JANUARY 2020

I-Connect007

DESIGN007 MAGAZINE

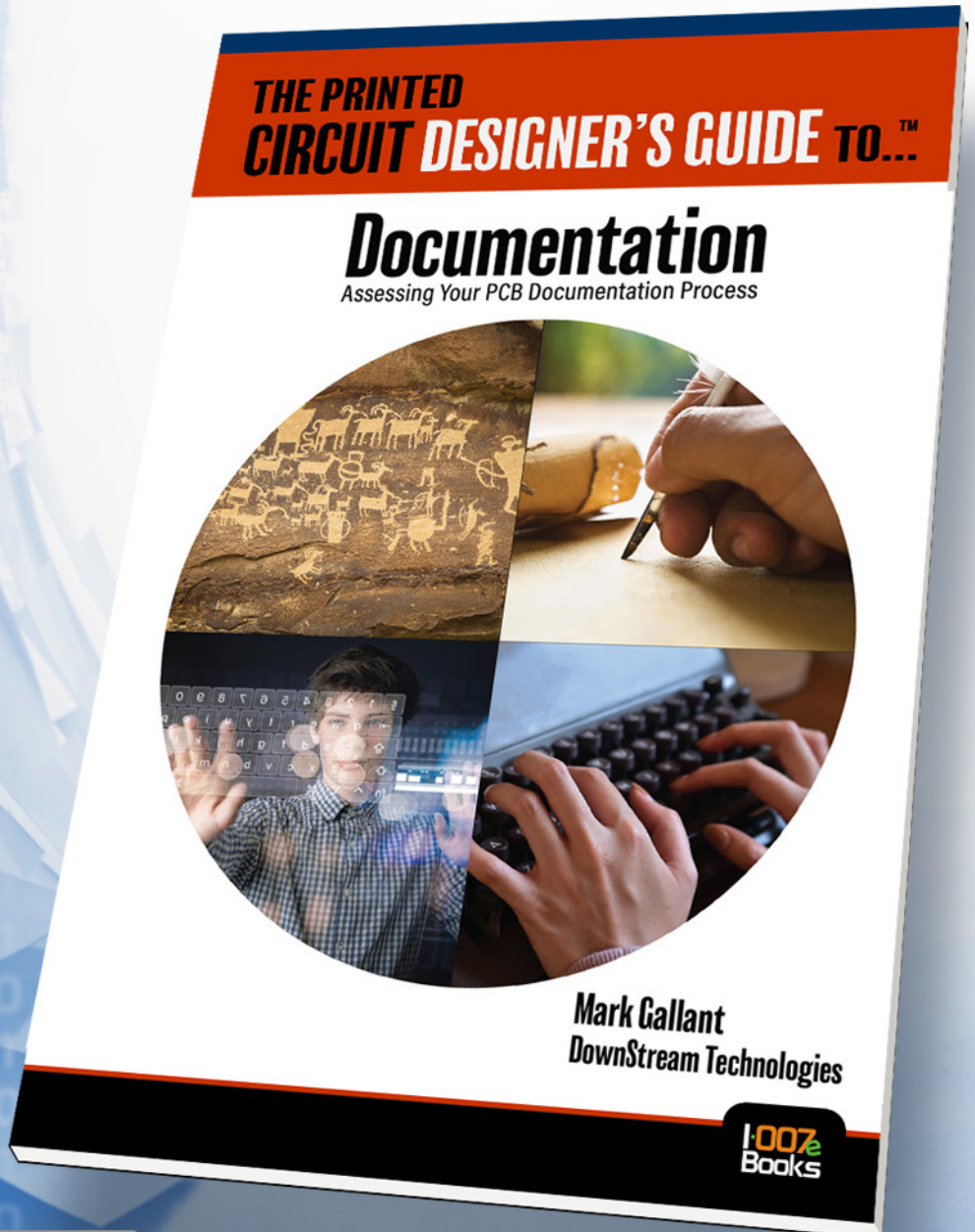
New Year's Resolution: GET INVOLVED!

FLEX007

SEE INSIDE!

Your life just got better.

This book will teach you how to save time and money, reduce frustration, and make your job easier!



I-007e Books

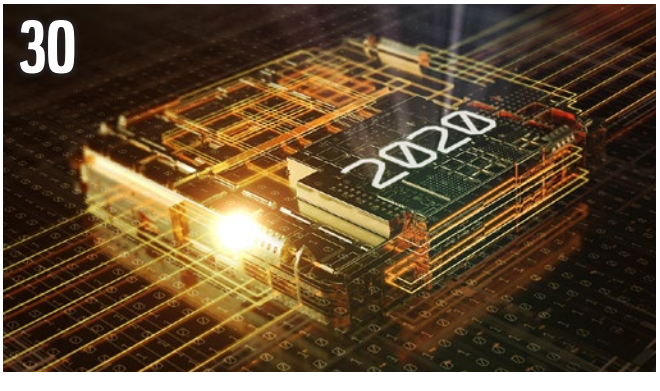
I-007ebooks.com/doc

DOWNLOAD NOW

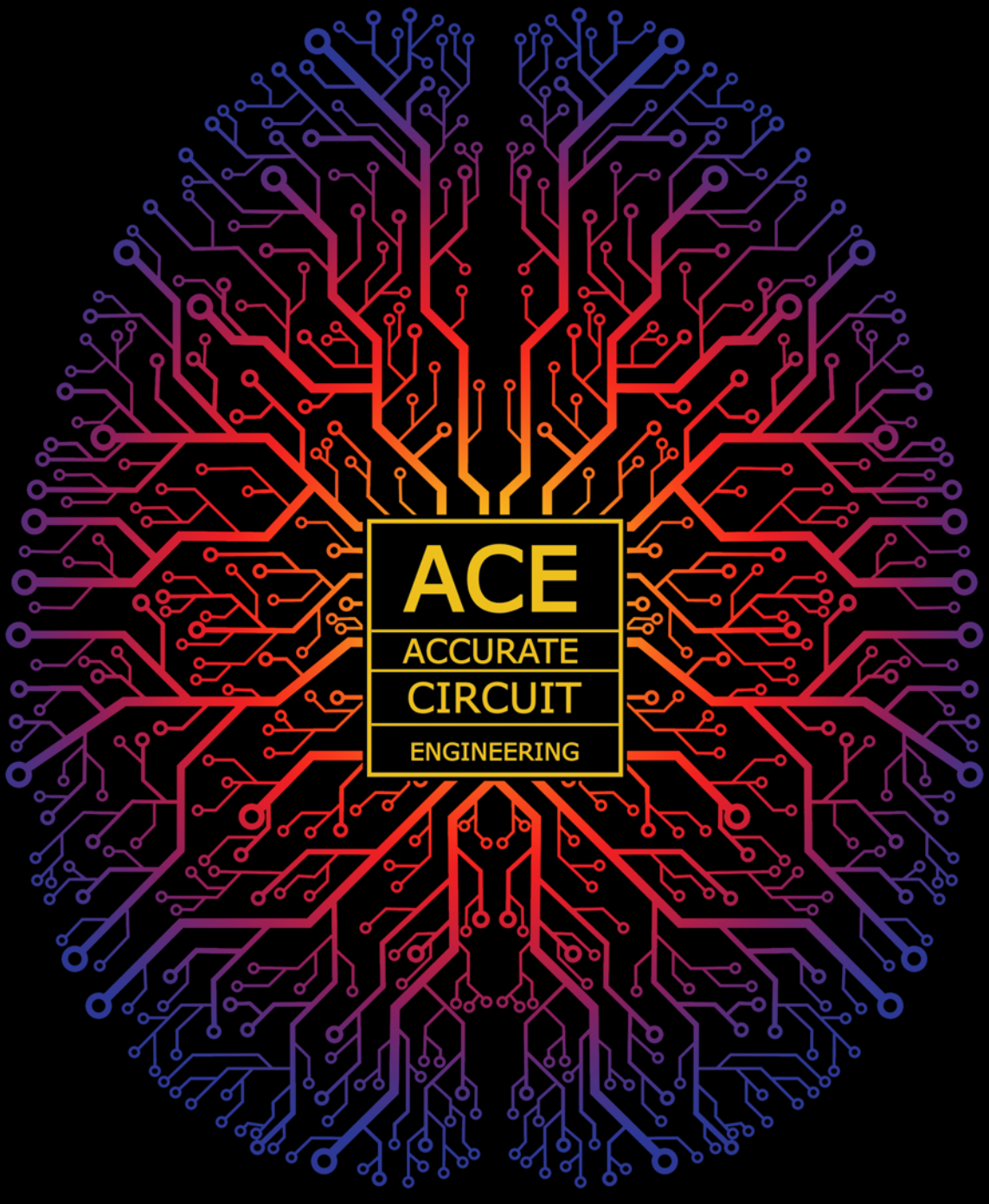
DESIGN007 MAGAZINE

Get Involved!

It's 2020, and it's time to hit the ground running. If you're not already networking with other designers or volunteering in our industry organizations, there's no better time to start. In this issue, we focus on getting involved with the PCB design community and working with your peers through training, standards development, or event planning.



Don't Overthink Your Prototype Builds



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

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS

ACE
ACCURATE
CIRCUIT
ENGINEERING

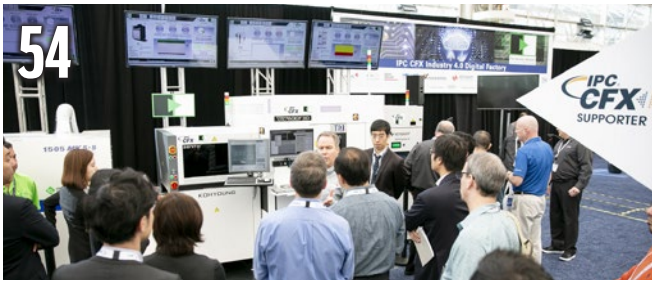
Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

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

www.ace-pcb.com

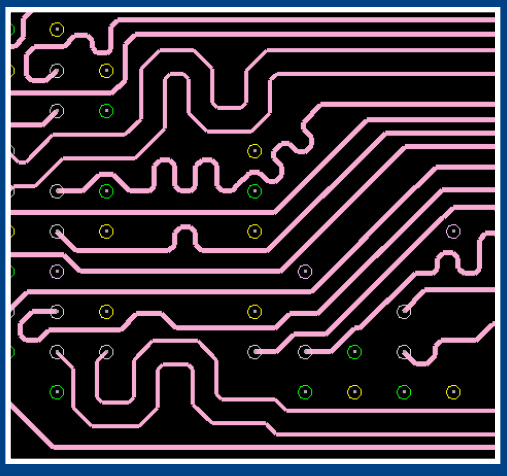
DESIGN007 MAGAZINE

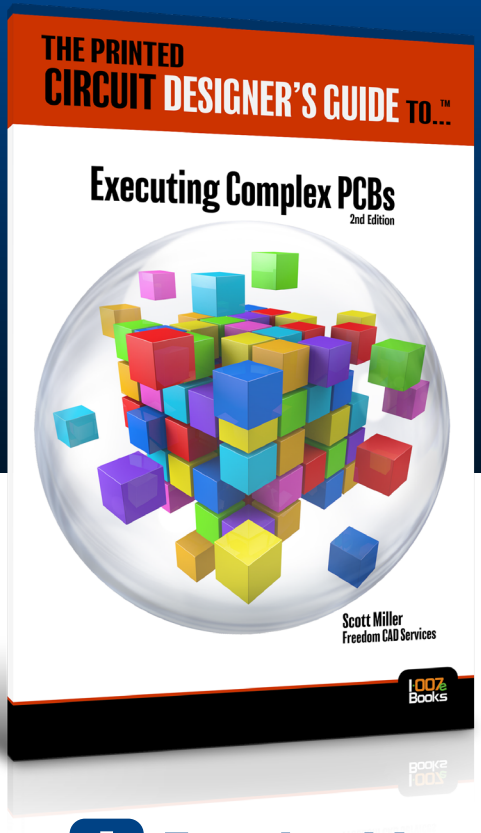


FREEDOM CAD EXPERT PCB DESIGN TIP #2:

DON'T MAKE YOUR HIGH-SPEED TRACES TOO LONG!

Today's PCB designs require more diligence in order to meet the requirements of today's high-speed microprocessors. At higher data rates, insertion loss becomes more critical. This results in signal length limitations due to insertion loss, very tight impedance matching, short stub lengths and the use of higher performance PCB laminates.





Free book!

"Freedom CAD drove the tasks to make the board design come together efficiently and effectively. They became a seamless extension of our team."

Laurie Fraser
Engineering Services Manager
Draeger Medical Systems, Inc.

FREEDOM CAD SERVICES

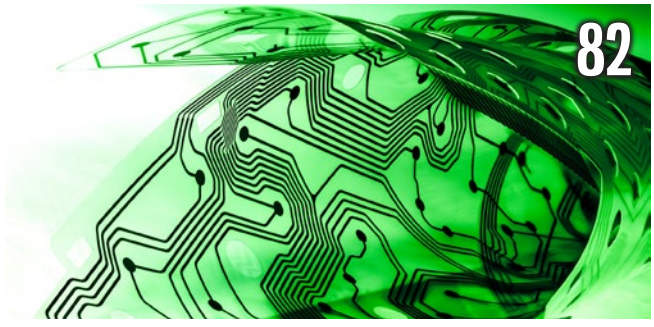
(714) 490-7600 West Coast
(603) 864-1303 East Coast

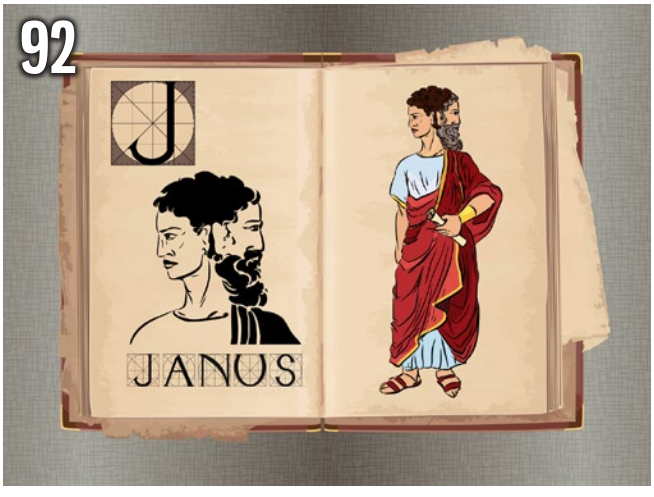
freedomcad.com

FLEX007

Focusing on Flex

It's a great time to be working with flex and rigid-flex, but it can be difficult just keeping up to date with all of the changes taking place in this growing segment. In this issue, we have you covered.





U.S. CIRCUIT
33rd
ANNIVERSARY
1985 ★ 2018

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

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

Take a look at our quality »

COMMERCIAL • MILITARY • MEDICAL • BAREBOARD

U.S. CIRCUIT

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

ISO 9001:2008 REGISTERED

DEPARTMENT OF DEFENSE · UNITED STATES OF AMERICA

ITAR International Traffic in Arms Regulations Registered

New Year's Resolution: Get Involved

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

As the new year dawns, let's start by wishing DesignCon a happy birthday. This year's event marks the 25th anniversary of a show that started out, as many good ideas did, in the offices of HP. Dave Belandi of HP gets much of the credit for getting the ball rolling for what eventually became DesignCon.

The show was driven primarily by electronics engineers who wanted to share their knowledge about a fairly new idea: signal integrity. Now, DesignCon has grown into much more than an SI show, focusing on topics such as power integrity, skew, crosstalk, and jitter.

The show was later acquired by IEC, who then sold it to United Business Media, the owners of EE Times. Thanks to its location in Santa Clara, California, the show always draws a good crowd, even during downturns. If you were an unemployed EE in 2008, you knew that DesignCon was the place to be.

Let's congratulate all of the people who have been involved with DesignCon since the very beginning, including engineers like Dave Belandi and our friends Istvan Novak and Eric Bogatin. These people volunteered their nights and weekends to get this show on the road, and the industry—especially in the signal integrity arena—is much better for it.

What's Up With the IPC Designers Council?

Speaking of getting involved, some of you may have heard that the IPC Designers Council has undergone a name change and much more. The new organization is called IPC Design, and it will be run much like the other groups within IPC. As you'll learn in this issue, IPC has plans to make the new group bigger, better, and more modern than before.

Whether you agree with this move or not, I doubt anyone would argue that the relationship between the Designers Council and IPC has ever been a smooth one. IPC didn't know what to do with the Designers Council, and each chapter acted more or less on its own.

The Designers Council began 28 years ago when Gary Ferrari founded the organization in Atlanta, Georgia. Since then, thousands of PCB designers have kept on top of their game by attending "lunch and learn" meetings. Some chapters (e.g., Scott McCurdy's Orange County Chapter, Bob McCreight's Silicon Valley Chapter, Luke Hausherr's San Diego Chapter, Tim Mullin's Cascade Chapter in Seattle, and Tony Cosentino's RTP Chapter in North Carolina) have drawn steady attendance to their meetings. But many of the other chapters in North American remain dormant, or nearly so.

As you'll learn this month, IPC has a plan for IPC Design to continue with what the DC started. (No, they're not going to start charging for membership, which was my first question.) I know; you're skeptical. I wasn't born yesterday either. But what if IPC really wants to make it easier for designers to get together, network, and share information? Let's see what IPC has planned; they'll be discussing this new organization at IPC APEX EXPO 2020 as well.

Meanwhile, a group of former leaders of the DC has launched their own organization, the Printed Circuit Engineering Association (PCEA). As Stephen Chavez explains in his column, the PCEA has pledged to continue working with IPC while helping to promote PCB design and design engineering through its own efforts.

Looking Forward to 2020

This is going to be an interesting year! In this issue, we focus on getting involved with the industry and working with your peers through training, standards development, or event planning. We start with an interview with Randy Faucette and Tony Cosentino, who discuss PCB Carolina 2019—a show that grew out of the RTP Chapter.

Then, we have a conversation with Dave Seymour of Ixia, who just passed his certification exam to be a CID+ trainer. Next, John Watson, CID, of Legrand explains why it's so important for designers to learn to adapt to adversity and stay active in their education. IPC's Teresa Rowe and Patrick Crawford discuss IPC Design, the new organization replacing the Designers Council, and how they hope to make this group better and more responsive to the needs of designers.

Next, we highlight an interview with Rainer Beerhalter of Squadrat, who discusses his job designing PCBs for large LED screens. We also have columns from our regular contributors, including Barry Olney, Bob Tise and Matt Stevenson, Vern Solberg, Stephen Chavez, Tim Haag, and Alistair Little.

We're getting ready to bring you coverage of DesignCon and IPC APEX EXPO. I hope to see you all on the road! **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.

How Real-time Tracking with IoT Keeps Your Food Safer and Fresher

Laura Rumbel works with Intel's customers to make IoT-based supply chain solutions that provide real-time data. That data allows enterprises to make decisions as food shipments are in transit and better predict the future needs of their supply chain.

Her simple explanation of IoT's promise in the food supply chain: Every year, $400 billion of food in production never reaches consumers, according to Bloomberg. "There is a major disparity between the amount of food that's never consumed versus the number of hungry people worldwide," Rumbel says. "This is where IoT comes into play."

Intel recently partnered with a produce distributor to help maximize efficiency and create less waste by tracking blueberries from harvest to the distribution center. Intel's IoT-based sensors tracked the berries' temperature, humidity, shock (which monitors damage, like being dropped), and changes in light (which indicate a pallet is being tampered with).

"Instead of berries going to waste if they are overripe, they can be sent to a consumer that can use the berries to make juice," Rumbel says. Real-time data allows these decisions to be made quickly, before the berries spoil, ultimately saving time and money and protecting consumers' health.

"Retailers are trying to grapple with the power of the consumer because ultimately consumers care more than ever before about the ethics and sustainability of products," Rumbel says. "Consumers want to make their own decisions; they want access to the data." But, she points out, it's important for the data to be broken down and contextualized for the consumer to make meaning of the raw data. (Source: Intel)


Tony Cosentino


Randy Faucette

PCB Carolina Show Continues to Grow

Feature Interview by Andy Shaughnessy
I-CONNECT007

At PCB Carolina 2019, I met with Tony Cosentino and Randy Faucette, both with Better Boards (a PCB design and engineering services company in Cary, North Carolina). They are also two leaders of the Research Triangle Park (RTP) Chapter of the IPC Designers Council and founders of PCB Carolina. I asked them to give us an update on this rapidly growing show and some of the changes going on at the RTP Designers Council chapter.

Andy Shaughnessy: Tony, I heard that you are now the outgoing president of the RTP Designers Council Chapter.

Tony Cosentino: I have been the president of the RTP IPC Designers Council Chapter for 11 years. I spoke with Randy at work and told him that 11 years is going to be the end of it. Maybe in a few years, if we feel like the chapter needs me to jump back into the leadership role, I can. But for now, I think that we need to switch captains of the boat and have somebody else steer for a while. I will take an ancillary position in the chapter and continue to support the organization, as I feel this is an important organization to the designers in the local area.

During the welcome at the keynote address, I announced that I was going to step down as president and that I am endorsing Randy. Then, I introduced Randy, and he came up to the podium and gave the PCB Carolina opening announcements. We will hold chapter elections for all the officer positions at our January meeting, and this will make it official.

Shaughnessy: Randy, do you think you're going to take the job? It sounds like it's there if you want it.

Randy Faucette: I'm not opposed to the role. I've been an officer with the Designers Council for 15 years. I've had the treasurer position for 14 years, and I became the vice president last year. For the past decade, it seems like there have only been three or four of us, and we've sort of rotated the officer positions. The chapter officers all rise to the occasion and do what needs to be done, so it's a team effort.

Siemens Technologies Deliver Smart Manufacturing for Electronics!

With Mentor and Siemens digital enterprise solutions, you can digitalize your entire new product development and electronics manufacturing processes

APEX EXPO IPC 2020

Visit Siemens Digital Industries Software at IPC APEX, San Diego, February 4-6, 2020

Hall D, Booth #3223

Don't miss these sessions at the show:

Martin Franke, Siemens AG, Corporate Technology - Industry 4.0 - How we transform a Buzzword into Manufacturing Excellence in Electronics:
Case Study for Improving the PCB Print Process Using Factory Data.
Wednesday, February 5th, 10:30 am to 12:00 pm.

Jay Gorajia, Director, Global Services, Siemens Digital Industries Software -
Optimizing Throughput and Cost with Manufacturing Simulation
Thursday, February 6th, 1:30 pm to 3:00 pm

SIEMENS
Ingenuity for life

Mentor®
A Siemens Business

Siemens.com/electronics

Hopefully, we can recruit some new people, which would be good; I also hope they won't all have the same color hair that we do. We need to get some younger people involved.

Cosentino: And if he takes over as president, that leaves the vice president position vacant.

Hopefully, we can recruit some new people, which would be good; I also hope they won't all have the same color hair that we do.

Faucette: There is also a vacancy in the secretary position. The person who was voted in last year had to relocate, and since they're not local, it didn't make sense for them to be part of the RTP chapter, so we have two officer positions open.

Cosentino: I am positive we will fill both positions at the January meeting elections.

Shaughnessy: How many active members do you have in the chapter?

Faucette: We have approximately 30 attendees for a typical RTP Designers Council meeting, and 50–60 members total throughout the year in chapter attendance.

Shaughnessy: At PCB Carolina, the show floor was bigger and sold out again.

Faucette: Yes. We had 78 exhibitors on the show floor, and 24 vendors on the waitlist, hoping a vendor would have to drop out. The show's popularity is evident.

Cosentino: We also had 16 technical sessions, which were all free, and an IPC class. We had two soldering workshops with 24 spots, which sold out too. Not only do we market to the local working community, but we also reach out to local universities and community colleges as well. This year, we added an event in conjunction with IPC and their STEM initiative. The STEM event brought in high school students in a continuing effort to get young people who are interested in engineering and math involved now because they're forming opinions about where they want to go in the future. In fact, some of these students are working on STEM projects and programming Arduinos even in middle school. It's amazing what is available to young people today.

Faucette: IPC coordinated the efforts early with the Wake County STEM High Schools to do this, and we supported them with the logistics of having it at this event. During PCB Carolina, half of the STEM students had a project where they took a board, mounted components on it, and did some programming while the other half walked the exhibit floor. Then, they had lunch and swapped activities, which worked out well.

Shaughnessy: It's cool that they have to create something. The show also seemed busier this year too.

Cosentino: There were over 1,000 people at PCB Carolina 2019, which was a 14% increase over last year.

Shaughnessy: I also noticed that most people stayed to the end. Having the evening reception and prize drawing at the end was a good idea.

Faucette: Many vendors say that they never stopped talking all day. If there weren't classes going on, there were three or four people deep waiting to talk to you, but if there were classes going on, they were still engaging people constantly. You want the traffic to be constant and busy, and that's what it was.

Shaughnessy: I talked to three or four vendors who said they couldn't believe they'd never done this show before.

(Attendee slaps Tony on the back: I love you guys!)

Shaughnessy: People come up and tell you they love you. I don't see that at other shows.

Cosentino: We seem to have a pretty good recipe for this, and it's running pretty well. Even now that the show is over, the attendees are gone, but all the volunteers are still here, helping.

Cosentino: And the McKimmon Center is ideal for our event and continuing education. The support McKimmon Center provides is beyond anything we have found anywhere else. It is absolutely the perfect location for our event.

Faucette: It's a good formula and a great venue for the event.

Cosentino: The Keynote address was on IoT and was held in the cafeteria with breakfast to accommodate the large crowd. IoT is the wave of the future; it is the fourth industrial revolution, so that was a great kickoff for PCB Carolina. The attendees can stay all day because we provide lunch and an evening reception with beer, wine, hors d'oeuvres, and two grand prizes. We feed and educate attendees and give them access to vendors they want to see. It's a reunion each year, and everyone wants to be here.

Shaughnessy: It's funny because I see people here that I don't see at other trade shows. Do you think you get a lot of people who don't travel to other shows?

Faucette: We are the only electronics trade show in a 300-mile radius. Many folks drive 2–3 hours and have access to all that PCB Carolina offers. I'm sure we bring in many folks that would otherwise need a large travel budget and a several-day commitment to attend one of the other shows.

We had an interesting entertainment exhibit this year from the Virginia International Raceway. The VIR offers a formula racing experience. You can race on their track by renting their formula cars or ride in your car with a professional and go around a live track. They brought a simulator here today and raffled off a couple of experiences at their track in Virginia, which was very well-received. There was always a line of people there.

Shaughnessy: It looked realistic. Is there anything else you'd like to discuss?

Cosentino: We'll start planning for PCB Carolina 2020, and exhibitor registration will open in February. The chapter work never stops; you think you're finished, but it just resets.

Shaughnessy: And it can be hard to keep a chapter active.

Faucette: Six months can go by in a hurry if you haven't done anything, and you can lose people and momentum. Life and work get busy. It takes effort to keep it going.

What keeps the momentum going is the involvement of the people in the chapter.

Cosentino: The only way to keep a chapter active is to engage them constantly, and if you stop having meetings that people want to attend, it will die. What keeps the momentum going is the involvement of the people in the chapter.

Shaughnessy: I think the designers feel a connection with you both. Congratulations, and best of luck as the new president, Randy.

Faucette: Thanks for the support.

Cosentino: I know it's no small effort to attend. We appreciate it. **DESIGN007**

Meet Dave Seymour, Brand-new CID+ Instructor

Dave Seymour

Feature Interview by Andy Shaughnessy
I-CONNECT007

At PCB Carolina, I had the chance to speak with newly minted CID+ instructor and longtime musician Dave Seymour. He had just passed his evaluation class as a Certified IPC Trainer (CIT) for CID+ instruction, and he was eager to talk about his experience as a PCB designer and EPTAC instructor. A few weeks after this interview, Dave received his formal approval to become a CID+ instructor. Congratulations, Dave!

Andy Shaughnessy: Dave, can you tell us a little about yourself?

Dave Seymour: I work for Ixia, a Keysight Business, as a PCB designer. Ixia was acquired by Keysight Technologies about two years ago. We have many top-notch engineers. Keysight has about 12,000 employees now. We have offices all over the world, including Morrisville, North Carolina, and Calabasas, California,

Shaughnessy: What's your background?

Seymour: I graduated from a two-year tech school with an engineering degree. I immediately went into a startup where I was employee #7. This was in the early '80s, and in '82, we were doing 300 baud modems using red and blue hand tape.

This was right after the breakup of the Bell operating companies when they finally unbolted the modems from the phones. You couldn't plug any equipment into the wall jack at that point. The Federal Government broke up the Bell companies, which meant you could plug a third-party modem into the wall jack. At our startup company, we went from 300 bauds to 1200–2400-baud modems. At that point, Rockwell International came up with a chipset modem that was basically two chips, an RJ-11 jack, and a transformer, and that put us out of business.

At that point, I started working for other companies. I worked for Tekelec Inc. for 14 years, doing protocol test equipment and interfaces. This was back in the days of ISDN, SS7, and other protocols. I was doing portable boxes as well as rack-mounted equipment.

Around 2001, Tekelec Inc. had a divisional change, and they sold the test division to Catapult Inc. I worked for Catapult for eight years. We did more rack-mounted test equipment and compact PCI proprietary backplanes and miscellaneous tabletop boxes. Catapult was

bought by Ixia, and if I draw a line, I've been working many of the same engineers for about 30 years.

Ixia continued the large format rack-mounted equipment as well as tabletop boxes. Now, with Keysight, we have moved into testing different interfaces, and we're up to 400G. These are the interfaces and technologies that enable 5G for your cellphones, WiFi communications between cars, etc. That's the path I'm on today.

Shaughnessy: That's some pretty high-speed stuff.

Seymour: It's very high-speed technology based on PAM4. If you've seen the three-eye waveform diagram associated with PAM4, in the near future, it will be PAM8 and PAM16, which will have more eyes, than you would have thought possible. Keysight has products based on PAM4 shipping now.

Shaughnessy: Tell us about your new certification. Yesterday, you were certified to teach the CID+ class at PCB Carolina.

Seymour: Yes. PCB Carolina is put on by the local RTP chapter of the IPC Designers Council, and I've been a member for about 18 years now. Better Boards founder Randy Fawcett is the main driver for the show, and I've been on and off the support team for the show as time permits. The local chapter plans the show, including finding vendors, technical sessions, etc. One of the things IPC has done is hold classes in association with PCB Carolina, such as soldering, IPC-600, CID and CID+ classes, etc.

The IPC contracts some training out to EPTAC, and I'm employed by EPTAC as an instructor. I was certified as a CID instructor three or four years ago. Pending my evaluation, I'll be a CID+ instructor. What that means is I taught a CID+ class this week under Cherie Litson's guidance. Cherie is a Master IPC Trainer (MIT). She evaluated my instruction in the class this past week, and if she grades me as such, then I will become a CID+ instructor.

Shaughnessy: Tell me about the class. What were some of the things that you focused on?

Seymour: The CID+ is the follow-on advanced class for the CID. It's designer-oriented, so we talk in more detail about quality control, laminates, balanced stackups, and copper balancing. Then, we get into high-speed design and topics like reflections, signal quality, and laminate characteristics. The class continues with data format interfaces to the assembly and fabrication shops, which is more in-depth than we talk about during the entry-level class.

Shaughnessy: I've heard that you really need to know your jargon and definitions to be a CID+.

Seymour: Absolutely. The IPC has specific terms for given topics. Whether it's a fab panel or a fab array, you need to know how IPC defines those terms and understand them, which is often the difference in how things are manufactured.

Shaughnessy: Because all of this is translated and "terminal" means one thing in English, but it might translate into something else in Spanish or Mandarin.

Seymour: Exactly. Even for English-speaking students, it's not easy. If I said to you, "I want an array," is that a fabrication array or an assembly array? Then, you think, "It's really a fab panel versus an assembly panel."

Shaughnessy: Right. And if it's a quarter ounce of copper, is it before or after processing?

Seymour: Correct. Because during the processing, you'll lose a lot of thickness.

Shaughnessy: What's next for you?

Seymour: Since I'm continuing to work full time, I'll teach a class when my schedule allows. I'll be working for EPTAC. I enjoy teaching PCB design. My major concern for the future is the supply of young designers. I started designing boards 35 years ago, and received my CID certification in 2001 and my CID+ in 2003.

The older designers have to teach the young designers so that, hopefully, they will pick up the torch and carry it. I'm sure you hear that all the time from the engineering community. I hope more young folks come into the profession; it's a great career, and you can make a good living. And no matter what you're doing in high-tech, we need more tech people in the United States.

Shaughnessy: And if we lose design, then we lose everything.

Seymour: We'd like to keep the edge.

Shaughnessy: If your evaluation goes well, will you be able to move up to the big leagues?

Seymour: Right. That's the process for becoming an instructor. You train and take the test. Yeah, I had to have a CID+ before I could even think about teaching it. I had to catch up on the material that they've changed over the last few years by either adding to or subtracting from the class and then study the new material and present. As new designs come along, there are always techniques and methods to stay on top of.

Shaughnessy: Do you have a pretty good feeling about it?

Seymour: I do. The next part is a formal approval where they give you the certificate and say you are approved as an instructor. Then, I'll look forward to a number of years of training young designers. I'd like to thank a number of EPTAC instructors who have contributed to my growth, including Steph Chavez, Cherie Litson, and Gary Ferrari, who are all master instructors. I think Gary was born in an etch tank.

Shaughnessy: Probably!

Seymour: But I haven't approached anywhere near the MIT level yet. I'm sure I have to teach for a certain number of years before I can think about becoming an MIT; then, I'll see what the future holds for the master level.

Shaughnessy: Very good. Is there anything else you'd like to discuss that we haven't covered?

Seymour: I'd like to say one more thing about PCB design. I'm a musician—a bass player on the weekends. I think that a PCB design is a great marriage of technology and artistry in the same way that that left and right sides of your brain think about problems. It's technology and artistry wrapped into one.

I think that a PCB design is a great marriage of technology and artistry in the same way that that left and right sides of your brain think about problems.

Shaughnessy: Sure. A lot of designers are musicians.

Seymour: Exactly. It's a creative outlet. Young potential designers need to understand that in this career, you draw all day long. There is a reason they call it "artwork." My mom thought I was wasting a lot of time in the video arcades during college. We'd get $6 worth of tokens for $5 on college night and play Centipede, Defender, Tempus, Stargate, and Asteroids. I was sitting in front of a big tube playing games and coordinating hand movement with a trackball, and my mom would say, "You'll never use that in the future." Now, what do I do? I use a trackball all day long and draw pictures.

Shaughnessy: That's too funny. It has been great talking with you, Dave. We need to put a band together for the next PCB Carolina.

Seymour: Thank you, Andy. Any time you want to visit and make some noise, let's do it. **DESIGN007**

Interconnect Impedance

Beyond Design

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

Arguably, the most critical factor in high-speed PCB design is the impedance of the interconnect. We know that transmission line drivers must be matched to the impedance of the line for the perfect transfer of energy. Energy is never lost but rather transforms into other forms of energy. Specifically, in the case of an unmatched transmission line, energy can be transferred into heat, coupled into adjacent elements, reflected, or radiated. In this month's column, I will look at why interconnect impedance is so important to the correct performance of the system.

Impedance is an extension of the definition of resistance to alternating currents (AC). Impedance includes both resistance (the opposition of the electric current) and reactance (the measure of opposition as the current alternates). Reactance also includes the effects that vary with frequency due to distributed parasitic inductance and capacitance of the transmission line.

Impedance is at the core of the methodology that is used to solve signal integrity issues:

1. Signal quality issues arise because voltage signals reflect and are distorted whenever the impedance changes along a transmission line.
2. Crosstalk arises from the coupling of electric and magnetic fields between adjacent traces or coupling between traces and return paths. The inductance and capacitance between the traces establish an impedance, which determines the amount of coupling.
3. Differential mode propagation can be converted to common mode by parasitic capacitance or any imbalance caused by impedance variation, signal skew, rise/fall time mismatch, or asymmetry in the channel. Common mode currents are the main source of electromagnetic radiation.

Not only are the problems associated with the signal integrity best described by the use of impedance, but the solutions and design methodology for good signal integrity are also based on the use of impedance. The two key processes—modeling and simulation—are based on converting electrical properties into an impedance and then analyzing the impact of that impedance on the signals.

The iCD Stackup Planner in Figure 1 illustrates the three most common transmission line structures of a multilayer PCB. For embedded

UNITS: mil — 12/2/2019 — Total Board Thickness: 44.2 mil

Differential Pairs > 50/100 Digital | 40/80 DDR3 | 90 USB

Layer No.	Via Span & Hole Diameter	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Characteristic Impedance (Zo)	Edge Coupled Differential (Zdiff)	Broadside Coupled Differential (Zdbs)
		Soldermask		PSR-4000 HFX Satin / CA-40 HF LPI ...	3.5	0.5							
1	8 8 4 4	Signal	Top	Conductive			2.2	12	4	0.43	51.67	98.65	Embedded Microstrip
		Prepreg		370HR ; 1080 ; Rc= 66% (1GHz)	3.97	2.9							
2		Plane	GND	Conductive			1.4						
		Core		370HR; 1-7628; Rc=42% (1GHz)	4.4	7							Asymmetric
3		Signal	Inner 3	Conductive			1.4	10	4	0.31	53.26	99.85	Stripline
		Prepreg		370HR; 7628; Rc= 50% (1GHz)	4.19	8							
4		Plane	PWR	Conductive			1.4						
		Core		370HR ; 1-1652 ; Rc=43% (1GHz)	4.4	5							
5		Signal	Inner 5	Conductive			1.4	16	4	0.31	51.23	99.63	Dual Symmetric 48.89
		Prepreg		370HR ; 2116 ; Rc= 56% (1GHz)	4.14	4.8							
6		Signal	Inner 6	Conductive			1.4	16	4	0.31	51.23	99.63	Stripline 48.89
		Core		370HR ; 1-1652 ; Rc=43% (1GHz)	4.4	5							
7		Plane	GND	Conductive			1.4						

Figure 1: Embedded microstrip, asymmetric, and dual symmetric stripline configurations.

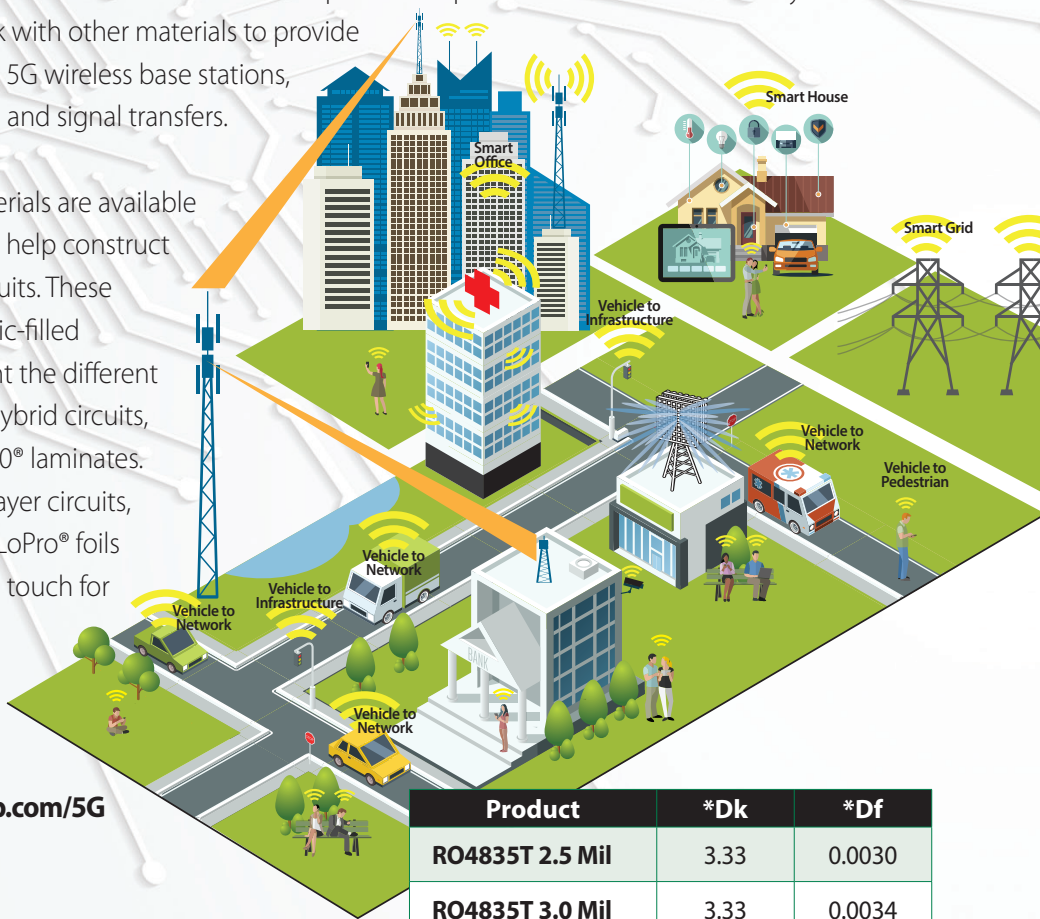
5G: Higher Frequencies!

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

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

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

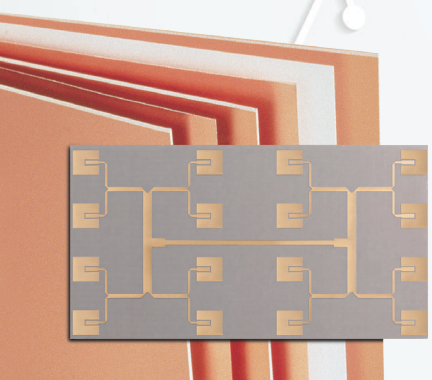
Rogers RO4450T bonding materials are available in 3, 4, and 5 mil thicknesses to help construct those 5G hybrid multilayer circuits. These spread-glass-reinforced, ceramic-filled bonding materials complement the different materials that will form these hybrid circuits, including RO4835T and RO4000® laminates.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



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

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

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



ROGERS CORPORATION
Advanced Connectivity Solutions

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

microstrip (solder mask coated microstrip), the electromagnetic field propagates partially in the dielectric material, solder mask, and air. Whereas in both stripline structures, the electromagnetic field propagates in the dielectric material sandwiched between the planes.

Interconnect impedance is a function of the geometry of the conductors and the dielectric constant of the material adjacent to or separating them. For PCB traces, the most critical dimension is the ratio of trace width to height above/below the reference plane(s). Impedance is also inversely proportional to the square root of the dielectric constant. Clearly, the accurate control over impedance requires precise management of both the physical geometries and the material characteristics along the entire length of the interconnect.

Figure 2 illustrates the variation of impedance with the three most influential variables: trace width, dielectric thickness, and dielectric constant. These impedance plots were simulated by multiple passes of the field solver in the iCD Stackup Planner. Note that the microstrip impedance (top row) varies almost twice as much as the stripline impedance (bottom row) to the same changes in the variables. Consequently, microstrip transmission lines are more vulnerable to change in impedance, which is another good reason not to route critical signals on the outer layers. Any slight variation in any of the total of five variables (including copper thickness and differential clearance) will dramatically change the localized impedance of a microstrip interconnect. These are physical properties of the multilayer PCB that the fabricator must control to maintain a constant impedance.

Having a PCB fabricated to controlled impedance specifications does not necessarily control the impedance of your routed traces; it only controls the impedance of the test coupons. Only you can control the impedance of the signal interconnect. As technology progresses, developers are specifying controlled impedance boards more frequently. The PCB fabricator does their best to control the impedance, of the bare board, given all the manufacturing variables. The fabricator will initially predict the stackup trace impedance using a field solver. They should then place impedance test coupons on the outer edge of the PCB to check that the manufactured product matches the predicted impedance using a time-domain reflectometer (TDR).

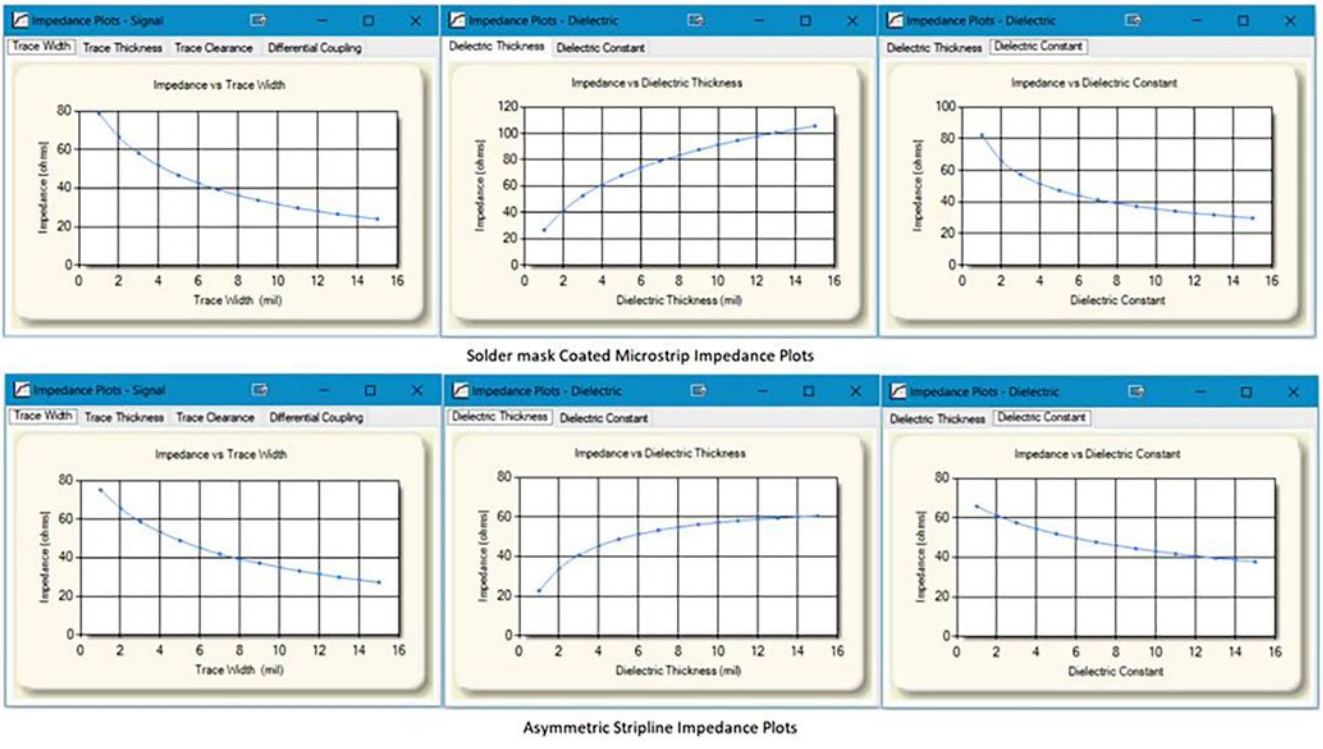


Figure 2: Comparison of microstrip and stripline impedance variations. (Source: iCD Stackup Planner)

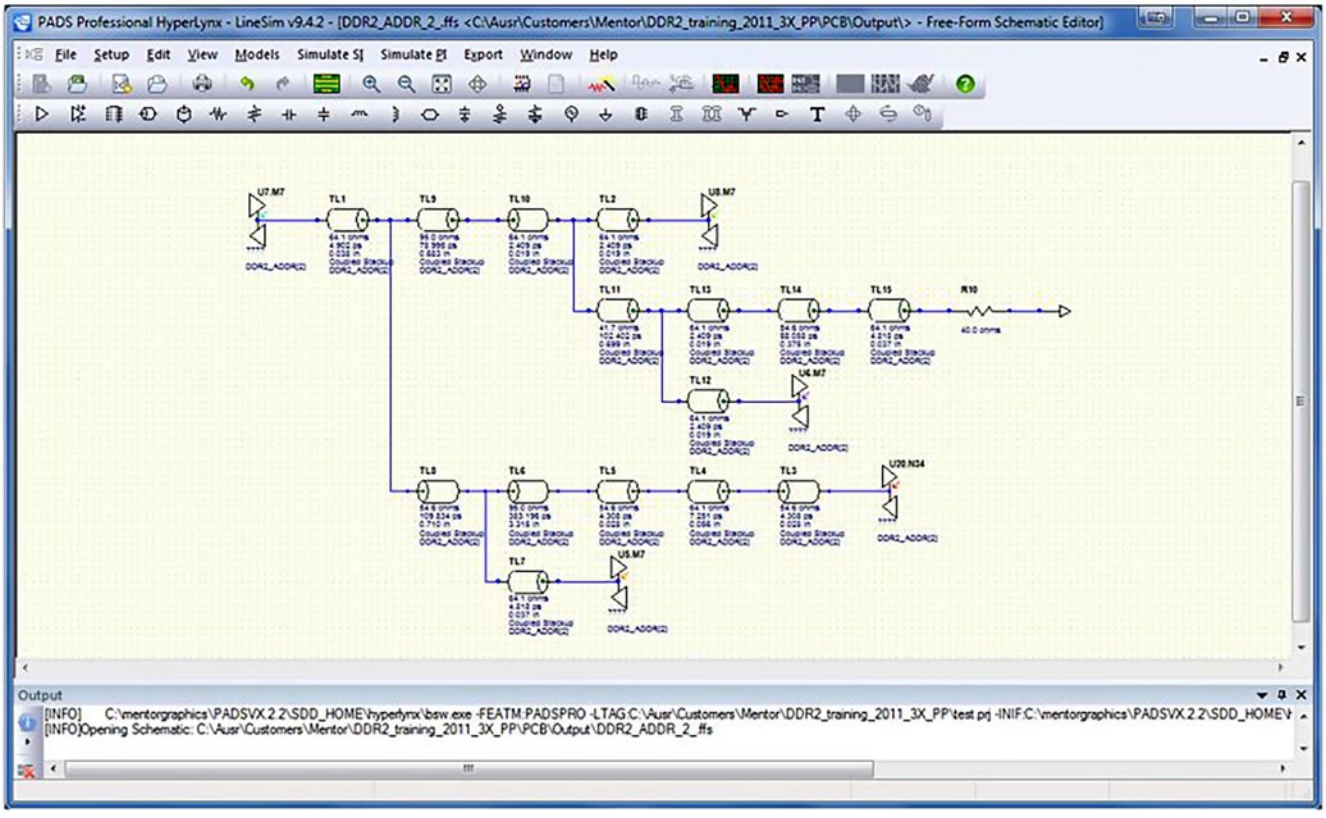


Figure 3: Free-form schematic model of a DDR2 address signal (simulated in HyperLynx).

Impedance variations along the transmission line are much more critical than a precise value of impedance. A flat impedance profile is vital, and it is the PCB designer's responsibility to ensure that there is no impedance discontinuity due to inadequate signal routing. Unfortunately, differential mode propagation can be converted to common mode by any imbalance caused by impedance variation.

Having a PCB fabricated to controlled impedance specifications does not necessarily control the impedance of your routed traces; it only controls the impedance of the inactive test coupons. The impedance test coupons do not take into account all of the possible issues that can occur throughout the maze of routing from driver to load. Only you can control the impedance of the signal interconnect. If you extract the interconnect topology (Figure 3), from a PCB layout to free form schematic models, the result can be terrifying—not quite that simple trace that was routed. In this case, any of the 15 individual transmission lines that form the entire interconnect can create issues if incorrectly routed.

The key to controlled impedance design is to maintain consistency along the entire length of the interconnect, providing a flat impedance profile.

1. Reflections occur whenever the impedance of the transmission line changes along its length. This can be caused by unmatched drivers/loads, layer transitions, dissimilar dielectric materials, stubs, vias, connectors, and IC packages. Terminate transmission lines, avoid layer transitions that don't have a common reference plane, and reduce the length of stubs.
2. These reflections augment crosstalk that is caused by close coupling of signal traces to other structures. Designers should couple traces close to the reference plane, avoid long parallel trace segments, and increase spacing to aggressor signals.
3. There are a number of recommendations to control skew caused by a glass-weave effect. But the simplest by far is to use two combined layers of 1067-style prepreg dielectric material between the signal and

reference plane. This ensures a constant percentage of resin to glass fiber in the dielectric material and controls the comparative propagation delay to < 2 ps/12 in.

4. Signal skew also occurs when differential pairs are not properly matched. Differential skew refers to the time difference between the two single-ended signals in a differential pair. Any mismatch in delay will result in changing part of the differential signal into common-mode current. If there is a mismatch (e.g., on a bend), it should be balanced by lengthening the appropriate trace where the bend occurs.
5. Do not route critical signals on the outer (microstrip) layers, as these are more vulnerable to change in impedance and also difficult for the fabricator to control the plating thickness.
6. Avoid placing copper pours next to signal traces, as the copper pour will lower the impedance on the adjacent trace segment. Use three times the dielectric height as an effective copper pour to trace clearance rule.

Establishing comprehensive design constraints can prevent many of the above issues from occurring in the first place and will certainly warn you when not enforced, depending on the level of your tool's electrical rule checking (ERC). (In addition, you could download the free HyperLynx DRC add-on, which can be used to identify PCB design issues affecting EMC and signal and power integrity.)

Key Points

- Energy is never lost but rather transforms into other forms of energy
- An unmatched transmission line's energy can be transferred into heat, coupled into adjacent elements, reflected or radiated
- Impedance is at the core of the methodology that is used to solve signal integrity issues
- Interconnect impedance is a function of the geometry of the conductors and the dielectric constant of the material adjacent to or separating them
- The most critical dimension is the ratio of trace width to height above/below the reference plane(s)
- Microstrip transmission lines are more vulnerable to change in impedance, which is another good reason not to route critical signals on the outer layers
- Impedance variations along the transmission line are much more critical than a precise value of impedance
- A flat impedance profile is vital, and it is the PCB designer's responsibility to ensure that there is no impedance discontinuity due to inadequate signal routing
- Having a PCB fabricated to controlled impedance specifications does not necessarily control the impedance of your routed traces; it only controls the impedance of the inactive test coupons
- The key to controlled impedance design is to maintain consistency along the entire length of the interconnect, providing a flat impedance profile **DESIGN007**

Further Reading

- B. Olney, "Beyond Design: Controlled Impedance Design," *The PCB Design Magazine,* May 2015.
- B. Olney, "Beyond Design: Transmission Lines–From Barbed Wire to High-speed Interconnects," *The PCB Design Magazine,* May 2014.
- B. Olney, "Skewed Again," *The PCB Magazine,* June 2013.
- B. Olney, "Differential Pair Routing," *The PCB Magazine,* October 2011.
- E. Bogatin, *Signal and Power Integrity: Simplified,* Prentice Hall, 2008.
- H. W. Johnson & M. Graham, *High-Speed Digital Design: A Handbook of Black Magic,* Prentice Hall, 1993.
- G Havermann & L. Ritchey, "[SI-LIST] Forum."

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

Create Comprehensive PCB Documentation to Drive Fabrication, Assembly and Inspection

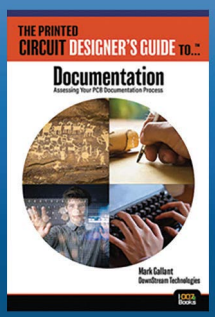
BluePrint-PCB®

BluePrint automates the PCB documentation process, creating comprehensive electronic drawings, successfully driving the procurement, fabrication and assembly process.

- 3D technology allows visualization of imported CAD data as a 3D PCB
- Lowers documentation cost and time
- Increases documentation detail & accuracy
- Shortens PCB design cycles
- Reduces documentation time by 60-80%
- Improves manufacturing & simplifies inspection

Download your free e-book "The PCB Designer Circuit Designer's Guide to Assessing Your PCB Documentation Process"

CLICK HERE TO DOWNLOAD



This book, written by Mark Gallant of DownStream Technologies, explains how an automated documentation solution like BluePrint-PCB® can eliminate post-processing errors and speed up time to market.

This book is a must-read for any PCB designers or design engineers who would like to adopt 21st-century PCB documentation processes.

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

DownStream Technologies

Teaching Designers to Adapt and Overcome

Feature Interview by the I-Connect007 Editorial Team

John Watson of Legrand speaks with the I-Connect007 editorial team about the design classes he taught at AltiumLive in Frankfurt, Germany, covering a variety of design challenges. John shares some of his insight and explains why it's paramount for designers of any level to adapt to adversity, stay active in the design community, and continue their PCB design education throughout their careers. As John says, "Put yourself out there."

Andy Shaughnessy: John, it's good to see you again. Tell us about the classes you're teaching.

John Watson: I did four sessions on AltiumLive's University Day. The main thing that we were talking about was their software package, Concord Pro, which is a library management system that helps to control your libraries and the parts that you're using, etc. There is so much information now on a part that it is unbelievable. Years ago, you would start a PCB design, and you'd have a schematic symbol and a footprint. Now, you have so much more information tied to that component that, when you lay it down on that schematic, it is an entire resource system developed to support your schematic and PCB. Even then, the other roles that are involved in the PCB design process, such as your MCAD person, are going to be interested in your 3D model. Then, it goes into procurement, and your sourcing information is in that component also.

All of this is brought in together and controlled/managed through a revision process, and it's all done through Concord Pro. We were talking about those features and looking at the component development, including what should be in a component and what should be in a project. Then, we also looked at some advanced features of Concord Pro by looking at some of the other things that Concord Pro covers; it's not just for components. Many people look at it as a component library, but it's a place also for your projects to live. In our library, we set up a resource library of articles, etc., for people to reference and read.

I had one other session on how to overcome the challenges of PCB design, specifically elec-

APCT

Passion | Commitment | Trust

Tired of Continually Juggling Your PCB Requirements?

APCT won't drop the ball… We are a complete PCB manufacturer.
From Prototype to Production – From Standard to Advanced Technology - From Commercial to Defense & Aerospace
Our People Continue To Be The Difference

Contact APCT Today 408.727.6442 | APCT.com

Printed Circuit Board Solutions

APCT.com

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

tronic shortage issues. I teamed up with Vince Mazur of Altium, and it was a pleasure working with him again. I dealt with a lot more of the challenges facing us. We have four major challenges that we are facing.

The first challenge is general advancements in technology. Technology is advancing so quickly that it is unbelievable that we used to joke about Moore's law and transistors doubling in an IC package every two years; now, that's down to almost 16 months. The transistor industry is doubling that. We're looking at building from the ground using single atoms and molecules as ICs.

The transistor industry is doubling that. We're looking at building from the ground using single atoms and molecules as ICs.

The second challenge that we're facing is related to changes in our ECAD software. The tools are constantly improving, which is great, and that's why I work with Altium; they're always giving us the best tools possible. The third challenge is the parts shortage issues. This is a conversation we had almost a year and a half ago now.

Shaughnessy: The 0402 shortage is over, but now there are probably more on the way.

Watson: Yes. There are waves of shortages that happen, and about the time it's tapering down, you think, "We got through this." Then, another one works its way into a product line. I don't think there's real stability yet in the component availability issue, so that's going to be something that is going to constantly be a challenge for PCB designers.

The fourth challenge that I gave to them was more of a challenge to designers. The industry does not need more good designers; we need great designers who are on the cutting edge and know how to face these challenges and keep moving forward. They need to continue to learn and adjust.

In "Heartbreak Ridge," Clint Eastwood's character earns the Medal of Honor, and the theme is "adapt and overcome." That should be the mantra of all PCB designers nowadays. We have to adapt our processes. The old ways of doing things are not going to work in the future, so we constantly need to keep learning and adjusting the way we do things.

Barry Matties: Some of the work that Happy has done—the "old way"—is becoming the new way because it was kept a secret, as in landless vias. In some cases, the new is the old coming around. We need to be mindful of that too.

