I-Connect007

MARCH 2019

DESIGN007 MAGAZINE

PCB Designers:

Here's who's filling your shoes...

DESIGN007 MAGAZINE

The Youth of the PCB Design Industry

There's something happening in this industry: Young people are once again entering the PCB design workforce. In this issue, you'll meet a variety of young people working in this industry. Is this the beginning of a trend? Let's hope so!

10 **George Fox University: Teaching PCB Design to EE Students**
Interview with Gary Spivey

18 **Putting University Training to Work**
Interview with Jake Whipple

24 **Preparing to Enter the Workforce With PCB Design Experience**
Interview with Alex Burt

28 **PCB Design is All in the Family**
Interview with Nicole Pacino

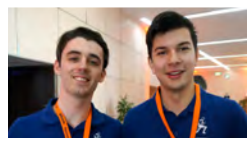
32 **The PCB Industry Must Work to Attract Young People**
Interview with Megan Teta

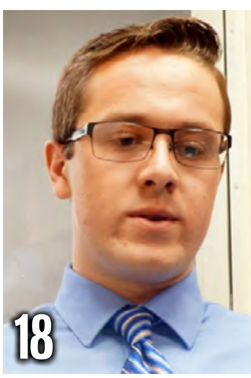
38 **An Experienced Millennial on Hiring the Next Generation**
Interview with Geoffrey Hazelett

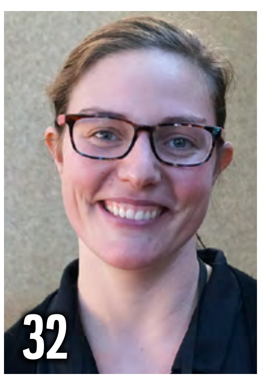
44 **EDA: A Great Field for Young People**
Interview with Andy Johnson

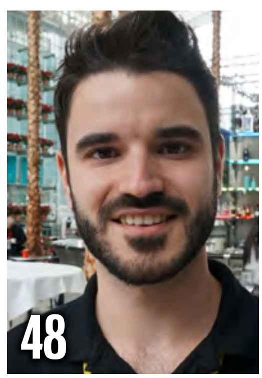
48 **From Spain to Austria: A Student's Drive to Be a Professor**
Interview with Pablo Sanchez Martinez

52 **Project MARCH Students' Exoskeleton Helps Paraplegics Walk Again**
Interview with Martijn van der Marel and Roy Arriens









DESIGN007 MAGAZINE

SHORTS:

31 Solar-powered Supercapacitors Could Create Flexible, Wearable Electronics

36 Materials Chemists Tap Body Heat to Power Smart Garments

42 Scientists Develop New Method to Revolutionize Graphene-printed Electronics

50 Researchers Make Breakthrough in Understanding Conductivity in Doped Organic Semiconductors

64 Tiny Silicon Nanoparticles Cement New Era for Ultra-high Capacity Batteries

69 Laser Pulses Light the Way to Tuning Topological Materials for Spintronics and Quantum Computing

83 Large, Stable Pieces of Graphene Produced With a Unique Edge Pattern

COLUMNS:

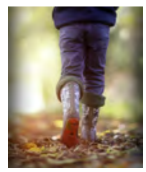
8 The Youth of the Industry
by Andy Shaughnessy

60 The Proximity Effect
by Barry Olney

66 Designer Highlights From IPC APEX EXPO 2019
by Stephen Chavez

70 MakeHarvard 2019: Bigger and Better!
by Bob Tise and Dave Baker

76 I Think I'll Go for a Walk
by Tim Haag



80 Important Considerations for Conformal Coating Selection and Performance
by Phil Kinner

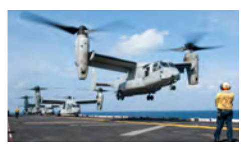
DEPARTMENTS:

87 Career Opportunities

96 Events Calendar

97 Advertiser Index & Masthead

HIGHLIGHTS:



58 PCB007

74 MilAero007

84 Top 10 PCBDesign007

The Youth of the Industry

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

When was the last time your company hired someone straight out of school, or even under 40? Until recently, I would have guessed 1985. But there's something happening, and I hope it's the beginning of a trend. Young people are once again entering the PCB design community workforce, and the overall PCB manufacturing industry as well.

It's not a flood of youngsters—yet. They're not joining us by the thousands, but certainly in the hundreds. You even see young designers and EEs at trade shows now, chatting with us silverbacks. And it's refreshing to talk to these "kids," as I term anyone under 40. They have an entirely fresh worldview, one that we don't hear about when we're talking with other middle-aged folks who have been in electronics design and manufacturing for decades. They're excited, not jaded and killing time until retirement.

Further, the young people of the industry discuss apps, altered reality, and drones, not the "good old days" of 1990, or 1970. (I know some of you started in this industry way before that!) They even chuckle at our war stories. They've never driven a rental car around Toronto while wrestling with a fold-out map on the way to that next appointment. They've also never used FedEx Overnight to send a file to a co-worker.

These young people look at us the way I looked at my grandfather when he told me about seeing his first automobile and checking if there were tiny horses under the hood. They're curious, and they know that we have some knowledge to pass down. They know they're in demand, which gives them the freedom to say what's on their mind. I enjoy hearing young people say, "You're older than my dad!" Good. That means they will be around to steer their company and this industry into the future after we all retire or slip the mortal coil.

Project MARCH team, 2019–2020.

Right now, we're still not seeing young people coming into PCB design in numbers that are even close to replacing the graybeards who are leaving over the next decade. There are still too many positions sitting open, from design through assembly. Think about it from the viewpoint of smart, young, would-be technologists. Why should they consider working in this field? Even more importantly, why should they be interested in working in your company?

For this issue, I spoke with a variety of young people working in this industry. The fact that I was able to find eight people with no gray hair speaks volumes—it would have been tough to publish an issue like this 10 years ago.

First, we bring you an interview with Professor Gary Spivey of George Fox University in Oregon. Gary discusses his electrical engineering department, one of the few in existence that provides students with real-world experience designing PCBs. Nolan Johnson also spoke with two current GFU students, Alex Burt and Jake Whipple, who discuss their PCB designing experience and future plans.

Next, we have my conversation with Nicole Pacino, CID+, a design team leader at Cobham and daughter of Mike Creeden of San Diego PCB. Nicole explains how she got into PCB design by working one of Mike's designs like a puzzle and never looked back. And Insulectro's Megan Teta, CID+, explains how she wound up in this industry after studying to be a pharmacist and offers advice for companies seeking to hire recent grads.

Further, I interviewed Geoffrey Hazelett of Polar Instruments who joked that he may be one of the more "elderly" millennials. Geoffrey explains why so many smart graduates are attracted to other fields that are considered "sexier." Then, I spoke with Altium's Andy Johnson about his work in EDA marketing, and he offered advice for young people considering working in this industry.

Circling back to professors, we have my conversation with Pablo Sanchez Martinez, a Spanish student studying engineering at an Austrian college who wants to become a full professor and teach PCB design. Then, we have an interview I conducted with Martijn van der Marel and Roy Arriens, students from Delft University of Technology in the Netherlands. They are members of Project MARCH, which designs and builds exoskeletons that can help paraplegics walk.

We also have columns by our regular columnists Barry Olney, Stephen Chavez, Tim Haag, Phil Kinner, and Bob Tise and Dave Baker.

Yes, the youth are making inroads into the world of PCB design and manufacturing, but we need a lot more of them. If you have any ideas about this, I'd love to hear them.

See you next month! **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.

GFU's campus includes buildings dating back to the early 1900s, such as Wood-Mar Hall.

George Fox University: Teaching PCB Design to EE Students

Feature Interview by Nolan Johnson and Andy Shaughnessy
I-CONNECT007

We've all heard the stories about engineering students entering our industry with no idea how to design PCBs. It's not their fault; most students aren't exposed to PCB design in their electrical engineering curriculum. But George Fox University (GFU) is an exception to the rule.

Gary Spivey is director of engineering projects at this Christian college in the Pacific Northwest, and his students learn to design and fabricate a PCB while also giving back to the community. Not surprisingly, these graduates get snapped up quickly. In this wide-ranging interview, Spivey discusses GFU's engineering curriculum, their cutting-edge lab facilities, and the need to teach students to think critically.

Andy Shaughnessy: What is your background and how did you end up working at George Fox University?

Gary Spivey: I was an EE major at the University of Arizona in '88, and I went to work for the National Security Agency, which I enjoyed very much. While there, I did a three-year stint in Scotland, and upon return, went back to grad school at the University of Maryland at College Park.

I received my master's and was working on my Ph.D.; I had everything done except for that little essay. Then, I took my family back to Arizona. I got a job in Tucson with a defense contractor and finished my Ph.D. from there. About that time, I thought, "Well, what now?" I was browsing the web one day and stumbled upon GFU, a small Christian school in the Northwest that I didn't know anything about; I wasn't even looking for a job as a teacher. I noticed that they had an engineering program and were looking for someone, and upon a more detailed look, realized that they were looking for me. My major and minor and the company I was working for were the top three things they wanted. I let it sit for a year, and the position was still open. So, we decided to

come up here and help start this engineering program. That was in 2003 when we had our first group of seniors. It had been a three-year program before that.

There were four engineering faculty at that point, including myself, and then the program got accredited with about 40–50 students; now, there are over 300 students and approximately 15 faculty. We began with concentrations in mechanical and electrical engineering, and now we also have computer, civil, and biomedical concentrations.

Nolan Johnson: Can you tell us more about your engineering program?

Spivey: Students are initiated into the program through our engineering principles courses. There is a two-semester sequence in the first year where students have a significant amount of hands-on education. In the fall, they are trained in machine shop equipment and solid modeling, and use these skills to design and manufacture a small air engine. In the spring, they learn basic electronics and programming, and use those skills to create and build an Arduino-controlled solution for a constrained theme (e.g., a new kitchen appliance or something to improve the dorm room). The theme changes each year.

Sophomore year is fairly standard and rigorous as students learn the fundamental skills in their concentration. In their junior year, students participate in our Servant Engineering Program. The program began in 2010 and allows every junior to participate in a service learning engineering project.

In their senior year, students participate in our senior design capstone experience, which is a full design experience with a professional company. There are about five people on a team, and we do it a little differently than many schools; instead of having the senior design teams supervised by a representative from the company sponsor, each team has two faculty mentors. Students work for the faculty, and the

GFU holds job fairs every fall to introduce students to companies in the electronics industry.

faculty are responsible for interfacing with the customer and maintaining the integrity of the project. This better reflects the industry model and also helps provide the teams with close mentorship during the project. Students are generally expected to deliver a prototype at the end of the year.

Shaughnessy: That's pretty cool. I understand you have a little more focus on PCBs and PCB design, and a lot of EE curriculum?

Spivey: I can't speak to what other schools are doing, per se. What we do is during their freshman year, students work with Arduinos and learn how to code and build some systems. Sophomore year focuses on FPGAs and digital object for the EEs. At the start of their junior year, students take a microprocessors class, learn how to code everything on a couple of different micros, start with an eight-bit AVR, and move to a 32-bit ARM.

In the spring of their junior year, EE students have an embedded systems class where they learn how to use Altium Designer. They design a simple board and have it manufactured using our Voltera PCB printers. Then, they use our LPKF pick-and-place system and reflow oven to assemble the board. Later, they design another slightly more difficult board and have that fabricated using a PCB house, such as Sunstone. They complete the semester by developing their own project. I help them do all of this, but each student does their own project. The goal is for students to design the schematic and layout, order the parts, have the board fabricated, solder it together, code up the microprocessor, and make the whole system work.

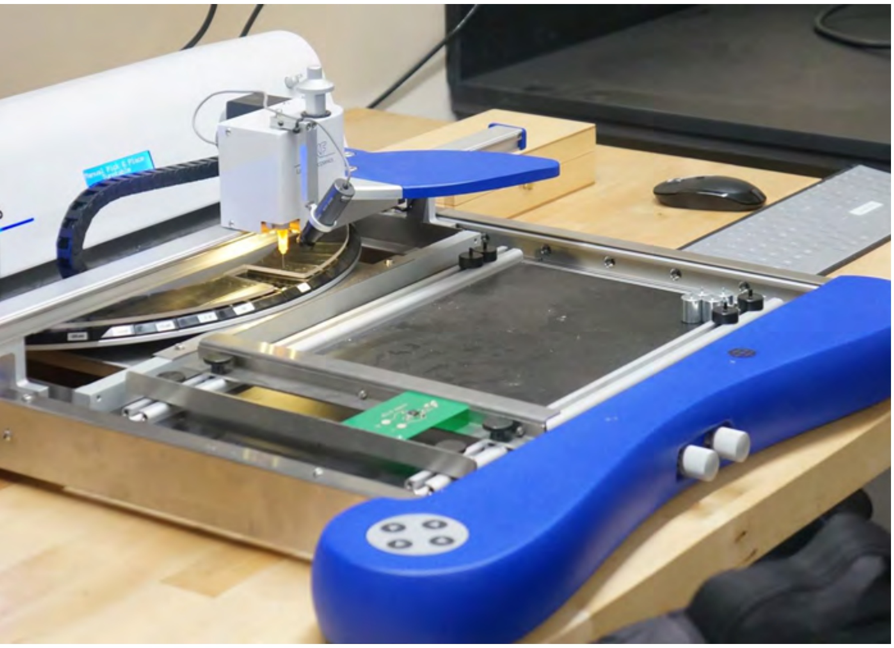
Also, in that same semester, for our applications of microelectronics course, students build a power amplifier and do another PCB design and layout. While learning all of this, they might also do a PCB as part of their Servant Engineering Program, and in the following year, it is quite common for them to build more PCBs as part of their senior design experience. Thus, many of our students come out having built five PCBs with all except the first couple being completely their own design.

Shaughnessy: That's definitely not the norm. We talk to EE grads and other students, and a lot of them have never designed a board or heard of copper pour. And of course, their instructors say, "Well, that's a manufacturing discipline."

Spivey: I think it's a thinking discipline. I teach digital system design, and I've done microcode and FPGAs for years. When I teach microprocessors and talk about coding styles, I think that's important to teach. It isn't just a style; it is a proper way of abstract thinking. Further, I don't use a textbook for microprocessors or embedded systems; we just use data sheets. Good, bad, or whatever, it's not my problem; I want them to read it and figure it out because that's what students will have to do in real life. So, I teach them how to think on their own and find answers.

I teach them how to think on their own and find answers.

In my professional career, I designed wire wrap boards using discrete integrated circuits, and moved on from there to ASIC design, FPGAs, and embedded coding, but I've actually never designed a PCB. So, that's my PCB knowledge. Of course, I've read about pour and other techniques, but at the level we're doing it, it's not rocket science. I tell the students to research it, figure it out, and just get it built. There's a whole career to learn details later, and we're not going to get to everything as undergrad. We're not going to get into EMAG and crosstalk, etc. But they should be able to figure out how to utilize the tools on their own with a little guidance from me when they get stuck. I grade it like a professional project. I don't have tests and such. I have some milestones that I want them to hit, but it is a competency-based outcome largely. I typically give students an A, a B, or an F. I can tell who has taken on a hard challenge and who has done well at it.



The engineering lab at GFU features some of the latest PCB equipment.

Shaughnessy: That's a good sign.

Spivey: Yes. TZ Medical is another local company that's almost full of GFU students. One of their engineers who is now the CEO hired one of our grads as the second engineer. Now, they have 10–12 engineers, and they're almost all GFU grads.

Johnson: I've been to your facility and seen your lab. Could you give us a verbal tour of the lab facilities you have for students?

Spivey: For the first few years, we started off with a perfboard in the first year, but we moved to doing PCBs. We used to use Electronics Workbench Tools, but I had students graduate and tell me, "You need Altium." So, we got Altium back and have been using it for years now. We would have the boards fabricated. In the early 2000s, we were doing through-hole stuff as much as possible, so we learned how to do the hotplate method.

Then, we learned about getting the templates from Pololu for the laser-cut Mylar. A couple of years ago, we acquired the old cafeteria at the engineering school, and now that's a 15,000 square-foot maker hub. We have a machine shop, wood shop, welding area, a prototype lab full of 3D printers and laser cutters, sewing machines, computer lab, meeting rooms, etc.

We also have a PCB lab in there. Now, we have Voltera PCB printers, which we use mostly for getting boards back from fabricators and printing the solder paste on them; they work very well for that. We also have an LPKF pick-and-place machine and reflow oven. We just added a microscopic camera as well, so we can look at the joints while we're soldering. We've also added some new rework tools for inevitable student mistakes.

Johnson: How long have you been doing PCBs in the lab?

Spivey: Around 13–14 years.

Johnson: How does that translate into employability for your alumni?

Spivey: We have close to 100% employment.

Shaughnessy: I saw the list of companies where your graduates have gone, and it's an impressive list with names like Intel, Boeing, etc.

Spivey: Tektronix has been a big destination for our students. They wouldn't hire from us 8–10 years ago. Then, they finally got one of our students in their embedded design group, and he won the presidential award that year. They hired another one of our students, and he also received their presidential award. After that, they hired another one of our students who got the presidential award again. They kind of quit giving it to that unit, but since then, they've been raiding us pretty heavily. We have students at a number of other places around as well, including Rockwell, Teradyne, A-Dec, Intel, and many more.

Johnson: How do you see your program and industry professionals working together?

Spivey: We rely heavily on industry input to help us decide what tooling and equipment to teach. Industry professionals are very knowledgeable on new trends and when it is appropriate for me to upgrade or alter my courses.

One challenge I've had with people from the industry over the years is the idea that we need to teach the students more about whatever their particular specialty happens to be. I generally respond, "No, you need to do that." I find that industry professionals often forget how little we all knew when we were undergrads. Most of these students didn't know Ohm's law two years ago. They are learning how to design. We are trying to teach them foundational skills as well as the ability to think well. This is why I focus so heavily on project-based experiences.

One of the beautiful things about the project-based experience is that you learn while you're doing as opposed to learning how to get through the assignment. The focus isn't the details of crosstalk as much as it is the understanding of how to process and apply new information. When you encounter crosstalk in your design and have to fix it, that lesson—as well as the thinking process or debugging it—stays with you for a very long time. If you understand how to think in the context of PCBs and electronics, you can learn crosstalk pretty quickly.

> **One of the beautiful things about the project-based experience is that you learn while you're doing as opposed to learning how to get through the assignment.**

Johnson: What I'm hearing you say about getting industry involved is making sure that experience professionals are aware that undergrad EEs are fundamentally learning how to think.

Spivey: The first three years in EE are pretty standard. Students cover signals, microelectronics, microprocessors, EMAG, etc. But in their senior year, it's generally all elective; there's not really a particular, required class. And I used to wonder, "If nothing is fundamentally required, why are we doing it?" After those first three years, students have the foundational skill set to be an EE. However, their thinking process is still weak. Before we let them go, we want them to take some more courses of their choice and learn to take those fundamental skills and apply them in different areas. That extra year is not an accident; it has evolved over time as we consider how to educate engineers. We need that fourth year of seasoning to get them thinking deeply.

Shaughnessy: It sounds like students are getting a good cross-section of experiences at GFU with EEs actually learning about PCB design.

Spivey: Right. The way we view it, we have freshmen come in and learn how to use a machine shop because you need to know how to build something; it's the same for EEs beyond all of the technical career stuff. You should know how to make a board. I think it's irresponsible to graduate engineers who can't build a board; that just seems odd to me. It is so readily available and easy to do, and it is such a powerful building block for so many problems, so why would we not empower students with this skill set?

Shaughnessy: Are you noticing changing attitudes of students when they come in? Have you had to change your teaching style over the last 10 years to "click" with the students?

Spivey: It's a little more difficult for us because we've gone from 50 to 300 students, and there has been a pretty drastic change in how we interact with that number. We have also changed our curriculum over the years. It used to be a common first two years for everybody, but now, it's a common first year. My digital logic class went from being electrical, mechanical, and civil engineers who hated being there but had to take it, to electrical and computer engineers and a bunch of computer science students who want to be there because it's in their major.

Given all of that, I always give them a very straightforward and fair first test on Boolean algebra, but the average score is always around 55. If you ask any student, "Was the test fair? Should we know how to do this?" They would say, "Oh, yeah." Something about learning digital logic is a major context switch for students. Basic mathematics in binary seems to confuse students who have been through semesters of calculus and differential equations. So, even with the students changing, this hasn't been any different; it's the same frustrating thing this year after year (laughs). I can't seem to change it no matter what I do. It just appears to be part of the learning process that sophomore-level students must go through.


Gary Spivey

Shaughnessy: Do you have programs set up with local companies?

Spivey: Yes, A-Dec has an internship with us. We don't do MECOP [Multiple Engineering Cooperative Program, the Oregon-based co-op program] because we're on semesters, but we invite companies to job fairs in the fall. Because it's a small school, students can receive more attention and direction than at a larger school, so that can be an advantage for our students. We have a lot of private communications with industry companies where we can promote our students to them.

Shaughnessy: I think it's great that they get to go out and give back to the community.

Spivey: The Servant Engineering Program has evolved continually over the years. The original idea was to get students to use their skills to help other people, and we realized fairly quickly that students have limited skills. We said, "That makes sense; that's why they're in school." So, we've had to revise to the program to give them more guidance throughout. We have worked very hard to keep the emphasis on teaching engineers how to use their gifts to serve others. We're happy with the direction of the program now and are excited to keep pointing engineers to a life of service.

Shaughnessy: This has been great, Gary. We talk to new EEs about manufacturing, and a lot of them say, "How would we learn about fabrication? We only learned formulas."

Spivey: There are a lot of details involved in getting into fabrication, but you just have to let the students learn them as you go. We have a 20-page design rule check that the students hate, but it's full of every stupid thing we've done.

I had a student once who had learned about blind and buried vias. That sounded useful the him, so he put some in. He had no idea that those are expensive and are not done with the fabrication methods we use. Now, we have a part of the design rule check that says "Are you using blind or buried vias? Stop it." The design rules are full of things like that. It is important to try and capture all of the details and resources and keep those available for students each year with each year learning from the previous years.

Johnson: That's great. Is there anything else you'd like to add?

Spivey: I don't think so. Thank you for the opportunity.

Johnson: Thanks again, Gary. DESIGN007

Putting University Training to Work

George Fox University senior Jake Whipple.

Feature Interview
I-CONNECT007

At a job fair on campus at George Fox University, Editor Nolan Johnson sat down with Jake Whipple, a computer engineering senior, to discuss the GFU engineering program. This is one of the few engineering programs in the U.S. that gives students experience designing PCBs before they enter the work force.

Nolan Johnson: Can you start by giving me some background on yourself, where you're at in your education, and any work history?

Jake Whipple: Absolutely. I was raised in southern California and came to Oregon to study computer engineering at GFU. As I moved through the curriculum, I've been enjoying embedded system and PCB design lately. I had a tendency even in high school towards learning about technology. So, I selected computer engineering as my major knowing it would probably be the best fit.

At GFU, we immediately hopped into making an air engine, which was a very mechanical project. The next semester, we built a robot, which was cool. I thought, "This is what I want to stick with." Fast forward a little bit, and it has been a very rigorous four years. I've learned a lot of life lessons, and it has been a crazy grind, but it's coming full circle at my senior year. Now, I'm looking into going into the industry. A big thing for me is I've been helping in the engineering labs a lot, and I get to work with a PCB machine. Last year, I had the microprocessors and embedded systems class. Now, I'm helping with that workflow for the next juniors that are going to be coming through that program.

LPKF benchtop convection oven used at GFU for lead-free reflow soldering.

George Fox University holds regular job fairs for its engineering students.

I get to train some volunteers to help with PCB design. We have a Voltera machine that prints traces and their whole thing is that they don't want to wait for a manufacturing lead time. So, you get to make your own boards in-house. And it's not meant to be a final design, but it's meant to be something you can prototype on before you send it off to Sunstone or OSH Park, etc.

It's all in-house. You can take substrate, pop it down, and I get to train other people. The juniors are going to get to come through and learn how that process goes. It's giving me a real look at the industry. I've decided that I've learned a lot in school, but I'm looking forward to getting a job, so that's where I'm at right now.

Johnson: Have you interned at all yet?

Whipple: I worked this past summer with Dr. Harder and Dr. Stillinger at GFU. Dr. Harder is the dean, and Dr. Stillinger is an electrical engineering professor. We did a renewable energy project with the U.S. Department of Agriculture (USDA), and I was able to work from campus remotely. We were into a renewable outreach program to local farms, which was a big thing. The USDA had some grants that they were able to promote to these people, and as an intern, I was able to make phone calls and do some bookkeeping. I saw what that process looks like for companies that want to switch over to solar panels and renewable energy sources.

I also worked at the college's engineering maker's space in the summer, which was enjoyable. I got to play with all sorts of machines. We have a laser cutter, a vinyl cutter, the PCB machine, lathes, mills—all sorts of toys. It's really fun stuff. I have two years of experience being here at the shop and learning all about our machines.

Johnson: Let's shift topics and talk a little bit about the curriculum and how PCB design and manufacturing is included in the curriculum for you as a computer engineer. Did you have any prior experience with PCBs before you got into that part of the class, and how did that go for you?

Whipple: I had no experience going in. It really starts in your junior year. If you're on the standard curriculum, when you're a junior in the fall at GFU, you take microprocessors. You're given a PCB that is already laid out, and then you have to code it and make programs for it.

George Fox made this board, and it's ready for all the programs that you do here; you just have to code it. The spring semester of your junior year is embedded systems. Now, you take your knowledge of coding and get to design the board. You use Altium Designer to lay out everything and learn how that process goes. You put all the vias down and traces and do that entire process, buy the parts that you need, and write the code for the project. What I loved about it was it was full circle. In the industry, you might only touch one of those things, but at school, we got to see how cool the whole process was when it came to fruition. We also got to tour Sunstone and see how that process looked, which was interesting.

Johnson: That changes your perspective. Just because the CAD tool can do it or represent it doesn't mean the fabricator can make it.

Whipple: That was something I learned just recently. We have a PCB machine that can drill holes. I thought, "I'll make this via this many millimeters, but then I go up in the machine, and I only have a drill bit that's this many millimeters. That's not going to work." I kind of take it for granted because when we were at Sunstone, they had a bunch of tools, so you can probably get by without looking at the specifications, but not here. We only have eight drill bits. You have to make sure that your via falls in that range.

It was really interesting learning about that process in the tour and see what manufacturing looked like. As we walked through, I wondered, "Where are we in the stage here?" There was a lot going on at the plant, and it was kind of overwhelming walking through. I felt like I didn't get to see a lot, and even though I didn't follow it quite correctly, you can design your board at GFU, pick the parts, code it, and ship it off. When it comes back to you, you get to see that whole process.

Johnson: That's a valuable part of understanding how PCB fabrication works. It helps to go there and see that the boards swish and agitate in tanks of chemicals. That's a part of the process. It isn't all digital, and it's not clean like 3D printing (laughs). From there you've moved into putting together your boards and then designing your own boards, how did that work?

Whipple: In that class, you have to choose a project that kind of needs to be run by Dr. Spivey, our instructor, because he doesn't want it to be super easy like just turning an LED on. He wants you to have some sort of goal with good functionality and multiple things that we've learned in the past implemented on it. I decided to go with embedded system design. It was a circuit board, and it's called a Bible quiz hub—a game show system with six buttons that represented the players. When you pressed it, the order that those six players responded to the question would light up in an LED matrix showing the order.

I went through the design process and sent my board off to Sunstone. I thought, "I hope I bought all my components correctly and I hope my schematic was right." I had a seven-segment digital display on there, and I completely wired it wrong. I don't know what I was doing that day, but the schematic was just completely wrong. I couldn't even get power

The engineering lab at George Fox University has the latest tools and equipment.

to it, and there were a lot of traces going, on so I didn't bother with that one. I wired a transistor wrong, so I had cut the traces and solder a little wire. You only get one shot in our class, so some people's projects in the past just didn't work. I heard about one student last year where nothing worked on the board. By the grace of God, I had a few components that got the job done. Not everything worked, but it was cool to see my schematic on the board, the board back, all the components, and the final project. I went from CAD to seeing Sunstone produce my board.

Johnson: Have you done anything more with PCBs since this class?

Whipple: That summer, we did a posture monitoring device, which I got to design; it was really fun. I learned a lot from embedded systems, and I had a little bit more time in the summer to make things nice and neat. I ordered it through Sunstone. It was a very simple circuit that monitored your posture in a chair. We enjoyed that. Then, I might have to make a PCB for my senior design project coming up, but I'm not sure yet.

Johnson: Looking at your next project, and I can see the change in your approach and the efficiency that's here already by the second pass. You figure it out and learn what not to do, and then you take that experience with you to the next design where you learn something else that's new to you.

Whipple: Right.

Johnson: These are great boards and solid designs.

Whipple: Thanks. And the speaker you see we designed on Altium. It was a through-hole board, but it was laid out in Altium, and then it went to Sunstone and back. This was a class project as well. The speaker was a hard project—definitely more of an electric circuit design. You're dealing with analog signals, so digital is out the window, and a lot of resistors for the proper gain on the speaker. It was challenging. There was one point where I dropped a metal screwdriver, and it hit two transistors and shorted them. I thought, "I have to go and find the right pair of transistors that kind of match again." Certainly, that was not the way I had planned that to go. That board was a full analog process, which was so different.

Johnson: And here you are, wrapping up your degree and ready to go into the industry. You have analog experience and multiple iterations of doing PCB design experience to go along with your computer engineering degree. That's pretty solid.

Whipple: Thank you.

Johnson: Are you talking to companies? How is recruiting going?

Whipple: It's going well. I talked to a few people here at this engineering expo, which was nice. I've made great connections from our senior design team and professors that I've worked with. I've also been a TA through a few professors. One of them has really been on my side as far as getting me plugged in. And I know some other people and opportunities I could pursue. I just had an interview with Tektronix based in Beaverton, Oregon, so that was encouraging. We'll see what comes from that. That's the most serious opportunity recently.

I spoke to a couple of other companies as well. I don't know if they're looking for my particular degree, but I'm still making connections. What I love about George Fox is that sometimes, people get hired right off their senior design teams. We also have the opportunity for mock interviews with other individuals, and sometimes real job offers come from those mock interviews.

Johnson: I'm sure that helps. Good luck with your interviews, and thanks for your time.

Whipple: Thank you. DESIGN007

Preparing to Enter the Workforce With PCB Design Experience

Feature Interview
I-CONNECT007

Editor Nolan Johnson spoke with Alex Burt, a computer engineering student at George Fox University, at a well-attended college career fair on the GFU campus in Newberg, Oregon. In the interview, Alex discusses his PCB classwork, challenges of design, and how it has impacted his internship experience as he prepares to enter the workforce upon graduation in the spring semester of 2019.

Nolan Johnson: Great to meet you, Alex. Can you tell me about the company you work for and what you studied in school?

Alex Burt: I currently work as a software intern at Summit Wireless in Beaverton, Oregon, and I am a computer engineering major at George Fox University. I'm graduating in the spring of 2019.

Johnson: Did you have the opportunity to go through the PCB process at George Fox as an undergraduate?

Burt: Yes.

Johnson: Tell us a little bit about that from the student's perspective. What was involved in that?

Burt: I was completely new to PCB design at the start. But through our electrical engineering class that Gary Spivey teaches at George Fox, we worked with Altium to understand how to create components on a PCB, create their footprints for a PCB, route these devices together, and create a full circuit not based on a breadboard.

Everything we had been doing in our electrical engineering classes had been based off a breadboard, and in one semester, we had two classes where we built two different PCBs. In one of the classes taught by Dr. Natzke, we built a custom amplifier housed with speakers, which is something that wouldn't have been possible on a normal breadboard. But through designing the PCB, we reduced noise levels and made it compact enough for permanent deployment in an actual device.

Overall, I knew nothing about PCBs before these classes, but now I can say that I could design an entire circuit on a PCB and have it sent out to a manufacturer or use our in-house PCB lab at George Fox. We can print out solder paste or print traces with the Voltera machine. We also have a pick-and-place machine to place surface mount components, and then we put it in the solder oven. But if we don't do it in-house, then we would put fiducials on it and send it out that way too. All of our surface mount components have gone through the Voltera to put the solder mask layer on; we've placed all the components with the pick-and-place machine, put it in the oven, and these devices all work.


Alex Burt

Johnson: At the table in front of us, there are four different boards. Can you tell us about them?

Burt: I have another one at home, but I have designed two of these, and I fabricated three of them.

Johnson: You haven't graduated yet, but you're working in the industry.

Burt: Correct. I'm an intern at Summit Wireless where some of my coworkers have to design PCBs for electrical engineering. I have to understand how those PCBs work, such as sizes and temperature limits, and how their circuits are laid out to interact with them via software. So, we all have to understand how it works, including all of the limitations with PCB manufacturing as well.

I have to understand how those PCBs work, such as sizes and temperature limits, and how their circuits are laid out to interact with them via software.

Johnson: Right. One thing that is interesting in our industry is that many of veterans comment that young designers don't really know what the limitations are, but you're getting that experience. You had to start basically from scratch by learning a CAD tool and how to create a schematic.

Burt: Exactly. Building a schematic and the actual circuit.

Johnson: And then translate that into a physical layout.

Burt: Yes, and in one class, we had to design it on a breadboard after the theoretical design. So, we wrote out the schematic in a simulation program such as Multisim or PSpice, and when we saw the simulation work, then we would breadboard it. After the breadboard would work, then we would design another circuit in Altium and have the components designed—the footprint—to be placed on a PCB and lay it out. It's a long process, but it's worth it. I still use the stereo speaker I built in that class to this day when I exercise.

Johnson: Awesome. How did the knowledge you gained on PCBs follow you into your work as an intern?

Burt: Understanding how printed circuits are designed and built, how they start with a basic schematic, and how they're laid out together on a PCB is important to bring together different chips. Also, how they interact with each other on a hardware level is essential. And the need to understand the circuits and size limitations with PCBs.

Johnson: For a moment, imagine that you didn't have the PCB education available to in your internship at Summit. Would your work be harder?

Burt: It would definitely be more difficult. I would look at the board and not understand what all the holes do. I wouldn't understand that there would be multiple layers on the board because you can have a board with many, many layers, and I wouldn't understand what a polygon pour is. And you see these different colors on the PCB—the darker and lighter—and I would have never even realized that one of the shades means that it's all one layer; it's all one wire in breadboard terms, at least.

Being able to see that you can jump from one trace to another layer, and then to another trace, or even to another plane and seeing all sides of the board is helpful. Without that knowledge, it would be unfortunate. PCBs are much more complicated than you would initially see because it's not just a flat, 2D board; it's 3D and could have multiple layers or buried vias. You could have so many different things inside of a PCB. If this was put in a breadboard scenario, it would be 20 times bigger physically.

Johnson: Out of the entire design flow—coming up with the concept of the circuit, building up a schematic, all the way through to getting an assembled board, bench testing, and now doing your own designs every day in the real world—what was the hardest part or the biggest cognitive step for you?

Burt: The most difficult part was the layout design of the components. To lay them out in a fashion to use the least amount of space while still being accessible, to be placed if it's a surface-mount component, or to be soldered if you're using a through-hole component can be challenging. Also, getting the traces to their proper destination is also a factor when you're placing components on a PCB and designing.

