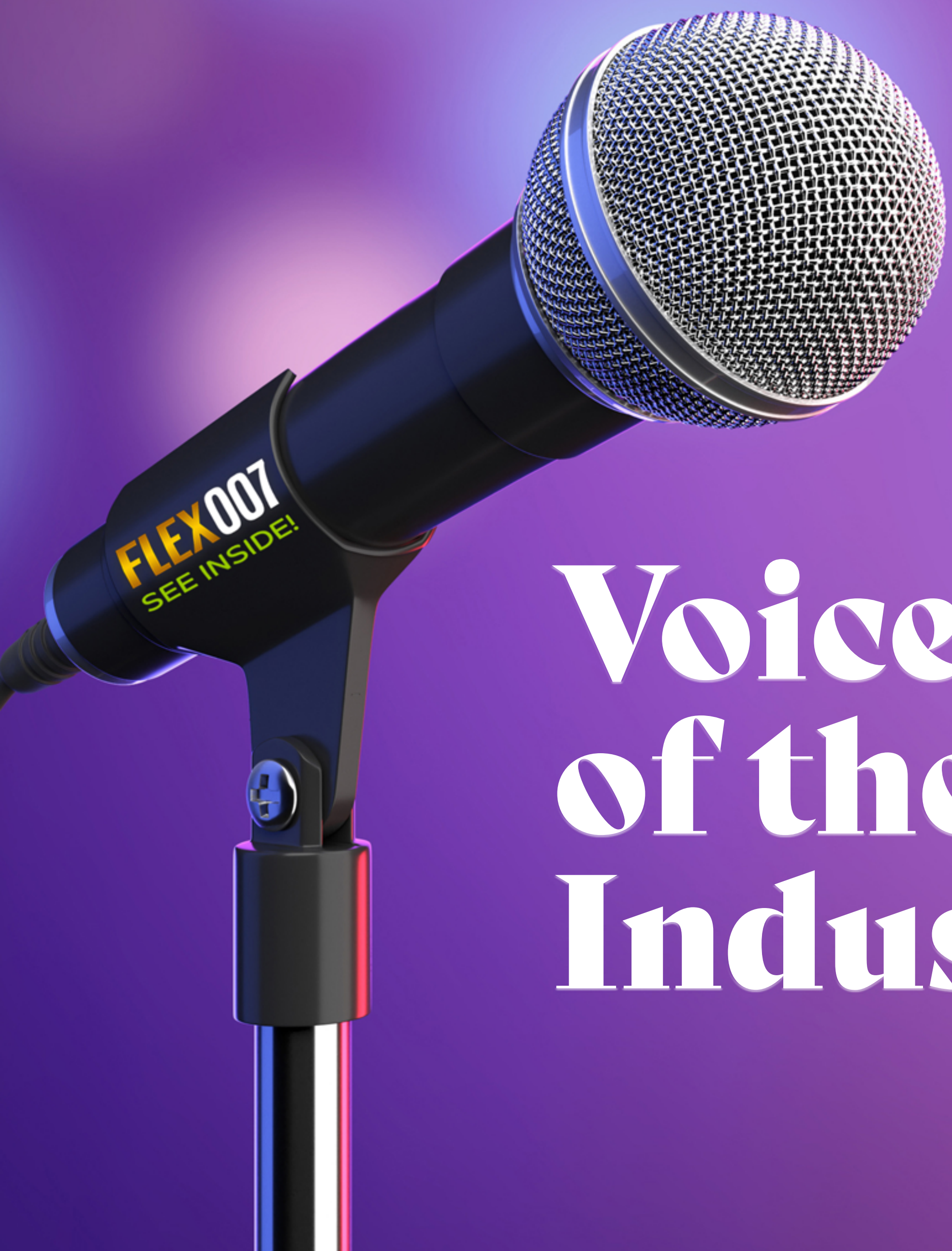
NOVEMBER 2019

DESIGN007 MAGAZINE



Voices of the Industry


OPEN
24/7
OUR LIBRARY
IS ALWAYS
OPEN


THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO... Advanced Manufacturing in the Digital Age
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Producing the Perfect Data Package
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO... Conformal Coatings for Harsh Environments
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Design for Manufacturing (DFM)
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Fundamentals of RF/Microwave PCBs


THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Documentation
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Executing Complex PCBs
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO... Low-Temperature Soldering
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Flex and Rigid-Flex Fundamentals
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO... Thermal Management with Insulated Metal Substrates

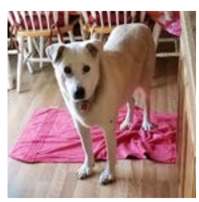
DESIGN007 MAGAZINE

Voices of the Industry

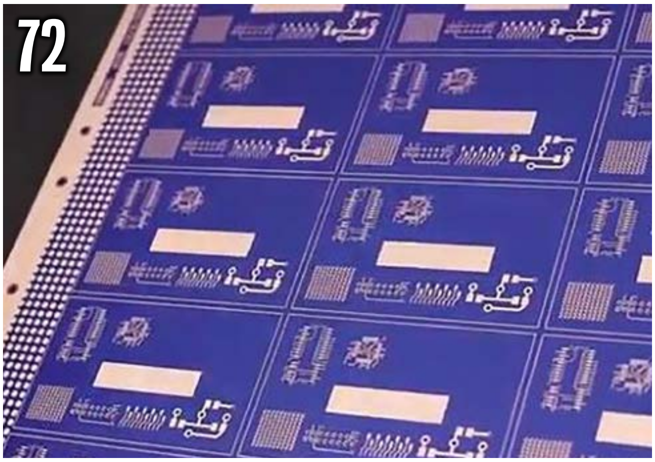
This month, we handed the microphone to a variety of designers and design engineers and asked them to share their thoughts about their jobs, technology, and the trends that they're seeing in the industry—the good, the bad, and the challenging. Read on for some eye-opening interviews.

FEATURE INTERVIEWS:

FEATURE COLUMN:



DESIGN007 MAGAZINE



SHORTS:

COLUMNS:

DEPARTMENTS:

HIGHLIGHTS:

FLEX007

Focusing on Flex

When a designer starts talking about using flex, the rigid board designers gather around, eager to hear more. Many rigid board designers and design engineers have their eyes on the flex space. For this month's Flex007, we look into some of the things designers should know before they begin designing flex or rigid-flex circuits.

Voices From the Road

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

I'm not usually a road warrior, not like some of you people in sales who live in airports. I cover four or five trade shows each year, and I attend a few conferences and other events.

But I've been traveling a lot for the past few months, and I've met with dozens of designers and design engineers. After talking to many of you, I'm beginning to believe that we'll look back on this period as the "Good Ol' Days."

If you ask a PCB designer for his opinion (or increasingly, her opinion), you're likely to get an earful. Designers are not a shy group of people; they are not afraid to "call the baby ugly," as the saying goes. But I'm not hearing much in the way of ugly lately.

Many designers that I've met recently say that they're having fun designing boards for the latest cutting-edge devices. They feel as if they're really contributing, and they're proud to be part of a great team. There are plenty of new, challenging technical problems popping up every day, but when I hear designers talking, it sounds like solving these problems is all part of the fun.

The return of the word "fun" is a good omen. For a decade or so, I don't remember many designers using the word "fun" to describe their jobs, the electronics industry, or their companies. Things were tough all over, and designers were worried about their jobs.

Contrast that with today. If you're a young designer looking for a job, odds are that you won't be for long. Fortunately, we are seeing more young people becoming designers and design engineers. Let's congratulate Michael Steffen and Kalen Brown, two designers who recently passed the IPC Certified Interconnect Designer exam. They also attended the Designers Council Executive Board meeting at PCB West, where they were given a warm welcome.

As Michael and Kalen explain in this issue, their managers are firm believers in sending them to conferences to get the training they need to stay current in this challenging field. We hope to see more young people like Michael and Kalen join the ranks of CID grads.

This month, we handed the microphone to a variety of designers and design engineers and asked them to share their thoughts about their jobs, technology, and that they think about this industry. We start with interviews with Michael and Kalen, who discuss their work as designers and why they decided to take the CID exam. Next, Todd Westerhoff of Mentor explains why the designers of today need to understand many concepts once left to the signal integrity engineers. Then, Bryan LaPointe of Cadence Design Systems discusses the satisfaction he feels working as a product engineer, as well as the diversity of the entire EDA segment. Matt Stevenson of Sunstone Circuits breaks down some of the reasons why this may be the most exciting time to be in this field.

EMA's Chris Banton focuses on new technology like 5G, AI, and printed electronics, and discusses their effects on PCB design. Columnist Tim Haag highlights the need for more communication between designers and manufacturers, and why assumptions can be your worst enemy. Tamara Jovanovich of Happiest Baby updates us on her first full year as a PCB designer, and she offers some advice for new designers and design engineers. Insulectro's Megan Teta explains why she enjoys being involved in so many different aspects of the job, and why she's glad to be an example for young girls considering STEM careers. And Curtis Scott of Current Products discusses his job designing PCBs for automated window coverings near the beaches of the Gulf of Mexico. We also have columns by regular contributors Barry Olney, Stephen Chavez, Vern Solberg, Bob Tise, and Phil Kinner.

Speaking of traveling, we'll be providing full coverage of productronica 2019. If you can't make it to Munich, don't worry; we'll bring you all the event information you need. See you next month! **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 19 years. He can be reached by clicking here.

Buzz Aldrin Named 2019, Space Cowboy

The EarthLight Foundation is proud to announce the 2019 Space Cowboy Award will be presented to Dr. "Buzz" Aldrin, member of the Apollo mission to the Moon, and longtime activist for the human exploration and settlement of space. The award will be presented to Dr. Aldrin during the Space Cowboy Ball on November 16, 2019, at the prestigious Bullock Texas State History Museum, in Austin, Texas.

"We are presenting Buzz with the Space Cowboy Award not so much for being part of the first mission to the Moon, but for what he has done ever since," explained EarthLight Foundation's founder and chairman of the board, Rick Tumlinson. "Ever since Apollo, he has advocated for the human breakout into space, and tirelessly worked to inspire young people to reach for the stars. Buzz is a known leader in the space community, and in the 50 years since he and Neil Armstrong's giant leap on the Moon, he has never slowed down in his quest to help humanity take our next steps into the space frontier."

The Space Cowboy Ball is a fundraiser for the EarthLight Foundation's new Endowment for Tomorrow, supporting STEAM Space education and research projects that further the human settlement of space while benefiting the Earth.

(Source: EarthLight Foundation)

Meet Two New CID Grads: Kalen Brown and Michael Steffen

Feature Interview by Andy Shaughnessy
I-CONNECT007

If you're a designer with 30 or 40 years of experience, you might be wondering who is going to replace you when you retire. If so, read on!

At PCB West, I spoke with two newly minted Certified Interconnect Designers: Kalen Brown of EaglePicher and Michael Steffen of Crystal Group. After passing the CID exam, they attended the IPC Designers Council (DC) Executive Board Meeting, where the more "seasoned" veterans were very happy to meet them.

Kalen and Michael shared their thoughts on the job, the PCB design community, and the need for designers to continue their education throughout their careers.

Andy Shaughnessy: Kalen, you are a newly minted CID recipient. Tell us about your class.

Kalen Brown: Yes. I attended Kelly Dack's class. We just finished, and it was a great class. I learned a lot of stuff I didn't know that I didn't know.

Shaughnessy: How long have you been in the industry?

Brown: I graduated and joined the industry about five years ago. My first job was at an EaglePicher Technologies doing software, electrical design, and board layout. We don't fabricate the boards; we just design and send them out to be built and assembled. EaglePicher makes mission-critical batteries and the group that I am in works on battery management systems to keep a battery safe. I have learned a lot and didn't know very much about circuit board design from out of school. It was an eye-opening experience getting thrown into it. Now, I realize that it wasn't always a job for electrical engineers. I had no idea that a board designer and an engineer were two different people; that was a very big learning experience too. Everybody here has been very passionate about what they do. It has been incredible to hear all of the stories, and everyone wants to help. It seems like it's a great group to be in, which makes me very excited.

Shaughnessy: We were just at the IPC DC Executive Board Meeting, which was open to the public. Was that your first IPC DC meeting?

Brown: Yes, this was my first time attending PCB West, my first CID class, and my first IPC DC meeting. It was a lot of firsts for me.

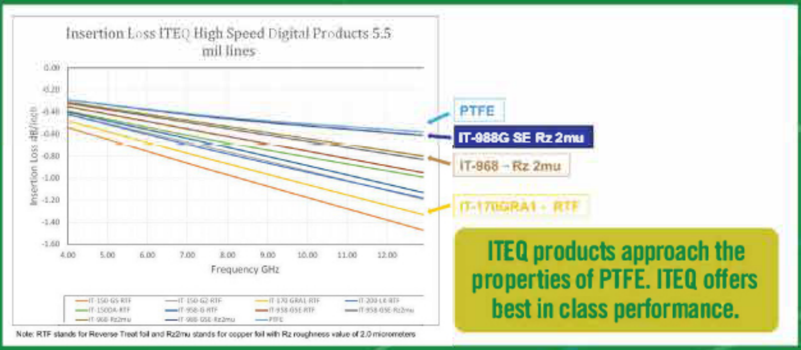
IT-968 / IT-968 SE

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

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

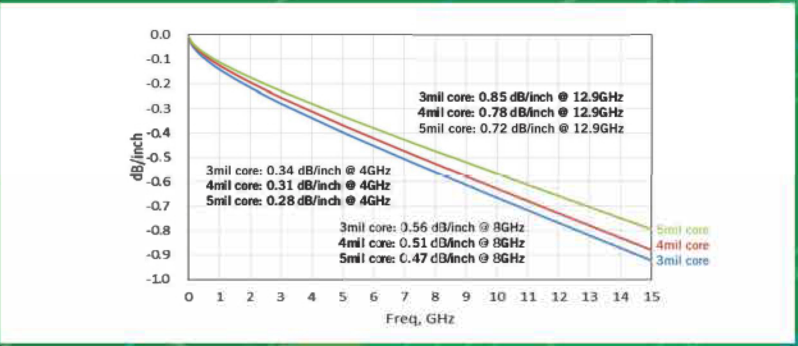
IT-968 SE
Dk @ 10 GHz - 3.30
Df @ 10 GHz - 0.0019

Insertion Loss - Measured

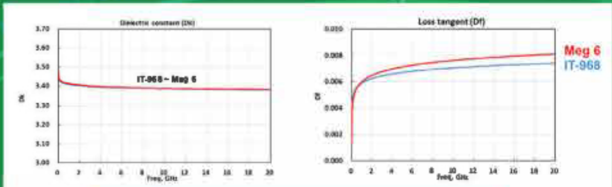


Industry Test Vehicle
32 layers, 0.140" thick
Four 2 oz copper internal layers
0.8 mm pitch, 9.8 mil drills
- Passed 1000 hours CAF, 10 V bias, 50 V
- 1ST - Passed 1000 cycles, 6x 260°C precondition
- No delamination after 8x 260°C reflow after 2 weeks at 35°C/ 85% RH

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

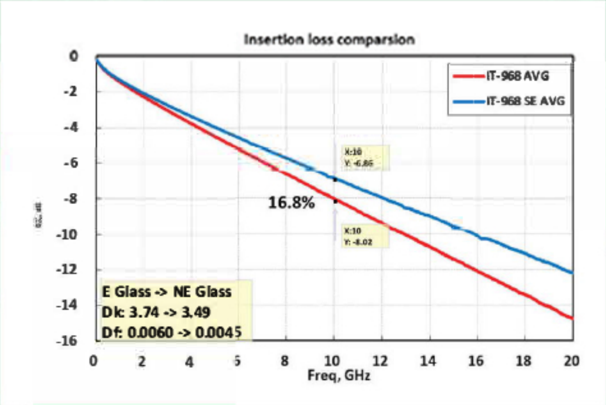
Features
High Tg, low CTE, High Thermal Reliability with Ultra Low Df < .005. Designed for High Speed Applications.

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 (°C)	DSC	185	185
T-288 (w/ 1 Oz Cu, min)	TMA	120+	120+
Td-5%(°C)	TGA 5% loss	390+	390+
CTE (%), 50-260°C	TMA	2.2	2.2
Peel strength (lb/inch)	1 oz	6	6
Water Absorption	D-24/23	< 0.1	< 0.1
Dk, 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	3.8 3.8-3.7	3.4 3.4-3.7
Df, 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	0.0032 0.0036-0.0050	0.0028 0.0031-0.0040

E-Glass vs Low Dk Glass



productronica 2019
Accelerating Innovation

B3.580

ITEQ

Shaughnessy: We're all glad you could come to the show. Are you the only designer at your company?

Brown: No, all four designers from our Joplin R&D group are here. We attended the CID class together, which has been great to bounce ideas around while we are thinking about them and learn all of the things we need to be doing going forward.

Shaughnessy: After five years, what has surprised you the most about this whole industry?

Brown: When I came out of school, I knew what circuit theory was and how I wanted to do a schematic, but the idea of physically embodying it into the real world was nothing that had even crossed my mind. If we're not going to put it on a breadboard, I didn't know how to do it. It's very nice to see all the tools that make this happen in a way that is effective. And that there are people who have been working on this to make it work well.

It's very nice to see all the tools that make this happen in a way that is effective.

Shaughnessy: What are some of the biggest challenges that you face every day doing layout?

Brown: Learning. As I said, I hadn't really done this, and we didn't really have very many mentoring opportunities to learn these things. It has definitely been a ground-up thing for us. The design idea is all new for me.

Shaughnessy: We hear more and more about mentoring now, but in a lot of companies, you're still on your own.

Brown: I hope we can get something together to help leverage all of that experience that's out there.

Shaughnessy: Do you have any advice for a designer who is just starting?

Brown: Keep your eyes open for learning opportunities, and if you can't find something, reach out and ask, and somebody will be there to help you.

Shaughnessy: I'm glad you all came here as a group. That shows that you work for a good company that supports you.

Brown: We're very grateful for the opportunity to attend PCB West and do this certification, back to back. A lot of people didn't get to stay for the whole week, so I really appreciate that. All of the things we didn't know we can take forward; we apply what we learned to everything we do.

Shaughnessy: It has been great talking with you, Kalen. Thanks for speaking with me.

Brown: Thank you, Andy.

Next I spoke with Michael Steffen, senior electrical engineer at Crystal Group, who just received his CID this week.

Andy Shaughnessy: Hi, Michael. You just received your CID this week. I believe Kelly Dack was your instructor.

Michael Steffen: That's correct.

Shaughnessy: Congratulations! How did you get into the PCB industry?

Steffen: I'm a 2012 graduate of Iowa State University in Ames, Iowa. I've been working with Crystal Group since 2010 as an intern doing a variety of electrical engineering, PCB design work, assembly, and troubleshooting and test.

Shaughnessy: So, you have an EE degree. How did you learn about PCB design?

Steffen: My dad was a PCB designer. He didn't have an engineering degree, but he knew enough to demonstrate basic electronics to me when I was a kid. I was exposed to electronics pretty early on, and I always found it exciting. But I was mostly interested in how it worked, so engineering was more appropriate for me, and I got my BSEE.

As for PCB design, that originally came out of necessity. You can only get so far with a breadboard. I started laying out and hand etching simple circuits in high school and graduated to professionally fabricated boards while working on university organization designs. Over time, I started appreciating the artistic aspects of it. There's a science to engineering, but there's an art to PCB design, which is interesting to me. There are an infinite number of different ways to design the same board, and having that freedom is a different experience compared to the more regimented circuit design process.

Shaughnessy: There's this idea, true or not, that designers are more artistic and electrical engineers are more scientific. Do you find that to be the case?

Steffen: I should clarify myself because referring to design as pure art is not exactly the right sentiment. There's just as much science in proper layout as there is in design. Particularly after listening to professional designers like Rick Hartley speak, you realize that there's physics involved that isn't covered in undergrad engineering courses. It's exciting to be able to come to an event like this and learn what goes into it from experts in the industry because it's immediately useful. You can take home these concepts and use them right away; you don't get that in such a practical format from the engineering curriculum. That said, I think design is still an artistic process, and it helps to have an artistic mind. Once the critical parts of the board are designed, much of the rest of the design comes down to what "feels right." From a purely scientific standpoint, there is not just one right answer, and that's where artistic design comes in.

Shaughnessy: You have been in the industry for about seven years. What do you like most about it, and what was the biggest surprise?

Steffen: It's always changing, and there's always something that's new. Companies continue to innovate the way people use electronics and interact with them, and there are always better components being developed to enable this. There are also many different parts and ways to assemble them that, a lot of times, you're doing something that nobody has ever done before. You are constantly inventing new things that don't exist anywhere else. It's fascinating work.

Shaughnessy: What is Crystal Group's main sweet spot?

Steffen: Crystal Group makes ruggedized computer hardware, largely for industrial and military applications. We've positioned ourselves mid-way between the standard commercial equipment manufacturers and the big government prime contractors; we're more rugged than the former but less expensive than the latter. We provide hardware for many major military platforms, as well as energy exploration, autonomous vehicles, and other industrial applications.

Shaughnessy: What advice would you give to somebody coming into the industry?

Steffen: Find a mentor. That's something that I struggled with for a long time and still do. There's a lot of information online and in books, but it's scattered and can be hard to find. Try your best to find somebody who is willing to take you under their wing and who you can ask questions of and get their opinions on how best to do things. There's so much breadth and depth to the industry that jumping into it blindly can just lead to frustration; I know that from personal experience.

Shaughnessy: And regarding tribal knowledge, people learn how to do one thing a certain way, and then when they take a test, they miss questions because their whole company did it wrong and no one noticed.

Steffen: Exactly. You get into the groove of doing the same thing year after year, with the same building blocks and the processes, and you get comfortable. But then you come to a conference like this, for instance, and think, "Wow! We can do all these other things that we never considered." It's very eye-opening.

Shaughnessy: What's your home base?

Steffen: We are based out of Hiawatha, Iowa, which is a suburb of Cedar Rapids.

Shaughnessy: We just left the IPC DC Executive Board Meeting. Is there an active chapter near you?

Steffen: I don't believe so. Cedar Rapids is also home to the headquarters of Collins Aerospace, as well as several other electronics design companies. If there's not a chapter there, it seems like that would be a prime area.

Shaughnessy: You could start a chapter. It's a lot of work, though. Talk to Scott McCurdy in Orange County. He has been doing it for a long time and spends part of each day working on it.

Steffen: I'd like to have local resources for the universities, at least. During my time at Iowa State, I didn't feel like I had much industry support, at least in my experience with university organizations. There's faculty support, but there's not a lot of industry support that I was able to access. Having access to this industry in those organizations would be great, if not an actual IPC DC chapter.

Shaughnessy: Had you been to an IPC DC meeting before?

Steffen: I had not. In fact, this is my first time visiting the Santa Clara area, so it has all been interesting.

Shaughnessy: The good thing is you can talk to any of those people, during or after a meeting, and ask them about anything; they won't mind if you ask questions. How many people work for Crystal Group?

Steffen: Crystal Group has almost 300 employees, but the electrical engineering group is only about a dozen people, so we end up doing all of the jobs. We get exposure to the whole gamut, from concept to assembly, testing, etc. It's helpful to be connected to the product all the way through the manufacturing process. You really feel like you own your designs.

Shaughnessy: That sounds like a good place to work, and they support you coming to events like this, which is a big plus. It was great meeting you, Michael. Thank you.

Steffen: Thanks for the opportunity, Andy. **DESIGN007**

A Robot Puppet Can Learn to Walk If It's Hooked Up to Human Legs

Getting robots to mimic how we move so effortlessly is the premise of a study from the University of Illinois and MIT. The team created a human-machine interface mapping an operator's movements onto a robot. The system also tracks the movements of their body, using a vest also wired up with sensors. The data captured from the torso and legs is then mapped onto a two-legged robot.

The system allows the operator to "feel" what the robot is feeling if it bumps into a wall or gets nudged, for instance. This means trainers can adjust their movements as required. This feedback includes safety measures that automatically cut out power if the robot experiences dangerous levels of force. (Source: MIT Technology Review)

Todd Westerhoff on the Value of Solid Design Skills

Feature Interview by Andy Shaughnessy
I-CONNECT007

I recently caught up with Todd Westerhoff, product marketing manager for Mentor's HyperLynx signal integrity (SI) tools. Todd discusses some of the challenges that he and his customers are facing and why good design skills have more influence on a PCB than any software tool.

Andy Shaughnessy: Hi, Todd. For those who may not be familiar with you, give us a quick background about yourself and your time in PCB design and EDA industries.

Todd Westerhoff: I've been working with system simulation tools for the past 40 years. I started with digital logic/timing/fault simulation at GenRad and HHB Systems, added circuit simulation at FutureNet, VHDL at Racal-Redac, RF/microwave analysis at Compact Software, and then got into high-speed design while consulting through ViewLogic. I've spent the past 23 years in SI, working for Cadence, Cisco, SiSoft, and Mentor, A Siemens Business in both design engineering and EDA marketing roles. I consider myself fortunate to have been both a user and provider of EDA tools; I think my experiences as a designer help me make better decisions with the capabilities we're developing today.

Shaughnessy: What do you see as the biggest challenge right now for PCB designers and design engineers?

Westerhoff: SI has been one of those things that people have typically "left to the experts." When I worked at Cisco, we had centralized SI groups supporting multiple design teams. The design team looked to the SI expert to determine what the routing rules should be and validate the design once routing was available. That worked when the number of critical interfaces in designs was small, but that's no longer the case. Just about every signal on every design now qualifies as "high-speed" if you compare the driver's edge rate to the net's electrical length.

But what's a designer supposed to do? There aren't nearly enough SI experts to go around, and the ones we do have are focused on the cutting edge stuff. Designers still need to make tradeoffs on high-interfaces when

they can't route "according to pre-defined rules," but they don't necessarily have access to the analysis support they need. That suggests that basic design simulation needs to be part of the designer's desktop toolkit, although progress has been slow in that area. I would maintain that the same is true for other disciplines. It's not just SI—you can add power integrity (PI), EMI/EMC, thermal, and others to that mix. A designer needs to know a lot about many different disciplines to be successful today.

In most companies, the "design review" is how designers get feedback on their work by having it reviewed by peers and experts. Design reviews take significant time and effort, but they're quicker and less resource-intensive than full-scale modeling and simulation. But there's still a rub; running a design review requires check-pointing the design, sending something out for review, and waiting for people (who are already busy with other tasks) to review the material, and then pulling all that feedback together. That represents a considerable delay in a project schedule that's already packed.

Ideally, we'd combine the ability to perform first-order analysis on the engineer's desktop, allowing the evaluation of design tradeoffs with automated multi-domain design reviews to provide feedback in near real-time. That way, designers could minimize the risk and delays they experience with the ways we do things today. I think the biggest challenge designers face today is simply trying to learn everything they need to know and analyze to make sure their designs work as intended. We'd like to help ease that burden a bit.

Shaughnessy: I know you like to take devices apart around the house. Have you had any more epiphanies, like the one you shared with me at DesignCon about the new Ethernet switch and the need for designers to know how to do a "cost-reduced" design?

Westerhoff: As it turns out, I've been both taking things apart and putting things together. As far as taking things apart, I had an LED bulb fail in my home office and decided to take a look inside (Figure 1). It's official: We've reached the point where we commonly have integrated circuits inside light bulbs! Thankfully, this particular bulb is a commodity item that doesn't take up an IP address on my wireless network, although I'm sure that some bulbs do.

