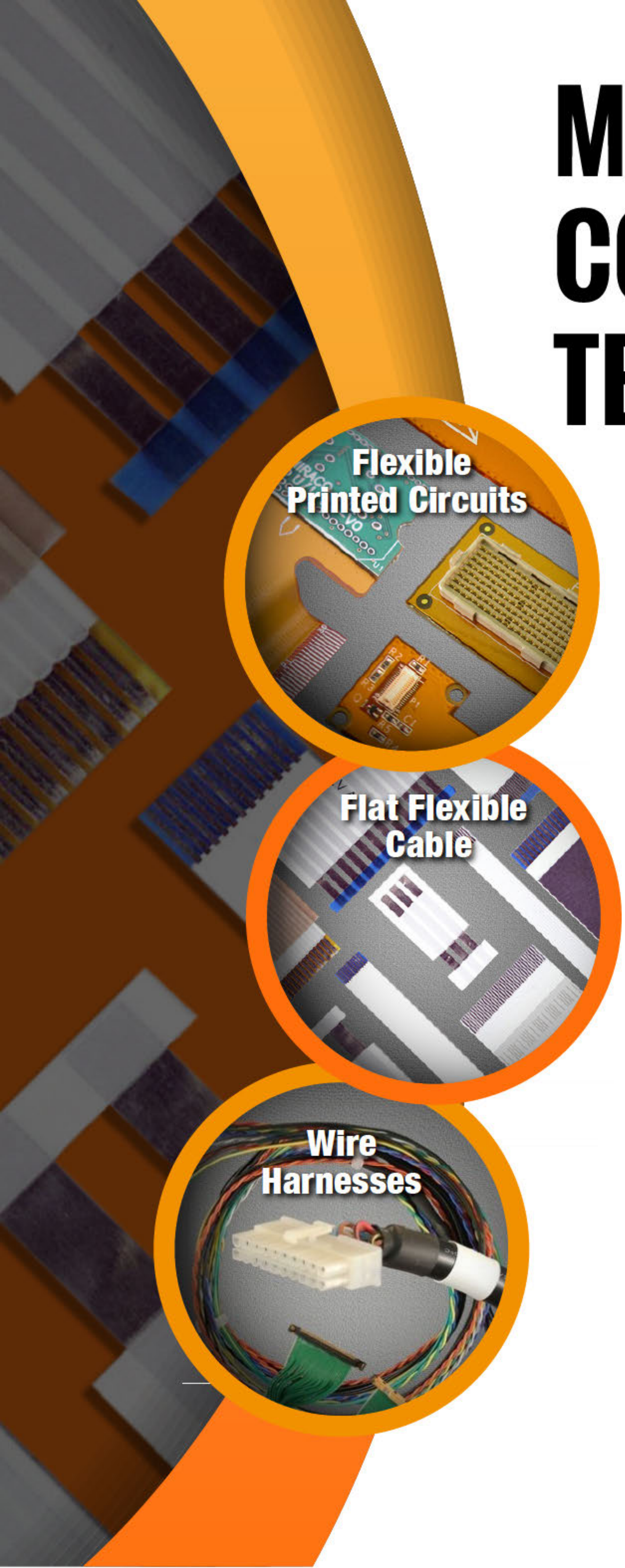
I-Connect007

JULY 2019

DESIGN007 MAGAZINE

THE SECRET LIFE OF VIAS

NEW MONTHLY SECTION FLEX007


Flexible
Printed Circuits
Flat Flexible
Cable
Wire
Harnesses

DESIGN007 MAGAZINE

Reliability & Vias

This month, we look into the secret life of vias as the culprit for failures and some of the reliability issues that technologists are facing today.

FEATURES:

ARTICLE:

COLUMNS:

FLEX007

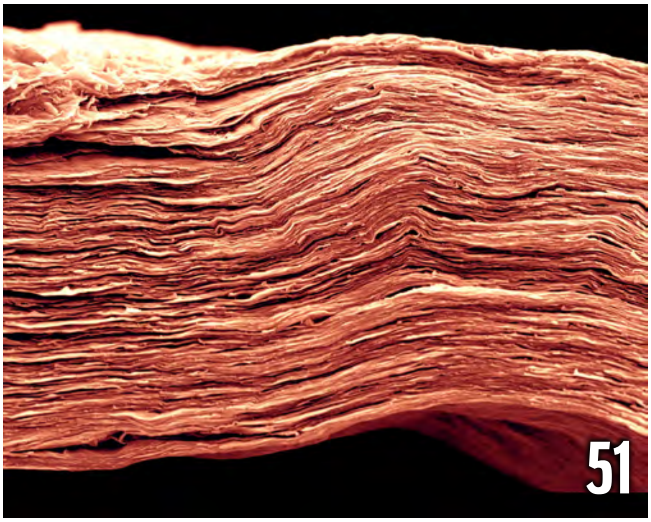
NEW Section!

Welcome the the new **FLEX007** section of *Design007 Magazine*. