaboration. Execution.

The people of MacDermid Performance Solutions stand united by a guiding principle: If it doesn't add value, don't do it. This belief inspires a unique culture where each team member has opportunities to imagine, create, hone and optimize. Do you have what it takes? Join our growing team of over 4,000 professionals across more than 50 countries with openings in research, finance, customer service, production and more.

MacDermid Performance Solutions and its affiliates are Equal Opportunity/ Affirmative Action Employers.

apply now

Career Opportunities

orbotech

FPGA Design Expert

Orbotech is seeking a FPGA Design Expert to join our electronics team, which develops multi-disciplinary systems including vision/laser, image processing and electro-optics.

What Will Your Job Look Like?

- Lead image acquisition and processing activities in the team
- Engage in all aspects of FPGA design activity: requirement phase, coding, synthesizing, verification support and LAB bring up
- Participate in system definitions for current and next generation products
- Collaborate with other teams: SW, algorithm and QA

What Do You Need to Succeed?

- BSc/MSc in Electrical Engineering from a well-recognized university
- Extensive knowledge of VHDL
- 5+ years of FPGA development experience (requirement, architecture, RTL coding, simulation, synthesis, timing analysis, P&R, board level integration and verification)
- Experience in designing and implementing low-latency, high-throughput FPGA designs utilizing PCIe Gen2/3, Gigabit Ethernet, SERDES, DDR3/4
- Experience in complex FPGA such as Altera Stratix-II and Arria 5&10 devices
- Authoring documentation experience such as FPGA specifications and FPGA verification plans

Who We Are

Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting-edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

ARLON

Arlon EMD, located in Rancho Cucamonga, California is currently interviewing candidates for **manufacturing** and **management positions.** All interested candidates should contact Arlon's HR department at 909-987-9533 or fax resumes to 866-812-5847.

Arlon is a major manufacturer of specialty high performance laminate and prepreg materials for use in a wide variety of PCB (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 9001:2008 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customer's requirements.

more details

PROUDLY INTRODUCING OUR PREMIER ISSUE

IPC APEX EXPO Show Coverage Magazine

Special coverage of the IPC Awards and Hall of Fame Inductees

Exclusive attendee and expert Q&A with designers, fabricators, and assemblers

Our columnists put the show into perspective

Our cameras take you to the show floor and beyond

Register Now

Events Calendar

IPC APEX EXPO 2018 Conference and Exhibition ▶
February 27–March 1, 2018
San Diego, California, USA

China International PCB and Assembly Show (CPCA) ▶
March 20–22, 2018
Shanghai, China

KPCA Show 2018 ▶
April 24–26, 2018
Kintex, South Korea

Thailand PCB Expo 2018 ▶
May 10–12, 2018
Bangkok, Thailand

Medical Electronics Symposium 2018 ▶
May 16–18, 2018
Dallas, Texas, USA

IMPACT Washington, D.C. 2018 ▶
May 21–23, 2018
Washington, D.C., USA

2018 EIPC's 50 Years Anniversary Conference ▶
May 31–June 1, 2018
Bonn, Germany

JPCA show 2018 ▶
June 6–8, 2018
Tokyo, Japan

PCB West Conference and Exhibition ▶
September 11–13, 2018
Santa Clara, California, USA

electronica India productronica India ▶
September 26–28, 2018
Bengaluru, India

SMTA International ▶
October 16–17, 2018
Rosemont, Illinois, USA

Additional Event Calendars

PUBLISHER: **BARRY MATTIES**
barry@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

EDITORIAL:
MANAGING EDITOR: **ANDY SHAUGHNESSY**
(404) 806-0508; andy@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@iconnect007.com

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **SHELLY STEIN, MIKE RADOGNA , TOBEY MARSICOVETERE**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © APINAN**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

February 2018, Volume 7, Number 2
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

myiconnect007.com

mediakit.iconnect007.com

EDITORIAL CONTACT

Andy Shaughnessy
andy@iconnect007.com
+1 404.806.0508 GMT-5

SALES CONTACT

Barb Hockaday
barb@iconnect007.com
+1 916 365-1727 GMT-7

www.iconnect007.com