Another hard thing for me was designing where each component needs to go because if I put a component in a certain place, other traces don't have very far to go to get to their next destination. The other part might have a harder time, so you might have to jump to another layer, but then more layers add more cost to the board as well. If you want a simple, two-layer board, then you have to come up with a pretty smart design to keep that cost down. Keeping a cost versus size versus organizational scheme when you're physically laying out the components in the program was also tough for me. The first time I built it, I changed my design multiple times—maybe three different designs—and ended up with the one with the least amount of trace length and the fewest board layers.

Johnson: Sounds to me like you spent a lot of your time getting a smart placement before you did the routing.

Burt: Yes. It's very important to practice because it takes a lot to get good. By my third PCB, I got quite good at it, and it took me just so much less time—an exponential drop in time. The first board I built took me so many days and weeks to get it right. The second board was a little less, but it still took me a long time. With the third board, I improved quite a bit and was able to do a simple design within hours. At first, even a simple design would take me so long, so practice is very important. To get started early is also very important as well as having a class on it to practice.

Johnson: You're transitioning into the industry now, and you interviewed with a test and measurement company in the area, right?

Burt: I did.

Johnson: How did your PCB design experience contribute to how that interview went?

Burt: I was able to show not only my software experience to the company but also my hardware and circuit design experience to them. With electrical engineers, it's important that they know how circuit designs and PCBs work because everything in the industry is on a PCB. There aren't breadboards; there's even testing usually done on the PCB. I was familiar with that and showed them that I had experience and an interest in it. They believed it was a benefit because I wouldn't have to learn how a PCB works for their designs. Instead, I can go in right away, interface with their designs, and continue where they've left off in the design process. I think that was important to them, and I definitely think it helped me versus if I had no idea how to do a PCB.

Johnson: Especially for a test and measurement company.

Burt: Exactly. All of their products are analog to digital. That's essentially everything that they do, which requires custom-designed circuits. Of course, those are going to be on a PCB, so it's always great to have them right in front of me to show them, so they can look and be impressed.

Johnson: Good luck with that opportunity, and thanks for taking the time to talk with us. This has been great.

Burt: You're welcome. Thank you. **DESIGN007**

PCB Design is All in the Family

Nicole Pacino

Feature Interview
I-CONNECT007

Some PCB designers warn their children never even to consider designing PCBs for a living, but some designers like to pass down their talents to their offspring. Nicole Pacino is one such offspring.

Editor Andy Shaughnessy shared a flight with Nicole on the way to AltiumLive in Munich, and she mentioned that her father was speaking at the show. In Germany, Andy asked Nicole to tell us about how she got into this industry, and what we could do to draw more young people into this career.

Andy Shaughnessy: I'm here at AltiumLive Munich with Nicole Pacino, a design team leader with Cobham, a defense contractor. She is also the daughter of one of our good friends and PCB design instructors, Mike Creeden, who is speaking at the show. Nicole, can you tell us about how you got involved in the industry?

Nicole Pacino: Sure. Obviously, you know my father has been in the industry for a long time. And he actually started me out when I was really young. I was 11-years-old and had a knack for mastering those old school, classic computer games like Tetris and Solitaire. I figured out in Tetris that at a certain point, the high score stops and starts counting backward, so my scoreboard in the game was actually all negative numbers. With Solitaire, I figured out the equation it uses to keep score and could win with any number someone requested before I started a new game.

So, my dad noticed that I was showing an aptitude toward the type of skills that might translate to PCB design. The first time he presented it to me to see what I would do with it, he explained it like it was another computer game because he knew I would want to try to master it like the others. Sure enough, challenging me to this new game worked, and I haven't stopped trying to master it to this day.

Shaughnessy: Where are you working now and what kind of designs are you working on?

Pacino: I've been with Cobham in Pennsylvania since last year. I work on defense contracts developing new technology in aerospace communications and electronic warfare systems for the U.S. government. Unfortunately, working

with the DoD means I can't give any further specifics about which programs I'm involved in.

Shaughnessy: I understand you're working on a design that has hundreds of pages of schematics.

Pacino: Yes, these systems are definitely some of the most complex and advanced designs I've ever seen. We're pushing every boundary in regard to capabilities, and then going another step further to explore how we can advance current standards and evolve the next generation of technology development.

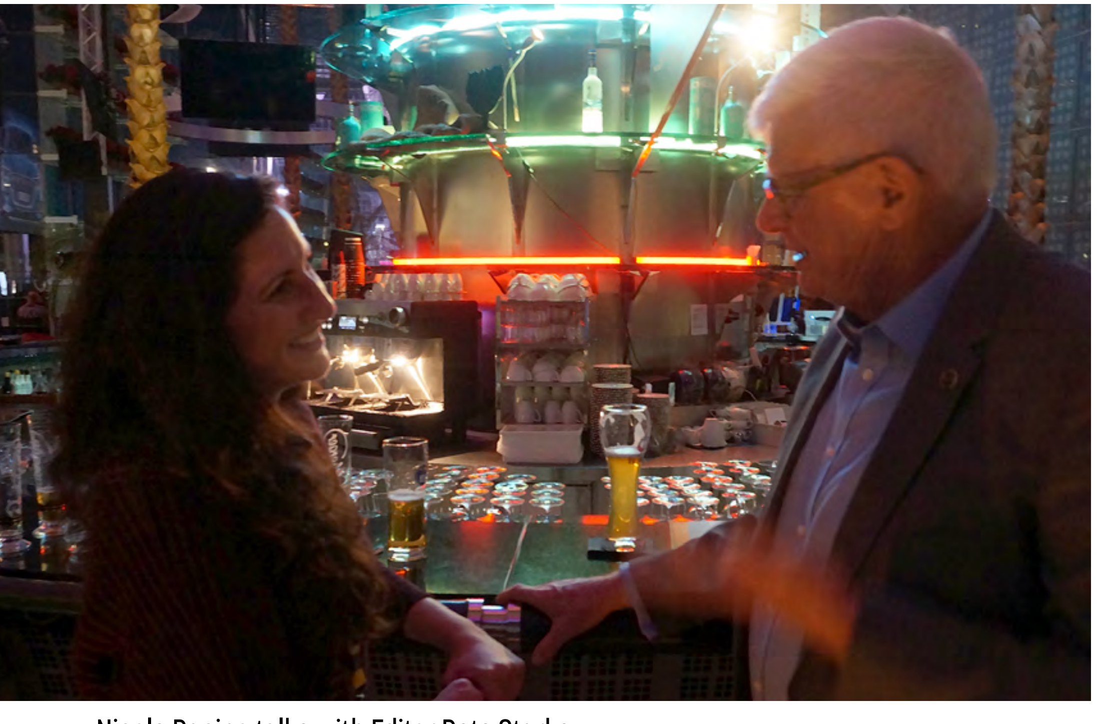
Shaughnessy: You're one of the younger people in the industry. What do you think we should do to attract more young people into this field? Do people your own age know about this as a career option?

Pacino: No, most people my age have never heard of what I do as something you can specialize in. As I get more involved in the industry, I've realized even more that there really isn't much of a younger generation here at all. I think most pursue the broader umbrella of career possibilities in electrical or mechanical engineering and only briefly learn the extreme basics in PCB design. I find it interesting that it isn't emphasized that this specific part of the process is extremely integral, necessary, and vital when creating new technology, and can be the sole determining factor in a project's success or failure. I'm not sure why PCB design is overlooked. It seems almost like it's a minimized requirement within our engineering education system. Everyone in the industry seems to only know how to design by "tribal knowledge," or they have become self-taught through years of trial and error.

Shaughnessy: Did you receive any sort of PCB design training in college?

Pacino: No. I attended Ohio State University and received two degrees that didn't involve engineering. I chose to pursue one in business and the other in communications. I do plan on going back to school in the very near future to pursue a master's degree in electrical engineering.

Shaughnessy: Some business education might come in handy. Do you think you'll start your own company?



Nicole Pacino talks with Editor Pete Starkey.

Pacino: That was my thought when I was still in my early twenties. I had already started my career in PCB design before going to college, so when I decided I wanted to go to college, it made more sense to go for business and equip myself with those skills and add that to my foundation since I was already mastering the technical side through informal education. And I think it was the best decision I could have made at that point. Having business knowledge and understanding all the ways it can be applied in almost every situation has been immensely beneficial in my career. Even if I don't end up starting my own company, having that business skill set will continue to be very impactful in my career.

Shaughnessy: Do you have any advice you would give to somebody just starting as a beginning PCB designer?

Pacino: As I always say, definitely listen to the advice of your seniors around you; they paved the way for everything we do and can teach you more from their experience than you'll ever read in a book. Take the time to be diligent and thorough in your work by ensuring you're using best practices to produce designs that are correct by construction the first time. Truly take pride in your work and be passionate about wanting to see your designs succeed because it's pretty cool to be a part of developing new technology for the future.

Shaughnessy: How do you stay up to date with your training? Do you go to many conferences like this?

Pacino: Yes. You learn a lot from these conferences. And you meet a lot of people who have an abundance of knowledge from all of the different areas of focus that are coming together. These conferences provide a really neat environment and atmosphere that is great for absorbing as much information as you can about new tips, tricks and an endless amount of new theories being worked on. I'm always looking for new conferences to attend so that I can continue to stay up as up to date as possible.

Shaughnessy: That's great. Thanks, Nicole.

Pacino: Thank you for the opportunity, Andy.
DESIGN007

Solar-powered Supercapacitors Could Create Flexible, Wearable Electronics

A team of engineers from the University of Glasgow has used layers of graphene and polyurethane to create a flexible supercapacitor that can generate power from the sun and store excess energy for later use. They demonstrate the effectiveness of their new material by powering a series of devices, including a string of 84 power-hungry LEDs and the high-torque motors in a prosthetic hand, allowing it to grasp a series of objects.

The research towards energy autonomous e-skin and wearables is the latest development from the University of Glasgow's Bendable Electronics and Sensing Technologies (BEST) research group led by Professor Ravinder Dahiya, professor of electronics and nanoengineering at the University of Glasgow's School of Engineering.

The top touch sensitive layer developed by the BEST group researchers is made from graphene. Sunlight that passes through the top layer of graphene is used to generate power via a layer of flexible photovoltaic cells below. Any surplus power is stored in a newly-developed supercapacitor, made from a graphite-polyurethane composite.

Similar supercapacitors developed previously have delivered voltages of one volt or less, making single supercapacitors largely unsuited for powering many electronic devices. The team's new supercapacitor can deliver 2.5 volts, making it more suited for many common applications. In laboratory tests, the supercapacitor has been powered, discharged, and powered again 15,000 times with no significant loss in its ability to store the power it generates.

The research, published in *Advanced Science,* was funded by the Engineering and Physical Sciences Research Council (EPSRC).

(Source: University of Glasgow)

The PCB Industry Must Work to Attract Young People

Megan Teta

Feature Interview
I-CONNECT007

Megan Teta, CID+, is a field application engineer with Insulectro. She graduated from college about five years ago and is now one of the few young female engineers in our industry. Teta discusses her job and shares her thoughts on ways that we can draw more young people into the PCB community.

Andy Shaughnessy: Can you start with a little background about yourself and how you got into this industry?

Megan Teta: I graduated from college in 2014 with a degree in chemical engineering. Then, I applied for a job through LinkedIn with a chemical company; they sell plating and final finish chemistry to the PCB world. I thought I was applying to work with their offshore business with oil, but I ended up getting in the electronics division. After that, there was an opportunity with Insulectro, a large distributor of Isola, DuPont laminate, and flex materials, as well as other PCB material suppliers, such as Pacothane, LCOA, and CAC, and some other PCB manufacturing materials. I've been with them for about a year and a half now.

Shaughnessy: When you were first going to college, did you have an idea of what you wanted to do or what you wanted to be when you grew up?

Teta: I actually started in the pharmacy program. From that, I transferred into chemical engineering, thinking I could do something in pharmaceuticals. I quickly learned that that's a lot more paperwork and a lot less fun stuff than actually changing and making an impact on anything. I wanted a job where I could work with more people versus being pigeonholed and only doing one thing day in and day out.

When I graduated in 2014, it was well after the recession, but it was still difficult to find jobs. I applied to anything and everything. I received three or four job offers, and I chose to take the position with MacDermid. It gave me a chance to get into a different type of indus-

try. I quickly learned that the PCB industry is a mature industry that's never going away and there are a lot of opportunities.

Shaughnessy: Absolutely. We need people your age. I bet you're always the youngest one in the room.

Teta: That's typically how it is. It's not uncommon for me to be the youngest and the only female, which isn't much different from college either.

Shaughnessy: Where did you go to college?

Teta: Rutgers University. Go Scarlet Knights!

Shaughnessy: Did you have any friends in school who were doing anything in electronics?

Teta: Because I came from the chemical engineering side and not electrical engineering, most of my peers ended up in pharmaceuticals or working for Johnson & Johnson. I don't know anyone who ended up working in electronics. It's a very small industry that most people don't know about, which is part of the challenge. You have all these students graduating, and we have a big need for young people in this industry that nobody knows about. Also, it's an older industry, so it's harder to find outlets to get younger people interested in this field. TTM does a great job recruiting interns and students from college, and other facilities do a good job, too, but a lot of companies don't. There's a huge need for engineers at just about every level in electronics, from design to manufacturing and assembly.

Shaughnessy: Did companies have job fairs at your school?

Teta: Merck and Johnson & Johnson both had a big presence. There were a couple of other big corporations that went to all of the job fairs too.

Shaughnessy: Especially in pharmaceutical because people are living forever now.

Teta: Exactly. I think that's something that a lot of people in this industry need to get better at is going to job fairs and trying to recruit younger talent. Obviously, it takes longer to train somebody, and it's a negative that you don't get somebody with experience, but you'll get somebody who will be around for a long time.

Shaughnessy: So, you're a chemical engineer by education. Do you find yourself doing any EE or ME stuff?

Teta: I do a little bit of electrical and a lot of mechanical. My title at Insulectro is field application engineer, but it's a lot of common sense, working through issues, and engineering from the logical standpoint. When I see a problem, I backtrack and try to find the solution. Understanding the whole process from the starting material to the end use has been helpful for finding solutions.

Shaughnessy: What's your favorite thing about your job?

Teta: One of the things I've been focusing on at Insulectro is Isola laminates as well as the DuPont flex materials. I'm not only working with the PCB fabricators but also the OEMs on the design side, trying to understand what their end use is and working between the OEMs as well as the fabrication shops. Insulectro has this program called ACT, which stands for "Accomplish Change Together." We want to work with the fabricators and OEMs and make sure we're recommending the right materials that will be good for their end use. I most enjoy going out and giving presentations and teaching people about these materials that they might have not been aware of.

Shaughnessy: And you have your CID+ certification, which is good.

Teta: When I first started with Insulectro, they wanted me to go ahead and get all of the extra education and the training. They've been fantastic with that. I received my CID certification a year ago, and I recently received the CID+. Obviously, it's the Certified Interconnect Designer program, but it's more about DFM, etc. I meet designers who know so little about the manufacturing process, and I try to work with them to enlighten them; it's not like you hit an "easy" button and a board spits out.

I meet designers who know so little about the manufacturing process, and I try to work with them to enlighten them; it's not like you hit an "easy" button and a board spits out.

Shaughnessy: Who was your CID instructor?

Teta: Gary Ferrari for both courses.

Shaughnessy: He's a great instructor.

Teta: I think everybody is aware that we need younger people in this industry. I know Insulectro is, but as I said earlier, it's harder to attract them and let them know what the industry is about and what they could be doing. Insulectro recruits both superstars and future superstars. We just hired another guy out in the San Jose area who's probably four or five years out of school. A lot of companies are trying to attract younger employees. We're getting there.

Shaughnessy: Have you visited any board shops?

Teta: I cover the whole East Coast, so I've been to just about every shop there—at least 50 of them. I've been to a lot.

Shaughnessy: Fantastic. Most of my readers are designers, and if they've been to a board shop, it was a long time ago, or a lot of them have never been.

Teta: What's interesting is there are many different ways to build a board and get the same final product. Some board shops will do certain things differently, which doesn't affect the end use but can make it either cheaper or faster to build. It all comes back to understanding the board shop's capabilities regarding certain aspect ratio holes, HDI builds, or buried and blind vias as well as how quickly they can do things. You also need to know what final finishes they have, especially with some of the high-speed and RF applications where you can't necessarily use just any finish because that's going to affect your end board. It's super important to understand their in-house capabilities too. Via fill is something that not everybody can do.

Again, Isola is one of our top suppliers, and we offer many of their materials that are glass-reinforced, which can help a lot due to the nature of the resin and how they're using it; then, they can use typical lamination cycles. A lot of the other materials, even some of the DuPont flex materials, need a higher temperature press. Knowing what facilities have that capability is important before you go and design something in. Something might be perfect for your application, but if you can only get it built by one or two fabricators, that's not always helpful either.

Shaughnessy: And then we hear horror stories of people designing a board for 3/3 spaces and traces, for example, that they didn't know was going to be built offshore. A designer might build and design a board, but then the shop in Asia can't do it, and the designer has to redo it.

Teta: I know this has been said a million times, but working closely with the board manufacturer is critical. You need to know what their capabilities are and what they can do versus just designing a board and sending it to them. They might have to make modifications, which delays the whole process. Make sure that you can manufacture it easily. Anything can be done, but it might take much more time and have a lower yield, which is never good. We work hand-in-hand with suppliers, designers, fabricators, and OEMs to ensure a viable pathway. We're all in this together.

Shaughnessy: Is there anything else you want to mention?

Teta: I'm very lucky. I entered the industry at a very good time, and there are going to be a lot of opportunities coming up.

Shaughnessy: Definitely. It was great talking to you. Thank you.

Teta: Thank you. I appreciate it. DESIGN007

Materials Chemists Tap Body Heat to Power Smart Garments

Trisha Andrew, materials chemist at the University of Massachusetts Amherst, and Linden Allison, her Ph.D. student, have developed a fabric that can harvest body heat to power small wearable microelectronics such as activity trackers.

Writing in Advanced Materials Technologies, Andrew and Allison explain that in theory, body heat can produce power by taking advantage of the difference between body temperature and ambient cooler air—a thermoelectric effect. Materials with high electrical conductivity and low thermal conductivity can move electrical charge from a warm region toward a cooler one in this way.

The researchers took advantage of the naturally low-heat transport properties of wool and cotton to create thermoelectric garments that can maintain a temperature gradient across an electronic device known as a thermopile, which converts heat to electrical energy even over long periods of continuous wear. This is a practical consideration to ensure that the conductive material is going to be electrically, mechanically, and thermally stable over time.

(Source: University of Massachusetts Amherst)

An Experienced Millennial on Hiring the Next Generation

Geoffrey Hazelett

Feature Interview
I-CONNECT007

During DesignCon, Editor Andy Shaughnessy met with Geoffrey Hazelett, VP of sales for Polar Instruments. Even though he has been in EDA for a few years, Geoffrey is still in his thirties, which makes him a youthful cherub in this industry. Andy asked Geoffrey what he thinks about the new PCB designers and EEs entering this field, and what more can be done to expose young people to the world of PCBs.

Andy Shaughnessy: We're finally starting to see more young people in this industry. Even though you joke about being on the "grizzled upper-end of millennials," you are younger than 50 and have all of your hair.

Geoffrey Hazelett: And I intend to keep my hair, so thank you.

Shaughnessy: What are your thoughts on young people entering the industry? When you were in college, was there any knowledge that this was something you could do for a career?

Hazelett: I recall asking my professors about this, and they told me, "Don't worry about PCB design. Somebody else will do that for you. You're going to do VLSI or ASIC design." I think what has changed is people with master's degrees and Ph.D.s in electrical engineering look at this and say, "This is how we're going to have to do PCB design." Now, it requires training on the education side that didn't formally exist even 20 years ago. And now, some universities—only a few—have signal and power integrity programs.

How are we going to pass on the information, knowledge, and experience from those people who fought the battles to develop this technology to the new designers coming into the industry? There are good resources out there—such as programs, training classes, and workshops at shows like this—but you have to be able to get your company to send you to a conference like DesignCon.

When I first joined, part of my hiring process with Polar Instruments was sending me to one of Eric Bogatin's last public signal and power integrity classes. Polar Instruments' President Ken Taylor said, "Think about the offer, but

either way, we want you to go to this because it's the last one that he's doing publicly." Now, Eric teaches at the University of Colorado and has an online academy.

Shaughnessy: Growing up, you must have had video games.

Hazelett: I remember before the internet was ubiquitous, the first computer I ever touched and played with was a 486—maybe an early Apple product. It wasn't until high school that I started seeing people with cellphones. Now, I have friends who have kids who are six years old and monitor their screen time.

Shaughnessy: It seems like today's young people would be more interested in a career like this because they've been exposed to handheld devices their entire lives. It seems like they would want to know how to design or built it

Hazelett: Unfortunately, a lot of this isn't flashy. Mark Zuckerberg is a few months younger than me. You're telling me I could have been a billionaire if I'd created a website? So, that leads to my generation saying, "I can just make an app and get a quick buck." You have a low barrier to possibly becoming a billionaire because you can have a good website or phone app versus PCB design and manufacturing, where there's a much higher cost of entry and infrastructure required.

Shaughnessy: How do we get more of the younger people involved? I see Susy Webb's PCB basics classes are full, but most of her students are currently EEs—not necessarily young people. Still, they're coming into the field.

Hazelett: That's what I was saying earlier. I'm an EE, and my professors told me, "A technician will do the PCB design." It was before the time when this was a specialized EE element. Fabrication shops are starting to acquire EEs to help them, not just chemical or mechanical engineers, but on the design side, and it's fascinating. There is a ton of exciting stuff that people are getting involved with.

Cellphones and video games have massively complex PCBs. That's what's really cool about my job: I get to go to companies that I'm a fan of and help them to make the next product that I'm probably going to buy. Or I get to find out what they're working on, which brings my excitement to a whole new level. I don't actually work for the company, but I support them and the chain because no one company does everything. I get to participate and think to myself, "They used the tools I trained them on to launch that product into the marketplace. I helped that device communicate with something in space. I helped this company with medical equipment."

Shaughnessy: We have to do a better job of tooting our own horn because half of the people in this industry are retiring in the next 10 years. How do we share this better with young students?

Hazelett: Polar Instruments started engaging that a number of years ago. I just celebrated five years at Polar, and six months before hiring me, the company hired Michael Bode who's even younger than me. We made that concerted effort, realizing there needs to be new blood welcomed into the industry.

Also, you have to be very careful because you can't just push out the older people either; then, you're approaching ageism. I've started to see some of that occurring in certain industry elements where companies remove an old-

er, more experienced person and hire two younger people to replace them. This doesn't permit a natural growing together of skill and knowledge transfer. I prefer how some large engineering companies will do a filter cycle, evaluating on merit and performance. If you're young, but you're screwing around, you're out.

Look at the newer tech companies that aren't necessarily designing hardware; they're software-based, which has been flashy and the sizzle. For our industry, it's not the hiring practices that are to blame; it's just that young people aren't drawn to it. Other tech industries are ironically facing the opposite side of the same coin: "How do we draw out wiser and more experienced people to our industry?"

Shaughnessy: What you're saying is that you see all of these things that are exciting, but they're just not as cool as the other technologies.

Hazelett: I think the reason some industries become very specialized for youth is that it's a new technology. Those who are older and are already doing something in the industry using their skills in, such as programming C, didn't need to retrain or learn the next popular programming language or system. Meanwhile, somebody younger came in who didn't bother learning C; they just went into Java. Then, when the company is looking for somebody with Java experience, the person who's been in the industry longer and is a good programmer but doesn't know that specific language yet gets dropped out. With PCB design, it's not really the same. A lot of these skills are transferrable. If you're using one layout tool or another, a transmission line is a transmission line, and connecting the chips together is still fairly similar. But as a friend of mine who runs a PCB design firm shared with me, "It's equally art and science to do layout right."

Shaughnessy: How did you get involved with Polar?

Hazelett: I stumbled into it really. I was introduced to Polar by the father of a friend of mine from early youth. I knew her dad who was a salesperson and used to work at Tektronix. Ken Taylor and a lot of the other guys at Polar have either worked at Tektronix or their spouses worked at Tektronix. He would stop by the office frequently and learned that Polar was hiring. He said, "I know someone who's an EE. He might fit." I had just come back from working in China for four years.

Shaughnessy: Before you came here, did you have a lot of PCB experience?

Hazelett: No. It's kind of funny because when I was an engineering student, my very first internship related to my major was a small engineering company called Engineering Design Team (EDT). A large entity bought them, but I was testing new cards and got some very specialized soldering training with surface mount—very small, hand-done parts—because it wasn't big lots. So, that was my first experience with it, which was even before college.

When I went to college, I had inquired with professors, saying, "I know a little bit about this. It would be cool when I'm working on a project or programming FPGA or some chips to be able to build this. They said, "No. It takes too much time and money. It's a waste. Focus on design." That was right around the time when FPGAs were starting to shine. I liked the flexibility of being able to reprogram a chip; that's firmware. Initially, in my senior year, I started a research project to work on some FPGA stuff, but then we shifted focus and built a robot instead.

Shaughnessy: Do you have any advice you would offer to young people who are maybe on the verge of entering this field?

Hazelett: I'm getting to do what I always wanted to do. I get to help people find solutions to their problems and equip those engineers and fabricators to solve them. I don't get bogged down in all of their problems. It's like they're building a house, and I get to hand them a hammer and some nails and show them how to use it. Then, I get to come back and say "Wow. That's a really cool house."

Shaughnessy: If you were talking with a random high school or college kid interested in electronics, what would you recommend?

Hazelett: I'd tell them to go make the next Facebook; I'm kidding (laughs). I'd say, "Imagine the potential. You could help design the next Xbox, Oculus Rift, Apple phone, or Google phone. There is the possibility that you could come up with the next hardware tool that transforms our lives." I would also encourage them to design and make their own projects or even follow the instructions someone else has put out there to make something.

Shaughnessy: A lot of engineering graduates don't realize that designing a circuit board is a career on its own. Maybe it's like your story where they were told, "That's what the techs will do."

Hazelett: Right. There are a lot of universities and professors. I can't speak that broadly, but there are some who know that this is important and this is a field, and they're going to be the ones to help educate and enlighten their students about this field—one where you could find enjoyment. Other universities have an ivory tower conception that they are educating silicon designers only, but they still need a way to connect that chip to other chips and devices!

Shaughnessy: Thanks for your time, Geoffrey. It's always a pleasure.

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

Scientists Develop New Method to Revolutionize Graphene-printed Electronics

A team of researchers based at The University of Manchester have found a new low-cost method for producing graphene-printed electronics, which significantly speeds up and reduces the cost of conductive graphene inks. The development of printed conductive inks for electronic applications has grown rapidly, widening applications in transistors, sensors, antennas RFID tags, and wearable electronics.

The team has found that using a non-toxic and environmentally friendly and sustainable material called dihydrolevogucosenone (also known as Cyrene) can also provide higher concentrations and conductivity of graphene ink.

"Materials characterization is crucial to be able to ensure performance reproducibility and scale up for commercial applications of graphene and 2D materials. The results of this collaboration between the University of Manchester and NPL is mutually beneficial, as well as providing measurement training for Ph.D. students in a metrology institute environment," said Professor Ling Hao, an NPL scientist.

(Source: NPOL)

Andy Johnson

EDA: A Great Field for Young People

Feature Interview
I-CONNECT007

At the recent AltiumLive event in Munich, Germany, Editor Andy Shaughnessy sat down for an interview with Altium's Andy Johnson. Andy is just a few years out of college, and he shared his thoughts on working in this industry and what we can do to draw more young people into the PCB design community.

Andy Shaughnessy: Andy, you're of the newer hires at Altium. Can you tell us a little bit about yourself, how you ended up working at Altium, and how you got into marketing?

Andy Johnson: Sure. Thanks for having me. Technically, I'm a content writer, which is how I got my foot in the door at Altium. But that's now a very small aspect of what I do here. I'm certainly new to the organization and the industry. I do come from the technology side of things, but more in the software sector, so the hardware industry is new to me. In my previous role, I was involved in everything under the sun in terms of marketing, which is really where my experience came from; I managed operations, and then took on events as well and really a lot of marketing in general. It was a natural transition into Altium's marketing department, but a brand-new industry to me, which has been just amazing so far. It has been about six months, but I'm constantly learning things, participating in events like this, meeting customers, and getting to experience a world that I had not been involved in before.

Shaughnessy: How old are you?

Johnson: I'm 25.

Shaughnessy: Well, we certainly need more 25-year-olds in the industry.

Johnson: I'm trying to start a wave here.

Shaughnessy: So, what made you get into marketing originally?

Johnson: That goes way back to college. Well, I guess I shouldn't say "way" back. My fa-

ther was always involved in global marketing, which is what initially piqued my interest in it. Freshman year, I took a couple of introductory classes and stuck with it. I studied marketing the whole way through and got my first job out of college at a very small company—a hole-in-the-wall marketing agency in San Diego. You could fit maybe four or five people in the office, and there was no A/C, so it was burning hot, but it allowed me to move from Wisconsin to California, so I wasn't complaining.

That was really my first experience with marketing, and I loved it. After that, I transitioned into a bigger company still in the startup world. I always stayed in the smaller startup world at first and found that I could move around and get a lot of different experience within one organization very quickly. That helped me tremendously when I was building my career early on and led me to opportunities like Altium where it's a much bigger, global organization, and I get to apply a lot of the things I learned in that startup space on a larger level.

Shaughnessy: When you were in college, was there any type of program to bring in people from industry to let students get exposed to them?

Johnson: Yes, in my marketing classes. It was more class-specific. It depends on the professors. But with two of my marketing professors, their curriculum heavily involved working outside of the classroom, finding businesses, connecting with them, and interviewing people who worked there. In certain situations, it involved bringing those individuals in to talk to the class on relevant topics, or to host a mock negotiation with students in front of the lecture hall. This wasn't a course that was just paperwork; it was more about the mental side of it, going out and actually learning about and participating in the industry early on.

Shaughnessy: What college was this?

Johnson: University of Wisconsin, Milwaukee. I'm from Wisconsin and moved to San Diego about four years ago.

Shaughnessy: You don't talk funny at all.

Johnson: A couple of my words have the Wisconsin accent, but I hide it pretty well. When I talk about the Packers or cheese, it comes out.

Shaughnessy: Among the people that are your age, are they aware of the EDA industry or PCBs?

Johnson: I'm sure they are, but it's minimal, and I think that that's my answer to the question of bringing youth into the industry—a lot of people just don't know enough about this industry. Unless an outside factor has influenced them like a relative or a friend that works in the industry, they're not going to know much if anything about it. Going into college, I never had any experience in the hardware industry—EDA or PCB design. I didn't know about any of it.

When I started at Altium, I was exposed to a whole new world, but I think it goes back to not being exposed to any introductory information in college. It was more about general tools like business, finance, etc. With computer sciences, such as software, applications, and definitely the hardware side, I didn't have a chance to see any of those avenues early on, really due to lack of exposure.

Shaughnessy: As a young person who's now part of this industry and doing well, do you have any suggestions about ways that we can draw more young people into the industry?

Johnson: It's about connecting with them as soon as you can, and I don't even mean just college. I think it should even be sooner than college. This should be in middle schools and high schools. We need to integrate more introductory computer science and application-based classes as early as we can. It's paramount.

Shaughnessy: That's one of the things that IPC has been trying to do for years. They want to inform high school guidance counselors about the industry because they often don't even know about it.

Johnson: Absolutely.

Shaughnessy: That's really where it starts because later in life, other careers just seems hipper, like being a network administrator, web designer, or in IT.

Johnson: Being a programmer does seem flashier nowadays, unfortunately.

Shaughnessy: Or they go to law school.

Johnson: Right, and then they change professions a year into their coursework because they want to get into tech anyway.

Shaughnessy: Is there anything you want to add? What would you tell somebody who's young and considering a career in the PCB world?

Johnson: Go explore now. If I could travel back in time and tell myself to explore something different, I would have gotten way more involved in this industry back in high school and college. Get involved in introductory classes or self-taught knowledge. Start exploring on the internet, and start using the tools that we have. There's so much free information out there; just start chipping away at it a little bit. That knowledge will do nothing but benefit you in your future career. And it doesn't matter where you end up; this information is valuable to everyone. So, start learning as soon as you can.

Shaughnessy: Okay, good. Thanks, Andy.

Johnson: Thank you. **DESIGN007**

Pablo Sanchez Martinez

From Spain to Austria: A Student's Drive to Be a Professor

Feature Interview
I-CONNECT007

The AltiumLive event in Munich drew several hundred PCB designers from around Europe, including engineering students interested in PCB design. Editor Andy Shaughnessy spoke with Pablo Sanchez Martinez, a student at FH Joanneum in Austria who is working toward being a full professor in hardware design. Pablo discussed his studies in engineering and PCB design, the classes he's teaching, and his plans for teaching the young engineers of the future.

Andy Shaughnessy: How are you doing, Pablo?

Pablo Sanchez Martinez: Hello, Andy. I'm great. Nice to meet you.

Shaughnessy: Nice to meet you too. I understand that you're a university student. Can you tell us a little about yourself and what brings you to AltiumLive in Munich?

Martinez: I'm originally from Spain but have been working for three years at FH Joanneum, a university located in Austria. I normally do power electronics. I work in a research center at the university, and we have a lot of products based on power electronics, power converters, transistors, etc.

Shaughnessy: Are you working toward being a full-time professor?

Martinez: Yes, until now, I've been mainly teaching and doing research, but I would like to become a professor. I am also striving towards getting a Ph.D. and I am using all of this experience that I am gaining through the years with research projects to prepare for my teaching courses.

Shaughnessy: And what is your original degree?

Martinez: I studied industrial engineering in Spain, and I did my master's thesis at FH Joanneum, which is how I got to know the research center.

Shaughnessy: And I'm guessing you use Altium tools?

Martinez: Yes. Altium is very important to us. We do high-powered converters. For example, we need to control the clearance and printed requirements in these power devices. In Altium, having a set of design rules with everything you can control is very useful for us. My whole work is based on Altium.

Shaughnessy: What classes are you teaching at the university?

Martinez: I'm teaching practical courses on Altium, actually. I'm teaching how to design PCB circuits as well as some courses and labs on power electronics, including semiconductor packaging.

Shaughnessy: How did you get into wanting to do this as a career? Were you just always interested in electronics when you were growing up?

Martinez: My studies in industrial engineering were very general, and I got the feeling that everything had already been invented there. But in electronics, everything is new and developing very fast. We have high-speed design and high-powered design, and I feel like there's so much I can do in this field. That's also how I got to know Altium. Altium's also developing very fast, and it's the perfect tool for doing what I like, which is hardware design.

Shaughnessy: Do you have any advice for young people who are just getting into PCB design or design engineering?

Martinez: For young designers, I would recommend reading all of the line documentation for Altium or whatever tool you use. You get the feeling for how this works and what you can do with the tool. That's what I did. It's a very good way to start. And go to conferences like this.

Shaughnessy: Do you attend a lot of events like this?

Martinez: This is the second year I've come to this conference in Munich, and I have the feeling that there is nothing similar to it in Europe. At this show, you can get to know designers with the same interests and experiences, and you can share what you have learned. You can get to know different points of view on hardware design.

Shaughnessy: Thanks for speaking with me, Pablo. Good luck with your studies. We need more professors who can teach PCB design.