Watson: True. I'm still hoping for spandex and polyester to make a comeback from the '70s.

Matties: You can start the trend (laughs).

Happy Holden: Disco will rise again.

Matties: Mike Creeden was showing that in his presentation yesterday about landless vias.

Holden: I was surprised to see that.

Matties: Landless vias were something that you were doing 30 years ago but as a secret.

Holden: Because we thought landless vias wouldn't work with our Japanese partners. They said, "Test it," and the test showed that we were right and, low and behold, it was good. Nobody in the industry would believe us, so we said, "Let's not write a paper. Let's keep it a secret because it's not worth our energy to convince the people who refuse to like it."

Matties: I appreciate what you're saying. But this is also a case in point where we need to be mindful of things that do work that we might improve on.

Holden: Yes. Unfortunately, I have around 15 other secrets that I won't divulge now. The problem is that I don't have the energy to debate with you. All I can say, "Test it. Build it, and then find out."

Watson: But we are building on the foundations that you put there for us. I have a copy of *Printed Circuits Handbook* sitting on my desk, and it's a guideline for everything I do. My real hope as we build on the foundation of what you've done, Happy, is that we develop new designers; that's part of my career as I fade into the sunset, and that's the main purpose of AltiumLive. The most exciting thing has been seeing young people's energy. Spend a few minutes with the Hyperloop racing team, and you'll catch it.

Matties: That was going to be one of my next points: You're dedicating a lot of your time to sharing your knowledge with the Hyperloop team and other young designers.

Watson: I am. I was one of the Hyperloop technical advisors last year, and I have the honor of doing that again with the new team. Each year, a new team comes on, and they asked me if I would be willing to do it again. My answer was, "Absolutely."

Matties: What advice would you give to other designers who want to follow that path and get involved with young people as a mentor or in some other way?

Watson: Put yourself out there. Everyone has something to offer. We're all at different levels in our knowledge and techniques. Another great thing about Altium is they give you a platform to work from; that's Altium's mentality for developing a community of designers that can then start meshing.

The one great thing I've seen at every single AltiumLive is a few hours into the event, people start opening their laptops. You'll have a group of people from all different areas and levels around the computer, cranking their necks to get the view of the screen, talking about things, and meshing together. That's what I envision and would like to see with the community of developers; novice or expert, we can all come together and share each experience, knowledge, and discuss challenges and issues we're facing.

Matties: What knowledge are young people looking for right now?

Watson: It varies. I came from the streets of designing, and I say that because I did not come through the conventional way of PCB design; I came in as an electronic technician. I then became interested in PCB design. I am not an EE, but I've worked my way up through the ranks. Once a lot of young people get out of school, they realize they haven't learned everything they need to know. Then, you start learning the street knowledge and learning the facts of life.

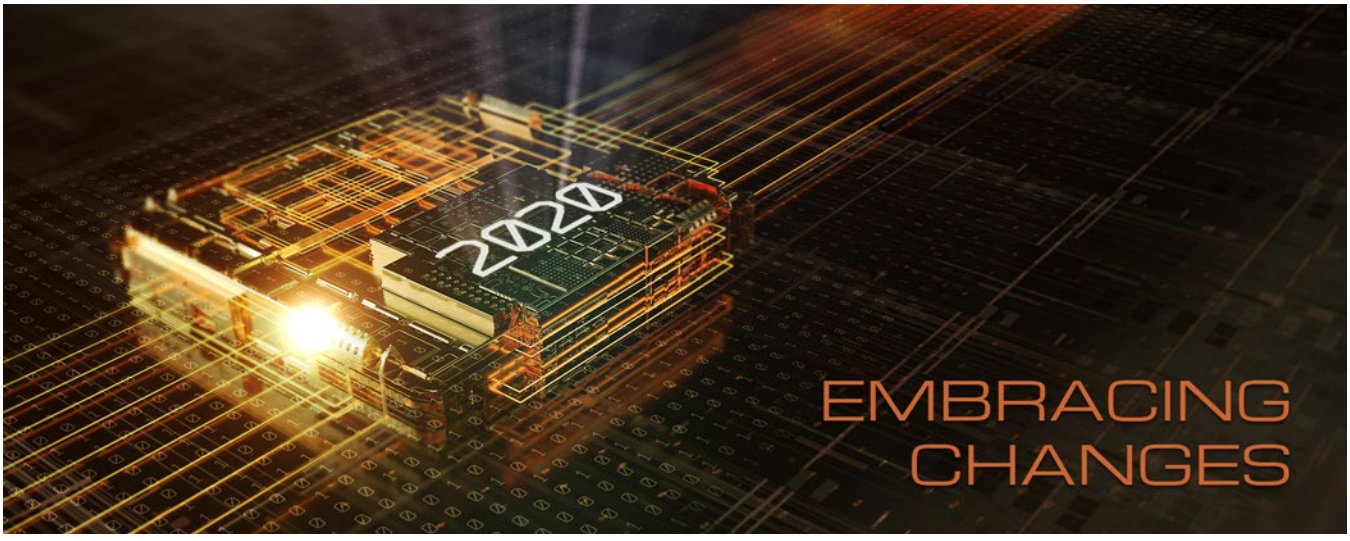
Once a lot of young people get out of school, they realize they haven't learned everything they need to know.

Matties: The facts of a designer's life.

Watson: Yes. It's not like what they might have told you in school, and you begin to learn the street knowledge that's out there. Many designers have worked their way through this. For example, maybe you don't want to know how to fully analyze something and or all the intricate details of everything, but how do you get from point A to point B? Companies are interested in if you can give them a PCB that works; they don't need all the intricate details.

Matties: Thanks for your time, John.

Watson: Thanks for the opportunity. DESIGN007



IPC Designers Council Has a New Name: IPC Design

Feature Interview by Andy Shaughnessy
I-CONNECT007

Yes, you read that right. The IPC Designers Council is now known as IPC Design. Many of you have heard secondhand stories about what this change will entail, so I asked IPC to shed some light on this subject. I recently spoke with IPC's Teresa Rowe and Patrick Crawford about what's changing, what's not, and IPC's plans to provide improved infrastructure for PCB design content and curriculum.

Andy Shaughnessy: Teresa, why and how is the IPC Designers Council (DC) changing?

Teresa Rowe: It has been about 15 years since IPC took a look at the design community at large. During that time, a lot has changed in the industry. It was time to look at where our pockets of designers were and how IPC's design products were being used in the industry and addressing the design programs, overall. It is changing in a way that will address the community at large.

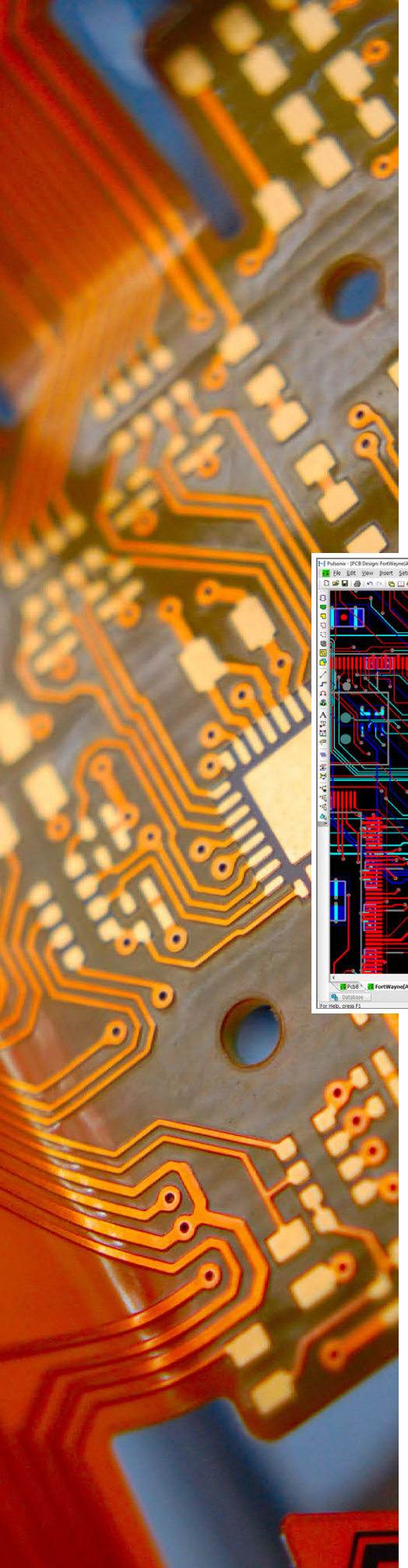
Shaughnessy: And what will the new program name be?

Patrick Crawford: It will be called IPC Design. We're going to be mixing the power of IPC and the design community.

Rowe: It embraces standards development, education, the foundation piece of it, designer community leadership, etc. It brings the design community together in one place to move forward.

Crawford: The core of the new program will be to empower and benefit more PCB designers, as I think it always has been. We want to bring more of IPC's resources to bear on the design community in the sense that we have a vast member network and a vast digital infrastructure that can track committees, create groups, and allow them to disseminate knowledge online. We have a large body of expertise in our IPC education and certification department. The point is that we have many parts of IPC that could be better channeled into the design community.

Looking at this, we love the idea of having these regional groups as the IPC DC—a forum by which designers can communicate and disseminate knowledge. What we want to do is take that idea and build a stronger scaf-

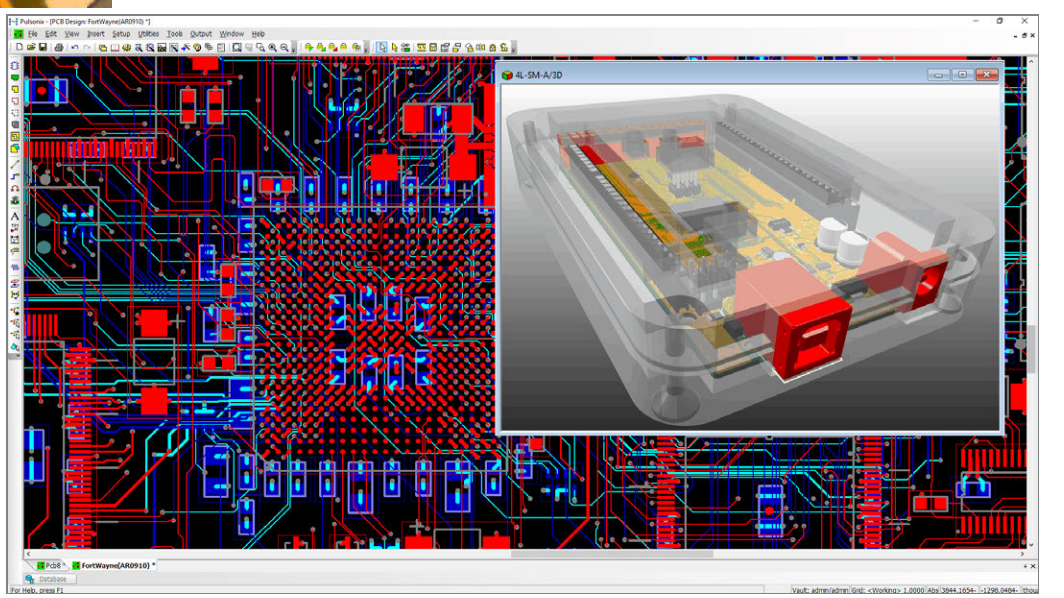


Pulsonix PCB Design Software

PCB design done right

Maximum Productivity in Minimal Time

With a modern, easy-to-use interface and advanced capabilities to automate your layout process, Pulsonix can be the key to a critical reduction in your design time. That's PCB design done right!



Advanced PCB Design Features

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

Try Pulsonix yourself at pulsonix.com/trial

PULSONIX

fold around it so it can be better supported by IPC. And to answer how the IPC DC infrastructure will change, it won't really. We're going to call them Design Chapters to better align with the nomenclature of IPC, but they are still going to have the same executive structure within those smaller groups, such as president, vice president, etc.

We're also going to have regional advisors, who will be responsible for more than one Design Chapter. They will be the liaison between the chapters and IPC, and your group of industry experts will help IPC drive this design program. This group will be called the Design Community Leadership Team and be comprised of seven individuals from the industry. We are modeling them after IPC's TAEC (Technical Activities Executive Committee) Global in that they will have two individuals from the Asia-Pacific region, two from the Americas region, two from the European region, and then one chair that can be from any region. The designers and the chapters will be supported by the regional advisors, who are then supported by IPC and the Design Community Leadership Team; all of them work together.

Patrick Crawford

We're looking to these existing chapters that have successful meetings, such as "lunch and learn" events that drew 50–100 attendees, as examples because they're doing something right. However, the issue was that success was not universal, and we want to learn from them. But if you are a president of a chapter right now, I'm not going to walk in and demand your papers. You're going to stay, and we're going to work with you to transition into a new model as a team.

Rowe: One of the other things we've adjusted is that our chapters have been pretty much on their own. They have been responsible for finding content for those meetings, and one of the things we want to do is look at content that we can push to all of our chapters as material that they can provide during their meetings. This is one of the reasons we're looking at the Design Community Leadership Team to help us to choose those topics to launch community discussion. We're also looking at ways to get them actively engaged and feeding back into IPC. It's a two-way street, and we don't want the chapters feeling like they're on their own.

Shaughnessy: It did seem like a handful of chapters had regular, well-attended meetings, while others were largely dormant.

Crawford: Exactly, and we're looking at that as an opportunity to bring all of these existing chapters to that level or benchmark of engagement.

Shaughnessy: Some designers were thinking that the local DC chapters were going to be giving up the ability to bring in certain speakers or share their own content. Will the chapters still have autonomy, or will they have to clear the content with IPC?

Rowe: They'll still be able to do that. We want to be able to provide them with some subjects and information and give them a way to draw in others. Maybe there's an important topic that everyone should be discussing, and the Design Community Leadership Team says, "We'd like to have input from these groups." Then, that's something we will ask them to do to get that two-way street going. But those chapters will still be able to continue on having local speakers as they choose.

Shaughnessy: The designers, as you know, are very passionate about their local DC chapters. A lot of them are worried about this change, but there was clearly room for improvement with the way the DC worked with the rest of IPC, and I think the designers all see that.

Rowe: As I said, it has been 15 years or more since we've taken a real long look at this. And as Patrick indicated, we use the success we've

had with IPC TAEC Global as a place to start looking at what we can do to bring the design community together in today's industry.

Shaughnessy: Tell us a little about IPC TAEC Global.

Rowe: IPC TAEC Global is a group of seven individuals that represents all corners of the world. The group comes together to look at the top activities in our standards development efforts.

Teresa Rowe

Shaughnessy: It would be great if we could get the chapters more connected and involved, including people in areas where there are no chapters.

Crawford: Absolutely, and that's part of what we want to do with having this new structure. If you have a group of designers, but the closest chapter is a five-hour drive away, now, you have a line to IPC through a regional advisor and could develop a chapter. That's not going to be an issue anymore because it will be more empowering for smaller groups to organize.

Shaughnessy: I think there needs to be some better guidelines on how to set up a chapter.

Crawford: One of our inspirations in going forward with this is the IPC Education Foundation, which has a great policy and procedure for starting a student chapter at universities. There's a great infrastructure built in there, and we're looking to do something similar so that it's a matter of filling out the right forms and calling the right people at IPC. We want to help the designer not waste time playing phone tag. We want to advance PCB design and help PCB designers better themselves.

Shaughnessy: Shifting gears a little, tell us about the CID and CID+ programs.

Rowe: The CID program is changing. If you've received a CID in the past, it's good forever. But going forward, the CID program will fall under the policies and procedures of our training and certification programs, and as part of that, there will be a program unveiled that will require designers to re-certify on an ongoing basis to remain current. Again, the same as our CIT and CIS courses now, so someone can take the training and receive their certification; it will be good for two years, and it expires at the end of two years. It doesn't mean the knowledge goes away, but the certification ends in two years.

Shaughnessy: It can be difficult to get designers to take the CID exam one time. Do you think they're going to come back every two years?

Rowe: It depends. I can't make up their minds for them, and our other programs are set up this way, so it's about staying relevant in the industry.

Shaughnessy: How is this going to affect EPTAC's classes on CID and CID+?

Rowe: The way it works for our other programs (i.e., assembly, board fabrication, and rework and repair) is that we have master training sites in the U.S. that are licensed to teach these courses. Right now, the plan is for the CID program to fall underneath the policies and procedures of those same requirements. And we're not saying it won't change in the future, but that's where we are today.

Shaughnessy: Are the members of the Design Community Leadership Team going to be designers and EEs?

Crawford: Yes. And we are consciously selecting professionals who are forward-thinking, open to those changes we talked about with the certification, and have been active members in leadership positions in IPC committees.

Rowe: It's important to remember that nobody likes change. Recently, we started to look for emerging engineers. We have embraced the younger community, so we're trying to bring designers together to move it forward into the future. We want to bring this whole group under an umbrella to move the whole industry forward in a way that is not disruptive but brings out the best in people.

And we recognize the fact that even we work virtually. We understand that we don't need to be face-to-face to have a productive meeting. From that perspective, we want to be able to get those people together who can't drive the five hours to the nearest chapter or even across town for a meeting in the evening. But designers might be able to carve out a slice of time to have a meeting if they're sitting at their computer at work or at home, so we want to include them as well.

Shaughnessy: Is there going to be a cost to any of this?

Crawford: No. We aren't doing this as a new way to make money; this is an opportunity to give back to the design community. If chapters want to help subsidize their activities or maybe some sort of endowment for scholarships, we'll still leave that up to them. The chapters will maintain autonomy in that regard, and we encourage them to do so.

Shaughnessy: Who is the leader of IPC Design?

Crawford: At the end of the day, the designers are going to be in charge; we want IPC Design to be from designers and for designers. The Design Community Leadership Team will be their earpiece to receive comments from the industry through the regional advisors. Then, using their expertise and knowledge about how this design engine works, they will recommend programs to IPC, and we will work with them to explore how we can implement those programs at the chapter level.

Shaughnessy: Who would a chapter president call for help or advice after this is all set up?

Crawford: Call the Design Community Leadership Team. You would talk to the chair, and they would help you. And in our technical standards committees, they use IPC Works, which is an online platform that allows groups to come together and share ideas, files, forums, etc. It's an awesome tool. We would like every design chapter to have its own IPC Works page. It could be where they share the meeting minutes, new activities coming up, or a centralized calendar. It could also be where they plan and schedule with groups from across the world, instead of having to use Skype, etc.

Rowe: IPC Works is an intranet-based tool, so it allows internal communications, messages to the group, places to store files, and places to post updates. There's a newsfeed, so if there's information they want to push to the group, they can. We rolled this out on March 4 of last year, and it has grown and been embraced very positively by our standards development activities.

Shaughnessy: That would definitely help—anything to make the communication more uniform. Thank you both for speaking with me about this.

Rowe: We're excited.

Crawford: Thank you for helping us get this information out.

If you're interested in becoming involved with IPC Design, contact Teresa Rowe (teresarowe@ipc.org) or Patrick Crawford (patrickcrawford@ipc.org). DESIGN007

Your circuit boards delivered...

1 DAY LATE

2 DAYS LATE

3 DAYS LATE

WHAT'S THE COST TO YOU WHEN YOUR BOARDS ARE DELIVERED LATE?

With Prototron, you don't have to ask that question. Serving customers for over three decades with a 98% on-time delivery record, we understand that providing you with high-quality PCBs on time, and right the first time, is critical.

Prototron Circuits

America's Board Source

www.prototron.com

Design Tips for Layout

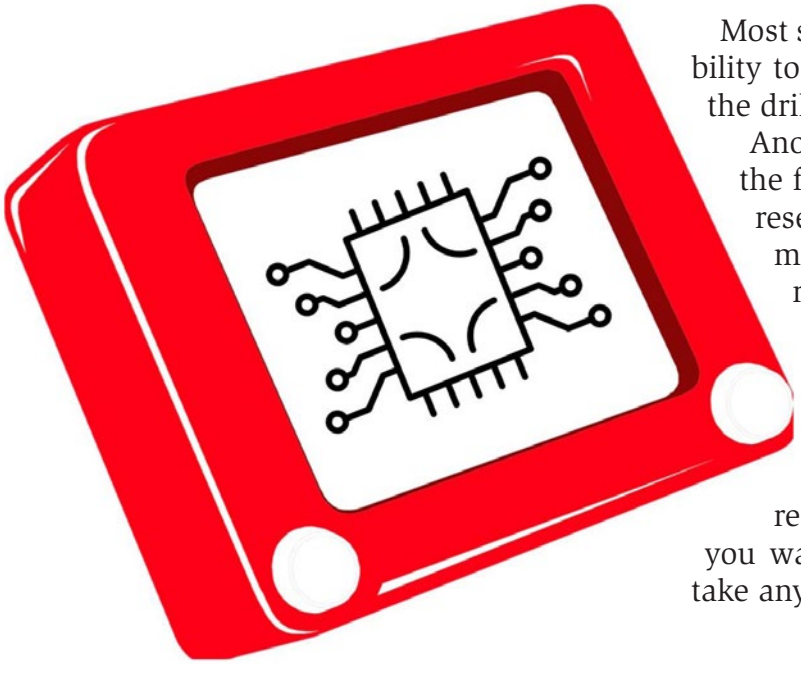
Connect the Dots

by Bob Tise and Matt Stevenson, SUNSTONE CIRCUITS

As a PCB manufacturer, we receive hundreds of PCB layouts represented in Gerber format every week. As you might expect, they're not all created equal. Some of the layouts check every box and roll straight into manufacturing, while others need work before they can be sent to the production floor.

We affectionately refer to these layouts on occasion as Etch-A-Sketch designs, meaning that they did not adhere to best practices nor pay close attention to detail during the layout phase. And yes, sometimes these designs really do look like they were created on a toy from the '70s and not in a CAD tool. We recognize there are many good reasons for not-quite-ready-for-primetime designs to be submitted, such as:

- A designer lacking layout experience
- Looming deadlines and rushed design processes
- A lack of attention to the details
- A need for something to test in a hurry



Often, these issues create delays and issues with yield or reliability. They can be avoided before they cause problems with your budget and production schedule. Here are some best practices for doing so.

Use the Tools in Your CAD Program

You can represent a feature in your design by almost any means you'd like, but adhering to a few best practices leveraging the many useful functions in your CAD design tools can help eliminate confusion for your PCB manufacturer. For example, you can represent slots as overlapping drilled holes in your design. It *is* a common practice, but it creates two potential issues for the manufacturer:

1. If left as is, it can create broken drill bits impacting yields and quality of the product.
2. The other possibility is that the tooler corrects the issue during manufacturing, which is a very inefficient process.

Most software for PCB layout offers the capability to make a slot and have it contained on the drill layer of the Gerber files.

Another common design issue comes in the form of circles drawn on a layer to represent non-plated mounting holes. Your manufacturer may easily interpret this representation to mean you want copper circles on your PCB. Creating the mounting holes in your design tool and assigning them the plating type that you need is a much better approach. This will ensure you receive the non-plated mounting holes you want, not copper circles, *and* it doesn't take any more time.

APEX EXPO IPC 2020 | Celebrating 20 YEARS OF EXCELLENCE IN ELECTRONICS

February 4-6, 2020
SAN DIEGO CONVENTION CENTRE
VISIT US: Booth 2413

Our winning formula for superior electro-chemicals

When every connection counts, you can count on Electrolube's electro-chemical expertise.

With a 77-year pedigree, a growing presence in 55 countries and production in 3 global locations, we have the products, research facilities, resources and personal expertise to engineer solutions to your manufacturing challenges.

Make contact and discover what makes Electrolube the solutions people for leading manufacturers worldwide.

Tel: 800-474-1472
www.electrolube.org

ELECTROLUBE
THE SOLUTIONS PEOPLE

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

Pay Attention to Your Gerber Files

In many cases, the Gerber files you submit for manufacturing will be the *gold standard* by which the manufacturer creates your PCB. Most of the higher end PCB layout tools do a pretty good job of creating Gerber files that do justice to the layout, but there are certainly exceptions.

Utilizing a Gerber viewer to display your files before submission can help you to identify potential issues ahead of manufacturing (Figure 1). This practice can even limit your need to troubleshoot a completed design and use the *blue* wires to make it work. Common issues discoverable using the Gerber viewer include:

- Overlapping thermal connections on adjacent pins (isolating intended connections)
- Drilled holes in an unintended copper feature (like through a trace)
- Oversized or missing solder mask clearances (potential to create solder shorts or unsolderable situations)

Just Because You Can Doesn't Mean You Should

Sometimes, you need to zoom in to see it. Many PCB layout tools are creatively accommodating and will let you do nearly anything digitally, but the consequences can be staggering during the transition to manufacturing. For example, we see many submitted designs that are virtually unbuildable due to the blind and/or buried vias in the design.

Blind and buried vias are one sure way to drive up the cost, extend lead time, and increase yield loss. We recommend that you do not build blind or buried vias into your design unless absolutely necessary. It goes without saying for this audience that comments like, "I was able to draw it, so you should be able to build it," are only made in jest. PCB manufacturing must still adhere to the laws of physics, and that makes some things impossible to build at any cost.

Conclusion

At the end of the day, it comes down to taking your time, checking your work, and applying thoughtful design concepts to your PCB

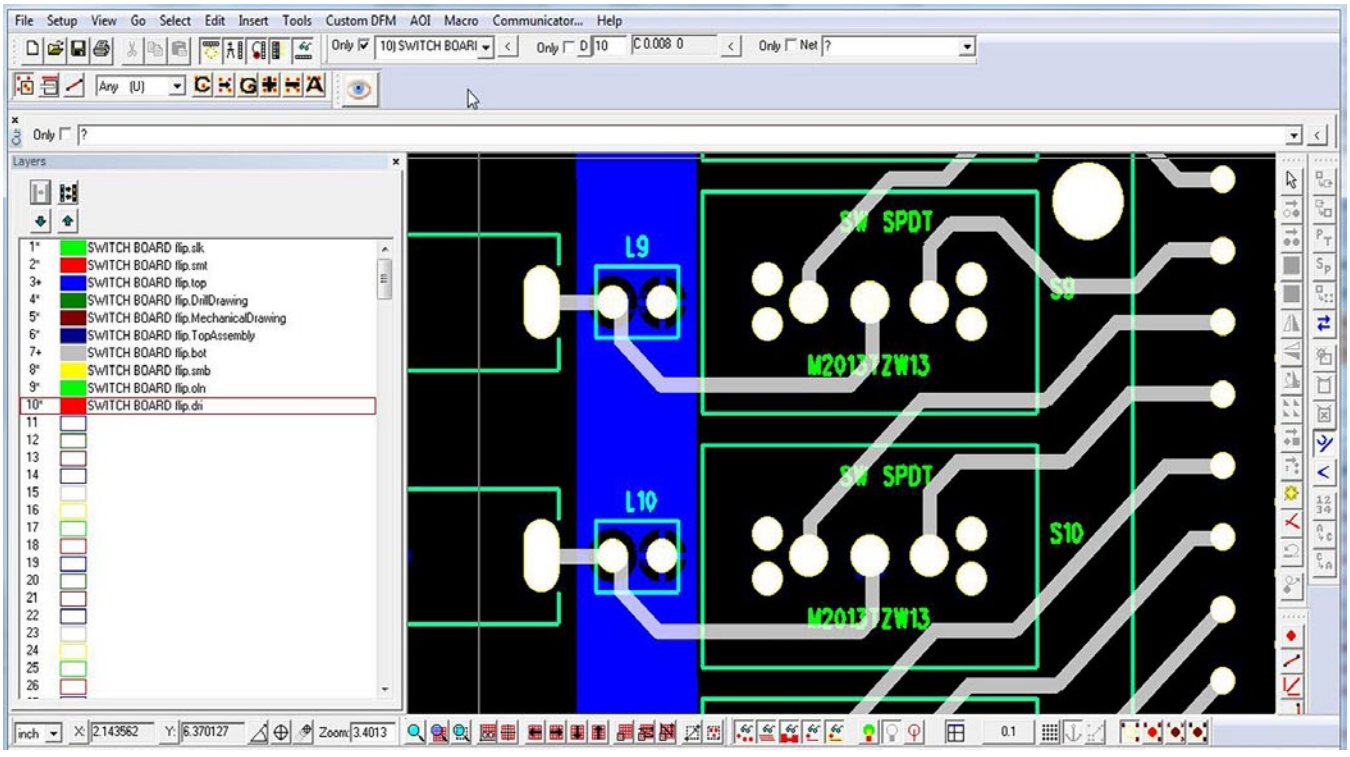


Figure 1: Gerber viewer.

layout project. PCB layout is not rocket science (well, sometimes it actually is). Applying a few best practices, and using common sense steps in the design phase, can be a serious time saver. It is like the old adage about health: An ounce of prevention is worth a pound of cure.