As far as putting things together, I decided to build a machine to run 3D electromagnetic simulations, and I didn't want to pay an arm and a leg for server-grade hardware. Thus, the task became seeing how far I could get with consumer-grade components. In the consumer computing market, performance and gaming are pretty much synonymous, so I started with a gaming motherboard and built it out as far as I could. I ended up with a 16-core, 32-thread,

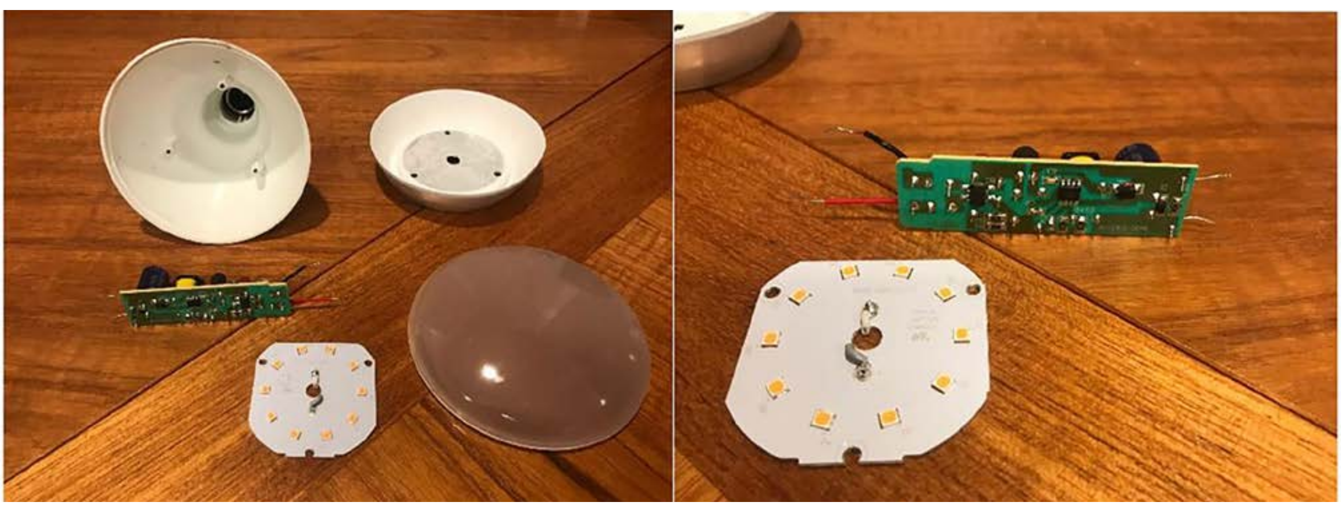


Figure 1: A disassembled LED bulb from Todd's home office.

water-cooled machine with 128 GB of RAM, an ultra-fast M.2 NVMe and 850W power supply, a modern graphics card, case, keyboard, etc., for about $3,000.

I learned some hard lessons about practical (as opposed to theoretical) memory bandwidth and how it drives compute throughput—things I've been exposed to for years but never fully understood. I ended up with a machine that does a decent job of running 3D EM simulations, which was the original goal, after all.

Shaughnessy: How do you think designers, and perhaps EDA tool companies, will be affected by new technologies like 5G and IoT?

Westerhoff: I think that we're living in an increasingly client/server world, where electronics technology is embedded in pretty much everything. That drives design in two different directions, even though they're usually two sides of the same coin.

On the server-side, designs are driven by performance, capacity, and reliability; cost is usually secondary. These are the design applications that EDA has typically targeted, and these are the applications with which SI/PI/EMC experts are usually associated. Each major new application—112G, 5G, take your pick—brings new design issues that the industry has to work through and bring to production. The design challenges keep getting tougher, but the broader pattern has been repeating since the dawn of EDA.

The client-side is more or less the opposite, even though the core technology is the same, with 5G, let's say. This is typically the consumer side of the equation; cost and time to market are the prime drivers. The product has to work, but that's it. Providing 10% more range in a cellphone won't substantially increase sales or justify a 10% increase in the phone's price, for example. These are also the applications where the designers may not have enough support from the SI/PI/EMC experts because they're busy on the other side of the fence.

So, how do these products get designed? Designers still need to make tradeoffs, and those tradeoffs are often more extreme due to space and cost constraints. The "rules of thumb" that work in the server world often won't work here, and I think this is an interesting opportunity for EDA tools. EDA tools have typically focused on design performance, not cost. What would EDA tools for a cost-sensitive design look like? I'm not sure I know, but I'm working on finding out.

Shaughnessy: What advice would you give a new designer or an EE just entering this segment?

Westerhoff: Try to take the time to understand how things work and build up your engineering intuition. Today's designers are surrounded by sophisticated simulation tools, but those tools are a poor substitute for human understanding and insight. Besides, if you don't have a strong grasp of the physics and the behavior you expect, how do you know if your simulation results are believable or not? Simulators are great, but they're still just tools. I don't expect a simulator to make a design decision any more than I expect a hammer will be able to drive nails by itself. Designers who can articulate the physics behind why something behaves the way it does—and combine that knowledge with practical advice of what design tradeoffs to make and how to verify them through analysis—will be highly valuable in any environment.

Shaughnessy: Is there anything you'd like to add?

Westerhoff: I have a great line from an old computer manual to share. By way of background, I'm a bit of a computer historian, fascinated by how computers evolved in the late 1950s and early 1960s. The IBM 650 RAMAC was IBM's first commercial computer introduced in 1954 with a "bi-quinary" architecture. In those days, computer manufacturers were still trying to explain to the public what computers were, in contrast to the "giant-brain" images that science fiction writers and the media had portrayed.

One of the programming manuals for the 650

RAMAC [1] has the best description of computers that I've ever seen. The author characterizes a computer as "…a remarkably fast and phenomenally accurate moron." A computer's job is to compute, not think. Granted, computers have come a very long way since 1958, and these days, they are a lot closer to thinking (or something like thinking) than they were back then. But even if the computer could think, that's no reason for the designer not to. A simulator is still just a tool, and design is the designer's responsibility. I've been running system simulations for 40 years, but the first thing I always ask myself is still, "Do I believe this?"

Shaughnessy: That's what it comes down to. Thanks for speaking with me, Todd.

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

Reference

1. Richard V. Andree, "Programming the IBM 650 Magnetic Drum Computer and Data-Processing Machine," Holt, Rinehart & Winston Inc., 1958.

Carbon Dioxide Capture and Use Could Become Big Business

Capturing carbon dioxide and turning it into commercial products, such as fuels or construction materials, could become a global industry, according to a study by researchers from UCLA, the University of Oxford and five other institutions.

The research is the most comprehensive study to date investigating the potential future scale and cost of 10 different ways to use carbon dioxide, including in fuels and chemicals, plastics, building materials, soil management and forestry. The study considered processes using carbon dioxide captured from waste gases that are produced by burning fossil fuels or from the atmosphere by an industrial process.

The research found that on average each utilization pathway could use around 0.5 gigatonnes of carbon dioxide per year that would otherwise escape into the atmosphere. (A tonne, or metric ton, is equivalent to 1,000 kilograms, and a gigatonne is 1 billion tonnes, or about 1.1 billion U.S. tons.)

A top-end scenario could see more than 10 gigatonnes of carbon dioxide a year used, at a theoretical cost of under $100 per tonne of carbon dioxide.

"The analysis we presented makes clear that carbon dioxide utilization can be part of the solution to combat climate change, but only if those with the power to make decisions at every level of government and finance commit to changing policies and providing market incentives across multiple sectors," said Emily Carter, a distinguished professor of chemical and biomolecular engineering at the UCLA Samueli School of Engineering and a co-author of the paper. (Source: UCLA Newsroom)

Bryan LaPointe: A Great Time to Be Working With Technology

Feature Interview by Andy Shaughnessy
I-CONNECT007

I recently interviewed Bryan LaPointe, a product engineer with Cadence Design Systems, and one of the younger people in the world of PCB design and EDA tools. I asked Bryan to discuss some of the challenges technologists are facing in PCB design and engineering, as well as some of the things that make this field so exciting right now.

Andy Shaughnessy: Hi, Bryan. Give us a quick background about yourself and your time in PCB design and the EDA world.

Bryan LaPointe: My journey into the EDA world started seven years ago after graduating college. I decided to put my forensic science degree aside and began working as an intern, designing PCBs in New Hampshire. That internship morphed into a full-time position as a PCB designer, presenter, and trainer before finding my current position at Cadence Design Systems as a product engineer. Since then, I've spent much of my time branching out to learn new areas of the EDA industry and even finished my M.S. degree in psychology.

Shaughnessy: What do you see as the biggest challenge right now for PCB designers and design engineers?

LaPointe: The biggest challenge I see for PCB designers right now is the growing complexity of their jobs. Engineers and designers are continually being asked to take on more roles, perform more duties, and learn more tools and technologies.

Shaughnessy: What is your favorite part of your job or this industry?

LaPointe: The best part about being a product engineer has to be the immediate feedback and impact I get from the market. When a product engineer does something well, the EDA world benefits as a whole and the product engineer sees and feels the buzz and excitement. If I had to choose one thing about the EDA industry that really excites me, the best part comes from its cultural and experiential diversity. The EDA industry has been responsible for pulling together more countries, people, and cultures than perhaps any other in the world.

Shaughnessy: I hadn't thought about that, but you're right. So, what new technologies are you interested in?

LaPointe: I am most interested in augmented reality (AR) and virtual reality (VR) tech-

nologies. I'm younger, and I play computer games—probably more than I should—and blurring the line between reality and computers is a very exciting prospect. This holds true not just for its implications for video games but also its practical implications. VR provides an extremely immersive environment that allows people to travel and experience things they could have never imagined. AR, on the other hand, provides practical productivity boosters and quality-of-life enhancers, which make day-to-day living more bearable.

Shaughnessy: What advice would you give a new designer or EE just entering this field?

LaPointe: Learn, learn, learn. Technology moves fast, and as a result, the employees who work in EDA must move fast too. Designers and engineers should aim to learn more, do more, and experience more every day to better prepare themselves for the next, and inevitably large, technological advance.

Shaughnessy: Thanks for speaking with me, Bryan.

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

Unlocking Turbulence

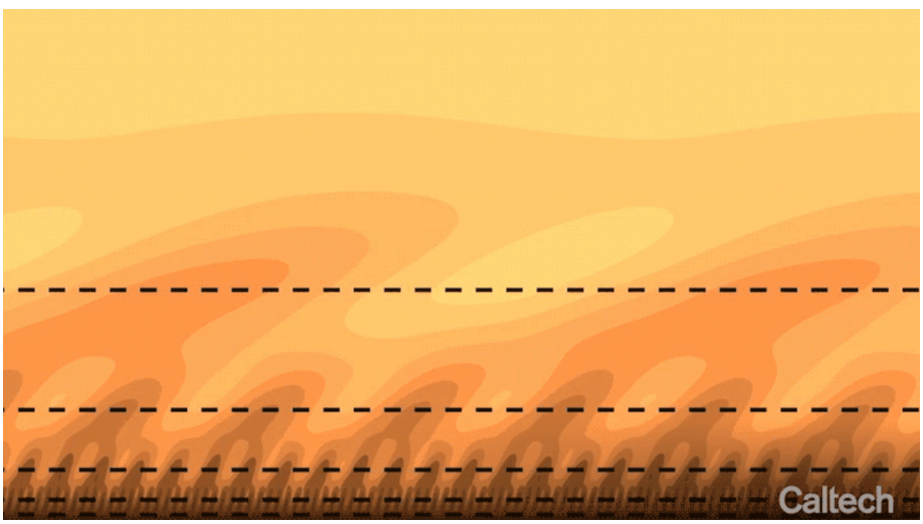
A Caltech engineer has unlocked some of the secrets behind turbulence, a much-studied but difficult-to-pin-down phenomenon that mixes fluids when they flow past a solid boundary.

Beverly McKeon, the Theodore von Kármán Professor of Aeronautics in the Division of Engineering and Applied Science, studies fluid mechanics. She specializes in turbulent flows. These types of flows are often seen in pipes and around aircraft.

At the boundary where a fluid flows over a fixed structure, a turbulent boundary layer is created where the fluid interacts with the wall, creating eddies in the current. These eddies may seem to be random at first glance, but they actually create distinct patterns. These eddies have a significant impact on the fluid flow, helping to determine features such as its pressure, velocity, and density, which are important to understand when engineering an aircraft or industrial piping, for example.

Like the weather, turbulence is a dynamic and ever-changing phenomenon. The shape and structure of the eddies in turbulence are geometrically self-similar, meaning that each of the eddies is identical, just on different scales, similar to a fractal pattern.

Mathematically quantifying these repetitions, McKeon was able to formulate a dynamical model that describes turbulence using a sort of shorthand, "We knew that, underlying these very complicated structures, there had to be a very simple pattern. We just didn't know what that pattern was until now," says McKeon.



The model could prove useful to engineers across the industry who are looking to more easily simulate turbulent systems. But more importantly, it represents fundamental research that will help scientists and engineers better understand what drives those turbulent systems.

McKeon's study is titled "Self-similar hierarchies and attached eddies" and was published by *Physical Review Fluids* on August 26. Her work was funded by the Office of Naval Research.

(Source: www.caltech.edu)


Matt Stevenson

Matt Stevenson: An Exciting Time to Be in the Industry

Feature Interview by Andy Shaughnessy
I-CONNECT007

I spoke with Matt Stevenson about some of the changes he has seen in the industry, from starting in a PCB shop chemistry lab 25 years ago, to his current position of VP of sales and marketing for Sunstone Circuits. Matt gives his thoughts on his decades spent working with designers and fabricators and explains why he thinks this is one of the most exciting times the electronics industry has ever seen.

Andy Shaughnessy: Today, I'm talking with Matt Stevenson, VP of sales and marketing for Sunstone. Can you give us a quick background about yourself and how you got into the business?

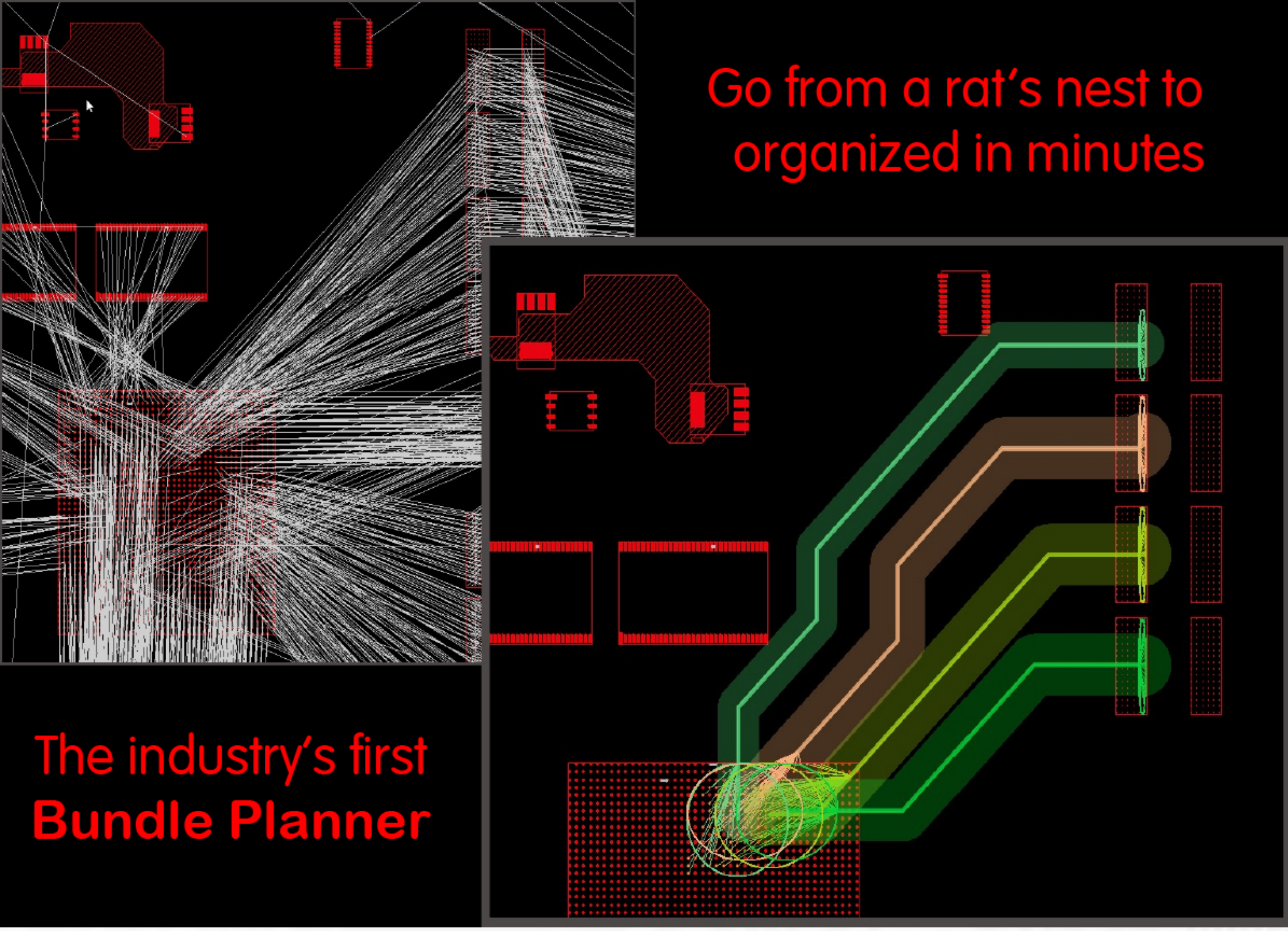
Matt Stevenson: It's great to see you again, Andy. I've been working with circuit boards for about 25 years now. I started innocuously with a job out of college in a chemistry lab, doing analytical chemistry for a board shop in Colorado; after they closed that shop, I moved to Oregon to work for another big board shop, where I did process, material, and quality engineering for about 10 years. Then, I moved to Sunstone about 13 years ago, starting out as the quality manager. After about six in that role, I had a chance to use my MBA and moved into marketing, and here we are today.

Shaughnessy: I know a lot of the Sunstone people have been there 10–20 years; there's not a lot of job-hopping.

Stevenson: I think our average length of service for the entire floor is approximately 12 years, so you could say that our people tend to stay quite a while. I was not planning a long-term stop at Sunstone, but I fell for the culture that they had developed, and I have not looked back.

Shaughnessy: Over the years in your job, you've seen a lot of things. What are some of the things that impress you about the industry? What do you think is important that we should note?

Stevenson: For me, right now is probably the most exciting time that we've ever had in the electronics industry. You can't find too many


Go from a rat's nest to
organized in minutes
The industry's first
Bundle Planner

things without some sort of electronics in them. Being in the manufacturing area for electronics opens the door for all of us to succeed and help build something. The altruistic view of it would be to help to build the next thing that will help to save the planet. We get to be a part of that every day, helping to create the next big or better thing, or even a planet-saving thing. In conjunction with that, everything electronic is getting smaller and faster and has more memory and more stringent requirements in terms of the materials or the manufacturing, especially the bare boards.

To fit everything required for the design, and all of that power into a small chip, you must have very dense circuitry. Gone are the days of scope drilling and big-format panels; now, we enter the world of high-density microvias and all of that other good HDI stuff, which is going to continue. It will get to a point eventually, where physics will be the barrier that needs a breakthrough to keep getting denser and smaller. But for the time being, probably for the next 25 years, things will continue to get smaller and more complex.

Shaughnessy: Everybody was worried about Moore's Law. Then, they predicted the end of copper.

Stevenson: They have gotten around a lot of that so far, in part due to the exponential cost increases to manufacture these newer technologies (Moore's second law) on the smaller scales.

Shaughnessy: Do you see any of the new technologies, like IoT and 5G?

Stevenson: We're starting to see it. Sunstone primarily has been used over the years for rapid prototyping, so we've seen a lot of the early design concepts come in and out. We're starting to see a little bit more of the production levels, especially in the IoT realm, but we haven't seen a lot of 5G stuff yet. I am curious to see some of the PCB layouts that will result from these.

Shaughnessy: A lot of people are waiting to see.

Stevenson: That's going to be an exciting program to see rolled out. It will be quite a feat getting all of that technology developed and implemented into the infrastructure.

Shaughnessy: Do you do automotive work?

Stevenson: We do business in about every industry. Automotive, for us, is not a huge player at this point, but we're going to make strides to get a little more in-depth and ingrained into the ever-growing automotive electronics industry. The number of electronics going into cars today is crazy, including everything from lidar and collision sensors to autonomous driving, electric engines, creature comforts in the cabin, monitoring and measurement under the hood, and everything in between. There's such a myriad of cool electronics going on in automobiles, and it is fun to be a part of it.

Shaughnessy: It's a good time to be in the industry. A lot of people don't even know this industry exists.

Stevenson: That's the truth. When I started out of college, it was a job, but now it's a whole industry and career.

Shaughnessy: Are you a chemist?

Stevenson: I am. My first degree was chemistry, which is how I broke into the PCB industry. After about three years of keeping all of the chemical processes stable, I went back to school and got my chemical engineering degree. Then, I moved into the engineering realm of PCB manufacturing (process, materials, and quality). After several years there, it was time to go back to school again, where I moved to "the dark side" away from science and toward the business side with an M.B.A.

Shaughnessy: What do you see going forward? Do you have any predictions on where we're going in the next three to five years?

Stevenson: I don't know if three to five years is the right timeframe for it, but from what I see in the news, graphene seems to have a leg up as a potential replacement to copper for circuit boards. The chemist and engineer in me wants to dig in and figure out how to make that work, but I am sure there are already hundreds of smart people that are way ahead of me.

Shaughnessy: We see graphene news every couple of days.

Stevenson: It seems like graphene has that type of potential. It's probably too expensive and cost-prohibitive at this point to replace copper, but I think it will get there eventually.

Shaughnessy: Sunstone is in a good spot because you have a design tool, which is pretty cool. People can use it and get PCBs and Gerbers out of it.

Stevenson: PCB123 opens up electrical engineering to the hobbyist if you will; it's a free tool that is powerful and allows you to do some pretty complex designs if your heart desires. It also takes the guesswork out of ordering PCBs and is nicely integrated.

Shaughnessy: We see more and more of these free or cheap tools, like KiCAD and some of these things, and people swear that they're good.

Stevenson: It's funny that you say that. Recently, I did a blind study where I took every free or almost free tool that I could find available to download and gave them all a try. I'm not a designer by any stretch, but as a novice, I took each of those tools (15 in all) and tried to replicate an existing schematic. I found that some tools were intuitive and easy to pick up; their parts library was great, and I was able to do it pretty quickly. With other tools, the parts editor frustrated me. It's pretty amazing what some of these new tools can do, though, and they keep coming out. But for me, if the parts creation was too cumbersome, I gave up and moved onto the next tool. People are busy and don't have a lot of time to invest in a tool.

Shaughnessy: Some of them look like they were created by college kids that came up with a tool that can design a few layers.

Stevenson: Exactly. Software is going to be the wave of the future in a lot of aspects. Even in manufacturing, people are going to utilize software a lot more to make smart factories as efficient as possible.

Shaughnessy: Thanks for your time, Matt.

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

Chris Banton Is Bullish on New Tech

Feature Interview by Andy Shaughnessy
I-CONNECT007

Chris Banton is the director of marketing for EMA Design Automation. We discussed the need to train the next generation of designers, advent technologies like AI and machine learning, and why companies like EMA are using these advancements to help designers with today's complex PCBs.

Andy Shaughnessy: Chris, it's good to see you again.

Chris Banton: Thanks for having me, Andy. I appreciate it.

Shaughnessy: What's your view of the industry from your perspective as a design tools guy?

Banton: It feels like we're in a state of transition in our industry, a lot of which has to do with the industry's demographics; there seem to be many new people and perspectives entering the industry. There's a high demand for electronics in all types of products. Companies we talk to can't find experienced engineers fast enough for the work they have. It's an exciting time. I believe the real question is, "How do we make sure we're getting enough engineers interested in PCB design and hardware design who want to fuel this innovation?"

One way is to leverage new technologies in the design space. A lot of what we do in hardware design/EDA is a key driving factor in the computing infrastructure and cloud infrastructure needed for AI, machine learning, cloud computing, etc. However, as a design community, we're just starting to think about how we use these new technologies to improve the hardware development process. It's weirdly circular. How do we benefit from what we enabled? How do bringing on these new technologies make hardware engineering more attractive to a larger group of engineers?

Shaughnessy: You mentioned AI, which kind of goes back and forth. You have a lot of designers saying, "We're the AI." They don't want to see more AI in the tools. What do you think?

Banton: It all depends on how you use it. It's like autorouters: one designer might say, "This is great. I can put my brain into the tool. It will

figure out how to run it for me," while another thinks it will never do what they need it to do. A lot of it has to do with personal preference, and it also depends on the designs you have. AI is the same way; it's a tool, and it's all about leveraging previous experience to automate the future. It's not saying we're going to replace the designer because you need to have the designer help teach it. Then, you don't have to do repetitive tasks over and over because the system will understand the thought process the designer went through and how it would accomplish them.

We're in the early days of AI or machine learning, and it's exciting. How much can you get the computer to do to analyze so that the designer can get quick and insightful feedback? That's the trick, I think; if you can get AI to be additive and context-aware to help problem-solve, you are giving the engineer a powerful tool to leverage. If the system could say, "I know this is a DDR channel. I have previous designs I can instantaneously reference for how this was done, apply that to this board, and suggest or perform a route strategy," that provides real value, and I think that is very additive to the design process.

Shaughnessy: And they say most of the AI that is in some of the tools now is more like machine learning; it's not exactly what you would call AI.

Banton: The definition of AI and machine learning can be a bit muddy. There is a lot of excitement around these technologies, so I believe the terms tend to be used more loosely. What I find interesting is how the boards and hardware advances within the PCB design community are what helped to make these AI/ML platforms possible. We are right at the forefront of these technologies, and I'm curious to see how they take shape.

Shaughnessy: Right. It is almost at a tipping point, and in a good way, with all of these different technologies coming online, like IoT and 5G.


Chris Banton chats with I-Connect007's Barb Hockaday at PCB West.

Banton: Let's take electric vehicles as an example. We see start-ups pop up and go from zero to EV; they design and build it in an amazing amount of time. However, the challenge we see from an engineering perspective is that designers don't have as much time as they used to. Therefore, they need more from their tools to help them focus on these complex engineering challenges. For instance, they don't have the support of a component engineer and librarian to manage their parts for them; they often have to do this on their own. The hardware engineering community is more important than ever, yet they're asked to do more than they've ever done.

Shaughnessy: And you're talking about new companies. It's interesting how many companies don't fit into any category, such as companies that are brokers, but then they start buying manufacturing facilities.

Banton: The industry seems to vacillate. In the IC world, you have fabless, but now we're flipping back to systems companies. Apple, Google, and Amazon are all doing the whole stack because they can see value there. That changes how a company looks at their tools because they own the whole process. How do those all talk together? How do they work? How does the chip in the package connect to the board in an optimal fashion? That's the value they can provide to their customers that no one else can give. In a good way, it changes all of the conversations we're going to have because it's less siloed.

Shaughnessy: We see some new people coming into the design sector. Do you see that?

Banton: It's happening, but not as fast as a lot of people would like, and part of it is figuring out how to catch up with what a modern engineering PCB designer needs to know. We might be stuck in the old paradigm where you had an EE and a PCB designer. What they teach you in school is EE-specific, but not necessarily PCB-specific. Signal integrity is a specialist thing that's "black magic" to many. How do we take those things and teach them in a way that designers need to know today?

Shaughnessy: People joke about how Howard Johnson's book, *High-Speed Digital Design: A Handbook of Black Magic,* is everyday stuff now. If you still think it's black magic, then you have problems.

Banton: Yes, because your board has all of that. If you don't understand it, then you're asking for trouble, which goes back to how much is being put on the PCB design engineer these days.

Shaughnessy: Earlier, you talked about how tools have evolved. Twenty years ago, you could do flex, but you had to coax the tool. Now, EDA tools come optimized for flex.

Banton: It has become table stakes. The company thinks, "I need to get more functionality into a smaller and smaller case, and I need it to be cost-effective. I need it to be fault-tolerant." And flex is a great choice for that, but it presents some unique design challenges. Then, they say, "We have two rigid boards we're flexing. What if I want to print it on some kind of non-planar surface or if I want to have it totally flexible?" It starts changing the way someone might think about how they do these things.

Shaughnessy: What do you think about printed electronics?

