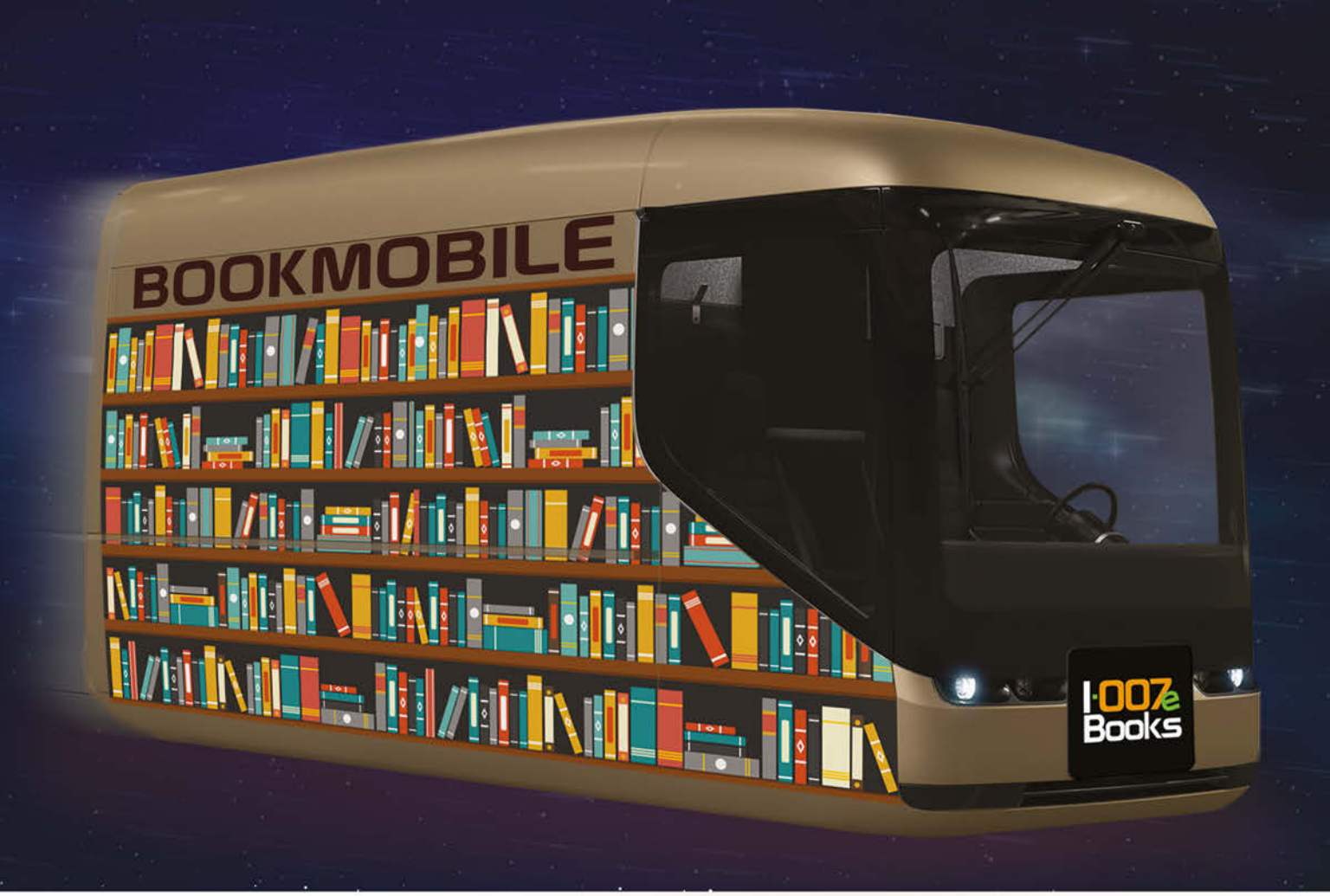
I-Connect007

DECEMBER 2018

DESIGN007 MAGAZINE



Our library is the bookmobile for the 21st century

Registered members of **my I-Connect007** have direct access to our ever-growing library of eBooks, which includes a wide range of topics, from low-temperature soldering and conformal coating in harsh environments to thermal management and data package perfection. Registered users simply download new titles free with the click of a mouse. It's pretty awesome!

I-007eBooks.com

APEX EXPO IPC 2019
Booth #3119

my I-Connect007
GOOD FOR THE INDUSTRY

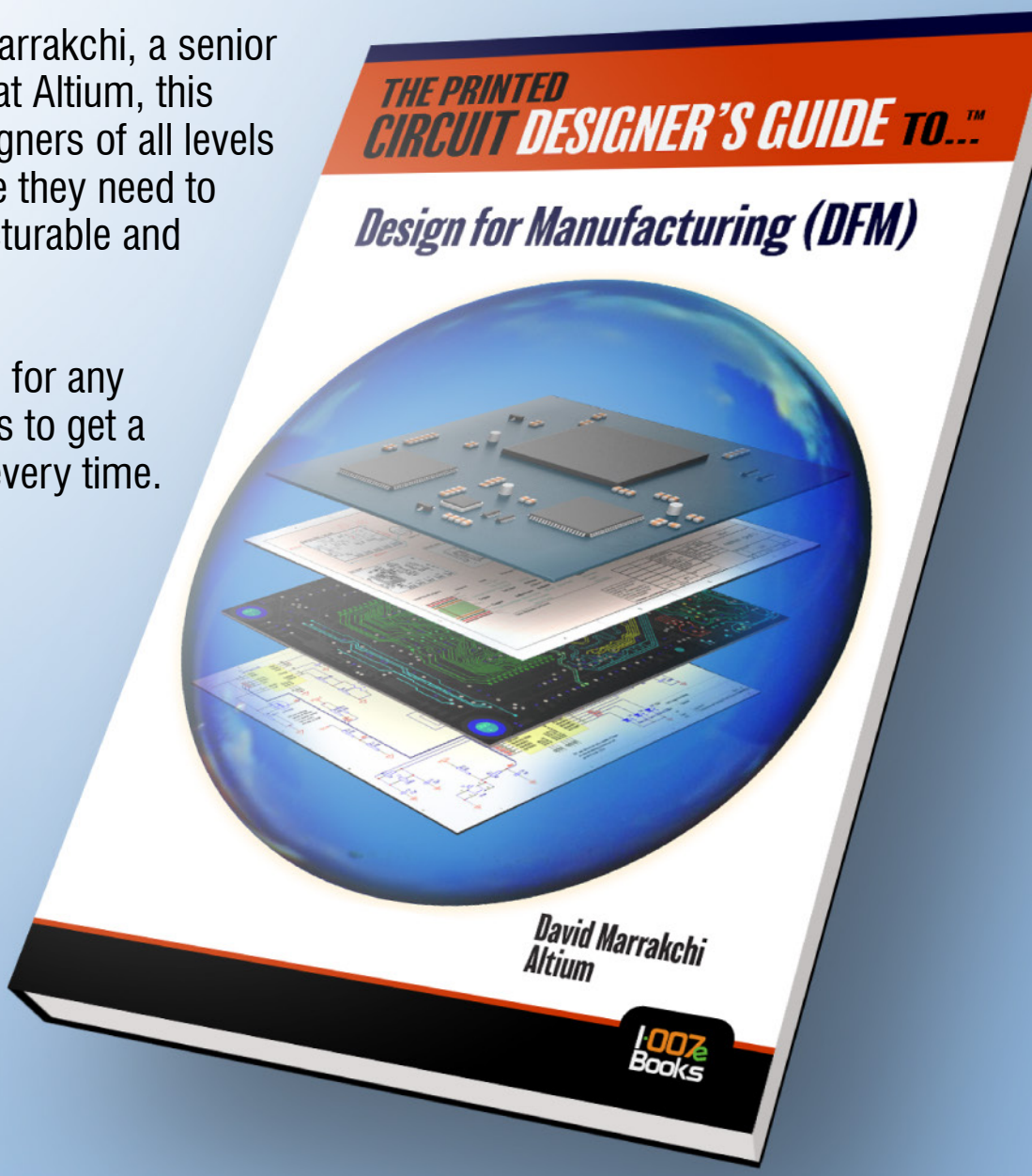
I-007e Books

“A guided tour through the entire DFM process.”

Kelly Dack
CID+, CIT, EPTAC Corporation

Written by David Marrakchi, a senior technical engineer at Altium, this book provides designers of all levels the DFM knowledge they need to produce a manufacturable and functional board.

This is a must-read for any designer who wants to get a good board back, every time.



DOWNLOAD NOW

I-007e Books

I-007ebooks.com/dfm

DESIGN007 MAGAZINE

THE IPC DESIGNERS COUNCIL

The IPC Designers Council is ready for the new year, new members, and maybe even new chapters. They've launched a new column in *Design007 Magazine,* "The Digital Layout," and they have a plan for the future. But they have plenty of work to do.

10 **Birds of a Feather: PCB Carolina and the RTP Designers Council**
Interview by Andy Shaughnessy

18 **Susy Webb: The History and Future of the Designers Council**
Interview by Andy Shaughnessy

22 **The Designers Council: A Chapter Primer From the Ground Up**
by Scott McCurdy

26 **Chapter Highlights and Certification Successes**
Column by Steph Chavez

32 **Why Should You Join the Designers Council?**
by Judy Warner

Also in this Issue: IPC APEX EXPO 2019 PREVIEW

Don't miss our special pre-show coverage of IPC APEX EXPO, scheduled for January 29–31, 2019, at the San Diego Convention Center. There's a lot going on at this show!

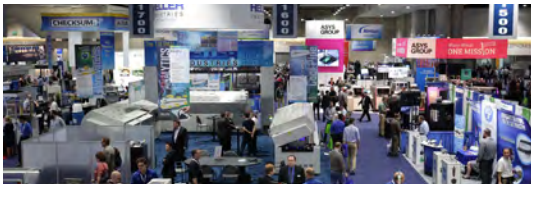
36 **IPC APEX EXPO 2019: Where Technology's Future Comes Together**
Column by John Mitchell

40 **IPC APEX EXPO 2019 Schedules, Stats, and Programs**
by Nolan Johnson

44 **Shopping at IPC APEX EXPO: Evolutionary or Revolutionary Products?**
by Andy Shaughnessy

46 **Dan Beaulieu on Making the Most of a Trade Show**
Interview by Barry Matties

48 **In the Studio: *Real Time with...IPC***
by Andy Shaughnessy



We have all the tools needed for advanced PCB solutions.

With the right tools, you can build anything. ASC has the experts, technology and experience necessary to build anything you might need, all under one roof.

More about our advanced solutions

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

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

DESIGN007 MAGAZINE

SHORTS:

25 Cutting and Pasting with Graphene

31 Supercomputers Without Wasted Heat

45 Sensitive Robots Feel the Strain

47 High-temperature Electronics? That's Hot

62 Pushing Lithium Ion Batteries to the Next Performance Level

89 Foldable Smartphones to Be Launched in 2019

HIGHLIGHTS:

50 PCB007

74 MilAero007

90 Top 10 PCBDesign007

DEPARTMENTS:

93 Career Opportunities

104 Events Calendar

105 Advertiser Index & Masthead

ARTICLES:

76 Altium Prepares for Munich Show as Growth Continues
Interview with Chris Donato

80 A Fractal Conversation with Jim Howard and Greg Lucas
by Barry Matties and Andy Shaughnessy

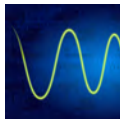
84 Fractals: New Tech and IP for Designing 5G PCBs
by Greg Lucas and Jim Howard

COLUMNS:

8 The Designers Council is Not Resting on Its Laurels
by Andy Shaughnessy

52 10 Fundamental Rules of High-Speed PCB Design, Part 4
by Barry Olney

58 Embedding Components, Part 6: Preparation for Active Semiconductor Elements
by Vern Solberg



64 Skin Depth and Its Interaction with Final Plated Finishes
by John Coonrod

66 Key Guidelines for Clean Schematic Designs
by Bob Tise and Dave Baker

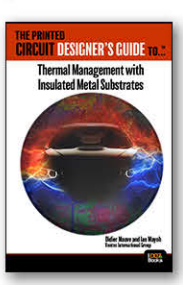
70 Top Five Tips to Protect PCBs from Harsh Environments
by Alistair Little

Flexible, reliable Supply Chain Solutions

high quality copper clad laminates & prepreg bonding materials

ventec
INTERNATIONAL GROUP
騰輝電子

Ventec International Group is a world leader in the production of high quality and high performance copper clad laminates and prepregs, with full R&D and product development capabilities. Our global distribution means we can readily supply our products to all markets in all geographic areas.



Get your free copy of *The Printed Circuit Designer's Guide to...Thermal Management with Insulated Metal Substrates*.

DOWNLOAD NOW

PLEASE VISIT US AT
APEX EXPO IPC 2019
BOOTH #417

Ventec International Group
179 Ward Hill Ave,
Ward Hill, MA 01835,
United States
T: 978-521-9700
E: ordersCA@ventec-usa.com
Follow @VentecLaminates

www.**venteclaminates**.com

Ventec - wherever technology takes you, we deliver

The Designers Council Is Not Resting on Its Laurels

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

There was a time when PCB designers were the odd man out. They're still a little odd, actually, but that's another story. In the 1970s and '80s, unless you worked in a hotbed of high-tech like Silicon Valley, you may not have known too many other PCB designers.

That's what happens when you work in a career that isn't its own industry. And it was usually one or two designers working with an office full of EEs, none of whom understood exactly what you did anyway.

Back then, it was easy for a designer to feel isolated.

But the IPC Designers Council has done quite a bit to erase that feeling of being out of touch. Founded in 1992, the Designers Council is a group created "by designers, for designers" with over 1,000 members and 33 chapters around the globe. The DC offers networking opportunities, technical education, and certification such as the certified interconnect designer program (CID and CID+).

There is one important distinction about the DC. Unlike IPC membership, which is awarded only to companies, DC members are all individuals. There is no cost to join the Designers Council.

One of the biggest benefits of being involved with the DC is networking; you never know who you're going to run into at a meeting. Some of the monthly "lunch-and-learn" chapter meetings draw a crowd of PCB designers and technologists—the Orange County and San Diego chapters routinely boast 50 or more attendees—who listen to guest presenters discuss some of the problems that designers face

every day. If you live near a Designers Council chapter, make it a point to attend the next meeting. There's a small fee to cover the cost of lunch, but you might learn something that helps you design your next board.

No, the DC isn't perfect. After all, it's made up of PCB designers! I'm kidding. In fact, bashing the Designers Council is almost a sport for some designers. But if you want to affect any sort of change, don't just talk smack about the group; get involved! And if you're not located near a DC chapter, why not start your own chapter? Our new columnist Stephen Chavez is a Designers Council Executive Board member, and he'd be happy to help you launch a chapter in your area.

PCB Carolina 2018 exhibition was held in Raleigh, North Carolina.

Best of all, you don't have to be a designer or EE. Anyone who wants to know more about PCB design is welcome to join or attend DC meetings.

As we found with our first feature interview, one DC chapter has actually developed its own trade show. During PCB Carolina, I interviewed show managers Randy Faucette, Tony Cosentino, and Lance Olive. They discuss how this show grew out of the RTP Designers Council chapter in Raleigh, and they track the show's growth over the years from a small meeting in a conference room to a show that now draws 80 exhibitors and 1,000 attendees.

Next, design instructor Susy Webb discusses how she first got involved with the Designers Council, and why she thinks designers should join their local chapter. She also makes some predictions about the role of the group in the future, especially if more engineers begin doing layout. And Orange County Chapter President Scott McCurdy details how he grew the OC chapter into one of the most popular in the U.S. with meetings that draw "rock star" speakers and a roomful of attendees.

Columnist Stephen Chavez highlights the work of the San Diego and RTP chapters, and some of the Designers Council events set for the upcoming new year. Also, Altium's Judy Warner explains how she succumbed to years of invitations and finally attended her first Designers Council meeting a few years ago, and why you should attend one near you as well.

Speaking of Altium, we also bring you an interview with VP Chris Donato who gives us a preview of the January AltiumLive event in Munich and traces the company's impressive growth over the past five years. Next, veteran engineers Greg Lucas and Jim Howard unveil a potentially revolutionary process—fractal design—which can reduce edge noise on a PCB and costs absolutely nothing to implement. You'll also want to read our interview with them.

We also have columns from our regular contributors Barry Olney, Vern Solberg, John Coonrod, Alistair Little, and Bob Tise and Dave Baker.

Our December issue wouldn't be complete without our special pre-show coverage of IPC APEX EXPO scheduled for January 29–31, 2019, at the San Diego Convention Center. Check it out. There's a lot going on at this show.

I can't believe another year has gone by, and I still don't have a flying car. Maybe next year! Have a great holiday, and I'll see you in 2019.

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.


Lance Olive


Randy Faucette


Tony Cosentino

Birds of a Feather: PCB Carolina and the RTP Designers Council

Feature by Andy Shaughnessy
I-CONNECT007

The PCB Carolina show in Raleigh, North Carolina, has grown quite a bit in the past decade from a local show to a regional event that now draws attendees and exhibitors from all over the U.S.

At PCB Carolina 2018, I met with show founders Tony Cosentino, Randy Faucette, and Lance Olive, who are all employees at the Better Boards service bureau in nearby Cary. I asked the trio to discuss the show's history, its relationship to the Research Triangle Park (RTP) North Carolina Chapter of the IPC Designers Council, as well as the show's tremendous growth over the past few years.

Andy Shaughnessy: I'm here at PCB Carolina. Tony, why don't you start and tell us about the RTP Chapter of the IPC Designers Council.

Tony Cosentino: I got involved with the Designer Council in 2002. I was on the board of directors for a year, then ran for VP and did that for a year. Then, I took on the president spot.

I learned about the mission of the Designers Council from the leaders that came before me. The mission is to network with our industry, promote our business (PCB design) as a profession, and teach people technical skills and share information.

We've taken that to heart and offer as many technical opportunities as we can for as little a cost as we can to keep the community engaged. It's all about networking.

Shaughnessy: Now, tell us how the RTP chapter figures into the launch of PCB Carolina.

Cosentino: Around 2000, in the early years of the IPC Designers Council, our local RTP chapter put on very small trade shows, about two hours long with maybe eight or 10 vendors, and it was to get PCB design software vendors and manufacturers in front of the Designers Council members, so that they could see new technologies, capabilities, and roadmaps.

That grew into something bigger over the years. In 2005, we came up with the idea of growing this into a trade show, and in 2008, we moved into the Raleigh Convention Center.

Candor
Circuit Boards

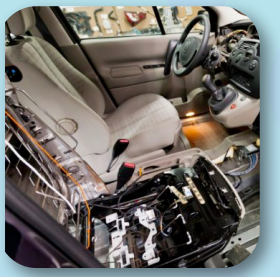
Innovative solutions for complex circuit board manufacturing

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

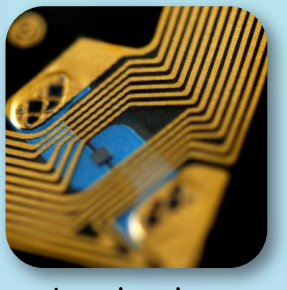
RF/Microwave PCBs

Power Electronics

High-Speed Data



Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

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

Randy came up with the name PCB Carolina. It was his baby from the start. Maybe I'll let Randy speak to that.

Randy Faucette: Sure. We took this show from something small—just meeting space at a local hotel—and moved to the convention center, which had more space. That meant we wouldn't be limited to the main software companies and a couple of fabrication shops, but could really expand it to benefit all of the engineering disciplines and put companies that had special products or services in front of potential customers.

We wanted to do it without large, expensive booth space to keep it to more of a tabletop show where the small million-dollar company could be represented next to a billion-dollar company. There are many local companies in this very high-tech area. There wasn't a PCB show here in the RTP area, so we felt like there was an opportunity to do something about that.

There are a lot of small companies here that just can't afford the big national shows. When I engaged my customers, I would hear, "Well, you know, I use this company in Ohio or Maryland." And I would say, "You know, there is a company just around the corner that does that, right? We have that here in this city." There was an opportunity for us to let the local community know that there are products and services in our backyard. With local companies, turn times are shortened without worry about shipping times. It's hard to put a value on face-to-face interactions with your vendor.

After growing the tradeshow, the next challenge was getting people to attend. The value of our trade show needed to be there so attendees felt like, "Yes, I can put my project on hold and go to this show." And we need their supervisors to say, "Yes, I want to shut down the department and send them to PCB Carolina." So, we needed technical content and other benefits to really draw attendees. Over the years, we developed a nice formula for that. The more success we had, the more the vendors saw that the attendees were quality; they weren't people just here to pick up a trinket.

Shaughnessy: What year was the first "big" show—the first time you had a show that wasn't just in a conference room?

Faucette: I think 2008 was probably the first step out into something big when we really had to have catering and needed to start printing some materials. We needed to print handouts so that people understand everything that was offered.

Shaughnessy: Then, you get into the whole logistics of show management.

Gary Ferrari speaking at PCB Carolina.

Faucette: That's right.

Cosentino: Once we realized it was more than just PCB design, we wanted electrical engineers, mechanical engineers, quality managers, all types of analysis engineers, and all the software vendors to come. We found out the market is really big, so why not invite all the vendors that make up the whole value chain from the creation of the CAD tool all the way to product delivery? We bring all of those to this show. Then, you have to offer technical content; you have to teach some-

thing important. Thus, technical sessions that are valuable and free bring people out.

Shaughnessy: This year, speakers included Dr. Bruce Archambeault, Susy Webb, Rick Hartley, and Gary Ferrari. That's a pretty good group of instructors.

Cosentino: We're blessed to have so many people willing to volunteer their time. We still have very competitive prices for the booths.

Lance Olive: As Randy said, if we don't, then how does everybody have an even playing field? We want the small guy to have the same advantage as the big guy.

Shaughnessy: Since this began, Lance, I know you had some data on how things have grown over the years.

Olive: That's right. I came from the software industry initially. I was a software engineer and a manager at Cisco. Then, I began working on my MBA. I wanted to run a small business. I was having lunch with my friend Randy, who was killing himself trying to keep his company productive and keep up with his deadlines. He said, "Why don't you quit Cisco and come run Better Boards' business side?" That way, Randy could focus on what he does really well, which is PCB design.

In 2012, Randy moved the show to the McKimmon Center here at North Carolina (NC) State for the first time. I saw the planning and execution of the show that year, and having recently come off of my MBA studies, I realized there was a big opportunity here, and a lot of what needed to be tapped wasn't being tapped yet. I said, "We should do things a little differently in 2013." We structured the planning of the show like a project. I managed it like a project, set up tasks, assigned tasks to people, and we were able to get things done a lot earlier, which reduced the costs. Planning ahead removes expediting.

Also, it gave us a little more time to reach out. In 2014, the person who was doing the outreach and mailers wanted to resign and turn it over, so I took that over. I took all of the various spreadsheets and emails and created one consolidated list for marketing. I began to manage that list and market to it actively.

I cultivated that list, and over time grew it from a few hundred to 2,000 quality email addresses. I also removed 1,500 email addresses that were lower quality. I could go around saying that I have almost 4,000 email addresses, but really, I have 2,000 quality email addresses. The reason that's important is that when I send my marketing emails out to people, I tap quality people who want to be informed, so it's not spam. Every year, I cultivate, renew, and refresh that list. I've also structured these emails so that they can be forwarded to coworkers or anyone who is interested. One of the best ways to find these second- and third-tier participants and attendees is to have their coworkers reach out to them.

One of the best ways to find these second- and third-tier participants and attendees is to have their coworkers reach out to them.

In 2012, we had 330 registrations. This year we've tripled that. We're pushing 970 in 2018. In terms of exhibitor growths, we've gone from about 36 in 2012 to 78 tabletops this year. I equate this growth to the two pedals on a bicycle. You need attendees so that the exhibitors feel like they get their money's worth. Then, you need the exhibitors there, so the attendees find the value in coming.

I told Randy and Tony, who did the outreach for the exhibitors, "Go ahead and tell them that we're going to set a new record for attendees this year. If they're on the fence, they'll come." It worked! Vendors wanted to exhibit. Then, I followed that with the press release that tells all of the vendors we set new records for attend-

The crowd mingles at Mentor's open bar.

ees, which then affirms in their minds that this was a valuable show, and if each vendor goes home with one quality contact at the end of the day, they make back their show money easily. Randy, how much does it cost to exhibit?

Faucette: We try to keep it under $1,000.

Olive: For under $1,000 for a tabletop space, and if you get one customer who brings in a $10,000 deal, you've already made your money back easily. Imagine if you come back with five good contacts.

One of the things that's really encouraging is that 40% of the people this year did not attend last year. That's a huge number. In fact, the first time I ran the report, I was not sure I did it right, and had to go back and check because it didn't feel like the right number. Last year, 35% of the attendees were new. Even as people attrition out of their companies, retire, or move to other states, we're replacing them with new people and different age groups too, which is encouraging.

Cosentino: As part of the keynote address, I wanted a pulse of the room. I asked, "How many people are here for the first time?" About 25% of the people raised their hands. I asked, "How many people came here from more than 50 miles away?" Probably 25% raised their hands. Then, I asked, "How many people here are 30 years or younger?" About 15% of the room raised their hands, which I thought was great. We're getting really old. I am all gray, Lance has half his hair, and Randy has two bad knees!

Shaughnessy: I think 15% is high for young people in the industry overall.

Cosentino: Young people in this industry are important. We would not be able to do a show like this without a bunch of volunteers. We reach out to local community colleges and universities and ask students to get involved with the show, volunteer, and take on some tasks, whether it's monitoring a door, working the front desk, handing out pamphlets, or carrying boxes. Then, they have an opportunity to meet the vendors. They're all students in the electronics industry, not English grads.

Shaughnessy: They're not drama majors.

Cosentino: No, and they're interested in what they see. They are studying to be electronics technicians or working toward an engineering degree. We also have professors from colleges who present papers. These students come in, get to see the industry vendors, gain exposure

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 deliver an integrated manufacturing data preparation solution and the most superior environment to visualize, verify and document a printed circuit board.

Visit downstreamtech.com
or call 800-535-3226

Or stop by booth #1716

APEX EXPO IPC 2019

DownStream Technologies

to the industry, and get to attend technical presentations. When a vendor on the floor says, "This is a great turnout, but there are a lot of kids here," I say, "Well, that's great, isn't it?" These are college students who are going to replace us later; it's a good thing!

Shaughnessy: And doing it at NC State has all kinds of benefits.

Cosentino: Absolutely. Since 2012, we've grown from filling the exhibit hall and maybe one classroom to now having four classrooms with technical sessions in each room. We've rented out the entire building this year, and we've overflowed out of the exhibit hall into the lobby. It's quite a nice growth, and we have been able to sustain it. This is in the continuing education part of NC State's campus, and they like having us here. We had 16 technical sessions, not counting the IPC Technical Education classes and CID certifications, and we have two hands-on soldering workshops onsite as well conducted by Circuit Technology.

Faucette: NC State's McKimmon Center is focused on continuing education, so they like our event as much as we like this venue. Since the costs are reasonable, we distribute the costs back out, so that the vendors get in at a very affordable rate. Even with all the free food, T-shirt, giveaways, and t door prizes, it all works out to be a very affordable opportunity for both exhibitors and attendees.

Shaughnessy: How do you have the show set up? Are you a nonprofit?

Olive: It's completely run by the RTP chapter of the Designers Council, which is a nonprofit group. The more exhibitors that we have and the more revenue that comes in, the more we're able to purchase better quality food, T-shirts, and printed guides. We have a few subcontractors—photographers, and caterers, etc.—but we pay all the bills and try to put a little bit of money back into our chapter so that we can pay to have speakers come in for the quarterly chapter meetings.

Shaughnessy: So, it's all via the RTP IPC Designers Council Chapter?

Olive: The finances for the show flow through the chapter, which is why Randy was so involved for many years being the treasurer of the RTP chapter. I started in 2012 with Better Boards. Tony was hired in 2013. Better Boards does do a lot to help make sure the show stays in production.

We find a lot of value in executing this show. As I described this to someone, even though Better Boards does work very hard, we see this as an effort to "rise the tide" for our area not just in Raleigh, Durham, or even Charlotte. It's beyond even North Carolina now. I can't tell you how many people I've met who came up from the greater Atlanta area at this show.

Shaughnessy: A few companies exhibiting came from California.

Olive: Absolutely, so people are starting to come from farther distances because they realize the gravitas that PCB Carolina has created is now worth making the trip for.

Shaughnessy: It's not just local reps.

Olive: That's right. It's a good and necessary mix. Due to the event's success, I think more things are starting to be scheduled adjacent to our event so that people can travel and have multiple benefits.

Shaughnessy: The CID class was here last year. Did you do that this year?

Cosentino: Yes. The CID certifications were held during the four days following our show—three days of lecture, and the fourth day is testing for CID and CID + .

Olive: I use the word gravitas. When IPC wants to do an event the night before, and there's a training exercise the day after and a soldering workshop during the event, it's like moons around a planet. We do this show at a very predictable time of year—usually in the first week

of November—so it's very predictable for people to put it on their calendar.

Cosentino: Now, as the president of the Designer Council chapter, I'm less interested in the treasury and marketing because that's not really my bag. I'm very interested in networking and education. Without the show, I think this area would be worse off for it.

Olive: It's definitely put the area on the map for sure.

Shaughnessy: Yes, but this is good.

Olive: We're actually growing at a rate of 100 attendees every year. How much more can we grow? We really don't want to move from this facility. The McKimmon Center is ideal for us.

Cosentino: We keep our fees reasonable, and the McKimmon Center helps enable that. Our fees cover all of our costs with a little left over to afford to bring in speakers for our chapter meetings the following year.

Olive: Yes, and those fees are put to good use. When you register and show up, you get a badge, tote, T-shirt, show guide, pen, and koozie. And don't forget, we feed you all day. A hot breakfast, outstanding lunch with barbecue, brisket, fried chicken, and banana pudding, and the evening reception is coming with more food and craft beer from local breweries.

Cosentino: We are grateful to have some sponsors cover some of the major costs. Better Boards covered the breakfast, Mentor, a Siemens Business, covered the costs for the evening reception, and PalPilot donated the grand prize—a 65" TV. I hope the winner brought a big truck; not kidding.

Olive: If we look forward, we'd like the attendee count to grow, and I believe that we're getting very close to fully tapping the local market, but I don't mind if we draw from nearby states and grow that past 1,000 registrations next year.

Faucette: What we don't want to do is leave this venue. We love being at McKimmon Center. The staff are awesome to work with, and the caterers are great. We are filling this space though. We've expanded the showroom floor and the number of tables several times. We're maxing out the number of tables for exhibitors.

One of our challenges is assigning the technical schedule. The four classrooms are not the same sizes. We try to put what we think will be the most attended sessions in the largest room, but you just never know what will happen on show day. Is it the topic or the speaker that draws the crowd? We also don't want competing topics at the same times. We try for an electrical track, a mechanical track, a manufacturing track, then usually a miscellaneous room (compliance, testing, PCB 101, etc.).