Martinez: Thank you, Andy. Maybe I will see you again. **DESIGN007**

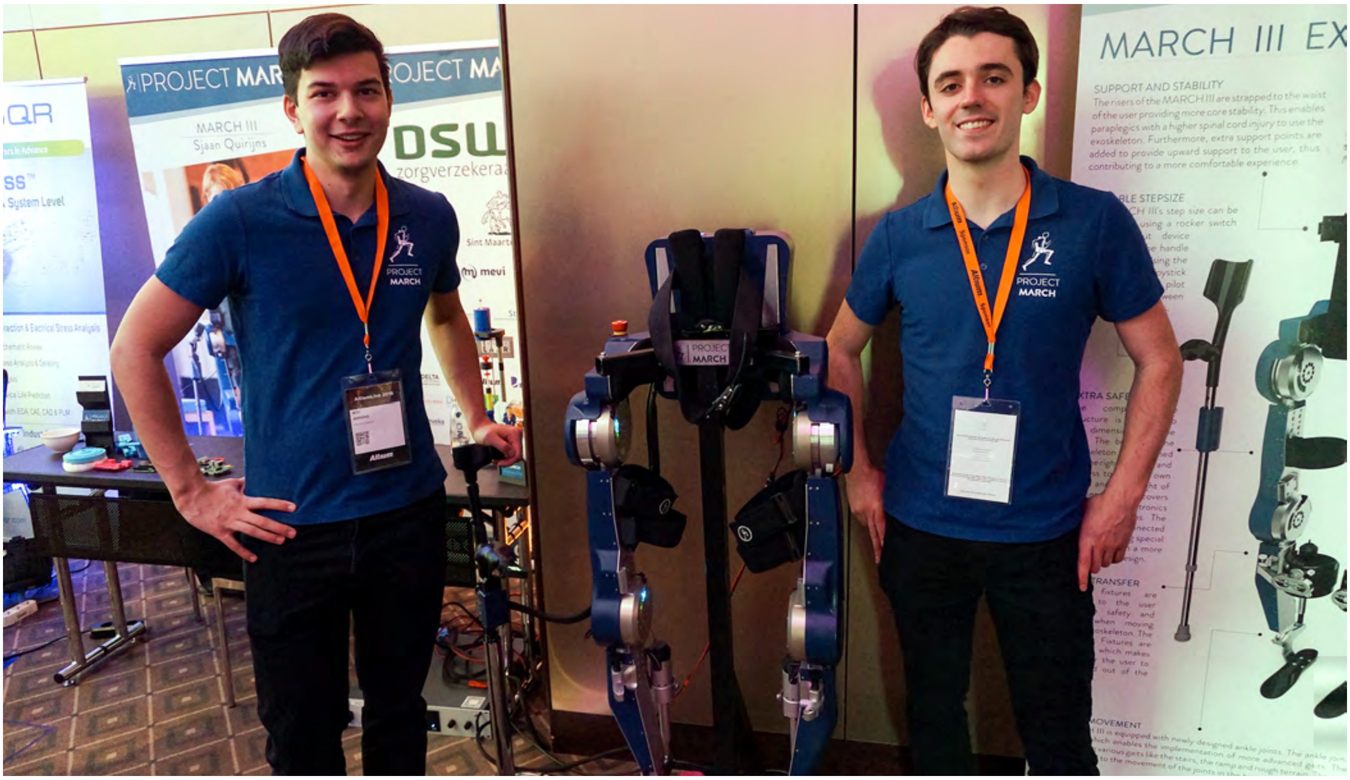
Researchers Make Breakthrough in Understanding Conductivity in Doped Organic Semiconductors

Organic semiconductors enable the fabrication of large-scale printed and mechanically flexible electronic applications. To break into further market segments, improvements in performance are still needed. Doping is the answer.

Researchers from the Dresden Integrated Center for Applied Physics and Photonic Materials (IAPP) and the Center for Advancing Electronics Dresden (cfaed) at TU Dresden in cooperation with Stanford University and the Institute for Molecular Science in Okazaki have identified key parameters that influence electrical conductivity in doped organic conductors.

The combination of experimental investigations and simulations has revealed that introducing dopant molecules into organic semiconductors creates complexes of two oppositely charged molecules.

(Source: Technische Universität Dresden)



Project MARCH Students' Exoskeleton Helps Paraplegics Walk Again

Feature Interview by Andy Shaughnessy
I-CONNECT007

When I was in college, I didn't think much about helping make the world a better place; I just focused on making it to graduation. But some students from Delft University of Technology have found a way to do both. Project MARCH is made up of student volunteers who design and build futuristic exoskeletons that can help paraplegics walk. The students do most of the work themselves, including designing the various PCBs.

I saw their latest exoskeleton up close when Project MARCH exhibited at AltiumLive in Munich, and I had to find out more about this program. Delft Students Martijn van der Marel and Roy Arriens sat down with me to discuss their work on the exoskeleton, including their PCB design experience, and whether they plan to pursue PCB design as a career.

Andy Shaughnessy: Can you tell us a little bit about Project MARCH and this exoskeleton?

Martijn van der Marel: We are a student team from Delft University of Technology, one of the biggest technological universities in the Netherlands. We volunteered for a year to build this exoskeleton. With the exoskeleton, we help paraplegics walk again. Each year, we participate in a competition against other exoskeletons to improve the technology and see how far we've come.

Shaughnessy: The exoskeleton is pretty impressive. Are you all engineering students?

van der Marel: All students have a technical background, but within the team, some of the students are in management and PR, but most of the students are in engineering roles.

Shaughnessy: How did this idea come about?

van der Marel: It started four years ago. Some other group initialized this exoskeleton project, and there were enough enthusiastic students who wanted to dedicate a year to building this exoskeleton. After that year, they continued their studies and recruited a new team of students to make an improved version. That has continued to this year.

Shaughnessy: Did you have to do your own fundraising to find the money necessary for this?

van der Marel: We find sponsors each year to do our own funding because we are self-dependent. We have dedicated people who attend events, find partners, and raise enough money to build an exoskeleton each year.

Shaughnessy: Approximately how much does it cost to build one of those?

van der Marel: It's kind of hard to estimate, but I can tell you that commercial exoskeletons cost approximately €80,000 or US$90,000. For us, we do not have the intention of selling our exoskeleton at this moment, so it's not really possible to put a price on it. Right now, we're focused on the development of the technology. We hope that in the future, the exoskeleton will be accessible for everyone who could benefit from it. After the competition, we retire them and use them for tests for the next year.

Shaughnessy: Roy, can you tell me about the circuit boards and the technology that goes into this exoskeleton?

Roy Arriens: We have multiple circuit boards in the exoskeleton that revolve around power management, data collection, and controlling the instrument itself. We primarily have three types of boards in our exoskeletons. One is the power distribution board, which distributes all the power from our main battery to all the different kind of types of joints.

Next, we have the data collection architecture, which is EtherCAT. We use EtherCAT to collect all of the data from the motor controllers, power distribution boards, and other types of sensors in the skeleton. And we have motor controllers, which are iMOTIONCubes, and we make our own PCBs for that, so it's a motherboard for this motor controller. We connect it with EtherCAT to different kind of boards called general EtherCAT slaves, which we use to interface with, for example, temperature sensors. We designed the power distribution board ourselves. For the slaves, we buy the motor controllers and design the motherboard in-house.

Shaughnessy: And you use Altium tools?

Arriens: Yes, primarily for the power distribution board. We design the whole PCB in Altium from schematics to layout.

Shaughnessy: And did you have access to Altium in the school? Is that how you were exposed to it?

Martijn van der Marel and Roy Arriens.

Arriens: No, we were exposed to it in the team. When we started the team, we didn't have any experience with Altium. We learned from our predecessors, who were already experienced in Altium, and we also have a consultant in Sintecs who helps us by reviewing our layout. We're not experts in layout; we only have about half a year of experience with layout. We have a lot to learn, so we get help from experienced experts in doing it.

Shaughnessy: Do either one of you want to be a circuit board designer? What are your plans after college?

Arriens: At the moment, I'm doing a master's degree in computer engineering, so I'm designing processors, but now that I'm in touch more with circuit board design, it is really interesting. But how can I do this in my career program? I'm not sure if what I find most interesting about it all because there are a lot of these aspects around circuit designing.

Shaughnessy: What's your major, Martijn?

van der Marel: Next year, I'm going to major in signals and systems, which is also next to an electrical engineering major, so I will definitely try to be involved with certain PCB designs in the future. It is a great experience.

Shaughnessy: It must be very satisfying to design this exoskeleton and give it to someone who's paralyzed and see them walk.

van der Marel: It's amazing, especially when someone who's paralyzed stands up for the first time. To finally see all the effort that you've put into the whole year finally pay off when someone walks again is an indescribable feeling.

Shaughnessy: How does it keep someone from falling over? Is there a gyroscope?

Arriens: At the moment, the person walking in the exoskeleton still needs crutches to keep their balance, which is a disadvantage because it's useful to be able to use your hands. We're still working on making the exoskeleton self-balancing. That would be a huge improvement, but it's very difficult.

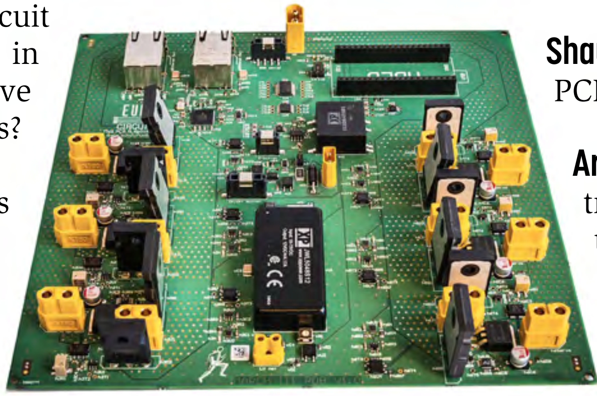
van der Marel: Right now, we have a full department working on trying to figure out how we can do the balancing statically or actively.

Shaughnessy: We see robots that are self-balancing, but that's talking about millions of dollars each, I imagine.

Arriens: And what's really difficult with exoskeletons is that you have two different systems. You have the user, which is the human, and the exoskeleton apart from that, so it's more than just a robot; it's a robot plus a human, and you must learn how to interface those things. It's different from the self-balancing Atlas from Boston Dynamics, which is one system; now, you have two systems that sometimes work together and often contradict each other.

Shaughnessy: Is there circuit board design being taught in your college? Do they have circuit board design classes?

van der Marel: No, besides the basic circuitry theory, we get very little circuit board design in college. I think that's missing.



Shaughnessy: PCB design education is hard to find. We see EE grads that get into the industry and don't even know basic PCB design.

van der Marel: It's one of the main reasons most of us joined this team because we have studied for four to five years, and we didn't get any practical applications to work on. With this team, we finally can put our studies to practical use and learn about stuff that we will find in the industry.

Shaughnessy: Have you done other trade shows with this exoskeleton?

Arriens: A couple. AltiumLive is a big event for us.

Shaughnessy: Is there anything else you want to add?

Arriens: We're still looking for sponsors, especially consultants who could help the electrical departments. We don't have a lot of experience, and we design all the technology ourselves, so we need experts to review them because sometimes there are still mistakes or practical uses or methods we don't fully understand.

van der Marel: One of the main issues is that we work in short terms of one year. We build an exoskeleton in one year, and because there are so many improvements in every version, we need to redesign every PCB every year, which makes it difficult to get it right in one go.

Shaughnessy: And how many PCBs are in each one?

Arriens: For each motor controller, we have a PCB, so there are currently six. There's one big board for the power. Then, there are six general slaves; that's 13 or 14 boards at once.

van der Marel: In the next version, that will probably increase. It's a challenge to fit it all in.

Shaughnessy: It sounds like a lot of fun, and a challenge too.

van der Marel: There are a lot of challenges. It's amazing that you can work with such a big product with 23 people for a whole year; it's a pretty good time. We're all very passionate students. We do this voluntarily for a full year without getting paid at all, but that doesn't bother us because the project is very inspiring and motivating.

Shaughnessy: Wow. And if someone wants to donate, where should they go?

Arriens: We have a website where you can donate yourself for Project MARCH. If you're passionate about the project and interested in our work, you can donate there.

Shaughnessy: Thanks, guys. Good luck with the competition.

van der Marel: Thank you for your interest, Andy.

Arriens: Thank you. DESIGN007

PCB007 Highlights

It's Only Common Sense: Eight Great Tips to Teach a New Salesperson—Listen!

Today, all of us are working diligently on getting more young people involved in our industry, and many of those people are going to be involved in sales. To help you with developing that training, here are eight great tips to teach a new salesperson.

Punching Out! Beware of Cultural Issues in M&E Deals

Company culture is hard to define and manage, but it is a critical factor in making an M&A deal successful. It is also often ignored or misunderstood during and after due diligence because culture is a "soft" science instead of a "hard" subject like finances, legal contracts, IP, or accounts receivable, among other things, which makes culture a difficult factor to deal with.

Standard of Excellence: Seven Things Your PCB Vendor Can Teach You

One of the great things about having a good PCB vendor is that they can teach you and keep you informed about when it comes to their technologies, products, and business. Just as you and your company are experts in your technology, they are experts in theirs.

RTW IPC APEX EXPO 2019: Super PCB Discusses High-end PCB Procurement Services

Kelly Dack speaks with Jessica Zhang, program manager for Super PCB, who focuses on serving customers with a small-business-type relationship. Based in Plano, Texas, Super PCB matches customer design requirements to a myriad of supplier capabilities through time-saving, direct communication with the supplier and best pricing.

North American PCB Sales Ends 2018 With 8.7% Growth

Total North American PCB shipments in December 2018 were up 7.7% compared to the same month last year. Shipment growth ended the year at 8.7%. Compared to the preceding month, December shipments increased by 17.1%.

Unimicron Raises Capex Budget for 2019

PCB and IC substrate supplier Unimicron Technology decided to increase its capex budget this year by NT$2.32 billion ($75.5 million) to total NT$8.3 billion, according to a resolution passed by the company's board of directors, Digitimes reports.

IPC Validation Services Introduces New QML Program

IPC Validation Services introduced a new qualified manufacturers listing (QML) program—the IPC-1791, trusted electronic designer, fabricator, and assembler requirements QML—to address gaps in current electronics industry trusted supplier accreditation programs.

NCAB Expanding in Malaysia

NCAB Group starts the first quarter of 2019 by establishing operations in Malaysia.

Flexible PCB Makers Plagued by Lackluster iPhone Sales

Among supply chain partners for iPhones, Taiwan's PCB makers have suffered significantly from the devices' latest wave of lackluster sales with flexible board suppliers Zhen Ding Tech, Flexium Interconnect, and Career Technology bracing for more negative impacts in 2019 as their revenues have already seen notable downturns since November 2018.

The Proximity Effect

Beyond Design

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

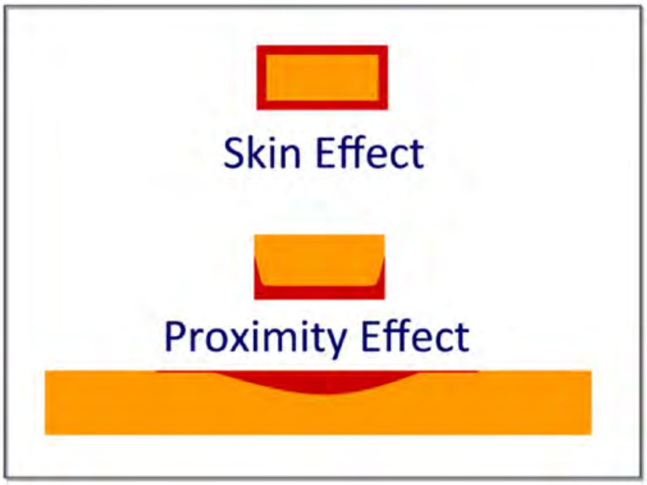


Figure 1: Skin vs. proximity effect.

Skin effect and the proximity effect are manifestations of the same principle—magnetic lines of flux cannot penetrate a good conductor. The difference between them is that skin effect is a reaction to the magnetic fields generated by current flowing within a conductor (Figure 1), while proximity effect is generated by current flowing in other nearby traces or planes. The frequency at which both effects begin to occur is the same. In this month's column, I will focus on the proximity effect. Please see "Beyond Design: Effects of Surface Roughness on High-speed PCBs" for further information on skin effect.

The Proximity Effect

In multilayer PCBs, these effects start to take hold at rather low frequencies on the order of ~30 MHz. Below that frequency, due to changing currents in the traces, the magnetic forces are too small to influence the pattern of current flow. In a low-frequency or DC circuit, the return current takes the path of least resistance filling the entire cross-sectional area of the trace. As it returns to the source via the power/ground planes, this current tends to spread throughout the wide, flat sheet of copper. However, as the frequency increases, the magnetic forces surrounding a trace become significant and the return current takes the path of least inductance. This high-frequency distribution follows a tight path directly above and/or below the trace in the reference plane(s).

As represented in Figure 1, magnetic fields distribute current to a shallow depth around the perimeter of the trace (red), increasing the apparent resistance of the trace; this is the skin effect. The magnetic fields also distribute current around the perimeter of the trace in a non-uniform manner when referenced to a plane; this is the proximity effect. This draws current toward the side of the trace facing the reference plane and forms the return current into a narrow band directly above and/or below the trace. Figure 2 shows microstrip return current density. In an asymmetric stripline configuration (Figure 3), the proximity effect draws current in an uneven distribution towards the near and far reference planes.

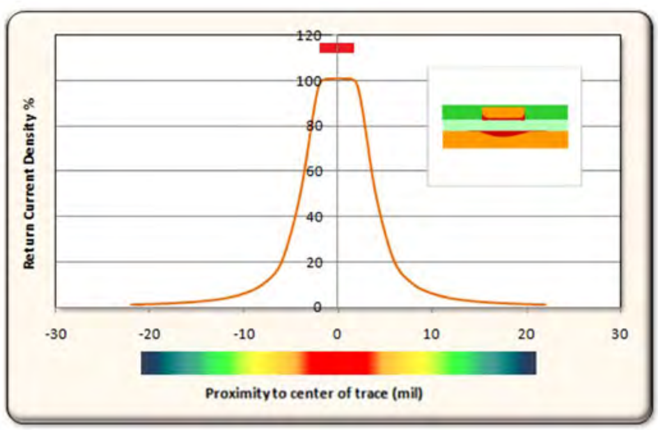


Figure 2: Microstrip return current density.

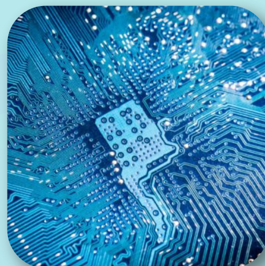
Candor
Circuit Boards

Innovative solutions for complex circuit board manufacturing

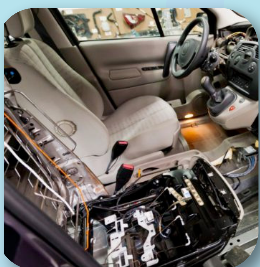
Click to see what we bring to each sector of the industry:

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

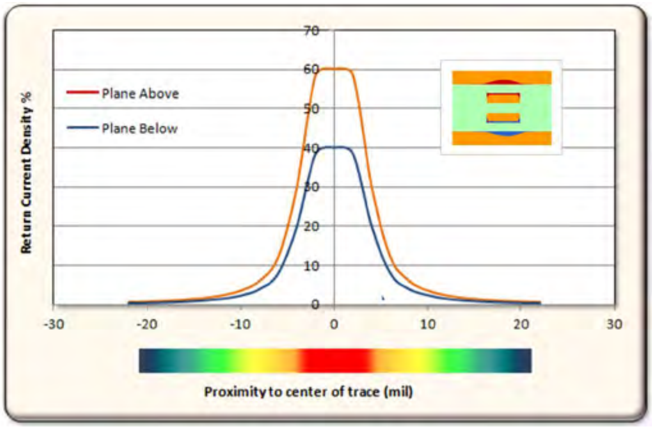


Figure 3: Return path current density for dual asymmetric stripline.

It is important to have a clearly defined return current path and to know exactly where the return current will flow. This is particularly critical with asymmetric stripline configurations where one or two signal layers are sandwiched non-uniformly between two planes. The question is not which plane does the return current flow on but rather how is the current distributed on each plane? Also, if a return path discontinuity (RPD) exists, then the current tends to divert increasing the loop area, inductance, and delay.

A via that provides the connection between signal traces referenced to planes of different DC potential creates RPDs. In other words, the return current has to jump between the planes to close the current loop, which increases the inductance and affects the signal quality. This return current can also excite the parallel plate resonance mode, causing significant electromagnetic radiation from the fringing fields.

If the reference planes are at the same DC potential, then they can be directly connected by stitching vias near the signal via transition to provide shorter paths for the return current. However, if the planes are at different DC potential, then decoupling capacitors must be connected across the planes at these points. Unfortunately, this can pass AC noise between power supplies. Two decoupling capacitors configured as the right example in Figure 4 is a much better solution because it eliminates the transfer of power supply noise from one supply to another. Although this does add a little loop area, it also provides additional decoupling to the planes, reducing power distribution network impedance. In addition, some of the return current flows through the interplane capacitance to close the loop.

Equations

The distribution of current for the three basic configurations depicted in Figure 5 is:

(a) The distribution of current *J(D)* on a solid microstrip plane is given by:

$$J(D) = \frac{1}{1 + (d/h)^2}$$

h = height of trace above/below the plane (mil)*
d = horizontal distance away from the center of trace (mil)*

* *um or mm can be substituted for mil providing the same units are used throughout*

Figure 4: Eliminating the transfer of noise in the return path of split power planes (right).

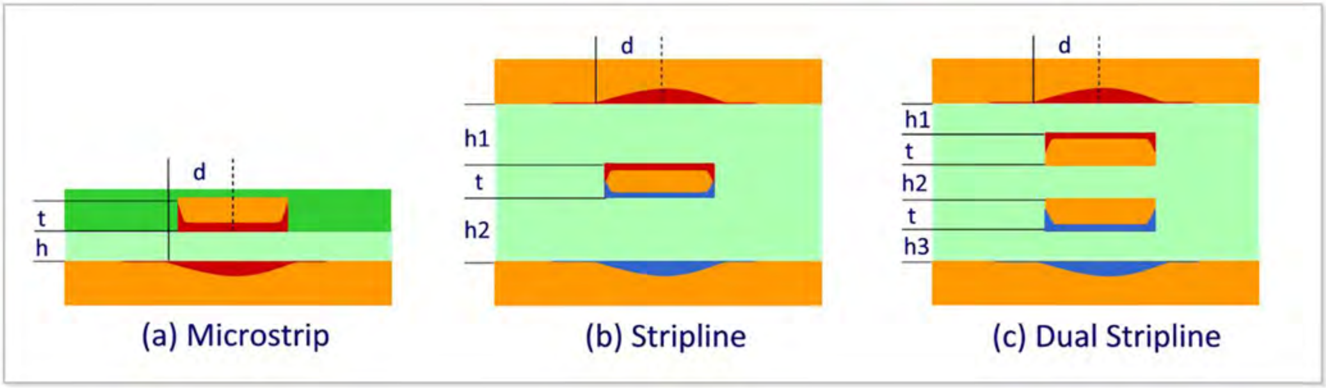


Figure 5: (a) Microstrip, (b) stripline, and (c) dual stripline configurations.

(b) However, for the stripline configuration, one needs to also take into account the ratio of *h1* (height of the plane above the trace) compared to *h2* (height of the plane below the trace). Then, h for the plane above becomes:

$$h = h1\,.\,(1 - \frac{h1}{h1 + h2})$$

And for the plane below, *h* is:

$$h = h1\,.\,(\frac{h1}{h1 + h2})$$

(c) You can easily extrapolate these equations to accommodate dual asymmetric stripline by adding the height of the appropriate dielectrics to each plane; *(h1 + h2)* becomes *(h1 + h2 + h3)* in the previous equations. This will result in a similar current distribution to that shown in Figure 3.

Also, to be more accurate, in the stripline configurations, the trace thickness *(t)* sinks into the prepreg material, bringing the trace closer to the plane and reducing the trace impedance. So, accounting for this resin flow adds a little more complexity to the equation. However, given that you know which dielectric material is core and which is prepreg, the height of the prepreg can be reduced by *(t)*.

The fundamental distribution of the current equation is also the basis for simple crosstalk estimates. Crosstalk changes very rapidly with distance and plummets roughly quadratically with increased separation *(d)* or decreased dielectric height *(h)*. For microstrip:

$$Xtalk = \frac{k}{1 + (d/h)^2}$$

Crosstalk is expressed as a ratio of noise voltage to the driving signal amplitude. The constant *(k)* depends on the circuit rise time and the length of the interfering trace segments. This is always less than one.

Surprisingly, these equations are based on Newton's 300-year-old inverse-square law: the force acting between two objects is inversely proportional to the square of the separation distance. It always amazes me how math, physics, electromagnetics, and other disciplines all align with the exception of subatomic quantum theory, which does not seem to comply with any established law of nature.

Conclusion

When modeling a trace above a solid plane, you will find that the current density is greater on the reference plane side of the trace than on the other. The same principle applies for two traces placed in close proximity in parallel segments—the current tends to concentrate on the two facing edge surfaces. The proximity effect is a simple manifestation of the general rule that high-speed current tends to concentrate near its return path.

Key Points:

- In a low-frequency or DC circuit, the return current takes the path of least resistance filling the cross-sectional area of the trace
- As the frequency increases, the magnetic forces surrounding a trace become significant and the return current takes the path of least inductance
- High-frequency return current distribution follows a tight path directly above and/or below the trace in the reference plane(s)
- The skin effect is the tendency for magnetic fields to distribute current to a shallow depth around the perimeter of the trace
- The proximity effect is the tendency for magnetic fields to distribute current around the perimeter of the trace in a non-uniform manner when referenced to a plane
- It is important to have a clearly defined return current path and to know exactly where the return current will flow
- Return path discontinuities tend to divert current increasing the loop area, inductance, and delay
- If the reference planes are at the same DC potential, they can then be directly connected by stitching vias near the signal via transition to provide shorter paths for the return currents
- If the reference planes are at different DC potential, then decoupling capacitors must be connected across the planes at these points to provide a return path
- Two decoupling capacitors spanning split power planes is a better solution as this eliminates the transfer of power supply noise from one supply to another
- The distribution of the current equation provides insight into where the return path current flows
- The fundamental distribution of the current equation is also the basis for simple crosstalk estimates

Further Reading

- Olney, B. "Beyond Design: The Dark Side–Return of the Signal," *The PCB Design Magazine*, May 2017.
- Olney, B. "Beyond Design: Effects of Surface Roughness on High-speed PCBs," *The PCB Design Magazine*, February 2015.
- Olney, B. "Beyond Design: Mythbusting–There Are No One-way Trips!" *The PCB Design Magazine*, April 2014.
- Olney, B. "Beyond Design: Plane Crazy, Part 1," *The PCB Design Magazine*, December 2015.
- Olney, B. "Beyond Design: Controlling the Beast," *The PCB Magazine*, December 2011.
- Olney, B. "Beyond Design: Return Path Discontinuities," *The PCB Design Magazine*, April 2017.
- Johnson, H., & Graham, M. *High-speed Signal Propagation: Advanced Black Magic*, Prentice Hall, 2003.

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.

Tiny Silicon Nanoparticles Cement New Era for Ultra-high Capacity Batteries

Scientists believe that silicon could be the answer to your battery woes with the potential for a charge capacity 10 times larger than current lithium-ion batteries. Now, University of Alberta chemists have published research that studies the effect of nanostructuring the silicon within lithium-ion batteries to understand the importance of size.

In their research, the researchers examined silicon nanoparticles of four different sizes within highly conductive graphene aerogels. The results show that the smaller the particle, the less likely it is to crack or fracture upon lithiation

The next steps are to develop technology for creating silicon nanoparticles in a faster and less expensive way, making these tools more accessible for industry and technology developers.

The paper was published in *Chemistry of Materials*. (Source: University of Alberta)

Designer Highlights From IPC APEX EXPO 2019

The Digital Layout

by Stephen Chavez, CIT, CID/CID+, IPC DESIGNERS COUNCIL

This month's column brings you designer highlights from IPC APEX EXPO 2019 in San Diego, California, and a brief report out from the IPC Designers Council Executive Board meeting that occurred on the opening day of the show floor.

This year's IPC APEX EXPO was another great show with 5,000+ attendees from 56 countries and 440 exhibitors. Overall, there were 9,796 visitors, attendees, and exhibitor personnel. Similar to past shows, the IPC subcommittees were filled with lots of collaborations, passionate discussions, debates, and positive synergies among talented engineers and subject matter engineers (SMEs), helping to improve our industry standards and make a difference. These are the people who volunteer their time and mold our industry.

A good example of one of these engineers and SMEs is Leo Lambert. Leo has been involved with IPC for many, many years in numerous subcommittees. He is the VP and Technical Director at EPTAC Corporation and became the 2019 IPC Hall of Fame inductee for all of his efforts and contributions to the industry (Figure 1). I know Leo personally and feel very blessed to call him a colleague and true friend. If you have not had the opportunity to engage with Leo or meet him at one of our many industry events, he is one sharp individual with tons of industry experience and knowledge that he wants to share with others. Congratulations, Leo!

Figure 1: John Mitchell, IPC president and CEO, presenting Leo Lambert the 2019 IPC Hall of Fame Inductee Award.

IPC DC Executive Board

The IPC DC Executive Board meeting took place on January 29. Executive Board members in attendance included Gary Ferrari, Mike Creeden, Cherie Litson, Kelly Dack, Thomas Romont, and me. We also had Michael Schleicher, a guest representative from Germany, in attendance. Further, Dr. John Mitchell, Dave Hernandez, Colette Buscemi, Teresa Rowe, Aaron Birney, and Charlene Gunter du Plessis represented IPC. After several exhausted attempts to no avail, we were unsuccessful in getting a video conference call pulled up for executive members calling into the meeting due to their unavailability to travel to attend in person, such as Rick Hartley, Susy Webb, Reiner Thueringer, and Soo Lan Cheah.

The meeting started with a lengthy discussion of the newly introduced IPC course, "Introduction to Printed Circuit Board (PCB) Design" made up of the following two sections:

1. IPC PCB Fundamentals, Schematic Capture—PCB Design 1
2. IPC PCB Fundamentals, PCB Layout—PCB Design 2

For more information on this course, visit IPC's website [1]. Another topic of discussion was the newly announced IPC Education Foundation that was unveiled at this year's IPC APEX EXPO.

Finally, it was agreed on that an initial kick-off meeting between the IPC DC Executive Board and the new IPC Education Foundation staff will be held in early March as we explore opportunities for collaboration around the IPC Student Chapter model and the broader STEM and education programs and initiatives that are currently being developed. Stay tuned for more information to follow.

IPC CID Certification Class

Cherie Litson, MIT, was the instructor who led the diverse IPC CID certification class made up of approximately 15 students at IPC APEX EXPO 2019. The class had both male and female students from different backgrounds and with varying experience who traveled both domestically and internationally to achieve their IPC CID certification. Cherie commented, "There was lots of energy and engagement within this group. After three days of lecture and an exam held on the fourth day, all student successfully achieved their IPC CID certification." Congrats to all of these students!

Real Time with... IPC APEX EXPO 2019 Show & Tell Magazine

After the show wrapped up, there were lots of activities to cover from IPC APEX EXPO 2019, including hot topics such as the successful IPC STEM Student Outreach program and more. I-Connect007 recently published out its second annual *Real Time with... IPC APEX EXPO 2019 Show & Tell Magazine*. This special publication is a supplement to their other monthly magazines and brings you exclusive, in-depth coverage of IPC APEX EXPO 2019. The pages are packed with tons of great content, including event photos, video interviews, attendees' thoughts, reviews from guest contributors and I-Connect007 editors, and interviews with award winners and other industry experts. If that's not enough, you will also enjoy our coverage of the Hand Soldering and Rework World Championship, the two executive forums, and all of the excitement around CFX. Click here to download your free copy.

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

- March 19–22: Kirkland, WA
- April 22–25: 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 21–24: Anaheim, CA
- November 2–5: Raleigh, NC
- November 5–8: Dallas, TX

IPC Advanced Certified Interconnect Designer CID+

- April 16–19: Markham, ON
- September 6–9: Santa Clara, CA

- September 10–13: Kirkland, WA
- September 19–22: Schaumburg, IL
- October 21–24: Anaheim, CA
- November 2–5: Raleigh, NC
- December 3–6: 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.

PCB Design Events

PCB2Day

- Controlling noise, EMI, and signal integrity in high-speed circuits and PCBs
- April 17–18: Seattle, WA
- June 13–14: Boston (Chelmsford), MA
- pcb2day.com

Realize LIVE

- June 10–13, 2019: Detroit, MI
- Realize LIVE

PCB West 2019

- September 9–11: Santa Clara, CA
- pcbwest.com

The Future

Next month, we will resume individual chapter spotlights and review feedback from the IPC DC Executive Board.

IPC DC

The IPC Designers Council is an international network of designers. Its mission is to promote printed circuit board design as a profession and to encourage, facilitate, and promote the exchange of information and integration of new design concepts through communications, seminars, workshops, and professional certification through a network of local chapters. DESIGN007

References

1. IPC–Association Connecting Electronics Industries. "Introduction to Printed Circuit Board (PCB) Design."

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.

Laser Pulses Light the Way to Tuning Topological Materials for Spintronics and Quantum Computing

Scientists at the U.S. Department of Energy's Ames Laboratory have discovered a means of controlling the surface conductivity of a 3D topological insulator—a type of material that has potential applications in spintronic devices and quantum computing.

Also, 3D topological insulators are emerging materials that hold great promise due to their unique electron conducting states on their surfaces, which is immune to backscattering, versus the bulk interior, which behaves as a normal insulator. But a challenge remains in underpinning and selectively controlling their high-frequency transport at the surface without an increased scattering from the bulk material.

By employing ultra-short mid-infrared and terahertz pulses of less than one trillionth of a second, researchers at Ames Laboratory were able to successfully isolate and control the surface properties of a bismuth-selenium (Bi2Se3) 3D topological insulator. The method provides what is essentially a new "tuning knob" for controlling the protected surface conductivity in this category of materials.

"We believe that this study could evolve into a benchmark method of characterizing and manipulating these materials, so they can be better understood and adapted for applications in new quantum technologies," said Jigang Wang, Ames Laboratory physicist and Iowa State University professor.

The research is published in *Nature Communications.*
(Source: Ames Laboratory)

MakeHarvard 2019: Bigger and Better!

Connect the Dots

by Bob Tise and Dave Baker, SUNSTONE CIRCUITS

The second annual MakeHarvard engineering makeathon was a huge success (Figure 1). The event brought over 370 top students from around the world to compete for prizes while building engineering prototypes. It was great to see such a diverse representation of different nations, cultures, and languages. The 2019 event was twice as large as last year, making everything seem bigger and better.

The maker movement is essentially a tech-influenced, do-it-yourself community. It was informally created around 15 years ago around Make Magazine—a geeked-out periodical featuring lots of DIY articles for creations, such as VCRs converted into cat feeders and 3D printers. Maker spaces and events have become more and more popular, inspiring startups and manufacturing innovation as well as clogging garages the world over with half-finished robots and potato cannons.

Sunstone Circuits was eager to return as a sponsor and creator of a competition category this year, also serving as both mentors and competition judges. If you were there, you saw us—we were hard to miss in our bright orange vests (Figure 2). As mentors, we were out and about helping students and answering questions.

Figure 1: MakeHarvard lanyards.

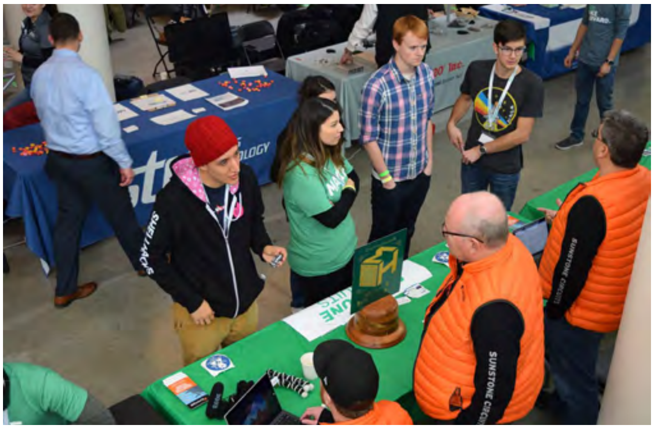


Figure 2: The Sunstone Circuits team talking with students at the event.

The student projects had all kinds of design elements, from software on complex platforms and different programming languages to hardware engineering of all kinds. It was truly amazing to see what these teams could do in a short amount of time. The myriad of skills on display ranged from concept design to advanced coding with real-world platforms, such as AWS. Some projects featured mobile apps and websites. Further, we saw real 3D objects designed, printed, and integrated into projects with documentation that would make most corporate projects envious.

The students were full of questions, and we were eager to help. Sometimes, there were questions we didn't have complete answers for at the ready, but usually, we were able to sug-

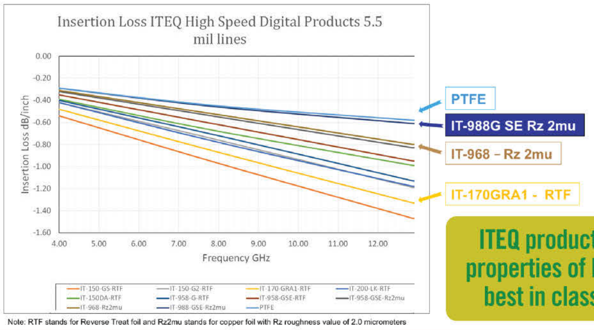
IT-968 / IT-968 SE

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

IT-968
Dk @ 10 GHz - 3.66
Df @ 10 GHz - 0.005

IT-968 SE
Dk @ 10 GHz - 3.16
Df @ 10 GHz - 0.0037

Insertion Loss - Measured



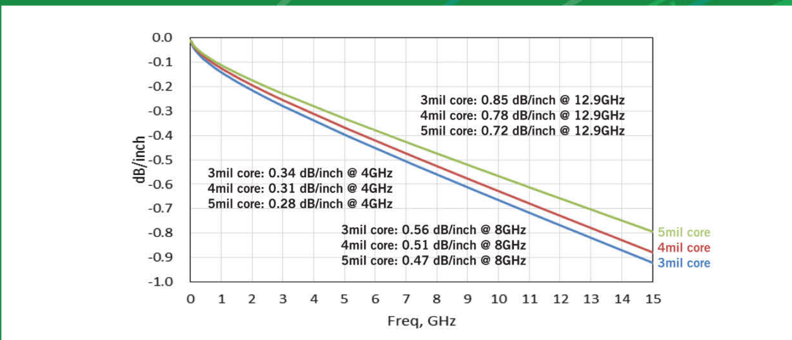
ITEQ products approach the properties of PTFE. ITEQ offers best in class performance.

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

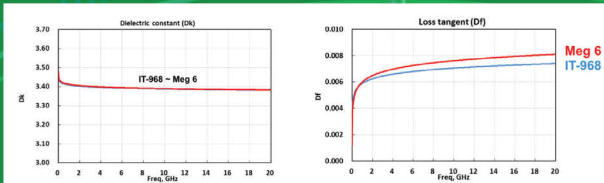
Industry Test Vehicle

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	0.0076	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

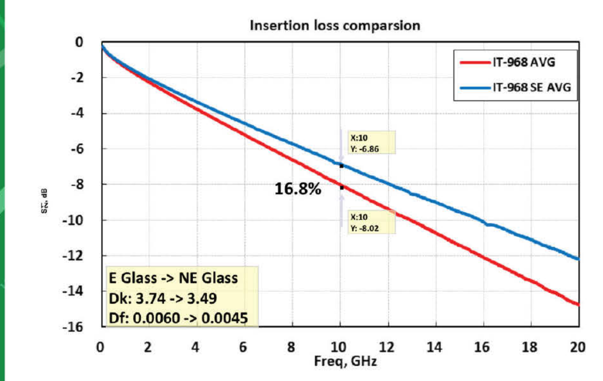
Features
High Tg, low CTE, High Thermal Reliability with Ultra Low Df < .005. Designed for High Speed Applications.

Applications
Server/Storage/Switch (10G/40G/100G), Backplane, Telecom, Base Station

Feature *High Speed , 25Gbs/path Solution*

Property	Test Method	IT-968	IT-968 SE
Tg (℃)	DSC	185	185
T-288 (w/ 1 Oz Cu, min)	TMA	120+	120+
Td-5%(℃)	TGA 5% loss	390+	390+
CTE (%), 50-260℃	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 0.0038-0.0050	0.0028 0.0031-0.0040

E-Glass vs Low Dk Glass



ITEQ

Figure 3: Sunstone's first-place winner, Team 42+2.

Figure 4: Sunstone's second-place winners, Sam Roquitte (top) and Maxwell Stigman (bottom).

gest a place to find the answer. All the students needed was direction, and they were all over finding the solution. Problem solved.

Sunstone sponsored Make It Matter—a competition focused on innovative prototypes that had significant social, personal, or environmental impacts. Teams had 24 hours to come up with an idea and bring their creation from design to prototype.

Judging was tough because there were so many great ideas out there! We scrambled to deliver the judging rubric on time—a task made difficult by the number of amazing prototypes and teams that we wanted to recognize. In the end, the first-place winner for Sunstone's competition was a robot pipe crawler designed by Team 42 + 2 (Figure 3). Their prototype could crawl along pipes and detect cracks or other faults in the material.

There were so many great submissions that at the last minute, we decided to award a second-place prize as well. This prize went to Sam Roquitte and Maxwell Stigman from Georgia Institute of Technology for their eL safety-enhancing adaptive lighting system for electric skateboards (Figure 4).

Here are some other great projects that really impressed us:

- CocoGogo, a hand-cranked coconut shredder
- Hydrophobe unmanned surface vehicle (USV) for filtering and cleaning oil spills
- Beam Print from Team 46 for laser-cutting without friction

By the end of MakeHarvard 2019, the entire Sunstone team was exhausted but overjoyed by the success of the event. We saw numerous prototypes and met so many enthusiastic students. We also gave away all of our "schwag"—Sunstone memory sticks, notebooks, and pens disappeared like smoke in a tornado! Overall, it was just a great event. We are already busy discussing next year's MakeHarvard makeathon and how we can help make it better. **DESIGN007**

An expanded version of this column will appear in the April issue of *PCB007 Magazine*.

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

MilAero007 Highlights

2020 Army Budget Begins 'Dramatic Shift' ▶

A U.S. Army budget drill that identified $30 billion in savings was partly about finding money for future modernization and partly to "do better with every dollar we have," said Undersecretary of the Army Ryan D. McCarthy.

NASA to Advance Unique 3D-printed Sensor Technology ▶

The technology is capable of sensing everything from minute concentrations of gases and vapor to atmospheric pressure and temperature, and then transmitting that data via a wireless antenna all from the same self-contained platform that measures just 2x3" in size.

BAE Systems Updates F-35 Electronic Warfare Systems ▶

BAE Systems has announced a critical program milestone with the successful insertion of new technology into its EW systems for the global fleet of fifth-generation F-35 Lightning II fighter aircraft.

NASA Seeks U.S. Partners to Develop Reusable Systems to Land Astronauts on the Moon ▶

Through multi-phased lunar exploration partnerships, NASA is asking American companies to study the best approach to landing astronauts on the Moon and start the development as quickly as possible with current and future anticipated technologies.

High-powered Fuel Cell Boosts Electric-powered Submersibles and Drones ▶

The transportation industry is one of the largest consumers of energy in the U.S. economy with increasing demand to make it cleaner and more efficient. While more people are using electric cars and designing electric-powered planes, ships, and submarines, it's much harder due to power and energy requirements.

Michael Cole Promoted to President and COO of AirBorn ▶

AirBorn Inc. has promoted Michael Cole to president and COO of the company.

Graphene-based Technology for Wearable Health Monitoring, Food Inspection, and Night Vision ▶

The Graphene Pavilion, organized by the Graphene Flagship and supported by the European Commission and GSMA, is returning to Mobile World Congress (MWC) 2019 with over 20 graphene-based prototypes—four of which are developed by the Graphene Flagship partner ICFO based in Barcelona.

Lockheed Martin, Diehl Defence, and Saab Unveil Collaboration ▶

Lockheed Martin, Diehl Defence, and Saab announced the Falcon air defense weapon system as the short- and medium-range air defense solution for current and emerging threats.

Collins Elbit Vision Systems Marks F-35 Helmet Mounted Display System Delivery Milestone ▶

With 1,000 deliveries, there's no sign of slowing down, according to Collins Elbit Vision Systems (CEVS), which recently celebrated the F-35 Helmet Mounted Display (HMD) System's milestone. CEVS is a joint venture between Collins Aerospace and Elbit Systems of America.

I Think I'll Go for a Walk

Tim's Takeaways
by Tim Haag, CONSULTANT

It has been a long time since I was in school—so long, in fact, that it was still called "junior high school" instead of "middle school" like it is today. Because it has been a while, I don't remember every single detail of what went on during those days. I remember a lot of the teachers, some of the classes, a few of the events, and of course, all of the girls I had crushes on. Other than that, it's mostly a blur.

It's Such a Beautiful Day

However, one thing I do remember was reading the sci-fi short story by Isaac Asimov called *It's Such a Beautiful Day.* In the story, technology has progressed to the point where people use something called a "Door" (capital "D") to travel in a way that was similar to Star Trek's transporter beam. The characters had stopped using regular doors (lowercase "d") to venture outside because that was considered a fate worse than death due to the germs, dirt, rain, mud, etc. You get the picture. Going outside was to be avoided at all costs, which was easy when you could simply step into your Door and be instantaneously transported to another Door. Then, it happened.

One day, the Door in the home of 12-year-old Richard Hanshaw Jr. suffered a malfunction due to a broken "field-modulator brake-valve" (you have to love 1950's sci-fi terminology). This catastrophe forced poor, young Richard to use the regular door to get to school. The problem was that once Richard took that first step outside, he enjoyed what he didn't know he had been missing.

After that, he began using the regular door on a daily basis. Soon, he stopped going to school, came home late, and a couple of times, he even had traces of dirt and mud on his clothes (gasp). Before long, school staff and his mother were seriously concerned that he had suffered some sort of mental breakdown because of these outside journeys. In an attempt to discover the cause of this unusual neurosis, a psychologist reluctantly agreed to go outside with him to investigate.

By the end of the story, the doctor was no longer concerned and explained to Richard's mother that nothing was wrong with him. The doctor came around to Richard's way of thinking, and instead of tak-

ing the Door himself when it was time to leave, he exclaimed, "You know, it's such a beautiful day that I think I'll walk."

The Importance of Work Breaks

When I read that story as a boy, I couldn't believe how dumb it sounded. It seemed to me that even if there was technology that could someday transport you like the "Doors" in the story, no one would ever favor it over the sheer joy of being in the great outdoors. The "Door" would certainly be a great time saver, but in the end, people would always prefer the natural beauty around them instead of being enslaved by technology.

That thought abruptly came back to me the other day when I was walking down the sidewalk and got shoulder slammed by someone whose attention was on their smartphone instead of looking where they were going. It made me realize that the very scenario I had scoffed at in junior high was now being played out in everyday life. This got me to thinking about other similar scenarios as well. How many times have parents taken electronic entertainment away from their kids only to be greeted with screams of anguish exclaiming, "What will I do now? I'm so bored!" As adults, we aren't much better when you consider just how easy it is to immerse ourselves in technology whether for work or entertainment.

Typically, I espouse the virtues of technology, which shouldn't be too surprising in a publication like this; I am a great fan of exploring new ways to leverage technology in our lives. But in an unusual reversal of my regular opinion, I'm suggesting that maybe we shouldn't be afraid to take a step back from all of this technology.

For those of us who design PCBs all day long, getting ourselves caught up in and even lost in all of this technology can be a very easy thing to do. I've known many designers who have worked long hours and not realized how much time had passed. You may remember the story I told in my column two months ago where I went back to work after a swim party to check on something, and without realizing it, ended up working the entire night through. I was extremely surprised to find that night had turned into morning, and people were showing up for work. It was made more embarrassing because I was only wearing a swimsuit and flip-flops, which wasn't exactly acceptable office attire even in Oregon.

I say all of this to myself just as much as I do to anyone else—take a break. You aren't at your best if you are exhausted, and you aren't doing your body any favors either. I recently read an article [1] that reported movement breaks are essential for your health and can also help prevent decision fatigue. Taking a break can also restore motivation, increase productivity and creativity, and help consolidate memories and improve learning. So, take a break; put away the technology and refresh yourselves from time to time.

Taking a break can also restore motivation, increase productivity and creativity, and help consolidate memories and improve learning.

Set an Example

In the story, Richard is a young man who learns the value of stepping outside and taking a break. With this month's issue focusing on the youth of our industry, I hope that they also learn the value of taking a break in addition to all of the other education, training, and practical experience that fledgling PCB designers must acquire. Too many times, we've heard about young people burning out and dropping out of technical programs and careers because they've become overloaded to the breaking point. This doesn't only happen to young people; all of us could learn from young Richard and learn to take better breaks. I also strongly believe that those of you who are responsible for new designers in our industry must insist

Henry taking his human for a stroll in the fall of 2017.

that they take healthy breaks every now and then, even if you have to make it a requirement.

One thing that used to help me to get away from the stress of technology was to go for a walk with my dog regularly. The annoying thing about walking a dog is that for some reason known only to dogs, they want to sniff and paw at the same bush that they sniff and paw at every time you walk them. I know that they sniff each other's waste, so I guess that they're simply checking their "pee-mail" (sorry, I just couldn't resist). It used to drive me crazy though when my dog stopped at every bush, but it forced me to slow down and be patient. Eventually, I would even laugh at his efforts and ask him what the heck he was doing. He never answered me, of course, but with forced moments of relaxation like these, I would feel the stress flow out of my body as I slowed down, walked, and saw the world through his eyes (and nose). Sadly, I no longer have him, but I still go for walks, and I try to take some of those same breaks that we used to take together (but I am never going to sniff those bushes, I'll tell you that).

My boss is also great at recognizing the need for taking breaks. At the end of our online group meetings, she encourages us all to stand, stretch, and hydrate. She understands how those of us who work at home without the normal distractions that occur in an office are especially prone to being glued to our desks.

Many years ago, a different boss at a PCB design service bureau had his own unique way of encouraging us to take a break. He would come through the design bay and call out in his deep baritone voice, "DARTS!" and we would all follow him into the break area for a quick game. In addition to the benefits of taking a break, forcing our eyes to focus in and out as we threw a dart was a great way to relieve us all from the eye strain of older CRT monitors.

Seriously, Take a Break

Please take my word for it. We all need to slow down, take a break, and get our head out of technology for a few moments. We all need breaks to stay healthy and ensure we're at our best. When you do take a break, you'll then be able to come back refreshed and ready for that next critical challenge that will eventually come your way as sure as rain falls in Oregon.

And now, if you will excuse me, it's such a beautiful day that I think I'll go for a walk. **DESIGN007**

References

1. Selig, M. "How Do Work Breaks Help Your Brain? Five Surprising Answers," *Psychology Today*, April 18, 2017.

Tim Haag is a PCB design consultant based in Portland, Oregon. To read past columns or contact Haag, click here.

Important Considerations for Conformal Coating Selection and Performance

Sensible Design

by Phil Kinner, ELECTROLUBE

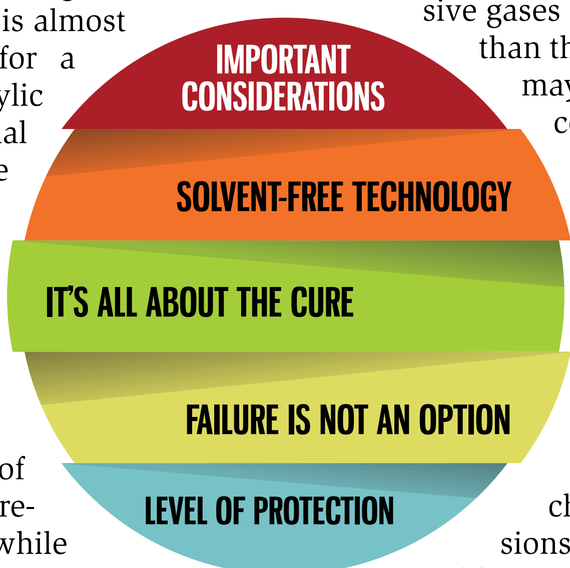
Having covered the subject of conformal coatings in depth over the past few months, now is an appropriate time to review some of the key pointers I have tried to share in my various columns. I present some of my thoughts on the essentials in this five-point guide.

Choosing a Coating: Three Prime Considerations

First, determine the expected operational temperature range for the circuit board—the highs and the lows. Should this be greater than 150–160°C, for example, it is almost certainly an application for a silicone rather than an acrylic or polyurethane conformal coating. Also, consider the temperature excursions. If thermal shock or cycling is not taken into consideration, it could lead to cracking, severely compromising a coating's protective capabilities.

Second, what degree of chemical resistance is required? Acrylic materials, while easily removed for rework, are usually highly susceptible to attack by solvents. Meanwhile, polyurethane materials provide more chemical resistance but are generally not amenable to rework. Assess whether immersion or splash resistance is required and whether the coating may be exposed to heated solutions of potential contaminants, which will increase their ability to act as a solvent.

Third, consider what level of corrosion protection is required. Humidity typically only becomes problematic when condensation occurs, which would require close attention to the thickness and coverage of the coating. But remember, while a thicker coating might provide superior protection in condensing environments or where salt spray or corrosive gases are present, anything deeper than the 50-micron target thickness may be prone to cracking under conditions of thermal shock or cycling.



Why Should I Consider Solvent-free?

Selecting solvent-free technology is a balance of ethics, performance, and process. Ethically, solvent-free materials are a smart choice because solvent emissions will be drastically reduced (if not eliminated), and workforce health will be better protected, ensuring easier compliance with local legislative requirements. Moreover, the energy required for curing these materials is significantly lower

than that needed for solvent-based materials, resulting in reduced energy bills and reduced CO_2 emissions.

From a performance point of view, solvent-free materials can be applied slightly more thickly, improving coverage and protection. Because they're easier to process and more readily compatible with rapid throughput manufacturing operations, solvent-free formulations are often technically superior and can meet the demands of challenging applications in the automotive and aerospace sectors where increased condensation and thermal shock resistance is required.

It's All About the Cure

The cure mechanisms of the main classes of coating materials include drying, oxidative, moisture, heat, chemical, and UV. Your choice will depend on a variety of factors, such as the performance requirements of the application and physical constraints, including the maximum permissible cure temperature and the time allotted for curing.

Acrylic polymers in a solvent can be air dried. Once the solvent has evaporated, the residual coating is physically dry, and no further reaction mechanisms are needed. Heat is often used to speed up solvent evaporation, but care must be taken to avoid solvent-entrapment and bubble formation.

Heat is often used to speed up solvent evaporation, but care must be taken to avoid solvent-entrapment and bubble formation.

Oxidative cure coatings based on solvent-based alkyd chemistry are dried as mentioned in the previous paragraph before undergoing a reaction with atmospheric oxygen, which initiates a cross-linking reaction that further develops the coating's protective properties. Cure times are typically much longer than for physical drying products, often requiring many hours at 80–90°C to develop optimum properties.

Moisture curing coatings are available in both silicone and polyurethane chemistries in solvent-based and solvent-free formulations. The materials absorb water from environmental humidity, which initiates cross-linking. These materials are used widely because they don't require any extra curing processes, although heat can be used to accelerate the reactions if required.

Heat-curing conformal coatings are largely silicone based and require a minimum temperature of 100–110°C for 10–15 minutes to achieve full cure. The main advantage of these materials is that they require no additional time to develop properties and are considered to be virtually 100% reacted, enabling coated boards to be safely bagged without fear of outgassing.

With chemical-cure coatings, such as urethanes and silicones), reactive oligomers are mixed with cross-linking materials immediately before application. Once these two species are mixed together in the correct ratio with a suitable catalyst, a chemical reaction occurs to produce a dry, cured coating. Again, heat can be introduced to increase throughput.

UV curable materials cure extremely rapidly (within seconds) when exposed to UV radiation of a suitable wavelength and intensity. However, the risk of shadowing by tall components means that a secondary cure mechanism, such as by heat or moisture, is often necessary.

When Coating Failure Is Not an Option

When coating failure isn't an option, it is important to consider the two principal failure mechanisms: corrosion and loss of insulation (leading to short circuits). Corrosion is a complicated, diffusion-controlled, electrochemical process that takes place on an exposed metal surface, usually in the presence of water and ionic contaminants. Cleaning before conformal coating will go a long way to removing these two prerequisite conditions for corrosion.

Conformal coatings help prevent the formation of electrolytic solutions by acting as moisture barriers. However, small voids in the coating that expose the PCB's metal surfaces can accelerate corrosion under the right environment. The challenge for a conformal coating is to achieve good coverage and adhesion to the complex, three-dimensional topography of a PCB.

Poorly performing coatings also risk insulation loss as the PCB surfaces when water condenses in combination with ionic impurities to form conductive pathways between PCB tracks. Without a doubt, condensation can severely test the insulation resistance of a coating!

Protecting Against Condensation, Immersion, and Salt Spray

The greatest test of conformal coating performance is posed during power up under wet conditions, whether this is due to condensation, immersion, or salt spray. Liquid water with soluble impurities is electrically conductive, and finding any weak spots in a coating will eventually lead to short circuits at the PCB surface. To provide protection in these circumstances, it is essential to achieve 100% defect-free coverage of the PCB's metal surfaces, which poses a real challenge for both the material itself and the application process.

Fortunately, a new class of conformal coating materials dubbed "2K" (two component) enable a much greater thickness and perfect application coverage to be achieved, resulting in a higher level of protection. Indeed, the performance advantages of 2K materials have been positively demonstrated in three of the harshest tests that these materials can be subjected to, including powered condensation testing and powered immersion testing in salt-water.

If you have any questions or would like more information about choosing and/or applying conformal coatings, there's a wealth of experience to call upon from our technical support team members who will be more than happy to answer your queries and offer expert guidance. **DESIGN007**

Phil Kinner is the global business and technical director of conformal coatings at Electrolube. To read past columns or contact Kinner, click here. Kinner is also the author of *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*. Visit www.i007ebooks.com to download this and other free educational titles.

Large, Stable Pieces of Graphene Produced With a Unique Edge Pattern

Graphene is a promising material for use in nanoelectronics. However, its electronic properties depend greatly on how the edges of the carbon layer are formed. Zigzag patterns are particularly interesting in this respect, but until now, it has been virtually impossible to create edges with a pattern.

A team of researchers from the Friedrich-Alexander-Universität Erlangen-Nürnberg (FAU) led by Dr. Konstantin Amsharov from the Chair of Organic Chemistry II succeeded in developing a straightforward method for synthesizing zigzag nanographene. Their procedure delivers a yield of close to 100% and is suitable for large scale production. They have already produced a technically relevant quantity in the laboratory.

First, the FAU researchers produced preliminary molecules, which they then fit together in a honeycomb formation over several cycles—a process known as cyclisation. In the end, graphene fragments are produced from staggered rows of honeycombs or four-limbed stars surrounding a central point of four graphene honeycombs with the sought-after zigzag pattern to their edges.

This approach allows scientists to produce large pieces of graphene while maintaining control over their shape and periphery. This breakthrough in graphene research means that scientists should soon be able to produce and research a variety of interesting nanographene structures.

(Source: Friedrich-Alexander-Universität Erlangen-Nürnberg)

Editor Picks from PCBDesign007

1 I-Connect007 Publishes Second Annual Show & Tell Magazine—All About IPC APEX EXPO 2019 ▸

This special publication is a supplement to our other monthly magazines and brings you exclusive, in-depth coverage of the recent event. These pages are packed with tons of great content.

2 Beyond Design: 10 Fundamental Rules of High-speed PCB Design, Part 5 ▸

The final part of the 10 fundamental rules of high-speed PCB design focuses on board-level simulation encompassing signal integrity, crosstalk, and electromagnetic compliancy. Typically, a high-speed digital design takes three iterations to develop a working product.

3 Joe Clark Discusses DownStream's Updated Tool Lineup and IPC-2581 ▸


Joseph Clark

DownStream Technologies recently revamped their entire product line, from CAM350 through BluePrint-PCB. DownStream co-founder Joseph Clark and Guest Editor Kelly Dack discuss some of these updates.

4 The Electronic Component Shortage Crisis: A Veteran Engineer's Perspective ▸


John Watson

From where we stand now, at the beginning of 2019, we see lead times for some components in the short range of up to 16 weeks; medium-to-high is 32 weeks, and long lead times are as far out as 80 weeks. In other words, if we ordered a component today, it would arrive in over a year a half from now (maybe).

5 Natasha Baker: Supply Chain Transparancy Inside the CAD Tool ▶

Natasha Baker, CEO and founder of SnapEDA, an online parts library, discusses the benefits of transparency in online libraries to designers, and discusses strategies on how to solve supply chain challenges, and more.


Natasha Baker

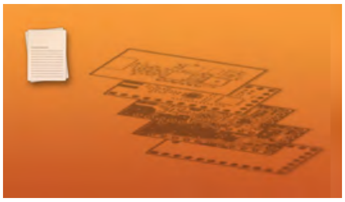
6 Carl Schattke on Stackup Design and Managing the Comonent Shortage ▶

At AltiumLive, I met Carl Schattke, CID+, a lead PCB designer with an American automaker. Carl and TTM's Julie Ellis taught a packed class on good stackup practices complete with plenty of slides showing examples of all kinds of stackups.


Carl Schattke

7 Mentor White Paper: Lean NPI at Optimum Design Associates ▶

This is the first in a series of three white papers which discuss Optimum Design Associates' business processes. For more than 20 years, Optimum Design Associates has offered its customers PCB design and layout services, and also bare-board assembly.

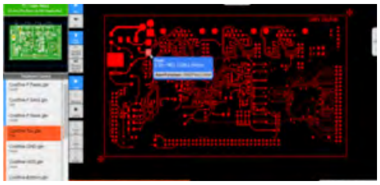


8 Cadence Reports Q4 and Fiscal Year 2018 Financial Results ▶

For the fourth quarter of 2018, Cadence reported revenue of $570 million, GAAP net income of $98 million, or $0.35 per share on a diluted basis, and non-GAAP net income of $147 million, or $0.52 per share on a diluted basis.

9 Ucamco Launches XNC PCB Drill Data Format ▶

Ucamco, KiCad and Pentalogix have developed XNC, a format for PCB drill data, with the support of Graphicode, Cuprum and ZofzPCB.



10 Casper van Doorne Discusses His AltiumLive Class, IoT, and More ▶

What's in a name? When PCB designer Casper van Doorne needed to choose a name for his service bureau, only one name would do—Doofenshmirtz Evil Incorporated, a villainous name familiar to fans of Walt Disney.


Casper van Doorne

PCBDesign007.com
for the latest circuit design news and information—anywhere, anytime.

Career Opportunities

Career Opportunities

KOH YOUNG AMERICA

Vision and Machine Learning R&D Engineer

Atlanta, GA or San Diego, CA

At Koh Young, we are focused on developing the future and continue to bolster our newly established R&D center near San Diego, California, with top talent focused on vision engineering and machine learning for electronics and medical applications. Currently, we are collaborating with top medical universities and hospitals across the U.S., Korea, and Japan to develop innovative neurosurgical robotic systems. With core technologies developed in-house, we expect to deliver neurosurgical breakthroughs.

The role will develop practical, scalable 3D machine learning solutions to solve complex challenges that detect, recognize, classify, and track medical imagery. Additional focus on the design, implementation, and deployment of full-stack computer vision and machine learning solutions.

The ideal candidates will hold a master's (doctorate preferred) in computer science or electrical engineering with at least three years of relevant experience. We desire a strong understanding of machine learning and computer vision algorithm application within embedded systems, plus significant vision expertise in multi-view geometry, 3D vision, SFM/SAM, and activity recognition.

Koh Young is the leading 3D measurement-based inspection solutions provider. We perform quality control and process optimization across a growing set of industries including electronics, final assembly, semiconductors, and most recently, medical imagery.

Join the 3D inspection leader as we expand. Forward your resume to Michelle.Hayes@KohYoung.com.

apply now

THERMAL SYSTEMS

Service Engineer Reflow Soldering Systems (m/f)

To strengthen our service team at Rehm Thermal Systems llc. in Roswell, Georgia, we are seeking candidates to fill the position of Service Engineer—Reflow Soldering Systems.

Your area of responsibility:

- Installation of Rehm reflow soldering systems at the customers' site
- Maintenance and repair work as well as technical service for our customers in the USA and Mexico
- Execution of machine training

Your profile:

- Completed education studies as an engineer in the field of electrical engineering/mechatronics or comparable education (m/f)
- Basic and specialist knowledge in the field of electronics and electrical engineering/ mechatronics
- High willingness to travel and have flexible employment
- Service-oriented and like to work independently

We offer:

- Performance-oriented, attractive compensation
- Comprehensive training
- A safe workplace in one successful group of companies
- Self-responsibility and leeway

Please send application documents online to Natalie Werner at n.werner@rehm-group.com.

apply now

Career Opportunities

Technical Service Rep
Waterbury, CT

Do you have what it takes? MacDermid Alpha Electronics Solutions is a leading supplier of specialty chemicals, providing application-specific solutions and unsurpassed technical support.

The position of the Technical Service Rep will be responsible for day-to-day support for fabricators using MacDermid Alpha's chemical products. The position requires a proactive self-starter that can work closely and independently with customers, the sales group and management to ensure that customer expectations and company interests are served.

- Have a thorough understanding of the overall PCB business, and specifics in wet processing areas.
- Prepare action plans for identification of root cause of customer process issues.
- Provide feedback to management regarding performance.
- Create and conduct customer technical presentations.
- Develop technical strategy for customers.
- Possess the ability to calm difficult situations with customers, initiate a step-by-step plan, and involve other technical help quickly to find resolution.

Hiring Profile

- Bachelor's Degree or 5-7 years' job related experience.
- Strong understanding of chemistry and chemical interaction within PCB manufacturing.
- Excellent written and oral communication skills.
- Strong track record of navigating technically through complex organizations.
- Extensive experience in all aspects of Customer Relationship Management.
- Willingness to travel.

apply now

Service Engineer USA

Limata GmbH, a provider of direct imaging system solutions for the global PCB manufacturing industry and adjacent markets, is looking for qualified candidates to fulfill the role of service engineer in the United States.

Duties:

- Assemble, install, service, and maintain our products
- Inspect the unit towards operating conditions
- Solve technical problems on-site
- Resolve problems with our customers and technical department
- Ability to support our customers in all technical questions

Qualifications:

- Proven experience in microelectronics is preferred
- Willingness to travel
- Strong verbal and written communication skills

To be part of our team, please click below and send your resume to karriere@limata.de.

apply now

Career Opportunities

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

Technical Support Engineer, Germany

We are looking for a technical support engineer to join our team at our German facility in Kirchheimbolanden. The successful candidate will assist potential and current customers in appreciating the benefits of using and optimizing the use of Ventec materials in their PCB manufacturing processes, enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. The technical support engineer will provide a two-way channel of technical communication between Ventec's production facilities and U.K./European customers.

Skills and abilities required for the role:

- Scientific and technical educational background
- Experience in the PCB industry in engineering and/or manufacturing
- Strong communications skills (German and English) with the ability to write full technical reports for group or customer distribution
- Ability to work in an organized, proactive, and enthusiastic way
- Ability to work well both in a team as well as an individual
- Good user knowledge of common Microsoft Office programs
- A full driving license is essential
- Willingness to travel regularly throughout Europe and occasionally to Asia

We offer:

- Excellent salary and benefits commensurate with experience

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

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

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Technical Sales Engineer, Germany

Want to advance your career by joining our globally successful and growing world-class CCL manufacturing company and help drive that success? Tasked with driving sales in the German-speaking markets, you will be a key member of the technical sales team. Your focus will be on Ventec's core market segments: military/aerospace, automotive, and medical offering a full range of high-reliability materials including polyimide, IMS, and thermal management products.

Skills and abilities required for the role:

- Seven to 10 years of experience in the PCB industry in engineering and/or manufacturing
- Strong communications skills (German and English)
- Project management experience
- Detail-oriented approach to tasks
- Ability to manage tasks and set goals independently as well as part of a team
- Knowledge of Microsoft Office products
- A full driving license is essential.
- Willingness to travel regularly throughout Europe and occasionally to Asia

We offer:

- Excellent salary and benefits commensurate with experience

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

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

apply now

Career Opportunities

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

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 careers.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

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

Career Opportunities

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

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

Events Calendar

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

EDI CON—Electronic Design Innovation ▸
April 1–3, 2019
Beijing, China

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

SMTA Atlanta Expo ▸
April 11, 2019
Peachtree Corners, Georgia, USA

Del Mar Electronics & Manufacturing Show ▸
May 1–2, 2019
San Diego, California, USA

Medical Electronics Symposium 2019 ▸
May 21–22, 2019
Elyria, Ohio, USA

IMS 2019 ▸
June 2–7, 2019
Boston, Massachusetts, USA

MD&M Medical Design East ▸
June 11–13, 2019
New York, New York, USA

PCB West 2019 ▸
September 9–11, 2019
Santa Clara, California, USA

SMTA International ▸
September 22–26, 2019
Rosemont, Illinois, USA

Additional Event Calendars

IPC

SMTA
Surface Mount Technology Association

pcb design 007

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 © JAMROOFERPIX**

DESIGN007 MAGAZINE

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

March 2019, Volume 8, Number 3
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering........ 29
Altium........ 33
American Standard Circuits........ 5
APCT........ 39
Candor Industries........ 61
Downstream Technologies........ 19, 81
Eagle Electronics........ 49
Electrolube........ 77
EMA Design Automation........ 59
General Circuits........ 23
I-Connect007 eBooks........ 2, 3, 65
In-Circuit Design Pty Ltd........ 73
IPC........ 43
ITEQ........ 71
Mentor, a Siemens Business........ 53
Meptec........ 37
Miraco........ 11
PCB007 China Magazine........ 51
Prototron Circuits........ 75
Pulsonix........ 67
Rogers Corporation........ 45
SimplifyDA........ 25
US Circuit........ 57
Ventec International Group........ 7
Whizz Systems........ 15