Banton: Printed electronics make sense for many reasons. Companies are asking about the advantages of printed electronics, not just because they can do it. But how does this benefit the industry and business? What new design problems are created? It's like your analogy to how flex was several years ago. Printed electronics is in that mode where people are making it work, but they're forcing it; it's not natural, at least not yet.

Printed electronics is in that mode where people are making it work, but they're forcing it; it's not natural, at least not yet.

Shaughnessy: A lot of things are going on now. We hear lip service from China that the government is cracking down on people pirating software like Cadence's. Do you see any evidence of that?

Banton: I don't have a ton of visibility there, but I think as these companies become more global with offices in North America and Europe, they see how these groups are able to leverage not just the tools but the companies that support it. If you pirate software, you're not going to get help; you're not going to be able to collaborate with the vendor to drive the development of the product. There are a lot of inherent benefits they don't realize they're missing out on. There may be some temporary monetary gains. If that's how it's trending, that's a good thing. There are a lot of bright engineers in China, and we want them contributing to that community in a positive and meaningful fashion as well.

Shaughnessy: I understand Cadence has a program for students to get their hands on OrCAD.

Banton: Yes, we started a new program. We've had various versions of this in the past. I think the best way we can help students get excited about electronics and what we do is to offer them the tools for free. Now, students can come to the EMA website (or the OrCAD website), and as long as they have a student ID, they can request a free license to OrCAD and use it in their studies. The response so far has been overwhelming; it's exciting to see that there's so much activity in the student community, and people are looking to do electronic design and learn how to use the CAD tools to make them successful.

Shaughnessy: That works out for everyone. They get free tools, and you get customers in on the ground floor.

Banton: Sure, it's not totally altruistic, but it's important because engineering isn't just the tools. However, that is a big part of what we can do to help make them successful. Our goal is to assist future engineers by ensuring they are comfortable with the tools. This helps them be an effective engineer and not have to spend time learning the software to make it do what they need it to do.

Shaughnessy: Is there anything else you'd like to add?

Banton: It's a good time for the industry and to be in EDA tools.

Shaughnessy: Thanks for your time, Chris.

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

The Future of the World is Truly in the Hands of Our Youth

by Barry Matties

The investments we make in our youth now will pay off for generations to come. We have to support and help wherever we can. At I-Connect007, we are proud to once again sponsor the STEM Outreach Program at the upcoming IPC APEX EXPO 2020. This year, IPC plans to double the number of participating students to 200. It should be a great event.

Another way we support our youth at I-Connect007 is by sharing the stories of young people who are doing amazing things. At 14-years-old Dylan Nguyen is an avid kite flier in the master class and is often a featured flier at kite festivals along with his younger brother, Cardin, who is age 12. Dylan started kite flying about four years ago. Now, he and his family can be found participating in and volunteering at kite events.

Dylan is also a great student, musician, and creative thinker. Recently, he shared with us the details of his school science fair project, "Kite: Powering the Future," which solved a problem that he faced. Dylan is a leader with a bright future, and the world is lucky to have him and other kids like him. **Read his interview in the November issue of SMT007 Magazine.**

PCB007 Highlights

Chinese Review: The 2018 NTI-100 Top Global PCB Fabricators ▶

NTI released the list of the world's top PCB manufacturers in 2018, and this information is of great interest to the industry to determine what is going on in the industry and analyze the global situation. I will examine the currency exchange rate, company rankings, global distribution of the top-ranked companies, global total PCB output, and the automotive sector.

It's Only Common Sense: The Modern Salesperson ▶

It's a new world, and the modern salesperson has to keep up or get run over; that's just that way it is. Dan Beaulieu shares five steps that every great, modern-day salesperson—from age 24 to 74—should be doing to be successful today.

Standards: Why We Have Them and Live by Them ▶

Have you ever designed a board but received feedback that it couldn't be manufactured unless changes were made? Or maybe you've designed a complex board and sent it to the factory only to find out that the manufacturer didn't build the board to your expectations? NCAB Group explains how PCBs are becoming more complex, factory options are growing, and expectations for product life cycles are becoming longer.

Printed Circuits Installs New Notion n.jet Direct Solder Mask and Legend Printer ▶

Rigid-flex circuit board manufacturer Printed Circuits has purchased and will install a new Notion Systems n.jet direct solder mask and legend ink printer.

Prototron Receives MIL-31032 and AS9100 Certifications ▶

Kim O'Neil, general manager of Prototron Circuits in Tucson, Arizona, discusses the company's recent MIL-31032 certification and how this experience prepared them for the AS9100 certification. He also explains why auditing is a good thing for any company's processes and highlighted some of the areas that the auditors inspected.

AT&S and Chongqing University Collaborate on Research and Education ▶

It represents a milestone for AT&S as a first high-end IC substrates manufacturer in China partnering with one of the universities affiliated to the Ministry of Education of China in research and education excellence.

Communication, Part 2: Design Data Packages ▶

Part 1 of this six-part series highlighted ways that PCB fabricators and designers can better communicate, starting with how to qualify a board shop. In Part 2, Bob Chandler and Mark Thompson talk about the importance of preparing, sending, and receiving comprehensive (and ideally, perfectly complete) design data packages.

The Plating Forum: Update on IPC-4552 ENIG Specification Revisions ▶

George Milad's columns will cover PCB plating, IPC specifications, and more. In this debut installment, he gives us an update on the IPC-4552 ENIG specification, including Revision A and B.

Communication Is Still the Key

Tim's Takeaways

Feature Column by Tim Haag, CONSULTANT

First Page Sage

There were times when I thought my dog Henry was part Vulcan. I would hear about how other dogs would bark at the front or back door to go outside or bark at their empty food dish, wanting dinner, but Henry would walk over to me and stare. After a while, I began to seriously wonder if he had some sort of telepathic thing going on like Mr. Spock on Star Trek. If so, I must have been too dense to get the message because the only thing I ever received was a feeling of unease at his creepiness.

Sometimes, Henry's intent was obvious. For instance, if I was eating something, he would invade my personal space and stare intently at my food in hopes of getting some leftovers (Figure 1). But most of the time, he would lock onto me with this zombie-like stare, which was absolutely unnerving. I tried to ignore him until I couldn't stand it any longer; then, I would finally yell at him in exasperation, asking, "What do you want?"

Did he want a treat, or did he need to go outside to take care of his personal needs? Did he want to play, or was he out of food or water? Perhaps the cattle had escaped from the pasture while the barn was burning down, and a little boy had fallen into a well, but probably not since we didn't have a barn or a well, and he wasn't Lassie.

I was taught that communication takes place in the mind of the receiver. Just because we think that we have clearly expressed our thoughts and ideas doesn't necessarily mean that we have successfully communicated. If the person we are talking to doesn't understand what we are trying to say, communication hasn't taken place. I'm sure that Henry thought that I was a complete idiot for not understanding what he wanted, but without a clear message from him, he wasn't going to get through to me.

Figure 1: Sometimes, there was no doubt about what Henry wanted.

This month, *Design007 Magazine* is presenting a variety of voices from our industry. You will find various accounts about how our industry has changed, what new changes are on the horizon, and what will drive those changes. These are exciting times for all of us as emerging technologies increase the complexity of PCB design and necessitate new tools and methodologies to get the job done. But no matter how much the design, tools, or the next new "thing" changes, there will always be one very human element that can get in the way if we aren't careful with it: communication. And while not understanding the zombie-like stare of a family dog can be funny, missing out on the important communication that takes place at work can result in disaster.

So, along with listening to the different voices of our industry, let's also look at five ways that we can make sure that our own voices are saying what we want them to say.

> **No matter how much the design, tools, or the next new "thing" changes, there will always be one very human element that can get in the way if we aren't careful with it: communication.**

1. Communication Should Be Clear and Complete

The first thing to consider for clear communication is if we are giving people everything they need for success. For instance, I have listened to many PCB manufacturers lately, and one common theme among these industry voices is that they are not getting consistent and accurate data with which to build circuit boards. Missing files, inconsistent data formats, and even different units of measurement are some examples of the problems they see, which results in frustration for the manufacturer as well as for the designer who doesn't get the results they expected. Fortunately, there are some things that we can do to help with communication problems like these, such as encouraging open relationships that foster dialog and questions and making sure that we prepare ahead of time and have all of the necessary information. There are some technological improvements with the tools that can specifically help with manufacturing as well.

More and more manufacturing outputs are changing from a collection of differently formatted files to a single-file solution. This ensures that consistent, accurate, and complete data is delivered to manufacturers for fabrication and assembly. The IPC-2581 file format, for example, contains all of the information that was previously sent out in individual Gerber, NC drill, drawing, netlist, and README files in one combined file. Additionally, the IPC-2581 format is bi-directional, giving manufacturers the ability to send important board configuration data to their customers first. By providing all of the necessary data for success, communication between PCB designers and circuit board manufacturers can be clearer now than it ever has been before.

2. Keep Everyone in the Loop

Even though you work in the same company or department, communication between team members has always been a challenge. It is very normal to think that everyone is on the same page when the person next to you might be in a different book altogether. Remember when we team designed PCBs by working around the clock in multiple shifts? There was an enormous potential for miscommunication between shifts, and serious problems crept up that often wouldn't be discovered until the following day.

Design notes, whiteboards, README files, and even the occasional phone call to an off-shift employee would help, although my ears still hurt from some of the tongue lashings I received from co-workers over the phone at 1:30 in the morning. However, all of this underscores the need for better communication between co-workers. Thankfully, there have been some new enhancements in PCB design technology that can help.

The team design concept has been around for a long time, but recent enhancements to team design tools are breathing new life into this process. Many PCB design systems now have methods for enabling multiple layout designers to work on the same design at the same time. This used to be a laborious process of writing out portions of the layout for each team member to work on and then merging the files back together again at the end of the day. While helpful, this also created many opportunities for confusion and error.

With the latest advancements in team design software, however, multiple designers can work together without fear of conflicting with each other's work. No longer do designers have to work on their portion of the design in the dark. Instead, each team member works together in the same environment allowing for real-time communication throughout the entire team.

3. Resist the Urge to Assume

Making assumptions about how others should do something is perhaps one of the largest communication traps that we can fall into because we know how we would approach a problem; therefore, we assume that others will perceive and resolve the problem in the same way.

Consider for a moment this simple example. Many years ago, on an older CAD system, I would create a surface-mount pad by drawing a polygon, yet my co-workers used line draws instead for the same task. The polygon was the preferred method to enable accurate DRC checks, and I assumed that they didn't care. But when I finally found out what was going on, I was pretty embarrassed to realize how wrong my assumption was. The simple truth was that no one had ever told them differently.

There was probably a lot of wasted work because I made a foolish assumption instead of following up on the problem by asking a simple question; it was simply a training issue. When we resist the urge to assume and look beyond the issue, sometimes we'll find out that there is more to the story than we realized.

Think about the co-worker that is habitually late; are they lazy, or are there problems at home that no one knows about? How about the vendor that always messes up your order; are they bad at their jobs, or are you not giving them enough information? My mom used to tell me that when I was angry at someone to close my mouth and count to three before I did anything. The older I get, the more I see the wisdom behind those words.

4. Document as Much as Possible

Those of you who have read my columns before know that documentation is one of my favorite communication drums to beat, and for a good reason. Without documentation, there isn't an accurate way to track work in progress, nor is there a reliable way to show that you did or didn't do those things that are required for your job. Instead, a well-developed documentation system can be very helpful. Some examples might include:

- **Design Processes:** Documented processes will provide a guideline to all for expected data input, design milestones, reviews, and department signoffs.
- **Change Requests:** Have a system in place where changes to design, manufacturing, or any other process are recorded and signed off by the appropriate participants.
- **Project Tasks:** Having a system that allows you and other team members to track milestones for different projects is a great way to keep team members, management, or customers updated.
- **Personal Tasks:** If you are dealing with requests that are outside the normal project workflow, it is important to keep track of what you are doing and for whom.

5. Listen to Yourself

There are many reasons for communication problems between people, and if you are having difficulty in trying to figure out what is wrong, take a moment to first listen to yourself. Put yourself in the place of your audience, and listen to what you just said. Does it make sense to you? Do any questions immediately come to mind? Did it give you enough information to successfully complete the task that needs to be done? By first listening to ourselves and looking to see how we can improve our own voice, we can do a lot better job at communicating our ideas with others.

Conclusion

It would be a lot easier if we could telepathically communicate with people as my dog thought he was doing, but let's face it—that just isn't going to happen. For now, we all need to work on improving our communication skills to make sure that the message we want to send is the message that is actually being received. Until next time then, keep on designing. **DESIGN007**

Tim Haag writes technical, thought-leadership content for First Page Sage on his longtime career as a PCB designer and EDA technologist. To read past columns or contact Haag, click here.

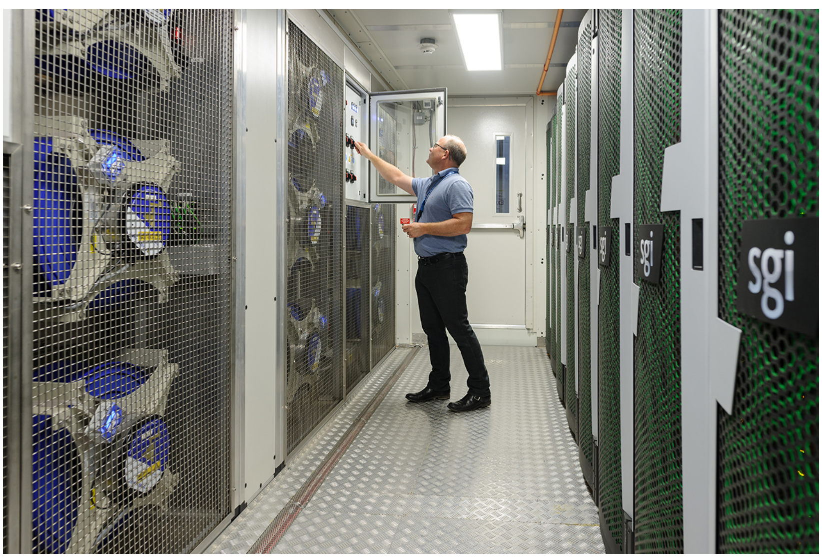
Google and NASA Achieve Quantum Supremacy

Google, in partnership with NASA and Oak Ridge National Laboratory, has demonstrated the ability to compute in seconds what would take even the largest and most advanced supercomputers thousands of years, achieving a milestone known as quantum supremacy.

"Quantum computing is still in its infancy, but this transformative achievement rockets us forward," said Eugene Tu, center director at NASA's Ames Research Center in California's Silicon Valley. "Our missions in the decades to come to the Moon, Mars and beyond are all fueled by innovations like this one."

The paper describes the experiments run by Google's Sycamore quantum processor to demonstrate quantum supremacy. Computations on a quantum computer are called "quantum circuits." These computer science abstractions work like programs, specifying a series of operations for the quantum processor to run.

Both the quantum processor and supercomputer were given increasingly complex and random circuits to compute until the supercomputer wasn't able to process them. To find that limit, researchers at Ames advanced techniques for simulating these random quantum circuit computations using NASA's supercomputing facilities. At a certain point, even with all the tricks NASA's quantum computing and supercomputing experts threw at it, this simulated "computer within a computer" wasn't able to handle the random circuits given to it, and that became the bar set for Google's quantum computer to beat.



Reaching quantum supremacy opens up the ability to experiment and develop quantum processing technology far more rapidly across the board—largely because of the unprecedented degree of control over the quantum operations possible in Google's hardware.

The achievement of quantum supremacy means that the processing power and control mechanisms now exist for scientists to run their code with confidence and see what happens beyond the limits of what could be done on supercomputers. Experimentation with quantum computing is now possible in a way it has never been before.

(Source: NASA)

Tamara Jovanovic Checks in After First Year as a Designer

Tamara Jovanovic

Feature Interview by the I-Connect007 Editorial Team

Andy Shaughnessy and Nolan Johnson caught up with Tamara Jovanovic, a PCB designer they met last year at AltiumLive in San Diego as a newly minted engineering graduate. She discusses her first year in the industry and explains how education and training events can help shape the careers of new designers.

Andy Shaughnessy: When we spoke one year ago, you were a newly graduated PCB designer and engineer. As an update to last year's interview, how has your first year been going?

Tamara Jovanovic: It has been great, and Happiest Baby is an awesome company. We make a smart baby bed that moves and plays sounds to soothe babies. The baby is placed in a special sleep sack, which clips into the bed; it's a safety feature so that they can't roll over. And the effect has been incredible. It has helped so many babies, parents, and families, and it has been great being part of that.

Shaughnessy: Is there a typical day for you?

Jovanovic: Not really; there's always something new that happens. We're currently working on a lot of projects and tests. Every day brings new challenges that I can try to solve, which I enjoy.

Shaughnessy: Are you designing boards that are going into some of the controllers and things that will make the crib start rocking or signal the parents that the baby is in trouble?

Jovanovic: Yes. The high-tech bassinet has a lot of electronics that make it work the way it does. For example, the current version of the bed has microphones inside, which will sense when there's a cry inside the bassinet. If the baby starts crying, the bed will automatically increase the level of motion and white noise to soothe the baby so that the parents don't have to get out of their beds in the middle of the night. It also works with an app. It's compatible with Android and iOS. You can change levels of movement, set different sensitivity levels, etc. We want to be able to let you tailor it to your baby's needs.

Shaughnessy: You're an electrical engineer, correct?

Jovanovic: Yes.

Shaughnessy: But you didn't have a lot of PCB design experience last year.

Jovanovic: Right, I didn't have any. When I started working last year, I realized that's something that I needed to get into, and this conference was a great starting point. I picked up many tips and tricks from experts at the conference and talked to a number of people who have a lot of experience. They gave me pointers, and it was a good base to get started. When I began designing boards, I mainly taught myself, and Altium's documentation online was of great help as well. My experience this year is completely different than last year at AltiumLive. I feel like I have a better understanding of a lot of things, I'm talking to people in the industry, and I'm sharing my experience.

Shaughnessy: How much did your degree prepare you for this?

Jovanovic: College prepared me for the hard work and the design part. Engineering was not an easy major, but a lot of the stuff that I'm doing at my job right now I learned in college. It was on a smaller scale, but now I feel like it has more impact and value because Happiest Baby has such a great mission. We had a very intense lab course in college that prepared me best because it was very demanding. I appreciate it a lot now; it definitely prepared me better for the real world.

Shaughnessy: What are some of the bigger challenges that you've seen in the last year or so? Did anything stump you?

Jovanovic: I definitely ran into some roadblocks, but I was able to figure it out with Altium's documentation online and other design resources. I didn't do any super complicated designs this year. Obviously, there are a few things that frustrate you and some things you wish you could change, but not particularly about the industry.

Jovanovich speaks with Editor Nolan Johnson at her second AltiumLive event.

Shaughnessy: So, you didn't have any boards fail the smoke test?

Jovanovic: Not yet. There have been some minor ones, but luckily, it wasn't due to board design. There have definitely been some interesting times in the lab during the proof-of-concept stage. During the whole fabrication process, I'm always a little stressed out, wondering if my design will work without issues.

Shaughnessy: Have you been able to see any of it assembled or fabricated?

Jovanovic: Yes. We're still in the prototype stage right now, so we're only building a few pieces at a time and ensuring that it all works before we go to production. I've been lucky that all my designs have been working or borderline working with minor improvements.

Shaughnessy: You mentioned last year that the company's founder and CEO is a famous pediatrician.

Jovanovic: Yes, his name is Dr. Harvey Karp, and he wrote a book called The Happiest Baby on the Block about soothing babies, where he discusses his method—the "Five S's"—to soothe babies: swaddling, side/stomach position, shushing, swinging, and sucking. The book did really well and is still popular, and his method has helped many people.

Shaughnessy: Does the bed plug into the wall or use batteries?

Jovanovic: You plug it into a wall and it rocks your baby all night long. This January, we started a rental program for SNOO. You can rent it for however long you want at an affordable price. Some people don't know if they're going to have more kids or when. It's a bassinet, and you're only supposed to use it from zero to six months; now, they can rent it and not worry about any of the stuff that comes after.

Nolan Johnson: This is your second AltiumLive. We talked to you a year ago as you were showing up with a deer-in-the-headlights look. Now that you've had a year of practical experience doing this kind of work, what's different?

Jovanovic: This year, the conference is a completely different experience. Last year, I was a little overwhelmed by what I didn't know, but I knew it was something I needed to start doing for my company, so I was ready to learn. The event was such a good place to get a basic understanding, and I learned a lot of good tips and tricks. I talked to the people that I met in the lectures, and they helped me understand the basics of the program. Those were the building blocks of my knowledge in PCB design. This year, I knew more about what was being presented. I understood more and had specific questions, and I found other people asking me questions about how to do certain things in Altium.

Johnson: And you had answers?

Jovanovic: Yes, which was weird to me; I could respond and help them. It's a nice reassurance that I've learned something compared to last year.

Johnson: Do you think you made the right choice with this profession?

Jovanovic: Absolutely. I've always liked science. Everyone always asks, "Why engineering?" Even with all the homework, all-nighters, and hard times, the big picture was that I enjoyed learning it. Now, I get to apply that experience into something meaningful. We're saving babies' lives! I'm lucky to have this job and have gotten to know all these incredible people on the way.

Shaughnessy: Did you keep up with the people you met last year?

Jovanovic: I didn't, but I've seen a bunch of people that I saw here last year, and we picked up right where we left off. It's great to reconnect with the people from the industry.

Shaughnessy: And it's good that you work for a company that will send you to educational events.

Jovanovic: I'm very lucky to have that support and to get the opportunity to learn. My boss was the one who suggested I go last year, and when I asked him about it this year, he was really cool about it.

Johnson: What would you tell a new hire that they need to know about PCB design?

Jovanovic: Be confident that your design will work. I'm very self-critical, so when something's wrong, I always tend to think that it's my fault. Also, do your research and use all the software available to you. In one of his lectures, John Magyar talked about impedance calculations. He said, "You don't even have to do calculations yourself now because Altium will do it for you." You can save yourself a lot of time if you understand all the resources available to you.

Also, attend conferences and lectures like this if you can. Connect with people from the industry and ask what their tips and tricks are because then you can use them and share them with others as well. It's awesome when engineers share their secrets. I wish I had more experience so that I could share it and help more people because it's much easier when someone shows you one on one rather than having to go through the book and do it step-by-step by yourself.

Shaughnessy: Do you ever get the chance to talk to young people and explain how cool your job is?

Jovanovic: Not really. I keep in touch with some of the younger engineering students at my school. We had a close community because we were a smaller program; we all knew each other.

Shaughnessy: I think you'll find that designers usually want to help other designers.

Jovanovic: I've discovered that, too. When you're working on a complicated design, and you can't find any documentation online, other designers will want to help you. It's nice to know that there's camaraderie between designers and that we're not alone; we're all going through this.

Shaughnessy: Thanks for the update, Tamara. It's always a pleasure talking with you.

Jovanovic: Thank you. Likewise! DESIGN007

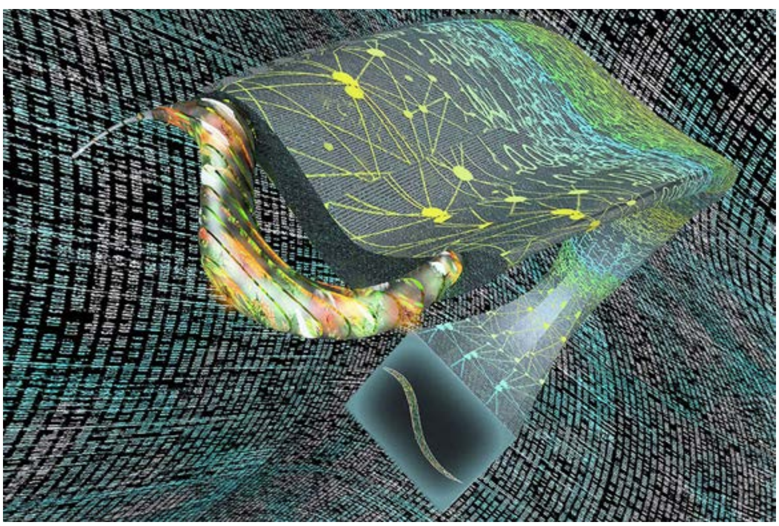
Researchers Convert 2D Images Into 3D Using Deep Learning

A UCLA research team has devised a technique that extends the capabilities of fluorescence microscopy, which allows scientists to precisely label parts of living cells and tissue with dyes that glow under special lighting.

In a study published in *Nature Methods,* the scientists also reported that their framework, called "Deep-Z," was able to fix errors or aberrations in images, such as when a sample is tilted or curved. Further, they demonstrated that the system could take 2D images from one type of microscope and virtually create 3D images of the sample as if they were obtained by another, more advanced microscope.

"This is a very powerful new method that is enabled by deep learning to perform 3D imaging of live specimens, with the least exposure to light, which can be toxic to samples," said senior author Aydogan Ozcan, UCLA chancellor's professor of electrical and computer engineering and associate director of the California NanoSystems Institute at UCLA.

Deep-Z was taught using experimental images from a scanning fluorescence microscope. In thousands of training runs, the neural network learned how to take a 2D image and infer accurate 3D slices at different depths within a sample.



Ozcan and his team showed that their framework could then use 2D wide-field microscope images of samples to produce 3D images nearly identical to ones taken with a confocal microscope. This conversion is valuable because the confocal microscope creates images that are sharper, with more contrast, compared to the wide field.

(Source: UCLA Newsroom)

Megan Teta

The World of PCBs: Anything But Boring

Feature Interview by Andy Shaughnessy
I-CONNECT007

I had the chance to catch up with Megan Teta, CID+, product manager of design and education services at Insulectro. Megan explains why she's excited to become more involved in training and why the world of PCBs is anything but boring, contrary to popular opinion.

Andy Shaughnessy: For anybody who didn't see last year's interview, give us a quick background on yourself and talk about your promotion. Congratulations, by the way.

Megan Teta: Thank you! I've been in the industry for five years, and a little over two and a half years with Insulectro. I received a promotion to product manager of design and education services. Before that, I was a field application engineer on the East Coast, which I'm continuing to do. Currently, our goals with PCB design and education are to get more designers informed on anything and everything, mostly focusing on materials set aside for the manufacturer, etc. We also partnered with two design services companies, and we work very closely with them. We make sure that anybody who has a bottleneck in their design can get through that, so that has been our primary focus.

Shaughnessy: You're not an old, grizzled veteran yet, but what were some of the things that surprised you about this industry?

Teta: People don't realize how many different aspects in which you can become involved. From day to day, I deal with a lot of the RF and high-speed digital stuff, along with aerospace and automotive. The different areas our PCBs go into is so large across the spectrum, and you need to learn exactly what type of applications they need to go through, whether that's high-temperature or down-hole drilling technology. Any electronic device you use has a board in it. We're going to 5G because circuit boards are going to be capable of that bandwidth. I'm still surprised at how involved PCBs are in every aspect of your life; it's not just the bare boards.

Shaughnessy: I have a lot of friends who think this sounds like a boring industry. Maybe the boards are, but the things they go into will change the world.

Teta: And depending on what area you're in, you can be involved from the design and innovation aspect all the way through assembly. Some people I work with deal with packaging hardware and software, so you can learn about every aspect of it if that's what you want to do. There are so many different things. If you decide you wanted to get involved in the manufacturing side, there are a million different processes, so it's not limited to one area. As soon as you get in the door and learn from some key people, you can do almost whatever you like. Even the marketing side of this segment is completely different from typical marketing, so you should never get bored.

Shaughnessy: When you were in college, did you imagine you'd be working with circuit boards?

Teta: I did not. I was convinced I was going to do something with chemistry, which I did in a roundabout way for a little while. I became involved with Insulectro and materials as well as manufacturing processes. But, as I mentioned earlier, I've been focusing on the design side and completed the CID and the CID+ course to make sure I understood exactly where the designers were coming from and what information they need to know so that I can talk with them more easily.