We have a converged on a good formula for PCB Carolina—a large vendor exhibition, multiple technical sessions, many attendees, food all day, and giveaways in a great venue. The cost is reasonable for vendors, and the event is free for all attendees—even parking. What more could you want from a trade show?

Shaughnessy: Cool. Well, thank you all for doing this.

Faucette: Thank you for the opportunity.

Cosentino: We appreciate it.

Olive: We'll see you next year. **DESIGN007**

SUSY WEBB: The History and Future of the Designers Council

by Andy Shaughnessy
I-CONNECT007

When we started planning this issue on the IPC Designers Council, I knew I'd have to speak with design instructor Susy Webb, a longtime DC member and currently an executive board officer. I asked Susy to discuss how she first got involved with the DC, why designers should join their local chapter, and what the future holds for this group.

Andy Shaughnessy: Tell us how you first became involved with the IPC Designers Council. Why did you decide to get involved with the DC?

Susy Webb: Houston had already started a chapter of the IPC Designers Council when I got involved. I joined because I always had the desire to learn more about the PCB design profession, and getting together with other people who felt the same was very enticing. When I happened to mention in a meeting that we should bring in speakers to teach us all, they appointed me to that task! It became a real growing experience for me.

Shaughnessy: What were the early DC meetings like?

Webb: What the early meetings lacked in knowledge, they definitely had in passion! People understood changes were coming to the designing of boards because of the new high-speed and signal integrity issues, and they were hungry to learn about them. Our first out-of-town speaker was Rick Hartley, and we had over 100 people come out in the middle of a tropical storm to learn from him.

Shaughnessy: What are some of the benefits a designer can gain by joining the DC? I know that IPC stopped charging $50 to join because so many designers said they couldn't afford to spend that much to network and help advance their careers.

Webb: There have always been some fairly good benefits. In the earlier days, one could attend

APCT

Passion | Commitment | Trust

"Your Lead Times Are What?!"

APCT Saves Customers Time & Money With Industry Leading Cycle Times

Printed Circuit Board Solutions

Sometimes going with the big guys isn't always in your best interest. Large enough to handle any requirement, yet small enough to remain quick and agile, APCT offers consistently best in class lead times. Wondering where to place that high-profile, quick turn project?

APCT, Delivers World Class Lead Times!

CYCLE TIMES	APCT Santa Clara	APCT Wallingford	APCT Orange County	APCT Cirtech
Through-Hole Technology				
Standard Lead Time	15 Days	15 Days	15 Days	4 - 6 Weeks
2 - 10 Layers Expedited Lead Time	24 Hours	48 Hours	24 Hours	-
12 - 24 Layers Expedited Lead Time	48 Hours	72 Hours	48 Hours	-
Filled Via in Pad Technology				
Standard Lead Time	15 Days	15 Days	15 Days	20 Days
Expedited Lead Time	2 - 3 Days	4 - 5 Days	2 - 3 Days	5 - 7 Days
HDI Technology				
2x Lam Cycles Standard Lead Time	15 Days	20 Days	20 Days	20 Days
2x Lam Cycles Expedited Lead Time	3 - 5 Days	5 Days	3 - 5 Days	10 Days
3x Lam Cycles Standard Lead Time	20 Days	20 Days	20 Days	-
3x Lam Cycles Expedited Lead Time	7 Days	10 Days	7 Days	-
4x Lam Cycles Standard Lead Time	25 Days	-	25 Days	-
4x Lam Cycles Expedited Lead Time	10 Days	-	10 Days	-

APCT.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

IPC functions for a discount or ask questions of industry experts on the DC forums. More recently, the benefits include classes brought to the local area by the chapter, meeting discussions on timely topics with others sharing experiences, and making friendships that carry over into other parts of our daily lives.

Shaughnessy: You're on the executive board for the Designers Council. What exactly does the board do?

Webb: The executive board as a group is the advocate for the DC to IPC in general. We encourage chapter organization and participation, brainstorm ways to encourage new designers to get into the industry, give our thoughts to IPC about issues important to designers, discuss ways to enrich CID and CID+ courses, and most recently, we will be reporting through the industry's design magazines what other chapters are doing and what classes are available so that all might use those same ideas.

Shaughnessy: If you could change anything about the Designers Council, what would you change? Or what needs to change, or be done differently, in your opinion?

Webb: There are so many issues related to new products that are the direct responsibility of the PCB designers. They make decisions that determine if the boards will work electrically, how easily their information is conveyed to and understood by the fabricator, whether the board can be manufactured with high yield at a reasonable cost, and if it can be checked, tested, and used in the field with high reliability. I believe it gives people a sense of pride to make the complex decisions that guide them to create a new product that meets those goals. To that end, I would like to see a true dedication on the part of the organizations out there to understand and meet the needs of designers because it ends up affecting everyone down the line.

Shaughnessy: What's next for the Designers Council in the next 5–10 years?

Webb: What's next will mostly depend on who the designer of the future is. Many are saying that engineers will be designing their own boards, and that may be true. But since engineers won't be designing on a regular basis, I believe there will still need to be dedicated designers too. Since many of the longtime mentors in the field will be retiring in the next 5–10 years, there has to be a way to reach all of those engineers and designers with the newest technologies and methodologies. Conferences, users groups and groups like the Designers Council can do that with dedication and leadership provided by the organizations they represent.

What's next will mostly depend on who the designer of the future is.

Shaughnessy: Thanks for your time, Susy. I'll see you on the road again.

Webb: Thank you, Andy. Always a pleasure. **DESIGN007**

The Designers Council: A Chapter Primer From the Ground Up

Feature by Scott McCurdy
FREEDOM CAD

I went to my first IPC Designers Council meeting in the fall of 2002. I was selling PCB fabrication a year after my company, McCurdy Circuits, ceased operations—one of the many casualties in the electronics industry following the dotcom collapse.

Thinking it might be a way to make new contacts with PCB designers in Orange County, I attended a couple of Designers Council meetings. The chapter president at that time was Paul Fleming who asked me to be part of his steering committee because he learned that I had spent decades in fabrication. Within a few months, he had gotten a job transfer to Arizona, so he twisted my arm to take over as the chapter president. I agreed, but little did I know that this would become a major turning point in my PCB career.

After a couple of years of building up our chapter, I took a job selling PCB design layout services, so I was suddenly smack dab in the middle of the design world. This was after 30+ years of building boards and receiving so many layouts wrought with terrible design practices that I would often exclaim that they had been created by designers with "no adult supervision" or the faintest idea of how boards were to be made.

Now, I had a platform to help put some real meaning behind DFM. Hoping to bring designers together with fabricators, assemblers, CAD vendors, and a variety of industry suppliers and gurus, our chapter officers came up with topics and speakers to provide educational opportunities at our chapter meetings. We wanted to inspire our audience to expand their knowledge and become better designers. It became a real passion for me.

In the ensuing 15+ years as president of the Orange County Designers Council chapter, we've grown to become one of the largest chapters in the country. To date, we've held over 60 events. We hold quarterly "lunch 'n learn" meetings, and with the help of our chapter officers, we've recruited terrific speakers covering a wide range of topics that are interesting and educational for our audience of designers and other PCB professionals. Our meetings typically draw 45–75 attendees, and

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

sometimes many more if it's a really great topic or a "rock star" speaker.

So, why do designers show up and make our chapter so successful? I believe there are several answers to that question.

1. Continuing Education

PCBs get more complex every year. Manufacturing processes advance as the fabricators improve their capabilities to meet ongoing technical challenges. A designer needs continue to learn more and more to have the skills to meet these challenges. Those who attend our meetings receive copies of the speakers' slide presentations to build their library of technical information to refer to and can reach out to that speaker in the future. You don't know what you don't know, so Designers Council meetings are a great resource for learning.

2. Networking

It's inspiring for a designer to sit in a room with 50–60 people who have the same job that they do. Our designer attendees often run into co-workers from the past, and they'll sit next to other designers during lunch and chat a bit. They take notice of the other designers who ask the speaker interesting questions and seek them out after the meeting to swap contact info. Quite a few fabricators and assemblers attend our meetings, so it's also a great opportunity to meet them. Take the initiative to come early or hang around for a bit after the meeting to mingle and meet other designers or ask questions of our speakers. We provide badges to all attendees that include your name and your company's name, which is helpful for meeting people.

3. Career Advancement

No one is looking after your career but you. Designers must make an effort to improve their understanding of design and manufacturing technology to continue being relevant. Attending Designers Council meetings is a very good way to learn and become inspired, knowledgeable, and skilled. IPC also offers a program for designer certification. I challenge designers to earn their CID, and ultimately strive for your advanced certification (CID+). No employer wants to hire a designer who is just "connecting the dots" or one who's trapped using obscure CAD tools. These events help designers keep their eyes open for new technologies and opportunities.

4. Employment Opportunities

At our meetings, I always make a point of asking the audience if their companies are hiring designers and if any designer is looking for a job. It warms my heart that dozens of designers have found a new job as a result of being at our meetings or contacting me and asking me to keep my ears open. There's no reason for a skilled designer to be out of a job when you have networking connections via your Designers Council chapter.

The Future

The points I've made here are just a few of the ways that I believe our chapter members have benefited from being part of the Designers Council, but there are lots of other ideas that we should try to address in the future. We need designers to have a bigger voice in what they would like to learn. All chapter leaders struggle with finding good speakers and relevant topics. We need input from the industry, designers, IPC, and the media. Do you have any ideas? If so, we'd like to hear from you.

What about the future of designers? I think about this all the time. I look around the room at our Designers Council meetings and don't see many young designers. The number of designers who are retiring or approaching retirement age is huge, and "greybearding" is going to leave a lot of CAD seats empty. All of today's designers should think about how we can find and promote the fact that PCB design is a very good career choice, is in demand, and is capable of providing a nice income once you acquire the skills.

I don't have the answers, but I'll throw some thoughts out there. I'm certain that there are engineering students who should be introduced to the layout profession where the puzzle solvers could use that electrical theory in their toolbox. What can IPC do to raise awareness of PCB design in schools, colleges, and universities? Can we find ways to put young designers under the wings of older, experienced designers as "masters" to mentor these apprentices?

OEMs are already struggling to find experienced designer talent, so it would be in their best interest to start training the next generation of designers. IPC could support chapters by arranging for videos to be made of the speakers' presentations at Designers Council chapter events, which could be made available to other chapters and designers who don't have chapters local to their areas. These videos could also be made available at little or no cost to share with new designers to help educate them too.

If our industry doesn't step up to find ways to recruit and train the next generation of designers, we run the risk of these jobs going away to countries other than the U.S. where the wages are lower. We desperately need to expand the number of designers here and build a stronger future for those of us in this industry. I believe IPC and Designers Council chapters can play a role in helping to educate and inspire better designers for the future ahead. **DESIGN007**

Scott McCurdy is the director of sales and marketing for Freedom CAD Services Inc. and can be reached at scott.mccurdy@freedomcad.com.

Cutting and Pasting with Graphene

To date, it has proved very difficult to convert the promises of the miracle material graphene into practical applications. Amedeo Bellunato, Ph.D. candidate at the Leiden Institute of Chemistry, has developed a method of cutting graphene into smaller fragments using a diamond knife. He can then construct nanostructures from the fragments.

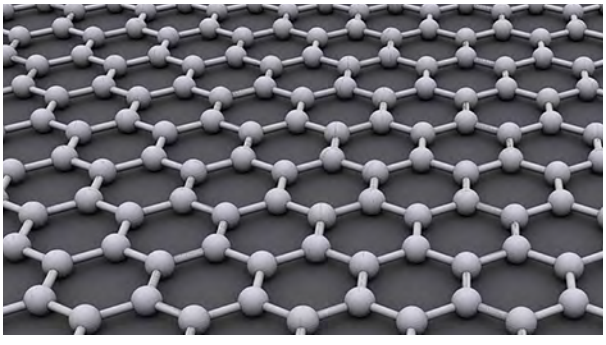
Graphene is a honeycomb structure of carbon atoms just a single atom thick. After its discovery in 2004, it seemed to be the ideal basic material for nanotechnology applications: it is super strong and it is an exceptionally good conductor of both heat and electricity.

However, in his dissertation, Bellunato states that making such nanostructures is still an extremely complex production process that does not lend itself well to serial production. Also, it has proven almost impossible to selectively 'functionalize' graphene chemically (i.e., to connect other chemical elements such as oxygen or nitrogen atoms) to the edges of a graphene nanostructure. It is important to be able to do this to make graphene into a versatile nanomaterial with multiple applications.

Inspired by earlier experiments, Bellunato decided to take a different approach, namely to take a sandwich of plastic and metal with a layer of graphene in the middle, and to literally cut it into fragments. He does this using a microtome, diamond knife that can cut fragments with nanometre precision.

The unconventional technique that he developed will not primarily be used in consumer products, he expects, but rather in advanced research instruments.

(Source: University of Leiden)



Chapter Highlights and Certification Successes

The Digital Layout

Feature Column by Steph Chavez, CIT, CID+, IPC DESIGNERS COUNCIL

This month's column highlights the San Diego chapter, which is also where IPC APEX EXPO 2019 will be held at the end of January. You will also find updates on recent CID and CID + certification success stories, a recap of the PCB Carolina event from the Research Triangle Park (RTP) North Carolina chapter, and upcoming events, so mark your calendars for next year.

Chapter Spotlight

by Luke Hausherr
SAN DIEGO CHAPTER SECRETARY

This month's spotlight is the San Diego chapter. Chapter officers include Bob Griffith (chairman), Ben Savage (vice chairman), Bill Gebhardt (treasurer), me (Luke Hausherr, secretary), John Carney (education coordinator), and Crystal Hardy (member-at-large) (Figure 1).

Figure 1: Bill Gebhardt, Crystal Hardy, and Ben Savage at the IPC Designers Council booth.

A main goal of our chapter is to bring the PCB design and engineering community together by holding local chapter meetings where industry leaders speak about various topics—such as design and manufacturing trends, capabilities, new technologies, and improving data exchange and communication—in an effort to engage and educate the community as well as take part in networking. Our largest annual meeting with the highest attendance—75 + people—is held at the Del Mar Electronics and Manufacturing Show, which takes place in San Diego at the beginning of May. This show typically runs for two hours and has one or multiple speakers in addition to thousands of dollars in raffle prizes given away.

The four to five other meetings throughout the year are hosted at various venues, and typically have approximately 25-40 people in attendance. There is usually a $10 donation at the door to help contribute towards food, refreshments, and raffle prizes. The meetings are held as luncheons or can be later in the evening.

Some of the various topics that the chapter has covered include:

- Multi-board system-level design
- PCB design calculators
- Signal integrity effects on different PCB technology
- CAD libraries
- Flex and rigid-flex overview and design guidelines
- Power-supply layout

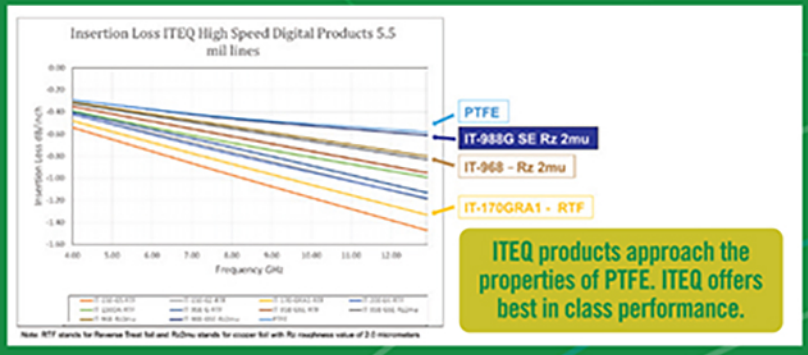
IT-968 / IT-968 SE

- High Thermal Reliability
- Designed for High Speed Digital Applications
  - Telecom, Backplane, Base Station, Server, Storage, Switch

IT-968
Dk @ 10 GHz - 3.85
Df @ 10 GHz - 0.0043

IT-968 SE
Dk @ 10 GHz - 3.30
Df @ 10 GHz - 0.0019

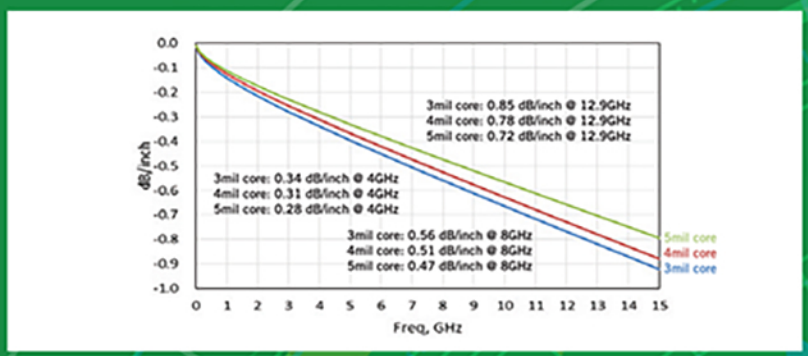
Insertion Loss - Measured



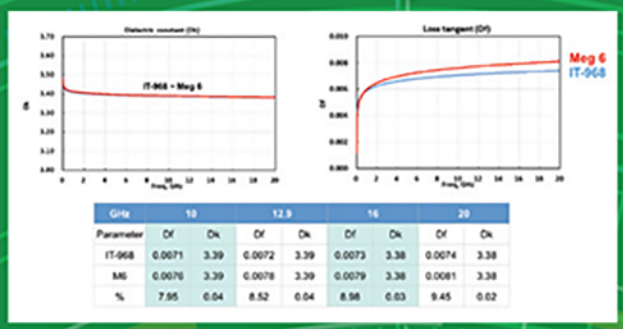
Industry Test Vehicle

32 layers, 0.140" thick
Four 2 oz copper internal layers
0.8 mm pitch, 9.8 mil drills
- Passed 1000 hours CAF, 10 V bias, 50 V
- IST - Passed 1000 cycles, 6x 260°C precondition
- No delamination after 8x 260°C reflow after 2 weeks at 35°C/ 85 % RH

IT-968 Loss Performance

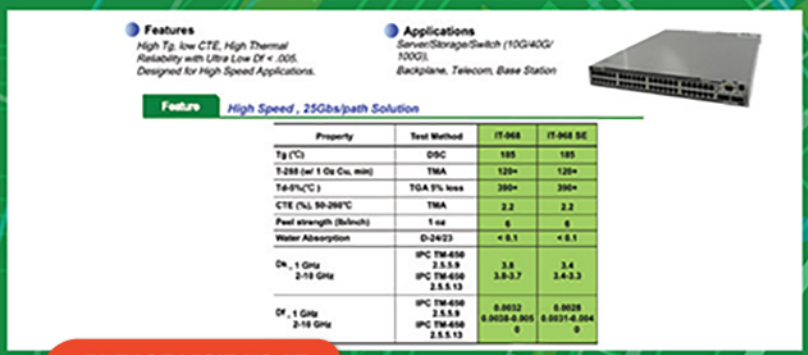


IT-968 SPP Comparison



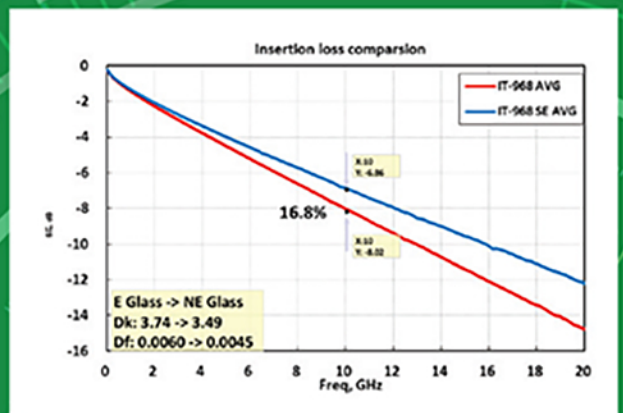
GHz	10		12.9		16		20	
Parameter	Df	Dk	Df	Dk	Df	Dk	Df	Dk
IT-968	0.0071	3.39	0.0072	3.39	0.0073	3.38	0.0074	3.38
M6	0.0070	3.39	0.0078	3.39	0.0079	3.38	0.0081	3.38
%	7.95	0.04	8.52	0.04	8.98	0.03	9.45	0.02

Ultra Low Loss IT-968 SE



Property	Test Method	IT-968	IT-968 SE
Tg (°C)	DSC	185	185
T-288 (w/ 1 Oz Cu, min)	TMA	120+	120+
Td-5%(°C)	TGA 5% loss	390+	390+
CTE (%), 50-260°C	TMA	2.2	2.2
Peel strength (lb/inch)	1 oz	6	6
Water Absorption	D-24/23	< 0.1	< 0.1
Dk , 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	3.8 3.8-3.7	3.4 3.4-3.3
Df , 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	0.0032 0.0038-0.005 0	0.0028 0.0031-0.004 0

E-Glass vs Low Dk Glass



PLEASE VISIT US AT
APEX EXPO IPC 2019
BOOTH #131

ITEQ

Our next meeting will be held in mid-January of 2019 where officer elections will occur. Our long-time running chapter chairman, Bob Griffith, is being promoted to the IPC Executive Board, so a replacement will be voted in.

Bob has rigorously contributed to the San Diego chapter for over a decade, providing not only his time and expertise but also his personal funds when needed for the chapter to continue to grow and thrive. His hard work and dedication have allowed the San Diego chapter to evolve and stay alive over the years, and he will be greatly missed from the group. It is with much gratitude that the IPC San Diego chapter bids Bob Griffith farewell and the best of luck with his new IPC ventures.

Bob has nominated the following people to take over these particular roles as he makes his transition, which will be voted on publicly with San Diego chapter members to make it official:

- Luke Hausherr: Chairman
- John Carney: Secretary
- Crystal Hardy: Education coordinator

The chapter would like to encourage any PCB professionals in the San Diego area to become a member of the group. Membership is easy, and best of all, it's free! Join us in January and take part in the officer election to help shape the outcome of the future for the chapter. If anyone else is interested in pursuing any of the above-mentioned roles, this is an open election, and all interested parties will be considered. If you're interested in learning more about the chapter, becoming a member, or pursuing a board position, please contact Luke Hausherr.

IPC CID and CID+ Success Stories
by Stephen Chavez

There were several recent IPC CID and CID+ classes throughout the months of October and November. Kelly Dack, Mike Creeden, and Gary Ferrari were the trainers for those classes and reported that their classes continued to have positive synergy with great results. Mike noted that his CID class held in Dallas had 18 students from Lockheed Martin. His other CID class held in Orlando had 23 students participate from Lockheed Martin as well.

Kudos to Lockheed Martin's upper management for recognizing the value of professional development of their staff. These two classes respectively involved more company focused and site-specific classes. Meanwhile, the fall IPC standards meeting and the PCB Carolina 2018 event, which always has IPC CID and CID+ certification classes, are more regional classes with students typically attending from all over the Midwest.

The two most recent regional IPC CID and CID+ classes were held in Anaheim, California, and the trainers were Paul Fleming and me. (Figures 2 and 3). Paul stated, "We had a great CID certification course in Anaheim earlier this month. It was a full class made up of students from all over the U.S. who brought with them a great variety of backgrounds and experience. Every class creates a unique experience for both the students and me. I'm always excited to see a new group of CIDs ready to step more fully into their role as a key part of the product design cycle. I'm proud to congratulate this new group of CIDs."

The CID+ class was another success with students attending from all over the Southwest

Figure 2: Paul Fleming teaching a recent CID certification class.

Figure 3: Stephen Chavez's CID+ class.

region of the U.S. and Mexico. It was one of the most collaborative advanced classes I have been involved in with lots of student engagement, which directly relates to the overall success of the class. What stood out the most to me in this particular class was the fact that all the students continued studying together after hours not only to achieve individual CID + certification success but for building relationships for future industry knowledge sharing and professional networking.

PCB Carolina Event Recap

by Tony Cosentino

RESEARCH TRIANGLE PARK (RTP) NORTH CAROLINA CHAPTER PRESIDENT, DIRECTOR OF TECHNOLOGY AND QUALITY AT BETTER BOARDS INC.

The RTP chapter's goals include bringing the PCB design community together by holding quarterly meetings, sharing ideas over meals, discussing chapter business, reviewing the current health of the job market, and fostering continuing education by holding technical presentations by guest speakers. We also created a local trade show, PCB Carolina, to offer our design community an opportunity to attend a trade show and technical presentations at no cost to them.

PCB Carolina 2018 was held on November 7 and was a huge success once again. This premier electronics trade show offers free access for all attendees to the showroom floor, keynote address, all technical sessions, food and drinks, and door prizes. Vendors pay for the privilege of a table and can purchase other promotional perks (ad hoc) or via a sponsorship. This event revenue is by far the major contributor to the chapter's annual budget.

This year's event was held at North Carolina State University's McKimmon Center for Continuing Education and offered the following:

- 75 exhibitors
- 987 attendees
- Two major, four gold, four silver, and 17 bronze sponsors
- One keynote speaker (Rick Hartley)
- 16 technical sessions
- Hands-on soldering workshops (Angel Deluna, Circuit Technology)
- IPC technical education (Susy Webb, IPC)
- CID/CID+ training (Dave Seymour and Gary Ferrari, EPTAC Corporation)
- Continental breakfast (sponsored by Better Boards)
- Buffet lunch
- Evening reception (sponsored by Mentor, a Siemens business)
- Door prizes including a grand prize 65" Samaung4K UHD TV (sponsored by PalPilot)

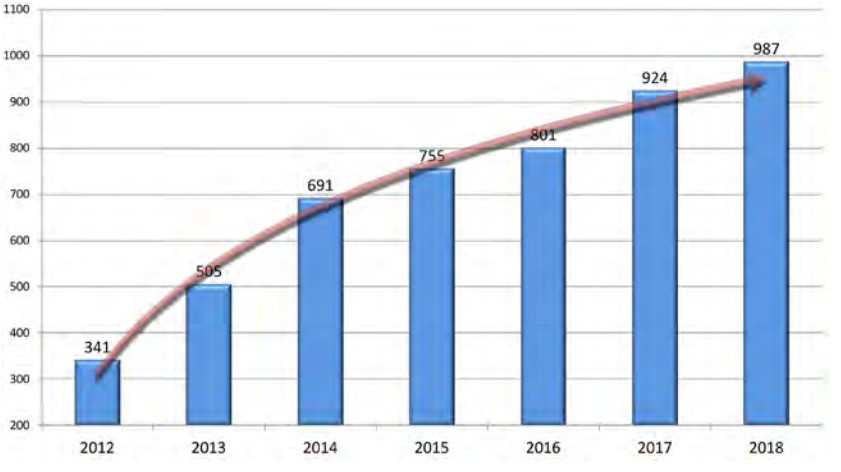


Figure 4: PCB Carolina yearly growth.

Figure 5: PCB Carolina 2018 show floor.

The event has grown over the years in both the exhibitors and the attendees. From 66 exhibitors and 800 registered attendees just two years ago, we reached 75 vendors with 987 registered attendees this year. Figure 4 demonstrates the event's yearly growth since 2012. Our goal for next year is to break 1,000 attendees (Figure 5).

Our chapter maintains open communications with area engineering and technical schools, including NC State University, Wake Technical Community College, and Coastal Carolina Community College. Since we hold PCB Carolina at NC State, we occasionally invite their professors to speak at the trade show. We often get volunteer support from local community colleges for the trade show to help set up, fulfill duties during the event, and some stay to help tear down. Getting the local community colleges involved gives them access to the experience of the show, industry vendors, job scouting, technical training, and the ability to network with people they might never have access to meet. They love the experience, and we keep getting volunteers year after year.

The local community is fortunate to have access to such a fulfilling trade show with a wide selection of training opportunities. The local trade show would not be possible without endless hours donated by the RTP chapter of the IPC Designers Council. Our chapter has 200+ members on email distribution, and meetings can have up to 40 attendees depending on the time of year and topic being discussed.

2019 Planning

In addition to our recent chapter activity with PCB Carolina 2018, we've started planning for 2019:

- **January 2019 chapter meeting**
  - Protolabs in Morrisville, North Carolina
  - Discuss rapid prototyping including injection molding, CNC machining, and 3D printing
  - The meeting will include a site tour of their facility
- **March 2019 chapter meeting**
  - CertifiGroup in Cary, North Carolina
  - Discuss environmental testing including HALT, HASS, product quality, and product durability
  - The meeting will include a site tour of their facility
- **June 2019 chapter meeting**
  - Not scheduled yet, but hope to cover the topic of antenna design
- **August 2019 chapter summer break**
- **September or October 2019 chapter meeting**
  - Not scheduled yet
- **PCB Carolina 2019**
  - November 13 at North Carolina State University's McKimmon Center for Continuing Education

Contact Tony Cosentino with any questions.

Recent Activities

- **Two-day hands-on power integrity workshop: Steve Sandler**
  - Occurred December 3–4 in Santa Clara, California

- **IPC CID+ certification class: Cherie Litson, MIT, CID+**
  - Occurred December 4–7 at Lake Washington Institute of Technologies

- **Cascade IPC chapter meeting**
  - Occurred after certification classes on December 7 at the same location
  - Microconnex (a local manufacturer) spoke on flex
  - The Altium user group met immediately after the meeting
  - cascade-ipcdc.org

2019 Training and Certification Schedule IPC Certified Interconnect Designer (CID)

- January 31–February 3: San Diego, CA
- February 26–March 1: Manchester, NH
- March 5–8: Santa Clara, CA
- March 19–22: Kirkland, WA
- April 30–May 3: Schaumburg, IL
- May 21–24: Pittsburgh, PA
- June 18–21: Kirkland, WA
- August 6–9: Baltimore, MD
- August 26–29: Markham, ON
- September 6–9: Santa Clara, CA
- September 19–22: Schaumburg, IL
- October 29–November 1: Anaheim, CA
- November 2–5: Raleigh, NC

IPC Advanced Certified Interconnect Designer CID+

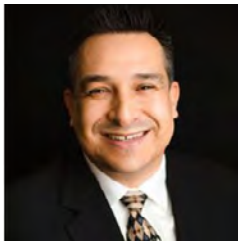
- January 31–February 3: San Diego, CA
- April 16–19: Markham, ON
- September 6–9: Santa Clara, CA
- September 10–13: Kirkland, WA
- September 19–22: Schaumburg, IL
- October 29–November 1: Anaheim, CA
- November 2–5: Raleigh, NC
- November 26–29: Manchester, NH

Note: Dates and locations are subject to change. Contact EPTAC Corporation to check current dates and availability. A minimum enrollment of seven students is required for a class to be held.

Upcoming Events

- **IPC APEX EXPO 2019**
  - January 26–31: Meetings and courses
  - January 29–31: Conference and exhibition
  - January 31–February 2: IPC Designer Certification CID/CID+
  - San Diego, California
  - ipcapexexpo.org

- **DesignCon 2019**
  - January 29–31
  - Santa Clara, California
  - designcon.com



Stephen Chavez is a member of the IPC Designers Council Executive Board and chairman of the communications subcommittee. To read past columns or contact Chavez, click here.