There are many resources available to help you be successful, from YouTube videos to colleagues and your PCB manufacturer, to name a few. Good luck out there! **DESIGN007**

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

Bob Tise

Matt Stevenson

Chemists Glimpse the Fleeting Transition State of a Reaction

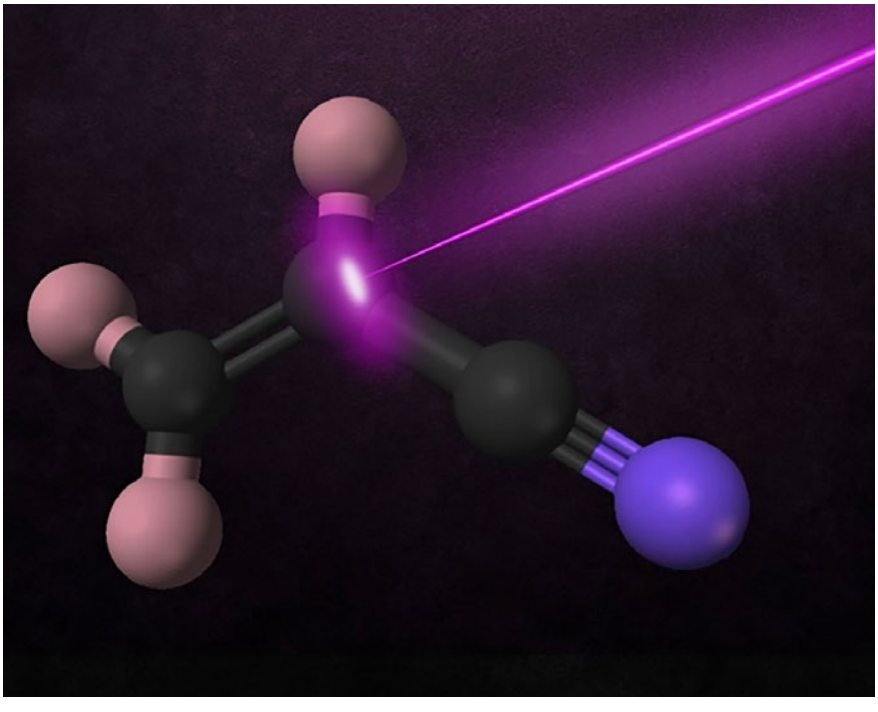
During a chemical reaction, the molecules involved in the reaction gain energy until they reach a "point of no return" known as a transition state. Until now, no one has glimpsed this state, as it lasts for only a few femtoseconds (quadrillionths of a second).

However, chemists at MIT, Argonne National Laboratory, and several other institutions have now devised a technique that allows them to determine the structure of the transition state by detailed observation of the products that result from the reaction. "We're looking at the consequences of the event, which have encoded in them the actual structure of the transition state," says Robert Field, the Robert T. Haslam and Bradley Dewey Professor of Chemistry at MIT.

In a paper published in 2015, Field and his colleagues used laser spectroscopy to characterize the transition state for a different type of reaction known as an isomerization, in which a molecule undergoes a change of shape. In their new study, the researchers explored another style of reaction, using ultraviolet laser radiation to break molecules of vinyl cyanide into acetylene and other products. Then, they used millimeter-wave spectroscopy to observe the vibrational level population distribution of the reaction products a few millionths of a second after the reaction occurred.

Using this technique, the researchers were able to determine nascent populations of molecules in different levels of vibrational energy–a measure of how much the atoms of a molecule move relative to each other. Researchers are now using this technique to study the reaction products of the pyrolytic breakdown of acetone.

This also allowed the researchers to distinguish between two slightly different products of the reaction – hydrogen cyanide (HCN), in which a central carbon atom is bound to hydrogen and nitrogen, and hydrogen isocyanide (HNC), in which nitrogen is the central atom, bound to carbon and hydrogen. (Source: MIT)



Designing PCBs for Large-scale LED Screens

Interview by Andy Shaughnessy
I-CONNECT007

I recently interviewed long-time PCB designer Rainer Beerhalter of Squadrat, an NEC company. He shined a light on his design specialty: Designing boards for large-scale LED screens that are installed in arenas and on the sides of skyscrapers around the world.

Andy Shaughnessy: Rainer, it's good to see you again. Give everybody a real quick background about yourself, your job, and the company where you work.

Rainer Beerhalter: I've been in the electronic design business for more than 25 years. I now have my own company, and we work as a design service. Since about two years ago, I've been in a contract situation with a big Japanese company that I design LED systems for. This is the domain I've been working in since 1996, so I've been designing LED systems for a long time. This is from the system architecture down to the LED implementation design, so every level of electronic design and systems design and parameter design is my job description. I work with a team of engineers from many disciplines. I'm a physicist, so I have a scientific background. My teams are electrical engineers or IT technology experts, who write all the firmware, so the systems are very complex.

And what makes it special is we do not think in square millimeters; we think in square meters. Each year, my responsibility is about 5,000–7,000 square meters of PCB production in the Far East. The main products are huge LED screens you see in football stadiums, public advertising, traffic signs, etc. The situation is this: how do you accomplish this job, designing these kinds of products, into PCB layout?

Shaughnessy: So, you're talking about the giant screens, like at The Bund in Shanghai.

Beerhalter: Yes, and other LED screens you see in the skyline in Hong Kong and Shanghai, for instance.

YOUR PCB PORTAL TO THE WORLD

DELIVERING GLOBAL-SCALE PCB SOLUTIONS

15
Subsidiaries Worldwide

80
Leading PCB Engineers

50
Countries Supplied

40
Thousand Different Items Annually

When you work with us, you have effortless access to a local partner who understands your short and long-term objectives, motives, environment, and concerns, while at the same time enjoying our global presence and the wide commercial advantages it brings.

VIEW CASE STUDIES

FINELINE
EXCELLENCE IN PCB

fineline-global.com

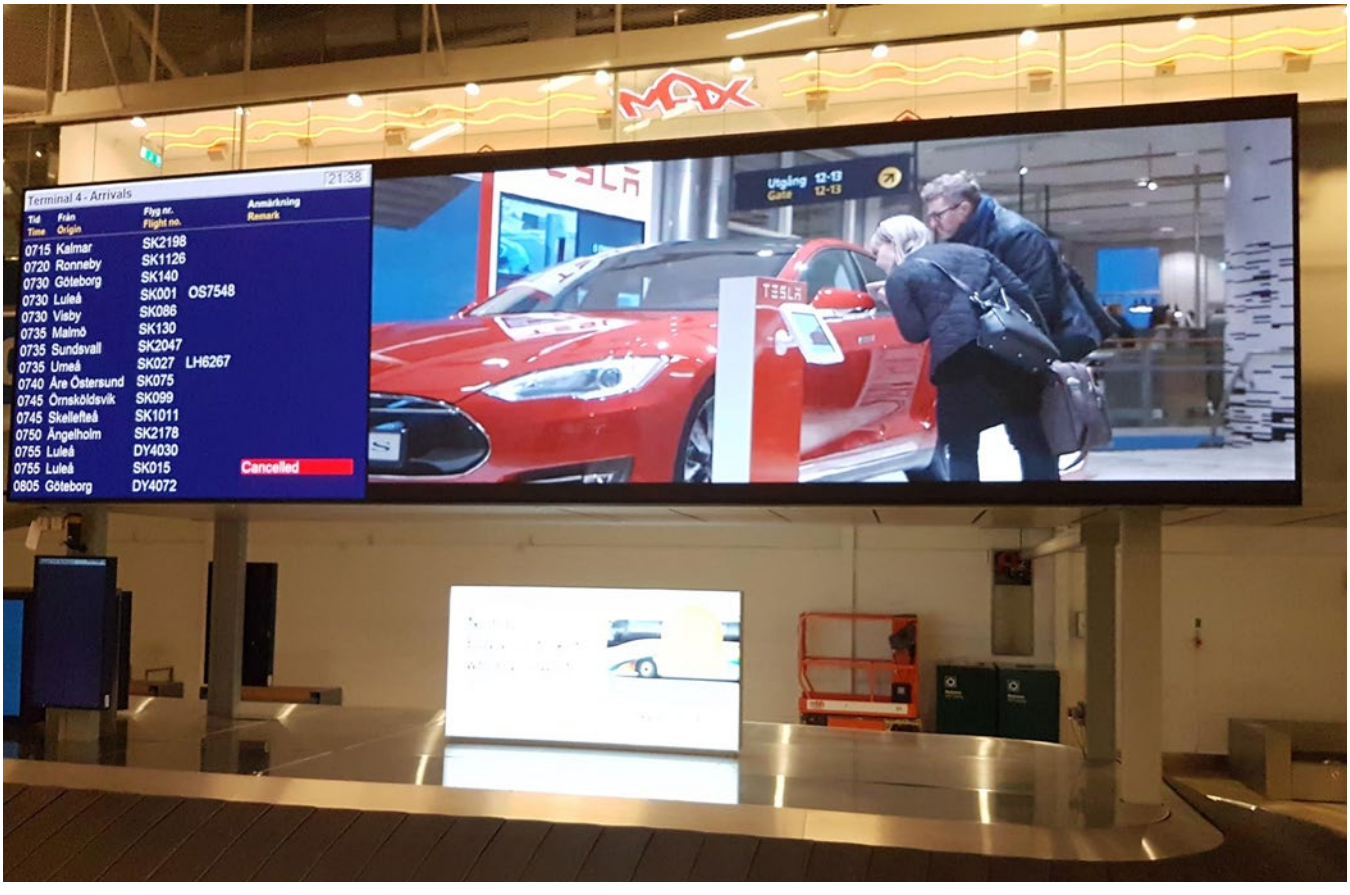
Shaughnessy: What are some of the unique problems that are associated with doing circuit boards for LED applications?

Beerhalter: There are two challenges, depending on the market you are serving. Most of the LED products go into the mass market, so the primary design target is cost-effectiveness. How do we bring down the cost? One of the biggest numbers I always talk about is that 22% of an LED system's costs go into PCBs. That's significant because if you imagine the type of problem, the screen size determines the amount of PCBs you use. Behind every LED you see in the front, shining up and lighting everything, there's a space on the PCB. The second challenge is design for manufacturing and bringing down the cost, and design ideas to bring down the design time because you want to avoid designing 50 square meters of LEDs. You want to design a small tile and replicate it. This can be a good challenge to figure out how to do more than copy and paste to get systems to designs.

Shaughnessy: I heard an engineer recently say that the trick is not how high-tech you can make the board, but how dirty you can make the signal and have it still work.

Beerhalter: That's absolutely right. Let's say the prime objective in every design job is functionality, so you design for function at the lowest cost possible. This is the objective we have in common with the car industry. But there's a second objective we have in common, and this is harder to achieve, and sometimes our competition disregards it. We design not only for function and signal integrity but also for EMI compliance. You don't have the chance to test that on a small scale; you have to test that at big scale. The whole display must be okay. Most designers have challenges bringing one small board to life. Imagine my situation: I have to bring 1,000 boards into the EMC chamber and have them pass EMC in one measurement.

Shaughnessy: You can't run one prototype and see if it fails the smoke test.



Beerhalter: I told one of the colleagues yesterday that the design target is very simple. Design one board, put it into the EMC, get EMI emission measurements, and you should see nothing. If you see some signature of the board, you won't have a chance to run that in 2,000 pieces on one screen. We must have the lowest emission possible at a reasonable cost; that is our objective.

I try to teach my engineering staff about more expensive and less expensive design decisions because it's very simple. When you have a PCB in your hands, my opinion is it's a result of thousands of decisions the design team makes during the development of this product. We always try to teach my younger engineers that every time they do that, the only component in your design that you have an influence on is the PCB; all others are commercially available parts. As long as you are not a chip manufacturer, by hobby or by profession, you have no impact on any of the parts; the only part you influence is the PCB, which is important to remember.

Shaughnessy: That's interesting. You're doing this on a whole different scale than everybody else. I bet you had to educate yourself on a lot of this because not many people design boards for what you do.

Beerhalter: There may be many designers that try to do that, but a lot less of them do it with the same attitude and objectives.

Shaughnessy: One designer I know designs boards for the gaming industry and casinos, and he had to do the LEDs that light up when you win. It was a lot smaller scale, but they had heat problems and things like that. Do you have to deal with heat?

Beerhalter: Yes, the biggest problem in our job is the heat system because even LEDs are told to be efficient. When you compare it to classical light bulbs with 3–5% optical efficiency, the LED is around 30%. Normally, for an outdoor display, you need around 800 watts per square meter of electrical energy. And when you think about 30% efficiency, this says 70% of this energy has to be cooled down or transported with a heat pipe system or however you solve that.

Rainer Beerhalter

But if you let it in the display, it heats up. I've seen products where you can boil eggs on them once they're switched on. They run up to 80°C or something like that; that is not our goal. Electromechanical design for cooling is the challenge, and getting all that done means interdisciplinary work with mechanical engineers. All the disciplines are involved: electrical, mechanical, electrical safety, etc. And this is most important in the Japanese design topology that I am trying to adopt. We have to follow 170 international standards to implement in the product. Reading those 4,500 pages could cost me three months of my life. But now I know what standards you have to follow in each country for a global product.

Shaughnessy: Do you have any advice for a PCB designer moving into the LED segment?

Beerhalter: Try to light and control one single LED in a perfect way. Then, the rest of the story is thinking about replication. What will change if two LEDs are on a board? What will change if a quad of LEDs is on the board? How does the visual perception system of the people work to get the optical quality? In the end, people don't pay for the LED; they pay for the visual experience, and they pay for emotions. Our sales figures are a transfer of emotions, not LEDs.

Shaughnessy: And everyone can see what you're doing. You're on display!

Beerhalter: Exactly. You cannot imagine the pressure you have when you have an audience of 70,000 people in a football stadium, and your system goes down because you've made

a mistake. It's not dangerous, like in avionics or the car industry, but the pressure can be fairly high.

Shaughnessy: How long have you been working with LEDs?

Beerhalter: I became interested in electronics at age 13, but I spent the first nine or 10 years of my "electronics life" without LEDs. After that, LEDs were my life's challenge. And when I was at university in 1993, I got one of the first LEDs of blue color from a Japanese scientist team who later received the Nobel Prize for the blue LED invention. It was extremely difficult at that time, from semiconductor processes, to create blue emissions of semiconductors.

Shaughnessy: What advice would you give to younger engineers?

Beerhalter: I like to design products because I like electronics and creating things. One of the best things in my life was being able to implement my ideas into products, and this is a luxury not everybody can have. I encourage young colleagues to think out of the box. There's more than routing wires on a PCB; it's about implementing an idea.

What always helped me is the story of when the U.S. president came to NASA before the Apollo 13 program and visited one of these big buildings where the rockets were created. They went down to the lounge area, and he saw an old man cleaning up the floor. The president asked that man, "What are you doing here? Are you cleaning up?" The old man said, "No, I'm helping men reach the moon." Our jobs have to do with motivation, and this story always helped me to think out of the box. Again, I'm not creating LED screens; I'm trying to sell emotion and a visual experience, which is a bigger scope.

Shaughnessy: Those large LED screens make you say, "Wow."

Beerhalter: Yes. We are the Louis Vuitton of electronics; nobody needs it, but everybody wants to have it.

Shaughnessy: Thanks for your time, Rainer. It has been great talking to you.

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

Zero-emission Flight Takes Giant Leap Forward

Introducing the E-Fan X, a hybrid-electric aircraft demonstrator that is 30 times more powerful than its predecessor. The E-Fan X is the flagship hybrid-electric aircraft demonstrator at Airbus. But its predecessor, the all-electric twin-propeller aircraft E-Fan 1.0, can be credited for playing a key role in laying the groundwork for the next generation of zero-emission aircraft technology.

On July 9, 2015, the world of aviation witnessed a remarkable milestone. At the Lydd Airport on the south-east coast of England, a twin-propeller aircraft was preparing to embark on an important journey. Its destination? The Calais-Dunkerque Airport–just over 74 km away by air.

Less than an hour later, the aircraft–known as E-Fan–landed safely in Calais. But the E-Fan was no ordinary aircraft; it was powered solely by lithium-ion batteries. The E-Fan became one of the first all-electric aircraft to successfully cross the English Channel. It also paved the way for the next generation of electric-powered and zero-emission aircraft technology at Airbus.

The original E-Fan has now been retired and is currently on display at the Aeroscopia Museum in Toulouse, France. But thanks to the lessons learned from the E-Fan 1.0 project, a new aircraft demonstrator has emerged: the E-Fan X. This hybrid-electric aircraft demonstrator is 30 times more powerful than its predecessor. With the E-Fan X, zero-emission flight takes a giant leap forward.

(Source: Airbus)

PCB007 Highlights

The ICT 2019 Christmas Seminar ▶

Since 2016, the Institute of Circuit Technology (ICT) has held its northern area Christmas seminar at the Majestic Hotel in Harrogate—the elegant and historic English spa town in North Yorkshire. Pete Starkey provides an overview of this popular ICT event.

Standard of Excellence: Respecting Your PCB Vendor Partner ▶

One of the more important aspects of any partnership is mutual respect for one another. And, of course, this is extremely important when it comes to your PCB vendor partner. Anaya Vardya shares five do's and don'ts to help build respect into your PCB vendor partnership.

Day 1 Coverage of International Electronics Circuit Exhibition in Shenzhen ▶

The newly named International Electronics Circuit Exhibition, formerly known as the HKPCA Show, kicked off in Shenzhen, China. This event is now part of a cooperation between the CPCA and HKPCA associations. This is the largest trade show serving the south China region with 3,537 booths from 621 exhibitors this year.

One World, One Industry: Rising Tariffs Put a Painful Squeeze on Electronics Manufacturers ▶

IPC recognizes that you are greatly affected by the ongoing tariff dispute between China and the United States. To date, the two countries have imposed tariffs on hundreds of billions of dollars in imports in a tit for tat that could escalate further if an agreement is not reached soon. Dr. John Mitchell discusses IPC's recent survey of U.S. members and the organization's perspective.

It's Only Common Sense: Making 2020 the Best Year of Your Sales Career ▶

Even if you didn't do your homework last year, it's not too late to set yourself up for success—all you need are two days of planning and a lot of hard work to make 2020 the best year ever. Dan Beaulieu shares 10 surefire steps to help you be successful in 2020.

Ventec Invests in Key Equipment to Support Expansion of PTFE Laminates Manufacturing ▶

Ventec International Group Co. Ltd. has significantly increased its PTFE laminate manufacturing capabilities following a strategic investment in a new state-of-the-art high-temperature press and lay-up/break-down line at its Suzhou (China) manufacturing plant.

Conductivity Laminate for High-frequency Applications ▶

Pete Starkey and Vitali Judin discuss how Rogers Corporation has responded to demands for high thermal conductivity combined with low loss in high-power RF applications with the newly launched. The PTFE-based TC350 Plus is capable of handling temperatures above 125°C and draws upon Rogers' long experience with the TC350 material line-up.

Looking Back on IPC's First Annual Electronics Materials Forum ▶

This year, IPC tried something new and developed a technical conference to bring together industry professionals focused on materials development. IPC's Electronics Materials Forum stretched through each industry silo and focused on the commonality of developing the materials we all use. Brook Sandy-Smith provides a recap of the event.


Charlene Gunter du Plesssis


Corey Lynn

Engaging STEM Students at IPC APEX EXPO 2020

Feature by Nolan Johnson
I-CONNECT007

At this year's IPC APEX EXPO, you're likely to see quite a few high school students moving amongst all the normal show activities thanks to the IPC APEX EXPO STEM Outreach Program. Launched two years ago at IPC APEX EXPO 2018, the 2020 version of the STEM Outreach Program will be larger and more immersive than ever before.

Charlene Gunter du Plessis, director of strategic partnerships and programs at IPC, shares: "The IPC APEX EXPO STEM Outreach Program has grown on an annual basis. This will be the third year hosting high school students at IPC APEX EXPO, and the program has evolved ever since." She adds, "200 high school students and educators from the San Diego region will join us during an interactive day of hands-on technical activities, career exploration, and industry engagement."

The STEM Outreach Program will take place on February 6, 2020. The organized activities will start at 8:00 a.m. and conclude at about 3:00 p.m. After a kickoff breakfast, students will move to the show floor, where they will rotate through four different education tracks throughout the morning. The tracks will provide students with real-world technical skills training in soldering, coding, design, and assembly of PCBs as well as an IPC APEX EXPO show floor tour. After a career panel luncheon, students will complete their day with one more education track before returning to their staging area for closing remarks and dismissal.

IT-988G SE / IT-988G

IT-988G

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

IT-988G SE

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

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

Insertion Loss - Measured

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

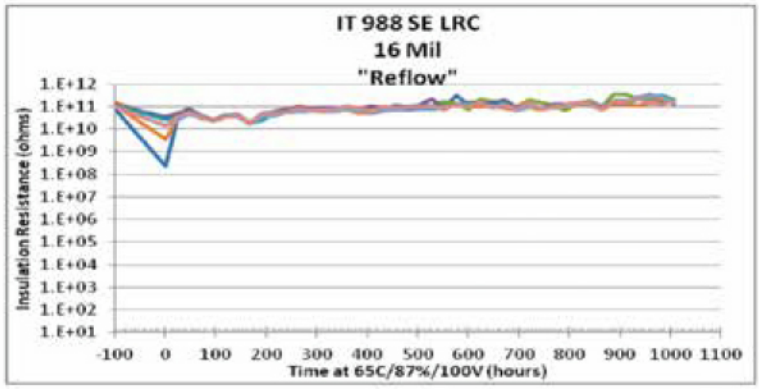
Sequential Lamination

7 Lamination cycle data

Lamination	DMA	DSC	TMA	T300 with CU	Solder Dip PCT: 1h @ 121°C	Td 2wt% / 5wt%
1	213	187 / 187	182	> 60	> 60	408 / 435
2	216	194 / 199	193	> 60	> 60	417 / 438
3	214	186 / 192	185	> 60	> 60	417 / 442
4	216	193 / 194	184	> 60	> 60	424 / 443
5	217	194 / 199	190	> 60	> 60	418 / 442
6	218	191 / 197	188	> 60	> 60	403 / 436
7	218	190 / 197	194	> 60	> 60	425 / 444

No thermal degradation observable after 7 lamination cycles

HDPUG MRT-6 LRC CAF Data



IT-988G SE

- High Tg / Lead Free / Halogen Free
- Ultra Low Loss Laminate & Prepreg
- 100G / 400G Solution
- Lower Dk (<3.24 @ 10GHz)
- Ultra low Df (<0.0014 @ 10GHz)
- Very Stable Dk-Df across frequency
- HDPUG MRT-6 - LRC and HRC passed all thermal requirements
- HDPUG MRT-6 - LRC and HRC passed CAF requirements
- Proven thermal stability beyond 7 lamination cycles and suitable for HDI applications

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

*53% resin content

ITEQ

The soldering stations are a regular highlight on STEM Outreach days.

Corey Lynn, education marketing manager at IPC, adds, "We're excited that we are doubling the number of students who joined us last year and we're doing more activities. Last year was mainly a tour of the show floor and some light soldering. This year, we're adding to the fun." Lynn continues, "There will be a session on PCB design, including evaluating the design of the pin, by Professor Kris Moyer from Sacramento State. We'll also have a session creating circuits using a breadboard and components with Dadre Rudolph from North Country Trade Tech High School in Vista, California." Further, Lynn points out, "And we're still going to solder. I just received the shipment of the IPC Education Foundation pin we'll be soldering. It has two leads and two resistors and was designed by Kevin Pintong from Oregon Tech."

Of course, the soldering portion seems to be a student favorite. Nichola, a high school sophomore who attended the STEM Outreach Program in 2019, shared, "The hands-on experience with all of the soldering was useful because if/when I get a job as an engineer, I can use the tips they gave us on how to solder better and make higher-quality products." Diego, a high school freshman, wrote, "The soldering was awesome. I never knew that it would be that easy. I thought that it would be super complicated, but the people there helped me out plenty." Further, Catherine, a high school junior, reported, "I really enjoyed the hands-on experience in soldering and learning more about how different companies solder components on their circuit boards."

Rudolph, who is also a high school teacher volunteer for the STEM Outreach Program, emphasizes, "Students get excited about the hands-on activities and gain new ideas about what they can do with their careers. Many have no idea about what electronics careers are available." She continues, "If students are successful at school and know there are jobs available, then they are more excited about going to University."

I asked Gunter du Plessis how teachers and students respond when reached about this program, and she said, "We've been receiving an overwhelming positive response from students and teachers, especially from the Career and Technical Education (CTE) front with a focus on the STEM and manufacturing tracks. Students enjoy learning from experts on how

to solder, and their faces light up when they complete the hands-on activity. Project-based learning is so important, and our hands-on soldering events have a strong focus on applications that support classroom teaching."

When I asked Rudolph how she became involved with the program, she said, "I have been teaching high school robotics for the last 5.5 years. I was asked how I plan to add more electronics education to my course by IPC. I told them about what I have been doing with circuits and breadboarding and how excited I am about adding more electronics to my course. As a result, I took a course to get an IPC J-standard certification in soldering. I have been working with IPC with the goal of sharing with other teachers and expanding my program for the last 1.5 years."

To that end, IPC has been reaching out beyond the San Diego local area as well. Gunter du Plessis elaborates, "We have expanded our community across the U.S. and established 26 IPC Student Chapters at local universities and community colleges." Gunter du Plessis continues, "We've hosted six hands-on soldering STEM-focused events in Pennsylvania, North Carolina, Alabama, California, and Illinois that engaged nearly 400 students. We've engaged with hundreds of CTE teachers and instructors in over 16 states."

Rudolph adds, "STEM projects take extra time to set up and figure out how to run in a classroom. They take money to buy the materials. And teachers have to learn to let students experiment and be willing not to know the answers all the time. It is a lot of fun, but it is something that takes experience that many teachers don't get a chance to acquire."

There are scholarship opportunities, as well. Gunter du Plessis shares, "We awarded $30,000 in scholarships and impacted 23 students, one educator, and six schools." She adds, "The IPC Education Foundation officially launched at the beginning of 2019 and has made excellent inroads. We have established a solid platform and will continue to create connections between individuals—such as students in high school and college, teachers, instructors, professionals—and IPC industry members."

Dadre Rudolph

For industry insiders who want to get involved, Gunter du Plessis explains, "We are always seeking volunteers to join us during the STEM events, depending on the activity. Let us know if you want to provide supplemental technical instruction and/or guidance in one of the technical learning tracks; engage with students (informally) about your career, job, and/or company to help expand their understanding of careers in electronics; or help coordinate student activities."

When I asked Rudolph about her biggest personal takeaway, she said, "Having students be proud of the projects they complete and be successful in school and gain more confidence in their future."

A final note from IPC Education Foundation is that this event is free for participating students, and they are thankful for the support from their sponsors: TTM, Nordson, Panasonic, Weller, and I-Connect007. **DESIGN007**

Learn More

- IPC Education Foundation Student Chapters
- IPC Education Foundation 2019 Scholarship Award Winners

IPC Government Relations Team Offering Education and Fun at IPC APEX EXPO

Feature by Chris Mitchell
IPC—ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

This year's IPC APEX EXPO is going to be especially interesting for anyone interested in government policies that affect the electronics industry and what IPC is doing to influence them. Your IPC Government Relations team is preparing a variety of activities to educate and engage you on these issues.

IPC places a high priority on government relations because there are so many public policy debates that have huge impacts on our members, from trade and tariffs to environmental regulations, research and development, minerals sourcing, and workforce skills.

Our "GR" team educates government officials on these issues from our members' perspective, advocating for policies that will help our members prosper and grow. We also serve as an information resource for you, answering your questions about what to expect and how to comply with new laws.

Here are the highlights of what IPC will be doing in San Diego from a GR perspective. We hope you will join us.

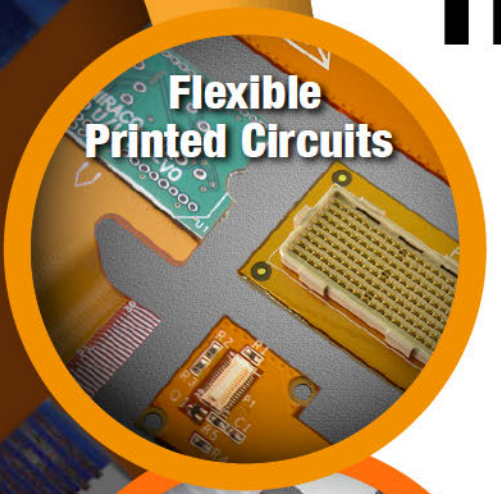
Calls to Action

At the highest level, IPC will be emphasizing several key themes.

Elevating the Industry's Excellence Through Government Policy

Our overall theme this year is "Elevating the Excellence of Electronics," and one of the ways IPC does that is through our government relations work. Because the electronics supply chain is so important to so many other industries and national economies, IPC is calling on governments worldwide to enact policies that enable a thriving, innovative electronics supply chain. Some nations and blocs, led by China and the European Union, are pursuing ambitious industrial policy programs with a view toward dominating 21st-century technologies. Others, led by the United States, are taking a more market-driven approach. While we can debate which approach works best, there is no doubt that all nations have an interest in staying ahead of the megatrends that are shaping tomorrow's economy, including Industry 4.0, automation, cyber threats, 3D printing, and the

MIRACO IS CONNECTING TECHNOLOGY



Since 1987, we have been dedicated to developing innovative high-density interconnect solutions for all electronics industries for our customers worldwide.

MIRACO

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

shortage of technically skilled workers, to name a few.

Workforce Champions

The electronics industry has a responsibility to train the workers of tomorrow, recognizing that many of those job descriptions haven't even been thought of yet. Over the last two years, IPC has doubled down on its longstanding commitment to addressing the skills gaps affecting the electronics industry (see my August column). At IPC APEX EXPO 2020, we will be unveiling new worker credentialing programs that will further drive excellence. We'll also renew our call for government policies and initiatives that more effectively link workforce education programs with job opportunities.

Industry Intelligence

Consistent with IPC's role as the eyes and ears of the electronics industry, we will be announcing several initiatives to expand our research and insight programs, including a landmark study by IPC's new Chief Economist Shawn DuBravac, on the many economic contributions of our industry. Many of the deliverables of our expanded research program will be useful in planning your business strategies and educating policymakers as well.

Opportunities to Engage and Learn

Throughout the show, we will have numerous opportunities to learn from and engage with each other.

For example, the IPC GR team will be participating in many of the industry standards discussions that are relevant to government policies, such as the groups that are developing materials declaration standards, halogen-free materials guidance documents, and the trusted electronic designer, fabricator, and assemblers standards. Please contact me if you have thoughts or questions on any standards-related issue.

Members of the IPC North American Government Relations Committee, who provide essential input for our advocacy work, will gather for their next meeting on February 3. The meeting is by invitation only, but anyone interested in the work of the GR Committee is invited to contact Ken Schramko, IPC senior director of North American government relations, for more details.

The electronics industry has a responsibility to train the workers of tomorrow...

The IPC Environment, Health, and Safety (EHS) Committee will gather on February 5 to discuss EHS policies, research priorities, and upcoming happenings that are applicable to electronics manufacturers. This meeting is also by invitation only, but anyone interested is invited to contact Kelly Scanlon, IPC director of EHS policy and research, for more details.

Speaking of EHS issues, on February 4, Kelly will lead an open session on California's environmental regulations, with expert contributions from Michael Easter, principal of EnSIGHT—a California-based consulting firm—and Carol Monahan Cummings, chief counsel of the California Office of Environmental Health Hazard Assessment (OEHHA).

And it wouldn't be a complete business conference without a networking reception! On February 4 from 4:00–5:00 p.m., all IPC APEX EXPO attendees are invited to join us for a GR-team-sponsored reception titled, "From D.C. to Brussels, Beijing, and Beyond: How IPC Advocates for Your Company and How You Can Get Involved." We'll have an open bar, light snacks, and a few tips and tools for you to make your voice heard by your government officials.

See you in San Diego! **DESIGN007**

Chris Mitchell is IPC's VP of global government affairs. Contact him at ChrisMitchell@ipc.org, and view his columnist page here.

APEX EXPO IPC 2020

Celebrating 20 YEARS OF EXCELLENCE IN ELECTRONICS

MEETINGS & COURSES: **February 1-6**
CONFERENCE & EXHIBITION: **February 4-6**
SAN DIEGO CONVENTION CENTER | CA

ELEVATE THE EXCELLENCE OF ELECTRONICS

Join Us to Celebrate 20 Years of IPC APEX EXPO!

The pursuit of excellence in electronics is year-round. But during IPC APEX EXPO 2020, the focus of the electronics industry will be on how collectively, we can elevate all aspects of our industry and the products we create.

Together, we'll celebrate the 20th Anniversary of IPC APEX EXPO, explore innovative ideas and share our experiences, all with an eye toward a future driven by success.

Plan now to elevate your excellence in San Diego at IPC APEX EXPO 2020.

ipcapexexpo2020.ipc.org | #ipcapexexpo

CFX Preview at IPC APEX EXPO 2020


Chris Jorgensen

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 Editorial Team chats with IPC's Chris Jorgensen about the advances CFX has made in the past year and where the standard is in the standardization process. Chris also previews the CFX line that will be on display at the upcoming IPC APEX EXPO and shares IPC's plans on educating and providing support solutions to potential users of CFX to implement it more easily.

Dan Feinberg: Chris, I knew absolutely nothing about CFX until about 18 months ago. I became involved in it by listening to some talks at IPC APEX EXPO 2018, and I was fascinated by it. And last year's exhibit was quite good; I went upstairs to the demo room and had a chance to look at it. What CFX progress are we going to see at IPC APEX EXPO 2020 this year?

Chris Jorgensen: At last year's IPC APEX EXPO, the IPC-2591 standard was not even published yet. Since that time, the subcommittee approved the first edition in spring and has put the standard on a twice-yearly version update schedule to support CFX for additional equipment or other needs communicated by the industry. The subcommittee chose to have the standard follow a version approach rather than the typical revision letters or amendments of other IPC standards because this is what IT professionals or software solutions providers understand. Once the subcommittee approved IPC-2591, version 1.0, they began work on version 1.1, which is currently under ballot. We expect to have that published in time for IPC APEX EXPO 2020.

Aside from the standard, we have also continued to bring real-world views of CFX to the industry through virtual demonstrations and live factory lines at trade shows in the U.S., Europe, and China. These demonstrations enable the industry to see the power and simplicity of CFX in mixed-vendor factory lines, as well as how CFX can enable Industry 4.0. IPC has also started collecting CFX implementation

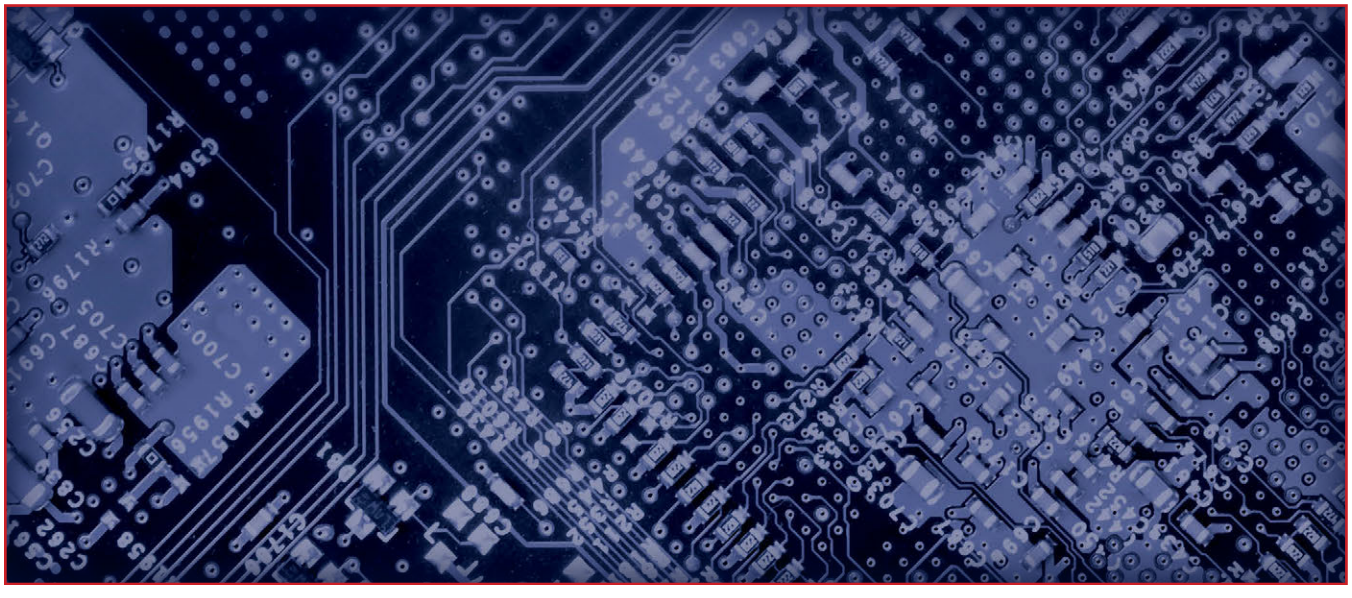
The All New EDA Store

Zero Impedance CAD Purchasing.

Visit the all-new EMA online storefront and shop our collection of CAD products anytime; day or night. Here you will find a broad array of products to help you design faster and more efficiently. Should you need it, our team of experts are here to help answer questions, provide demos, and support you through CAD tool selection.

Best of all, you can rest easy with our upfront, transparent, and hassle-free purchasing process.

Start Shopping Now



EMA Design Automation®

roadmaps from the industry so that EMS and OEM companies can see CFX plans for their equipment and software suppliers. IPC publishes these roadmaps to the CFX website.

In addition to that, IPC also published a joint standard with a Hermes initiative, IPC-HERMES-9852, which is the SMEMA standard replacement. This joint standard is a positive and exciting step in the work IPC and The Hermes Standard Initiative are doing together to support industry. We have been working together on the live factory lines and taking steps to ensure the two standards work together seemed like a very natural next step.

Feinberg: As I understand it, the main thing with Hermes is that it tracks the transfer of each circuit board from machine to machine when you're using CFX. Is that correct?

Jorgensen: Yes. Hermes can work on its own, but it also can work with CFX. If you look at what we demonstrated at IPC APEX EXPO earlier this year, the Hermes part of it was with the line control and the multiple board sizes that we used. We're able to do a transfer between board sizes without human interaction and without having to stop equipment. It tells the machine when a different board size is coming through.

Feinberg: One thing you commented on was that this is software that has been updated a couple of times. That's normal because software is going to be updated much more often than a standard of metal thickness or all the other IPC standards that we have had for decades.

Jorgensen: Right, and we have the software development kit (SDK) that goes with CFX and is available on GitHub as a free download. We also have a way for industry professionals to submit messages that need to be added or comments on existing messages on GitHub. Those comments and suggestions that come in through GitHub form the basis of looking at how the subcommittee wants to make any changes to the standard. Again, some of the changes might be to a message or an update in the SDK that doesn't have any change in the standard, but sometimes it will reflect changes in the standard.

Feinberg: Can they see your comments through GitHub, and then other people that are using the CFX can see them and comment back?

Jorgensen: There is a way to view the comments because they're within the GitHub community. Additionally, any changes that need to be reflected in the standard are tracked through comments submitted through the IPC standards process as well as tracking changes in the standard.

Feinberg: How often do you see the software and these standards being updated?

Jorgensen: The SDK can be updated regularly free of the version updates to the standard, but the subcommittee has indicated that it will publish version upgrades to the standard twice per year. This keeps the flow of messages to the SDK moving and also provides the industry with a regularly updated consensus standard to match the relevant SDK changes.

Feinberg: Looking at the new standard that has been out since last year, what do you think the next major changes will be?

Jorgensen: For version 1.1 that should be published after January 1, the changes are primar-

ily additions to support messages for working with Hermes and many new messages for materials management, test and inspection, and assembly process. To help the industry understand the updates for version 1.1 and future versions, the subcommittee has created an appendix of change updates. This way, when somebody gets version 1.1 of the standard, they will be able to see what changed from one version to the next.

Feinberg: When are you expecting the next standard change?

Jorgensen: I don't want to hold the committee to six months, but the goal is twice a year.

Nolan Johnson: Let's talk about what the update request process looks like.

Jorgensen: The update request process can go one of a couple of ways. One way is the GitHub direction as I was talking about, where somebody posts a message request to the GitHub community. Any message requests or comments that come through GitHub are reviewed by a triage team from within the subcommittee, or what we call an A-Team. The second way is when comments come directly to the subcommittee. With either approach to submitting comments, the committee will review those comments and figure out if and how the standard needs to be changed.

Johnson: Walk us through the line that you'll have at IPC APEX EXPO 2020. What can we expect to see in the line, and what's going to be engaging about it?

Jorgensen: Last year, we had two lines in the sales pavilion, which was disconnected from the show floor. We had feedback from the committee and the line participants that we need to be on the main floor. So, we found a spot on the main floor, and we're going to have one line with 10 pieces of equipment. Like last year, we are going to do live soldering, but to demonstrate the power of Hermes for line control, we will run two-component placement variations on the same process sequence. The committee is also planning to show more about the Industry 4.0 connection for CFX.

Johnson: That suggests that there's a bit of a scope change to the application of the data.

Jorgensen: I'd say it's more of a scope expansion. The focus of the lines that we had at IPC APEX EXPO earlier this year was to demonstrate CFX. The power of CFX is that you have a multi-vendor or a mixed-vendor line and can load CFX across all of them; everything runs through fine. This year, the committee wants to demonstrate not only the equipment messaging and line control but also what any manufacturer can do with CFX in just a few days to take their company to Industry 4.0.

Johnson: There's no argument that the equipment manufacturers are responding to CFX and Hermes and implementing them. How are customers responding to this?

Jorgensen: We are aware of some companies that are implementing CFX or have plans to soon, but we can't share the company names. We expect some of them to make announcements soon. We have heard from several equipment vendors that customers are asking for

CFX, and that's why you see so many equipment vendors that have their roadmaps on the CFX website.

IPC has also recognized that as more EMS companies decide to start implementing CFX on their factory floors, they will need validation that equipment they are buying meets their CFX needs, as well as educational programming for their IT staff to successfully implement CFX in their facilities. IPC is going to provide a service where we'll have third-party validation for equipment, as well as a self-evaluation service for any equipment vendor to test their equipment for CFX before shipping to the customer.

Feinberg: The number of exhibitors at IPC APEX EXPO last year showing support for CFX was impressive. What do you expect this year?

Jorgensen: The factory line was confirmed weeks ago, but we are still gathering volunteers for the virtual CFX demonstrations from exhibitor booths, and we can accept new equipment for the demos pretty much up to the date of the show. Last year, we had a company or two that decided last minute that they wanted to participate, and they were able to get CFX loaded on their machine to be up and running in a few hours. The goal is to have more virtual demonstration participants than what we had the last year because it's good marketing for those equipment vendors, and it's a way to push CFX across the show floor.

Feinberg: If you wanted to get a message through to those who have heard about CFX, but don't understand what it is, or to those who haven't heard about it, what are some of the messages that you'd like our readers to know?

Jorgensen: The three words that always pop to mind for me with CFX are that it's standardized, free, and easy. CFX was developed by the subcommittee for the industry to provide a true plug-and-play Industry 4.0 solution that is based on a standard set of message sets which are available via a free SDK which was developed to allow any IT department to set up a CFX line with no or very minimal hardware requirements. The subcommittee has delivered.

Happy Holden: With the demo line for IPC APEX EXPO, does the product information come over from IPC-2581 seamlessly, or is there an interim step in between?

Jorgensen: That's a good question. For the CFX line for this year, the committee is planning on using IPC-2581 data for the boards and for the assemblies.

Holden: Are there any handouts or data available?

Jorgensen: We're in the early stages of planning. We just had our first meeting with the vendor/line participants. They're all aware that we're using IPC-2581, so there's going to be a guided tour to walk people through each step of the process and show the messaging. I would guarantee that there will be a discussion of IPC-2581 as part of that, but I still think it would be a good idea to have a takeaway for people. We have members of the CFX committee that are also involved with DPMX, IPC-2581, as well as the traceability standard for IPC.

Holden: There is also going to be other software that people will want to tout related to why their equipment should be selected because of how it uses the CFX data for Industry 4.0 for higher profits or better equipment utilization, etc.

Jorgensen: Those are some of the metrics that would be great to have. As we get more utilization of CFX through the EMS community, we hope they will share what they have experienced as well as their improvements in productivity and reliability, reduced costs, lack of downtime, being able to get their fingers on data for their customers for the status of projects, etc. Those are going to be great stories to tell, and they're going to come over time. One of the reasons IPC is working with the industry to develop the three programs I discussed for self-evaluation, validation, and education is to support this. It's important to have these support solutions because the committee and industry put a lot of time and effort into developing CFX, so IPC will provide as much support as necessary for the industry to implement CFX.

Holden: And will someone be available to answer questions on CFX as an expert?

Jorgensen: As part of the solutions IPC will provide to support CFX implementation, we will make available on-site engineering expertise for a fee. Even though there could be a cost for this, our hope is that it will be insignificant or significantly lower than if you were trying to have a mixed vendor line with middleware connecting all of the devices together, etc. Ideally, the company's IT and equipment techs will pick up enough knowledge from the on-demand web-based CFX education that IPC will provide.

For example, an IT person or equipment tech may only need a handful of 30-minute or one-hour web education courses to help guide them through one specific area of utilizing CFX. This alone would be a tremendous cost saving to manufacturers who have to pay time and travel for software vendors and equipment vendors to set up a line or make adjustments to it. The reality is that there will be some cases where a company needs some added assistance for implementation or ongoing support. That's where the on-site engineering assistance would kick in.

Johnson: Is there anything you'd like to add that we haven't talked about yet?

Jorgensen: We covered the revision of the standard coming out, but I would say that also in addition to the CFX Hermes line and the virtual demonstrations, there is a CFX subcommittee meeting on Monday from 8:00 a.m. to 12:00 p.m. at IPC APEX EXPO that's open to anybody to attend. It would be great to have anybody who is interested in learning more or wants to join the subcommittee to drop by because you can get involved right away.

Johnson: Chris, this has been great. Thank you for your time.

Jorgensen: Thanks for the opportunity. **DESIGN007**

PCB Design and HD Semiconductor Packaging

Designers Notebook

by Vern Solberg, CONSULTANT

To better meet their performance and miniaturization goals, manufacturers are looking for higher functionality for their semiconductor packages. For that reason, many manufacturers will rely heavily on more innovative IC package solutions, often integrating a number of already proven functional elements within a single-package outline. This capability has been stimulated by the rapid deployment of new semiconductor packaging methodologies from a broad number of both domestic and offshore companies that understand that new product time-to-market can be the difference between leading and following.

The key enablers for providing interconnect for these new generations of multifunction semiconductor elements is choosing the best package substrate or interposer structure for the specific application. PCB designers will realize that the base material and interconnect metalization processes utilized for the traditional multilayer glass-epoxy component mounting base materials are very different from base materials common to semiconductor fabrication. Furthermore, the design rules for via formation and circuit geometry will have a significant difference. On the other hand, these high-density semiconductor package platforms are essentially miniature printed circuits requiring the same tools and skills developed for PCB design.

The following describes examples of both mature and evolving single-die package variations:

- Single-die ball grid array (BGA) and fine-pitch BGA (FBGA) packaging
- Die size and flip-chip package technologies
- Fan-in wafer-level packaging (FIWLP)
- Fan-out wafer-level packaging (FOWLP)

Single-die BGA and the FBGA package families commonly rely on either traditional face-up wire-bond or facedown flip-chip processing for interconnecting to a package substrate. The substrate is designed to redistribute the die terminals on the top surface of the package substrate to a PCB-compatible array contact pattern on the bottom surface. The die elements mounted onto the top surface will be coated or molded over with a polymer composition, and for board-level assembly, alloy spheres or bumps are furnished on the bottom surface.

Die-size BGA (DSBGA) generally adopts the die facedown approach. The interface between die and interposer may utilize flip-chip processing, but wire-bond and lead-bond processing are more common. A good example of the die-size package using a wire-bond interface is the center-bond memory family of products illustrated in Figure 1.

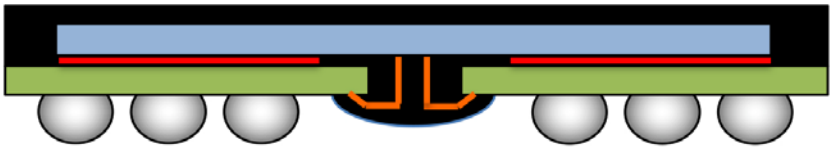


Figure 1: Center wire-bond die-to-substrate interface.

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

VIEW OUR CAPABILITIES

EAGLE electronics Inc.

www.eagle-elec.com

Wafer-level Packaging

WLP technology is commonly utilized to accommodate die elements having very high pin count. Developers have found that by redistributing terminals inside and/or outside the die perimeter, they can accommodate the redistribution on the die terminals to a wider substrate terminal pitch that is more compatible with an organic substrate or PCB circuit routing capability. Silicon wafers are commonly utilized for a broad range of these single- and multiple-die packaging applications, and, although considered an emerging technology, even glass panels are expected to be a viable low-cost alternative to the silicon material. Even though the silicon- and glass-based materials are considered to be an ideal match for HD semiconductor packaging, glass-reinforced B-T epoxy material has been successfully used in manufacturing package substrates for a significant portion of single- and multiple-die packaging markets. Examples of FIWLP and FOWLP are illustrated in Figure 2.

Preparing die elements for direct chip attachment (flip-chip) to the wafer-level base requires several metalization procedures. While the die elements remain in the wafer format, they are subjected to a rather complex sequence of plating and chemical etching processes to form the conductors and terminal lands required for solder bump or sphere attachment.

The first process step employs a sputtering procedure to furnish a barrier metal and enable the deposition of a copper seed layer over the active surface of the wafer. A coating of photoresist is then applied over the wafer surface, imaged and subjected to a secondary copper electroplating process to provide the interconnect pattern. After copper plating to form the interconnect pattern and the barrier and seed layer, metalization is chemically removed, leaving only the copper redistribution layer (RDL) ready for passivation and further imaging processes to expose the copper contact pattern for terminal formation.

Following terminal formation, the die elements are singulated from the wafer format and transferred to systems specifically designed for precise die placement onto the substrate or interposer panel. 2D WLP system-level microcircuits have wide appeal when packaging products require the use of multiple bare die elements from several sources. The semiconductor die elements will be mounted onto the base material facedown (flip-chip) to accommodate reflow solder or alternative joining technologies. Passive elements may also be placed onto the interconnect substrate while in the wafer format.

System-level Packaging

Many companies have realized that integrating mature multiple-die elements into a 2D or 3D configured package proves to be superior to a single, multiple-function die (system-on-chip, or SoC) concept because it minimizes risk and significantly reduces both development time and cost. Although integrating several semiconductor functions onto a single die element may appear to provide a viable solution for companies producing products in very high volume, the cost and time required to develop the multiple function SoC semiconductor have often proved excessive. The challenge the developer faces is selecting the chipset that will meet the designated functional goals and

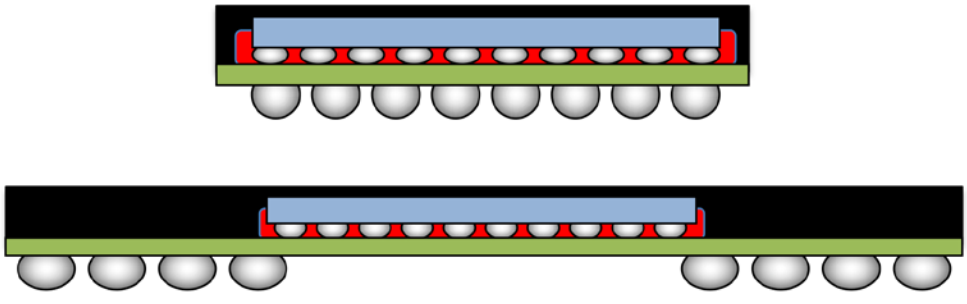


Figure 2: Comparing fan-in and fan-out WLP variations.

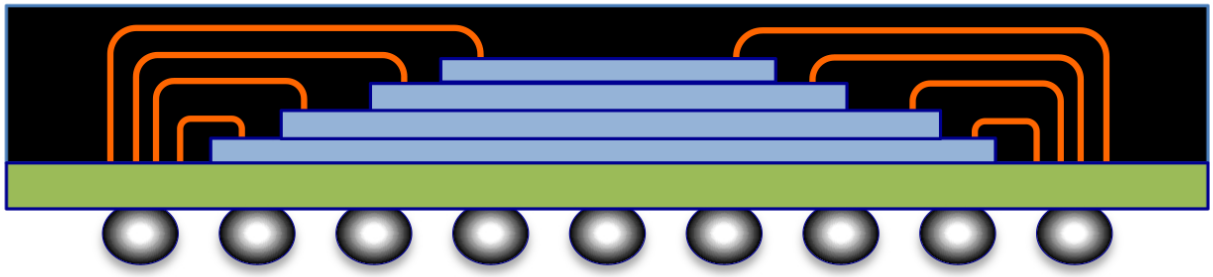


Figure 3: Stacked die packaging.

choosing a package solution that can achieve the performance objectives and cost constraints.

3D die-stacking technologies have evolved as a practical, low-risk solution for a number of homogeneous memory and heterogeneous, system-in-package applications. The vertical integration of proven, high-yielding die elements onto a single substrate will enable shorter development cycles and minimize overall development cost. In regard to reaching product objectives, the short interconnect between stacked-die elements will improve functional performance and minimize power, which is a key issue for portable and handheld products (Figure 3).

3D package-stacking offers practical solutions for pure memory applications as well as mixed-signal and logic and memory applications. Typical applications include the integration of high-density flash, DRAM memory, digital baseband, and processors within a single package outline. Vertically mounting one or more pre-packaged die elements (package-on-package) is preferred by many over die-stack packaging because each level in the stacked package configuration can be pretested before joining (Figure 4).

2.5D represents a viable approach for integrating very high I/O, fine-pitch semiconductors for both single- and multiple-die package applications. A primary challenge to the PCB design professional is dealing with the excessive increase in the package I/O and the shrinking space between terminals.

Current examples include a semiconductor die with a terminal pitch range of 40–60 µm (~0.0016–0.0024") and a terminal geometry as small as 20–30 µm (~0.0008–0.0012"). Although the individual die elements may be furnished with a uniform array terminal format, the terminal size and pitch are often far too small for conventional PCB fabrication capability.

Users have realized that mounting one or more uncased die elements onto a silicon, glass, or TCE-matching organic-based interposer enables higher-density circuit routing and significantly shorter interconnect for critical signal paths. And with a majority of the in-package interconnect accomplished on the interposer's surface, the interface between the component(s) and package substrate can be significantly less complex (Figure 5). This, in turn, allows the contact pitch on the package substrate to increase, simplifying the design

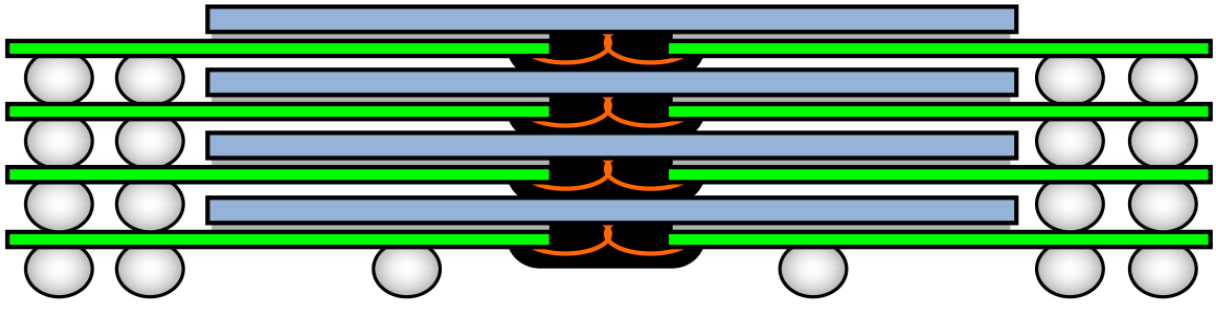


Figure 4: 3D package stacking.

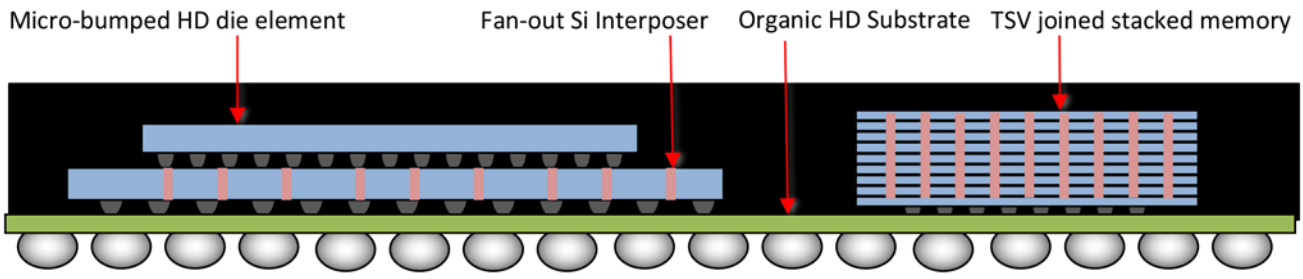


Figure 5: 2.5D interposer-enabled interconnect.

of the host PCB, more efficient circuit routing, fewer circuit layers, and improved power utilization and ground distribution.

Key Planning Issues for 2D, 3D, and 2.5D Packaging

A great deal of progress has been made in semiconductor package process refinement and system development; however, methodologies can vary significantly. Issues that will need to be resolved before initiating package development include:

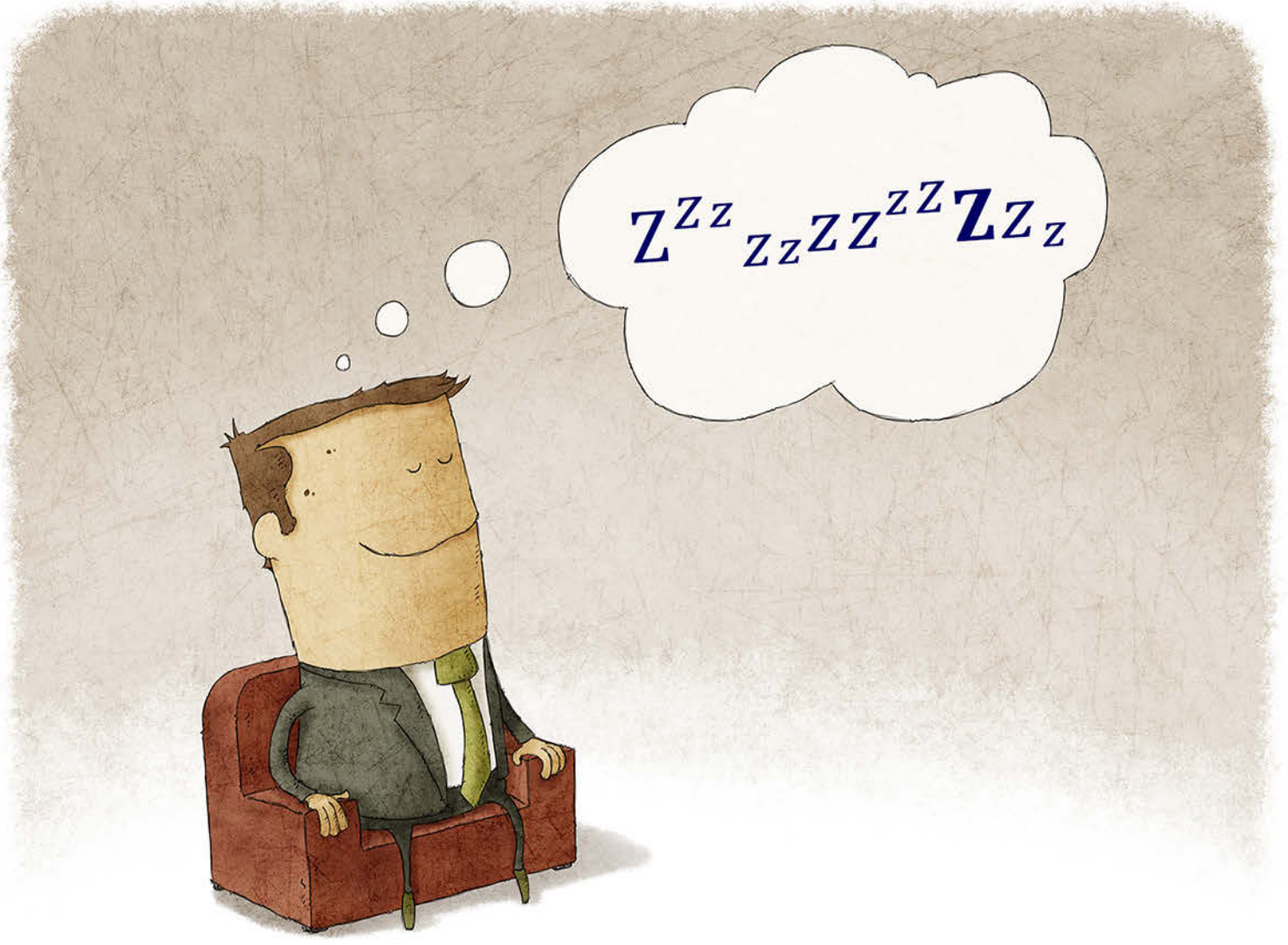
- Selection of suitable semiconductors for multiple-die packaging
- Establishing reliable sources for semiconductor elements
- Specifying physical and environmental operating conditions
- Defining package design constraints and process protocols
- Stipulating electrical test method and post-assembly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

Upcoming Presentation

Vern will be conducting a half-day tutorial workshop on "PCB Designers Guide to Flip-Chip, WLP, FOWLP, and 2D, 2.5D, and 3D Semiconductor Package Technologies" at IPC APEX EXPO 2020 to be held at the San Diego Convention Center on Monday, February 3. This course addresses the design and assembly challenges for developing and implementing flip-chip and multiple function system-in-package (SiP) technology. To register for this timely tutorial workshop program, visit www.ipc.org. 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.

We DREAM Impedance!



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

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

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

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

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

iCD
In-Circuit Design Pty Ltd

www.icd.com.au

A New Beginning

The Digital Layout

by Stephen V. Chavez, MIT, CID+, PRINTED CIRCUIT ENGINEERING ASSOCIATION (PCEA)

In this column, I share a letter from the Legacy Officers and Board Members of the IPC Designers Council, introduce the formation of the Printed Circuit Engineering Association (PCEA), and invite readers to consider future professional development and event opportunities.

Gratitude, Acceptance, and Future Efforts

To: ***The PCB Industry***

From: ***Legacy Officers and Board Members of the IPC Designers Council***

All,

It is with appreciation we are writing this letter to express our gratitude for the opportunity to serve the designers in the electronics industry through our affiliation with the IPC Designers Council. Many of us have contributed a significant amount of our careers to the betterment of our industry with this involvement. We have done so with the attitude of an educator with a servant's heart. Through all the decades of this effort, we have encountered success and challenges. We have no regrets in looking back at our efforts.

We are also writing to convey our acceptance that we have been summarily dissolved effective immediately, at the request of senior management at IPC. We have respectfully ceased all actions in relation to our efforts as an IPC Board. We will make all efforts to continue to represent IPC in a professional and positive manner, as professionals serving in our industry. Many of us continue to serve the IPC community in other capacities and plan to do so with excellence.

With our determination to continue to be involved in the electronics industry, we are informing IPC of the intentions of the Legacy Board Members, along with many others in the engineering/design community, to continue to serve the design professional. We have determined to form a new entity called the Printed Circuit Engineering Association (PCEA). In this new organization, we seek to be strong advocates of


KNIGHTSCOPE
RECORDING
United States
THE
ULTIMATE GOAL
IS TO MAKE THE
UNITED STATES
THE SAFEST
COUNTRY
ON THE PLANET.
Visit the Altium Stories YouTube
channel to watch Knightscope
Altium

IPC and all their efforts to better the electronics industry. We will encourage all we encounter to embrace the IPC standards along with other organizations and standards bodies in the electronics community.

It is our goal and hope to have a professional cooperative relationship. We truly wish IPC continued success and seek to be professional advocates.

Respectfully (alphabetic order),

Stephen Chavez	***Michael Creeden***
Kelly Dack	***Gary Ferrari***
Richard Hartley	***Luke Hausherr***
Cherie Liston	***Bob McCreight***
Scott McCurdy	***Susy Webb***

Professional Development and Events

As we start the new year with excitement and anticipation for up and coming industry events regarding professional development opportunities, this section of the column will continue to list details of opportunities that will be available, such as IPC APEX EXPO, the Del Mar Electronics Show, DesignCon, PCB West, PCB Carolina, Realize LIVE, AltiumLive, etc., and CID and CID+ certification courses. If you have any local or regional industry event coming up in your area and would like to announce it, please feel free to submit the details to be listed in an upcoming column. For 2020 CID and CID+ certification schedules and locations, contact EPTAC to check current dates and availability (dates and locations are subject to change).

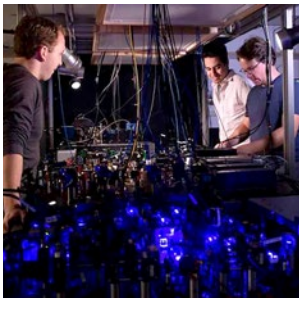
The Printed Circuit Engineering Association (PCEA) is an international network of engineers, designers, and anyone related to printed circuit development. Its mission is to promote printed circuit engineering as a profession and to encourage, facilitate, and promote the exchange of information and integration of new design concepts through communications, seminars, workshops, and professional certification through a network of local and regional PCEA association groups. **DESIGN007**

Stephen Chavez, MIT, CID+, is a Master Instructor of PCB Design for EPTAC, an SME in PCB design for a major aerospace corporation, and is a member of the Printed Circuit Engineering Association (PCEA). To read past columns or contact Chavez, click here.

'Tweezer Clock' May Help Tell Time More Precisely

Atomic clocks are used around the world to precisely tell time. Each "tick" of the clock depends on atomic vibrations and their effects on surrounding electromagnetic fields. Newer atomic clocks that measure optical frequencies of light are even more precise, and may eventually replace the radio-based ones.

Now, researchers at Caltech and the Jet Propulsion Laboratory (JPL), which is managed by Caltech for NASA, have come up with a new design for an optical atomic clock that holds promise to be the most accurate and precise yet (accuracy refers to the ability of the clock to correctly pin down the time, and precision refers to its ability to tell time in fine detail). Nicknamed the "tweezer clock," it employs technology in which so-called laser tweezers are used to manipulate individual atoms.



"This approach bridges two branches of physics–single-atom control techniques and precision measurement," says Ivaylo Madjarov, a Caltech graduate student and lead author of the new study. "We're pioneering a new platform for atomic clocks."

Madjarov explains that, in general, the atoms in atomic clocks act like tuning forks to help stabilize the electromagnetic frequencies, or laser light. The team says that the new system is ideally suited for future research into quantum technologies. The atoms in these systems can become entangled, or globally connected, and this entangled state can further stabilize the clock.

(Source: Caltech)

MilAero007 Highlights

Defense Speak Interpreted: The Continuing Resolution ▸

The topic of the continuing resolution (CR) has been sneaking past other hot Washington topics, such as impeachment, candidate debates, and why the Redskins are so bad. Dennis Fritz provides an update concerning a CR and the 2020 fiscal year.

Lockheed Martin Partners With The Common Mission Project to Support Hacking for Defense (H4D) ▸

Lockheed Martin announced a corporate donation to The Common Mission Project to support the adoption of innovative solutions that solve critical national security challenges through the National Security Innovation Network (NSIN) program Hacking for Defense (H4D), a nationwide academic course.

Designing for Complex PCBs ▸

The I-Connect007 editorial team sat down with Freedom CAD's Scott Miller to talk about the industry's demand for more increasingly complex PCBs, and the challenges this presents. They also discuss Freedom CAD's in-house training programs, the company's recent book authored by Scott, and why communication is such an important tool in a PCB designer's toolbox.

NASA's X-59 Quiet Supersonic Research Aircraft Cleared for Final Assembly ▸

NASA's first large-scale piloted X-plane in more than three decades is cleared for final assembly and integration of its systems following a major project review by senior managers held Thursday at NASA headquarters in Washington.

Lockheed Martin Paints Helicopter for Capt. Kimberly Hampton Memorial ▸

Lockheed Martin completed painting the OH-58D Kiowa Warrior helicopter that will be a part of the memorial honoring Capt. Kimberly Hampton in Easley, South Carolina.

Sikorsky and United Rotorcraft Deliver Three Firehawk Helicopters to CA Firefighters ▸

Sikorsky, a Lockheed Martin company, and United Rotorcraft, a division of Air Methods Corporation, announced deliveries of three new S-70i Firehawk helicopters to California fire agencies: one each to the Department of Forestry and Fire Protection (CAL FIRE), the Los Angeles County Fire Department (LACoFD), and the City of San Diego Fire-Rescue Department.

Czech Republic Acquires Mixed Fleet of AH-1Z and UH-1Y Helicopters ▸

The Czech Republic becomes the first international customer to purchase a mixed fleet of H-1 aircraft. The H-1 mixed fleet shares 85% commonality between parts, reducing the logistics, maintenance, and training costs of the AH-1Z and UY-1Y helicopters while offering a lethal combination of integrated weapons systems.

Vahana Comes to an End: New Chapter at Airbus Begins ▸

Nearly four years after the Vahana concept was sketched on a napkin, the flagship program that launched the urban air mobility initiative at Airbus has come to a close. Vahana's key learnings are now providing Airbus Urban Mobility with invaluable insight on the design of its future urban air vehicle.

Rounding Up a Posse

Tim's Takeaways

by Tim Haag, FIRST PAGE SAGE

The history of the old West is full of stories about people who lived their lives mostly alone. There were immigrants from the East, looking to build a new life in the West, who would travel along primitive trails with all of their earthly belongings packed in a wagon. There were also mountain men who spent months exploring remote wilderness territories; soldiers who were stationed at small, isolated outposts; and lonely homesteads on the prairies. And then there was the cowboy.

By day, cowboys were saddled on their horse, herding cattle through millions of acres of majestic countryside untouched by man, and by night, they slept under the stars alone, except for their horse and a pot of coffee brewing over a campfire. For some, this would be the most ideal setting imaginable: peace, quiet, and solitude. Not for me, though. I'm very happy that I was born and raised amongst the background noise of the suburbs.

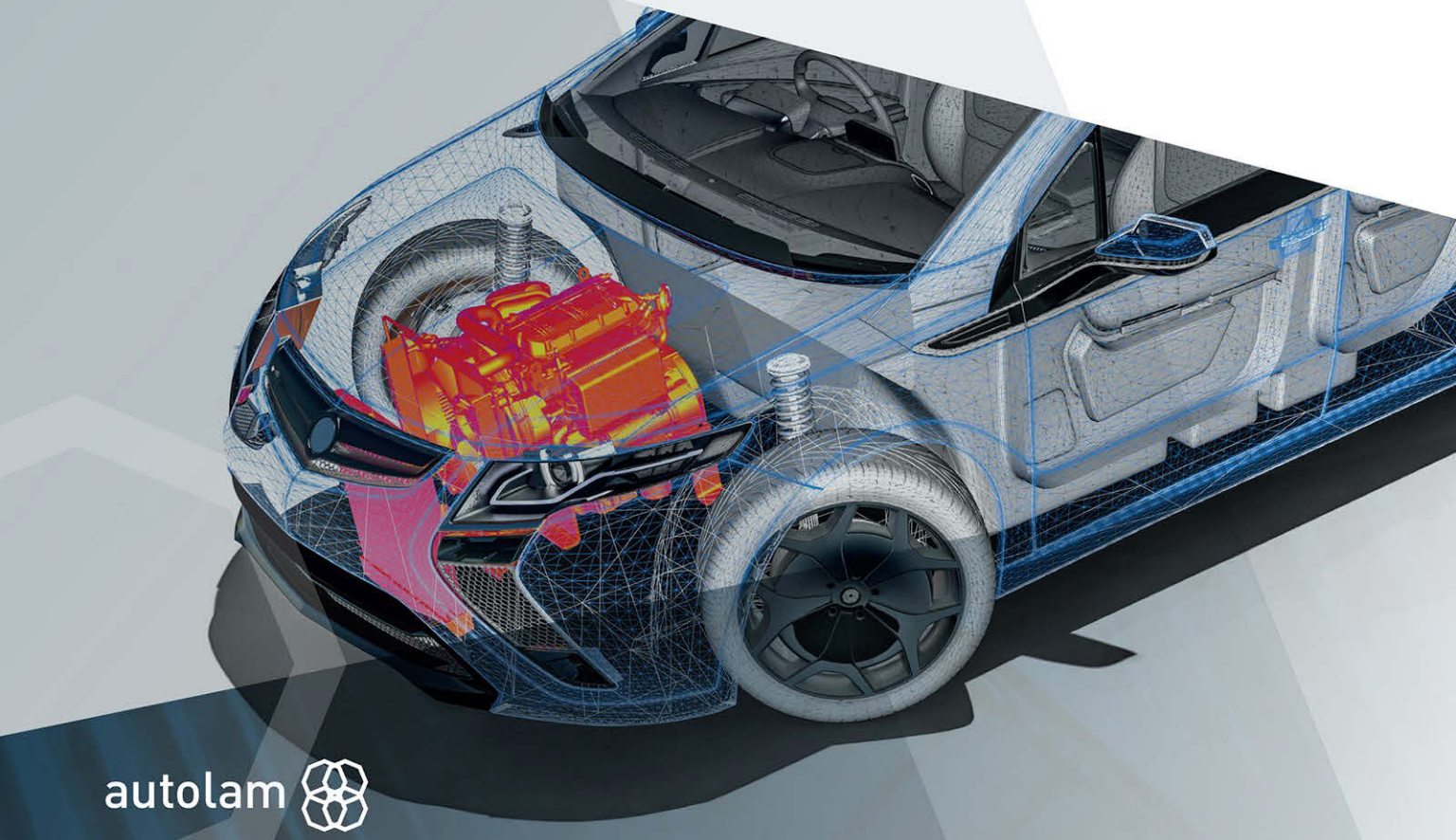
I like that my life is filled with people. Yes, I do understand the romance and intrigue of making it on your own in a pioneering lifestyle. There aren't any interruptions, after all, and there's no one to bother you; you aren't accountable to anyone. On the other hand, there also isn't any help nearby if you need it, there's no one to talk to, and your only companion is loneliness. I have always been happy to leave the Wild West to Marshal Dillon, Curly, and the good folks of the Ponderosa while I stayed where there was plenty of family, friends, and a healthy professional network of co-workers and industry peers.

Unfortunately, life sometimes does imitate art, and for a while, I found myself in a period of isolation, not unlike the solitude of a lonesome cowboy. No, I didn't suddenly decide to head out West in search of the meaning of life. For one thing, I already live out West, and if I were to go much further, I would find myself treading water in the Pacific Ocean.

Instead, my career path abruptly changed, and for a short time, I lost contact with many of my peers in the industry. And not only was

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

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

IATF 16949:2016 CERTIFICATION

venteclaminates.com

Visit us: DESIGNCON® 25TH ANNIVERSARY 2020 WHERE THE CHIP MEETS THE BOARD

APEX EXPO® IPC 2020

I no longer in regular contact with all of these people, but I also was unable to attend the design conferences and other events that I had enjoyed for many years. They say that silence can be deafening, and with the absence of the networking activity in my life, I completely agree.

Fortunately, that period of time went by faster than a six-shooter horse, and I am now back in the saddle again, as they say. However, the relative isolation that I experienced made a huge impression on me, and I came to realize just how important having a network of professional peers to rely on actually is. The truth is that a good peer network is more than important; it is absolutely essential. Here are six benefits that I have come to depend on with my professional network of peers, and perhaps some of these may resonate with you as well.

The truth is that a good peer network is more than important; it is absolutely essential.

1. Help

Let's face it; sometimes, we are going to run into problems where we just don't know the answer. Maybe you need help creating a special shape in your CAD tools that you are unfamiliar with, or you need help calculating a PCB layer stackup. In those moments, you need the help of your co-workers and professional peers to get you back on the right track. I can't even begin to count the number of times that people have helped me when I found myself barkin' at a knot, and I would be willing to bet that each of us relies on help from our networks far more than we realize.

2. Training

There have been several instances in my life where help has turned into full-blown training. I am very thankful for the folks in my life who have spent extra time to patiently take me through whatever process it was that I needed to learn. I've even had people spend time outside of their regular shifts at work to make sure that I completely understood what it was that I was doing. A few years back, as I was preparing to work on a new schematic capture tool, one of my peers made a special trip to my house to drop off his own training materials on that tool to help me.

3. Professional

There are also many aspects of our careers that aren't specifically PCB design related, and your network of peers can be of great help with that as well. I've had peers help me with financial questions, writing resumes, and learning the ropes at a new company. I've even had times where I've whined and complained about work situations that I couldn't clearly see through, and a valued coworker was able to help me view the situation from a different perspective. Having someone to bounce ideas off of that you can trust is a treasure that can't be bought, and must be protected at all costs.

4. Career

When you are looking for a job, having a large and diverse network of professionals that you can reach out to is worth more than gold. Time and time again, I've had referrals from my network of professionals that have opened up opportunities that I would never have known about otherwise.

5. Social

Industry trade shows are not only great places for seeing and learning new design advancements and methods, but they also provide a wonderful venue to engage with your peers on a social level. I absolutely love my family, friends, and neighbors, but when I start a discussion with them about microvias, I can see their eyes glaze over right before they collapse on the floor in a deep coma.

With trade shows, however, you have a unique opportunity to spend time with people who share the same interests as you, including microvias. Additionally, the times away from

the technical sessions at those shows can be more valuable than you realize. You will meet new people who will become part of your network of peers and be able to relax with old friends. And if you're really lucky, you'll wind up in a hotel lobby where you can listen to Andy Shaughnessy and friends play a little music. That truly was a mighty fine shindig.

6. Support

Not only do your peers understand what you do, but they also understand how you are motivated by your career and how that contributes to who you are. This puts them into the position of being able to help and support you in ways that others may not fully understand. I'm not suggesting for a moment that your network of peers should replace your family or friends—far from it. But I do believe that your peers provide another important part of your life that you need in order to be content and successful.

Connect and Pay It Forward

There's a lot more to networking than what you receive from your peers, however. To bring it all full circle and fully leverage the power of your professional network, you need to give out just as much in return. In other words, pay it forward. You may not realize it, but there are many others out there that could use your help and support just as much as you need theirs. And what is truly great is that when helping others with their questions and problems that, in turn, you end up helping yourself. There have been many times while helping a co-worker with a problem that I have ended up accidentally finding a solution for one of my own problems, and I wasn't even looking for it. So, don't be afraid to reach out and lend a hand when you can; it can be well worth the effort.

Now, it is possible that you are in a position of being isolated yourself. Maybe you are between jobs, work in a small company, or have co-workers who aren't very sociable. Fortunately, you have a lot of options. You can connect with others in your profession through social media, or look up people that you went through school or training with. You can also find PCB design specific groups such as an IPC local IPC Designer's Council Chapter. There are also other professional groups that meet via online sessions and industry trade shows. And if none of those avenues are available to you, send me a message. I can't promise that I will be able to help you with any specific problems, but I would be happy to at least say "howdy" in return.

Conclusion

Most of us are not designed for going through life in isolation, and I know that very well from my own experience. The good news is that there are lots of others out there in the PCB design community that we can connect with to build our own professional network of peers.

The good news is that there are lots of others out there in the PCB design community that we can connect with to build our own professional network of peers.

If you haven't done so already, reach out to your fellow designers. A solid network of professional peers that you can rely on will serve as a firm foundation to help you keep your footing steady under you in this industry. And with that, I suddenly have a hankerin' to find an old Western on TV to watch, so I'm going to skedaddle. Yee-haw, everyone, and 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.

Five Top Resin Tips

Sensible Design

by Alistair Little, ELECTROLUBE

For this month's column, I'm going to provide some useful advice on encapsulation resins to help designers achieve even higher levels of reliability when it comes to protecting circuitry. Potting compounds are increasingly important in the electronics industry where they serve to protect sensitive components from moisture, dust, and damage, and help to extend the lifetime of devices. Appreciating the subtle characteristics of resin systems will go a long way in helping you choose the most suitable resin for your application requirements, making the process a lot less complicated and, more importantly, helping you avoid any serious implications for the performance of your application. Let's take a closer look at five critical factors affecting encapsulation resins, which will include potential contaminants, no-clean/cleaning processes, adhesion, and best mixing practice.

1. What potential harm can contaminants wreak if present on a PCB? Can this cause long term problems ultimately resulting in board failure?

There are a number of different contaminants that can cause long-term problems for a PCB, ranging from dust and grease to flux residues. Since the resin will stick to the contaminant rather than the board substrate, this produces weak points within the PCB-resin interface, which, over time, can lead to cracking of the resin or adhesion failure between the resin and PCB. Once a fault appears, it can grow in size to allow other contaminants to attack the

Candor
Circuit Boards

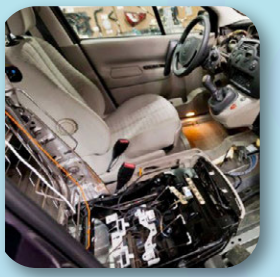
Innovative solutions for complex circuit board manufacturing

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

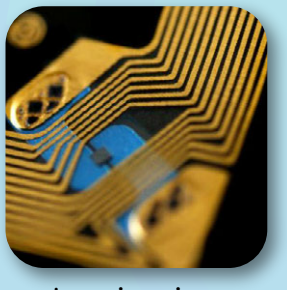
RF/Microwave PCBs

Power Electronics

High-Speed Data



Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

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

PCB and components under the resin leading to failure. These contaminants can be chemicals, moisture, and/or corrosive atmospheres. In the case of silicone resins, contaminants can prevent the resin from curing properly, as the catalysts used in silicone resins are very sensitive.

2. Why do some applications involve a no-clean process? Is this common within production cycles?

The no-clean process is very common for medium- to high-volume PCB manufacturing, as it takes out the cleaning stages and hence saves time and money. However, it should be noted that no-clean solder and solder pastes usually mean that there is a low amount of flux in the formulation that is left behind. Paradoxically, the flux residues left behind are harder to remove than the traditional systems and adhere well to the PCB substrate.

3. What are the potential causes and consequences of a lack of adhesion between the resin/PCB/outer casing?

Encapsulation resins are designed to have excellent adhesion to a range of substrates varying from FR-4 laminate, copper and tinned tracks, the plastics used for components, and the wide range of plastics and metals used for the enclosures/housings. The biggest cause of poor adhesion between a resin and the substrate is dust and/or grease.

To gain optimal adhesion, the surface of the substrate should be cleaned to remove any loose debris and to degrease the surface. In the case of plastic housings, if they are extruded mouldings, the plastic might have a layer of mould release agent on the surface which needs to be removed. If the surface is highly polished, then light abrasion of the surface is often sufficient to improve the adhesion. There are certain plastics that are very hard to get resins to adhere to, including polypropylene, in particular. It is possible to get surface-treated materials or use a silane or plasma pretreatment, which can help in these cases.

4. How important is the mixing and dispensing process in the performance of an encapsulation resin?

With all two-component systems, mixing is absolutely critical to ensure that the resultant resin performs as described in the TDS. The aim of mixing is to obtain a homogeneous mix with parts A and B distributed uniformly. Incomplete or poor mixing will result at best in a poor performing resin, and, at worst, an uncured mixture. Selecting the correct sized static mixer for the machine and resin will enable the resin to be mixed consistently with each shot of resin dispensed.

Next, the mixed resin has to be placed in the right location to allow it to flow across the PCB, displacing air from any voids and to coat the components to the required thickness. Air trapped in the resin is another source of potential weakness within the resin as this can lead to a physical weakness due to thermal cycling, physical shock, chemical ingress, or in the case of high voltages, act as a concentration point where corona can build up.

5. What are your top tips for the manual mixing process of a resin pack?

Firstly, select the right size resin pack for the job. If a lot of small volume units are to be potted, it might be easier to use a couple of small resin packs than a single large one to allow more time to dispense the material accurately, and the useable life/pot life of a small volume of resin is usually longer than with a large volume.

In the case of polyurethane and silicone resins, make sure that the foil pack is sealed before use. Do not open the outer packs until ready to mix and dispense. Remove the resin pack from the outer packaging and remove the middle clip separating the two halves of the pack. Use the clip to push the resin from one half of the pack into the other. Take the resin pack between two hands and mix vigorously for a couple of minutes.

Lay the pack on a flat surface and, using the clip, push the material into the centre of the

pack. Pay particular attention to the corners. Continue mixing vigorously for another couple of minutes.

On a flat surface, use the clip to push the mixed material into the half of the pack with the angled seal. Roll up the other end of the pack to the bottom edge of the mixed resin.

Cut the corner of the pack off and slowly decant the resin over the PCB or into the enclosure. Allow the resin to flow so that any air under or between components or wiring can be displaced. For complicated geometries, it might be better to add the required amount of resin in multiple stages to allow the air to be released, control the exotherm experienced by the components, and to obtain the desired thickness. If adding a second layer of resin, wait until the first layer has gelled before adding the second.

If in any doubt, it is always advisable to discuss with suppliers which type of resin system is most appropriate for your application. The technical support teams of reputable suppliers have a wealth of experience to call upon, and should it become necessary, they have the expertise to modify chemical formulations to meet your particular application needs.

I hope the points covered this month have been informative, and please look out for my next column, where I'll cover more issues on getting the most out of encapsulation resins for seamless circuit protection. In the meantime, please contact us if you have any questions you would like to ask. **DESIGN007**

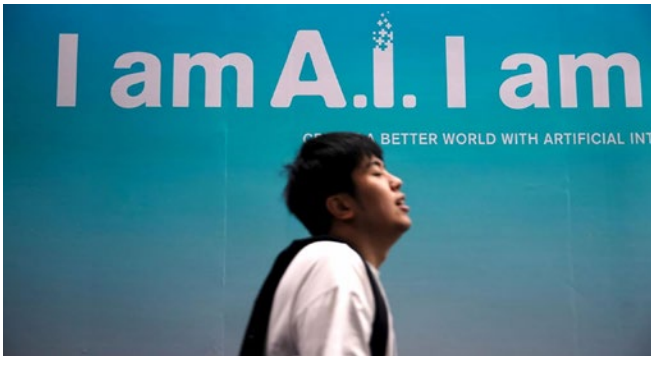
Alistair Little is global business/technical director–resins–at Electrolube. To read past columns from Electrolube, click here. Also, visit I-007eBooks.com to download your copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*, as well as other free, educational titles.

White House Proposes Regulatory Principles to Govern AI Use

The White House proposed regulatory principles to govern the development and use of artificial intelligence (AI) aimed at limiting authorities' "overreach" and said it wants European officials to likewise avoid aggressive approaches.

In a fact sheet, the White House said federal agencies should "conduct risk assessment and cost-benefit analyses prior to any regulatory action on AI, with a focus on establishing flexible frameworks rather than one-size-fits-all regulation."

The Trump administration said agencies should "promote trustworthy AI" and "must consider fairness, non-discrimination, openness, transparency, safety, and security." As an example, the White House cited the Food and Drug Administration, which is currently considering how to regulate the use of AI and machine learning technologies by medical device manufacturers.



Some U.S. states have raised concerns about AI applications. California's legislature in September passed a three-year ban on state and local law enforcement using body cameras with facial-recognition software, the latest curb on technology that some say poses a threat to civil liberties. Some U.S. cities have also voted to bar facial-recognition technology by law enforcement.

A 2018 study from consultancy PwC said 30% of jobs around the world are at risk of automation by the mid-2030s, including 44% of workers with low education. The study also found automation could boost global gross domestic product by $15 trillion by 2030.

The White House held a meeting on AI in 2018 with over 30 major companies from a variety of industries, including Ford Motor Co., Boeing, Amazon, and Microsoft, vowing not to stand in the way of the technology's development.

(Source: Reuters)

FLEX007
A SPECIAL DESIGN007 MAGAZINE SECTION

Flex Segment Starts Strong in 2020

What the Flex?

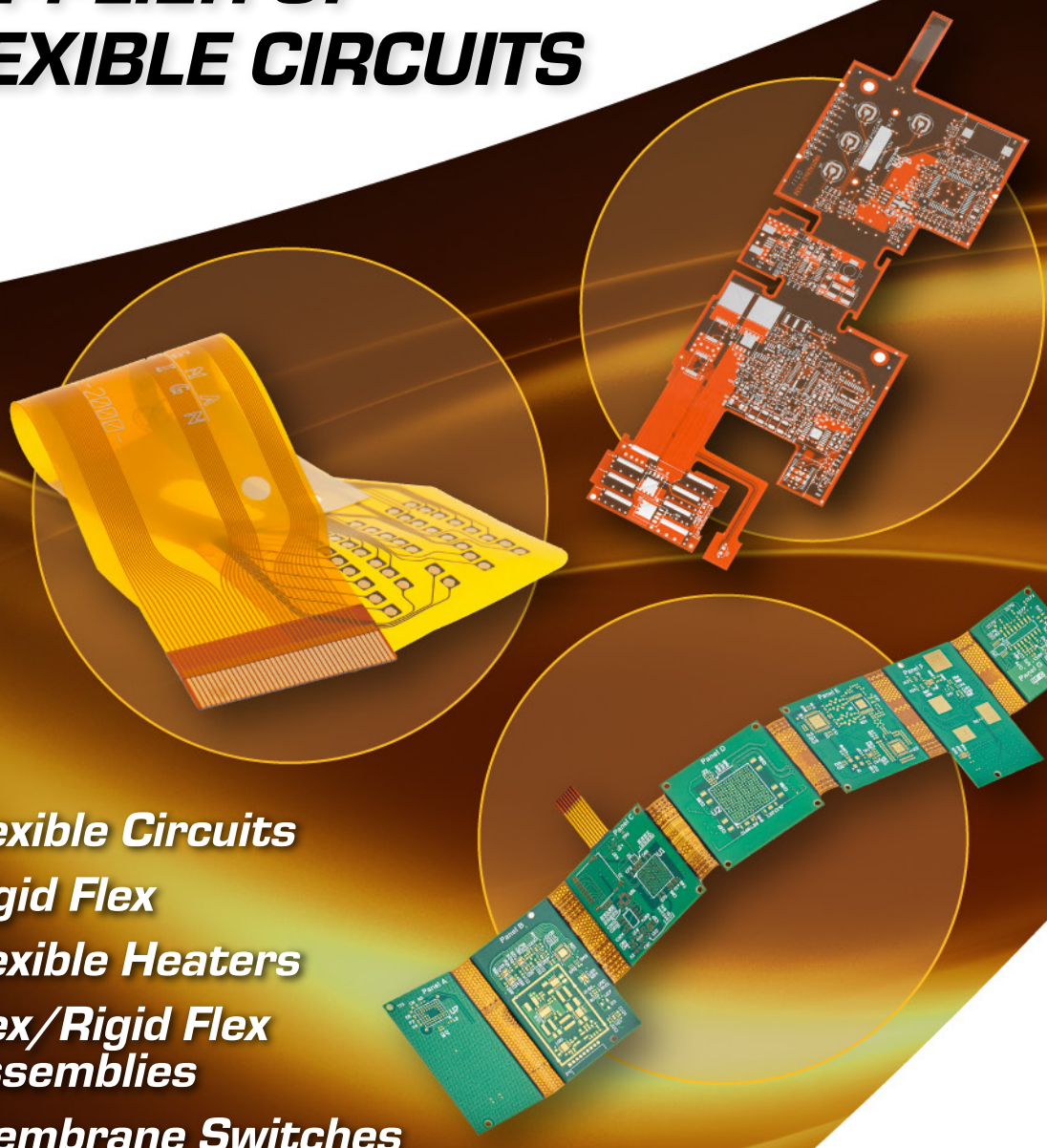
by Andy Shaughnessy, I-CONNECT007

I hope you all had a great holiday season and a happy new year. Can you believe it's 2020 already?

The flex market is looking good as we move into the new year. Flex is still the most dynamic segment in our industry, and it's expected to remain strong in 2020 and beyond. The flex market isn't bulletproof, but it's spread out across a variety of industries and end-products, so it's not likely to take a major hit if one sector tanks.

Many analysts were worried that the slowdown in the automotive sector—the first since the 2008 downturn—would lead to a drop in demand for automotive flex and rigid-flex. But flex keeps finding its way into new vehicles with automakers adding bells and whistles, such as advanced safety and lighting features, that require flex. Even if fewer new cars are sold this year, I imagine they will probably utilize more flex and rigid-flex than last year. In the meantime, other segments—medical,

PREMIER GLOBAL SUPPLIER OF FLEXIBLE CIRCUITS



- Flexible Circuits
- Rigid Flex
- Flexible Heaters
- Flex/Rigid Flex Assemblies
- Membrane Switches
- Product Module to Complete Box Builds

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

FLEXIBLE CIRCUIT® Technologies

WE GO WHERE OTHERS WILL NOT

www.flexiblecircuit.com

wearables, industrial, and consumer/handheld devices—are exploding with new products that require flexible circuits.

I think flex brings out the creativity in OEM design teams in a way that a standard PCB just can't. Flex is almost fun compared to rigid boards. As the miniaturization trend continues, more and more companies are finding themselves forced into using flex and rigid-flex because rigid boards just aren't reliable enough for their products.

Flexible circuits are also changing much faster than rigid boards. I've said before that the flex community reminds me of the Wild West because it's constantly evolving. It can be difficult to keep up with all of the changes in materials, chemicals, and processes.

One of the best ways to stay on top of these changes is by getting involved with some of the IPC flex standards committees. If you're attending IPC APEX EXPO 2020, join one of the flex standards committees. There are four subcommittee meetings during the show, and they'd love to have your input. If you work with flex and rigid-flex circuits, you can help shape the standards that technologists around the world rely upon, and you'll stay ahead of the curve.

This month, we bring you an article from Flexible Circuit Technologies that is based on their flex design guidelines with tips and tricks for designing single-layer, double-layer, and multilayer flex and rigid-flex, including advice on cutting costs. We also have a column from Joe Fjelstad of Verdant Electronics that focuses on the effects that unpredictable events have had on shaping our technology and some of the newest innovations coming that are just starting to ramp up.

I'll be heading to DesignCon and IPC APEX EXPO later this month, and I hope to see you there. If you can't make it, don't worry; we'll bring you front-to-back coverage of these events. See you next month! **FLEX007**

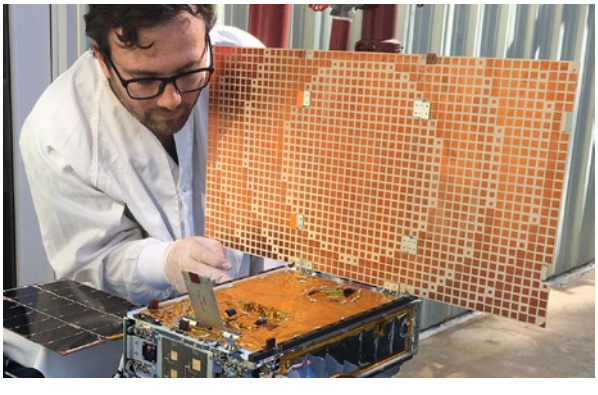
Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 19 years. He can be reached by clicking here.

NASA's Briefcase-size MarCO Satellite Picks Up Honors

Aviation Week & Space Technology is bestowing a prestigious Laureate award on NASA's pair of briefcase-size Mars Cube One spacecraft. Known as MarCO, they're the first CubeSats–compact spacecraft made up of cube-shaped units–to travel into deep space.

Designed and built at NASA's Jet Propulsion Laboratory in Pasadena, California, as a technology demonstration, MarCO launched to the Red Planet last year with NASA's InSight lander. Using experimental radios and antennas, the pair relayed signals back to Earth that enabled InSight's team to observe the spacecraft's November 26, 2018, entry, descent, and landing on Mars in near real-time.

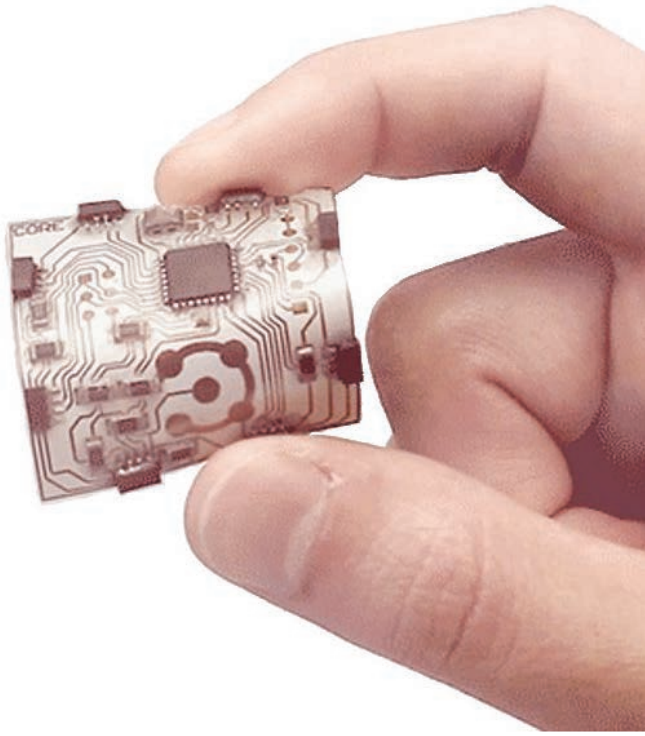
They also sent back stunning images of Mars and performed some simple radio science, transmitting signals through the edge of Mars' atmosphere, all within a low-cost mission. "It's gratifying to see MarCO honored in this way. The accomplishments of the pair highlight the tremendous potential of small spacecraft to enhance our exploration efforts," said JPL Director Michael Watkins.



This isn't the only honor for MarCO: It was named Small Satellite Mission of the Year in August by the American Institute of Aeronautics and Astronautics. "This is the MarCO team's second major aerospace award," said John Baker, JPL's program manager for small spacecraft. "It's an honor to be recognized in our field. But the real pleasure has been seeing how the team's achievements are inspiring future space projects around the world."

(Source: NASA JPL)

DUPONT®



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

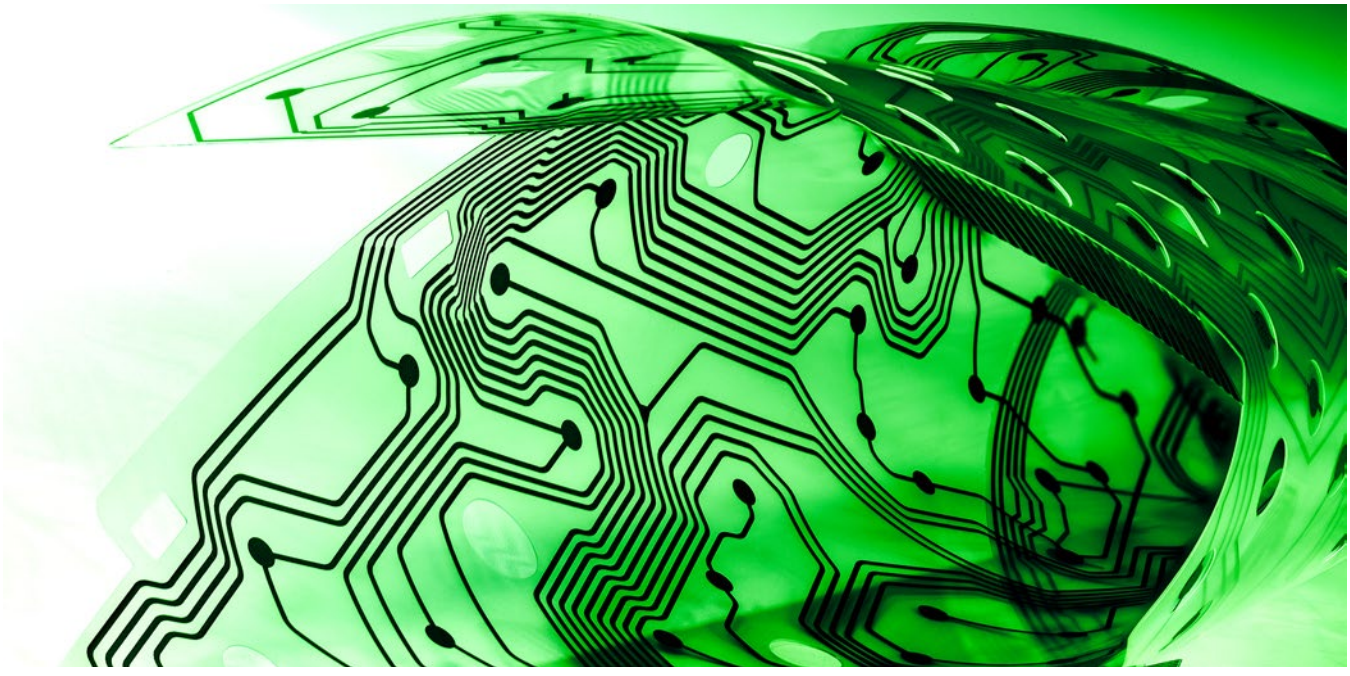
DuPont offers a comprehensive, industry-leading portfolio of Pyralux® flexible circuit materials and systems designed to meet today's and next generation design challenges.

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



Flexible Circuit Design Guidelines

Feature by Flexible Circuit Technologies

Flexible circuit designs share many of the same challenges of rigid PCB designs, but there are also many differences and additional challenges. The very nature of a flex circuit being able to bend and flex make it as much a mechanical device as an electrical one. This creates a special set of requirements unique to flexible circuitry. Understanding how these requirements interact will allow the PCB designer to create a flex circuit that balances the electrical and mechanical features into a reliable, cost-effective interconnect solution.

Overview

Scrutinize your design for stress concentration features. Stress concentration features are the single predominant cause for mechanical failures in flexible circuits (i.e., cracked/broken conductors, torn insulating material, etc.). To avoid stress concentration points, the construction of the circuit should not change in, or immediately adjacent to, the bend area. In a bend area, there should be no change of conductor width or thickness or direction, no termination of plating or coatings, no openings in covers or outer insulating materials, and no holes of any kind in a bend zone.

Bend Ratio

Determine and evaluate the minimum bend ratio of your design. This will be your single best indicator of whether your flex circuit may experience problems in service. Bend ratio is bend radius: circuit thickness (Table 1).

10:1	Single-layer
10:1	Double-layer
20:1	Multiple layers

Table 1: Preferred bend ratios.

Conductor Routing

When possible, conductors should be routed through bending or flexing areas with the conductors perpendicular to the bend (Figure 1). This will minimize stress on the conductors during flexing and maximize circuit life. Conductors should always change directions with soft curves rather than sharp corners. When

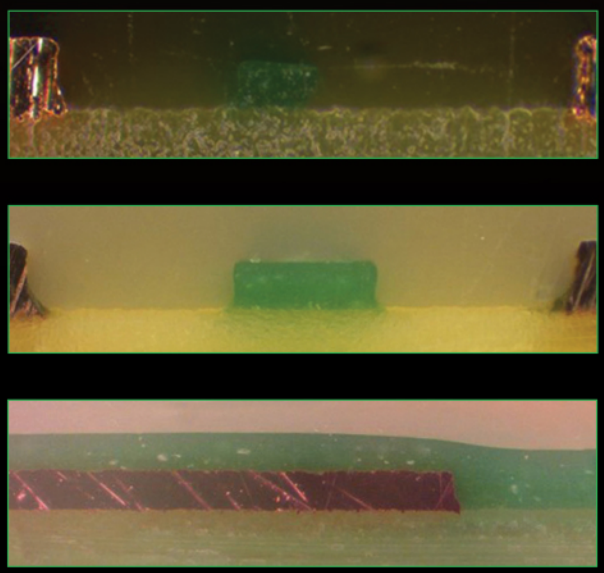
FLEXIBILITY NEVER LOOKED SO GOOD

FLEX SOLDER MASK

BY TAIYO

PSR-9000 FLX03G LDI

- Designed specifically for Orbotech LDI equipment
- Dark green glossy finish
- Fast exposure time
- UL listed VTM-0 on 2 mil polyimide
- Straight sidewalls



TAIYO AMERICA

UNMATCHED INNOVATION

Phone [775] 885-9959 • www.taiyo-america.com • info@taiyo-america.com

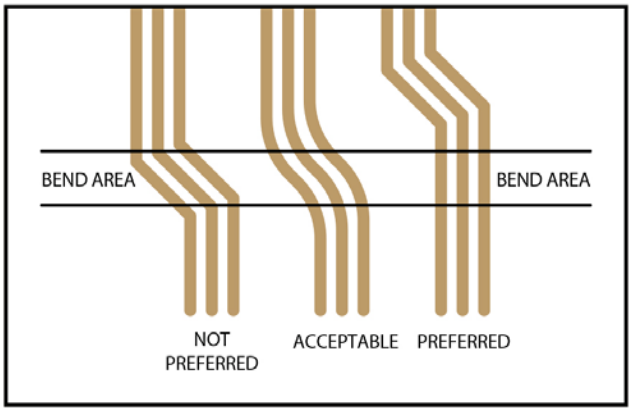


Figure 1: Conductors should be routed through bending or flexing areas when possible.

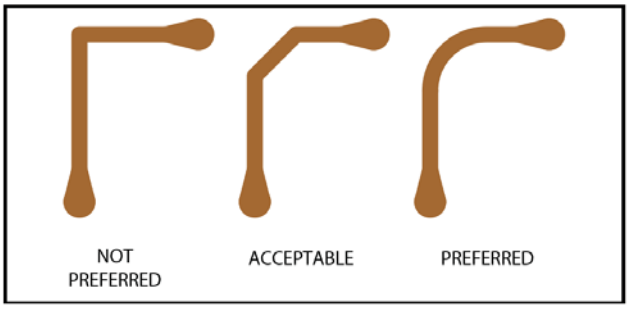


Figure 2: When soft curves are not possible, corners of 2-45° are better than corners of 1-90°.

curves are not an option, two 45° corners are preferred over one 90° corners (Figure 2).

When possible, route small conductors on the inside of a tight bend. Small conductors (<0.007") will tolerate compression better than stretching. Placing these conductors on the inside of a bend will reduce or eliminate tension forces. Do not stack conductors on top of each other on multiple layers creating an I-beam effect. Stacking conductors will essentially increase the overall circuit thickness, thereby decreasing flexibility and the circuit's ability to bend reliably.

Conductors

Flexible circuit conductors are manufactured using a photo-etch process, which starts with a full sheet of copper. Conductors are formed by masking the desired conductive paths, and then chemically removing all unwanted copper, leaving the desired circuit patterns. As the etchant dissolves the unmasked copper, it also

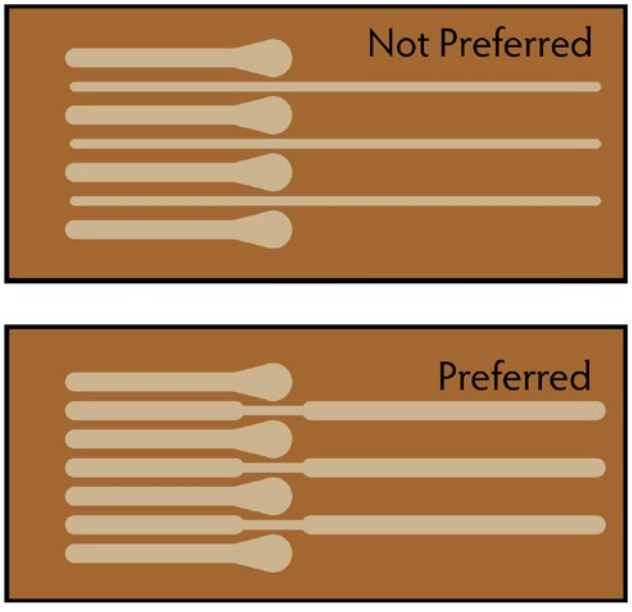


Figure 3: If a conductor must shrink to fit between pads in an isolated area, the conductor should flare back out after clearing the constricted area.

attacks the edges of the conductors, resulting in what is referred to as "under-cut."

As copper foil thickness increases, so does the amount of undercut. This makes it very difficult for the flex manufacturer to create very small conductors on very thick foil. There are also variations in the etching process (primarily etchant strength, which varies with the amount of copper in the solution). For this reason, the designer must factor in a processing allowance for strand width (and spacing). For optimum etch yields, conductor widths should be at least 5X greater than the thickness.

It is advisable to maximize conductor width wherever possible. For example, if your design requires 0.005" conductor width to squeeze between pads in an isolated area, the conductor should flare back out to 0.010–0.012" once the conductor clears the tight area (Figure 3). This will improve the manufacturing etch yields, which in turn means a lower overall circuit cost to you.

Pad Fillets

It is a good idea to insert fillets on pads at each location where a conductor enters a pad. Pad fillets will reduce or eliminate potential stress concentration points.

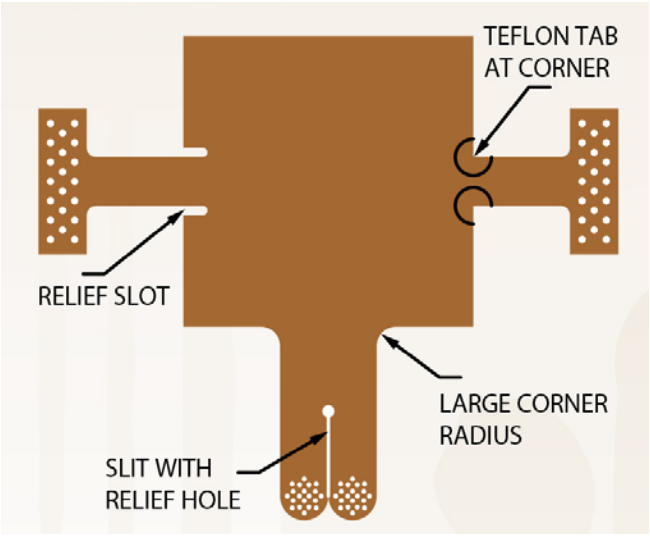


Figure 4: A proper tear-relief design option.

Tear Relief

Figure 4 shows the most common and effective method of eliminating tears in a flexible circuit. Copper tear stops are not advised; they have been shown to have limited value in keeping a tear from starting or propagating.

Vias

Through-holes can connect all layers at a via point. Blind vias connect outer layers to adjacent layers but do not extend through the circuit. Buried vias connect internal layers; however, they do not extend to the outer layers. Blind and buried vias will increase the cost of the circuit but can increase usable PCB real estate on non-drilled layers.

SMT Access Openings

The two most common cover materials are polyimide film and flexible solder mask. The methods for creating access openings in the two materials are very different and carry different design requirements. Access openings in polyimide film are created by drilling, routing, or punching, which limits the size and shape of the openings to what can be done with a round bit or a tool. For this reason, SMT access openings in polyimide film are either round or oval. Also, gang access of multiple SMT pads is a common design practice on flex circuits.

Flexible solder mask, like regular PCB solder mask, is photo-defined, so any shape opening is possible. Solder mask openings should be made slightly larger than the SMT pads to ensure that the mask does not get on the pads if there is any misregistration in the printing process.

Controlled Impedance and Signal Integrity

The speed at which electronic devices are operating is continually increasing. The result is that the characteristic impedance of all parts of the electronic assembly, including any flex or rigid PCBs in the system, need to have matching impedance. Impedance mismatches will cause signal reflections and degradation at each mismatch point, which results in erroneous signals and, ultimately, device malfunction.

The characteristic impedance of flex can be determined before manufacturing using an impedance calculator. Your flex fabricator can assist you with these calculations, or you can buy or download an impedance calculator. A number of factors will affect the characteristic impedance of a flex PCB. The main contributors are:

- The dielectric constant of the insulation materials used to construct the circuit

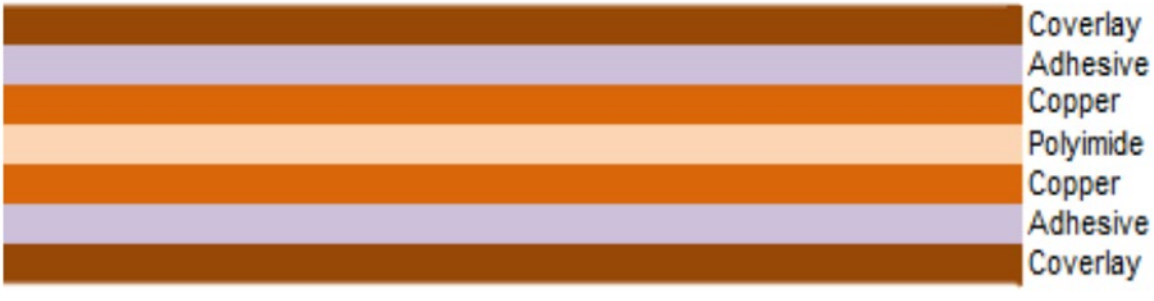


Illustration of a standard two-layer circuit construction.

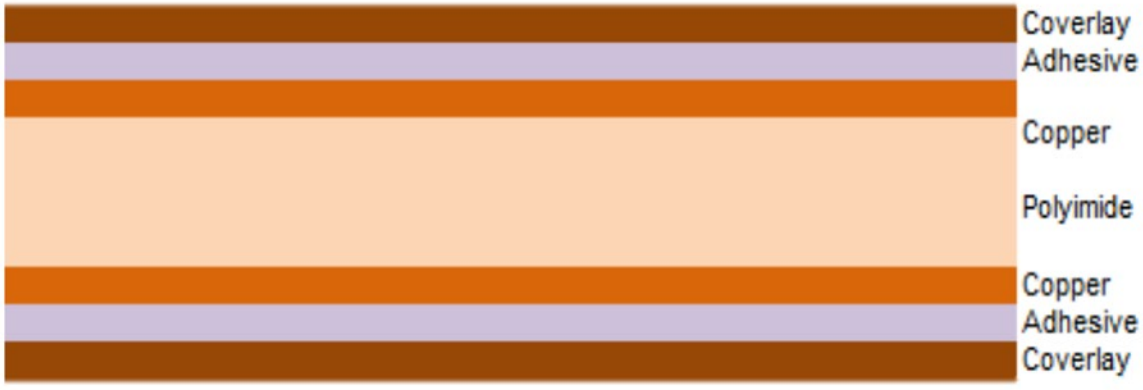


Figure 5: Illustration showing a two-layer construction with higher impedance requirements. The added thickness of the controlled impedance section makes circuits less flexible.

- The width of the traces carrying the signal
- The distance of the signal traces from the reference plane layer(s)
- The thickness of the traces carrying the signals
- The distance between signal traces in differential impedance applications

The most common impedance requirements range from 50–75 ohms (single-ended) or 100–110 ohms differential. Achieving these impedance values in flex circuitry requires the use of thicker dielectric materials than are normally used, resulting in an overall thicker and stiffer circuit (Figure 5).

Plane Layers and Shielding

Reference plane layers and external shielding play a key role in both impedance control and signal integrity. Fabricators can add plane layers using:

- Additional etched copper layers
- Screened conductive epoxies or inks
- Laminated conductive films

Copper plane layers are the standard for internal planes that require a connection through plated vias. Copper planes will cause a flex to hold a pre-form better, while screened epoxies and inks and laminated conductive films will produce a more flexible circuit.

Stiffeners

It is wise to rigidize SMT, connector, and other termination areas on your flex circuit with mechanical stiffeners. Your flex fabricator can add stiffeners of various thickness made from epoxy-glass laminate (FR-4) or polyimide film. In SMT applications, stiffeners should be applied to the side opposite the SMT components. On through-hole connectors and other through-hole applications, stiffeners should be applied to the same side as the connector or through-hole component. Stiffeners applied to connector areas will require holes that match the connector footprint. Holes in the stiffener should be sized at least 0.015" larger than the access hole in the circuit.

Thermal Pads

Thermal pads should be used on any solder pad that is surrounded by a large amount of copper. Large areas of copper will sink heat away from a non-thermal pad and make it very difficult to solder.

Rigid-flex Design Guidelines

Since rigid-flex circuits are a hybrid of rigid and flexible PCBs, there are special guidelines that apply to this type of construction (Figure 6).

- On rigid-flex circuits, ensure that all plated through-holes are in a rigid area (no PTHs in flex areas)
- Specify adhesiveless flex materials and "cut-back" or "bikini" cover construction for rigid-flex designs. Acrylic adhesive is the "Achilles heel" of a plated through-

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

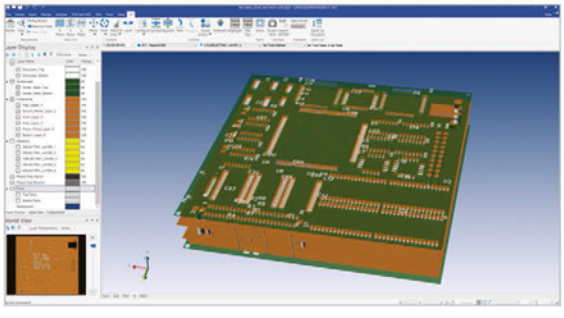
Ensure that manufacturing data is accurate for PCB construciton.

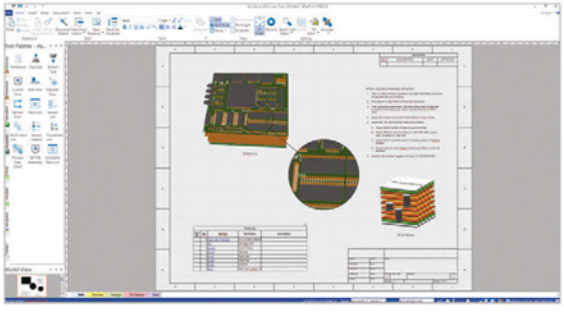
Panelize

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

Document

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





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

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

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D

hole in a rigid-flex circuit. Eliminating acrylic adhesive from the plated through-hole area will greatly increase the reliability of the PTHs
- Rigid sections connected by flex should be a minimum of 0.375" apart and preferably 0.5" or more
- Utilize "unbonded" construction to increase flexibility. When using unbonded construction on impedance controlled circuits, you must ensure that signal and reference plane layers are not unbonded from each other. When the circuit is bent, the unbonded areas will buckle, which will cause an impedance mismatch if the signal and reference plane layers are not bonded together
- When specifying a carrier panel or "pallet" for component installation, contact your manufacturer to make sure that the carrier panel fits efficiently on their processing panel. Failure to do this can result in a major cost increase

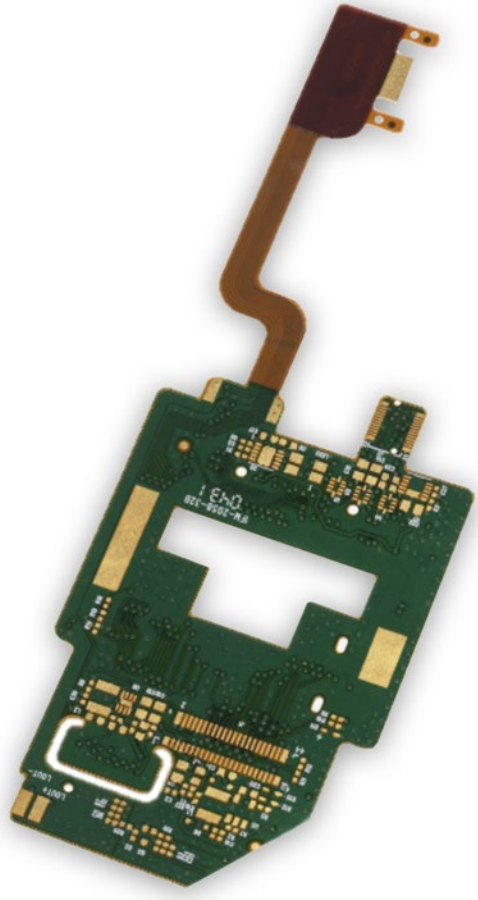


Figure 6: A typical rigid-flex circuit.

Cost Drivers

Every designer is looking for ways to decrease costs without sacrificing performance. IPC research has shown that PCB designers drive over 75% of the circuit cost based on the decisions they make. It is imperative that the flex designer understand what features add value and what features add only cost. Designers should never sacrifice reliability to save costs, but at the same time, many flex circuits are over-specified, resulting in additional costs that add no additional value. Here is a list of the features that drive the majority of your circuit cost.

- **Layer Count:** As the number of layers increase, so does the cost. More layers will require additional materials and processing time. Processing high layer count flex or rigid-flex can also be very technically challenging, which may result in reduced yields
- **Circuit Size and Shape:** Most flexible circuits are constructed in panel form. The greater the panel area a circuit occupies, the greater the cost. There are instances where even a small change in outline can result in a large cost decrease. A slight modification to the flex shape allows for a better nesting of the flexes on the panel, resulting in two more circuits per panel
- **Circuit Type (i.e., Type 3 vs. Type 4):** Rigid-flex circuits are typically more expensive than multilayer flex with stiffeners. Scrutinize your design to determine if your application requires a rigid-flex construction or if a multilayer with stiffeners will work. If in doubt, call your flexible circuit manufacturer and ask
- **Circuit Class (i.e., Class 3 vs. Class 2):** Class 3 circuits require additional testing, inspection, and construction requirements, which make them more expensive. Review the requirements of your application to determine the proper class for your flex circuit

- **Drawings Overly or Too Tightly Dimensioned:** It is important to remember that you are purchasing a flexible circuit, not a machined part. The materials used to manufacture flexible circuits both permit and require looser tolerances than rigid PCBs. Each dimension placed on a drawing will have to be verified, so ask yourself, "Is this dimension adding value or just cost?" All non-critical dimensions on your flex circuit drawing should be designated as reference
- **Dissimilar Layer Counts in PTH Areas:** All areas that have plated through-holes should have the same layer count and construction
- **Multiple Final Finishes:** While multiple final finishes can certainly be accomplished, it usually requires a series of hand masking operations that will add cost
- **Small Features:** Because of the inherent dimensional instability of flex circuit materials, small circuit features (i.e., via pads) can cause processing difficulties and reduced yields. There are instances where it would be less expensive to add additional layers with larger features than to design with very small features. For this reason, it is advisable to contact your flex supplier early in the design stage for guidance
- **Blind and Buried Vias:** These are significantly more expensive than through-holes

Flexible Circuit Technologies is a flex fabricator with locations in Minneapolis, Minnesota, and Shenzhen, China. This article was adapted from their flex design guidelines, available at www.flexiblecircuit.com. **FLEX007**

NIU Launches the Future of Urban Electric Motorcycles: 5G Connected, Autonomous and Self-Balancing

NIU Technologies introduced two new electric vehicles, the RQi-GT and TQi-GT, at this year's Consumer Electronics Show in Las Vegas.

The new RQi-GT is an urban performance electric motorcycle, allowing riders to reach the outer limits of their city at top speeds of estimated 160km per hour. Designed and built around NIU's advanced technology, the motor provides a peak output power of 30 kW and the two removable batteries (combined 7 kWh) can take riders up to 130km on a single charge.

The TQi-GT is NIU's first self-balancing electric three-wheeler, which comes standard with autonomous driving functionalities, including self-parking. Designed to provide urban commuters with an advantageous way to enjoy their city, the TQi-GT can reach top speeds of 80 km per hour and has a range of up to 200 km to provide more than a week of urban commuting range.

At present, NIU IoT can use smart sensors to detect information in real time and aggregate cloud computing analysis to recognize people, vehicle, cloud connectivity. With the arrival of the 5G era, NIU IoT will have a lot more space for innovation. And NIU aims to build a transportation network platform called the NIU FLEET and it can be the perfect urban mobility model that NIU envisions for the new decade.

The mass production of the new models, RQi-GT and TQi-GT, is expected in the second half of 2020 and will be made available to consumers a few months later.

(Source: Business Wire)

FlexOO7 News Highlights

Insulectro Works to Bridge the Fabricator/Designer Gap ▸

Barry Matties sat down with Insulectro's Megan Teta and Mike Creeden to discuss trends they see in the materials market and how they're working to bridge the gap between fabrication and design, including helping designers understand what they can do to make a board more manufacturable.

Decreasing Bend Radius and Improving Reliability, Part III ▸

Many flex novices will design a circuit that calls for bending the flex in too tight of a bend radius, which can cause damage to the circuit and lower the reliability of the end product. This is the third and final installment in a series of articles that will focus on the seven key aspects to consider when designing for maximum durability and maximum flexibility.

Is 3D Printing the Future of Battery Design? ▸

One of the enabling technologies of our 21st-century lifestyle is the lithium-ion battery. These energy packs make possible mobile phones and electric cars, laptops, and healthcare devices, robots and remotely operated sensors, and much more.

Eurotech Expands With Acquisition of Lyncolec ▸

U.K. PCB manufacturer The EuroTech Group has acquired Lyncolec, based in Poole, Dorset. The deal will see the enhancement of EuroTech's manufacturing capability to include flex and flex-rigid PCBs. This complements its current offering and will facilitate EuroTech's strategy to provide a more diverse product range to its customers.

Worldwide Wearables Surge 94.6% in 3Q Led by Hearables Market, Says IDC ▸

Global shipments of wearable devices totaled 84.5 million units in the third quarter of 2019 (3Q19), a year-over-year increase of 94.6% and a new record for shipments in a single quarter, according to IDC.

Meet Super PCB at IPC APEX EXPO 2020 ▸

Super PCB plans to exhibit at IPC APEX EXPO 2020, scheduled to take place February 4–6 at the San Diego Convention Center in California. The team will display samples of a number of Super PCB products for the fast-moving PCB industry, including single-layer, two-layer, multilayer rigid PCBs, flexible PCBs, rigid-flex PCBs, aluminum PCBs for LED and Rogers PCBs for RF applications, HDI, and others.

AltiumLive Frankfurt 2019: Carl Schattke Keynote ▸

"How many here can remember manually taped artworks?" No more than three hands were raised in an audience of over 230 at the AltiumLive 2019 European PCB Design Summit in Frankfurt, Germany, as IPC Advanced Certified Interconnect Designer Carl Schattke introduced his keynote, entitled "Making and Breaking the Rules."

Toray Creates Revolutionary PPS Film for 5G Circuit Boards ▸

Toray Industries has created a polyphenylene sulfide (PPS) film that maintains the outstanding dielectric characteristics, flame retardancy, and chemical robustness of that polymer while remaining thermally resistant at 40°C higher than conventional counterparts. Employing the new film in flexible printed circuits offering 5G and other fast data rates would offer several benefits.

LENTHOR ENGINEERING

MAKING THINGS FLEXIBLE IN A RIGID WORLD®

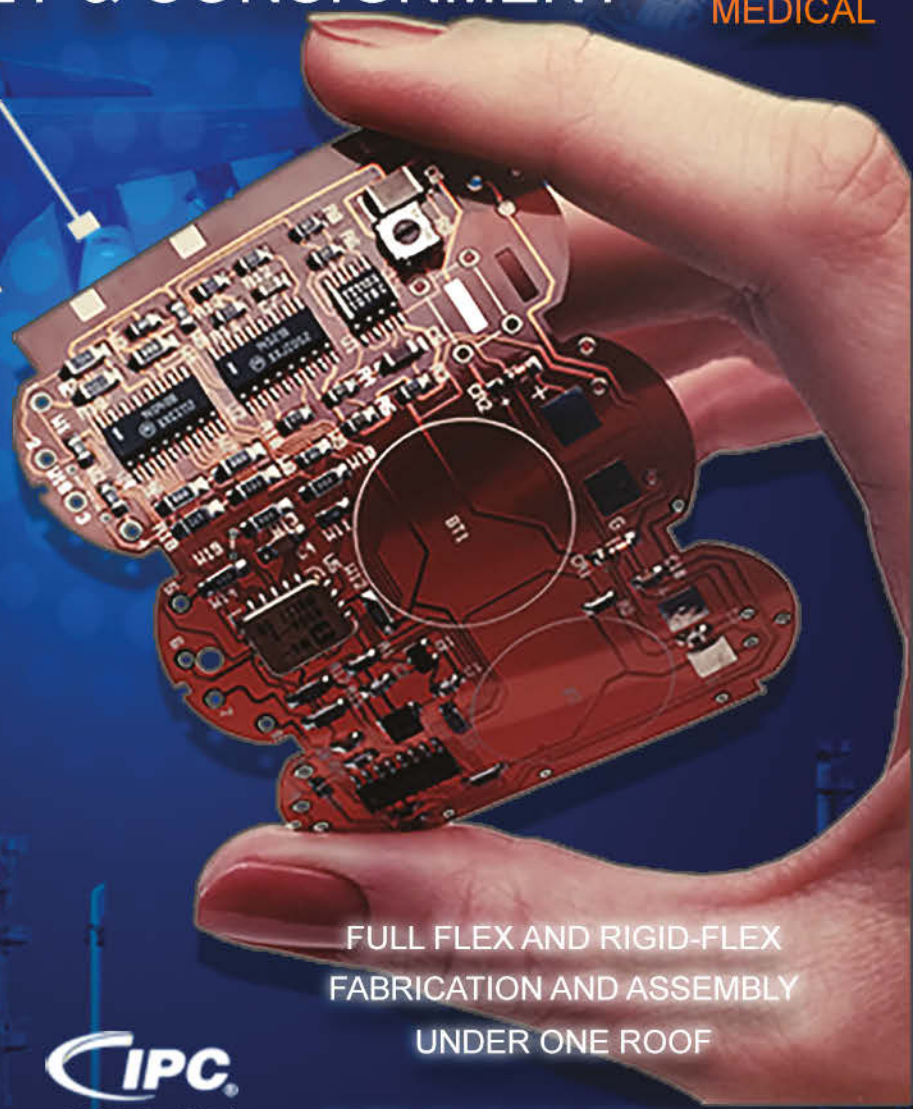
FLEXIBLE CIRCUITS ASSEMBLY

- 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

OVER 32 YEARS SPECIALIZING IN FLEX AND RIGID-FLEX PWB / PWA TECHNOLOGIES

IPC J-STD-001

ITAR

RoHS Compliant

ISO

MADE IN USA

Looking Back and Looking Forward

Flexible Thinking

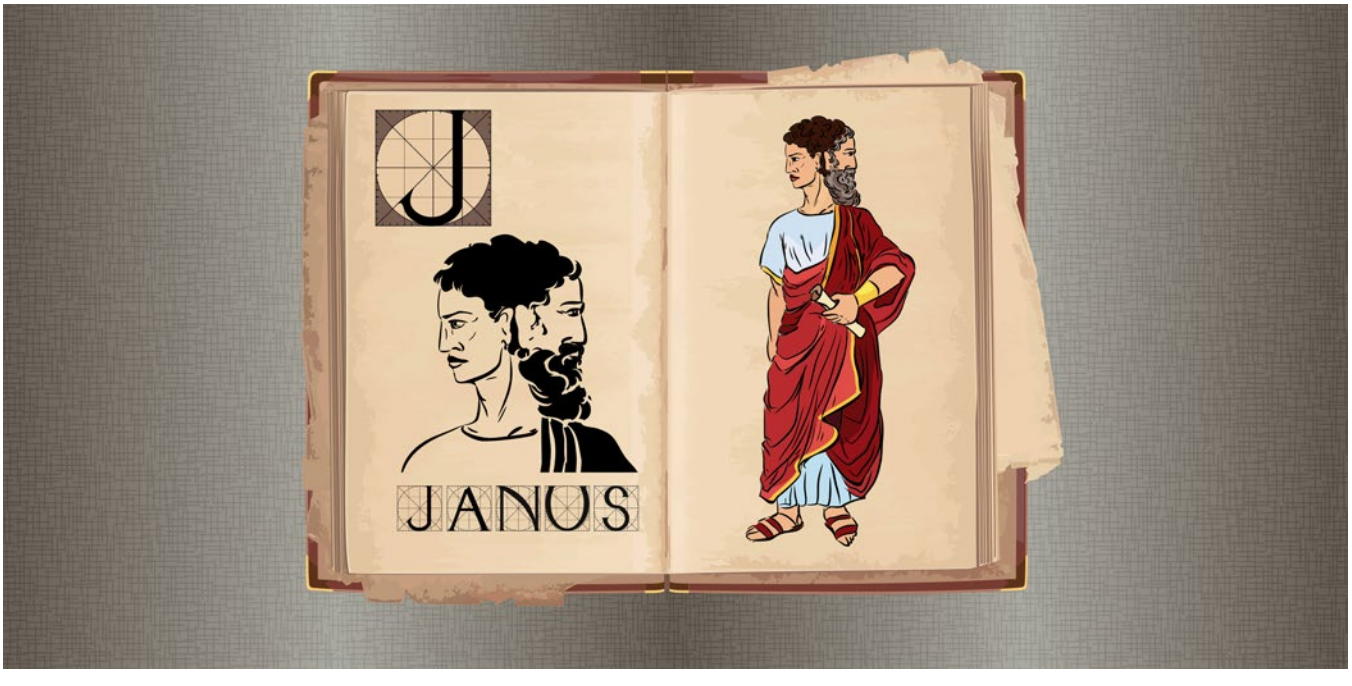
by Joe Fjelstad, VERDANT ELECTRONICS

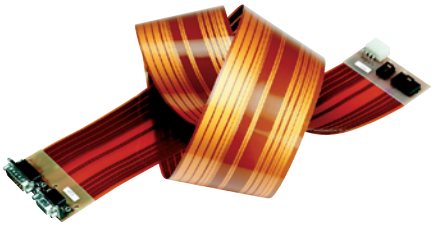
The month of January is upon us once again. The month is named after the Roman god Janus. According to Wikipedia, Janus is the god of beginnings, gates, transitions, time, duality, doorways, passages, and endings. He is usually depicted as having two faces: one on the front of his head, and one on the back since he looks to the past and future.

The electronics industry, like every other industry, needs to recall its past even as it looks forward to its future. Guidance can be found and should be taken from the past. As George Santayana warned, "Those you don't remember history are condemned to repeat it." However, while history can provide a measure of guidance as to the past, it is not infallible. It does not predict the future with certainty. Things happen that are not always predictable.

The universe has, for the last 14 billion years or so, been about never-ending change. The ancient Greek philosopher Heraclitus clearly understood this, having said, "There is nothing permanent except change."

Change is not always welcome but can be reasonably predictable as we connect dots from the past and extrapolate them into the future. However, in virtually every field of human endeavor and experience, there are inevitably found some rare events that defy experience and prediction by simply arriving on the scene, seemingly out of nowhere. This is the theme of the book *The Black Swan: The Impact of the*

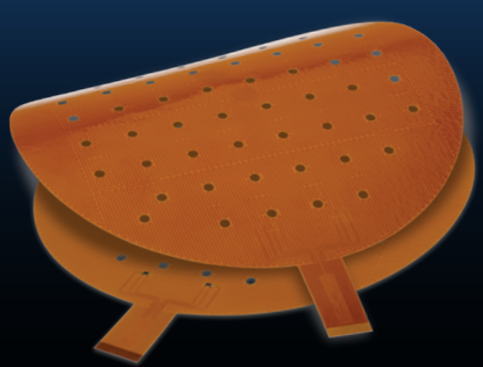




all flex

Flexible Circuits • Heaters • Assemblies

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



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

allflexinc.com

Highly Improbable by Nassim Nicholas Taleb. It is a topic I have explored in the past, but it is also one that needs to be revisited from time to time so that the lessons of history are not forgotten.

The term "black swan" is a reference that has been traced to the Roman poet Juvenal who wrote, "A good person is as rare as a black swan." While not necessarily a misanthrope, it is evident that he was at least a reasonably well-studied observer of human nature and would likely have taken the current political situation in Washington D.C. as predictable.

Fast forward a couple of thousand years, and the use of the black swan analogy as a reference to something exceedingly rare surfaces in 16th century London, at which time the term was used to describe an occurrence or event deemed impossible. This can be attributed to the fact that the commonly held perception at the time was that all swans were white and that black swans did not exist. However, that perception was eventually proved wrong. There are indeed black swans, just not all that many.

The Black Swan explores and examines the disproportionate effect of rare and largely unpredictable events that have ultimately wound up redirecting and even fundamentally reshaping science, technology, and history itself. Taleb also discusses the various human psychological biases that blind us to the very existence of uncertainty and even conspire to make us sleepily unaware of the impact of such rare events even as they are occurring right under our noses.

Interestingly, when they are finally recognized for what they are, the events are then typically viewed retrospectively as predictable, with many suggesting that they knew all along that the event was bound to happen. It conforms well to the adage "Hindsight is 20/20." Thus, while such events are rare, in general, they often play a larger role in reshaping or redirecting the arc of progress than routine or normal evolutionary events.

Consider the following events in our world of electronics. Switches were always mechanical until they were electrical, and they were then based on vacuum tube technology until they were based on transistor technology. What might be next? People are working with photonics now, for example.

Electrical connections were originally made point to point using wires, and then they made using printing technology and conductive inks, followed by them being etched; now, they are increasingly being printed again. Things cycle. But what might be next? With the continuous reduction of semiconductor devices and the rise of wireless communications and IoT, the spaces that surround us in everyday life could represent the next generation of connection.

Further, through-hole component technology was overtaken and replaced by surface-mount component technology. Peripheral leaded components are being supplanted by area array components that use board area more efficiently. Then came stacking and 2.5D and 3D package structures. Again, what might be next?

Chiplets is the latest topic of high interest. It is a concept I have a personal interest in having proposed "disintegrating circuits" almost a decade ago to fit into the Occam process concept. Positing the prospective benefit of enabling designers to execute their designs using the least number of transistors possible, and even using early nodes of semiconductor technology, helped to make electronics much more

reliable than the ones being made today.

First proposed in 2007, the Occam process represents a potential electronics industry "black swan" technology, quietly being developed and refined. It is one that could greatly and positively impact the cost, reliability, and environmental friendliness of electronics manufacturing by simply eliminating the soldering process. The concept has proven intriguing to many over the last several years, but new concepts are not always openly welcome because of their disruptive nature. For those interested, download *Solderless Assembly For Electronics: The SAFE Approach*—a free, educational book on the subject.

Nature may favor evolution, but business tends to favor stability. Still, when new technologies ultimately prove their ability to deliver on their promises—whether it's cost reduction, improved performance (e.g., the integrated circuit), or some other positive attribute—they can have a compelling and profound effect on the industry.

Summary

To summarize, Taleb suggests that black swan events are events that meet three basic criteria. First, they are surprises and not generally viewed as predictable by the average individual. Second, they normally have a significant and even profound effect on any given group, from a society to an industry. Third, their occurrence is normally rationalized by hindsight, and it is often treated as if it was expected. In short, if we expect the unexpected, when change arrives, we can all say, "I was certain change was coming. I just didn't know when." **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 185 patents issued or pending. To read past columns or contact Fjelstad, click here.

Is 3D Printing the Future of Battery Design?

Battery performance is the result of numerous different factors. Electrochemists know only too well how delicate this balancing act is. Where are the big improvements we need likely to come from?

We now have an answer of sorts: batteries of the future will be made via 3D printing, say Vladimir Egorov at the University of Cork in Ireland, and a few colleagues. These folks have surveyed the various new printing techniques for batteries and suggest that this will make possible a new generation of smaller, more capable devices.

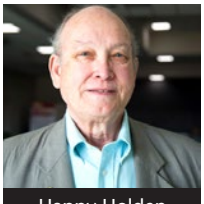
Materials scientists have also begun to experiment with ways to print electronic circuits using polymer inks and a silver polymer for traces, so soldering is no longer needed. However, a significant limitation is the need to incorporate conventional batteries, which come in specific sizes and shapes.

The ability to print 3D batteries will change that. "If they can be printed to seamlessly integrate into the product design, for aesthetic as well and comfort or functional reasons, the bulkier and fixed form factor standard battery need not be accommodated at the product design stage," say Egorov and company.

This is easier said than done. The electroactive materials used in batteries are inherently reactive, and structures such as anodes and cathodes are physically complex. Most important of all, batteries must be safe. One of the biggest and most important challenges for the battery industry is in making its products recyclable. The flexibility that 3D printing allows has the potential to kick-start and accelerate this much-needed revolution.

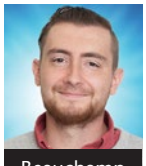
(Source: MIT)

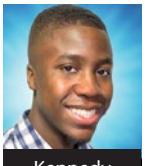
1 AltiumLive Frankfurt 2019: Happy Holden Keynote ▶


Happy Holden

Nobody left early! Altium had wisely kept Happy Holden's keynote presentation on "PCB Trends that Will Impact Your Future" until the end of the final day of the AltiumLive 2019 European PCB Design Summit in Frankfurt, Germany. Pete Starkey presents the highlights of Happy's presentation.

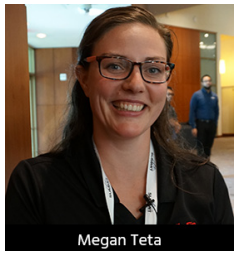
2 Fresh PCB Concepts: Designing a PCB for Telecom Applications ▶


Beauchamp


Kennedy

Jeff Beauchamp and Harry Kennedy discuss PCBs for telecommunication applications, including key factors to consider, such as design and material considerations. They also recommend involving your PCB supplier at the time of design to help ensure manufacturability at the lowest possible cost.

3 Insulectro Works to Bridge the Fabricator/Designer Gap ▶


Megan Teta

In this interview, Megan Teta and Mike Creeden discuss trends they see in the materials market and how they're working to bridge the gap between fabrication and design, in order to help designers understand what they can do to make a board more manufacturable.

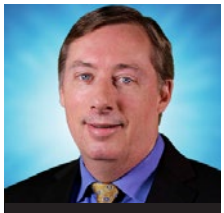
4 Connect the Dots: A Penny for Your Thoughts on Copper ▶


Bob Tise

Copper is the primary metal for standard PCBs. And while standard PCB capabilities depend on what materials are used and how they are constructed, copper is the go-to choice.

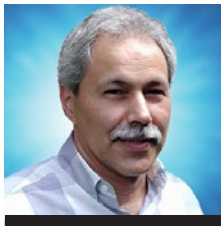
Bob Tise explains some of copper's applications, advantages, and challenges.

5 Lightning Speed Laminates: Test Vehicles for Materials Evaluation ▸


John Coonrod

There are many different types of PCB designs and constructions that can be used as a test vehicle to evaluate electrical properties. John Coonrod explains how a proper test vehicle to compare different circuit materials would be a design and construction that takes into consideration the different material properties and have the least amount of PCB fabrication variables that can impact the results.

6 Columnist Istvan Novak Nominated for Engineer of the Year ▸


Istvan Novak

Design007 columnist Istvan Novak, a principal signal and power integrity engineer with Samtec, has been nominated for the DesignCon Engineer of the Year Award.

7 Mentor Paper: Concepts of Signal Integrity—Impedance ▸

Successful signal integrity analysis depends on a fundamental concept: impedance. Without a thorough understanding of the impedance values that a signal encounters along the way, designers cannot maintain good signal quality from source to receiver. This Mentor paper defines the difference between characteristic and instantaneous impedance and explores the essential principles of impedance and design-controlled impedance.

8 DesignCon Celebrates 25th Anniversary With 2020 Keynote Lineup ▸

DesignCon 2020—the nation's largest event for chip, board, and systems design engineers—announced three keynote speakers that will headline the conference and expo set to take place January 28–30 at the Santa Clara Convention Center.

9 The Digital Layout: Fall 2019 Recap ▸


Stephen Chavez

As the second half of the year is quickly passing by, the IPC Designers Council has 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. Stephen Chavez provides a brief breakdown of fall activities within local chapters.

10 Zuken Unveils High-speed Features for Internet-connected PCB Design ▸

eCADSTAR Advanced HS extends eCADSTAR's connected 3D PCB design capabilities with powerful high-speed routing tools, such as length and delay-based impedance and skew control and comprehensive constraint management capabilities.

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

CAM Engineer

Eagle Electronics is seeking a CAM engineer specific to the printed circuit board manufacturing industry. The candidate should have a minimum of five years of CAM experience and a minimum of two years of experience in Frontline InCAM software. The candidate should also be fluent in PCB and CAM language pertaining to customer and IPC requirements. The ideal candidate has experience with scripting Frontline InCAM software.

This is a first-shift position at our Schaumburg, Illinois, facility; this is not a remote/offsite position. Any offer would include relocation costs to the Schaumburg, Illinois, area along with competitive salary and benefits.

If interested, please submit your resume to HR@eagle-elec.com and include "CAM Engineer" in the subject line.

About Eagle—Since 1979, Eagle Electronics has provided customers with the highest quality printed circuit boards at fair and competitive prices. From providing customers with short standard lead times to very low premiums on quick turns, Eagle strives to provide the best total value in high technology rapid turn-around PCBs in the industry.

apply now

orbotech®
A KLA Company

West Software Application Engineer

This position reports directly to the Orbotech West software support manager and works with customers to support Orbotech's pre-production software products. Acts as a focal point for technical issues, manages product implementation projects, provides customer training, and supports the sales process. Advanced knowledge of Frontline PCB products, including InCam, InPlan, InStack, InSight, Genesis, and Genflex. Ability to travel and manage time to maximize results. Requires both written and oral technical communication skills. Skilled in the use of scripting languages, including C-Shell, Perl, or Python. Knowledge of relational databases and HTML/XML highly desirable. Knowledge of PCB manufacturing processes. Familiar with the processes used in front-end engineering departments at PCB fabrication sites. Requires use of project management skills to organize and complete projects that involve the implementation of sophisticated software tools used in printed circuit fabrication facilities.

An expected average of 35%+ travel. College degree or equivalent technical education, in addition to a minimum of five-plus years of related experience. Experience supporting sales and sales activities is a plus. U.S. citizen with the ability to work and travel within the U.S., Canada, and internationally.

apply now

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

OEM Sales Manager
Chicago/Home-Office-Based

Want to advance your career by joining a globally successful and growing world-class CCL manufacturer and help drive that success? We are seeking to hire an OEM sales manager to grow and manage key customer accounts with OEM's and Tier 1 manufacturers in the USA, focusing on Ventec's core market segments: mil/aero, automotive, and medical, offering a full range of high-reliability materials, including polyimide, IMS, and thermal management products.

Skills and abilities required for the role:
- Non-negotiable: Drive and tenacity!

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

Full product training will be provided.

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

Please forward your resume to jpattie@ventec-usa.com and mention "Technical Sales Engineer—Chicago" in the subject line.

apply now

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

Career Opportunities

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

GO Gardien

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

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

INSULECTRO

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

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

INDIUM CORPORATION®

Multiple Positions Available

The Indium Corporation believes that materials science changes the world. As leaders in the electronics assembly industry we are seeking thought leaders that are well-qualified to join our dynamic global team.

Indium Corporation offers a diverse range of career opportunities, including:

- Maintenance and skilled trades
- Engineering
- Marketing and sales
- Finance and accounting
- Machine operators and production
- Research and development
- Operations

For full job description and other immediate openings in a number of departments:

www.indium.com/jobs

apply now

Manncorp™

SMT Field Technician
Huntingdon Valley, PA

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

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

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

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

apply now

Career Opportunities

U.S. CIRCUIT

Sales Representatives (Specific Territories)

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

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

ZENTECH

Zentech Manufacturing: Hiring Multiple Positions

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

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

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

Please email resume below.

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Master Instructor

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

For more information, click below.

apply now

YOUR JOB AD HERE

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

I·Connect007
GOOD FOR THE INDUSTRY

WHAT DO ALL OF THESE LEADING COMPANIES HAVE IN COMMON?

atg LUTHER & MAELZER

ATOTECH

CHEMCUT
BOUNDLESS INNOVATION | UNBEATABLE PRECISION

CIMS
Creativity in Motion

CYBEROPTICS

DownStream Technologies

ELECTROLUBE
THE SOLUTIONS PEOPLE

esi
Designed for Brilliance. Engineered for Production.

KIC Making Ovens Smarter™

KYZEN

MacDermid Alpha ELECTRONICS SOLUTIONS

Mentor
A Siemens Business

MIRTEC

NANODIMENSION
Electrifying Additive Manufacturing®

orbotech
The Language of Electronics

pluritec

PULSONIX

rehm
THERMAL SYSTEMS

schmoll asia pacific

ventec
INTERNATIONAL GROUP
腾辉電子

VJ ELECTRONIX
Process Control Solutions
CONFIDENCE DELIVERED

YXLON

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

Should your name be on the leaders' list?

GET STARTED NOW!

I·Connect007
GOOD FOR THE INDUSTRY

pcb007china.com

Events Calendar

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

CPCA Show 2020 ▶
March 16–18, 2020
Shanghai, China

electronica & productronica China ▶
March 18–20, 2020
Shanghai, China

LOPEC Exhibition and Conference (Driving the Future of Printed Electronics) ▶
March 24–26, 2020
Munich, Germany

KPCA and KIEP Show ▶
April 22–24, 2020
Kintex, Korea

Additional Event Calendars

Coming Soon to *DesignOO7 Magazine*

February 2020: PDN: Power Distribution Network
Turn on the power! We focus on the best methods for designing a power distribution network and the effects that your PDN can have on EMC.

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

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

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 © FINAL09**

DESIGN007 MAGAZINE

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

January 2020, Volume 9, Number 1
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

my I-Connect007

myiconnect007.com

MEDIA Kit007

mediakit.iconnect007.com

EDITORIAL CONTACT

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

SALES CONTACT

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

pcb design 007 | pcb 007 | smt 007 | mil aero 007 | flex 007 | ein 007 | RTW

www.iconnect007.com