Also, we recently started working with Mike Creeden, who is a master trainer for the CID and the CID+ courses for EPTAC. We're working with him and their team to do training sessions on materials at the front or back end of his classes. That's very exciting and goes along with our education portion to make sure that young designers, and even people who have been in the industry for a while, understand the basics of the materials.

Shaughnessy: It seems like we see more women now in this industry, and a lot of the speakers in PCB design are women.

Teta: Yes. As an industry, we're doing a good job trying to get more women involved. Women in Electronics is a new organization that started less than a year ago. They conduct professional training for women in electronics, making sure that they have access to professional development, which has been awesome. They have several branches across the U.S. There's a Boston branch that I'm going to get involved in and several on the West Coast as well. Several customers have a lot of women that are in lead engineering positions and doing a lot of different things, such as NPI, so we're getting there. It's still a male-dominated industry, but we're slowly infiltrating. And more people are looking at me as an equal. Occasionally, I'm still asked if I'm in marketing, but more and more, they assume that I'm a technical resource. People reach out to me that wouldn't have a couple of years ago. I've proven myself now within the industry.

Shaughnessy: Do you ever speak to young women who are surprised to see that you're doing this?

Teta: Yes, and it can be kind of funny. I was at a Jets game this weekend with my whole family, and I always enjoy talking to some of my female cousins; they think it's neat that I'm involved in electronics in general. It's always going to seem like a more technical field; it doesn't have to be, but there's definitely some excitement.

Another cool thing is that engineering toys are coming just for little girls, which are focused on chemistry, etc. One toy is called GoldieBlox. Further, I see more T-shirts with science references on them in the girl's section of Target, etc. We're getting past the idea that women need to be secretaries, etc., and men get to do all of the technical stuff; there are technical options available for women, which is awesome.

Shaughnessy: What's your favorite part about the industry?

Teta: I've enjoyed getting involved in every aspect, from design and front-end engineering through the manufacturing process, including troubleshooting there and after all the manufacturing assembly is done. I like looking at sections of boards and saying, "Maybe if we change this, it would be a little bit easier to build." Working with Isola's materials always makes it easier for DFM, so we've been able to switch some things with that. This is the time of year where we do a lot of training events. We just did one with 25 design engineers, so we're hoping to hit 100 before the end of the year.

Shaughnessy: Are you doing any of the talking or teaching?

Teta: Not right now, but we've talked about it. I will be doing the "lunch and learn" training on materials. I spend a lot of time training on different items, such as copper foils, laminate materials within board tops, and resists.

Shaughnessy: What would you like to change about the overall industry if you could?

Teta: Probably keeping more manufacturing in the U.S. I think these tariffs are going to move more things back to the United States and change people's perspectives on Chinese manufacturing. There are still so many boards built in the U.S., especially for DOD medical, so there is a market for it. A lot of board shops are investing and making sure people understand that you don't have to go overseas for that. This year, many of our fabricators have been investing a lot of CapEx into new equipment, such as LDI or additive, or to make smaller lines and spaces. We still haven't hit our limitations to manufacturing.

Shaughnessy: Are you doing additive stuff now?

Teta: From the printed electronics side of it, we do a lot of additive and inks being printed on and used. This is a lot of lower-temperature stuff for the most part. DuPont has some high-temperature, co-fired stuff that's also printed. I'm not involved in it personally, so I can't speak to it, but different parts of the industry are moving that way.

Shaughnessy: Have you gone to many IPC standards meetings?

Teta: I have not been involved in the standards meetings recently. I've been on some committees in the past that ensure all the information on standard is correct. I need to be more involved in them; it's one of the other things on my ever-growing list that I could be doing. Everyone's a volunteer.

Shaughnessy: And you still have to do your day job.

Teta: Exactly. More people need to be involved. IPC, suppliers, and fabricators have a lot riding on it. So, it's making sure the right people are making those decisions where it gets a little bit trickier. Having everyone on board is also a little difficult. IPC does the best they can, but they need to get more people involved.

Shaughnessy: I've covered the meetings, and some of it is tedious, arguing over one word because it won't translate into Spanish or Mandarin. But it has to be done, and the volunteers do a great job.

Teta: I know. It's tricky because it has to be translatable, as you stated. And I'm not saying I could do a better job. It's never going to be perfect, so make sure you're on board with the assembly house and the manufacturing houses. Specifications are important, especially as we get into tighter geometries.

Shaughnessy: It's nice seeing you again, Megan. Thanks for your time.

Teta: Thank you, Andy. It was good to see you too. DESIGN007

Curtis Scott Opens the Blinds on PCB Design

Feature Interview by Andy Shaughnessy
I-CONNECT007

I recently spoke with Curtis Scott, a young PCB design engineer with Current Products, a manufacturer of automated window coverings in Pensacola, Florida. I'm always excited to hear about OEMs in Pensacola; I lived there for 17 years, and it's a great town, but you won't find much industry other than tourism. I asked Curtis to share his thoughts on the industry and his job designing PCBs in that scenic city on the Gulf of Mexico.

Andy Shaughnessy: Curtis, why don't you start with a little background about yourself and how you got into circuit boards?

Curtis Scott: I'm an electrical engineer. I went to school at the University of West Florida and graduated in 2012. I started as an intern with another company, QMotion, designing boards, and doing general electronic testing. I knew this was something I liked, and I enjoyed the consumer electronics side of things. I stuck with it, and eventually, the owner sold the company and started a new one, so I went with him to Current Products. I'm enjoying making products for them.

Shaughnessy: Tell us about Current Products.

Scott: Current Products is a home automation company. We primarily develop and manufacture window coverings and automated drapery, such as retrofit window blinds.

Shaughnessy: So, you're designing boards for the motors that make the blinds open and close?

Scott: Yes, I design most of the boards that we do. We have remotes, hub devices, and controllers for the drapery motors. And then we have other people who do programming, make all of the mechanical parts, and assemble it all together.

Shaughnessy: That's cool. Did you know you wanted to work with boards when you were getting your degree?

Scott: Once I started in my degree, it piqued my interest. Communications didn't really pique it, and power didn't really either, but the consumer electronics side of things did.

Shaughnessy: Is the only job you've ever had in the industry?

Scott: Pretty much.

Shaughnessy: How did the company find you?

Scott: The company is big on hiring interns; we hire a lot of them. Many decide they either want to do something else or they just leave, but if they're good and we like them, we hire them. With interns, you constantly get a fresh pool of people, which keeps the company full. I came in as an intern under more experienced engineers, and have been there ever since.

Shaughnessy: I'm happy to hear that there's innovation like this going on in Pensacola. What do you like most about the industry or your job?

Scott: In general, home automation is a very quick-moving industry right now. Even in window coverings, there are constant changes. At the old company, we did roller shades, which are constantly evolving. I think we helped push that forward with other companies in the industry. Now, we're into track draperies and rod draperies. We're hoping to push that market forward too.

Shaughnessy: You were able to stay in your hometown, which a lot of Pensacola people will tell you is hard to do. I had to move to Atlanta. Are you the only designer in the company?

Scott: We have a few interns, as I said, which helps. But for the most part, I design the boards that we use from start to finish. I figure out what's going to be on the board, how it's going to work with the program, and how it's going to work with the mechanics. I lay out the board, send the board out to be manufactured, get it tested, ensure everything's working, and prepare for production. Then, once it gets to the production stage, we hand it off.

Shaughnessy: You send it out to be fabricated, but you do the assembly?

Scott: We assemble the final components; the board itself is normally assembled by the board house. But once we get the board in, we assemble the whole product.

Shaughnessy: What are some of the changes you see in your technology?

Scott: The ZigBee controller is a little older at this point, but it's still growing. Hopefully, we'll stay innovative with things like Bluetooth and Wi-Fi. There's a lot going on, even in window coverings.

Shaughnessy: Do you have automated blinds at your house?

Scott: I do. I tested them out when I first started, and I like them.

Shaughnessy: That's cool. How many people are in the company?

Scott: We have about 25 people. We're just now getting into the full swing of production, so we're going to slowly build up production and a call center with more jobs. The nice part is we've been an R&D company up to this point. As I said, it's growing quite a bit still, and probably will continue to grow for the next two years.

Shaughnessy: Do you ever go talk to students at UWF to get young people into the industry?

Scott: Occasionally, I'll attend job fairs, talk to students, and find out what work interests them. Working with all the interns, we also get to see the direction that the school is going.

Shaughnessy: That's great to hear because there's not a lot of industry in Pensacola; most people I know are lawyers or in hospitality.

Scott: It's picking up slowly. There are only a handful of engineering companies in the area. Hopefully, we'll see more industry soon.

Shaughnessy: Thanks for your time, Curtis.

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


DAVE CLARKE
R&D MANAGER,
VOX AMPLIFICATION LTD
I WAS LUCKY ENOUGH TO TAKE MY LOVE OF ENGINEERING, ELECTRONICS, MUSIC, AND PUT THEM TOGETHER.
Watch Dave's Story on
Altium's YouTube Channel
Altium.

The Frequency Domain

Beyond Design

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

As system performance requirements increase, the PCB designer's challenges become more complex. The impact of lower core voltages, higher frequencies, and faster edge rates has forced us into the frequency domain. At first, signal integrity (SI) can look quite daunting, but if we take the time to absorb the key concepts, then it is like visualizing a multilayer PCB from a different perspective. In this month's column, I look at the frequency domain.

Perhaps one of the most fundamental steps in the process of gaining proficiency in high-speed digital, RF, and microwave design is learning to think in the frequency domain. For most of us, the vast majority of our early experience with electrical circuits and signals remains within the context of voltages and currents that are either static or dynamic with respect to time.

Digital design, on the other hand, is a world of frequencies, so we need a different paradigm. The frequency domain can provide valuable insight to understand and master many SI effects, such as impedance, lossy transmission lines, and the power distribution network (PDN).

In the time domain, the system is evaluated according to the progression of its state with time. In the frequency domain, the system is analyzed according to its response for different frequencies. In a linear system, a transformation (usually Fourier transform) can convert the model into the frequency domain from the time domain. The system is changed from time to frequency to make it easy to understand the response of the system because the time domain is more complex for higher orders.

Put simply, a time-domain graph shows how a signal changes over time (Figure 1), whereas a frequency domain graph (Figure 2) shows how much of the signal lies within each given frequency band over a range of frequencies (bandwidth).

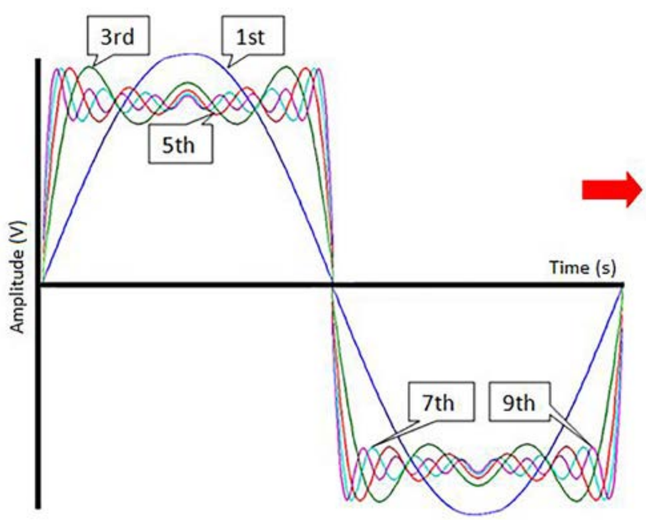


Figure 1: A square wave created by odd harmonics.

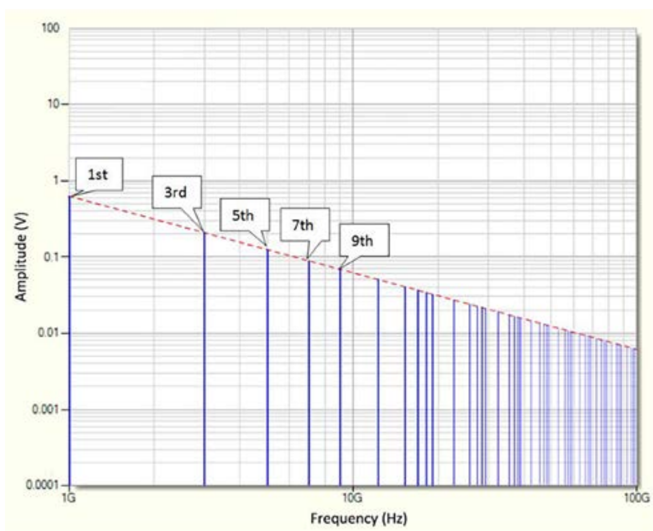
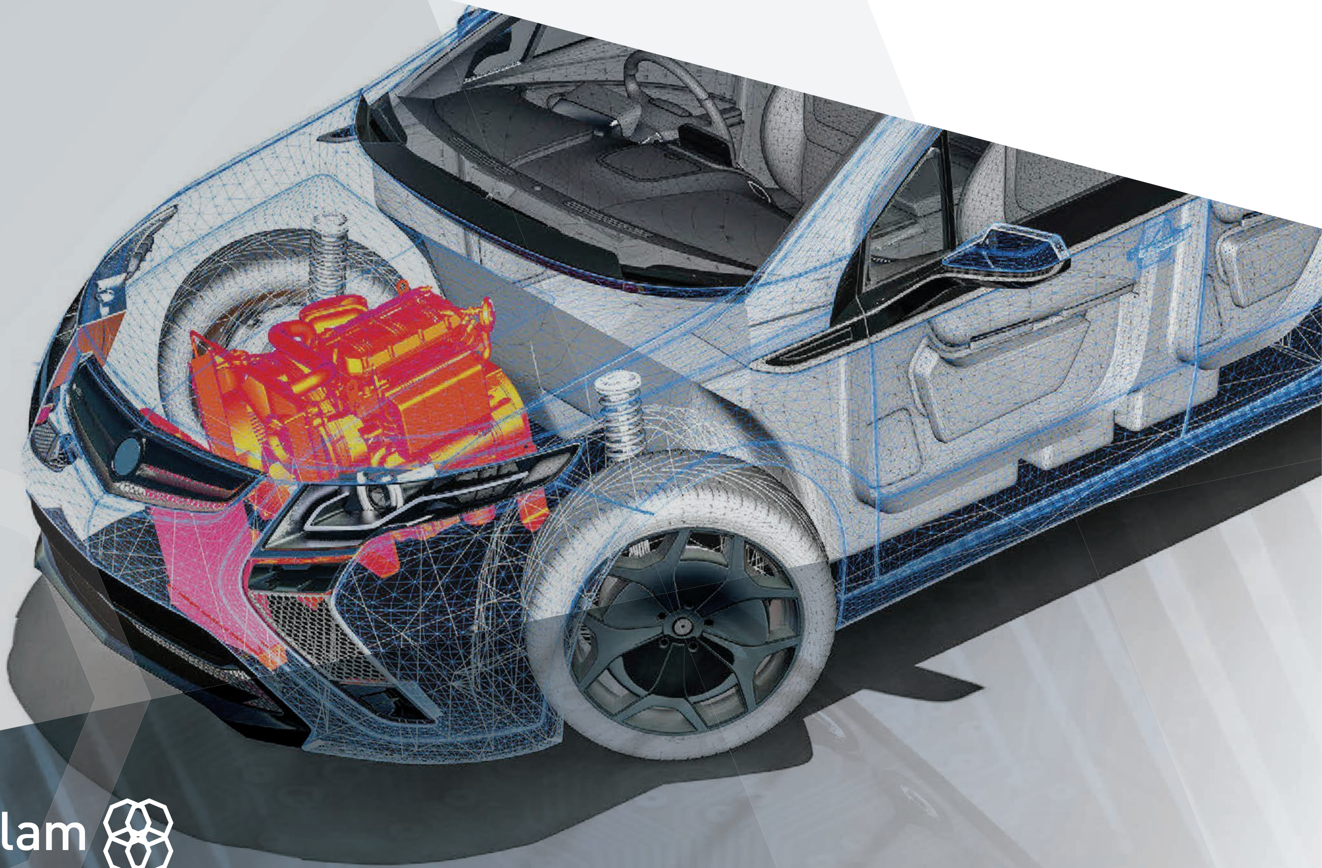


Figure 2: Frequency transform of a square wave.



The Fourier Theorem states that every function can be completely expressed as the sum of sine and cosine waves of various amplitudes and frequencies. The Fourier series expansion of a square wave is made up of a sum of harmonics. If the waveform has an even mark to space ratio, then the even harmonics cancel. Also, as the frequency increases, the amplitude decreases.

A square wave can be expressed as Equation 1:

Equation 1:

$$F(t) = \cos(\omega t) - \cos(3\omega t)/3 + \cos(5\omega t)/5 - \cos(7\omega t)/7 + \cos(9\omega t)/9$$

Any waveform in the time domain can be completely and uniquely described by a combination of sine waves. If we switch to the frequency domain and use sine wave descriptions, we can solve problems faster than in the time domain.

Impedance is defined in both the time and frequency domains. However, it is far easier to understand and apply the concepts of AC impedance in the frequency domain. Decoupling capacitors are spread throughout the PDN. Thinking in the time domain, you can say that decoupling capacitors store and supply charge on-demand to the loads. However, in the frequency domain, decoupling capacitors also lowers the impedance at different frequencies to help to meet the AC impedance target. So, there are two distinct functions of capacitors that work in unison but in different domains.

Further, when dealing with electromagnetic compatibility (EMC) issues, both FCC/CISPR specifications and methods of measuring the emissions of the product are more readily performed in the frequency domain.

The Fourier series expansion of a square wave is made up of a sum of harmonics. Figure 3 shows the conversion of a square wave from the time to the frequency domain and the resultant amplitudes of the frequency components. If the waveform has an even mark to space ratio, then the even harmonics cancel. The Zeroth harmonic is the DC value, and the fundamental (first harmonic) frequency is the largest in amplitude tapering off as the odd harmonic frequency increases. If the AC waveform is offset (from zero), then the DC voltage component will appear at DC (0 Hz).

The high-frequency content of a square wave is significantly affected by the rise time of the waveform. A fast rise time results in higher frequency components. Also, as the frequency increases, the amplitude decreases. In the real world, one needs to consider the maximum bandwidth of a signal, including harmonics, rather than assume the perfect square wave fundament frequency model. For example, a 200-MHz clock may have harmonics up to the fifth, meaning we need to consider the bandwidth up to 1 GHz.

Technology moves fast, and much has changed over the years since I have been in high-speed multilayer design, particularly advances in lithography that enable IC manufacturers to ship smaller and smaller dies on chips. In 1987 we thought that 0.5-micron technology was the ultimate, but today, 5-nm technology is at the cutting-edge. As of 2019, Samsung and TSMC have begun commercial production of 5-nm nodes.

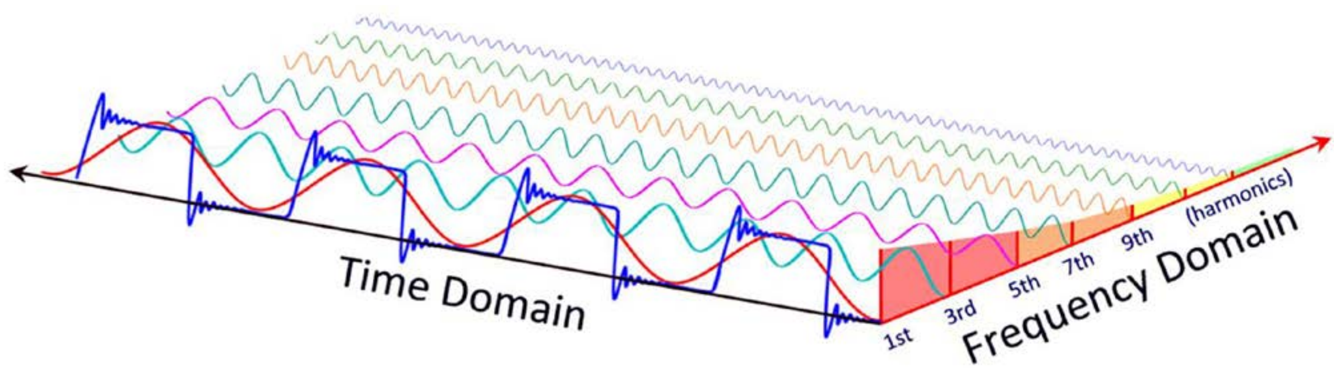


Figure 3: Harmonics of a square wave transformed from the time to the frequency domain.

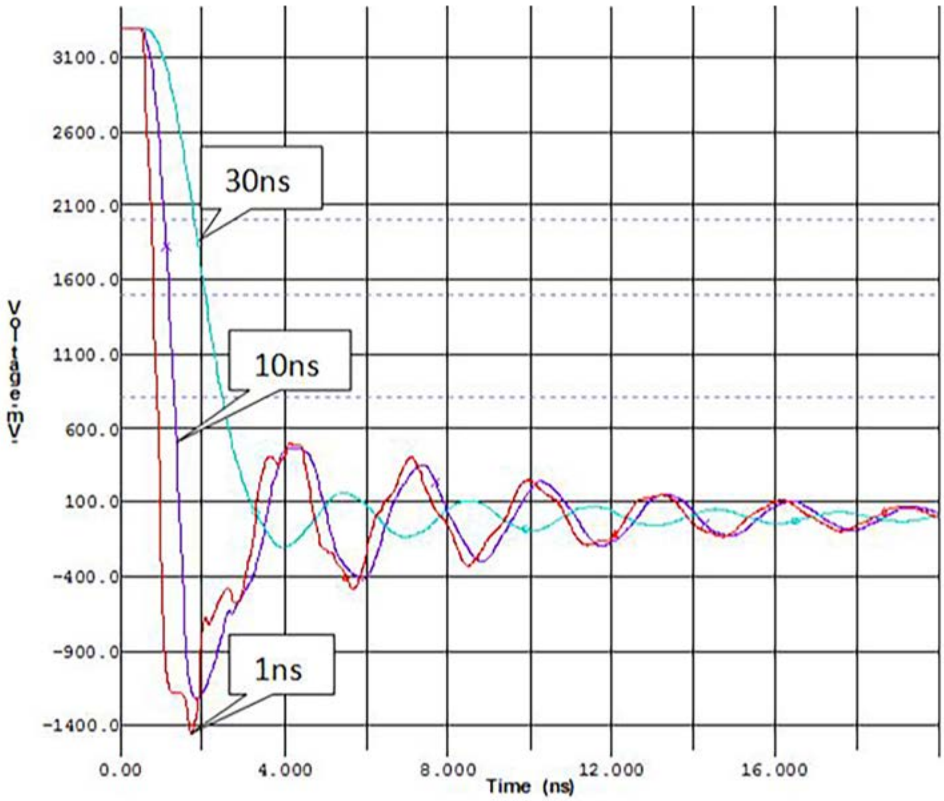


Figure 4: Edge rate changes in the time domain over the years (simulated in HyperLynx).

Power consumption in FPGAs has also become a primary factor for FPGA selection. Whether the concern is absolute power consumption, usable performance, battery life, thermal challenges, or reliability, power consumption is at the center of it all. To reduce power consumption, IC manufacturers have moved to lower core voltages and higher operating frequencies, which means faster edge rates, of course. And faster edge rates mean reflections and signal quality problems. The enhancements in driver edge rates have a significant impact on signal quality, timing, crosstalk, and EMC.

Figure 4 illustrates the change in edge rates over the years from 30 ns back in the early 1980s to less than 1 ns in 2010. Sub-nanosecond rise times are now common. For the same frequency and length trace, the faster edge rate creates ringing in the unterminated transmission line. This also has a direct impact on radiated emissions. Figure 5 shows the massive increase in emissions from the slowest to

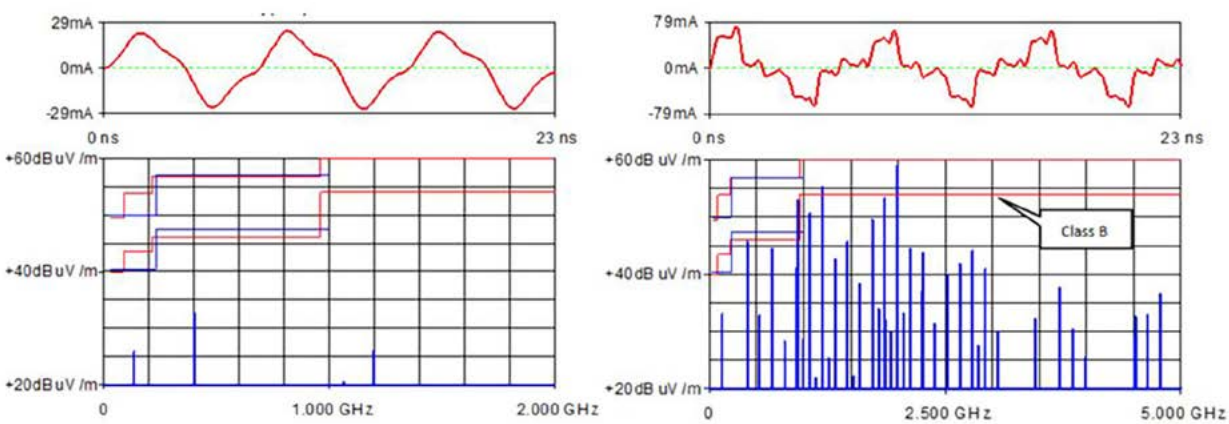


Figure 5: Radiated emissions in the frequency domain from 10 ns edge rate (L) and 1 ns (R).

the fastest rise time. When dealing with 1-ns rise times, the emissions can easily exceed the FCC/CISPR Class B limits for an unterminated transmission line.

At high frequencies, traces on a PCB act as a monopole or loop antennas. Unfortunately, the high-frequency components of the fundamental radiate more readily because their shorter wavelengths are comparable to trace lengths (particularly stubs), which act as antennas. Consequently, although the amplitude of the harmonic frequency components decreases as the frequency increase, the radiated frequency varies depending on the antennas/traces characteristics.

Computer-based products tend to radiate on the odd harmonics. High emissions are generally detected at the third, fifth, and sometimes seventh harmonic of the fundamental clock frequency. If this also occurs where the AC impedance of the PDN is high, then the radiation is projected even farther.

Being able to view a problem in the frequency domain is a powerful tool that provides another perspective that often reveals structure to a problem that isn't obvious in the time domain alone.

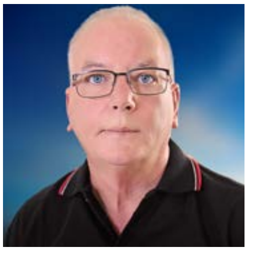
Key Points

- The frequency domain can provide valuable insight to understand and master many SI effects
- In the time domain, the system is evaluated according to the progression of its state with time; however, in the frequency domain, the system is analyzed according to its response for different frequencies
- The system is changed from time to frequency to make it easy to understand the response
- The Fourier series expansion of a square wave is made up of a sum of harmonics
- Impedance is defined in both the time and frequency domains; however, it is far easier to understand and apply the concepts of AC impedance in the frequency domain
- If the square wave has an even mark-to-space ratio, then the even harmonics cancel
- The high-frequency content of a square wave is significantly affected by the rise time of the waveform; a fast rise time results in higher-frequency components
- One needs to consider the maximum bandwidth of a signal, including harmonics, rather than assume the perfect square wave fundamental frequency model
- Power consumption in FPGAs has become a primary factor for FPGA selection
- To reduce power consumption, IC manufacturers have moved to lower core voltages and higher operating frequencies, which of course mean faster edge rates
- A faster edge rate creates ringing in the unterminated transmission line; this also has a direct impact on radiated emissions
- The high-frequency components of the fundamental radiate more readily because their shorter wavelengths are comparable to trace lengths (particularly stubs), which act as antennas

Further Reading

- B. Olney, "Beyond Design: When Do Traces Become Transmission Lines," *The PCB Design Magazine,* October 2017.
- B. Olney, "Beyond Design: Signal Integrity, Part 1," *The PCB Design Magazine,* October 2014.
- E. Bogatin, *Signal and Power Integrity: Simplified,* Prentice Hall, 2008. **DESIGN007**

Editor's Note: Figures 1 and 3 drawn by Barry Olney.



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

MilAero007 Highlights

SMTAI 2019: To the Moon and Beyond ▶

W. Michael Hawes, D.Sc., gave a great keynote at SMTAI 2019 titled, "To the Moon! Orion's Next Giant Leap Into Deep Space." Dr. Mike Hawes is currently the VP for human space exploration and the Orion Program manager for Lockheed Martin. Barry Matties spoke with him afterward to discuss the technology innovations that empower the next-generation spacecraft to take astronauts to explore farther than humankind has ever ventured.

Catching up With Niche Electronics ▶

Niche Electronics President Frank Bowman speaks with Dan Beaulieu about his company, their services, how they help their customers, and their plans for the future.

WWII Meteorologist Turned Material Scientist Shares Nobel Prize in Chemistry 2019 ▶

The Air Force Office of Scientific Research congratulates John B. Goodenough, professor in the Cockrell School of Engineering at The University of Texas at Austin, for recently being awarded the Nobel Prize in Chemistry 2019. Goodenough is the eldest recipient of a Nobel Prize at age 97.

Defense Speak Interpreted: PCB-related OTAs from NAVSEA Crane ▶

In my previous column, I described how Other Transaction Authority (OTA) projects were speeding up the development of new technology for the Defense Department. Much of this improvement has to do with the speed of contracting and the less restrictive selection and payment process involved. Specifically, I would like to call out projects under the National Security Technology Accelerator (NSTXL).

Combination of Techniques Could Improve Security for IoT Devices ▶

A multi-pronged data analysis approach that can strengthen the security of IoT devices—such as smart TVs, home video cameras, and baby monitors—against current risks and threats has been created by a team of Penn State World Campus students pursuing master of professional studies degrees in information sciences.

Boeing Building the Future of Space at International Astronautical Congress 2019 ▶

Boeing will participate in International Astronautical Congress (IAC) 2019 next week at the Walter E. Washington Convention Center in Washington, D.C., to celebrate our shared mission with NASA and global partners in this new age of space exploration and transformation.

Research Shows Military Service Can Hurt Some Job Seekers' Prospects ▶

New research from Duke University's Fuqua School of Business suggests veteran job candidates can be typecast as agentic and unemotional and are likely to be overlooked for jobs that leverage emotional intelligence and interpersonal and leadership skills.

Global Market for Counter Unmanned Aircraft Systems to Exceed $2B by 2024 ▶

Frost & Sullivan's recent analysis, Global Counter UAS Market, Forecast to 2024, reveals that heightened demand for commercial unmanned aerial systems (C-UAS) by the military for expensive, technologically advanced, multiple-sensor systems is driving innovative C-UAS market growth opportunities.

Fall 2019 Recap

The Digital Layout

by Stephen V. Chavez, MIT, CID+, IPC DESIGNERS COUNCIL

As the second half of the year is quickly passing by, we have seen lots of continued activities within our industry regarding PCB design. PCB West was held in mid-September in Santa Clara, California, and as usual, the show exceeded expectations. AltiumLive 2019 was held in early November in San Diego, California, and also had a great turnout. We also have some activity within our local IPC Designers Council (DC) chapters, such as the San Diego Chapter, which held its final chapter meeting for the year in late September. Here is a brief breakdown of these recent fall activities.

San Diego Chapter, California

Chapter Leader: Luke Hausherr, CID+

The San Diego Chapter held another successful meeting on September 24 at San Diego PCB with 25 attendees. OrCAD EMA sponsored the event to help offset the cost of lunch. The highlight of the meeting was a presentation given by Jeff Jenkins, which covered conformal coatings (Figure 1). The content of the presentation was very thorough and received positive feedback from all who attended. Jeff is a PCB chief technologist for L3Harris and has been involved in the workings of the PCB design, fabrication, and CCA industries for over 23 years.

His presentation covered:

- The purpose of conformal coatings
- Types of conformal coatings and the pros and cons of each
- Methods of application and the pros and cons of each
- Items not coated
- Engineering drawings and notes

This was also the last San Diego Chapter meeting for 2019, but we will convene in January to plan for 2020.

Silicon Valley Chapter, California

Chapter Leader: Bob McCreight

The Silicon Valley Chapter held a chapter meeting on October 24 and will report its continued success in next month's column.

Figure 1: Luke Hausherr presenting certification of appreciation to Jeff Jenkins.

Research Triangle Park (RTP) Chapter, North Carolina

Chapter Leader: Tony Cosentino

In next month's column, the RTP Chapter will report on PCB Carolina 2019, which will take place on November 13 in Raleigh, North Carolina.

AltiumLive 2019

This annual event—which takes place in both the U.S., as mentioned earlier, as well as in Europe (Frankfurt, Germany)—creates lots of industry buzz and has gained tons of traction over the past few years. This year's keynote speakers included Eric Bogatin, Robert Feranec, Joe Grand, and Bob Martin in addition to industry technical sessions led by Rick Hartley, Gary Ferrari, Mike Creeden, Susy Webb, Tara Dunn, Pete Wilson, Linda Mazzitelli, Max Seeley and Carl Schattke, Ari Mahpour, Shashank Samala, and many others. The Altium Technical Team also covered the University Day courses, and the feedback from this year's attendees has been overwhelmingly positive, as reported to me from my dear friend and colleague, Judy Warner, director of community engagement at Altium. As always, it's great to see and hear of the continued activities, from professional development to knowledge sharing and networking, taking place at various industry events.

IPC CID/CID+ Certification Success

We also continue to have successful IPC CID and CID+ certification classes. The feedback from attendees of these classes is extremely positive. They are an excellent source of professional development. If you are not yet CID/CID+ certified, I highly recommend these certification courses as a path for continued education in PCB design.

The remaining IPC Advanced Certified Interconnect Designer CID+ training session for 2019 is scheduled for December 3–6 in Manchester, New Hampshire. Note: Dates and locations are subject to change, and a minimum enrollment of seven students is required for a class to be held. Contact EPTAC Corporation to check current dates and availability. **DESIGN007**

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.

Stephen Chavez, MIT, CID+, 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.

Artificial Skin Creates First Ticklish Devices

A new interface developed by researchers in Bristol and Paris takes touch technology to the next level.

The Skin-On interface, developed by researchers in the Faculty of Engineering at the University of Bristol in partnership with Telecomm ParisTech and Sorbonne University, mimics human skin in appearance but also in sensing resolution.

In the study, researchers created a phone case, computer touchpad and smartwatch to demonstrate how touch gestures on the Skin-On interface can convey expressive messages for computer-mediated communication with humans or virtual characters. The authors are inviting developers with an interest in Skin-On interfaces to get in touch.

(Source: University of Bristol)

Design Challenges for Developing High-density 2.5D Interposers, Part 2

Designers Notebook

by Vern Solberg, CONSULTANT

Author's Note: Read Part 1 here.

2.5D Interposer Design

A typical 2.5D substrate application supports the interfacing of one or more high-density semiconductors. While the upper surface will accommodate a majority of semiconductor redistribution and/or die-to-die interface for multiple die applications, the primary I/O channels and power and ground terminals are transferred to the bottom surface of the interposer through plated or filled microvias for routing to an array pattern of terminals designated to interface with the intermediate package substrate or host PCB structure requiring significantly lower circuit density. Although the overall circuit density of the 2.5D interposer is significantly greater than the mainstream HDI circuit board, the commercial CAD tools already available for PCB circuit routing should accommodate most interposer development activity.

In preparation for developing the interposer, designers are advised to prepare a description of the proposed substrate, detailing the semiconductor element(s) physical parameters, intended use environment, and timeline anticipated for development. The 2.5D substrate fabricator can then recommend a suitable base material (organic, silicon or glass) and define the fabricator's via forming methodologies, metalization process capabilities, and circuit geometry limitations.

Base Materials

The primary base materials utilized for the 2.5D interposer applications include glass-reinforced organic laminates, silicon (wafers or panels), and ruggedized glass panels.

1. Organic

The high Tg, low CTE organic base material outlined in Part 1 is commonly furnished with a seed layer of thin copper foil bonded onto both surfaces. Following laser ablation of the via holes, the surface is coated with a photoresist, and the circuit pattern is imaged, developed, and made ready for a semi-additive plating process to fill via holes and form the conductor pattern. The copper seed layer that remained on the substrate's surface after pattern plating is chemically etched away, leaving only the conductor pattern. Additional copper circuit layers can then be built up onto the base to further increase circuit density, utilizing the more conventional and well-established PCB fabrication processes.

2. Silicon

Silicon-based interposer fabrication requires a rather specialized and complex sequence of processes that begin with via-hole formation. Although laser ablation can be adopted for forming the microvia holes, the process most commonly employed for volume applications uses a deep reactive-ion etching (DRIE) process, often referred to as the Bosch process. This methodology can provide very small hole diameters that range from 5–20-microns. In preparation for conductor forming and via filling, a seed layer of copper or tungsten is applied to enable electroplating the additional copper required to complete the via-fill operation. Further pattern imaging and plating processes are engaged to provide interconnect features on the outer surfaces of the silicon substrate.

3. Glass

Glass may be furnished in a wafer format but is more commonly furnished as panels as large as 500 mm square. Although the initial fabrication panels are rather large, users may specify segmenting the large panel in order to furnish a size that is more suitable for their existing assembly systems. In regard to substrate fabrication capability, microvia hole diameters can range between 10–30 microns with a pitch in the range of 50–100 microns (depending on the hole diameter).

For the initial hole-forming process, the fabricator may employ one or a combination of ablation technologies: laser ablation (CO_2, excimer, UV), electrostatic discharge, mechanical drilling, chemical etching, and/or micro sandblasting. In regard to conductor forming, several methods are available for metalizing glass, including copper alloy plating similar to that described for the silicon-based substrate, deposited silver paste, and precision printing using silver and copper impregnated inks.

Design Guidelines

The design guidelines furnished in Table 1 relate to copper alloy via filling and conductor formation. The geometries furnished were developed from research I did and consensus among several colleagues involved in the technology. The data shown may not reflect the capability of all suppliers in their respective categories, but supplier companies will generally furnish the designer with alternative design guidance related to their material sets and specific process capabilities. The supplier-developed interposer design guidelines will generally reflect factors derived from their experience, ensuring that they will likely furnish a reliable product with a high degree of quality and process yield.

As detailed earlier, the platform for mounting the die is typically one of three material sets: high Tg glass-reinforced epoxy laminate, super stable silicon wafers, and high impact glass panels. Because of the relatively high I/O required to interface multiple die, these interposers commonly adopt a uniform array-configured ball or bump contact design for the next level interface.

Limiting the semiconductor package size continues to be a factor as well; however, the overall package outline will always be controlled by the size of the largest die and number of interface contacts. The base structure (interposer) selected must furnish a surface area large enough accommodate the I/O interface and mechanically stable enough to withstand

Attribute	CTE Matching Organic Material	Silicon Base Material	Glass Base Material
Substrate Panel Thickness	100–800 µm (~0.004–0.032")	200 µm to 3 mm (~0.008" to 0.120")	200 µm to 3mm (~0.008–0.120")
µVia Hole Diameter	10-µm min. (~0.0004")	5–20 µm (~0.0002–0.0008")	10–30 µm (~0.0004–0.0012")
µVia Land Diameter	20-µm min. (~0.0008")	15–25 µm (~0.0006–0.001")	25–50 µm (~0.001–0.002")
µVia Hole Pitch	40-µm min. (~0.0016")	30–50 µm (~0.0012–0.002")	50–100 µm (~0.002–0.004")
Base Core Line/Spaces	3-µm min. (~0.00012")	2-µm min. (~0.00008")	1–5µm (~0.00004–0.0002")
Build-up Lines/Spaces	3-µm min. (~0.00012")	2-µm min. (~0.00008")	1–5µm (~0.00004–0.0002")

Table 1: 2.5D interposer design guidelines.

the process temperatures of the entire package assembly process (bumped die mounting and/or wire-bond processing and encapsulation).

Technical Challenges

A number of 2.5D interposer-based products have already materialized, but there are several technical challenges that need to be addressed: cost-effective through-silicon-via (TSV) and through-glass-via (TGV) formation, thin wafer and panel handling solutions, thermal management, and the lack of a broad 2.5D infrastructure network. Although 2.5D package technology has reached a level of maturity, a number of issues remain that must be considered: selecting the specific material and methodology to be utilized, semiconductor procurement and die quality, and the area required for single or multiple die mounting and interconnect.

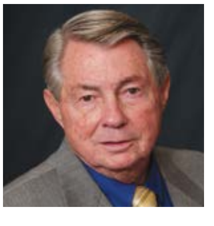
Key Planning Issues for 2.5D Packaging

Assembly process methodologies will vary a great deal. Issues that will need to be resolved before beginning the development process include:

- Selecting suitable semiconductors for multiple die packaging
- Establishing reliable sources for semiconductor elements
- Specifying physical and environmental operating conditions
- Defining package design constraints and process protocols
- Stipulating electrical test method and post-assembly inspection criteria

Conclusion

Semiconductor packaging methodology will continue to evolve, and market analysts project a fairly steady growth in semiconductor package applications requiring the 2.5D interposer elements. Although very difficult to predict, new semiconductor package innovations and solutions will likely continue to emerge. **DESIGN007**



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

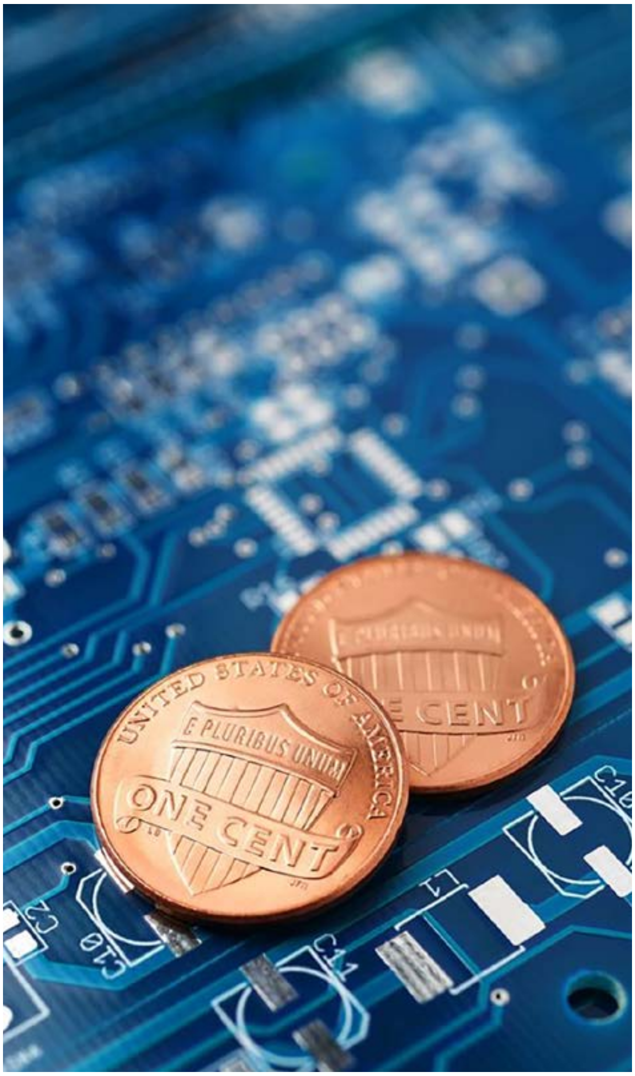
A Penny for Your Thoughts on Copper

Connect the Dots

by Bob Tise, SUNSTONE CIRCUITS

I know what you're thinking: "He can't possibly write an entire article dedicated to the use of copper in PCBs." I say, "Hold my beer."

Copper is the primary metal for standard PCBs, which you can find in pretty much every type of electronic device on the planet. And while standard PCB capabilities depend on what materials are used and how they are constructed, copper is the go-to choice.



Copper works very well for the conduction of electrical current in any environment. In fact, it has the highest electrical conductivity rating of all non-precious metals, making it highly effective for performance and cost. For a vast majority of PCB designs, copper is the conductor of choice. Copper conducts the signal and power for just about every electronic item in your life.

Way to go, copper! You are doing a great job. Keep it up.

There are, however, challenges associated with its use. Though incredibly conductive, copper is a relatively soft metal and susceptible to corrosion. Insufficient copper-to-edge clearances can result, potentially causing exposed copper, shorts, or corrosion. Resist can flake off of very narrow traces. During etching, long slivers can wander around in the bath. These loose cannons of conductivity can affix themselves to a board, keeping unwanted copper from being etched away, creating unwanted circuits and failed boards.

So, how do you get copper to function in a nice, orderly manner the way your design calls for it? First, keep an eye out for potential problems during the design phase. To properly control resistance and temperature rise, use a PCB trace width calculator to determine how thick and wide traces need to be. If your board is large enough, wider traces increase reliability. Skinny traces can get hot and release all of your magic

Figure 1: Trace width.

smoke. Once it gets out, it's really hard to put back in (Figure 1).

In design, you can reduce the likelihood of shorts, damaged traces, and exposed copper by keeping adequate distance between traces and mounting holes. Make sure to leave room for screw heads, washers, and other sorts of fasteners. Allow as much annular ring as possible for vias and plated holes. Within reason, more is always better. Vias need a bare minimum pad 0.010" larger than the drill. Other plated holes need 0.016" larger than the drill size. You can design to reduce the chance of slivers as well; keep widths and spacing above 0.006 inches.

Copper: The Key to the PCB Process

For PCB manufacturing, copper is integrated into the board in three different ways. The raw laminate material (dielectric) is copper-clad, meaning there is a consistent thickness of copper adhered to both sides of the dielectric material. This copper has either a plating mask or an etching mask applied that defines all circuitry and pads for that layer. Eventually, all the copper that is between and surrounding the desired circuitry and pads will be chemically etched away, leaving only the needed copper.

For the external and other plated layers—where there are plated holes—the second application of copper is needed and added after the holes are drilled. This copper is a very thin layer of catalytically deposited copper used primarily to metalize the plated through-holes to the point that they will conduct current for electroplating. This deposition of copper is a critical step to ensure that all the holes and vias will successfully plate with copper.

Once these holes are metalized and the plating mask is applied, it is time to electroplate all the circuitry, pads, and through-holes with copper (Figure 2). On a typical PCB, the electroplating process increases the thickness of the base copper foil by approximately 1.0 mil of copper and deposits a total thickness of 0.8

Figure 2: Plate.

mils of copper into the holes for normal Class 2 designs. This gives the circuitry the current-carrying capacity needed and makes the plated through-holes robust enough to withstand thermal expansion during operation and maintain an electrical connection to all of the necessary internal layers.

Whew! I'm out of breath now. Pass me back that beer!

Counting the Ways to Love Copper

We lean on copper for a lot of reasons. Along with great conductivity, it is very tolerant of heat (aka "thermal stress"). This is increasingly important as more boards use exotic materials like ceramic, Teflon, or next-gen laminates that are often part of devices that need to function to their full potential in high temperatures.

Last, but not least, we should recognize copper's other big selling points. It's plentiful and relatively inexpensive. Over the past year at Sunstone, we have dedicated a fair amount to assessing the cost of quality. With supply chains in so much flux with respect to cost and availability, copper's reliability and usability make standard PCB use that much more compelling for designers, engineers, and entrepreneurs. Copper is pretty cool. **DESIGN007**

Bob Tise is an engineer at Sunstone Circuits. To read past columns or contact, click here.

IDTechEx Latest Research on Autonomous Cars and Robotaxis 2020–2040

Autonomous driving holds great promises for increasing road safety and efficiency, as well as for decreasing future travel cost. Many companies have already been operating testing fleets on public roads and have announced plans to commercialize robotaxi services in the next 2-3 years. How will autonomous driving shape the future of the automotive sector? IDTechEx has recently published its latest report on autonomous mobility titled "Autonomous Cars and Robotaxis 2020-2040." According to IDTechEx's forecasts, global autonomous car and robotaxi services will become a $2.5 trillion market by 2040.

The autonomous driving ecosystem requires a different approach to traditional automotive development, in that it focuses on the full technology stack of hardware such as sensors, computing platforms, and non-hardware components, including AI software and HD maps. The report offers an in-depth analysis of key enabling technologies, including lidars, radars, cameras, AI software, HD maps, teleoperation, cybersecurity, and 5G and V2X. Key players with their latest technologies and product commercialization plans are presented as the case studies of this report.

Autonomous driving technologies will accelerate the shift of private car ownership towards shared mobility or mobility-as-a-service (MaaS). Mobility services enabled by autonomous driving technology, which allows fleet operators to get rid of the biggest operation cost—the human driver—will offer a cheaper alternative to purchasing and owning a private car. IDTechEx forecasts that in a moderate scenario, 30% of the total travel demand will be provided by MaaS by 2040 and global passenger car sales are expected to peak in 2031.

(Source: IDTechEx)

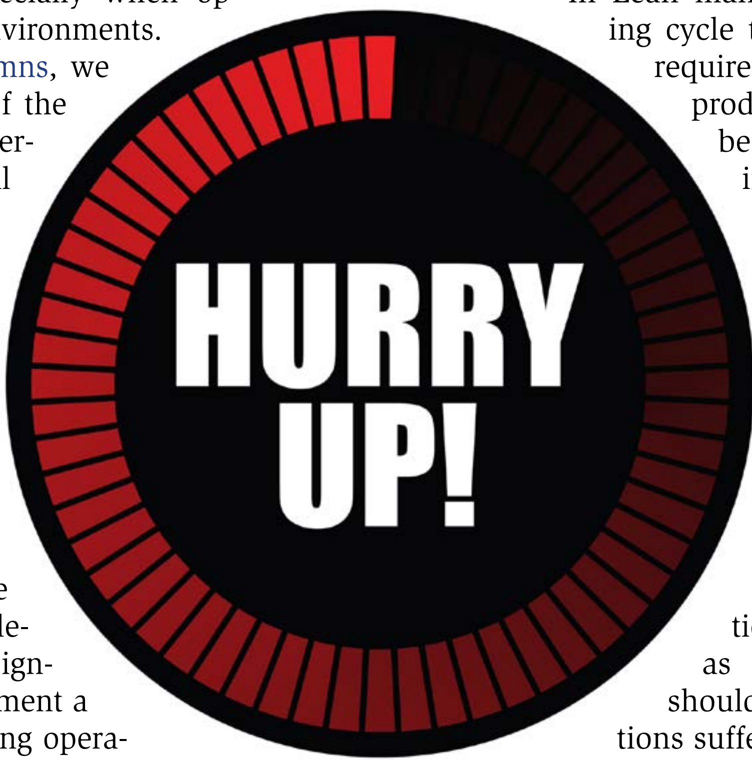
Design Challenges and the Impact on Coating Success

Sensible Design

by Phil Kinner, ELECTROLUBE

As assemblies become ever more densely populated, and housing/casing designs become more permeable to save weight, the use of conformal coatings is becoming essential to protect the assembly from its operating environment and ensure acceptable reliability for the application intended, especially when operating in hostile environments.

In our previous columns, we have covered some of the fundamental considerations of conformal coating selection and performance, examined key benefits of different types of coatings, and explored their limitations. In this month's column, I am going to concentrate on essential factors regarding the challenges board designs can pose on designers to help you implement a more successful coating operation. Let's look at issues that may arise with coating coverage, cycle time, and coating flow.

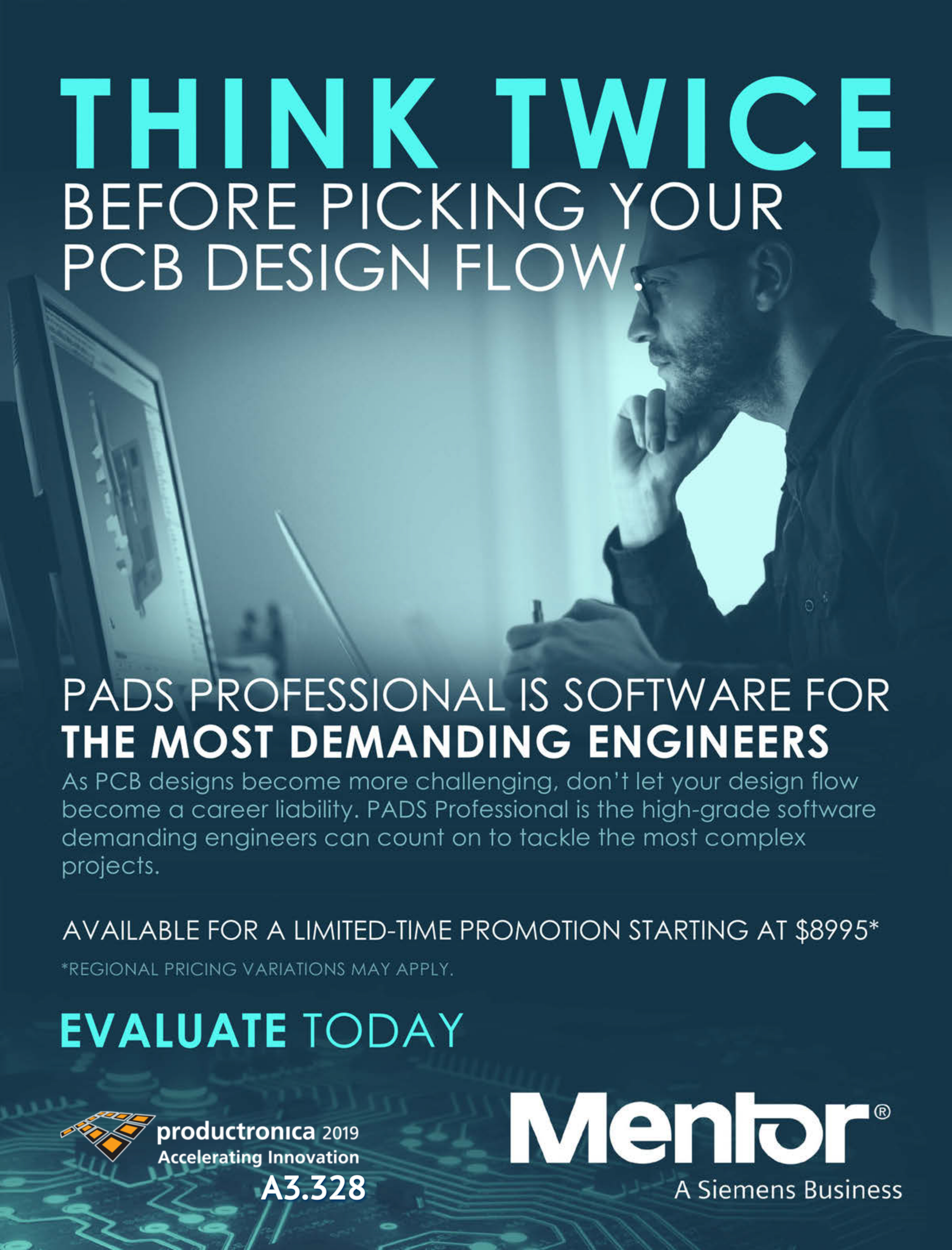


Board Design Challenges

The design of the board can have a huge impact on a successful coating operation, both by determining which methods can actually be used, as well as the degree of coverage that can be achieved within the required cycle time. I'm going to focus on selective coating, being the most common application method, where masking is minimised if not eliminated, but most of the principles will translate to other coating application methods.

In Lean manufacturing, the coating cycle time is often the key requirement, ensuring the production takt time can be met, thus maintaining a balanced line. Given that coating is often one of the later processes in the production line, it is often advantageous for the coating cycle time to be faster than the overall line takt time to ensure any accumulated production can be cleared as rapidly as possible, should any previous operations suffer a stoppage.

In general, the fewer areas there are on a board that must be free from coating, the faster the coating cycle time will be. Should the coating cycle time be too long, then a decision can be made to either determine the most vulnerable parts of the assembly and focus on that area in the available cycle time, or to split the coating across several workstations.