Supercomputers Without Wasted Heat

Researchers from the Department of Physics at the University of Konstanz have now demonstrated that the lossless electrical transfer of magnetically encoded information is possible. This finding enables enhanced storage density on integrated circuit chips, and, at the same time, significantly reduces the energy consumption of computing centres. The results of this study have been published in the current issue of the scientific journal *Nature Communications.*

"The combination of superconductivity, which operates without heat generation, with spintronics, transferring magnetic information, does not contradict any fundamental physical concepts, but just naïve assumptions about the nature of materials," Elke Scheer says. Recent findings suggest that by bringing superconductors into contact with special magnetic materials, electrons with parallel spins can be bound to pairs carrying the supercurrent over longer distances through magnets. This concept may enable novel electronic devices with revolutionary properties.

(Source: Universität Konstanz)

Why Should You Join the Designers Council?

by Judy Warner
ALTIUM

Let's get right down to business. Most of you know what IPC Designers Council is, and perhaps where your local chapter holds meetings, but have you ever been to one? If so, have you ever considered volunteering? No judgment here—I had the largest chapter in the country in my backyard for years and I never darkened their door until fairly recently.

Why? Well, although I've worked alongside PCB design professionals for the entirety of my professional life, I didn't think I belonged at a DC meeting. I'm not a board designer or an electrical engineer. After all, I was busy and wasn't confident I had anything to gain, much less contribute.

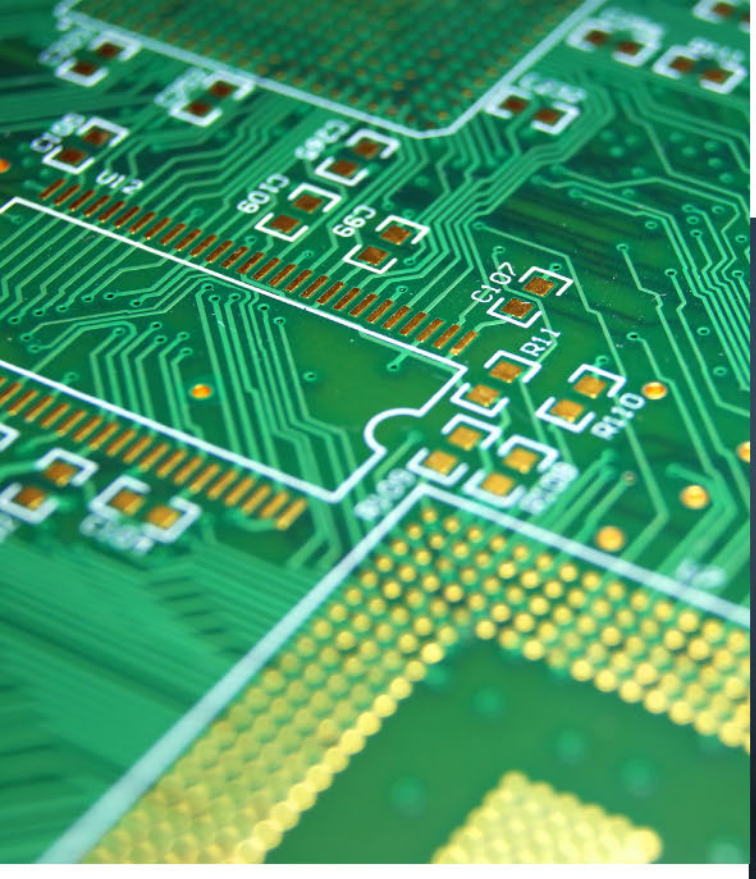
Thank goodness for the persistence of my friend Scott McCurdy of Freedom CAD who has been running the largest IPC Designers Council chapter in the country for over 15 years, Orange County. He consistently invited me, and I finally attended my first DC meeting a few years ago. At that first meeting, I realized how much I had been missing. For this reason, I'd like to do my best to dispel three myths about the Designers Council and help inspire you to get involved in your local DC chapter.

Myth #1: The IPC Designers Council is exclusively for PCB designers.

Well, yes and no. The chapters are primarily designed to provide learning and networking opportunities for PCB designers. However, if you are a material provider, fabricator, or EMS provider, you will learn so much about current challenges the design community faces and how you might better serve them. You can also provide a talk or speaker who can help inform on issues related to designing for performance, reliability, manufacturing, etc. We often complain about working in "silos" in this industry; this is one place where we can all step out of our silos, open up highly valuable conversations, and make connections.

Myth #2: DC meetings take too much time out of my busy schedule, and they won't benefit me anyway.

You have to eat lunch, right? So why not do it at a DC meeting where you can get some education and do some professional networking at the same time? Most meetings are in the lunch-and-learn format that provides lunch for attendees for a nominal cost. Do you have a design challenge that's hanging you up? Where better to get some advice from fellow designers



Offshore pricing. Exceptional quality. Local service.

We strive for excellence in everything we say and do.

Our finger is on the pulse of the latest technologies, systems and solutions in an effort to be a total PCB solution provider for our customers.

Why choose Eagle as your total PCB provider:

- Dedicated offshore team with local quality and engineering support
- Seamless transition from prototype to production
- The same exceptional service and support that you've come to expect from Eagle

Click here to learn more about our offshore option

EAGLE electronics Inc.

www.eagle-elec.com

or EDA providers who are often in attendance?

Do you have a DFM or DFA question? Did you lose a supplier recently? Bring these challenges to a meeting where manufacturers or assembly company representatives can help you. Are you between jobs? A DC meeting is a great place to find one! Think of the DC like a local neighborhood of electronics professionals. We all do better in a community then we do in those pesky silos, especially when it comes with lunch.

Myth #3: I don't have a chapter in my area, so there's no opportunity for me.

The opportunity may be greater for you than anyone else if you're willing to roll up your sleeves and find a few cohorts to help you start your own DC chapter. At this point, I don't want to sound like Pollyanna. It takes work to get anything off the ground, like pushing a rock uphill. Sharing the load with two or three others makes it far easier.

A couple of years ago, I interviewed Scott McCurdy, and he shared his "playbook" for running a successful DC chapter. I encourage you to read this article if you are thinking about starting a chapter as it will help shorten your path to success. Remember, when it comes to IPC, it is an "us" not a "them." So, don't be hesitant to jump in and take this on. The industry is only as good as the people who contribute to it.

One thing that I think has helped McCurdy be so successful is that he made phone calls and visits personally while he was going about his business. We are flooded with information these days, and one more email invitation isn't going to be effective. At this particular point in time, we are overconnected online and underconnected in person simultaneously. I am trying to reverse that equation in my personal and professional life. The IPC Designers Council offers designers a great way to create a platform for rich in-person relationships and networking in addition to the ongoing education that is the centerpiece of it all. For information about becoming a member for free or starting a chapter, visit the website.

The Designers Council: The Switzerland of EDA Company Engagement

One of the mandates I received when joining Altium almost two years ago was to support the DC chapters actively; this is one part of my job that I particularly enjoy. I help find speakers (from within Altium or not) and provide lunch when I'm able to for any active chapter. Currently, Altium has the privilege to support the following chapters: San Diego, Orange County, Silicon Valley, Salt Lake City, and the Cascade Chapter (Oregon). If you run a chapter and I don't know about you, please contact me by clicking here.

What I particularly appreciate about the DC is the fact that all major EDA companies can contribute, which they often do. In my humble opinion, it is our collective responsibility to serve these chapters that are made up of all of our customers. It is a pleasure to volunteer alongside wonderful people like Terri Kleekamp from Mentor, a Siemens Company, and John Carney from Cadence Design Services in service of regional DC groups. We may compete in the marketplace, but when it comes to the DC, we are friends, allies, and fellow contributors.

I hope the stories I've shared have provided some fresh insight and inspiration that will lead you to engage in the IPC Designers Council. Whether you are a PCB designer or supply-chain provider in the electronics industry, get involved with a local DC chapter. Your participation and presence bring and deliver much value. I invite you to join me in connecting with the DC. You'll be glad you did.

To learn about additional resources and initiatives from IPC, and to hear about their recent visit to the White House, listen to this recent podcast with John Mitchell, IPC president and CEO. **DESIGN007**

Judy Warner is director of community engagement for Altium. To download your copy of Altium's micro eBook, *The Printed Circuit Designer's Guide to… Design for Manufacturing (DFM),* click here.

GET YOUR 20% DISCOUNT!

WHEN YOU REGISTER BY
12.21.2018

APEX EXPO® IPC 2019
IPCAPEXEXPO.org
#IPCAPEXEXPO

SAN DIEGO CONVENTION CENTER, CA, USA

MEETINGS AND COURSES: *JANUARY 26–31, 2019*
CONFERENCE AND EXHIBITION: *JANUARY 29–31, 2019*

TECHNOLOGY'S FUTURE COMES TOGETHER

THROUGH NETWORKING

Technology's future will literally come together in San Diego at ***IPC APEX EXPO 2019*** on the exhibit show floor, in meetings, sessions and courses, and during networking events. The in-person exchange of ideas and know-how between you and your peers powers the industry's future.

Whether you are established in your career or just starting out, ***IPC APEX EXPO 2019*** will be the event to:

- Network with thousands of electronics industry peers
- Make new connections
- Strengthen existing relationships
- Become a mentor/meet a mentor

Strengthen the opportunities for yourself and the entire industry—register today for IPC APEX EXPO 2019 at ipcapexexpo.org.

IPC APEX EXPO 2019: Where Technology's Future Comes Together

One World, One Industry

by John Mitchell, IPC–ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

The electronics manufacturing industry continues to grow and change, and there's no better place to participate in the excitement than at IPC APEX EXPO 2019. We've built a strong community of enthusiastic individuals passionate about electronics manufacturing, and I'm eager to work with exhibitors, speakers, standards committee leaders, members, and attendees alike to ensure that "Technology's Future Comes Together" at the 2019 event.

While at the show, attendees will have the opportunity to experience and compare equipment from nearly 500 of the industry's top innovators and suppliers, discover new processes to gain greater efficiency, and find new suppliers while uncovering new solutions that will prepare them for tomorrow's opportunities.

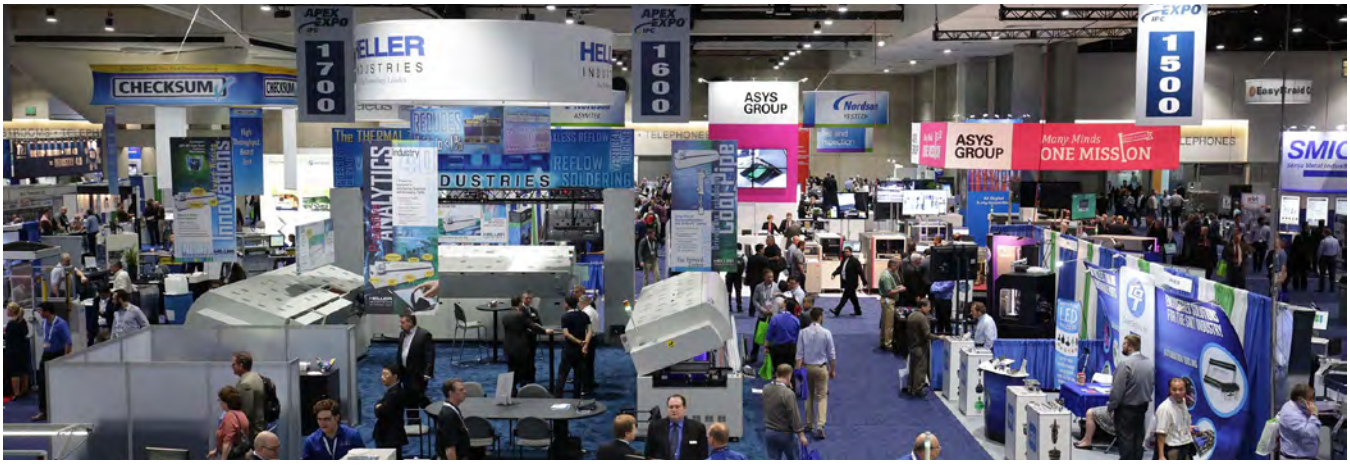
Educational program offerings will help keep attendees and their companies productive in 2019. Here are my top eight reasons to take part in IPC's technical conference sessions and professional development courses:

1 All IPC APEX EXPO 2019 programming is designed to help address current and future industry challenges such as the expanding importance of automation and data exchange, the increasing miniaturization of assemblies, the need for low-temperature soldering to lower energy consumption, and the growing complexity of design.

2 The education is not theoretical—it's grounded in and driven by real-world applications that are happening right now.

3 All submitted abstracts are a snapshot of what the industry is interested in, and the education program committee is comprised of people working in the industry. Selected submissions are always based on how they will truly offer value to the attendee.

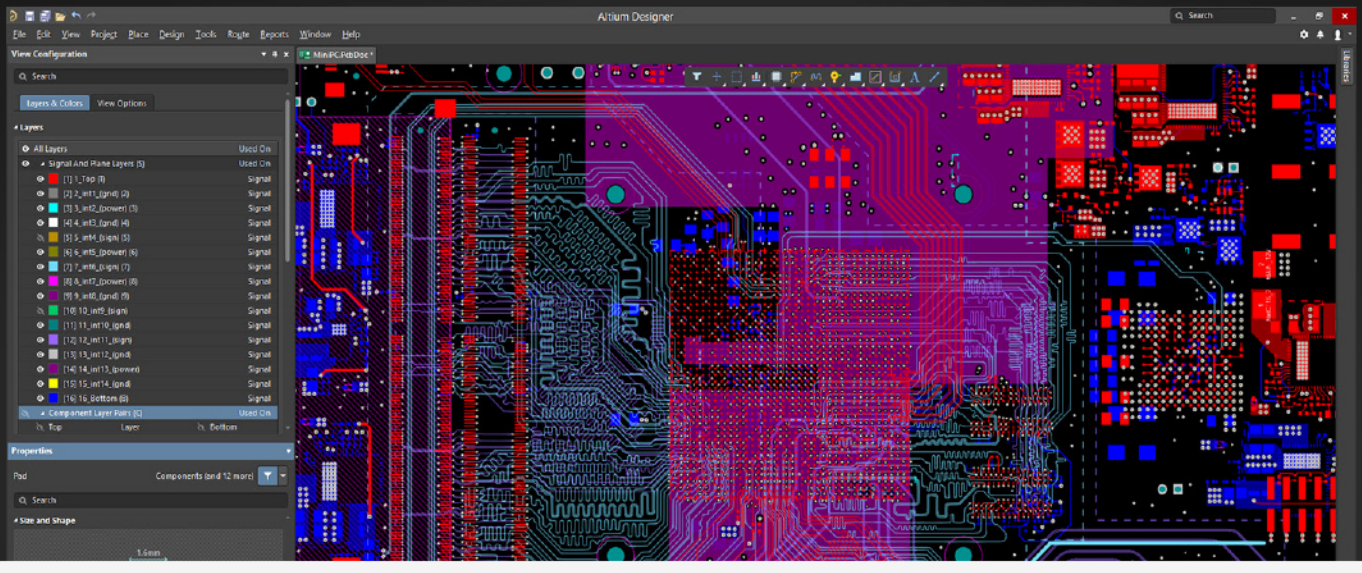
4 The content that participants experience during our technical conference is presented here first and is completely unique to IPC APEX EXPO.



ALTIUM DESIGNER19

The next generation of Easy, Powerful, Modern PCB Design

DECEMBER 17



Explore Altium Designer 19, available December 17th

https://www.altium.com/ad19

Altium

5. IPC APEX EXPO is a multidisciplinary event. This is an opportunity to learn while interacting with others who represent different functionalities within the supply chain. The conference also provides a greater, more well-rounded picture of the industry when compared to other events available.
6. Attendees will have the opportunity to connect with equipment and solution providers on the show floor to discuss how current or future technology may impact what they learned in the classroom.
7. Learn how updated IPC standards around reliability, assembly, and other concerns address changes in both traditional areas such as PCBs and emerging technologies like printed electronics.
8. 83% of IPC APEX EXPO attendees say the training and education delivered met or exceeded their learning objectives.

Further, part of the fun of attending IPC APEX EXPO is learning from individuals who are influential in the industry. IPC APEX EXPO 2019 will feature JB Straubel, chief technical officer (CTO) and co-founder of electric vehicle maker Tesla Inc. Straubel will present "Accelerating and Disrupting Innovation: The Tesla Story" on January 29. In his presentation, Straubel will draw on his extensive experience and share the history of Tesla, detailing that innovation was not an add-on to its operations but woven into every element of the business. He will also answer the perennial question, "How can my organization re-engineer itself to create products and ideas that answer 21st-century needs?"

An attendee participating in a buzz session.

Attendees connecting with exhibitors on the show floor.

From the show floor to the classroom and all points in between, IPC APEX EXPO 2019 is the place to be to learn, network, conduct business, and create new relationships while building on established ones.

Every year, IPC APEX EXPO provides me with unique ways of learning things I didn't realize that I needed to know. I look forward to seeing you and learning along with you in sunny San Diego in just a few short weeks!

A few noteworthy happenings:

• Show Floor Reception: January 29

You're invited to the industry's largest networking event—the IPC APEX EXPO show floor reception. This is your opportunity to network with industry colleagues, make new connections, and interact with nearly 500 exhibitors in a dynamic environment.

• New Products Corridor

View cutting-edge products and services in the New Products Corridor located in the Sails Pavilion (upper-level exhibit hall). Get a sneak preview of tomorrow's equipment, materials, and services that are breaking new ground in our industry.

• Connected Factory Exchange (CFX)

IPC-CFX is an electronics manufacturing industry developed standard forming the foundation and backbone of Industry 4.0 applications. Participate in a live demonstration of IPC CFX on the show floor with a larger presence of supporting exhibitors for 2019.

• IPC Hand Soldering World Championship and Rework Competition: January 29-30

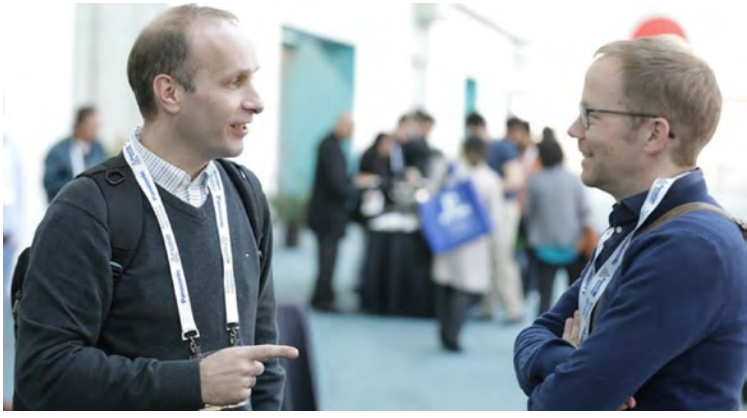
Competitors will be presented with a soldered assembly that will be partially populated with components. Contestants will be required to remove six specific components, remove the old solder, and clean the area of removed components. This will take place in the Sails Pavilion (upper-level exhibit hall).

• Ice Cream Social on the Show Floor: January 30

Satisfy your sweet tooth while networking with the innovators and suppliers of the electronics industry. Check the IPC APEX EXPO mobile app for the specific time, so you don't miss out on the fun!

• Passport to Prizes: January 31

Travel around the show floor collecting stickers from participating exhibitors and enter for a chance to win an exciting prize including an Apple Watch, GoPro, VR Headset, Ring Doorbell, and Star Wars Hero Droid BB-8. Submit your completed card to the IPC Bookstore on or before Thursday, and attend the drawing giveaway at 12:30 p.m. **DESIGN007**



Attendees networking with industry professionals.

For more information on IPC APEX EXPO 2019, visit ipcapexexpo.org.

John Mitchell is president and CEO of IPC—Association Connecting Electronics Industries. To read past columns or contact Mitchell, click here.

IPC APEX EXPO 2019 Schedules, Stats, and Programs

Feature by Nolan Johnson
I-CONNECT007

IPC APEX EXPO 2019 will open its doors at the San Diego Convention Center in late January. IPC APEX EXPO is regarded as the largest event for electronics manufacturing in North America, attracting more than 9,000 professionals from 45 countries.

Attendees at all career stages can expect to access new research and best practices; learn about trending materials, applications, and processes such as Industry 4.0 and wearables; address real-world problems to help with job success; experience the largest electronics industry collection of top suppliers, live demonstrations, and extreme innovations; and make connections in educational sessions on the show floor and during networking events.

Read on to find schedules, stats, and programs in preparation for IPC APEX EXPO 2019.

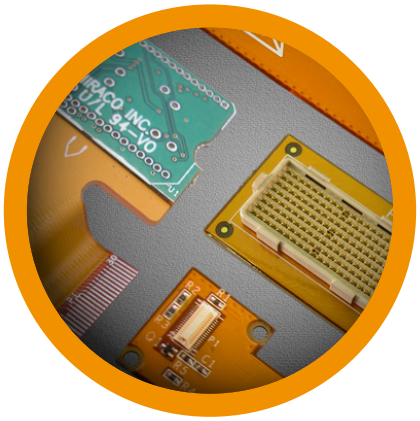
Event Information

- Meetings: January 26–31
- Courses: January 26–31
- Conference: January 29–31
- Exhibition: January 29–31
- IPC CID/CID+: January 31–February 3

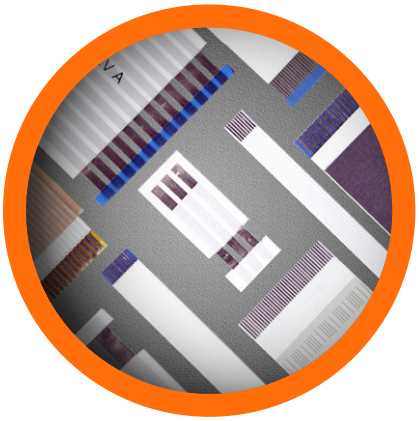
EXPO by the Numbers

- 8 Buzz sessions
- 16 Special events
  - Chairman's reception
  - Event awards luncheon
  - Hall-of-fame breakfast
  - IPC first-timers welcome
  - IPC annual meeting
  - Opening keynote
- 5 Management sessions
- 30 Professional development sessions
- 36 Technical conference sessions
- 105 New product announcements
- 117 Standards committee meetings
  - 23 Assembly and joining
  - 17 Base materials
  - 11 Cleaning and coating
  - 10 Electronic product data description
  - 4 Flexible and rigid-flex printed boards
  - 6 High-speed/high-frequency interconnections
  - 4 Management
  - 6 Packaged electronic components
  - 4 Printed board design technology
  - 13 Product assurance
  - 6 Rigid printed boards
  - 3 Testing
  - 10 Other topics

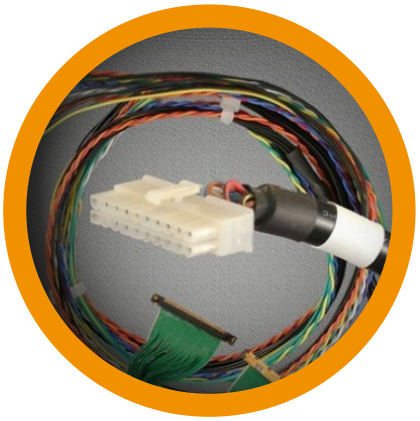
30+ YEARS EXPERIENCE. MORE THAN 30 PATENTS. JUST ONE GOAL.



Flexible Printed Circuits



Flat Flexible Cable



Wire Harnesses

Providing a quality, cost-effective solution on time has been our goal since day one. We achieve this through unparalleled customer service and by designing, sourcing and finishing interconnection systems for our customers worldwide.

PLEASE VISIT US AT
APEX EXPO IPC 2019
BOOTH #4027

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

MIRACO

Activities

Innovation Awards

The IPC APEX EXPO 2019 Innovation Awards are a celebration of the innovators and forward thinkers who are changing the technological landscape of the electronics industry.

Buzz Sessions

Buzz sessions are free, educational, and informational and are meant to bring participants up to speed on the topic. They are usually moderated and may include one presenter or multiple depending on subject matter. Questions from participants are encouraged; for some sessions, questions can even be submitted ahead of time. Here is a list of the current buzz sessions:

Export Control

- January 28, 10:30 a.m.–12:00 p.m.

E-textiles—The Fourth Industrial Revolution

- January 29, 1:30–3:30 p.m.
- Send your electronic textile integration questions in advance to etextiles@ipc.org

IPC-2581

- January 29, 3:30–5:00 p.m.

Perm Buzz Session

- January 30, 9:00–10:00 a.m.

Impacts of Warpage on SMT Processes

- January 30, 10:30 a.m.–12:00 p.m.

Shoring Up the U.S. Defense Electronics Industrial Base

- January 30, 1:30–3:30 p.m.

Findings from IPC's PCB Technology Trends

- January 30, 3:30–4:30 p.m.

iNemi Next-Generation Solder Materials

- January 31, 8:00 a.m.–2:00 p.m.

Management Programs

EMS Executive Management Meeting

- January 28, 7:30 a.m.–5:00 p.m.

Executive Forum on Advancing Automotive Electronics

- January 28, 7:30 a.m.–5:00 p.m.
- A must-attend event brought to you by the IPC Hall of Fame Council

Technical Information for Designers

Ray Prasad, Ray Prasad Consultancy Group

- Design and assembly process challenges for bottom-termination components such as QFN, DFN, and MLF in tin-lead and the lead-free world

Dale Lee, Plexus

- Design for excellence: DFM, DFR, DFA, and more—Parts 1 and 2

Martin Goetz, Northrup Grumman Corporation

- Designing additive manufacturing/3D printing to PCB fabrication and assembly
- High-speed/high-frequency laminate materials used in design
- Electrical and thermomechanical design constraints affecting system and component performance

IPC Designer Certification (CID and CID+)

- Tutorials January 31–February 2
- Exams February 3

Professional Development Program Highlights

Topics include reliability, reflow soldering, troubleshooting PCB related defects, Lean practices, design for testing (DFT), thermal management, high reliability for lead-free solder joints, electrostatic discharge, design for manufacturing (DFM), contracting with the customer, PCB fabrication basics, and process and specification.

Show Floor Highlights

January 29

You're invited to the industry's largest networking event—the IPC APEX EXPO show floor reception. This is your opportunity to network with industry colleagues, make new connections, and interact with nearly 500 exhibitors in a dynamic environment.

January 29-30

The IPC World Championship Hand Soldering and Rework Competition will require steady hands, nerves of steel, and a will of iron—only at IPC APEX EXPO 2019.

January 29-31

Participate in the live show floor demonstration of the IPC CFX Industry 4.0 protocol. Look for CFX supporting booths to answer all your questions.

January 29-31

The New Products Corridor will be the place to find cutting-edge products and services. This showcase of innovation will be located in the Sails Pavilion (upper-level exhibit hall).

January 30

Satisfy your sweet tooth at the ice cream social while networking with the innovators and suppliers of the electronics industry. Check the IPC APEX EXPO app for the specific time. You don't want to miss out on the fun!

How to Connect

Available mid-December on Google Play and the Apple Store, the IPC APEX EXPO smartphone app allows you to plan for sessions and events at the expo, review the show floor exhibitors, stay informed on schedule updates, and receive event alerts as they happen. You can also use the schedule at a glance from the IPC APEX EXPO conference guide. **PCB007**

For more information, visit the
IPC APEX EXPO 2019 website: ipcapexexpo.org
email: registration@ipc.org

or call the attendee hotline:
Toll-free: 877-472-4724
Outside of the U.S. and Canada: +1 847-597-2861

Shopping at IPC APEX EXPO: Evolutionary or Revolutionary Products?

Feature by Andy Shaughnessy
I-CONNECT007

Every year, managers and technologists descend upon IPC APEX EXPO, shopping list in hand, scouring the aisles in search of deals on capital equipment, software, alloys, chemistry, and whatever else they can find. Sure, many of the big pieces of equipment on display are already sold. On the flip side, there are plenty of deals to be made at IPC APEX EXPO because no company wants to pay to ship a machine twice.

As you're finalizing your shopping list, take a second to consider this: Are you in the market for products that are evolutionary or revolutionary? And what do those terms even mean for someone looking for new DFM software or a new pick-and-place machine?

Evolutionary vs. Revolutionary

The term "revolutionary" gets thrown around pretty often in the electronics world. But most new products—even the most expensive—fall into the evolutionary category; they feature logical updates and improvements over the last revision of that particular product. You could see these updates coming, and they make a good product a better product.

If you look back on a product's life cycle, you can track the new functionalities as they were incorporated each year or so, much like parents chart their children's growth by marking the kids' height on the wall on their birthday. Evolutionary change is slow and gradual, and fairly predictable. I imagine that most of the equipment and software in your facility contain evolutionary improvement and upgrades over the previous models.

But revolutionary change is disruptive to the status quo. Revolutionary change is unpredictable, an upset applecart that can't be set right until the full effects of the shift are quantified and understood. Sometimes the reverberations from revolutionary change make it almost impossible to measure its long-term effects for years.

You know your product is revolutionary if it forces other companies to change the way they operate. I liken revolutionary change to a quantum leap; it's not exactly an apples-to-apples comparison, but it's close.

The Model T was clearly revolutionary, putting the average Joe into an automobile and blacksmiths and wagon builders out of work almost overnight. EDA software and the SMT process were likewise revolutionary, changing the way PCBs were designed and assembled, and

allowing employees to be more productive.

Then again, some products are "leaners" that are tough to categorize.

The Wright brothers' first airplane was revolutionary, but there is some debate about whether the first jet plane was too since it's really just an airplane with a more powerful engine. Similarly, the Tesla seems revolutionary to most of us, but electric vehicles (EVs) have been around since the 1800s, and one EV held the world land speed record until 1900. Is a new type of fastener or squeegee capable of being revolutionary? Undoubtedly, some companies will say so, especially if they increase their profit margin.

Products at IPC APEX EXPO

Keep all of this in mind as you roam the aisles at IPC APEX EXPO. You'll probably see some "leaners," somewhere between evolutionary and revolutionary. IPC's Connected Factory Initiative (CFX) comes to mind. The open-source CFX standardizes machine-to-machine communications, allowing one person to monitor all of the machines on a line through a smartphone. But is CFX a truly revolutionary? You be the judge. Try it out in San Diego next month and see.

To help you craft the ultimate trade show shopping list, IPC has created this handy listing of all 311 new products on display at IPC APEX EXPO. It's not too late to thumb through these pages and start making a list of new products that could put your company light years ahead. Whether your company needs the latest cutting-edge tools or not, you can find what you need at IPC APEX EXPO 2019. **DESIGN007**

Sensitive Robots Feel the Strain

An artificial soft skin imbued with flexible electronics could enhance the way robots sense and interact with their surroundings, KAUST researchers have shown. The team has discovered how to program electrical conductivity and strain sensing into a single material embedded in a stretchy polymer skin. The discovery could also have applications in wearable electronic devices.

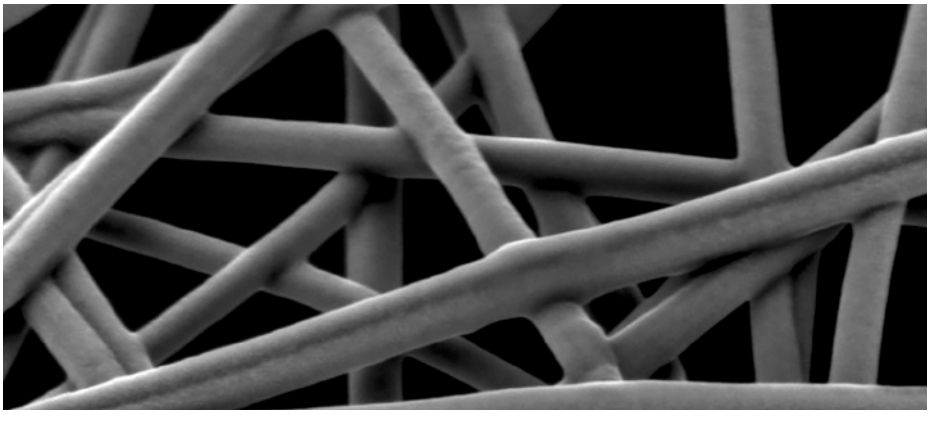
When an animal stretches a limb, a network of nerves and sensors within the skin provides feedback that help it orient the limb in space and interact with its surroundings. Embedding a network of strain sensors and connective wiring into a flexible artificial skin would give soft robots similar sensory feedback, helping them autonomously navigate their environment, says Gilles Lubineau, who led the research.

Until now, researchers have used different materials for the sensing and conductive wiring components, adding cost and complexity to the fabrication process, explains Ragesh Chellattoan, a Ph.D. student in Lubineau's team. "Our objective is to get both sensing and wiring connectivity in the one material," he says.

The researchers created a stretchy skin for a toy action figure to demonstrate their material. They coated one of the figure's legs with the artificial skin and then applied DC voltage only to the leg's left side before flexing the leg at the knee and observing what happened.

The next step, Chellattoan says, is to gain greater control over where nanowire welds form. This would give researchers the ability to draw precise conductive patterns into the artificial skin.

(Source: KAUST)



Dan Beaulieu on Making the Most of a Trade Show

Feature Interview by Barry Matties
I-CONNECT007

Dan Beaulieu, president of D.B. Management Group, has over 30 years of experience in the PCB industry. In this interview, Beaulieu provides his expert knowledge on selling strategies for companies as well as getting the most from your trade show experience as an exhibitor.

Barry Matties: Dan, IPC APEX EXPO 2019 is coming up fast. This industry has been going to trade shows forever, and we've seen the exhibitors grow and contract through good and bad times. Right now, we have a good extended period in our economy, which means a lot of people are going to be at the trade show. What advice would you give your customers in terms of preparing for a trade show and making it as successful as possible?

Beaulieu: The key is preparation. Barry, so often we hear, "I went to that trade show for years and got nothing out of it." Like everything else, it's what you put into it. I like to set clients up with a schedule. The first question to ask is, "Why are you going to this trade show? Why did you pick this event? What are you hoping to get out of it?"

Set your goals, whether that's introducing a new product or going to meet potential customers. You just want to know what's going on in the industry. Some people say a trade show is basically a branding tool. I think it is, but I like to be much more effectual. Start with establishing why you're going to this show. It's just like marketing—it's getting the story for the show, and showing off your marketing and sales abilities.

I like to work backward as well. At the end of the show, do a debrief. What happened? What worked well? What didn't work well? Even look at what you brought. Did you forget anything? Do you want to attend that show again?

The other thing is, as you start preparing for the show, put together a schedule 14 weeks out. At week 14, you should start talking about what your graphics will be and what you want to share at the show. At week 13, you're down to basic stuff like solidifying hotel and travel details, who's coming to the show, and what you want to talk about? Many times, it drives me crazy when people spend $8,000–9,000 on a show, and you walk by, they're sitting in the back of the booth reading a magazine or something. That's not what you should be doing. There are so many other valuable things you could do.

It's all in the preparation and deciding what you're going to get at the show, right down to the core of who are you going to meet. Set up those appointments. Send invitations and email blasts announcing what you're going to do. It's always great if you have a white paper to work around. Do things with intention. Make sure that every minute of that expensive show is worthwhile for you and your company's resources even down to who you invite to breakfast, lunch, or dinner.

Another important thing is don't load the booth with all of your employees so that there are 20 people wearing the same outfit and you feel like you're crashing a party if you try to go in and talk to somebody. If you have a big company, work in shifts to ensure there are two or three individuals in the booth at once. Always be thinking of that person who's coming to see your product. Try to load the deck. You don't just say, "Well, they'll walk by, and we'll bump into them." No, you should reach out to the kind of person or company who wants your service, and you make sure that they're going to come to the booth.

Also, if you're giving something away, give away something you do. Don't give away golf balls—you're not selling golf balls.

In summary, plan, prepare, organize, have a system, and be intentional.

Matties: That's really good advice. Further, one thing we would suggest people look at is the flow of your booth. Some companies have a booth that's 10 x 10 feet. Do you want them to step into that booth, or do you want them in the hallway behind a counter?

Beaulieu: That's right. Don't block the way in; instead, let people come in. Try to have a table and a couple of chairs in the corner of it to have a place where you can have a conversation. If that's not possible in the booth, then stake out a place where you can have individual conversations.

Matties: Great advice, Dan. Thank you very much for your time.

Beaulieu: Thank you. It's my pleasure. DESIGN007

Dan Beaulieu is president of D.B. Management Group and an I-Connect007 columnist. To read past columns or contact Beaulieu, click here.

High-temperature Electronics? That's Hot

From iPhones on Earth to rovers on Mars, most electronics only function within a certain temperature range. By blending two organic materials together, researchers at Purdue University could create electronics that withstand extreme heat.

This new plastic material could reliably conduct electricity in up to 220°C (428°F), according to a paper published Thursday in the journal *Science*.

"Commercial electronics operate between –40°C and 85°C. Beyond this range, they're going to malfunction," said Jianguo Mei, an assistant professor of organic chemistry at Purdue University. "We created a material that can operate at high temperatures by blending two polymers together."

The researchers discovered a few properties that are essential to make this work. The two materials need to be compatible to mixing and should each be present in roughly the same ratio. This results in an organized, interpenetrating network that allows the electrical charge to flow evenly throughout while holding its shape in extreme temperatures.

Most impressive about this new material isn't its ability to conduct electricity in extreme temperatures, but that its performance doesn't seem to change. Usually, the performance of electronics depends on temperature. The performance of this new polymer blend remains stable across a wide temperature range.

(Source: Perdue University)

In the Studio: *Real Time with...IPC*

Article by Andy Shaughnessy
I-CONNECT007

It's almost time for IPC APEX EXPO 2019 at the San Diego Convention Center, and that means another *Real Time with…IPC* video program bringing you interviews with the electronics industry's top movers and shakers, engineers, and managers. It's hard to believe that *Real Time with…IPC* has been a staple of IPC APEX EXPO since 2007.

I still remember my first *Real Time with…IPC* at IPC APEX EXPO 2008 in Las Vegas quite well. I had just begun doing video interviews, and to say I was a little rough is an understatement. Publisher Barry Matties had given me some training beforehand, but I was still trying to find my groove.

My first interview went along pretty well; so far, so good. Then, the engineer I was interviewing stopped talking and I couldn't think of a good follow-up question to ask. I looked around, trying to find inspiration for another question, and saw Barry raise one eyebrow at me and start smiling. We all started laughing. Finally, he said, "Don't look at me, focus on who you're interviewing."

That turned out to be good advice because an interviewer has to tune out everything except the person being interviewed, no matter where you are, and I'm usually surrounded by distractions at IPC APEX EXPO. Pick-and-place machines and vacuums always seem to be running. If we're doing a show wrap-up interview as the event is breaking down, a forklift driver will usually pull up next to us and drop a half dozen pallets. Once I tuned out the distractions, I could focus on the interview like a hunter's gun-proofed retriever.

Then and Now

Things have certainly changed since 2008. For one thing, I've gotten much better at doing

on-camera interviews; of course, after those first attempts, there was really nowhere to go but up.

We've also added a whole range of guest editors—many of them are people just like you—who conduct fantastic interviews with individuals in their segment of the industry. No matter what topic the interviewee wants to discuss, we probably have a guest editor who is familiar with that topic. Some of our guest editors have been doing *Real Time with…IPC* interviews for a decade, and we couldn't do it without them.

The technology we use today is also light years ahead. In 2008, we used our big old-school camcorders that still ran on tape, and our video editing operations sometimes took well into the night. Now, we shoot interviews with digital recorders; editing is finished before the show closes each day, and most of that day's interviews are posted on the *Real Time with…* site on the same day.

Another thing that I've noticed is that people in this industry are much more open to doing on-camera interviews than they were 10 years ago. It used to be tough finding someone to sit in front of the camera. Back then, a company might have had one person—perhaps in marketing communications—who was accustomed to doing video interviews. Now, forward-thinking companies consider video interviews to be another part of the marketing process, and they come to IPC APEX EXPO with a handful of camera-ready technologists.

Each year, we set up a studio at IPC APEX EXPO that allows us to conduct two video interviews at the same time. Jo Ann Sotelo is the conductor in the middle of this synchronized madness, rapping her baton on the dais, making sure that interviewers and interviewees are ready to go when the time comes. I still don't understand exactly how she makes all these interviews work out; I just do what she tells me to do!

So, stop by our *Real Time with…IPC* studio at IPC APEX EXPO 2019 and say hi. You just might meet some of the biggest names in the industry. And if you'd like to be a guest editor, all you have to do is ask. **DESIGN007**

PCB007 Highlights

Substrates for Advanced PCB Technologies: What Will the Future Hold? ▸

The UK chapter of the global IMAPS community of electronics and microelectronic packaging engineers shared a wealth of knowledge and wisdom about PCB substrate technology trends, developments, and future requirements in a webinar on the first of November.

Increasing Productivity for Flex Fabricators ▸

Barry Matties and Nolan Johnson of I-Connect007 met with Shane Noel and industry veteran Mike Jennings of ESI to discuss the introduction of their CapStone laser tool, a product aimed at doubling their flex circuit fabricators' throughput. Mike also shares advice for fabricators who are looking to move into the ever-growing flex market.

It's Only Common Sense: What the Customer Really Means ▸

As salespeople, one of the things that we spend a great deal of time doing is trying to interpret what the customer is telling us. We all can hear the customer and what they are saying, but so often, we don't know what they mean.

Jeff Waters: Isola Updates ▸

During PCB West 2018, Nolan Johnson and Barry Matties sat down with Jeff Waters, Isola CEO, to catch up on company activities, including the recent sale of the factory in Chandler, Arizona, the plan to build a new facility, product developments, current market dynamics, a new CFO, and much more.

Powering the Flexible World ▸

From 2019 we will come to a world with flexible electronics and batteries to power these devices have attracted tremendous attention. These devices may require batteries with special mechanical properties or form factors.

IDTechEx Highlights World Firsts in Printed Electronics in 2018 ▸

With the end of the year in sight, it's interesting to look back and review what has been new in the world of printed electronics in the last 12 months. This analysis is taken from the new IDTechEx Research report, "Flexible, Printed and Organic Electronics 2019–2029" covering the entire sector in great detail based on analyzing the industry for over 15 years.

IPC Signs White House Pledge to the American Worker ▸

IPC has signed the President's Pledge to the American Worker and committed to creating at least one million new training and workforce development opportunities in the electronics industry over the next five years.

150+ Years of Experience: Reflections with Three Industry Icons ▸

You would be hard-pressed to find a more knowledgeable and experienced group than that of Gary Ferrari, Gene Weiner, and Happy Holden. In a brief interview with Barry Matties, these three industry icons consider the past, present, and future state of electronics manufacturing while also offering advice to the newest generation of manufacturers.

Trouble in Your Tank: The Art and Science of Photoresist Stripping, Part 1 ▸

Photoresist stripping has become a complicated process due to many unique resist formulations on the market. The first part of this column series looks at some of the most common problems in photoresist stripping and offers strategies on how to address them.

U.S. CIRCUIT
33rd
ANNIVERSARY
1985 ★ 2018

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

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

ISO 9001:2008 REGISTERED

DEPARTMENT OF DEFENSE · UNITED STATES OF AMERICA

ITAR International Traffic in Arms Regulations Registered

10 Fundamental Rules of High-speed PCB Design, Part 4

Beyond Design

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

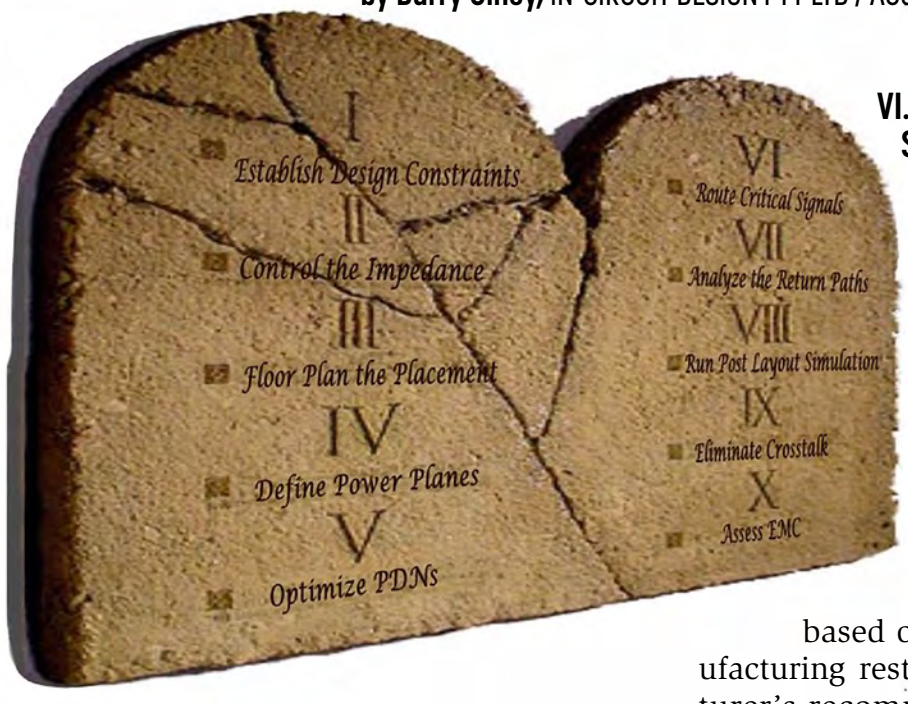


Figure 1: The 10 fundamental rules of high-speed PCB design.

Part 4 of the 10 fundamental rules of high-speed PCB design (Figure 1) deals with the routing of critical signals and return path discontinuities. Needless to say, matched delay and length, differential pairs, and other critical signals should be routed first with the precision they require before less important low-speed and static signals are completed. Maintaining this priority is imperative.

Note that the logic schematic diagram masks details crucial to the operation of unintentional signal pathways vital to the understanding of signal performance, crosstalk, and electromagnetic radiation emanating from the board. Therefore, it is just as important to understand the flow of the return current path of critical signals because these can influence the signal integrity and electromagnetic compliancy (EMC).

VI. Route the Board Based on Critical Signals: Adhere to the defined routing strategy. Clock signals should always have the longest delay of the group. Differential pairs should maintain constant impedance along the entire length.

As mentioned in Part 1 of this series, before starting placement and routing, detailed interconnect routing constraints should be established. These constraints—based on pre-layout simulation, manufacturing restrictions, and the IC manufacturer's recommendations and guidelines—will control the placement and routing processes. Online design rule checks (DRCs) will warn the designer when a constraint is violated.

When we draw a schematic by functionality, I think that we should also place and route by functionality. By doing this, you can add your own creativity and make decisions on the fly while still taking advantage of the automation. The most efficient approach is to cross-probe between the schematic and the PCB/routing editor (Figure 2). Also, having two extended monitors or a new 34-inch curved screen monitor—as I just purchased—makes this very easy to implement. I read that curved monitors make a straight line look bent and are not suitable for CAD applications. However, that is definitely not the case as the screen curvature follows the natural profile of the eye and the graphics appear crystal clear and undistorted over the entire screen.

RISE ABOVE THE AVERAGE

100+ PAGES of valuable PCB knowledge they didn't teach in school, straight from industry veterans.

Ready to Learn? Download the Guide ›

EMA Design Automation®

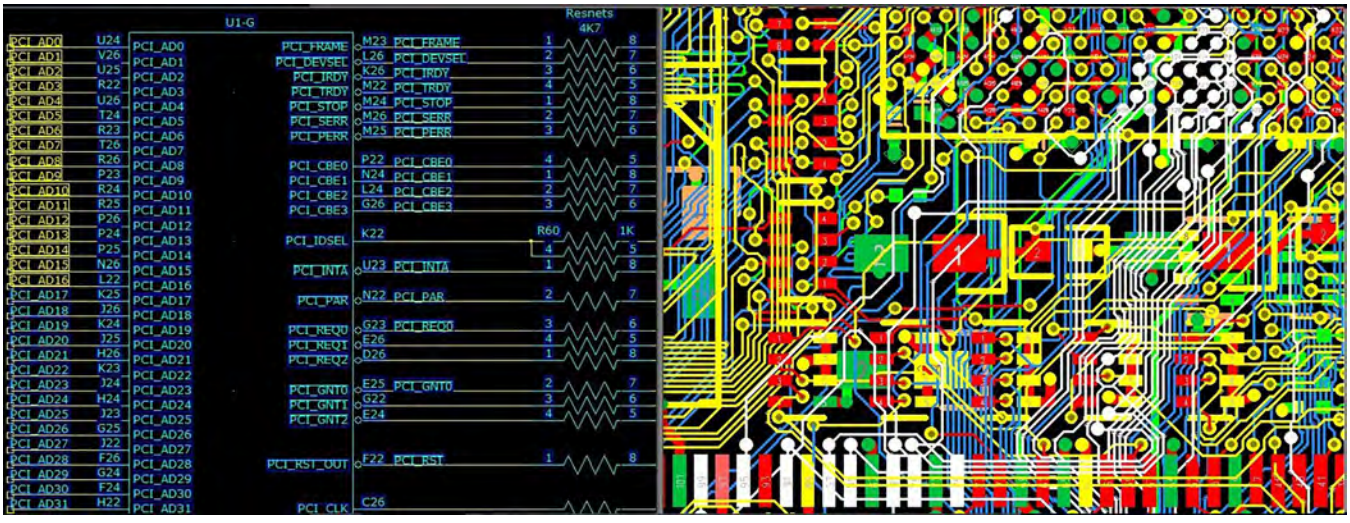


Figure 2: Cross-probing part of an address bus from schematic to router. (Source: Mentor Graphics, PADS)

On a multilayer PCB, critical signals should be routed on a stripline (inner layer) adjacent to a solid reference plane to reduce radiation. The spacing between the signal trace and return plane should be as small as possible to increase coupling and reduce loop area.

The three constraints to keep in mind include:

1. Making the mark to space ration of the waveform equal because this eliminates all the even harmonics
2. Routing high-speed signals between the planes and fanout out close to the driver (200 mils), dropping to an inner layer and route back up to the load again with a short fanout
3. Using the same reference plane (GND if possible) for the return signal because it reduces the loop area and radiation

As you can see from Figure 3, the trace routed on inner layer three exhibits between four to 10 dB less noise than the trace routed on the top layer. Note that there are radiating harmonics above 40 dB on the top layer routing. Also, high-frequency components radiate more readily because their shorter wavelengths are comparable to trace lengths, which act as antennas.

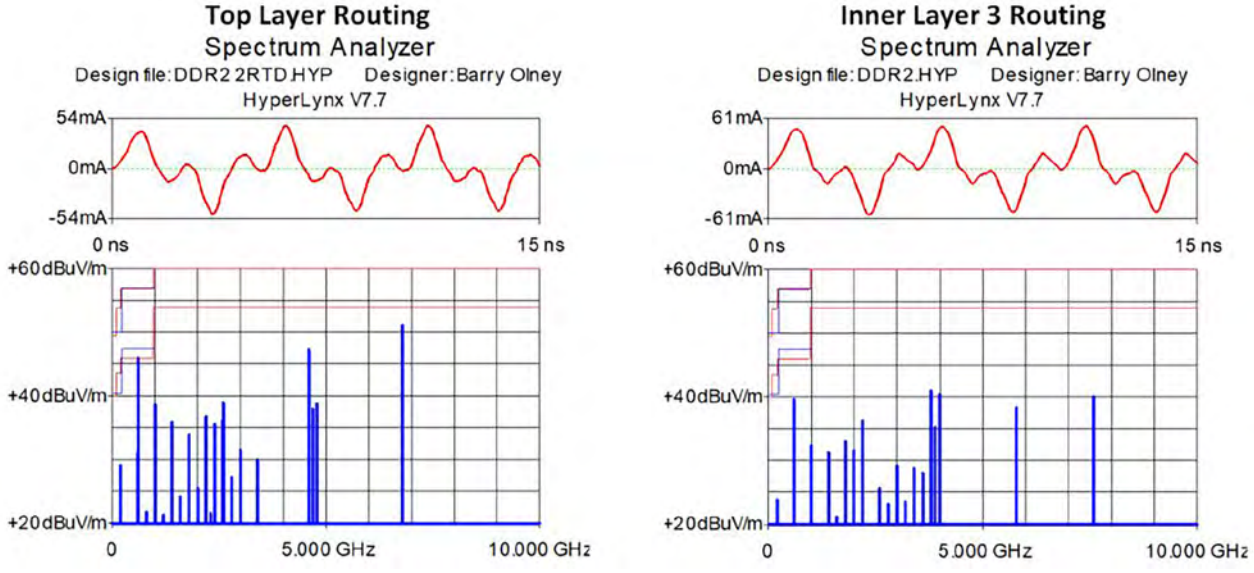


Figure 3: Comparison of signals routed on the top versus inner layer three. (Source: HyperLynx)

Consequently, although the amplitude of the harmonic frequency components decreases, as the frequency increases, the radiated frequency varies depending on the traces characteristics.

For a synchronous bus—providing the receiver waits sufficiently long enough for the crosstalk to settle—before sampling the bus, the crosstalk on data and address signals within each group has little impact on the signal quality at the receiver. This can be ensured by always making the clock or strobe the longest signal of a matched length group.

For a perfectly balanced differential signal, the radiation from one trace exactly cancels the radiation from the other as they are equal and opposite. However, a common-mode signal represents an average of the two signals in a pair. The radiation is identical on both traces, and therefore, does not cancel but instead reinforces. The transformation from differential to common-mode takes place on bends and nonsymmetrical routing near via and pin obstructions. This also impacts the impedance of the pair. To minimize radiation and crosstalk, one must think explicitly about the common-mode component of the differential signal; skew creates this common-mode signal.

Arguably, the principal source of imbalance is time-delay skew between the two traces. The easiest way to minimize this skew is to match the electrical lengths and to correct any shift immediately after it arises by adding length (hence the delay) to the shorter trace.

VII. Analyze the Return Current Paths: All signal traces should be tightly coupled to a contiguous reference plane and have a clearly defined minimum loop inductance return current path.

High-speed design is not as simple as sending a signal from the driver to the receiver over an interconnect. Rather, one should also consider the presence and interaction of the power distribution network (PDN) and how and where the return current flows.

Generally, PCB designers take great care to ensure that critical signals are routed exactly to length from the driver to the receiving device pins, but they take little care of the return current path of the signal. Current flow is a round trip, and the important issue is delay—not length. If it takes one signal longer for the return current to get back to the driver—around a gap in the plane for instance—then there will be skew between the critical timing signals. Return path discontinuities (RPDs) can create large loop areas that increase series inductance, degrading signal integrity and increasing crosstalk and electromagnetic radiation.

Small discontinuities, such as vias and nonuniform return paths on a bus, are becoming an important factor for the signal integrity and timing of high-speed systems. RPDs produce impedance discontinuities due to the local return inductance and capacitive changes. Impedance discontinuities create reflected noise, contribute to differential channel-to-channel noise, and may promote mode conversion. In the case of differential pairs, the transformation from differential-mode to common-mode typically takes place on bends, and asymmetrical routing near via and pin obstructions, but can also be caused by small changes in impedance due to RPDs.

Each signal layer should be adjacent to—and closely coupled to—a contiguous reference plane that creates a clear, uninterrupted return path and eliminates broadside crosstalk. As the layer count increases, this concept becomes easier to implement, but decisions regarding return current paths become more challenging.

Although power planes can be used as reference planes, ground is more effective as local stitching vias can be used for the return current transitions rather than stitching decoupling capacitors, which add inductance. This keeps the loop area small and reduces radiation. As the stackup layer count increases, so does the number of possible combinations of the structure. But, if one sticks to the basic rules, then the best-performing configurations are obvious.

Figure 4 shows the electric and magnetic fields emanating from a signal trace in both a microstrip and stripline configuration. Electric fields (blue) terminate when they come into contact with a solid plane while magnetic fields (red) are shielded by the planes, but the fringing fields still tend to radiate from the board edges.

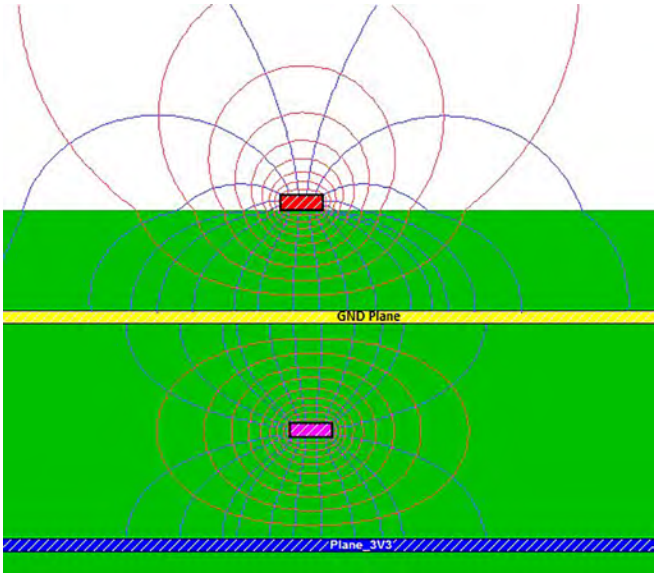


Figure 4: Microstrip and stripline electromagnetic fields. (Source: HyperLynx)

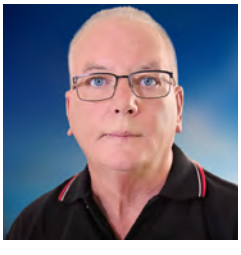
Unfortunately, return path discontinuities can never be totally eliminated, but we can take steps to minimize their impact significantly. As with PDN planning, it is all about inductance. If the return path loop area is increased in any way, then the inductance will also increase.

Key Points:

- Critical signals should be routed first with the precision they require
- It is important to understand the flow of the return current path of critical signals because these can influence the signal integrity and EMC
- Before starting placement and routing, detailed interconnect routing constraints should be established
- The most efficient approach to placement is to cross-probe between the schematic and the PCB/routing editor
- Critical signals should be routed on a stripline (inner layer) adjacent to a solid reference plane to reduce radiation
- The spacing between the signal trace and the return plane should be as small as possible to increase coupling and reduce loop area
- Timing can be assured and crosstalk ignored by always making the clock or strobe the longest signal of a matched length group of a synchronous bus
- One should consider the presence and interaction of the PDN and how and where the return current flows
- Current flow is a round trip, and the important issue is delay—not length
- RPDs produce impedance discontinuities due to the local return inductance and capacitive changes
- The transformation from differential-mode to common-mode typically takes place on bends and asymmetrical routing
- Each signal layer should be adjacent to—and closely coupled to—a contiguous reference plane, which creates a clear, uninterrupted return path and eliminates broadside crosstalk
- Although power planes can be used as reference planes, ground is more effective as local stitching vias can be used

Further Reading: "Beyond Design" Columns by Barry Olney

- Common Symptoms of Common-Mode Radiation, *Design007 Magazine,* May 2018.
- Crosstalk Margins, *Design007 Magazine,* July 2018.
- The Dark Side—Return of the Signal, *The PCB Design Magazine,* August 2011.
- The Dumping Ground, *The PCB Magazine,* August 2011.
- Embedded Signal Routing, *The PCB Magazine,* September 2011.
- Return Path Discontinuities, *The PCB Design Magazine,* April 2017.
- Routing Techniques for Complex Designs, *The PCB Design Magazine,* January 2013.
- Uncommon Sense, *The PCB Design Magazine,* August 2011.



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 read past columns or contact Olney, click here.

A NEW QUARTERLY MAGAZINE

With original content dedicated specifically to flex system and PCB designers, electrical engineers and those responsible for integrating flex into their products at the OEM/CEM level, you won't want to miss a single issue of ***Flex007 Magazine****!*

Subscribe **Contribute** **Advertise**

I-Connect007
GOOD FOR THE INDUSTRY

Embedding Components, Part 6: Preparation for Active Semiconductor Elements

Designers Notebook

by Vern Solberg, CONSULTANT

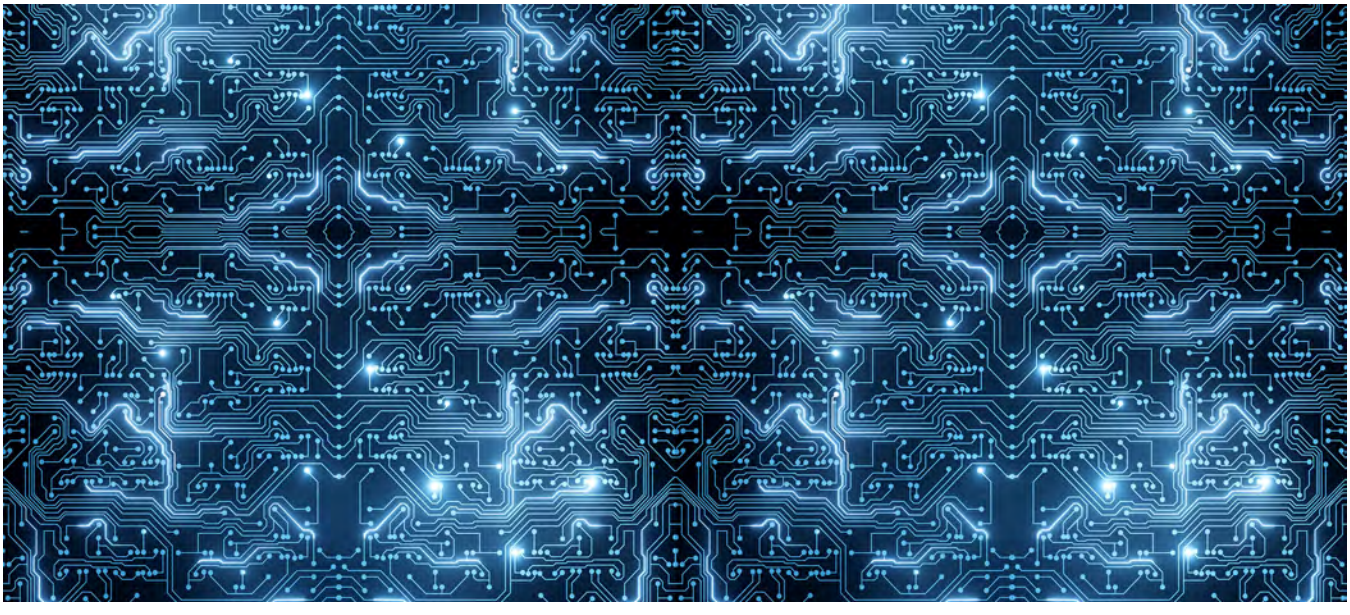
Designers are well aware that a shorter circuit path between the individual die elements, the greater the signal transmission speed, which significantly reduces inductance. By embedding the semiconductors on an inner layer directly in line with related semiconductor packages mounted on the outer surface, the conductor interface distance between die elements will be minimized.