THINK TWICE
BEFORE PICKING YOUR PCB DESIGN FLOW.
PADS PROFESSIONAL IS SOFTWARE FOR
THE MOST DEMANDING ENGINEERS
As PCB designs become more challenging, don't let your design flow become a career liability. PADS Professional is the high-grade software demanding engineers can count on to tackle the most complex projects.
AVAILABLE FOR A LIMITED-TIME PROMOTION STARTING AT $8995*
*REGIONAL PRICING VARIATIONS MAY APPLY.
EVALUATE TODAY
productronica 2019
Accelerating Innovation
A3.328
Mentor®
A Siemens Business

Coating Cycle Time

Coating cycle time is very important to the balance of the production line, so what kills coating quality and cycle time on a typical selective coating line? Selective coating machines deposit a stripe of coating material that can be programmed to stop and start on demand. Stripes of coating can be deposited to create a coating pattern, avoiding areas such as switches, connectors, and test points, which must not be coated to prevent interference with form, fit, or function.

The coating stripes deposited are usually in the 8–15 mm range for optimum accuracy and minimizing overspray and splashing. When the area to be coated is less than 8 mm in width, then it is necessary to utilise a dispensing step, which is a cycle time killer.

Due to the combination of machine X/Y positional accuracy, material fluid dynamics, and component topography, 2–3 mm is usually as close to keep-out areas as anyone would be comfortable coating for a repeatable process. In summary, must-coat and keep-out areas positioned within 2–3 mm of each other present a problem. Dispensing is also required, representing another process step adding to cycle time.

Coating Flow

In manufacturing, for many coatings, it is difficult to limit the flow of coating from the application site to adjacent sites. Most coatings will have some level of capillary flow (often referred to as wicking), taking the coating to unanticipated areas. Therefore, components that must be coated should not be placed in proximity to components that must not be coated.

Coating flow is also an important consideration on low-standoff bottom-terminated devices, such as BGAs or QFNs, which may also have microvias under them. Coating can wick under the parts and then down the vias to the opposite side of the assembly. To avoid this problem, the vias under low standoff components can be tented with solder mask or have the vias filled with solder or suitable via-fill material. Of course, since conformal coating can reduce the solder joint lifetime of bottom-terminated devices, deliberately coating beneath these devices should be avoided. Keep in mind that liquid coatings will flow somewhat during the coating application process, so placing a keep-out zone at the base of a tall component, such as an electrolytic capacitor, will create a lot of problems in manufacturing as the coating flows down the tall part into the keep-out zone.

Engineering drawings or quality control documents should be very specific on what coating anomalies are defects, requiring disposition, and those that are primarily cosmetic. For many years in the industry, anything that looked different in the coating was considered a defect, requiring aesthetically perfect assemblies. While possible, such requirements often drive up the cost of coating and often have no impact on assembly reliability.

Designers should be aware that coating the edges of an assembly is not considered as a value-add for most applications. Most design standards do not allow inner layer circuit traces to be closer than 0.63 mm from the edge of the board. The act of routing the board edges results in smearing the epoxy resin over the cut glass fibers or the reinforcement material, sealing off the inner layers. A designer might specify coating board edges where such sealing does not occur, such as with V-scored, punched, or sheared edges.

Understanding the dynamics of what affects what on the surface of the board during the early stages of design will go a long way in helping you achieve a successful conformal coating operation. This will also help avoid potential disasters in other areas of production. **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 I-007eBooks.com to download this and other free, educational titles.

FLEX007

A SPECIAL DESIGN007 MAGAZINE SECTION

Flex Technology Advancing Almost Daily

What the Flex?

by Andy Shaughnessy, I-CONNECT007

I've been out covering industry conferences and trade shows lately. Most of the designers I meet are rigid board designers, but at every event, I run into a handful of designers who are designing flex and rigid-flex circuits.

One thing I noticed: The flex folks can monopolize a conversation without even trying. When a designer starts talking about using flex, maybe during a break at a conference, many of the rigid board designers gather around, bagels and coffee in hand, eager to hear more. The rigid designers often start lobbing questions at the flex designers, usually about the materials, process, and cost. They ask for business cards, and a recommendation for a good flex fabricator to help get their plans off the ground.

Flexible circuits still make up only a small minority of the total, but many designers and design engineers have their eyes on the flex space. They're designing boards for products

with increasingly smaller form factors, and when reliability becomes a big problem, flex starts to look better and better.

This is another case of the OEM and the final product leading the way. One senior designer who does a mix of flex and rigid boards told me that his company had to go to Korea to find a flex fabricator who could handle their next-generation flexible circuits. They really wanted to keep everything in the U.S., but it just wasn't possible.

Flexible circuit technology is changing so rapidly that flex developers sometimes wind up revamping their business plans. Case in point: CelLink, a company founded eight years ago with a plan to make large, high-conductance flexible circuits for the solar, battery and LED segments. But the universe had other plans.

Now, BMW has invested $22.5 million in the California company, along with Ford Motor Company and Robert Bosch Venture Capital. CelLink is poised to begin developing cutting-edge flexible circuits that will replace the antiquated wire harnesses in the Bavarian company's cars. According to reports, these flexible circuits are low in cost, reliable, and lightweight, potentially reducing the weight of the car's circuitry by up to 90%.

If you had told a group of PCB technologists 20 years ago that a car's wire harnesses could be replaced by flexible circuity, they would have laughed at you. I've said it before: Flex is like the Wild, Wild West right now. PCB designers and design engineers have a lot of questions about flex and rigid-flex, but Flex007 has the answers.

This month, we begin with a column by Tara Dunn of Omni PCB. Tara discusses why she believes that technologists will have to embrace change more quickly than ever before, and not just with flexible circuits, in order to keep up with innovations in the industry. Next, Dominique Numakura of DKN Research shares his thoughts on the use of electroless plating to produce high-density flexible circuits, and why the time is right for companies to consider electroless plating. Then, Joe Fjelstad explains why additive manufacturing may be on the verge of taking off, and how additive processes can help companies save one very precious commodity: Time. And Outi Rusanen, et al., of TactoTek make the case for smart molded structures, and the need for updated standards that can keep up with this evolving technology.

We'll be covering productronica 2019, and before you know it, we'll be heading to IPC APEX EXPO and DesignCon. I hope to see you on the road. See you next month! **FLEX007**

Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 19 years. He can be reached by clicking here.

Technique Helps Robots Find the Front Door

Standard approaches for robotic navigation involve mapping an area ahead of time, then using algorithms to guide a robot toward a specific goal or GPS coordinate on the map. While this approach might make sense for exploring specific environments, such as the layout of a particular building or planned obstacle course, it can become unwieldy in the context of last-mile delivery.

MIT engineers have developed a navigation method that doesn't require mapping an area in advance. Instead, their approach enables a robot to use clues in its environment to plan out a route to its destination, which can be described in general semantic terms, such as "front door" or "garage," rather than as coordinates on a map. For example, if a robot is instructed to deliver a package to someone's front door, it might start on the road and see a driveway, which it has been trained to recognize as likely to lead toward a sidewalk, which in turn is likely to lead to the front door.

The new technique can greatly reduce the time a robot spends exploring a property before identifying its target, and it doesn't rely on maps of specific residences.

This research is supported, in part, by the Ford Motor Company.

(Source: MIT)

The Challenge of Change

Flex Talk

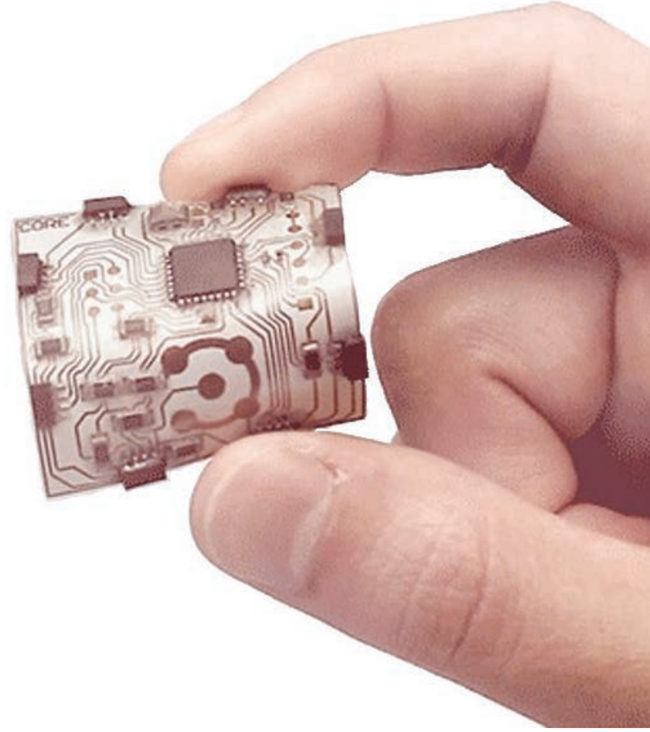
by Tara Dunn, OMNI PCB

I recently kicked off a presentation on flex and rigid-flex by asking for a show of hands of those who had never worked with flex materials or considered themselves to be just learning how to design with flex. Over half of the room raised their hands, which excited me because I could help them learn something new, but it also surprised me a bit. After all, flex is the fastest-growing portion of the market and has been for several years. But change can be hard and uncomfortable, and there is a natural tendency to avoid change until there is no other choice, especially in a work environment.

Over the next few years, one thing that I think is going to change in our industry is more and more people will be forced to embrace change and adopt new technologies and materials at a faster pace than we have had the luxury of in the past. The industry is moving at a rapid pace, with new materials and processes rolling out quickly. Electronics are becoming increasingly complex in smaller and lighter packages.

We all serve different areas of the electronics market and work with varying levels of technology. For some, change could mean moving to flex and rigid-flex to take advantage of space, weight, and packaging benefits. For others, change could mean leaning into semi-additive PCB processing because line width and space requirements need to be much smaller than we are able to reliably achieve with traditional subtractive processes. Fabricators serving that technology segment will deal with changes and adjustments to processes and work with new technologies and materials that challenge how we think about PCB design.

In my opinion, the best way to lessen the challenges and uncertainty of change is to arm yourself with as much knowledge as possible. Yes, I think internet searches and YouTube vid-



PYRALUX® FLEXIBLE CIRCUIT MATERIALS

PYRALUX® AP
high reliability flex & multilayer circuitry

PYRALUX® AC
for thin, light, high density with chip-on-flex attachment

PYRALUX® HT
for high temp, high performance flex

PYRALUX®TK
Teflon®/Kapton®

PYRALUX® LF
industry standard for high reliability

PYRALUX® FR
flame-retardant for products requiring UL rating

PYRALUX® APR
reliable resistor, temperature tolerant

DuPont offers a comprehensive, industry-leading portfolio of Pyralux® flexible circuit materials and systems designed to meet today's and next generation design challenges.

The Pyralux® portfolio includes a diverse collection of core dielectric materials and customized dimensional constructions that enable the designer and manufacturer of complex circuits to deliver high performance solutions.

DuPont Pyralux® copper clad laminates, bondplys, coverlays, and adhesive systems offer excellent functional performance and high reliability and allow for the fabrication of thin, solderable, high density electrical interconnects for single and double-sided, multilayer flex and rigid flex applications.

PROUDLY DISTRIBUTED BY
INSULECTRO
20362 WINDROW DRIVE • LAKE FOREST, CA 92630 • 949.587.3200 • INSULECTRO.COM

eos are going to have a place on our learning curves, and I challenge fabricators and suppliers to continue to create educational content that we can have available at our fingertips whenever the urgent need to answer a question arises. But I also think that we are going to have to rely more on each other and have dialog about new processes, discussing both the challenges and the opportunities they afford.

Let me share an example. I was recently told a “war story” about a “flex board that didn’t flex.” These stories are often some of my favorite stories, but this one was different. It wasn’t a story of someone misjudging how stiff the flex would be with solid copper planes; this was someone using thin-core rigid material in place of flex materials. The board was for a wearable application, and the circuit would be attached to a patient’s clothing.

After some internet research, the team decided to use a flexible circuit, which would be their first flex design. As they continued researching flex materials, they also learned that rigid materials come with very thin cores, as well. They were used to that material, and anything only two or three mils thick should flex fine, right? Well, no.

They ran through two different prototype versions before understanding that the material was not intended to be used in the manner they were designing it. At that point, they contacted a flexible circuit manufacturer and learned about common materials and what would best meet their needs. The good news was, once they moved to flex materials, the design worked perfectly!

I share that story because it is so easy to make similar missteps. Today’s world runs at breakneck speed, and it is easy to do a little, or even a lot of internet research, to think you fully understand the concepts and then get caught by something that you hadn’t learned yet. Thus, I challenge people to work on building their network. And I don’t just mean the number of followers or connections on social media, although that has its place also.

My challenge is directed at getting out, being active, and participating in our industry events. Attend an SMTA chapter expo with speakers addressing new high-speed materials. Attend AltiumLive and listen to presentations on best practices for flex and rigid-flex design. Attend IPC APEX EXPO, and sit in on the session on semi-additive PCB processes. Attend your local IPC Designer’s Council meetings or IEEE events.

While you are at these events, introduce yourself and meet other people in the room who are interested in the same topic. There is an excellent chance that the people you meet will have knowledge you don’t yet have, or they could learn something new from you. Don’t forget to exchange business cards and connect on LinkedIn if that is your thing. This network of industry friends can be a great resource when you need to dive into a technology that’s new to you.

At an event not that long ago, an industry friend was talking about an assembly issue they were having. I was listening with my fabrication mindset and realized that we were across the room from the material supplier. All three of us quickly huddled together, did a little brainstorming, and agreed on a few things that may be causing the issue. That, to me, is the power of networking with others in the industry and expanding your resources. One conversation provided options that days of emails would have had trouble accomplishing.

In my opinion, both the pace of technology and the frequency of needing to rely on technology that’s new to us to accomplish our goals is something that will be required at an increasingly fast pace, throwing us all out of our comfort zones. By expanding our own knowledge when attending industry events, being willing to share our knowledge with others, and being purposeful about meeting others in adjacent segments of the industry, we can shorten the learning curve and ease into those new technologies with more confidence. **FLEX007**

Tara Dunn is the president of Omni PCB, a manufacturer’s rep firm specializing in the PCB industry. To read past columns or contact Dunn, click here.

Electroless Plating for Flexible Circuits

EPTE Newsletter

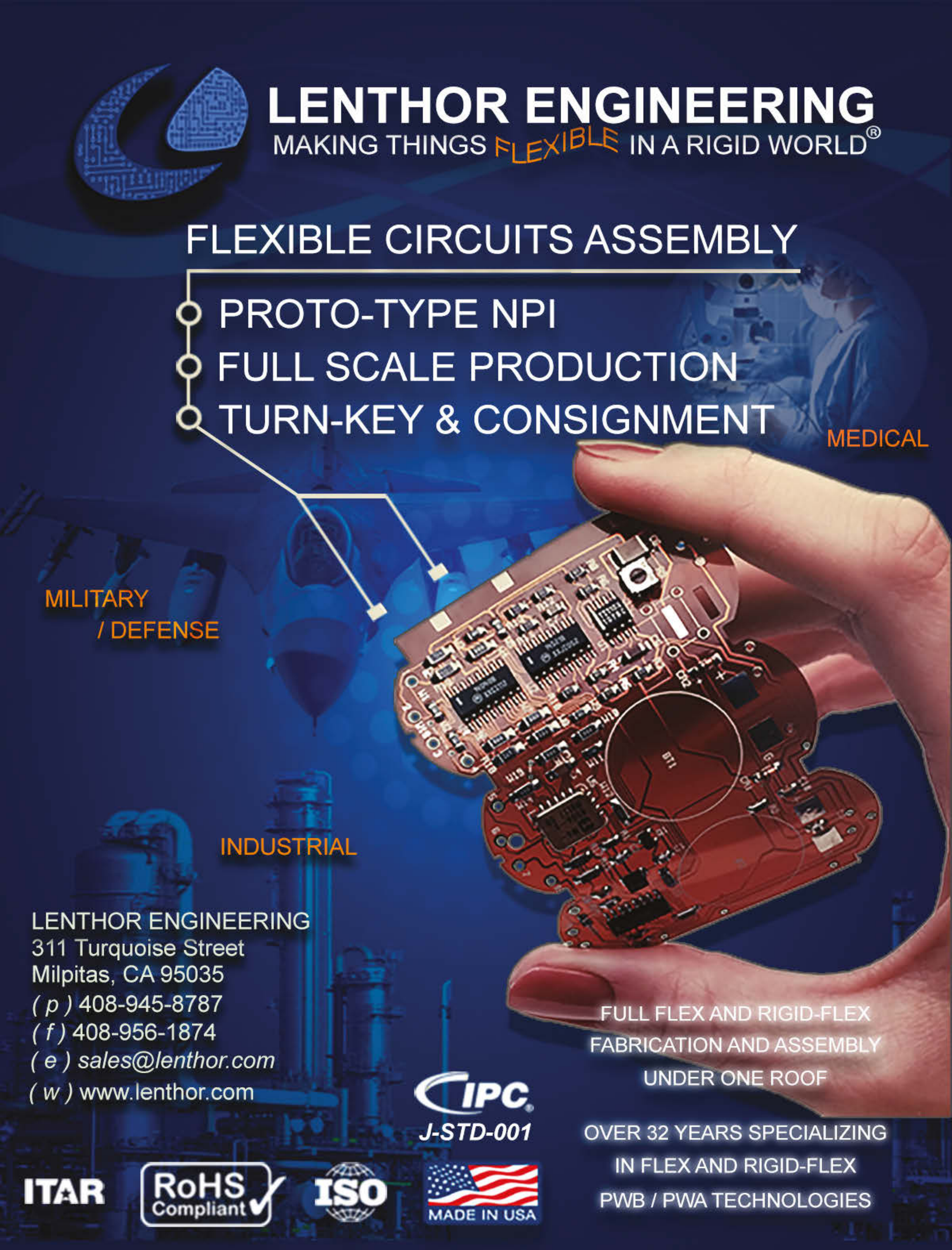
by Dominique K. Numakura, DKN Research LLC

Electroless plating is a popular method in manufacturing printed circuits or flexible circuits. The electroless plating concept is not new; it is used to protect conductors against oxidation. However, it is not the default process for printed circuit manufacturing. Many circuit manufacturers do not have in-house plating capabilities and rely on plating shops for surface finishing.

Over the last decade, there has been remarkable progress with electroless plating. Nowadays, material supplies have many new chemicals available for the plating process, especially for surface treatments. Chemical suppliers are confident that circuit manufacturers can secure metallizing on plastic substrates without making a large investment.

I participated in R&D projects during the '90s, where we tried to produce flexible copper laminates without adhesive layers. Manufacturers employed different processes, such as casting and sputtering, to generate new laminates. The combination of electroless plating and electrical plating was considered a candidate because it called for very little investment. Unfortunately, the chemical plating processes could not provide a secure enough bond strength between the base film and copper conductors and was not worth pursuing.

New plating chemicals are available to use with the plating process and provide a reliable bond strength with flexible laminates. However, plating shops do not follow the recipe line by line and create their own process conditions by adding supplemental treatments to increase the cost performance. Usually, chemical costs are significant.



LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®
FLEXIBLE CIRCUITS ASSEMBLY
PROTO-TYPE NPI
FULL SCALE PRODUCTION
TURN-KEY & CONSIGNMENT
MEDICAL
MILITARY / DEFENSE
INDUSTRIAL
LENTHOR ENGINEERING
311 Turquoise Street
Milpitas, CA 95035
(p) 408-945-8787
(f) 408-956-1874
(e) sales@lenthor.com
(w) www.lenthor.com
FULL FLEX AND RIGID-FLEX FABRICATION AND ASSEMBLY UNDER ONE ROOF
IPC J-STD-001
OVER 32 YEARS SPECIALIZING IN FLEX AND RIGID-FLEX PWB / PWA TECHNOLOGIES
ITAR
RoHS Compliant
ISO
MADE IN USA

Material suppliers have an advantage from electroless plating—they can experiment with new laminates using base materials other than polyimide films or PET films at a significantly lower cost. The plating process could be available for other materials such as rubber sheets, cloth, and papers.

Let's not forget the biggest advantage with electroless plating is the metallization of the non-conductive surface of plastic materials. This could be the key process from the semi-additive process to generate ultra-fine traces in high-density flexible circuits. The process is very simple; just dip flexible items in the plating baths few minutes, and that's it!

In my opinion, the electroless plating is the key to build high-density flexible circuits, but it is not easy to create stable plating conditions. Experiments and trial-and-error activities are necessary to create these conditions. You could consider the process condition and handling as intellectual property owned by the manufacturer.

If you are interested in learning more about electroless plating, feel free to contact haverhill@dknreseach.com or me with any questions or comments.

Headlines

1. Asahi Kasei Electronics (device manufacturer in Japan)

Rolled out small-size, 3D, magnetic smart switch "AK09970D" for wearable devices with 1.35 mm^2 WL-CSP package.

2. Toshiba (electric and electronics company in Japan)

Changed the name of Toshiba Memory to KIOXIA with a new logo and corporate color (silver); Kioku means "memory" in Japanese.

3. Mitsubishi Electric (electric and electronics company in Japan)

Developed a trench-type SiC-MOSFET with the lowest resistance rate of 1.84 milli-ohm/cm^2 for the power devices of home and industrial appliances.

4. Tokyo University (Japan)

Developed a cooling device, utilizing thermionic cooling effects of heterostructure GaAs/AlxzGaxAs instead of Peltier cooling devices.

5. TDK (Component supplier in Japan)

Will display seven categories of products at CEATE 2019, including some related to IoT, mobility, wellness, connection, energy, experience, and robotics.

6. NICT (R&D organization in Japan)

Successfully demonstrated 1 petabit per second switching with the collaboration of multi-core, optical fiber and fundamental optical network.

7. NTT Docomo (cellular phone carrier in Japan)

Started 4G LTE service with glass antennas developed with AGC, a glass product supplier in Japan.

8. Murata (component supplier in Japan)

Built two manufacturing plants in Thailand to expand the production capacity of EMI filters and antenna coils for mobile products.

9. Renesas (semiconductor manufacturer in Japan)

Unveiled the arm 32-bit microprocessor RA Family Series for IoT systems.

10. Kioxia (semiconductor manufacturer in Japan, formerly Toshiba)

Completed the building of the first plant in Iwate Prefecture; it will start the production of 3D NAND Flash memory in 2020.

11. YDB (market research company in Japan)

Forecasted the market of automobiles displays at 201.98 million units in 2023 from 166.46 million units in 2018. FLEX007

Dominique K. Numakura is the managing director of DKN Research LLC. Contact haverhill@dknresearch.com for further information and news. To read past columns by Numakura, click here.

Flex007 News Highlights

Trouble in Your Tank: Working With Flexible Circuits ▶

Even though they are a smaller part of the circuit board industry, flex and rigid-flex circuits have been growing in popularity over the last decade, and for good reasons. These circuits are made to be thin, flexible, and durable. However, in addition to the opportunities that come with flex and rigid-flex circuits, there are also challenges. Find out more here.

Flexible Thinking: Standards—An Industrial-strength Glue ▶

Just as there are many government bodies around the globe, there are hundreds of standards bodies around the world with sometimes conflicting missions in terms of the generation and guidance in the enforcement of industrial standards. In this regard, just as laws help to hold societies together, standards serve the vital purpose of holding industries together. They are an industrial-strength glue that holds the industry together.

Development of Flexible Hybrid Electronics ▶

This article will present a hybrid manufacturing process to manufacture FHE systems with a two-layer interconnect structure utilizing screen printing of silver conductive ink, filled microvias to connect ink traces at the different layers, and use of the traditional reflow process to attach the semiconductor chips to the printed substrates.

Insulectro and DuPont Experts Talk Flex Design ▶

Mike Creeden recently spoke with Insulectro's Chris Hunrath and DuPont's Steven Bowles at the DuPont Technology and Innovation Center in Sunnyvale, California. They discussed a variety of topics related to flex design, including the support structure that's needed in flex design, the everchanging world of flex materials, and the need for working with a flex fabricator as early as possible in the flex design cycle.

Decreasing Bend Radius and Improving Reliability—Part 1 ▶

Many of the issues that arise when using a flex circuit come from a lack of knowledge about how to properly design one, especially when the circuit is required to bend.

Amphenol Printed Circuits Recognizes Panasonic as a Qualified Vendor for Flexible Laminates ▶

Amphenol Printed Circuits has qualified Panasonic's Felios Flexible Laminates and will now increase usage on new part numbers.

Flexible PCB Maker Zhen Ding Likely Revise 2019 Revenue Forecast Upward ▶

Flexible PCB specialist Zhen Ding Technology is expected to revise its revenue forecast for 2019 to growth from flat as projected earlier, having posted strong sales for September and the third quarter.

BMW i Ventures Invests in Flexible Circuit Tech with CellLink ▶

BMW i Ventures has recently invested in CelLink, a San Carlos, California-based manufacturer of flexible circuit technology that delivers high-conductance, large-area, lightweight, and low-cost flexible circuits through a proprietary combination of manufacturing processes, designs, and materials.