Terminal Interface

The semiconductor fabrication process typically furnishes the die with an aluminum bond pad for the traditional gold-bond interface process. When alternative interface methods are required, the semiconductor terminals must be furnished with metalization that will be compatible with the attachment material or interface method. Companies using solder or conductive polymer for attaching the die element in the facedown orientation, for example, will need to employ techniques that enable liquidus material deposition, a solder reflow process, or a method for thermal curing followed by the dispensing of underfill polymer to negate air from between the die and laminate surface. The underfill material will require further thermal and vacuum processes to evacuate any trapped air and complete curing procedure.

Bond Pattern Redistribution

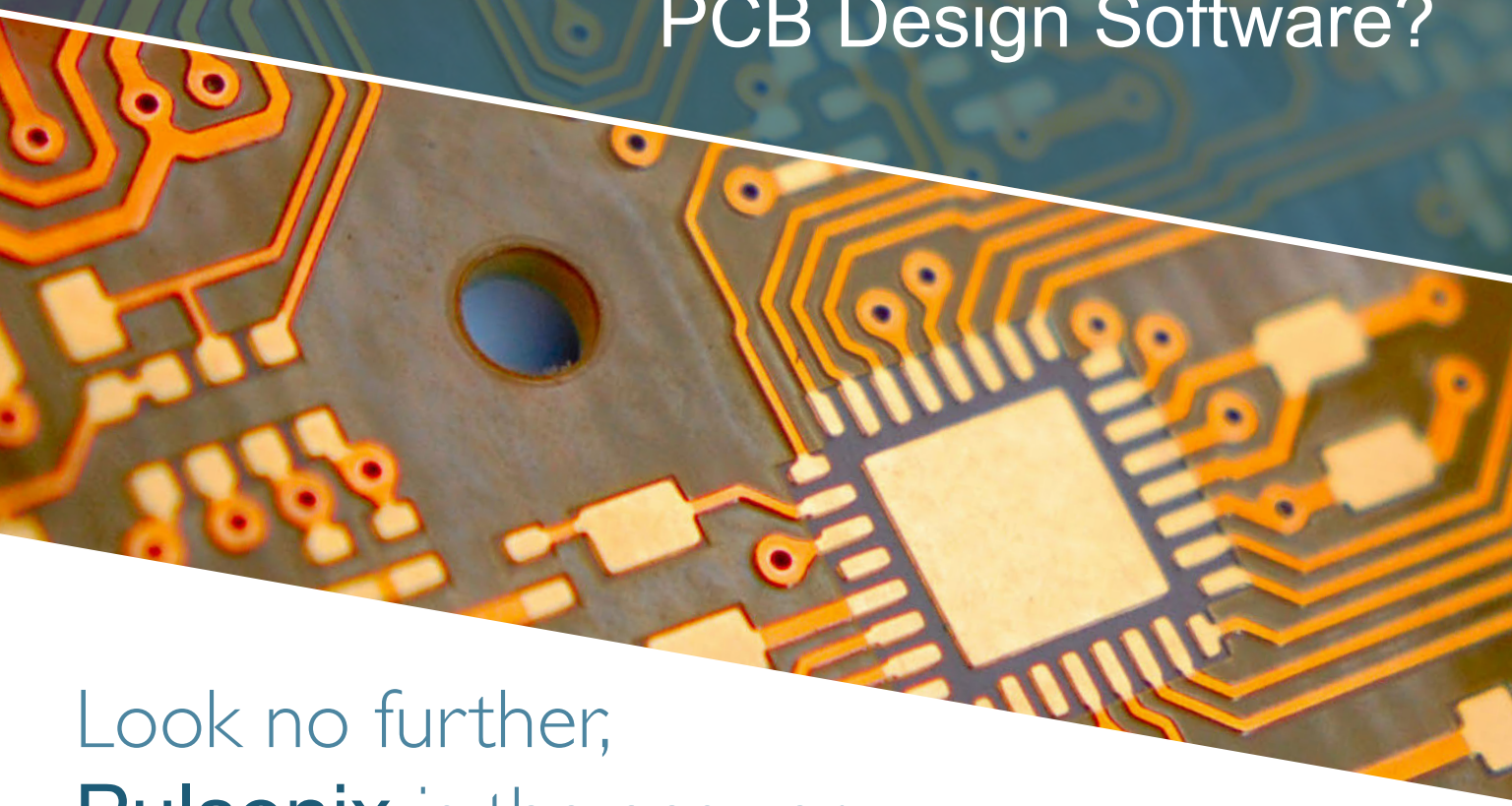
To enable efficient circuit routing, the semiconductor elements (while they remain in the wafer-level format) will likely be subject to additional metalization processes to redistribute the closely spaced wire-bond sites on the die perimeter to a more uniform array pattern within the central area of the die. In preparation



Powerful Schematic Capture
and PCB Design Software

PULSONIX

Are you looking for cost effective PCB Design Software?



Look no further, Pulsonix is the answer

- Very easy to learn and use
- Co-design in integrated PCB/3D design environment
- Import design data from your existing CAD tool
- High Speed design with Constraint Manager
- Free access to over 13 million library parts
- STEP integration with Mechanical CAD
- Reverse Engineer feature to PCB from Gerber files

Download a FREE trial from
pulsonix.com

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

Email sales@pulsonix.com

www.pulsonix.com

for adding the redistribution layer (RDL), the fabricator will first sputter-coat a metal alloy adhesion layer onto the wafers active surface. Adhesion-promoting metals include nickel (Ni), molybdenum (Mo), chrome (Cr), tungsten (W), and titanium (Ti). Resist is applied over the wafer's surface and photo-imaged to delineate the interconnect pattern and component termination sites (land patterns). The wafer(s) is then made ready for the electroplating process, building up additional metal over the exposed RDL pattern. Copper has become the preferred alloy for RDL circuit plating.

Following pattern plating, the resist coating is removed, and the remaining thin adhesion layer metalization is chemically etched from the silicon surface, leaving only the interconnect pattern and terminal lands. After cleaning, a photo-imaged passivation layer is applied to define the termination pattern and insulate and protect the conductive circuit pattern. The general RDL process sequence is shown in Figure 1. Further plating or coating processes may be applied to prevent oxidation of the exposed bare-copper terminal surfaces.

Wafer Thinning

Silicon wafers are furnished in diameters ranging from approximately 50–400 mm with a thickness range of 280–775 µm. Following the metalization (RDL) process, the wafers are commonly subjected to backside thinning and polishing. To prepare for thinning, the wafer is placed in a vacuum chuck with the active surface down and processed on the backside surface using both coarse and fine-grinding steps to reduce wafer thickness. Dry polishing follows to achieve the desired surface finish, ensure stress relief, and provide a degree of strength for the thin and somewhat fragile wafers. The preferred finished thickness of the semiconductor die element prepared for embedding will be between 130–150 µm (0.005–0.006").

Singulation Process

Saw dicing is widely employed for separating the individual semiconductor die elements from the wafer-level format. For low-volume or moderately varied processing, wafers are first mounted onto a UV tape-based film furnished on a tape frame (ring). Conventional dicing usually utilizes high-precision diamond enriched resin-bonded saw blades mounted onto high-speed spindles to cut through the silicon. A typical problem associated with controlling sawing accuracy of the semiconductor wafer is mechanical deformation or substrate warpage. To minimize material distortion during the saw process, high-pressure water nozzles flood the workpiece and blade to provide cooling.

The blades used in conventional dicing saws vary in size depending on the material thickness and "saw-street" width, which is the distance between the outer edge of each die on a wafer or panel. Blade selection is highly dependent upon street width. Material thickness plays a critical role in blade selection as well; thick materials require wider blades to provide adequate blade strength. As the blade

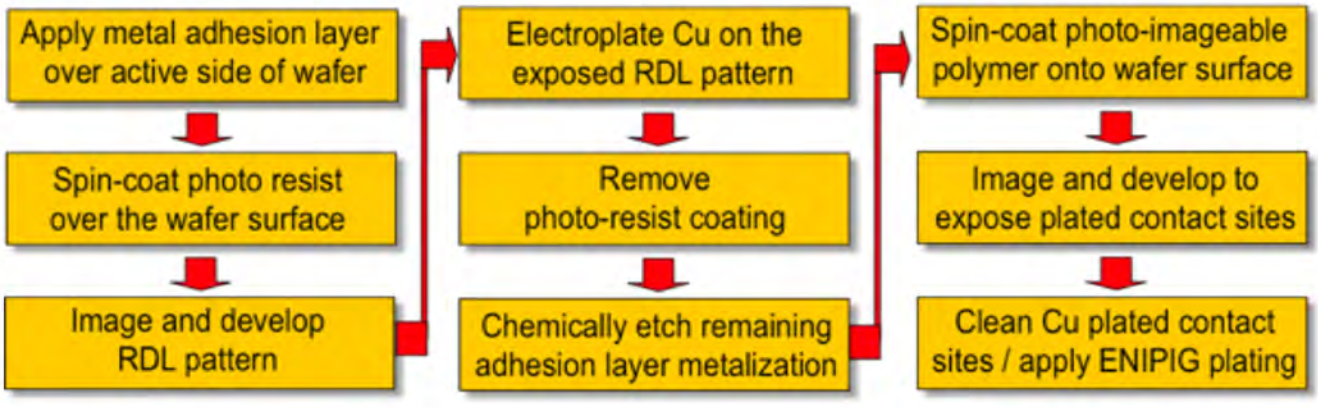


Figure 1: Basic RDL pattern imaging, plating, and etching process sequence.

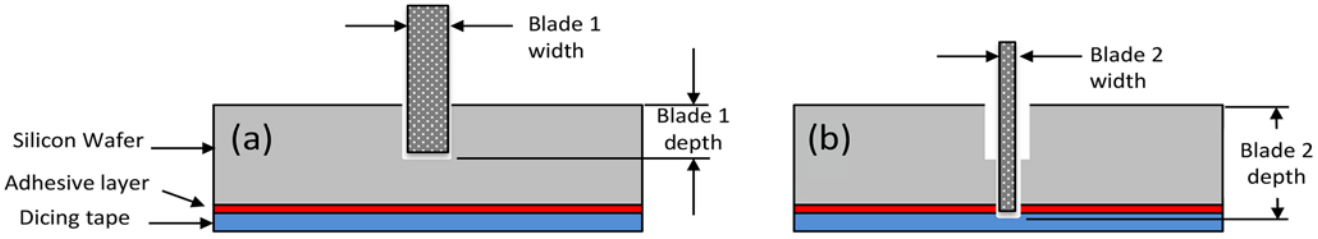


Figure 2: Two-stage saw singulation. (Source: (L) Spectrum Micro, (R) Photonics Media)

cuts through the material, a saw "kerf" slightly wider than the blade width is generated. The kerf will add an additional 10–20 µm to the street width. On a 40-µm wide blade, for example, the total material removed would be 60 µm. Saw blade width can impact process throughput as well. A tall, skinny blade will be less stable and prone to wobble and breakage when cutting thick materials. As a rule of thumb, thicker materials will require a wider blade, and in turn, the wider street width must be provided between package elements.

With conventional saw blade dicing, a single blade will cut all the way through the material. Edge chipping within the narrow streets, however, is a key quality concern. Chip propagation can be minimized by controlling the feed rate, cut mode, and blade width, concentration, and grit. Single-pass sawing is preferred because it provides greater throughput; however, if the feed rate is not optimized, there will be a greater potential for chipping.

To minimize chipping and provide a narrow street width on the active side of the die, the fabricator may adopt a "step-cut" procedure, which uses two blades to sequentially cut at different depths in the wafer (Figure 2). With the active surface facing down, a wide blade is employed to make the first cut from the backside surface of the wafer (L). A second pass follows using a narrow blade that continues through the silicon, stopping at the surface of the dicing tape (R).

Laser Singulation

Laser-dicing systems specifically designed for efficient semiconductor backend processing are available. The programmable scanner-based cutting technology is fast and flexible and provides extremely precise and small kerf widths (down to 30 µm) and very smooth edges. The systems can provide cutting speeds of up to 200 mm/s for thin wafers with a tolerance in the range of ±3 µm. Because these systems utilize continuous water jet cooling during ablation, there is no heat impact on the wafer during processing. The examples furnished in Figure 3 compare the finished

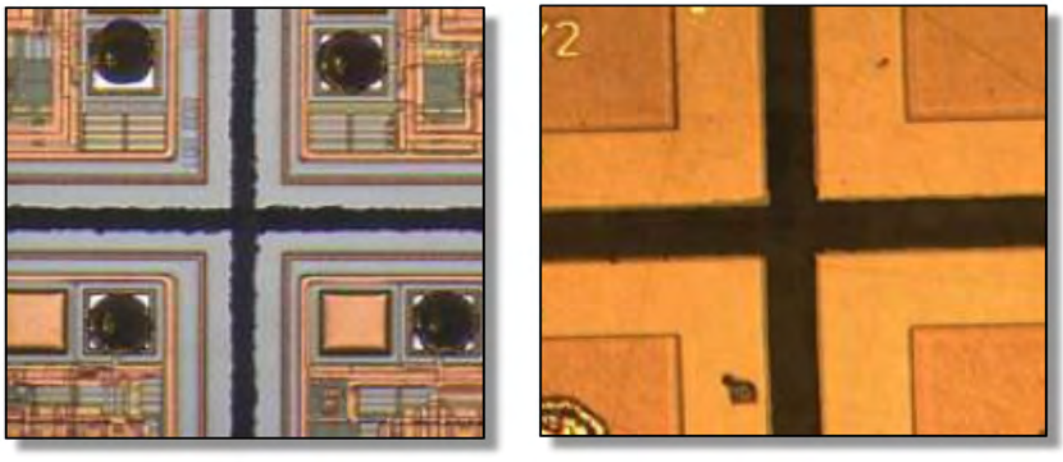


Figure 3: Comparison of a saw cut to a laser cut street-edge quality on a silicon wafer.

street-edge quality of saw and laser singulation. Following singulation, the die elements are subject to physical inspection and electrical testing.

Test and Reliability Screening

Although wafer-level testing will sort out most defective die elements, testing and screening are commonly performed while the singulated elements remain on the tape frame carrier using a flying probe tester or dedicated test platform. Testing will ensure the semiconductor meets the manufacturer's quality and reliability specifications. The phrase "known good die" (KGD) designates bare die elements that could be relied upon to exhibit the same quality and reliability as the equivalent single-chip packaged device. Meanwhile, "known quality die" (KQD) is based on general yield expectations and does not always reflect defect levels or early failure rates.

Semiconductor Die Placement

Embedded semiconductor elements can be mounted onto the core substrate with the active surface in the faceup orientation or facedown. Facedown mounting is typically selected for a direct interface (flip-chip) to the core surface. The faceup orientation will be preferred if the semiconductor requires a more direct path to the backside of the die for thermal dissipation.

Part 7 of this series will examine alternative techniques for embedded semiconductor die placement and termination methodologies. **DESIGN007**

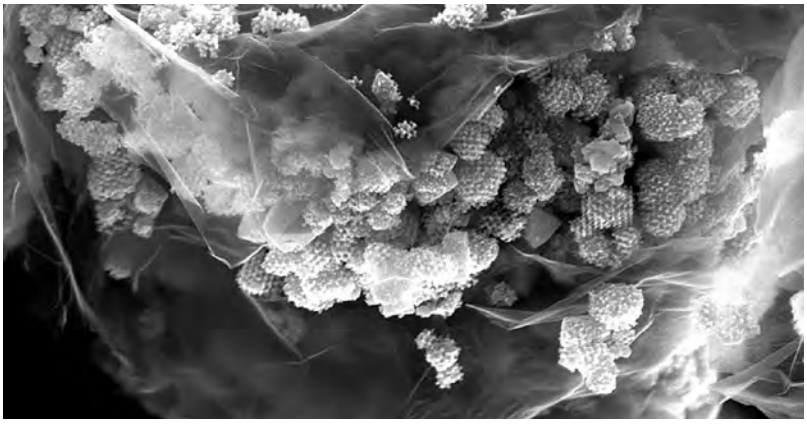
Vern Solberg is an independent technical consultant specializing in surface mount technology, microelectronics design, and manufacturing technology. To read past columns or contact Solberg, click here.

Pushing Lithium Ion Batteries to the Next Performance Level

Materials chemist Freddy Kleitz from the Faculty of Chemistry of the University of Vienna and international scientists have developed a new nanostructured anode material for lithium ion batteries, which extends the capacity and cycle life of the batteries. Based on a mesoporous mixed metal oxide in combination with graphene, the material could provide a new approach how to make better use of batteries in large devices such as electric or hybrid vehicles.

Nanostructured lithium ion battery materials could provide a good solution," says Kleitz of the Department of Inorganic Chemistry—Functional Materials of the University of Vienna, who together with Claudio Gerbaldi, leader of the Group for Applied Materials and Electrochemistry at the Politecnico di Torino, Italy, is the study's main author.

As a first step, based on a newly designed cooking procedure, the researchers were able to mix copper and nickel homogenously and under controlled manner to achieve the mixed metal. Based on nanocasting—a method to produce mesoporous materials—they created structured nanoporous mixed metal oxide particles, which due to their extensive network of pores have a very high active reaction area for the exchange with lithium ion from the battery's electrolyte. The scientists then applied a spray drying procedure to wrap the mixed metal oxide particles tightly with thin graphene layers.



"Compared to existing approaches, our innovative engineering strategy for the new high-performing and long-lasting anode material is simple and efficient. It is a water-based process and therefore environmentally friendly and ready to be applied to industrial level," the study authors conclude.

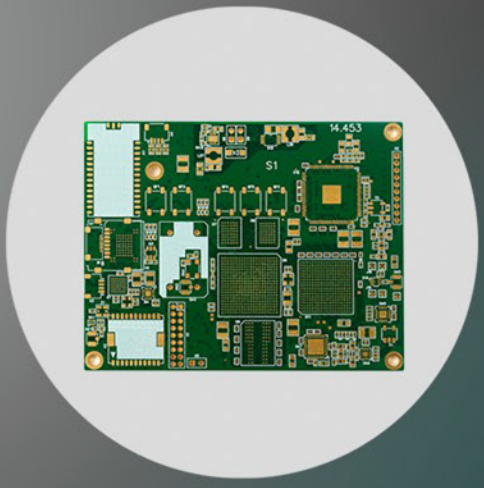
(Source: University of Vienna)

PCBCART

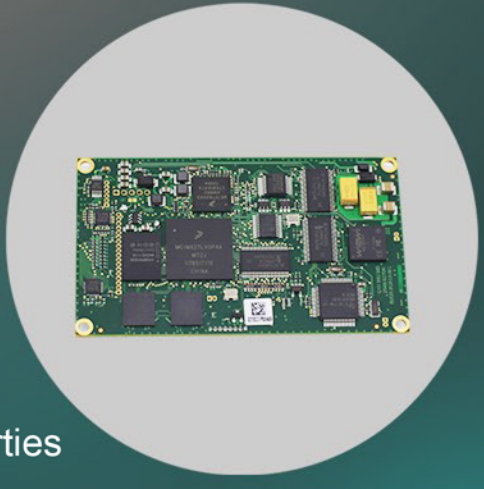
Visit www.pcbcart.com or email sales@pcbcart.com for more details and get an exclusive $200 discount!

Undoubtedly, China has become the largest supply base of PCBs and Printed Circuit Board Assemblies. With so many self-styled “reliable cheap PCB Manufacturing Solution suppliers”, are you capable of picking up a long-term partner for your projects?

CHECK HOW TO MAKE THE BEST OF CHINA'S PCB MANUFACTURING RESOURCES



PCBCart is among the very first Chinese suppliers who take care of international customers’ PCB manufacturing demands. Till now, we’ve been manufacturing PCBs for over 10,000 customers from 80+ countries, most of them have been working with us since our establishment. Here are our commitments:



- We never offer poor quality PCBs
- We give the best balance on price-quality-service
- We never share customers’ project design with third parties

Skin Depth and Its Interaction with Final Plated Finishes

Lightning Speed Laminates
by John Coonrod, ROGERS CORPORATION

Skin depth is the depth within a conductor where the majority of the radio-frequency (RF) current resides. Imagine looking at a cross-sectional view of a circular wire and being able to see how much current is within that cross-section. If the current is supplied by a battery and is direct current (DC), the amount of current is distributed evenly across the cross-sectional area of the wire. The current density is the same everywhere in the area of that wire.

If you change the current source to a sinusoidal alternating current, then you will find that there is more current on the outer edges of the wire than in the middle of the wire. If you increase the frequency a lot, you will notice at some point there is no current in the middle of the cross-sectional area of the wire, and the majority of the current is on the very outside edge (skin) of the conductor. That is the basic concept for skin depth.

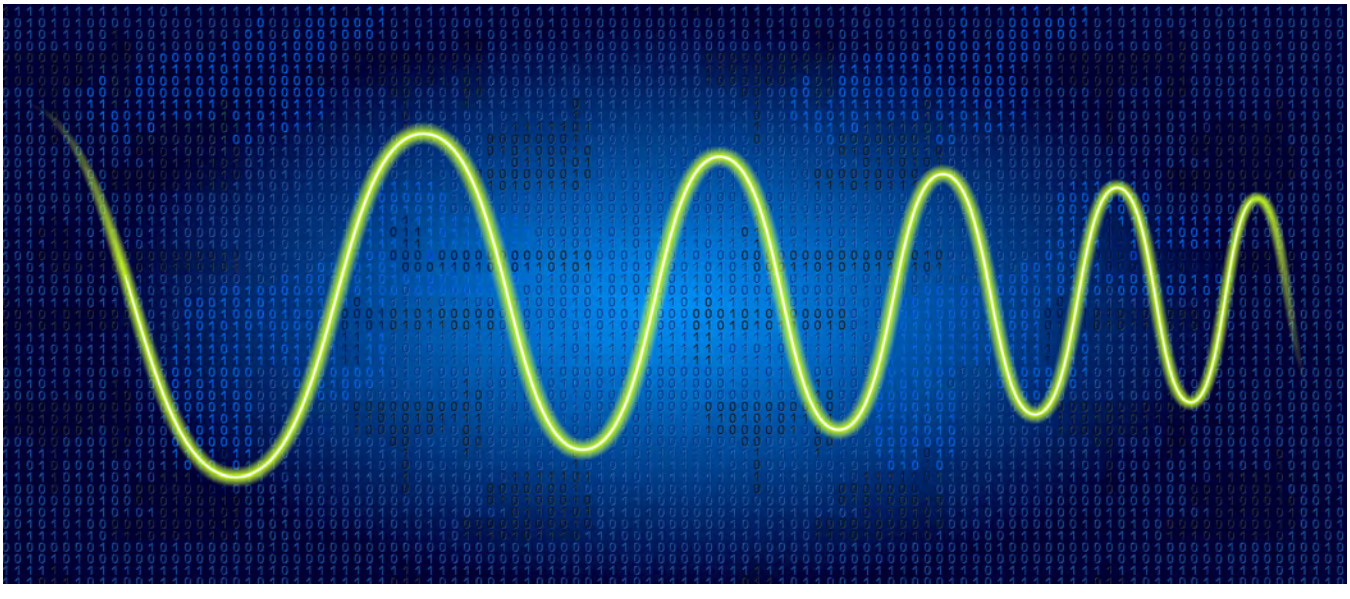
This formula will help explain some of the relationships with skin depth. A simple definition for skin depth (δ) is:

$$\delta = \sqrt{\frac{1}{\pi \cdot f \cdot \mu \cdot \sigma}}$$ Equation 1

ϖ = a fixed number
f = frequency
μ = permeability
σ = conductivity

I know some of these formulas can be a little confusing when first looking at them, but this is actually pretty easy. The "μ" value is related to the magnetic properties of the metal and copper has a relative value of about one, so the magnetic property of copper really doesn't have any effect on the equation. The symbol "σ" is related to how conductive the metal is, and copper has one of the best conductivities (high conductivity) for metals.

The reason I gave the formula is that relationships can be easily seen. For example, if you increase "f" (higher frequency), then "δ" (skin depth) will decrease. If a metal is used that



is lower in conductivity, then the skin depth will increase, which is what happens when some types of final-plated finish are applied to the conductor for a PCB. The influence of the plated finish is a rather complicated issue, but I wrote on this subject a few months ago if you want more information.

One particular plated finish that is used a lot in the industry is electroless nickel immersion gold (ENIG), and the effects of the ENIG are related to an edge effect of the conductor. At the edges of the conductor where the conductor meets the substrate, there is naturally higher current density, and the difference in the conductivity of metal at these edges will cause differences in RF performance. In the case of ENIG—assuming very low frequency where the skin depth is thick—the conductivity at the edges of the conductor will be a composite conductivity made up of copper-nickel-gold. When the frequency is increased, there will come a point where the composite conductivity will be nickel-gold. At much higher frequencies, the conductivity will be mostly related to the gold layer.

To give you a sense of the different conductivities for these metals and with the unit being 10^7 S/m, copper is 5.8, nickel is 1.5, and gold is 4.5. These values are for the pure metal. In reality, these metals are usually an alloy for how they are used in PCB processing and are slightly different for conductivity, but these are good reference values. Nickel is approximately ¼ the conductivity of copper, which is a double-edged sword for RF issues. The lower conductivity will cause more insertion loss and also increase the skin depth, which means the RF current is using more of the lossy metal.

There is another twist for ENIG that is a potential issue related to magnetic issues. Pure nickel has a very high relative permeability (μ) of about 500, but the nickel used for ENIG is an alloy with a lower μ value than pure nickel—although it is still significant. With an increase in μ, it can be seen in the skin depth formula that the skin depth will decrease. This is an offsetting factor to the lower conductivity of nickel. There are also magnetic losses associated with the metal, which are akin to the losses associated with dielectric. The dielectric losses are related to the dissipation factor (Df), and the magnetic losses are similar—but related to—the magnetic properties of the metal. Nickel does have higher magnetic losses than copper.

The following is an example of a real-world issue related to ENIG and skin depth. We had a customer tell us they were experiencing significantly different RF losses for their circuits when looking at the performance of many circuits of the same design. It was basically circuit-to-circuit insertion loss variation. It turned out that the operating frequency was at 800 MHz (0.8 GHz), which is an interesting frequency for skin depth as it relates to ENIG.

The skin depth in copper at that frequency is about 2.3 microns (approximately 92 microinches), and would be a little thicker for ENIG. The nickel layer of the ENIG can vary from 50–250 microinches depending on many factors. Normal circuit-to-circuit variation for ENIG is not that extreme, but there is a normal nickel thickness variation for ENIG that will be different for many different reasons.

What was found was that the nickel thickness variation was in the right thickness range to impact skin depth variation, which was related to the composite conductivity of copper-nickel-gold changing from one value to another with a different nickel thickness. At this frequency, a change in the nickel thickness had a significant influence on the skin depth and associated insertion loss. If the same nickel thickness variation occurred for an application operating at 24 GHz where the skin depth was about 17 microinches, the composite conductivity would not be impacted because the composite is made up of approximately eight microinches of gold and the rest is all nickel. Again, this is just at the edge of the conductor where the ENIG influences insertion loss. **DESIGN007**

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

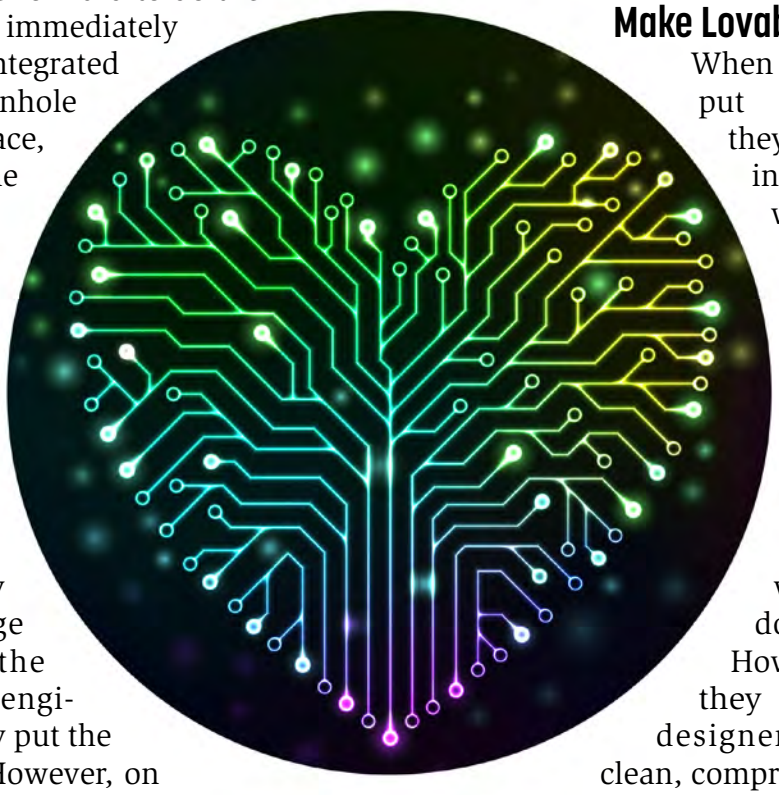
Key Guidelines for Clean Schematic Designs

Connect the Dots

by Bob Tise and Dave Baker, SUNSTONE CIRCUITS

The smoke has not even cleared, but they already know what the problem is. It was supposed to be a celebratory moment for the team, plugging in the first board from the initial shipment. Instead, the room is as silent as a morgue.

The engineer steps forward to do the postmortem and immediately sees the issue. The integrated circuit (IC) has a pinhole melted into the surface, telling the story of the destruction within. This doesn't melt the part usually; it just quietly kills it and is more of a noise thing. They trace back along the board, and the decoupling capacitor is nowhere to be found. A completely predictable voltage spike overcame the fragile circuit. The engineer knows that they put the capacitor in place. However, on closer examination, it is not close enough to the IC pad to make a difference.



Everyone turns to the PCB designer. "But I put it right where the schematic said to!" they say. The fix is easily implemented. It takes 15 minutes to produce a new design. Unfortunately, the break room already has an ample supply of coasters, and that's all this batch of boards can now be used for. The project will lose days and dollars.

How did this confusion come about?

Make Lovable Schematics

When engineers start to put together projects, they are not just making circuits—they are writing a message to the PCB designer. Their work product—and thus, their process—needs to be a helpful part of a team effort. The schematic is, at first, the space in which an engineer does their thinking. However, by the time they hand it off to the designer, it needs to be a clean, comprehensible document that eliminates vagaries.

In the nightmare scenario above, the designer ends up in the crosshairs, but what could the engineer have done to prevent the problem and save that first run of boards from

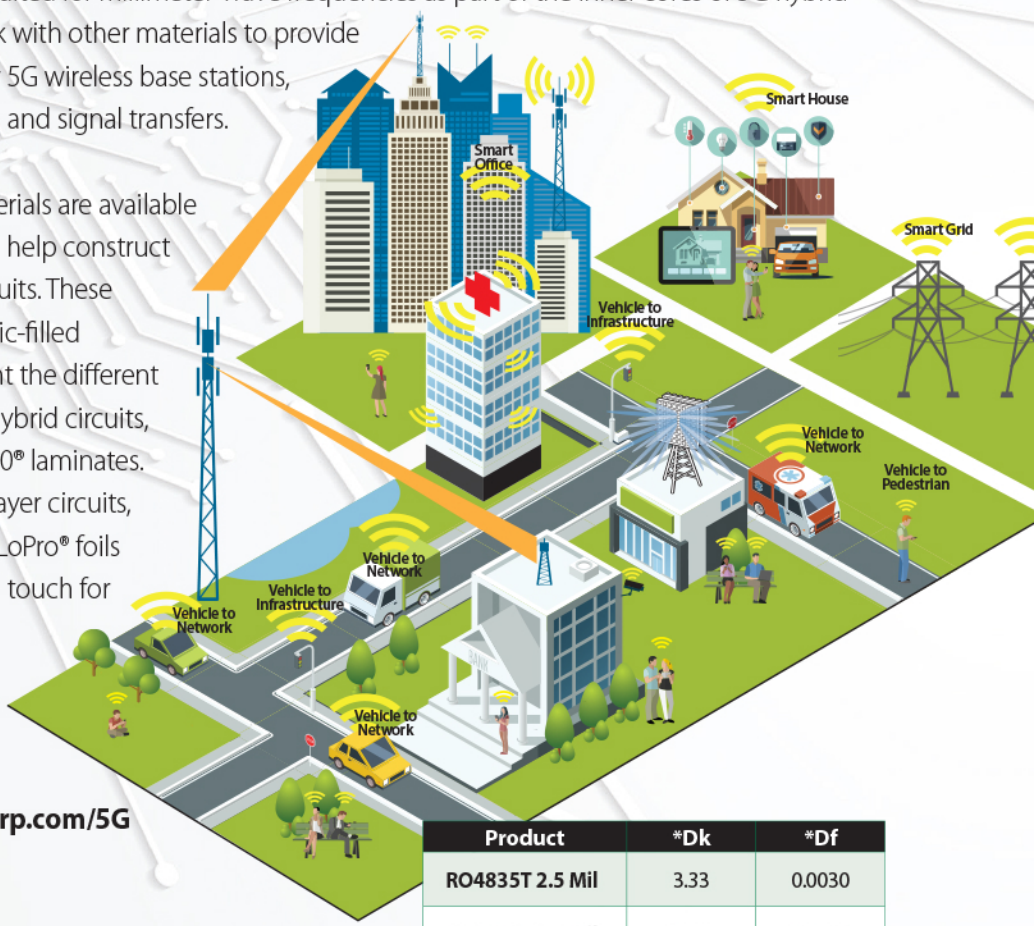
5G: Higher Frequencies!

Do you have the **right** circuit materials?

Frequencies at 28 GHz and higher will soon be used in Fifth Generation (5G) wireless communications networks. 5G infrastructure will depend on low-loss circuit materials engineered for high frequencies, materials such as RO4835T™ laminates and RO4450T™ bonding materials from Rogers Corporation!

Rogers RO4835T spread-glass-reinforced, ceramic-filled laminates are low-loss materials in 2.5, 3.0, and 4.0 mil thicknesses. They are well suited for millimeter-wave frequencies as part of the inner cores of 5G hybrid multilayer PCBs. They can work with other materials to provide the many functions needed by 5G wireless base stations, including power, signal control and signal transfers.

Rogers RO4450T bonding materials are available in 3, 4, and 5 mil thicknesses to help construct those 5G hybrid multilayer circuits. These spread-glass-reinforced, ceramic-filled bonding materials complement the different materials that will form these hybrid circuits, including RO4835T and RO4000® laminates. And for many 5G hybrid multilayer circuits, Rogers CU4000™ and CU4000 LoPro® foils will provide a suitable finishing touch for many hybrid multilayer circuit foil lamination designs.

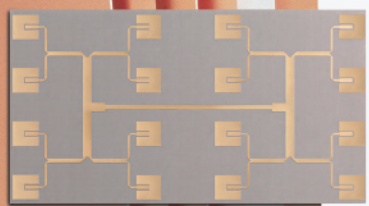


5G is coming! Do you have the right circuit materials?
Learn more at **www.rogerscorp.com/5G**

PLEASE VISIT US AT APEX EXPO IPC 2019 BOOTH #117

Product	*Dk	*Df
RO4835T 2.5 Mil	3.33	0.0030
RO4835T 3.0 Mil	3.33	0.0034
RO4835T 4.0 Mil	3.32	0.0036
RO4450T 3.0 Mil	3.23	0.0039
RO4450T 4.0 Mil	3.35	0.0040
RO4450T 5.0 Mil	3.28	0.0038

** IPC TM-650 2.5.5.5 Clamped Stripline at 10 GHz - 23°C*



ROGERS CORPORATION
Advanced Connectivity Solutions

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

a life of coffee stains? Good schematics are good team play.

Best Practice Every Time

Every day at Sunstone, we get calls from teams that have sent a design in prematurely and need to make a last-minute change. If they get to us in time, we will pull the design back. At the very least, they may lose some productive days fixing and resubmitting the design.

We all know that there is a best practice for the work, but we slip from it, often in the effort to save time. In teams that we work with, we want the mindset to be, "I was in a hurry, so I did everything right." What is right? It is starting at high-altitude block diagrams, breaking each block into one or more schematic sheets, checking flow and accuracy carefully, and then designing the board.

We all know that there is a best practice for the work, but we slip from it, often in the effort to save time.

Of course, the engineer and the designer go over the design carefully to make sure that it matches the specifications, right? One would hope so, but busy people on tough deadlines can easily decide to lean on automation tools.

Good Designs Need Good Inputs

Designers don't read minds—they read schematics. Engineers need to make that document as functional as possible. The engineer may understand the locations for all their bypass capacitors implicitly; they end up listed in the corner of one page, but that relies on the designer to interpret that intent and properly place the elements. A more thoughtful engineer will locate devices in roughly the manner of the final design to help the designer avoid bad interconnects, placement assumptions, or other errors.

In addition, it never hurts to provide explanatory notes about the elements. We have never heard a designer say that the engineer provided too much guidance for their work. A key element is to utilize proper naming conventions for connections (i.e., net names). We all know that the automated default labels that our design software provides are far from intuitive. Help the designer avoid embarrassing connection mistakes that will slow the project down by creating—and consistently using—connection labels a human can understand.

Working in "Expert Mode"

Speaking of interconnects, the designer's job is made easier if they have software on their side. Some engineers—again, often thinking they are too busy for best practices—will do what we call "working in expert mode." They give implicit connections using port symbols for the entire schematic, thereby creating no trail for the designer to follow. While it saves the engineer a couple of minutes, that time is effectively lost while the designer sorts out the nest of connections. The software tracks and confirms those connections for a reason. It may feel like an obstacle for the engineer, but for the overall project, these tools are a lifeline.

Use DRC and Resolve Errors

Everyone runs the design rule check (DRC), but not everyone actually resolves those errors. It is too frequently used "just for documentation issues" that won't theoretically affect the performance of the board to be left in place. Over time, these errors build up and create a fog of confusion that makes it hard to maintain quality. A team should expect that every build is accompanied by a DRC report that reads "no warnings or violations."

Recommendations for Clean Schematics

Below are our top six recommendations for clean schematics:

1. Start with a block-level diagram

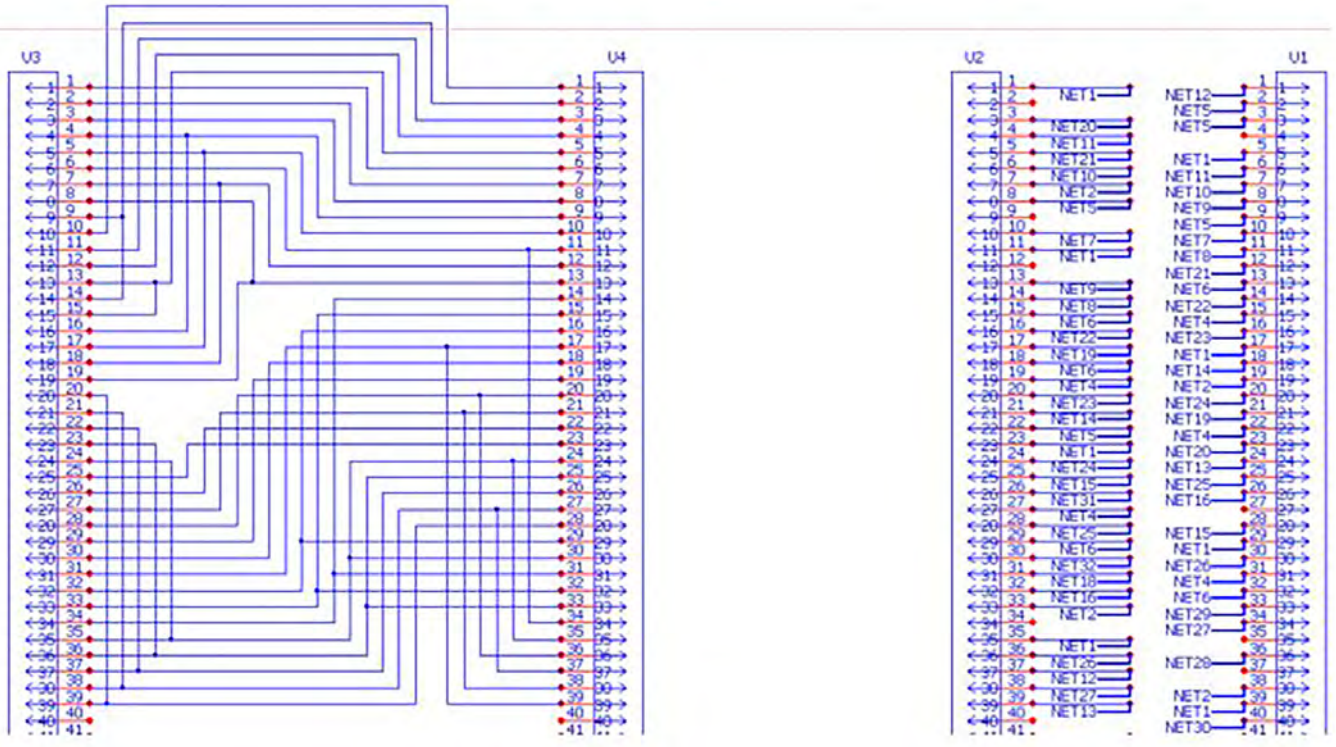


Figure 1: Schematic examples.

2. Design the schematic in the design program, or at the very least, label all connects with comprehensible net names and avoid using autogenerated names
3. Lay out the schematic in a manner that clarifies locations; particularly in complicated designs, it keeps the designer confident and quick
4. Leave a legacy by labeling the schematic so that the next person can understand it
5. Pride cannot get in the way of good work; designers should be comfortable asking for clarification if they are uncertain of how to proceed
6. Use the DRC and do not allow any design with schematic errors or warnings to be built

Winning the Long Game

Well-designed schematics pay off not just for the current project but also into the future. If your schematic is clear and nice to look at, you and other engineers can utilize blocks for other similar projects. Teams that commit to good schematic practices win the long game.

Commit to Process

We are all thinking about making great final products. Designers want to make great boards. Engineers want to make great schematics. We also need to think about making the work of the next person easier to ensure that they do not make avoidable errors. Everyone—from management to manufacturing—suffers from go-it-alone engineering and benefits from clean schematics.

In Figure 1, the schematic on the right looks neater, but you can easily trace the interconnects in the one on the left. Which would you rather try to follow? **DESIGN007**

Bob Tise and Dave Baker are engineers at Sunstone Circuits. To read past columns or contact Tise and Baker, click here.


Bob Tise


Dave Baker

Top Five Tips to Protect PCBs from Harsh Environments

Sensible Design

by Alistair Little, ELECTROLUBE

Last month, I began my new series of columns on encapsulation resins by selecting five frequently asked customer questions regarding resin chemistries and properties, and typical resin applications and their limitations. In this column, I'm taking this introduction a step further by listing what I believe are the top five tips for circuit designers and manufacturers who seek to ensure that the in-service reliability and longevity of their electronic assemblies and products are fully addressed.

Five Top Tips

1 First, think very carefully about the sort of environment your PCB is likely to encounter. It is easy to over-engineer a product so that it will survive the very worst of conditions, but worst conditions may only be fleeting or transient. Therefore, a resin solution with a lower temperature performance specification will often cope. Take temperature extremes as an example; your application may experience occasional temperature spikes of up to 180°C, which you might feel deserves treatment with a special resin. However, such excursions may only be short-lived. Under normal operating conditions, the PCB might only be subjected to a maximum temperature of 120°C, opening up a wider choice of resin types and methods of application.

Similarly, the required chemical resistance of your chosen resin will depend on the duration and/or extent of the chemical contamination. For example, there is a considerable difference in terms of the extent of chemical damage between a thin layer of a contaminating chemical on the resin surface that is wiped off within five minutes, and 500 ml of a chemical present on the resin surface for one hour or more. Complete immersion takes the requirements to an even higher level. Furthermore, the range of chemicals that a PCB might eventually be exposed to is often quite limited, and almost certainly not the broad range that is frequently listed at the design stage just to be on the safe side.

2 Environmental factors that normally affect a PCB include temperature, chemical attack, physical shock (vibration), and thermal shock. The trick is to decide which of these is likely to have the greatest impact upon your PCB and then concentrate on making an appropriate resin choice. Each of the three main resin types (epoxy, polyurethane, and silicone) have strong points as well as weaknesses.

Silicone resins have the broadest continuous operating temperature range of any of the resin chemistries, so they are a natural choice

BLACKFOX
Premier Training & Certification

Experience the Blackfox Difference!

Premier IPC Training & Certification

- High quality training and customer service
- Offering 6 certification programs in four locations
  - *Colorado, Arizona, Mexico, Malaysia*
- Military Veterans Advanced Manufacturing Program
- **NEW!** IPC/WHMA-A-620 Space Addendum for trainers and specialists
- Online store supporting the training industry

PLEASE VISIT US AT
APEX EXPO IPC 2019
BOOTH #2807

CLICK FOR COURSE SCHEDULE

For more information contact us at
888.837.9959 or sharonm@blackfox.com

www.blackfox.com

for both high- and low-temperature applications as well as those subject to thermal shock. They also maintain their flexibility over this temperature range with very little sign of degradation over time. On the downside, silicones have poor adhesion on certain substrates, and their chemical resistance is not as good as that provided by an epoxy resin.

As well as offering excellent chemical resistance, epoxies provide good temperature performance. However, due to their rigid nature, epoxies are not as good at protecting against physical shock. Polyurethanes, on the other hand, have excellent moisture and physical shock resistance but deliver poor high-temperature performance. As a result, polyurethanes are best restricted to applications operating in the -40 to +120°C region. However, they do provide similar levels of flexibility and better adhesion to many substrates compared with silicones at a lower cost.

3 Where the PCB is mounted in an enclosure (into which resin is poured to encapsulate it fully), the enclosure material must be considered. In the case of plastic enclosures, these usually are injection moulded, so there might be traces of the release agent on the surfaces, which will result in poor resin adhesion unless the agent is removed beforehand. Some plastics are very moisture sensitive and likely to undergo dimensional and other physical changes in humid conditions, which will impose physical stresses on the enclosure, encapsulation resin, PCB, and components. Where the enclosure is made from steel, aluminium, or other metals, then the differences in the coefficient of thermal expansion (CTE) between the enclosure and resin will have to be taken into account as well as the surface treatment used, which might affect the adhesion of the resin.

4 Typically, the thicker the resin layer, the better the level of protection. However, unless all of the components on the PCB are of a uniform height, then the thickness of the resin layer will vary across the board and potentially slightly different levels of protection will be provided for individual components. Good board design and component selection will go a long way towards mitigating this type of problem, but the thinnest resin layer must be assumed to be the level of protection offered across the board. Naturally, with the desire and need to reduce weight and/or volume, designers are inclined to reduce the amount of resin applied. Nonetheless, the expected service life needs to be factored in with thicker layers generally providing better long-term protection.

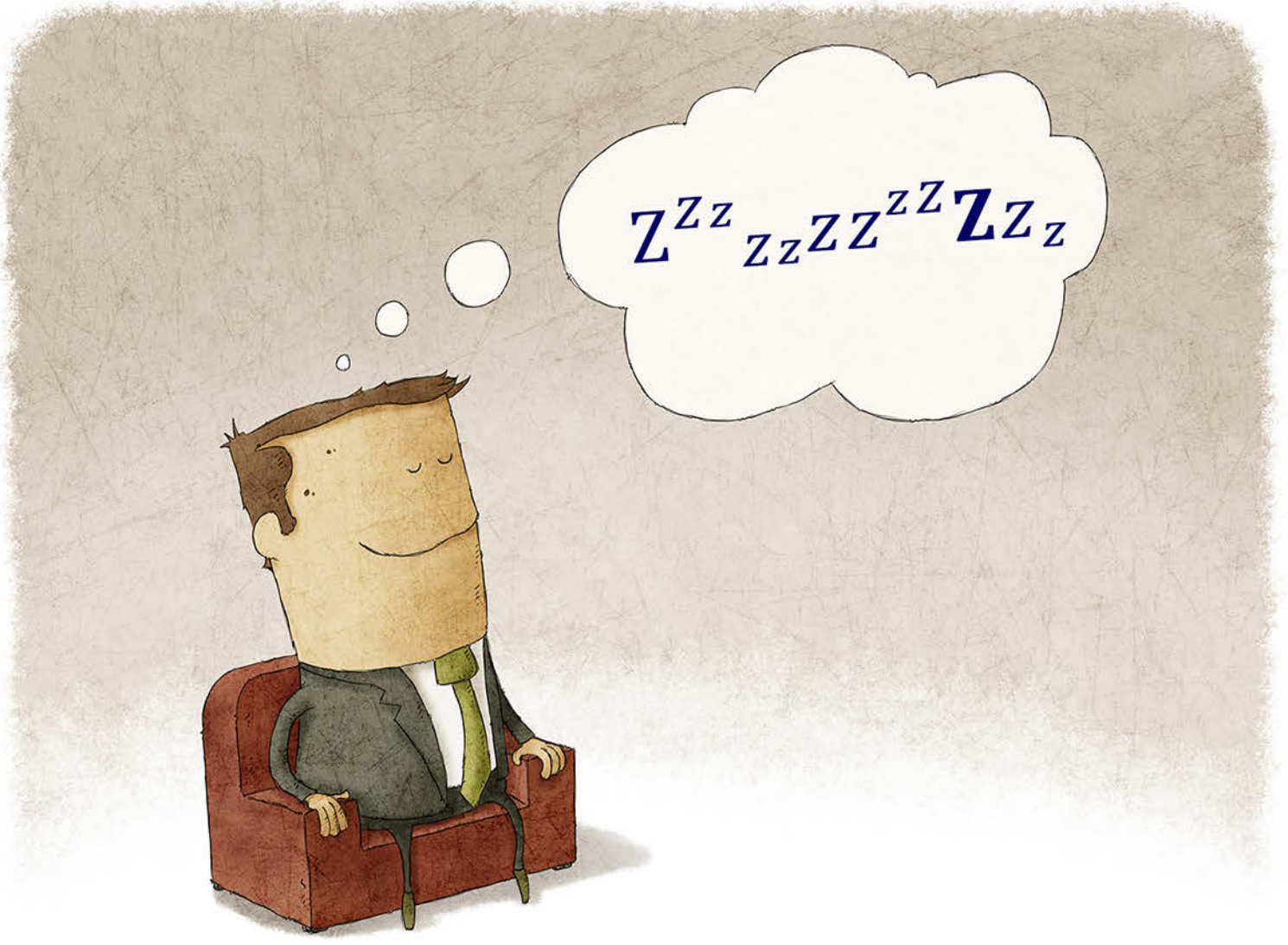
5 Remember, before you even consider resin encapsulation or potting, the PCB needs to be thoroughly cleaned. Surface contamination can have a negative impact on the protection levels offered by encapsulation, particularly in cases of chemical resistance as it provides an easier route for chemicals to penetrate. In addition, contaminants will adversely affect the resin's ability to absorb physical and thermal shocks due to the weak layer formation between the resin and the PCB, which ultimately leads to delamination. Of course, after cleaning, any solvent or cleaning solution must be removed, and the PCB needs to be thoroughly dried before resin application.

Conclusion

By paying attention to these basic design pointers, you are likely to achieve the levels of reliability and long service life that will ensure happy relationships with your customers. Over the coming months, I will take a look at a wide range of issues concerning the correct choice and application of resins, so be sure to check out this column in future issues of *Design007 Magazine*. **DESIGN007**

Alistair Little is global business/technical director of resins at Electrolube. To read past columns from Electrolube, click here. To download your copy of Electrolube's micro eBook, The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments, click here.

We DREAM Impedance!



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

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

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

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

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

iCD
In-Circuit Design Pty Ltd

www.icd.com.au

MilAero007 Highlights

Rick Hartley is Bullish on PCB Design, 3D Printing ▶

At the recent PCB West in Silicon Valley, Consulting Technical Editor Tim Haag met with long-time design industry veteran Rick Hartley to discuss the changing landscape of circuit board design, the layout designers of the future, and how designers can benefit from 3D printing of circuit boards.

Substrates for Advanced PCB Technologies: What Will the Future Hold? ▶

The UK chapter of the global IMAPS community of electronics and microelectronic packaging engineers shared a wealth of knowledge and wisdom about PCB substrate technology trends, developments, and future requirements in a webinar on the first of November.

Standard of Excellence: Forging Partnerships Through Adversity and Problem Solving ▶

For the past few months, this column has discussed how to find and work with a great PCB vendor, and most importantly, how to form a strong, productive partnership. This month, Anaya Vardya will address how adversity can forge a great partnership between you and your PCB vendor that will last for life.

NASA Looking to Tiny Technology for Big Payoffs ▶

NASA is advancing technology that could use large amounts of nanoscale materials to launch lighter rockets and spacecraft than ever before. The super-lightweight aerospace composites (SAC) project seeks to scale up the manufacturing and use of high-strength carbon nanotube composite materials.

Increasing Productivity for Flex Fabricators ▶

Barry Matties and Nolan Johnson of I-Connect007 met with Shane Noel and industry veteran Mike Jennings of ESI to discuss the introduction of their CapStone laser tool, a product aimed at doubling their flex circuit fabricators' throughput. Mike also shares advice for fabricators who are looking to move into the ever-growing flex market.

Ventec at electronica 2018: No Compromises for High-frequency Materials ▶

Ventec's IMS material families, thermally conductive and standard laminates and prepregs for multilayer PCB's, Ventec has teamed up with EMI Thermal to provide a range of thermal interface materials (TIM) to the European market.

ERAPSCO Inks $40M in Navy Sonobuoy Contracts ▶

Ultra Electronics Holdings plc (ULE) and Sparton Corporation announce the award of subcontracts valued at $39.6 million to their ERAPSCO joint venture, for the manufacture of sonobuoys for the United States Navy.

Nano Dimension Partners with Productivity Inc., Expands Reseller Network ▶

Nano Dimension announced a new reseller agreement with Productivity Inc., significantly expanding the company's North American channel partner ecosystem.

Sparton Receives 2018 Asia Pacific Entrepreneurship Award ▶

Dung Tran, managing director of Spartonics Viet Nam Co. Ltd, has been presented with the 2018 Asia Pacific Entrepreneurship Award for exemplary leadership and innovation in the electrical and electronics industry.

ISO 9001 Quality Management

AS-9100D

PROTOTRON.COM
(888)847-7686

Your Passport to Global PCB Sourcing

We are pleased to announce that with the same fine service, quality, delivery and attention to detail that our customers have grown to rely upon, we are now adding a complete global solution to meet all of your needs, both today and in the future.

Our Solutions:

- *FLEX & RIGID-FLEX*
- *HDI / MICROVIA*
- *COMPETITIVE VOLUME PRICING*
- *PRE-QUALIFIED OFFSHORE PARTNERS*
- *QUICK-TURN PROTOTYPES*
- *COMPLEX MULTILAYER*
- *MIL-SPEC PRODUCTION*
- *SPECIALTY MATERIALS*

MORE INFO
Global PCB Sourcing
PDF DOWNLOAD

FREE BOOK
DOWNLOAD
I-007eBooks.com

Prototron Circuits

A COMPLETE GLOBAL SOLUTION

REDMOND, WASHINGTON • TUCSON, ARIZONA • GLOBAL PARTNERSHIPS

ALTIUM Prepares for Munich Show as Growth Continues

Article by Andy Shaughnessy
I-CONNECT007

It's been just two months since the AltiumLive event drew several hundred designers to San Diego, California, and Altium is already gearing up for the next show in Munich, Germany (January 15–17, 2019). I recently spoke with Chris Donato, VP of global sales for Altium, about the upcoming AltiumLive show as well as the company's growth over the past few years.

Andy Shaughnessy: Chris, welcome. Why don't you start by telling us a little about January's AltiumLive event in Munich? What can attendees expect at your sophomore event?

Chris Donato: Thanks, Andy. AltiumLive is quickly becoming one of the largest educational events that focuses exclusively on PCB design. Attendees in Munich can expect four fantastic keynote presenters: Thomas Wischnack of Porsche Engineering, Dan Beeker of NXP Semiconductors, Alun Morgan of Ventec, and Max Seeley of 3M.

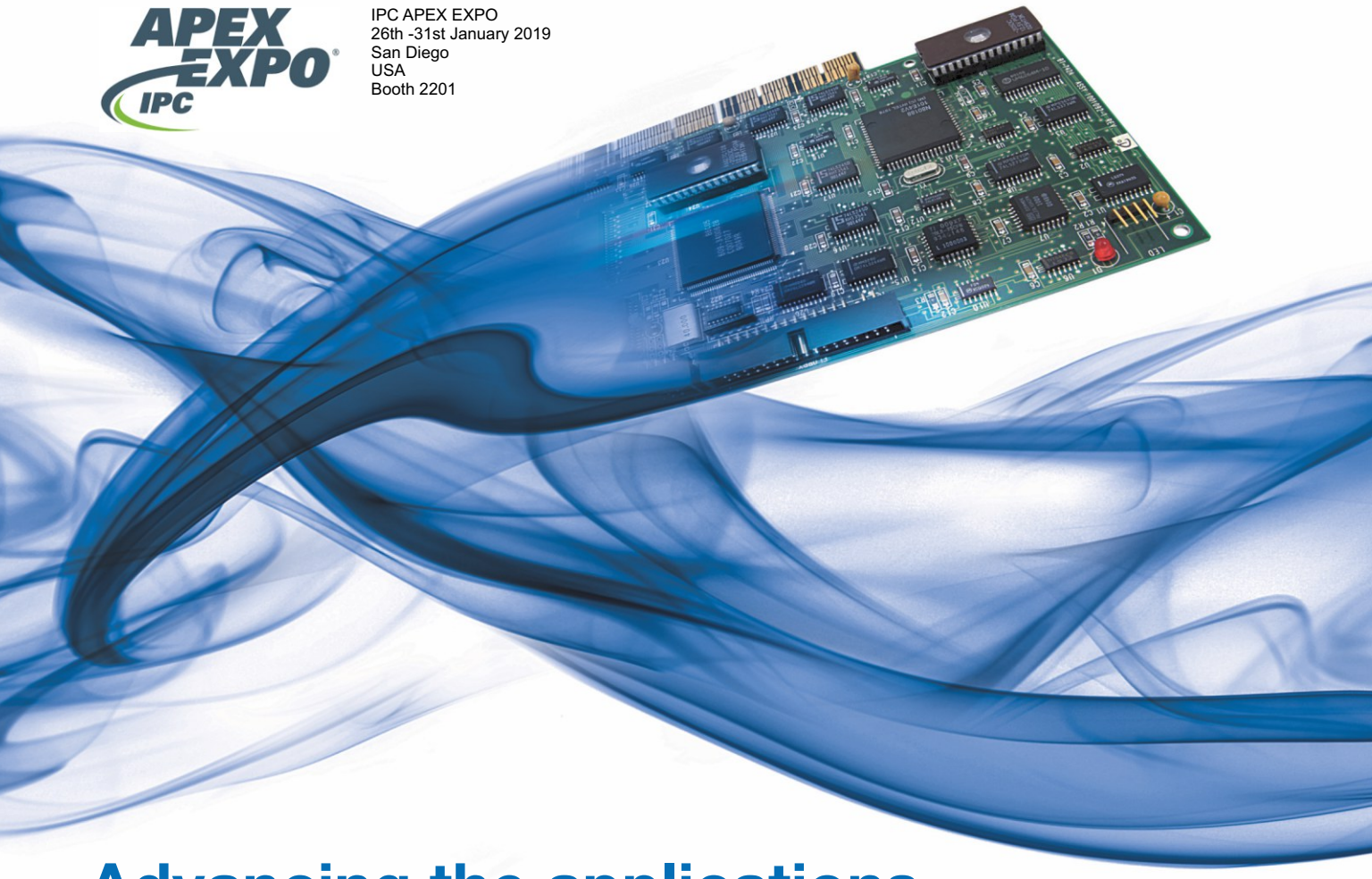
We also have a full-day high-speed class taught by Lee Ritchey on the front end. That class runs in parallel with our University Day, which features a variety of different Altium Designer courses. In addition to that, 12 presentations from Altium Designer users and supply chain experts are scheduled, and six professional development courses are available on topics like multi-board, advanced design rules, and constraints and documentation. That's 30 sessions in total, and they're all filled with some amazing education opportunities not only for our users but for us as well.

Lastly, we'll host two sessions to present Altium Designer 19, and finally, Altium 365—a cloud-based platform that allows for unprecedented collaboration between all stakeholders in the design-to-realization process. It's certainly an exciting time to be a part of Altium.

Shaughnessy: Now, let's segue a little. How are things going at Altium? I understand you've had some real growth lately.

Donato: First, thank you for giving me this opportunity. I've been here for close to 15

IPC APEX EXPO
26th -31st January 2019
San Diego
USA
Booth 2201



Advancing the applications of conformal coatings

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

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

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

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

Tel: 888-501-9203

www.electrolube.com

ELECTROLUBE
THE SOLUTIONS PEOPLE

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

years, and I've never felt as excited as I am now. We've really set our sights high. Since 2014, we've had over 7,100 new customers come on board, which has been quite amazing. That comes out to more than 10,000 new licensed users of Altium Designer.

Chris Donato

Shaughnessy: So, that's from FY 2014 to 2019, basically.

Donato: Yes, almost five years coming up here shortly. During that time, the company has doubled our revenue from $70 million to over $140 million, closing in on our goal of $200 million by 2020. Our subscription tools, which are the number of active licenses that we have, have grown from 26,000 to 40,000. It's been exciting to see this continued growth.

Shaughnessy: A couple of years ago, you were just excited to hit the $100-million mark, and now you're at $140 million.

Donato: Our fiscal year ends this coming June. Then, the year after that, we plan to be at $200 million. By 2020, we are aiming to have doubled our revenue again. I can map these growth years with our continued innovation that our R&D team has shown every year. Last year, we had this easy, powerful, modern update of the Altium Designer product—Altium Designer 18. That brought on a whole new world of opportunity with its 64-bit capability, and here we are on the precipice of yet another release shortly.

Shaughnessy: What do you think is driving this growth?

Donato: One big factor I've seen as a company is our singular focus on PCB design; that's what has gotten us to this point. We've been extremely focused on PCBs in our user community. I think that's our big differentiator as a company. We are very committed to users; we talk to our users, take feedback from them, and interact with them. Recently, we just finished an AltiumLive event in San Diego, which was a phenomenal experience for not just users but the PCB community as a whole. This singular focus has brought us to where we are now, but I think the next leg of this journey begins with Altium 19 and 365 where we will take this singular focus on PCB and expand it throughout the connecting ecosystem. We strongly feel that Altium 19 and 365, which have cloud-enabled capabilities, are the right vehicles to not only take us to the next level but also to our goal of reaching 100,000 subscribers.

Shaughnessy: When people think about Altium as a company, they often think of Altium Designer, but you have other products too. What's your strategy for those products and the acquisitions of the last few years?

Donato: That's allowed us to not only continue our singular focus on PCB, which has gotten us here today, but also to address the transformation in the electronics industry that we see with all of these connected devices. We see lots of new companies having to connect devices that never had to be connected to the internet before. To continue this innovation and help enable our customers to take full advantage of this trend, we need to connect beyond the PCB design space and into the ecosystem that surrounds it into disciplines and domains like manufacturing and the bill of materials. Some of our acquisitions have been for the technology, and we've taken that technology and put it into some of these new capabilities. Some of these acquisitions are also running business units, such as Octopart, which is—if you will—the Google search of components.

Shaughnessy: Acquisitions like Octopart are integrated into Altium Designer, for instance.

Donato: Exactly. Some of these acquisitions stand alone, and some of them partner with the Altium Designer family of tools; the technologies get shared to complement and give access across the user community. It's really exciting to see that we've continued in this fashion of PCB focus, and yet we've been able to move outside of PCB design and bring in these other companies that give us the ability to expand into the ecosystem and start to connect design to manufacturing.

Shaughnessy: One thing that I notice when I talk to designers is that the lines are blurring dramatically between ECAD and MCAD. Beyond that, there are a variety of stakeholders, as you said: the manufacturer, EMS company, purchasing agent, and manager. Is it possible to please all of these different stakeholders?

Donato: One of the next major capabilities in Altium 19 is the seamless ability to connect PCB design to the mechanical space—not just one mechanical vendor, but multiple mechanical vendors in an effortless way that we haven't seen in the industry yet. Altium was probably one of the first—if not the first company—to take PCB into the 3D space, so we have a bit of a history going down this road. Looking beyond ECAD/MCAD integration, Altium 365 also addresses this by enabling real-time collaboration between designers, fabricators, material suppliers, EMS partners, managers, and every contributor or stakeholder involved in the design-to-realization process. Instead of DFM, users will be able to design *with* manufacturing in real time, and it all happens right in their browser. You don't have to install special software; it's just there, and it works.

As you said, Andy, the lines are blurred. We're not throwing designs over the fence anymore. Stakeholders are working in parallel now and have tight constraints—not only design constraints but business pressures that more and more customers are feeling on a regular basis, which we see in our daily interactions. We need to have tighter collaboration between PCB designers and all of their parallel disciplines or counterparts straight down to the manufacturing floor itself.

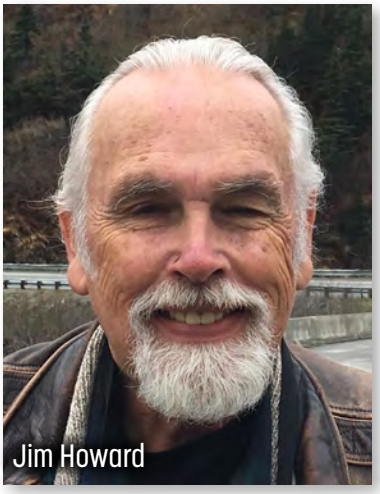
In Altium Designer 19 and 365, we'll see all these capabilities take things to the next level. The possibilities are truly endless here.

Shaughnessy: As we both said here, the lines are blurring, and time-to-market is a driver. What are some other things that you see from where you are sitting that are driving this need for seeing seamless integration as we've never seen before?

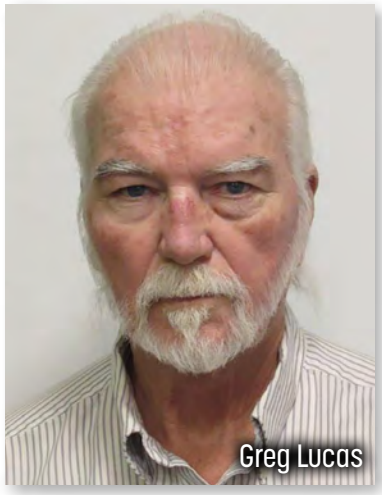
Donato: I'd say it's just the world pushing it. What I mean by that is that everything has to be connected to the internet. If a company doesn't make a smart device, they need to make one, or they won't have a company anymore. We see it constantly from our customers; they are being pushed by the global economy. It is pushing the demand to have smaller, better, cheaper, faster products, which is pushing our engineers who then push us. I think this big, massive wind in our sails is pushing everybody and driving the need for faster, tighter designs done right the first time. I think that's the other economic pressure from companies. They know that if they don't get their product out faster, somebody else will.

Shaughnessy: Thank you again, Chris. Maybe I'll see you in Munich.

Donato: Great. Thank you, Andy. **DESIGN007**


Jim Howard

A Fractal Conversation with Jim Howard and Greg Lucas


Greg Lucas

Interview by Barry Matties and Andy Shaughnessy
I-CONNECT007

Veteran PCB technologists Jim Howard and Greg Lucas have made an interesting discovery: Certain shapes of copper planes make a PCB run more efficiently than other shapes, a process they dubbed fractal design. It doesn't appear to cost a penny more, and testing suggests that fractal design techniques could eliminate edge noise.

Barry Matties and Andy Shaughnessy asked Jim and Greg to discuss the fractal design process, as well as some of the advantages of using this technique on your next PCB design.

Barry Matties: Gentlemen, tell us how you all got started with fractals and fractal design.

Jim Howard: Greg and I were in the midst of developing a technology known as buried capacitance, and we had a number of tests that we needed to run on PCBs to determine exactly what the effects of having this very thin distributed capacitance within the board would accomplish. We discovered some effects that really didn't gel with what we were doing at that time, but they were of interest.

A few years later, we went back and thought, "These things could be useful," because we discovered that certain shapes on the copper planes of the PCBs—which, at that time, were only dictated by anti-pads and various cuts and things in the copper—either increased or decreased its ability to operate efficiently as a power/ground plane. We didn't know precisely what that effect was.

Then, I spent a few years working for a Chinese company developing some technology for them, and they wanted to see if this was something that I could work on in the future. I found we could create an effect with a specific shape of putting anti-pads, cuts, and other things where we wanted to, but the problem was that no designer was ever going to let you do that. That's an impossible scenario.

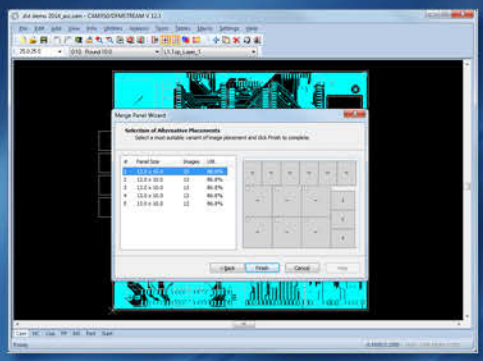
So, I discarded that for a moment, and after I left that company as their technical director, I started to think about this particular problem again. I thought, "What related fields can I get information from?" because as an engineer, I can never be too proud to beg. I started investigating unusual bits of science, came across fractals, and immediately recognized the fractals exemplified the patterns we had been creating in their simplest form. The thing I was looking at, at the time, was a Cantor set, which is one of the very early fractal forms.

I researched that, talked to Greg about it, and he thought about it for a while. Then, we realized, "Hey, there's a reason to pursue this." We initially pursued it from the point of view of something very simple that we could initially find a reason for: If we could etch this around the outside of a PCB, perhaps we could

Solutions to Ensure PCB Manufacturing Success!

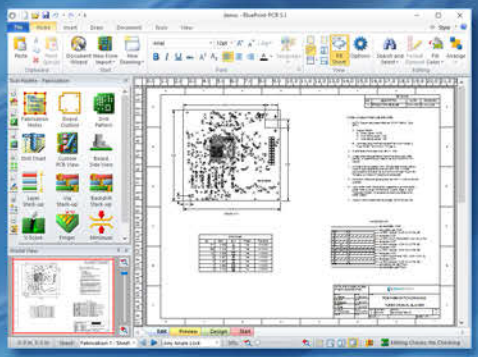
CAM350

Verify and Optimize PCB Designs for Successful Manufacturing.



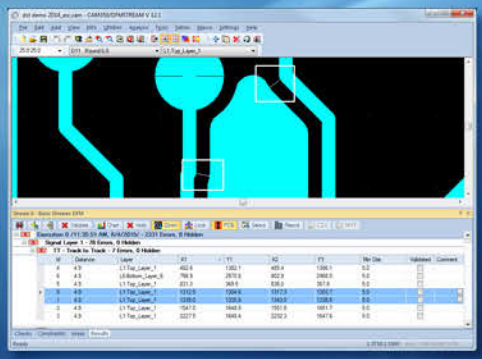
BluePrint·PCB

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 or stop by booth #1716 APEX EXPO IPC 2019

DownStream Technologies

reduce the noise given off by the edges of the board. Edge noise was a significant issue with buried capacitance. We started working with it and developing some IP to give us a history in the field, and also to serve as a vehicle for further study.

Next, we created some samples. We went through testing in Silicon Valley at Dr. Earl McCune's laboratory. We did a lot of analysis with Dr. McCune. From that, we derived and sent the samples that gave us the very most interesting results to an FCC testing laboratory with a set of engineers at that end to help us interpret the responses. The responses were as we expected in terms of reducing the noise from the PCB for virtually no cost or no cost that we could think of.

The responses were as we expected in terms of reducing the noise from the PCB for virtually no cost or no cost that we could think of.

We reduced the edge noise and avoided potential fixes like edge plating, etc., that would have to be done on a board. We also discovered something else that was a little harder to explain, and we didn't notice it in our first analysis. However, a group of engineers from a company—and we can't use the name because we never made it public that we talked with these folks—said, "This is absolutely astonishing."

What happens to the point at which this becomes resonant is it simply moves, not a little, but five, six, or seven GHz and up the range. One GHz doesn't sound like much to us when we're talking about 10 or 15 meters, but a GHz covered the entire FCC range that was talked about for all of the years since electronics has existed; it's an enormous range of change. In doing that, we finally came upon what we think is the right bit of information that everybody needs to know to determine whether this is something that might be of help to them in terms of developing their technologies.

This is potentially a method of, at virtually no cost, increasing the ability of the board to operate at higher frequencies, and, at the same time, reducing the edge noise that's radiated by the same board. I felt that there was an interest within the 5G community for a solution to the problem of really high-speed traffic over standard PCBs. The research and development groups of material laminators simply made a statement that above three to six GHz, the typical materials used by the PCB industry will not operate or give them any results that they can use.

Looking at that, I'd have to say that's a pretty accurate analysis. We've done our own analysis on that. When you add in special materials—the PTFE group and some of the hybrids and other special materials created by the industry for these purposes—you can increase the speeds of surface signals and do other things in that regard. However, you really can't do anything about the electrical energy necessary to operate the devices and act as a ground for those high-speed signals on the system.

So, when looking at the integration of this with a high-speed PCB, we arrived at three basic ideas. One, if you wish to obtain very high-speed operation on a PCB, you may wish to include optical systems within that PCB, at least to accomplish theoretical objectives, but at a very large cost. Or you can use special materials to create all of the desired Dk and Df objectives at a significant cost. However, fractals give a third option. You can not only use regular PCB materials to create higher-speed boards, but you can also add it to the special materials available to create an even higher-speed board that may allow the production of the kinds of PCBs that they would need to fully carry out their mission in terms of high-speed latency networks like 5G.

Now, this is not to paint a panacea. We're not in the position of saying all the research has been done and that we know precisely

what's going to evolve once all of the development, research, and testing are looked at. This appeared to us—as people who have been in the industry for a while—to be something that could be a significant advantage to a particular manufacturer or a group of manufacturers. I'm speaking of OEMs predominantly, but I'm not sure exactly who would find their way into this marketplace. I think it would present a very large advantage to anyone when compared to something that was simple and not very involved like varied capacitance, which ended up being a very profitable thing for a company that was involved with it. So, that's the thumbnail sketch in terms of the development.

Andy Shaughnessy: That's great. Reducing edge noise is a big challenge, and it's always one of the topics at DesignCon.

Greg Lucas: Yes, of the people I've talked to, that's their main interest as well.

Howard: It's an interesting area because every one of the boards we're talking about has a certain distance between the planes. Each one of those distances dictates a set of frequencies that will be emanated in a resonant fashion from the edge of those planes, and it's just a law of physics. Nobody is going to avoid it.

Lucas: This also increases voltage switching speed, so maybe it's that for some people.

Shaughnessy: Jim, you worked with the ZBC process, and fractals seem to be in a similar ballpark.

Howard: We believe that's one of the bases of this technology. I didn't talk about it, but all of the testing that we did found these unusual and very useful results when done on thin distributed-capacitance layers of the types that we developed in the 1990s.

Shaughnessy: The accompanying article really explains what fractals are. I didn't realize that they're basically shapes selected by nature.

Howard: You contain quite a number of fractals yourself. Your blood vessels, lungs, and brain neurons are all part of the fractal universe, and I'm quite amazed by what nature did with it before anybody caught on. It's a fascinating thing when you look at the shape of space. We don't get into these discussions with normal printed circuit folks, but when you look at the scientifically accepted dimensions of space and electromagnetic forces and the way that Maxwell explains those, it's fascinating. People with far better credentials than I do in this area have done enough work to determine that without those additional dimensions—such as gravity and electromagnetism—the weak and strong forces couldn't work the way we see it work.

And from that, even though it's not changing the basic structure of the PCB, there probably would have to be some work done to ensure that the reliability was within the structure of whatever they're certified to, and then they can take it into a market usage. Big companies could skip some of those steps, but they don't. They don't for very good reasons, and I understand the reasoning behind it. That was sort of the process that Unisys took varied capacitance through before we started shipping it to them for use in real products.

Lucas: Yes. The real cost of doing this is simply going back to CAD, adding the fractals to the power and ground plane, and testing it; that's the cost they're looking at.

Howard: If you were to compute what we know to be the cost today, forgetting about any testing that has to be done as pre-production testing, yeah, that's basically it.

Lucas: It's almost costless.

Howard: Right. If you know you have the appropriate design that you can do this with and you add these specific elements to the designs on the power and ground layers, you get this effect; you don't have to do anything else. Now, I'm not saying the etching is going to be easy because when we get down to very small kinds of etching, that's when we have

to call in Greg and ask him the appropriate methods for attacking those problems, but in essence, I think that's pretty right. Your suggestion of finding an OEM and letting them subcontract the making and designing of the boards—they may design them in-house—and the testing and so forth, it is probably the right place to go.

Matties: What is it that you want to do to carry this out to the marketplace?

Howard: I think we'll probably have some responses to this—not necessarily from the person who will end up to be the end-user, but from someone who's heard from someone who wants to know how the hell they're going to get to 20- or 25-GHz PCBs. That's the question out there, and I don't think anyone's solved it yet.

Shaughnessy: All right, guys. Thank you. It sounds like it could be a real game-changer.

Howard: Very good. Thank you so much.

Lucas: Thanks for asking us. DESIGN007

Fractals: New Tech and IP for Designing 5G PCBs

Article by Greg Lucas and Jim Howard
CIRCUIT SOLUTIONS

Fractal shapes have been shown to speed up and improve the performance of raw PCBs. To understand whether this will help in your next design, you must follow some basic fractal design ideas as they apply to PCB structures.

Why Use Fractals?

Fractal shapes are those selected by nature because they have infinitely long perimeters for the dissipation of heat, as well as unique electrical properties that can be derived from Maxwell's equations for the best operation of electrical devices.

In the last few decades, much has been

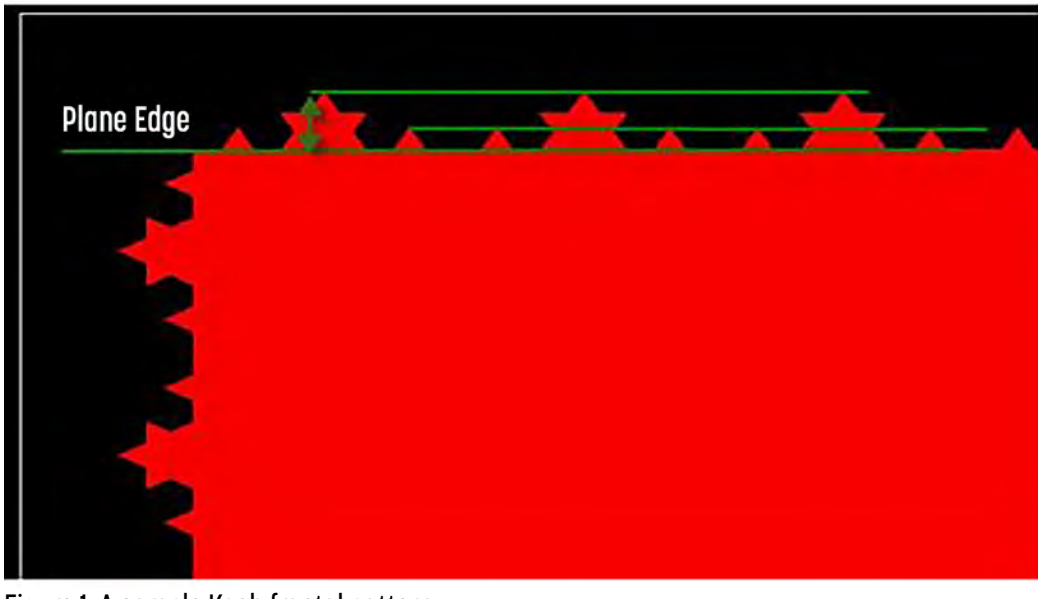


Figure 1: A sample Koch fractal pattern.

WHAT DO ALL OF THESE LEADING COMPANIES HAVE IN COMMON?

ATOTECH

CIMS
Creativity in Motion

Cerambus
Technology Group

COMETGROUP
Technology with Passion

DownStream
Technologies

esi
Designed for Brilliance. Engineered for Production.

KIC Making Ovens Smarter™

KYZEN

MacDermid Enthone
ELECTRONICS SOLUTIONS

Mentor
A Siemens Business

PULSONIX

orbotech
The Language of Electronics

ventec
INTERNATIONAL GROUP
騰輝電子

環球集團
World Wide Group

They all benefit by advertising with us–PCB007 China Magazine.

Should your name be on the leaders' list?

GET STARTED NOW!

I-Connect007
GOOD FOR THE INDUSTRY

pcb007china.com

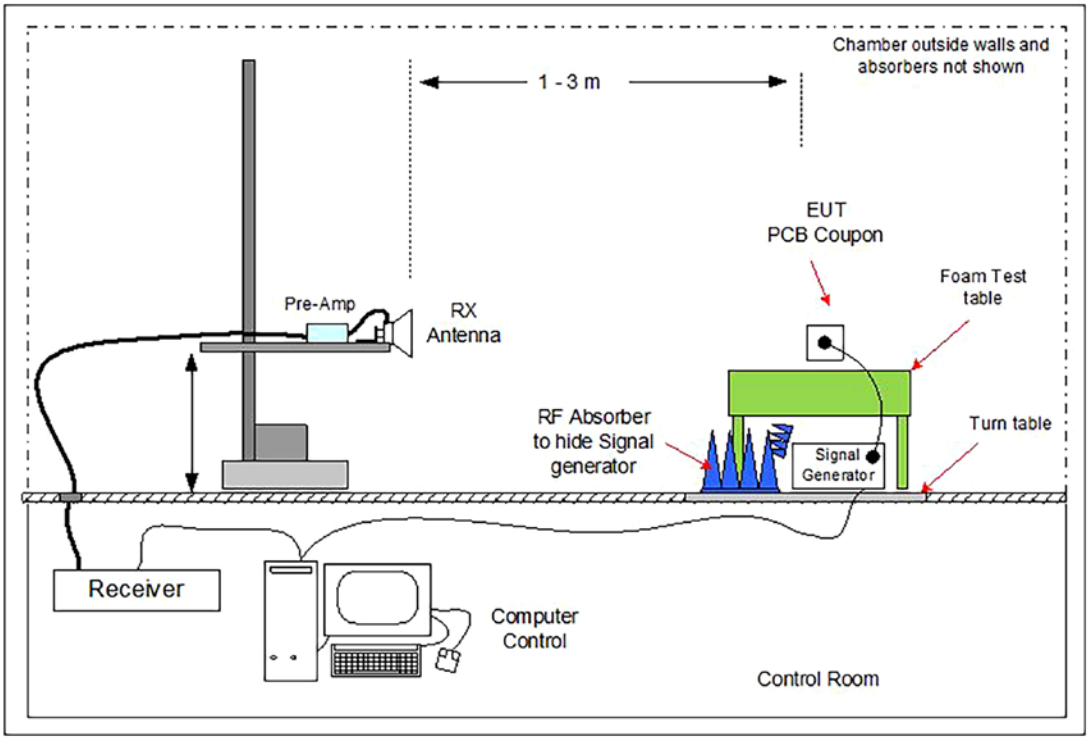
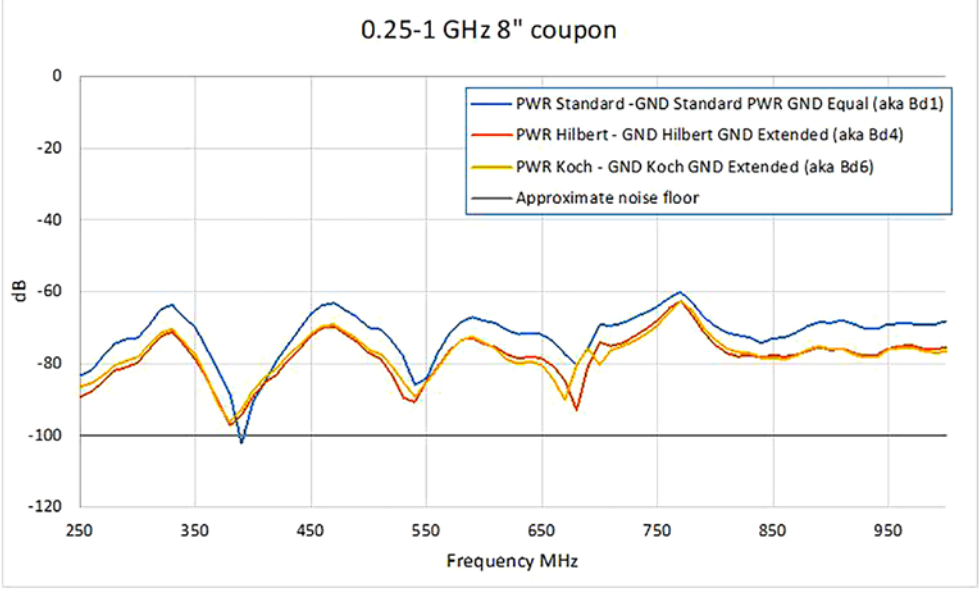


Figure 2: The test apparatus at a licensed FCC test and analysis laboratory.



Typical reduction in edge radiation: Blue *normal edge*, yellow Koch Fractal Edge, Red/Brown *Hilbert Fractal Edge*

Figure 3: The test result for the FCC test range of frequencies, including a standard layer, Hilbert fractal space-filling curve, and Koch fractal.

done through science regarding this very new field of fractal technology, including the fractalization of most antennas used in cellphones and other similar applications. They are much smaller and more efficient than the older style of antennas, which were crude and simple applications of what we learned from the earliest electronic communications devices.

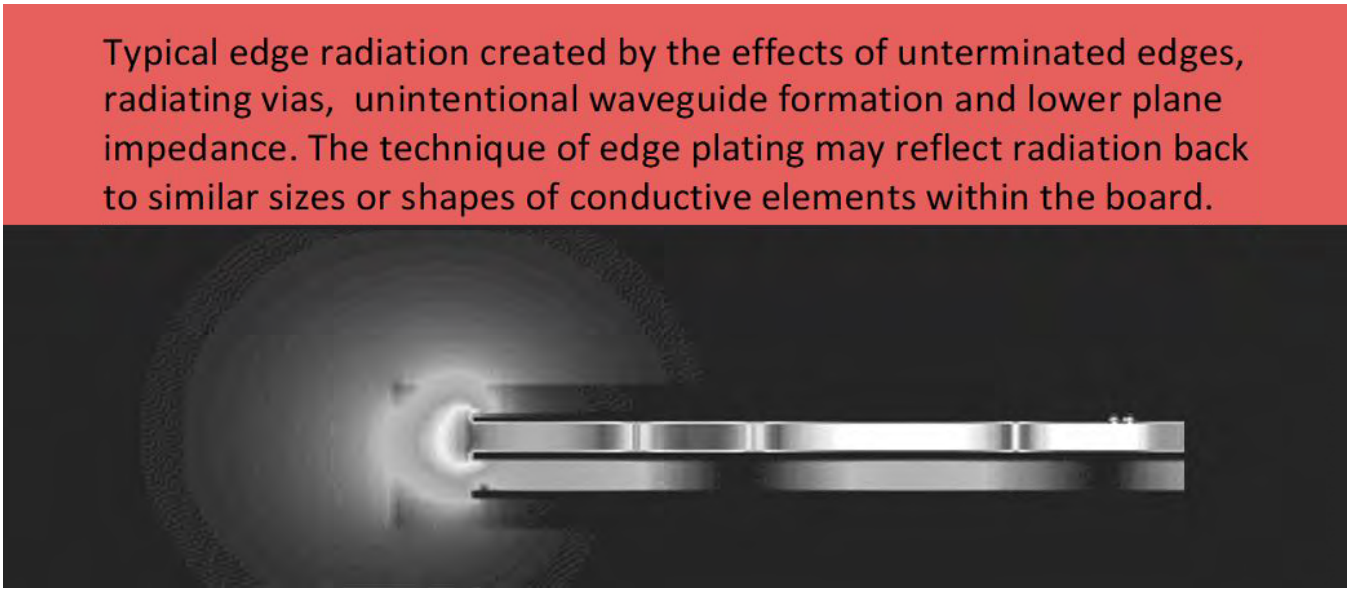


Figure 4: Description of the generating surfaces contributing to PCB edge noise.

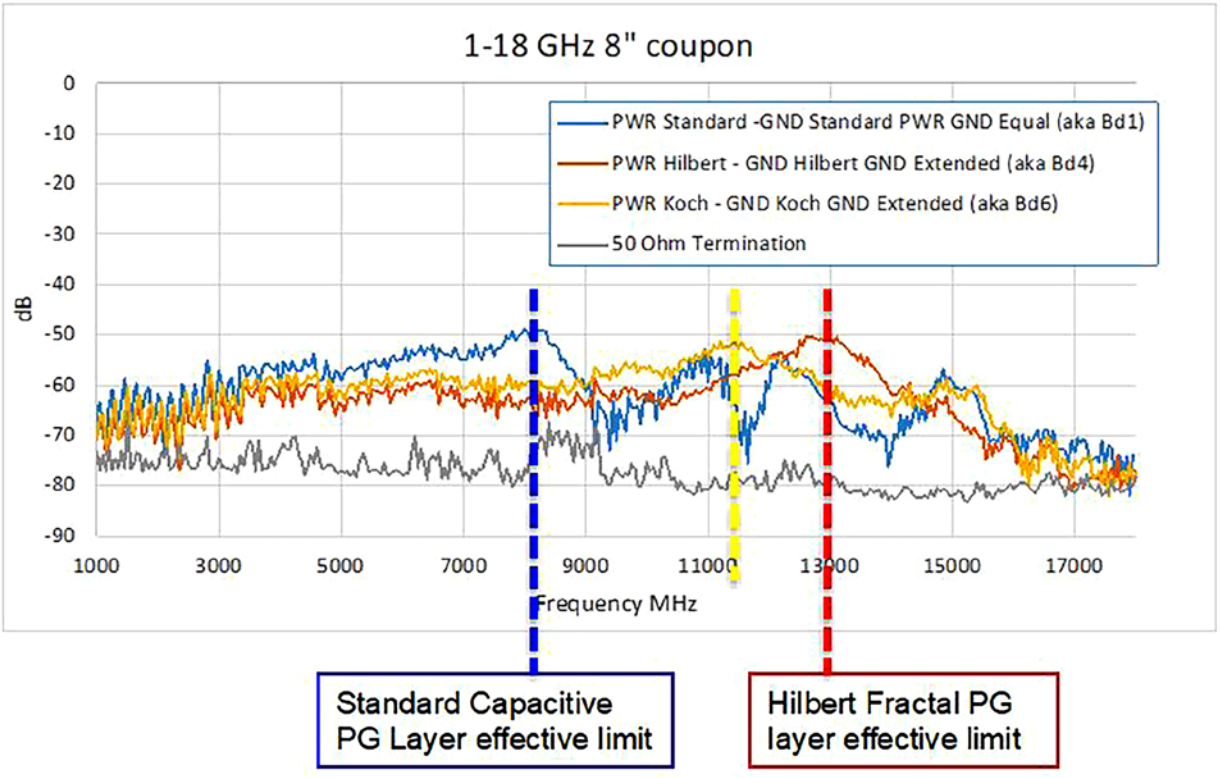


Figure 5: Changing small, unused features on a conductive layer can alter the PCB's electrical response.

Starting from the basic parts of PCBs, testing has been done on power, ground, and circuit layers to determine if including fractal design principals in their design changes how they perform electrically. The results were quite dramatic. Teams of engineers pointed out changes that point to higher-frequency operations with less radiated noise with the same materials, indicating that no matter what materials are chosen for construction, the design will be improved by incorporating fractal design techniques.

There is a general reduction in the amount of radiated noise when proper rules are used

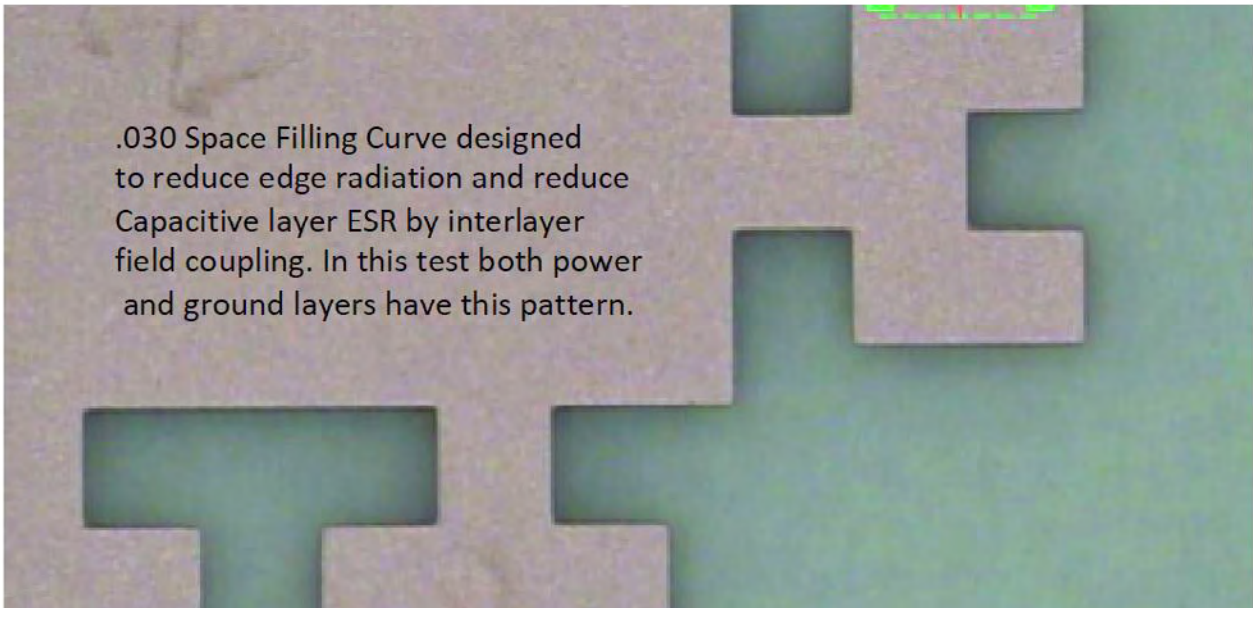


Figure 6: An experimental 0.030" proprietary Hilbert fractal edge pattern.

in the design of the electrical elements. These rules are not obvious from the basic information given about fractal shapes in the available data; however, it becomes evident in the analysis of test data.

This capability indicates that this reduction is useful in many existing products to eliminate the need for special requirements or fixes to reduce radiated emissions. Only a simple, no-cost design change is needed to implement this improvement, which will eliminate processing such as edge plating.

Why does the application of fractal technology design rules have such an effect on current printed circuit designs? It's very much like understanding why airplanes are better designed with modern aerodynamics than with the original biplane concepts.

PCBs are the biplane of the electronics industry, having changed little since the 1950s in their interactive design. These experimental designs demonstrate the ability to improve high-speed operations using the normal rules of electrical engineering with new physics.

Eventually, PCBs will have to conform to interactive scientific design rules to use less expensive and more easily manufactured designs or convert to optical technology, which presents a plethora of nightmarish problems.

Here is a quick review of some of the testing done thus far, using a test board constructed with a small 0.030" space-filling curve around the perimeter. Adding this type of feature will have virtually no impact on the size or cost of the raw board while changing its electrical response significantly.

Figure 5 demonstrates that the electrical response of a PCB component can be altered by simply changing small features on a conductive layer that are otherwise unused. This capability may allow the coupon above to operate at a higher frequency with lower radiated noise at targeted frequencies. The cost difference in making this change is zero. The value of this technical performance can be decisive.

Fractal shapes by their very nature will present a form that has more dissipation of heat than other shapes. In addition, electrical fundamentals—such as impedance, crosstalk, and other features—can be controlled with the symmetry of fractal design.

What might these design features be useful for? Currently, 5G technologies featuring 20 gigabit/s at speeds of 15 GHz are not achievable with standard PCB technologies and materials. If your plan is to change materials or morph to a different technology like optical circuit boards, the cost difference is astronomical.

The Next Step

From past experience, we believe that the most successful form of fractal development would a single company that can take the existing IP from our engineering team, fully develop the IP, do design to product testing, implement it through products in its current supply line, and then relicense the developed IP to its competitors and customers. A single entity should own the rights and license or use them as they see fit.

Obviously, we cannot carry the discussion further without an NDA in place to enable the sharing of patent protected information. But if you're interested in increasing the ability of a PCB to operate at a higher frequency while reducing edge noise without adding any cost, fractals may be just the technique you need. **DESIGN007**

Greg Lucas began his PCB career at Cinch Graphic in 1963 while earning his bachelor's degree in chemistry. He went on to run PCB engineering at Storage Tech, Zycon, and Pragetzer. He is now retired, but still involved in exploring new ideas such as the use of fractals in PCB design.

James Howard is a principal research engineer who has served Silicon Valley and foreign firms as a technology director and innovator for 40 years. He holds a BSEE and eight basic patents relevant to high-speed PCBs utilizing distributed capacitance fields and fractal mathematic design features.

Foldable Smartphones to Be Launched in 2019

According to The Future of Smartphone Era Webinar by WitsView, a division of TrendForce, the global smartphone market has been approaching saturation, with less room for product differentiation. Thus, smartphone makers have shifted their focus to next-generation foldable models. WitsView expects the first foldable smartphone to be launched in 2019, accounting for only 0.1% in the global smartphone market. The penetration rate is estimated to reach 1.5% in 2021.

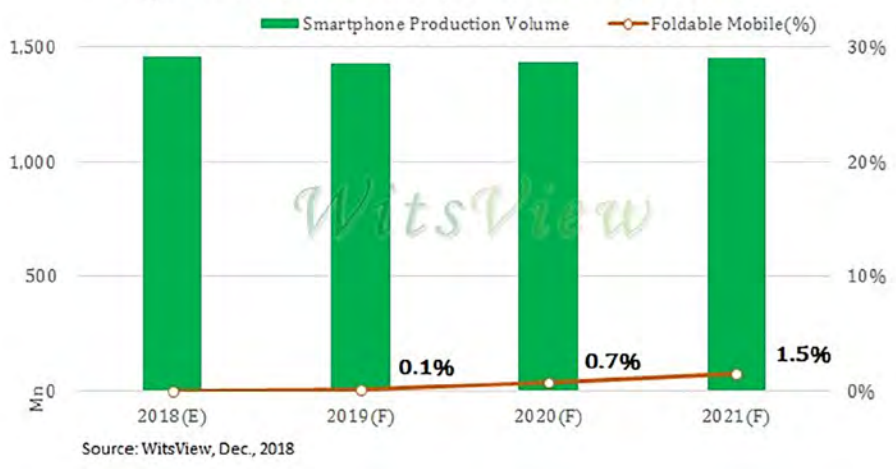
According to Boyce Fan, the research director of WitsView, a few manufacturers tried to develop foldable models in the past using a dual-screen design, but most of them failed. However, as breakthroughs continue to be made during the development of flexible AMOLED panels, single-screen foldable smartphones gradually emerge. Among the makers, Samsung takes a leading role with its advanced display technology.

Although foldable phones have become the focus of the market, they still remain in the early stage of product development, so manufacturers are bound to improve the designs continuously, says Fan. Meanwhile, there is also room for improvement in terms of the thickness of the phone and the border, as well as the battery life.

WitsView believes that the initial demand for foldable smartphones will not be very strong, considering the need for product optimization, lack of flexible AMOLED panel suppliers other than Samsung, and the looming demand in the market. The segment of foldable smartphones will not have a chance to expand until the second half of 2019 or after 2020, when more manufacturers follow suit to launch foldable models and get more feedback from the market.

(Source: TrendForce)

Figure: Penetration of Foldable Phones in Global Smartphone Market



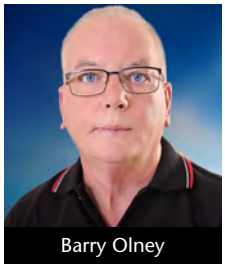
Source: WitsView, Dec., 2018

Editor Picks from PCBDesign007

1 PCB Carolina 2018 Draws a Crowd of Technologists ▶

PCB Carolina 2018, the one-day tabletop show based in Raleigh, North Carolina, drew quite a crowd to the McKimmon Conference and Training Center at NC State University on November 7.

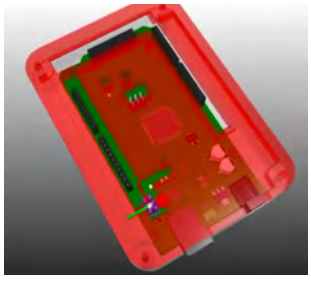
2 10 Fundamental Rules of High-speed PCB Design, Pt. 2 ▶



Barry Olney

In last month's column, I introduced the 10 fundamental rules of high-speed PCB design. This month, I will elaborate on the importance of controlling the impedance and floor planning the placement based on connectivity.

3 Pulsonix Expands into North America with Version 10 Release ▶



At PCB West in Santa Clara, California, Guest Editor Tim Haag and Publisher Barry Matties sat down with Ty Stephens of Pulsonix on the eve of the software developer's much-anticipated version 10 release date.

4 Max Seeley's PCB Design Career is No Fish Story ▶

Max Seeley

Max Seeley, senior electrical, PCB, and manufacturing engineer with 3M, was an instructor at the first AltiumLive event in 2017, and he made a return visit to this year's event in San Diego.

5 The Bare (Board) Truth: Getting on the Same Page—A Data Story ▶


Mark Thompson

Thickness callouts for single-sided or double-sided orders are even more critical. As a fabricator, we can control the thickness of the multilayer by using different combinations of prepregs/cores.

6 EMA Launches Ultra Librarian Reference Design Cloud ▶

EMA Design Automation announced the release of its Ultra Librarian Reference Design Cloud, giving engineers easy, web-based access to reference design schematics enabling faster design starts.

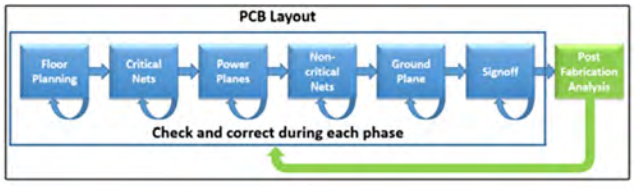
7 Julie Ellis: Communication and Fabrication Knowledge Critical for Designers ▶


Julie Ellis

Field Application Engineer Julie Ellis of TTM sees it all: good designs, bad designs, and everything in between. Her classes on proper DFM techniques are always a big draw. She taught at the inaugural AltiumLive in 2017 and was back at this year's event. I caught up with Julie and asked her to discuss some of the things she covered in class. As she points out, many issues could be eliminated if designers communicated with their fabricators and had a better understanding of how PCBs are manufactured.

8 New Mentor Xpedition Platform Delivers 'Shift-Left' Multi-Dimensional Verification ▶

Mentor, a Siemens business has announced the new Xpedition printed circuit board (PCB) design platform focused on multi-dimensional verification.



9 Life Beyond 10 Gbps: Localize or Fail! ▶


Yuriy Shlepnev

Ideally, all interconnects should look like uniform transmission lines (or wave-guiding structures) with the specified characteristic impedance. In reality, an interconnect link is typically composed of transmission lines of different types (microstrip, strip, coplanar, coaxial, etc.) and transitions between them such as vias, connectors, breakouts and so on.

10 DownStream Technologies Unveils Updates of CAM350, DFMStream and BluePrint-PCB ▶

DownStream Technologies has announced global availability of Release 2018 for CAM350, BluePrint-PCB, and DFMStream.

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

Career Opportunities

Sales Partner / Agent for North America

TSK Schill GmbH is a specialized partner in horizontal wet process equipment for the printed circuit board and chemical milling industries.

We are looking for a sales representative in North America who will support us in the sales of our HMS Höllmüller by TSK wet-process machines in the market.

Job responsibilities:

- Acquire customers in the field of chemical wet process
- Ascertain customer needs
- Support customers in the PCB/PCM business
- Spare parts support for horizontal wet process machines

Candidate profile:

- Company or individual contributor
- Experience in the field of chemical wet process
- Communication and organization skills
- Sales attitude

apply now

Field Service Engineer West Coast

Pluritec North America, ltd., An innovative leader in drilling, routing and Automated Inspection in the Printed Circuit Board industry, is seeking a full-time Field Service Engineer, located on the West Coast.

This individual will support service for North America in Equipment installation, training, maintenance and repair. Candidate must be able to handle trouble shooting electronic and mechanical issue's as well customer applications in the field. A technical degree is preferred, along with strong verbal and written communication skills. The position requires the ability to travel 2-3 weeks per month.

Please send your resume to:
Carolina.zeppieri@pluritec.org

apply now

Career Opportunities

Director of Operations, New Hampshire

The role is based at the Gardien facility in Nashua, New Hampshire. It is a managerial role, requiring a sense of ownership of the designated territory and directly reporting to the vice president, North America.

Major Responsibilities:

- Represent Gardien and manage relationships with customers with particular regard to fulfilling customer needs
- Ensure customer requirements are fulfilled and optimum service levels maintained
- Ensure that the quality management system is fully implemented
- Takes ownership of all processes in the specification location
- Responsible for all P&L tasks within that location
- Define operational resources and capabilities within that location
- Liaison with customers and technical staff to resolve in-process problems
- Responsible for effective and successful management of staff and personnel matters (e.g., employee training and awareness is continuously maintained and updated)

Interested? Please contact us at career.na@gardien.com with your resume.

apply now

Sales Personnel, Japan

The Gardien Group is looking to expand the sales team in Tokyo, Japan, and seeking highly motivated team players with a positive attitude. Prior experience in the PCB industry is an advantage but not necessary for the right candidate.

The role involves working closely with the customer to identify their needs and deliver the right solution. The candidate should be able to offer a high level of customer satisfaction to ensure ongoing sales.

Training will be provided along with a competitive benefits package, excellent growth opportunities, and periodic bonuses.

Interested candidates, please contact us at info.jp@gardien.com with your resume.

Kindly note only shortlisted candidates will be notified.

apply now

Career Opportunities

U.S. CIRCUIT

Sales Representatives (Specific Territories)

Escondido-based printed circuit fabricator U.S. Circuit is looking to hire sales representatives in the following territories:

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

Prototron Circuits

New England Territory PCB Sales Position

Prototron Circuits, a leading manufacturer of quick-turn PCBs for over 31 years, is seeking representation in our New England territory. Prototron Circuits is ITAR registered, ISO 9001:2015 and AS9100D certified, and MIL-PRF-31032 qualified with two facilities located in Redmond, Washington, and Tucson, Arizona.

Candidates should have:

- Experience selling PCBs or industry related products
- The ability to work from a home office and travel to and from customer facilities
- A proven track record prospecting and building sales in an existing territory

Prototron Circuits has maintained a 98% on-time rating every year and is well known for both outstanding quality and reliability. If you desire spending the day selling and not apologizing, this is the opportunity you've been looking for.

Direct position includes salary, incentive plan, full benefits package, and company covered expenses.

A rep agreement will be on a commission basis for a principal with a proven track record of reliability in an established territory.

To be considered for this opportunity, please send your resume to Russ Adams at russa@prototron.com.

apply now

Career Opportunities

Manncorp™

SMT Surface Mount Technician / SMT Operator

Huntingdon Valley, Pennsylvania

Manncorp, a leader in the electronics assembly industry, is looking for an additional SMT field technician to install and support our wide array of SMT equipment.

Duties and Responsibilities:

- Manage on-site installation of equipment and training of customers
- Must be capable of running SMT production lines on-site
- Post-installation service and support includes trouble-shooting and diagnosing technical problems by phone, email, or additional on-site visits when necessary
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Produce service reports
- Cooperate with the technical team and share information across the organization
- Assist with the crating and uncrating of equipment

Requirements and Qualifications:

- Three to five years of experience with SMT equipment or an equivalent technical degree
- Proven strong mechanical and electrical trouble-shooting skills
- Proficient in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Organized, detail orientated, and capable of multitasking
- Have strong written and oral interpersonal skills with an ability to work under minimum supervision
- Ability to work with little supervision while traveling
- Frequent travel
- Ability to arrange and schedule service trips
- USA citizenship required

We Offer:

- Health and dental insurance
- Retirement fund matching

apply now

MivaTek Global

Multiple Positions Available

Want to work for a fast-growing company? MivaTek Global may be the place for your next career move. 2018 has brought significant growth, increasing sales and revenue. And, we are just getting started! To support the current customer base and fuel further expansion, we are looking for bright and talented people who are energized by hard work in a supportive and flexible environment.

Open Positions:

- Technical Service Technicians
- Regional Sales Representatives
- Regional Leader for Asia Sales and Support

Proven experience in either PCB or Microelectronics and willingness to travel required for all positions.

More About Us

MivaTek Global is a distributor of manufacturing equipment with an emphasis of Miva Technologies' Direct Imager, Mask Writer, Flatbed Photoplotter imaging systems and Mach3 Labs X-Ray Drills. We currently have 45 installations in the Americas. Expansion into Asia during 2018 has led to machine installations in China, Singapore, Korea, and India.

To be part of our team, send your resume to n.hogan@kupertek.com for consideration of current and future opportunities.

apply now

Career Opportunities

FREEDOM CAD SERVICES

Sr. PCB Designer – Allegro

Freedom CAD is a premier PCB design service bureau with a talented team of 30+ dedicated designers providing complex layouts for our enviable list of high-tech customers. Tired of the commute? This is a work-from-home, full-time position with an opportunity for overtime at time and a half.

Key Qualifications

- EXPERT knowledge of Allegro 16.6/17.2
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Experience using SKILL script automation such as dalTools
- Excellent team player that can lead projects and mentor others
- Self-motivated, with ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem solving skills
- Other design tool knowledge is considered a plus (Altium, PADS, Xpedition)

Primary Responsibilities

- Design project leader
- Lead highly complex layouts while ensuring quality, efficiency and manufacturability
- Handle multiple tasks and provide work leadership to other designers through the distribution, coordination, and management of the assigned work load
- Ability to create from engineering inputs: board mechanical profiles, board fabrication stack-ups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

MacDermid Enthone ELECTRONICS SOLUTIONS

Product Group Field Manager Waterbury, CT

The Product Group Field Manager is responsible for creating and driving the regional product line strategic plan in coordination with the global product line managers, strategic account manager and regional business managers. The successful candidate must balance commercial obligations to assist the sales teams in closing new business, perpetuating technical expertise throughout the field and develop best practices for the region.

Education: Bachelor's degree; 5 years of related experience; or equivalent combination of both.

Responsibilities

- Thorough understanding of the PCB business; specifics in wet processing areas.
- Facilitate developing commercial and technical strategy for customers.
- Create and deliver customer facing presentations.
- Training.
- Create and execute a product rationalization program aligning with global product managers.
- Develop roll-out packages for new product introductions, including operating guides.
- Excellent written and oral communication skills.
- Expert in chemistry and chemical interaction within PCB manufacturing.
- Willingness to travel globally.

MacDermid Enthone is an E-Verify Company and provides reasonable accommodation for qualified individuals with disabilities and disabled veterans in job applicant procedures. "Equal Opportunity Employer: Minority/Female/Veteran/Disabled/Gender Identity/Sexual Orientation."

apply now

Career Opportunities

MacDermid Enthone
ELECTRONICS SOLUTIONS

Global Application Specialist
Waterbury, CT

Qualifications: Bachelor's in Chemistry, and seven years progressive experience in related field. Expertise preferably in ENIG and ENEPIG. Global travel required: up to 40%.

Responsibilities

- Chemical analysis and experiments of final finish chemistries; characterize new processes from research prior to beta site installations, establishing operating parameters, problem solving tools and analytical guidelines.
- Recommend product, process, and analytical method improvements; including changing composition of compounds.
- Develop final finish product line. Install products at beta sites; collect data.
- Lead technical teams during beta site installations of new products and problem-solving groups at customer locations.
- Train personnel.
- Set up tests of final finish chemistries and products for laboratory personnel to identify customer problems, analyze result to resolve customer issues, and communicate results to customers.
- Oversee laboratory analysis and processing of customer samples through our global technical centers; summarize data, make recommendations and write reports.
- Document technical bulletins.

MacDermid Enthone is an E-Verify Company and provides reasonable accommodation for qualified individuals with disabilities and disabled veterans in job applicant procedures. "Equal Opportunity Employer: Minority/Female/Veteran/Disabled/Gender Identity/Sexual Orientation."

apply now

MacDermid Enthone
ELECTRONICS SOLUTIONS

Director of Final Finishes
Waterbury, CT

Education: Advanced practical knowledge–formal education and experience in chemistry or related sciences. Knows all technology within the business area and has knowledge of end use processes and OEMs.

Responsibilities

- Collects and analyzes market information, understands the competitive landscape, identifies potential gaps in product portfolio and effectively communicates needs to the product development group.
- Oversees product development activities, and reviews projects as they reach PDP milestones.
- Responsible for customer presentations and participation in trade organizations and other industry activities.
- Constructs release package information for the introduction of new products and sets pricing guidance for the commercial teams.
- Responsible for customer presentations and participation in trade organizations and other industry activities. High-level customer interaction required.
- Has successfully demonstrated the ability to manage professionals and nonprofessionals in a technical and marketing environment.
- Develops and responsible for budgets and goals of the group.

MacDermid Enthone is an E-Verify Company and provides reasonable accommodation for qualified individuals with disabilities and disabled veterans in job applicant procedures. "Equal Opportunity Employer: Minority/Female/Veteran/Disabled/Gender Identity/Sexual Orientation."

apply now

Career Opportunities

Careers with Gardien

The Gardien Group, a leading solutions provider in the PCB industry, is looking to fill multiple openings in their China, Japan, Taiwan and the United States service centres.

We are looking for electrical engineers, operations managers, machine operators and sales executives. Prior experience in the PCB industry is beneficial but not essential. Training will be provided along with excellent growth opportunities, benefits package and periodic bonuses.

Our global teams are from diverse cultures and work cohesively as a tight-knit unit. With performance and initiative, there are plenty opportunities for professional growth.

Interested candidates please contact us at careers@gardien.com with your resume and a cover letter. Kindly note that only shortlisted candidate will be contacted.

About Gardien Group

Gardien is the world's largest international provider of independent testing and QA solutions to the PCB industry with a global footprint across 24 service centres in 5 countries and we cater to a whole range of customers, from small, family-owned PCB shops to large international fabricators, and everything in-between. Gardien's quality solutions and process standards are trusted by leading high-tech manufacturers and important industries including aerospace, defense and medical technology.

apply now

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

CAM Operator

American Standard Circuits is seeking a CAM Operator for its Phoenix, Ariz., office. Qualified applicants will need experience in using Valor/Genesis (GenFlex) CAD/CAM software with printed circuit board process knowledge to edit electronic data in support of customer and production needs.

Job Requirements:

- At least 5 years' experience in PCB manufacturing
- Process DRC / DFMs and distinguish valid design and manufacturing concerns.
- Modify customer supplied data files and interface with customers and engineers
- Responsible for releasing manufacturing tooling to the production floor
- Prepare NC tooling for machine drilling, routing, imaging, soldermask, silkscreen
- Netlist test, optical inspection
- Work with Production on needed changes
- Suggestions on continual improvements for engineering and processing.
- Be able to read write and communicate in English
- Must understand prints specifications
- Must be US Citizen or permanent resident (ITAR)
- High School Graduate or equivalent

Join our Team!

Founded in 1988, American Standard Circuits is a leading manufacturer of advanced circuit board solutions worldwide. Our ongoing commitment to leading-edge higher-level interconnect technology, cost-effective manufacturing and unparalleled customer service has put us at the forefront of advanced technology circuit board fabrication.

We manufacture quality rigid, metal-backed and flex printed circuit boards on various types of substrates for many applications.

apply now

Career Opportunities

ELECTROLUBE
THE SOLUTIONS PEOPLE

We Are Recruiting!

A fantastic opportunity has arisen within Electrolube, a progressive global electro-chemicals manufacturer. This prestigious new role is for a sales development manager with a strong technical sales background (electro-chemicals industry desirable) and great commercial awareness. The key focus of this role is to increase profitable sales of the Electrolube brand within the Midwest area of the United States; this is to be achieved via a strategic program of major account development and progression of new accounts/projects. Monitoring of competitor activity and recognition of new opportunities are also integral to this challenging role. Full product training to be provided.

The successful candidate will benefit from a generous package and report directly to the U.S. general manager.

Applicants should apply with their CV to
melanie.latham@hkw.co.uk
(agencies welcome)

apply now

Ucamco
Former Barco ETS

International Field Service Engineer located in ITALY

The successful candidate will:

- Install and service our plotters and direct imaging machines at customer sites Europe-wide
- Carry out maintenance in the field
- Frequent travel: 4 to 5 days a week, 3 to 4 weeks a month
- Assist product manager

We are looking for a team player who is:

- Strongly customer-oriented and experienced in on-site support
- Accustomed to travel, and willing to travel frequently
- Motivated, independent and enterprising
- Technically-minded with training/background in electromechanics/electronics
- Experienced with software (setup, configuration, and usage of Windows-based CAM front-end software and Linux-based RIP software)
- Fluent in Italian and English (German and/or French is a plus)
- An analytical thinker
- Capable of problem solving

The right candidate will be a valued member of a friendly, team-oriented, growing international company that is a leader in its field, dedicated to excellence in all it does. Dynamic and fun, the company offers a great working atmosphere, and this new position is forward-looking and open, with plenty of opportunities for enterprising individuals whose results could be rewarded with prospects for progression in technical development.

Apply to Anja Ingels after clicking below.

apply now

Career Opportunities

Role: Vice President Gardien Taiwan
TAOYUAN COUNTY, TAIWAN

Gardien Taiwan is a service provider of circuit board (PCB) quality solutions, including electrical testing, AOI optical inspection, engineering (CAM), fixture making, repair and rework. Gardien Taiwan operates service centers in Taoyuan and employs about 100 employees and is currently seeking a vice president to manage and oversee the entity.

Candidate Profile:

- Proficiency in Chinese and English (written and spoken)
- Excellent communication and organization skills
- Experience in change management
- PCB background appreciated, but not mandatory
- Management experience in internationally operating companies
- Savvy in standard office software (Word, Excel and Power Point)

If this sounds like you, please click here to send us an email with your attached CV.

About Gardien Group - Gardien is the world's largest international provider of independent testing and QA solutions to the PCB industry with a global footprint across 24 service centres in five countries and we cater to a whole range of customers, from small family owned PCB shops to large international fabricators. Gardien's quality solutions and process standards are trusted by leading high-tech manufacturers and important industries including aerospace, defense, and medical technology.

apply now

Zentech Manufacturing:
Hiring Multiple Positions

Are you looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add Manufacturing Engineers, Program Managers, and Sr. Test Technicians. Offering an excellent benefit package including health/dental insurance and an employer-matched 401k program, Zentech holds the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485.

Zentech is an IPC Trusted Source QML and ITAR registered. U.S. citizens only need apply.

Please email resume below.

apply now

Career Opportunities

Manncorp™

Sales Associate - Mexico

Manncorp, a leader in the electronics assembly industry for over 50 years, is looking for an additional sales associate to cover all of Mexico and to be part of a collaborative, tight-knit team. We offer on-the-job training and years of industry experience in order to set up our sales associate for success. This individual will be a key part of the sales cycle and be heavily involved with the customers and the sales manager.

Job responsibilities:

- Acquire new customers by reaching out to leads
- Ascertain customer's purchase needs
- Assist in resolving customer complaints and queries
- Meet deadlines and financial goal minimums
- Make recommendations to the customer
- Maintain documentation of customer communication, contact and account updates

Job requirements:

- Located in Mexico
- Knowledge of pick-and-place and electronics assembly in general
- 3+ years of sales experience
- Customer service skills
- Positive attitude
- Self-starter with ability to work with little supervision
- Phone, email, and chat communication skills
- Persuasion, negotiation, and closing skills

We offer:

- Competitive salary
- Generous commission structure

apply now

Mentor®
A Siemens Business

PCB Manufacturing, Marketing Engineer

Use your knowledge of PCB assembly and process engineering to promote Mentor's Valor digital manufacturing solutions via industry articles, industry events, blogs, and relevant social networking sites. The Valor division is seeking a seasoned professional who has operated within the PCB manufacturing industry to be a leading voice in advocating our solutions through a variety of marketing platforms including digital, media, trade show, conferences, and forums.

The successful candidate is expected to have solid experience within the PCB assembly industry and the ability to represent the Valor solutions with authority and credibility. A solid background in PCB Process Engineering or Quality management to leverage in day-to-day activities is preferred. The candidate should be a good "storyteller" who can develop relatable content in an interesting and compelling manner, and who is comfortable in presenting in public as well as engaging in on-line forums; should have solid experience with professional social platforms such as LinkedIn.

Success will be measured quantitatively in terms of number of interactions, increase in digital engagements, measurement of sentiment, article placements, presentations delivered. Qualitatively, success will be measured by feedback from colleagues and relevant industry players.

This is an excellent opportunity for an industry professional who has a passion for marketing and public presentation.

Location flexible: Israel, UK or US

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

YOUR JOB AD HERE

For information, please contact:
BARB HOCKADAY
barb@iconnect007.com
+1 916.365.1727 (PACFIC)

I·Connect007
GOOD FOR THE INDUSTRY

Events Calendar

48th NEPCON JAPAN ▶
January 16–18, 2019
Tokyo, Japan

IPC APEX EXPO 2019 Conference and Exhibition ▶
January 26–31, 2019
San Diego, California, U.S.

DesignCon 2019 ▶
January 29–31, 2019
Santa Clara, California, U.S.

SMTA Pan Pacific Microelectronics Symposium ▶
February 11–14, 2019
Kauai, Hawaii, U.S.

EIPC 2019 Winter Conference ▶
February 14–15, 2019
Milan, Italy

China International PCB & Assembly Show (CPCA Show 2019) ▶
March 19–21, 2019
Shanghai, China

MicroTech 2019 ▶
April 4, 2019
Cambridge, U.K.

Medical Electronics Symposium 2019 ▶
May 21–22, 2019
Elyria, Ohio, U.S.

PCB Pavilion @ LCD EXPO Thailand ▶
June 27–29, 2019
Bangkok, Thailand

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

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

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

ASSOCIATE EDITOR: **KIERSTEN ROHDE**
kiersten@iconnect007.com

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © FLAMINGO IMAGES**

DESIGN007 MAGAZINE

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

December 2018, Volume 7, Number 12
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering..... 23
Altium..... 37
American Standard Circuits..... 5
APCT..... 19
Blackfox Training Institute..... 71
Candor Industries..... 11
Downstream Technologies..... 15, 81
Eagle Electronics..... 33
Electrolube..... 77
EMA Design Automation..... 53
FLEX007 Magazine..... 57
General Circuits..... 63
I-Connect007 eBooks..... 2, 3
In-Circuit Design Pty Ltd..... 73
IPC..... 35
ITEQ..... 27
Mentor, a Siemens Business..... 21
Miraco..... 41
PCB007 China Magazine..... 85
Prototron Circuits..... 75
Pulsonix..... 59
Rogers Corporation..... 67
US Circuit..... 51
Ventec International Group..... 7

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