Additive Manufacturing of PCBs

Flexible Thinking

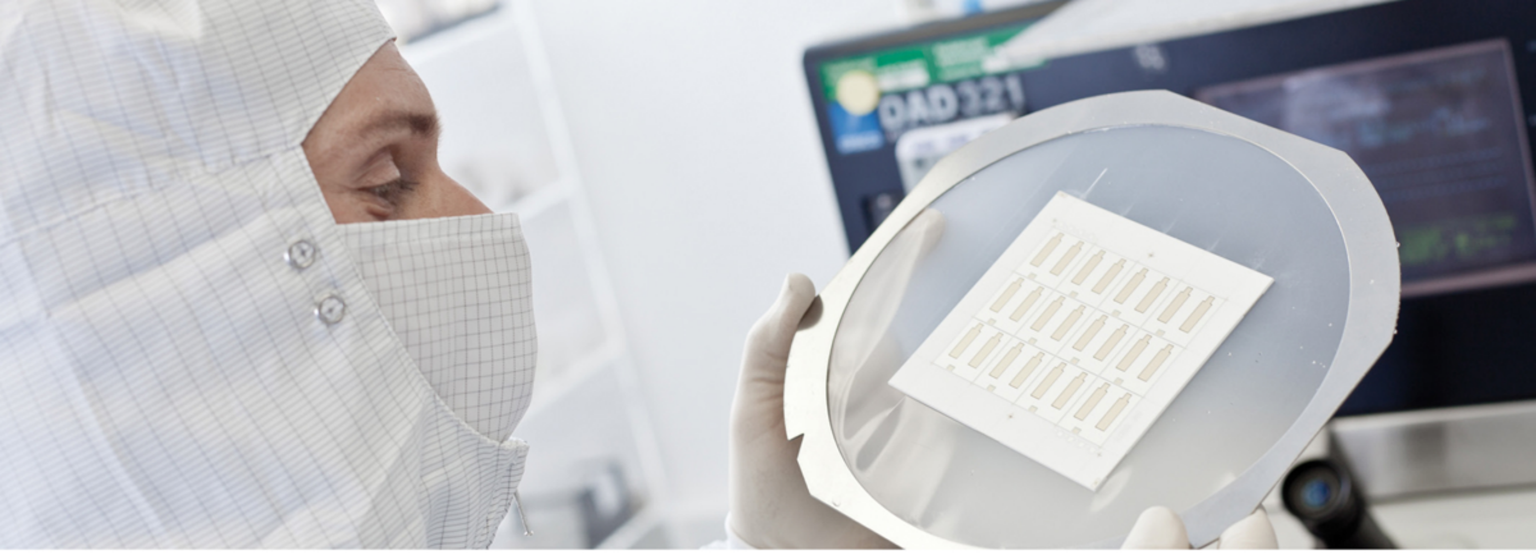
by Joe Fjelstad, VERDANT ELECTRONICS

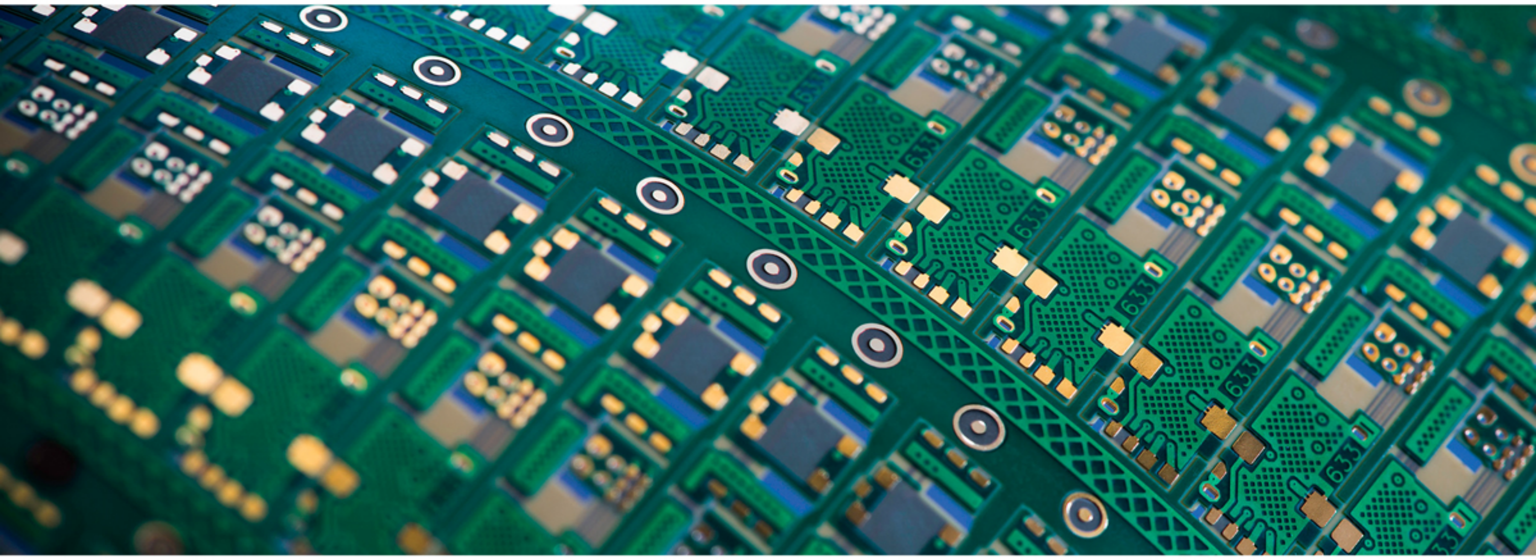
"It's hard to make predictions, especially about the future." –Yogi Berra

Yogi Berra and several other more scientifically minded individuals (including no less than pioneering physicist Nils Bohr) have been proven right time and again with their simple but astute observations about predicting the future. The future is indeed very difficult to predict, but certain things can be intuited with a modicum of logic and a bit of wild-eyed speculation. Some fellow soldiers and I, serving with the 101st Airborne, "invented" what is now called virtual reality (VR) and/or augmented reality (AR) in Vietnam 50 years ago during some of our less frenetic off-hours. It was based on our steel pot helmets, and it had a curved display that gave a 180° field of view and speakers placed around it that provided 3D sound (we did not, however, envision stereoscopic vision to provide 3D imaging, as I recall).

Of course, we had no way to implement our outrageous ideas because the technologies required to support our collective vision were not available until now. I am willing to assert without proof that there were likely others who may have had a similar and contemporary inspiration. Most certainly, there have been numerous others who have had similar ideas since, as evidenced by the Oculus VR headset, which is now available.

We are seeing increasing interest in technologies that will allow one to make electronic substrates in near real-time using additive processing techniques and 3D printers. It is a true game-changer in product development. The surge in interest in additive manufacturing technologies shown in recent times—as indicated by the significant increase in published articles and press releases—suggests that the electronic intercon-






109C

nection manufacturing industry could be on the verge of a manufacturing renaissance.

The fundamental concept of additive manufacturing of circuits is quite old, with roots that extend back to the birth of the industry in the 1940s. At that time, engineers and technology visionaries were exploring many new and different methods for printing electronic circuits, using combinations of conductive and insulating materials. They had the right idea at the time, but because the available equipment, materials, and processes were limited in terms of their capabilities, it remained largely a fringe technology limited to circuits that operated at relatively high voltages and low currents, such as keyboards for electronic products and methods that are still used today.

The fundamental concept of additive manufacturing of circuits is quite old, with roots that extend back to the birth of the industry in the 1940s.

The additive concept was picked up again by major OEMs, such as AT&T's Western Electric was making full additive PCBs in 1963 The processing was slow, however, taking many hours to build the copper metal to the desired thickness. It was advanced by the concept of semi-additive processing, which is heavily used today to manufacture high layer count PCBs for high-density circuit products used in many high-performance products.

Following my personal journey, I joined a small startup company (ultimately called Extend Length Flex (ELF) Technologies) in 1990. The founders had developed a technology for laser printing a catalytic toner which they intended to use to make interlayers; however, I convinced them that the technology would be ideally suited to laser printing the catalytic toner on a web of material to create flexible circuits of unlimited length as well as an economical run unit of one circuit because the data was all digital. That simple idea got the company funded, and while the process was demonstrated, the market was not ready. Today, technologies for direct web printing of circuits are available that allow those same possibilities but with capabilities in terms of circuit feature sizes that we did not dare to dream of back then.

Fast forward to the early 2000s when Seiko Epson showed off a multilayer printed circuit where each layer of dielectric and conductors was printed in sequence, producing some very fine circuit details. The technologies have continued to advance and improve, and it was technologies like this that inspired personal confidence that it would be possible to print circuits directly onto the planar termination of component contacts, bypassing the soldering step altogether.

The inspiration was provided to me as a result of my years-long opposition to the EU's lead-free mandate. In the end, I christened the new solderless approach to manufacturing idea "The Occam Process," owing its simplicity and adherence to the principle of Occam, who said, "It was vanity to do with more that which could be done with less." The process eliminated the most vexing process step of all in the manufacture of electronics—soldering and all of its negative attributes.

And for those who doubt the assertion of solder being a highly problematic technology, I invite you to look at any issue of any electronics journal, magazine, or newsletter, past or present, and count how many articles and papers have solder as a central theme: materials, equipment processes, defect detection (and continually promised mitigation), and failure analysis. Then, you can determine what percentage of the total number of articles they represent.

Today, it is possible to build electronic assemblies without the use of solder with existing equipment (à la Occam). It involves a simple reversal of the process, that is to build a component board first and then additively apply the circuits in a buildup fashion. For those with greater interest, I encourage you to read my series "The Occam Files" on my LinkedIn page.

Conclusion

In conclusion, today's additive electronics manufacturing is poised to deliver on promises made long ago. It is a way to deliver to the consumer the most precious of all in this physical world of ours: time. Time may itself be infinite, but everything else in life is finite.

If a manufacturer can develop and deliver a prototype product faster than their competition, they will own the market, even if only temporarily (big companies tend to steal ideas if they are good enough as industrial icons Bill Gates and Steve Jobs both correctly observed and arguably did themselves). Speed is the only defense the small innovative company has over its behemoth competitors. Embrace change and take additive technology to heart; it will give small companies a fighting chance in an increasingly brutal business world.

To end, I would like to offer my take on Yogi's opening observation: "The future will never be what it used to be, so embrace change."

FLEX007

Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 150 patents issued or pending. To read past columns or contact Fjelstad, click here.

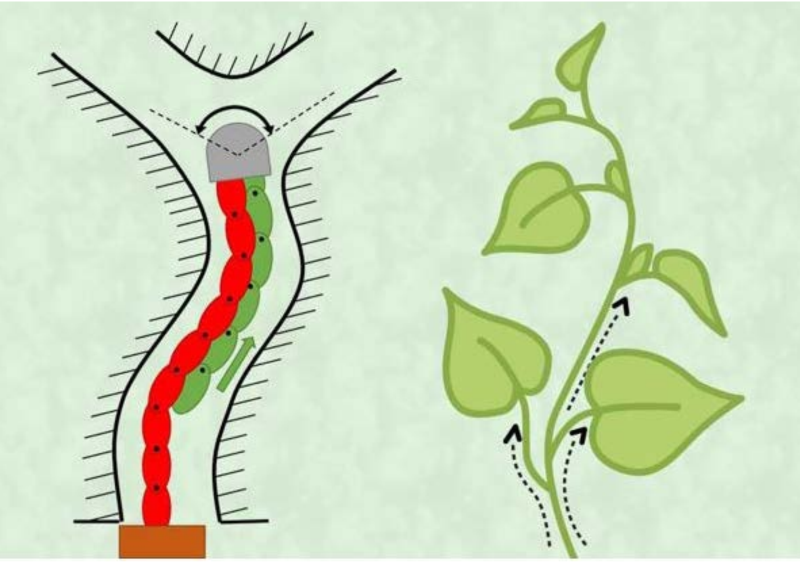
Flexible yet Sturdy Robot Is Designed to 'Grow' Like a Plant

In today's factories and warehouses, it's not uncommon to see robots whizzing about, shuttling items or tools from one station to another. For the most part, robots navigate pretty easily across open layouts. But they have a much harder time winding through narrow spaces to carry out tasks such as reaching for a product at the back of a cluttered shelf or snaking around a car's engine parts to unscrew an oil cap.

Now MIT engineers have developed a robot designed to extend a chain-like appendage flexible enough to twist and turn in any necessary configuration yet rigid enough to support heavy loads or apply torque to assemble parts in tight spaces. When the task is complete, the robot can retract the appendage and extend it again, at a different length and shape, to suit the next task.

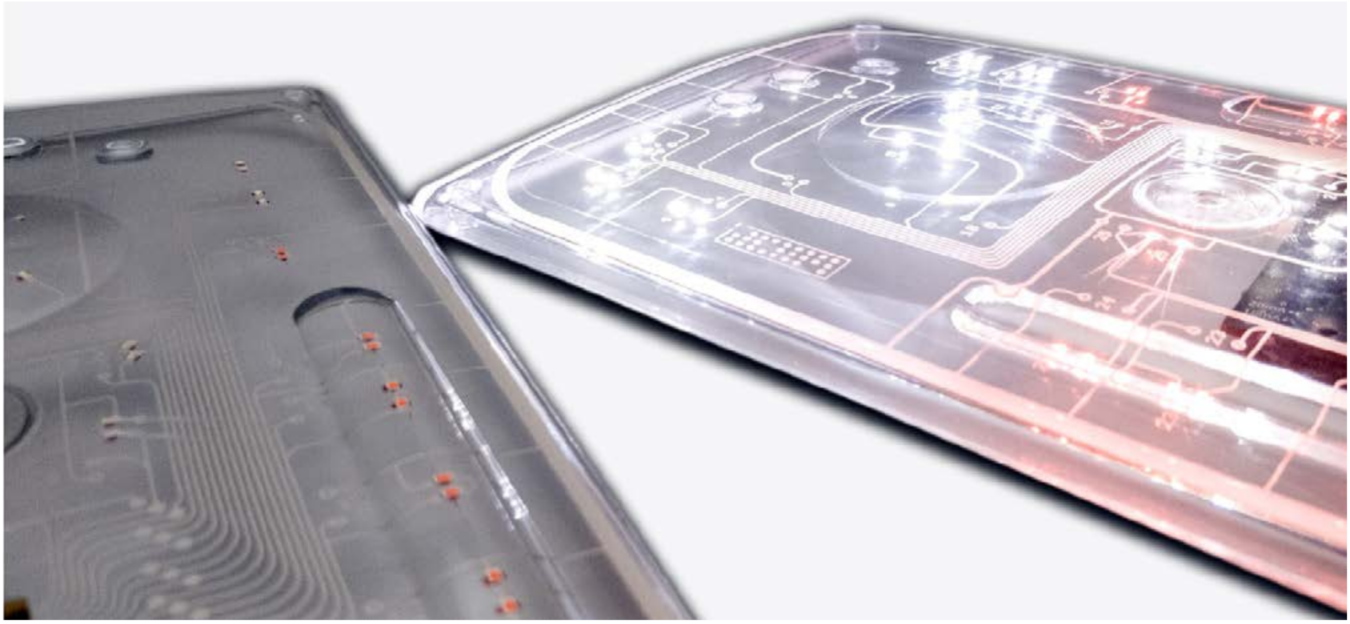
The appendage design is inspired by the way plants grow, which involves the transport of nutrients, in a fluidized form, up to the plant's tip. There, they are converted into solid material to produce, bit by bit, a supportive stem.

Likewise, the robot consists of a "growing point," or gearbox, that pulls a loose chain of interlocking blocks into the box. Gears in the box then lock the chain units together and feed the chain out, unit by unit, as a rigid appendage.



The researchers presented the plant-inspired "growing robot" recently at the IEEE International Conference on Intelligent Robots and Systems (IROS) in Macau. They envision that grippers, cameras, and other sensors could be mounted onto the robot's gearbox, enabling it to meander through an aircraft's propulsion system and tighten a loose screw or to reach into a shelf and grab a product without disturbing the organization of surrounding inventory, among other tasks.

(Source: MIT News)



Smart Molded Structures Bring Surfaces to Life

Article by Outi Rusanen, Janne Asikkala, Mikko Heikkinen, Paavo Niskala, and Tomi Simula
TACTOTEK

Abstract

This article introduces structural electronics technology, enabling smart molded structures. It also presents a case for developing industry standards specific to structural electronics materials, processing, and testing.

In Part 1, we outline the benefits of structural electronics technology as well as manufacturing processes. Smart molded structures are made by integrating and encapsulating printed electronics and standard electronic components within durable, 3D injection-molded plastics. Structural electronics technology and processing differ significantly from conventional electronics. Thus, we describe how these differences influence component and materials certification for use within injection molded plastics. In Part 2, we discuss the lack of suitable standards for structural electronics. Two technologies, bearing a resemblance to structural electronics, have standards. However, they do not cover all aspects, and we see a need for further development.

Part 1: Structural Electronics

Introduction to Structural Electronics Technology and Manufacturing

Structural electronics technology enables design innovation by integrating electronic functions into 3D injection molded plastic structures. Features, such as controls, sensors, illumination, and communications, are embedded in thin 3D structures with plastic, wood, and other surfaces (Figure 1). The structures are light, thin, and durable.

In conventional use cases, such as an in-vehicle control panel, a single part replaces a multi-part conventional electronics structure and eliminates labor-intensive electro-mechanical assembly. The part also weighs less

Figure 1: Structural electronics can also have natural surface finishes.

Figure 2: TactoTek has demonstrated structural electronics designs with 70% weight and 90% thickness reduction when compared with conventional multi-part assemblies.

and is significantly thinner. TactoTek has demonstrated structural, electronic designs with 70% weight and 90% thickness reduction when compared to conventional multi-part assemblies (Figure 2).

Core manufacturing processes for structural electronics are printing, surface mounting, forming, and injection molding (Figure 3). Taken individually, these processes are mature, and we use standard equipment suitable for mass production. However, the standard processes are combined in a unique way during the manufacturing of structural electronics.

Printing is the first core manufacturing process. Electronics and decoration (graphic inks) are printed onto plastic film or another suitable substrate material. Electronics are typically printed using silver (Ag), conductive inks, and dielectric inks to insulate between layers of circuitry. The outputs are two kinds of films: electronic and surface. The latter films are used for decorations, such as icons for the human-machine interface. In some cases, a single film can be used for both decoration and electronics. Both films can also be a substrate for electronics.

Surface-mounting technology (SMT) is the second core process. Components are placed and bonded, mechanically and electrically, onto electronic films. The output is a 2D film substrate with components. Forming is the third core process. 2D electric and graphic films are thermoformed into a 3D shape and trimmed as needed. Outputs are 3D electric films with components and 3D graphic films.

Injection molding is the fourth core manufacturing process. 3D electric films and 3D graphic films are used as inserts in an injection molding tool, and a plastic resin, such as polycarbonate (PC), is injected between the films, resulting in a single molded part. The output is a strong and durable structure in

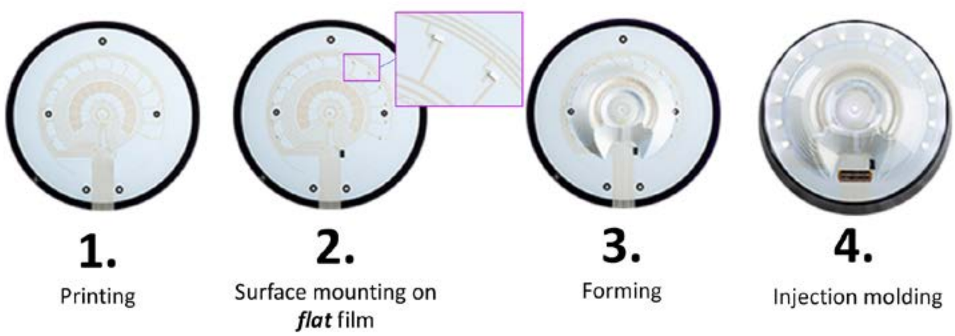


Figure 3: Core manufacturing processes for structural electronics.

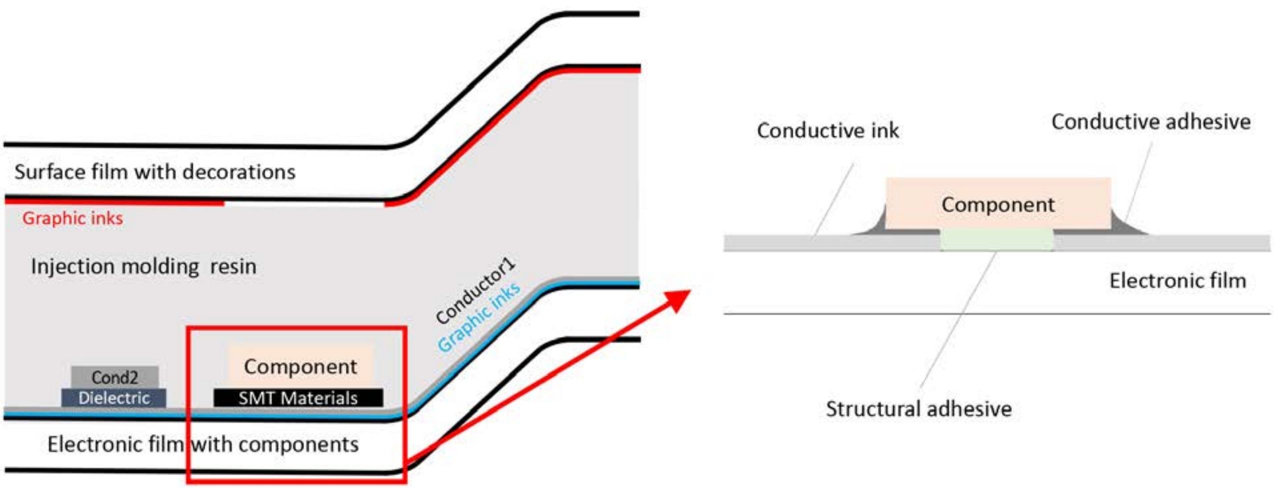


Figure 4: Typical material stack in structural electronics.

which electronics are encapsulated within the molded plastic. Figure 4 illustrates the material stack. Structural electronics manufacturing often includes pre-assembly and the final assembly of control electronics as well.

Introduction to Component Certification Process

Structural electronics technology differs significantly from conventional electronics. Currently, most electronic components are optimized for conventional electronics manufacturing that do not anticipate the temperature and pressure exposure of thermoforming and injection molding processes. Thus, TactoTek certifies all electronic components that are embedded inside injection molded polymers. Certification has three steps (Figure 5).

TactoTek has defined the requirements for ideal component packaging [1]. During pre-screening, component data is compared with the ideal package. The component package does not need to fulfill all requirements to pass this step. However, there are some items that cause failure at pre-screening—for example, a package with moisture sensitivity level (MSL) of four or higher fails.

If a component passes the pre-screening, TactoTek manufactures test platforms with that component, using internal company standards

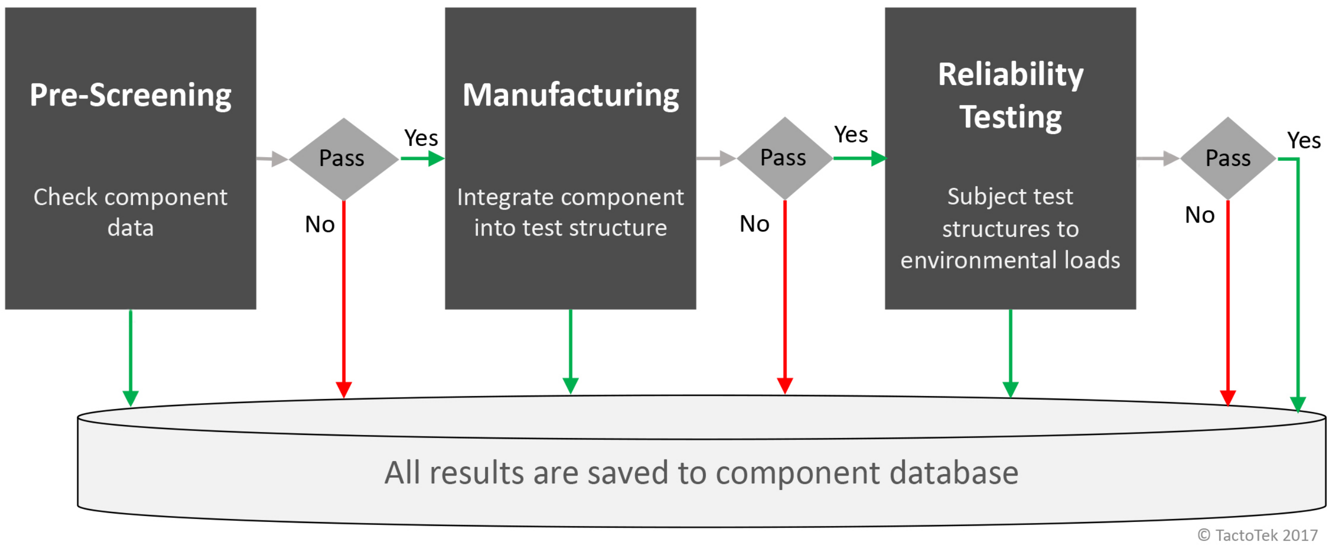


Figure 5: Component certification process flow.

for certification layout and material stack. Components undergo surface mounting, forming, and injection molding, and they are tested after each process step. The testing system is also standardized. If the manufacturing yield is sufficient, the test platforms are subjected to reliability testing. Typical environmental loads include change of temperature as well as elevated temperature humidity. Based on test results and physical failure analysis, components can receive certification for use in commercial projects.

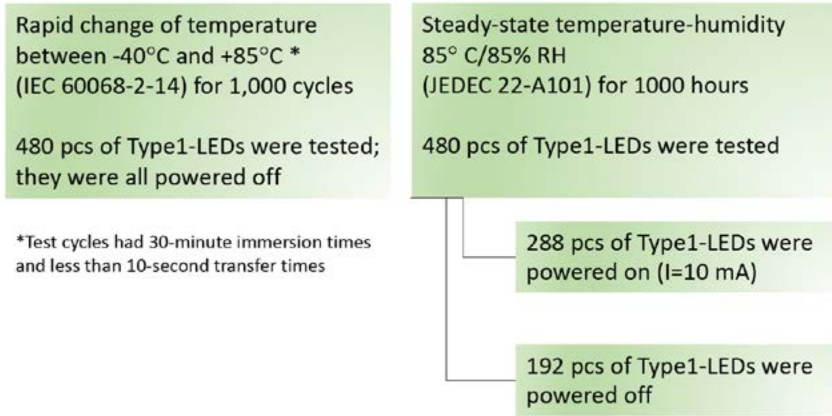


Figure 7: Summary of tests and sample sizes in Type1-LED reliability testing.

Reliability Testing Results

The results presented here are part of the certification process for Type1-LED that was tested using an internal certification platform (Figure 6). Each certification platform contains 48 Type1-LEDs. We have surface-mounted some of the Type1-LEDs on top of small radius 3D curves. This is against our design guidelines, but we wanted to push the limits of the technology and gain an understanding of potential failure modes under adverse conditions. Figure 7 shows a summary of the tests and sample sizes.

At the end of the reliability tests, we checked the functionality of the Type1-LEDs. The rapid change of temperature test did not cause any Type1-LED failures; all of the 480 Type1-LEDs were functional after 1,000 cycles between -40°C and +85°C. Steady-state temperature/humidity (85°C/85% RH) caused one Type1-LED to fail. The failed LED was in a certification platform that had been powered on during testing. The other 479 Type1-LEDs were functional after 1,000 hours of elevated temperature and humidity. We also measured light luminance and color coordinates from some of the tested Type1-LEDs. Reliability testing did not cause any adverse effects.

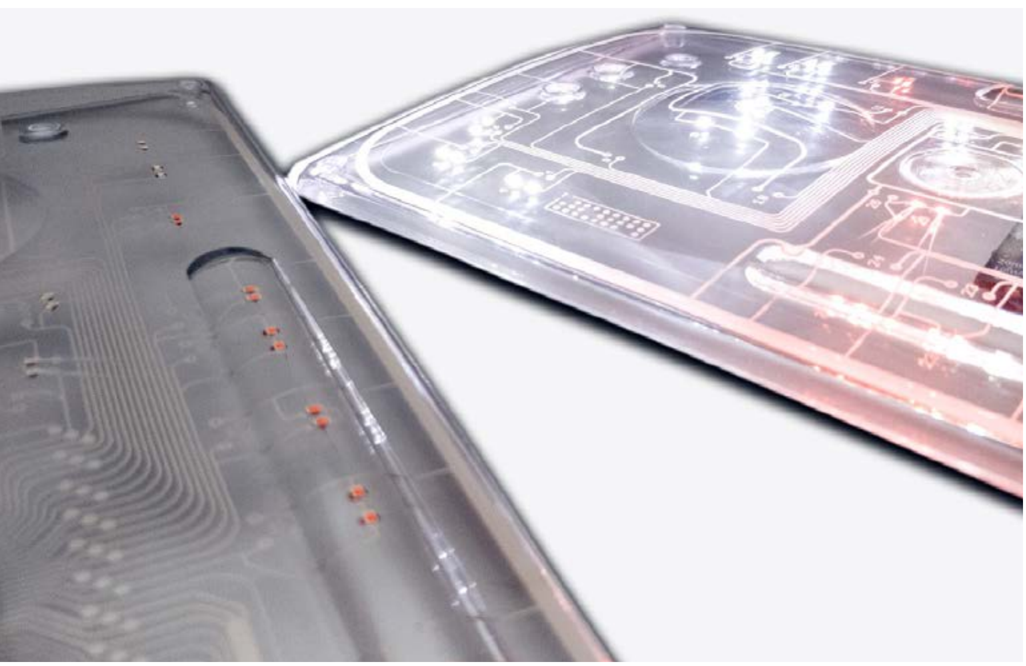


Figure 6: Photo of two certification platforms that were subjected to extended reliability testing.

Physical Failure Analysis Results from Type1-LED

Electrical measurements indicated that the failed Type1-LED had an internal component short-circuit. Nevertheless, we made a cross-section of the failed component because it had been mounted on top of the thermoformed 3D-curve. The cross-section showed that the component base had bent slight-

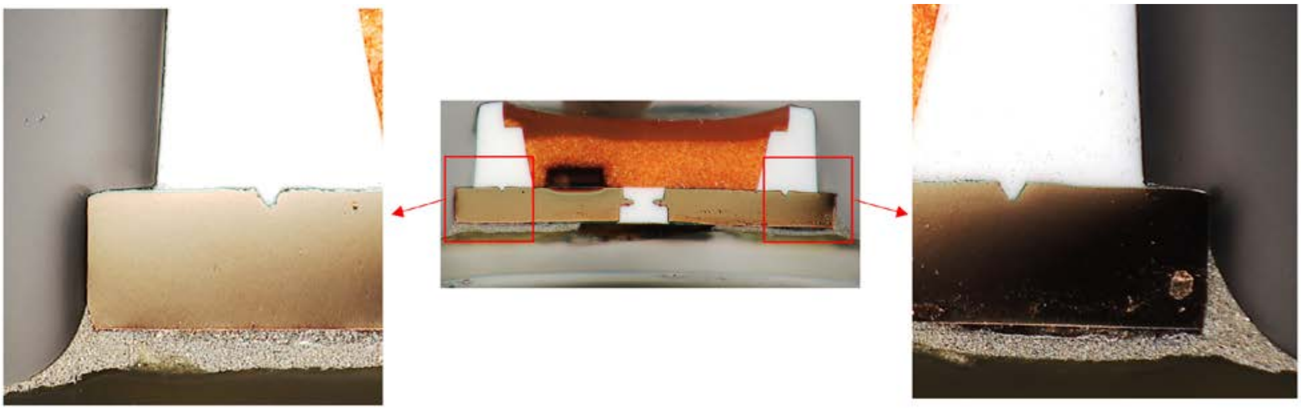


Figure 8: Cross-section shows a slight bending of the component base and partial delamination of conductive adhesive.

ly. In addition, the conductive adhesive bond on one side had partly delaminated (Figure 8). As stated before, component mounting to 3D-curves with a small radius is against our design guidelines.

Extended Reliability Testing Results

Testing until failure is useful for better understanding structural electronics reliability. That is why we tested one of TactoTek demonstrator products for 3,000 cycles in the change of temperature test. The demonstrator product, shown in Figure 9, contains 20 pieces of Type2-LEDs. Figure 10 shows a summary of the tests and sample sizes.

Two Type2-LEDs failed during 3,000 cycles. Failures occurred at 702 and at 2,988 cycles. When we fit this data into a Weibull distribution, we predict a 50% failure point at over 10,000 cycles (Figure 11). Such a high value demonstrates the reliability of the technology.

Figure 9: Photo of a demonstrator product that was subjected to extended reliability testing.

However, the number of tested components was small in the extended reliability testing. Thus, we have subjected more certification platforms to extended reliability testing.

Change of temperature
between -40°C and +85°C *
(IEC 60068-2-14) for 3,000 cycles

20 pcs of Type2-LEDs were tested;
they were all powered on (I=15 mA).

*Test cycles had 1-hour exposure (i.e., dwell) times and 1-hour ramp times; one test cycle lasted for four hours

Figure 10: Summary of the test and sample sizes in extended reliability testing.

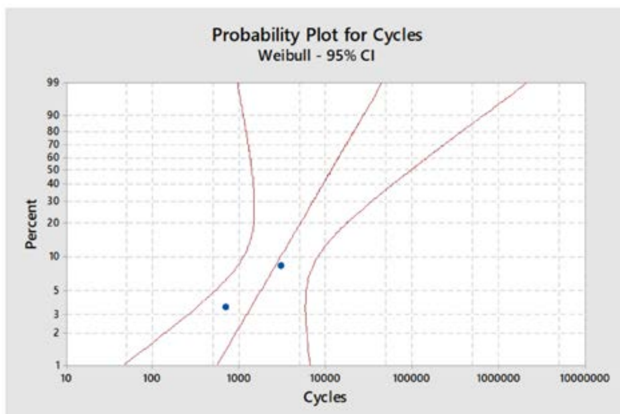


Figure 11: Extended reliability testing failures in Weibull distribution.

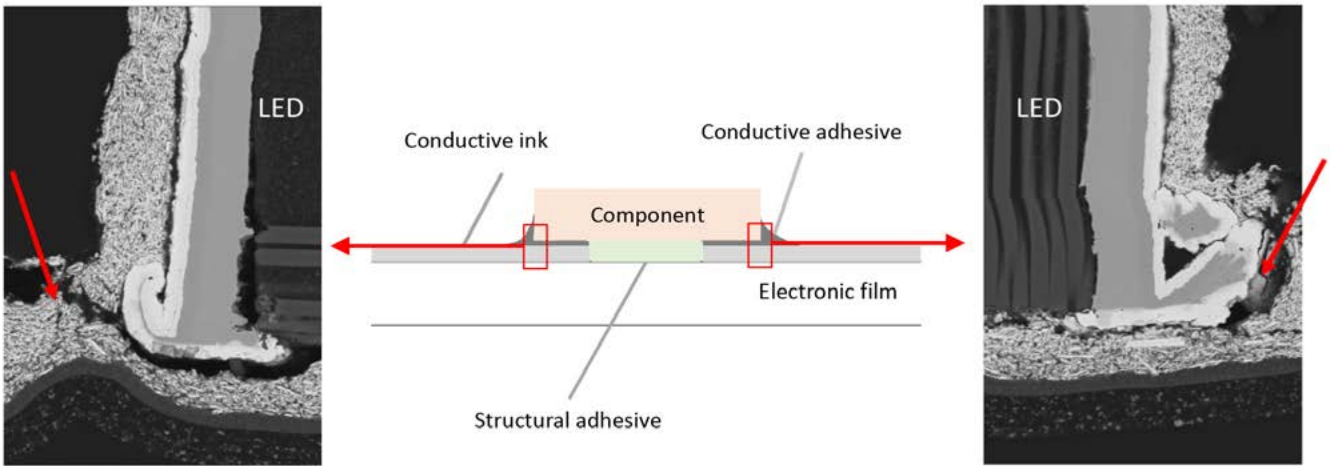


Figure 12: SEM-images of the cross-section show a fracture in the conductive adhesive used for surface mounting.

Physical Failure Analysis Results from Failed Type2-LED

We performed cross-section and scanning-electron-microscope (SEM) analysis on a failed Type2-LED. It showed that the failure mode is a fracture in the conductive adhesive used for surface mounting the component to the electronic film, Figure 12. The failure mechanism is conductive adhesive creep caused by thermo-mechanical stresses during thermal cycling.

Conclusions

We have demonstrated structural electronics designs with 70% weight and 90% thickness reduction when compared with conventional multi-part assemblies. Structural electronics is also a reliable technology. The tested certification platforms and demonstrator products endured thermo-mechanical stresses and elevated temperature-humidity. Injection molding resin strengthens the structures and also protects electronics from environmental conditions, such as moisture, dust, and mechanical impacts.

Part 2: Standardization for Advanced Technologies

Many PCBA Standards Are Not Applicable for Structural Electronics

Structural electronics technology differs significantly from conventional electronics. This means that many PCBA standards are not relevant to structural electronics. For example, the IPC-9704 Printed Circuit Assembly Strain Gage Test Guideline defines how to measure strain in rigid PCBs during the assembly process. Even if IPC-9704 defines only the measurement method, some OEM manufacturers have defined strain limits and require compliance from their suppliers. TactoTek substrates are thin and flexible plastic films. They inherently bend and cannot comply with the strain limits made for rigid PCBs.

PCBs also tolerate higher temperatures than most plastic films. The typical maximum operating temperature for FR-4 is around 130°C. Many elevated temperature and thermal cycling tests assume that the substrate material is FR-4. Thus, they have a maximum temperature of 125°C or even 155°C. Usually, structural electronics substrate materials cannot tolerate those temperatures. Furthermore, temperatures, such as 125°C or 155°C, may not be relevant for the application environment.

Two Similar Technologies Have Standards

Printed electronics and device-embedded substrate technologies bear a resemblance to structural electronics. Figure 13 shows their definitions. TactoTek follows printed electronics standardization developments in IEC and IPC. We have also acquired relevant IEC standards for device-embedded substrates.

Printed Electronics

Devices and systems using printed electronics processes during manufacturing.

Device Embedded Substrates

A substrate in which active device(s) (semiconductor device) and/or passive devices(s) (e.g., resistor or capacitor) are formed using thick-film technology or embedding them within the substrate.

Figure 13: Definitions for printed electronics and device-embedded substrates.

We welcome the fact that standards for device-embedded substrates recognize the thermal limitations of substrate materials. For example, standard IEC 62878-1 ED1 states, "Compared to the test severities applied to bare printed wiring boards, limitations exist, which are determined by the sensitivity of embedded components and substrate material. Requirements and severities shall be defined between user and supplier." In our opinion, this is a good practice because users and suppliers can select temperatures that are relevant to the application environment.

Two Organizations Create Printed Electronics Standards

IEC and IPC are both active in printed electronics standardization. They have organized printed electronic standard development work into (technical) committees. Figure 14 shows the working groups and sub-committees in them. The standardization scopes are similar even if the naming differs. By June-2018, IEC had published around 20 standards and IPC around 10 standards for printed electronics [2 & 3]; both organizations have many standards in the pipeline.

IEC has Printed Electronics Technical Committees (TC-119) with the following working groups:

- TC 119-1: PE Terminology
- TC 119-2: PE Materials
- TC 119-3: PE Equipment
- TC 119-4: PE Printability
- TC 119-5: PE Quality Assessment

IPC has Printed Electronics Committee (D-60) with the following subcommittees:

- D-61: PE Design
- D-66: PE Processes
- D-66A: 3D PE Processes
- D-62: PE Base Materials / Substrates
- D-63: PE Base Functional Materials
- D-64: PE Final Assembly
- D-64A: PE Terms and Definitions
- D-65: PE Test Method Development and Validation

Figure 14: IEC working groups and IPC sub-committees in printed electronics [4 & 5].

Structural Electronics Technology Needs New Standards

Current printed electronics standards lack information that is important for structural electronics. An example is a method for resistance-strain measurements. During thermoforming, 2D films with conducting layers are formed into 3D shapes. The films and conducting layers deform plastically. Printed electronics standards have some deformation related items [6–9]; however, the deformations and stresses are in the elastic region only.

For companies to design reliable structural electronics solutions, they must know how conductive layer resistance changes as a function of plastic strain. This information is seldom available from ink suppliers. Moreover, there is no shared measurement or analysis methodology. Thus, results cannot be compared to ink suppliers. TactoTek made resistance-strain measurements for conductive inks in 2015, has continued developing the method since and is looking to refine it into an international standard.

If a test method standard becomes an IEC or IPC standard, will the other standard organization develop a similar one but with different test methods and requirements? TactoTek does not want this to happen. In a truly global industry, such as (structural) electronics, it would be a waste of effort to have overlapping work in two standardization organizations, with co-operation and similar standards being the goal.

Conclusions

Structural electronics use flexible plastic films as substrates. Thus, the standards that are based on typical PCB stiffness and temperature endurance are not applicable to structural electronics. In TactoTek's opinion, IEC device embedded substrate standards have selected a good approach that does not hinder the utilization of new and evolving technologies, which could be adopted in other standard documents. These standards allow the user and supplier to select temperatures that are relevant to the application environment. We welcome this approach and hope to see it in other standards going forward.

Acknowledgments

The authors thank Tuomas Kallio, Antti Keränen, Pasi Korhonen, Dave Rice, and Johannes Soutukorva for their much-valued contribution to this work. FLEX007

References

1. T. Simula, P. Niskala, M. Heikkinen, & O. Rusanen, "Component Packages for IMSE (Injection Molded Structural) Electronics," NordPac Proceedings, 2018.
2. IEC Webstore, "IEC 62899," accessed June 2019.
3. IPC Online Store, "Printed Electronics," accessed June 2019.
4. IEC, "IEC Standard Development Working Groups," accessed August 2018.
5. IPC, "IPC Public Groups," accessed August 2018.
6. Standard IPC-4921, "Requirements for Printed Electronics Base Materials (Substrates)."
7. Standard IPC-4591, "Requirements for Printed Electronics Functional Conductive Materials."
8. Standard IEC 62899-201-2, "Printed Electronics–Part 201-2: Materials–Evaluation Methods of Stretchable Substrates."
9. Standard IEC 62899-201-4, "Printed Electronics–Part 201-4: Materials–Evaluation Methods of Stretchable Functional Ink (Conductive Ink and Insulator Layer)."

This paper was first presented at the IPC APEX EXPO 2019 Technical Conference and published in the 2019 Technical Conference Proceedings.

All of the authors are with TactoTek in Oulu, Finland.
Outi Rusanen is senior interconnect specialist.
Janne Asikkala is prinicpal research chemist.
Mikko Heikkinen is chief engineer and co-founder.
Paavo Niskala is VP of advanced engineering.
Tomi Simula is principal integration specialist.

Outi Rusanen

Janne Asikkala

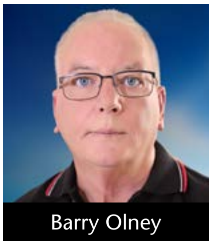
Mikko Heikkinen

Paavo Niskala

Tomi Simula

Editor Picks from PCBDesign007 and Flex007

1 Beyond Design: My 100th Column ▶


Barry Olney

Believe it or not, Barry Olney has written 100 "Beyond Design" columns. To wrap it up, he looks back over the past 99 columns and reflects on what he believes to be the most enlightening for high-speed PCB designers, counting down in reverse order of preference.

2 ODB++: Transforming Ideas Into Products ▶

The ODB format originated with the objective of delivering on this need. The format was originally introduced for use by PCB fabricators, eliminating the need for a collection of CAM files in multiple formats—such as Gerber, Excellon, IPC-356, or even IPC-350, which was an early attempt to simplify this process. The key to the success of ODB was that it obtained industry acceptance.

3 Tim's Takeaways: Realizing a Higher Standard for OCB Design ▶


Tim Haag

To the untrained eye, one circuit board may look pretty much like any other, but as we know, there are major differences between them. Not only are they different in purpose and design but also in how they are manufactured for specific industries. If you are designing medical equipment, for instance, you will have to meet many different regulatory requirements from organizations, such as the FDA, WHO, and IEC, among others.

4 Cadence Reports 9% YoY Revenue Growth in 3Q19 ▶

Cadence reported third quarter 2019 revenue of $580 million, compared to revenue of $532 million reported for the same period in 2018.

5 Time to Market: The Importance of Timely NPI ▶

In this new column, Imran Valiani plans to address ways to get products to market as quickly as possible.


Imran Valiani

6 Fresh PCB Concepts: Getting It Right From the Start ▶

When faced with critical time-to-market situations, it is all too easy to say, "It doesn't matter because this is just the prototype; we can fix this later." However, if the design is perfected from the beginning, cost savings can be applied, and manufacturability can be ensured. Perhaps most importantly, the design can be adapted with reliability in mind, leaving a seamless transition from prototype to production. How do we get it right from the start?


Jeff Beauchamp

7 EMA Design Automation Celebrates 30th Anniversary ▶

EMA Design Automation, a full-service provider and innovator of electronic design automation (EDA) solutions, is celebrating 30 years since it began selling EDA tools in 1989 and has since continued to serve the North American EDA industry with innovative products, support, and services.

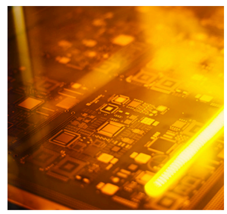
8 IPC-2581 Continues to Flourish ▶

Because standards adoption in this industry tends to be akin to turning around a battleship, what is often seen as new has generally already been in production for 10 years or more. Often, there are outside influences and dependencies that cause this, so most of us tend to take a "wait-and-see" approach—let others shake out the issues. And even then, we only look at adoption if the mandate comes from higher-ups in the organization. It works the way we do it now, so why change?


Linda Mazzitelli

9 Printed Circuits Installs New Notion n.jet Direct Solder Mask and Legend Printer ▶

Rigid-flex circuit board manufacturer, Printed Circuits, has purchased and will install a new Notion Systems n.jet direct solder mask and legend ink printer.



10 AltiumLive 2019 Frankfurt: A Perfect Mix of Education and Fun ▶

The AltiumLive PCB Design Summit in Frankfurt, Germany, has come to a close, with Happy Holden's keynote signaling the end of this three-day event. Here's a quick wrap-up of this year's event.


Andy Shaughnessy

Career Opportunities

Pssst!
Are You Looking for Someone?

Place your notice in our Help Wanted section.

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

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

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

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

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

Quality Engineer

Scope of Responsibilities

Under the supervision of the director of quality, the quality engineer is responsible for various documentation and reports that support customers and internal systems. This individual will be key in supporting our current quality systems (AS9100, ITAF16949, and ISO9001) and our MIL certifications. This person will also be expected to use tools like PPAPs, FMEA, control plans, and MSA. This individual will ensure that all internal quality systems are effective via auditing processes and procedures. Reporting results to management is a critical part of this job. Following through to resolve any deficiencies or opportunities is required.

Attributes Desired

- Manage time (and have organizational skills) that will assist in taking on high-profile tasks and completing them in a timely manner
- Work independently with minimal supervision
- Advanced computer skills
- Effective oral and written communication skills

Qualifications

- Internal auditing experience with an emphasis on ISO9001, AS9100, and IATF 16949
- Certified internal auditing is desired
- A bachelor's degree in a related field is desired; an associate degree with a minimum of five years of manufacturing experience will be considered
- U.S. citizenship or proper documentation to work legally within the U.S.

apply now

INDIUM CORPORATION®

Technical Support Engineer III

The technical support engineer III is responsible for providing leading-edge, high-level technical support to Indium Corporation's customers, potential customers, and sales staff. Due to their senior position and experience, their role also includes conceiving and devising projects, assisting with staff career development, marketing guidance, and more. The technical support engineer III has learned, mastered, and demonstrated unique and specific skills and information throughout their career. They are responsible for at least one sales territory and for leading other engineers. They train and evaluate colleagues on unique and general information. Continuing education/training is critical.

Requirements

- Technical undergraduate degree (B.S. in engineering, chemistry, physics, metallurgy, or materials science)
- 15 years of direct technical experience in applied materials science, electronics assembly techniques, and/or electronics assembly technical service
- Demonstrated technical competency
- Strong interpersonal, communication, and presentation skills
- Ability to work, with ease, with executive-level counterparts
- Strong alignment with the corporate and departmental missions
- Ability to work cooperatively and effectively in a cross-functional team environment
- Ability to travel with limited notice
- Proficient in Word, Excel, and PowerPoint
- Experience with JMP or Minitab preferred
- Special consideration is given to candidates with language skills in Spanish and/or Chinese

apply now

Career Opportunities

Career Opportunities

FREEDOM CAD SERVICES

Sr. PCB Designer—Mentor Xpedition

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

Key Qualifications

- EXPERT knowledge of Xpedition VX 2.x
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF, and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Excellent team player who can lead projects and mentor others
- Self-motivated with the ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem-solving skills
- Other design tool knowledge is considered a plus (Altium, Allegro, PADS)

Primary Responsibilities

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

apply now

ROGERS CORPORATION

Advanced Connectivity Solutions

Senior Development Engineer

Rogers Corporation is seeking a senior development engineer accountable for the development of more complex products and processes, the establishment of sound technical bases for these developments, and effective interaction with technology, process, and platform innovation; operations; sales and marketing; and process engineering personnel to commercialize these developments.

Essential Functions:

- Design and conduct experiments and interpret the results
- Report on projects in both written and verbal formats at all levels of the organization
- Perform technical troubleshooting of new products and processes; act as new product/concept incubator for new technologies and platforms, identifying opportunities for improvement and incorporation design for manufacturing requirements resulting in a viable, scalable product
- Provide ongoing process and manufacturing support to newly launched products as applicable
- Provide support in terms of analytical equipment maintenance, methods development, material analysis, and documentation of new process or products
- Manage capital projects for the purchase and installation of new process or support equipment; train employees in new processes

Required Education and Experience:

Ph.D., Ch.E., M.E., or material science, or B.S. or higher in a technical discipline with accomplishment in product development and project management.

Rogers Corporation provides equal employment opportunities to minorities, females, veterans, and disabled individuals as well as other protected groups.

apply now

Career Opportunities

Field Service Engineer
(Location Negotiable)

Are you passionate about delivering an exceptional user experience? Come work as a field service engineer at the industry's leading inspection company that offers great benefits with opportunities to advance while learning alongside accomplished business leaders.

The Company: Koh Young is the leading 3D inspection solutions provider in the electronics manufacturing industry. With is new offices in Atlanta and Guadalajara, it helps its customers optimize their printed circuit board assembly process.

The Position: Deliver technical services–including installation, support, and maintenance–to elevate the user experience. Location is flexible, but OH, IN, IL, MA, MI, FL, CA, or Toronto are desired.

The Reasons: An opportunity to apply leading-edge inspection technology to products you know and use every day. A great environment that supports its team and treats everyone like family.

Join the industry's leading provider of
true 3D inspection solutions

Interested? Submit your resume below.

apply now

Gardien Is Hiring!

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

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

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

Gardien is an equal opportunity employer. Employment decisions are made without any regard to race, color, religion, national or ethnic origin, gender, sexual orientation, age, disability, or other characteristics.

Interested candidates, please contact us with your resume and a cover letter. Kindly note that only shortlisted candidate will be contacted.

Apply at careers@gardien.com.

apply now

Career Opportunities

TAIYO AMERICA

Assistant Department Manager, Operations, Carson City, NV

This is an entry-level professional management trainee position. Upon completion of a 1-2-year apprenticeship, this position will be elevated to facility/operations manager. Primary functions during training: shadow incumbent staff managers to learn and understand the operations and personnel of the operations department. This position will train and learn, develop, implement, and coordinate strategies related directly to the manufacture of Taiyo products. Additionally, this position will be learning all about the facility, environment, and health and safety functions. Eventually, this position will be responsible for the administration, security and maintenance of the facility and warehouse

Required Experience/Education:

- 4-year college degree in industrial engineering or another similar science discipline combined with work experience in ink or coatings manufacturing
- Ability to read, analyze, and interpret common scientific and technical journals, financial reports, and legal documents
- Ability to respond to inquiries or complaints from customers, regulatory agencies, or members of the business community
- Ability to develop and implement goals, objectives, and strategies
- Ability to effectively present information to top management, public groups, and/or boards of directors
- Ability to apply principles of logical or scientific thinking to a wide range of intellectual and practical problems
- Knowledge of governmental safety, environmental, transportation regulations/laws

Preferred Skills/Experience:

- Bilingual (Japanese/English)
- Toyota Production System (TPS)

Working Conditions:

- Occasional weekend or overtime work

See complete job listing for more information.

apply now

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

Career Opportunities

Technical Account Manager Chicago/Minneapolis

Insulectro, the largest national distributor of printed circuit board materials, is seeking a talented sales superstar for a Technical Account Manager role based out of either our Chicago or Minneapolis office. This role will focus on maintaining the existing customer base and developing new business within the assigned territory in both the printed circuit board and printed electronics industries. We are looking for the perfect fit of education, experience, and attitude that matches our company culture and enhances the service level to our customers.

Qualifications:

- A self-motivated business professional who is driven to succeed with a minimum of 3 years outside sales experience in the PCB or PE industry
- Proven sales/business development record
- Excellent communication and interpersonal skills
- OEM and electronic assembly experience is a plus

We offer:

- Competitive salary and commission plan with a comprehensive benefits package
- A fun, high-energy company with an entrepreneurial spirit
- A great group of people to work with!

apply now

Analyst Programmer, Hong Kong

We believe in caring about our people because they are our greatest asset. CML works with multicultural stakeholders daily to achieve more and bring them the best solutions. That's why we continuously invest in optimizing our culture and focus on providing our team with opportunities to develop their skills (e.g., through professional coaching to achieve their highest potential).

The analyst programmer will assist the IT and ERP manager in Hong Kong to support the company's BI systems, ERP systems, and other related IT-landscape applications.

In addition, this post will participate in system development projects and provide support including, but not limited to, user requirement collection and analysis, user training, system documentation, system support and maintenance, enhancement, and programming.

- Develop and enhance related IT systems and applications
- Prepare functional specifications
- Transfer the relevant business and interface processes into IT systems and other applications to get a maximum automation degree and prepare all required business reports
- Conduct function testing and prepare documentation
- Manage help desk/hotline service

CML is a leading provider of printed circuit boards. We develop tailor-made sourcing and manufacturing solutions for our customers worldwide with strong partnerships and reliable connections.

apply now

Career Opportunities

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

Development Chemist
Carson City, NV

Develop new products and modify existing products as identified by the sales staff and company management. Conduct laboratory evaluations and tests of the industry's products and processes. Prepare detailed written reports regarding chemical characteristics. The development chemist will also have supervisory responsibility for R&D technicians.

Essential Duties:

- Prepare design of experiments (DOE) to aid in the development of new products related to the solar energy industry, printed electronics, inkjet technologies, specialty coatings and additives, and nanotechnologies and applications
- Compile feasibility studies for bringing new products and emerging technologies through manufacturing to the marketplace
- Provide product and manufacturing support
- Provide product quality control and support
- Must comply with all OSHA and company workplace safety requirements at all times
- Participate in multifunctional teams

Required Education/Experience:

- Minimum 4-year college degree in engineering or chemistry
- Preferred: 5-10 years of work experience in designing 3D and inkjet materials, radiation cured chemical technologies, and polymer science
- Knowledge of advanced materials and emerging technologies, including nanotechnologies

Working Conditions:

- Chemical laboratory environment
- Occasional weekend or overtime work
- Travel may be required

apply now

Career Opportunities

Career Opportunities

Career Opportunities

IPC EMERGING ENGINEER

IPC EMERGING ENGINEERS

- **Grow your technical knowledge and organizational skills with participation in IPC committees for development of industry standards.**
- **Build connections for technology interchange.**
- **Become a valued contributor to the industry — like IPC Mentors.**

BECOME AN IPC EMERGING ENGINEER

If you have worked in the industry less than 5 years or are a student in an engineering or engineering-related program at a College or University, and are ready to invest in your career.

You can commit to:

- Learning about new technologies and the industry standards development process from subject matter experts
- Participating in IPC committee work on industry standards development.
- Attending IPC APEX EXPO for the next 3 years.

Individuals selected as IPC Emerging Engineers will be awarded an All-Access Package to IPC APEX EXPO (approximately $1,500 value) to learn about all that IPC has to offer the industry.

Send your resumé to: CareerDevelopment@ipc.org.

IPC MENTORS

- **Develop your leadership skills by providing guidance to Emerging Engineers new to the industry, demonstrate your technical and organizational expertise.**
- **Expand your professional network.**
- **Ensure that your IPC committee work continues.**

JOIN THE IPC MENTORS TEAM

If you have at least 7 years of industry experience, and have worked on an IPC committee for at least 5 years.

You can commit to:

- Spending time with an Emerging Engineer sharing experiences and providing guidance on IPC-related activities
- Attending IPC APEX EXPO for the next 3 years.
- Helping the next generation become tomorrow's standards development leaders.

Mentoring responsibilities include: participation in the candidate selection process, briefing your assigned IPC Emerging Engineer before and after meetings, making introductions to peers, participating in networking opportunities, and providing feedback to IPC twice each year.

Send your professional biography and ideas for this program to: CareerDevelopment@ipc.org.

ACCEPTING APPLICATIONS

Events Calendar

productronica 2019 ▸
November 12–15, 2019
Munich, Germany

PCB Carolina ▸
November 13, 2019
Raleigh, North Carolina, USA

Space Coast Expo & Tech Forum ▸
November 20, 2019
Melbourne, Florida, USA

2019 International Electronics Circuit Exhibition (Shenzhen) ▸
December 4–6, 2019
Shenzhen, China

DesignCon 2020 ▸
January 28-30, 2020
Santa Clara, California, USA

IPC APEX EXPO 2020 ▸
February 1–6, 2020
San Diego, California, USA

Medical Design & Manufacturing ▸
February 11–13, 2020
Anaheim, California, USA

Embedded World ▸
February 25–27, 2020
Nuremberg, Germany

Additional Event Calendars

Coming Soon to *Design007 Magazine*

December 2019: What You Need to Know
In December, we ask a group of industry experts: What do designers and design engineers need to know (technologically or not) going into the new year?

January 2020: Networking & Education
Our Designers Council issue will also feature previews of IPC APEX/EXPO and DesignCon.

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

CONTRIBUTING TECHNICAL EDITOR: **DAN FEINBERG**
baer@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@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 © ES SARAWUTH**

DESIGN007 MAGAZINE

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

November 2019, Volume 8, Number 11
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX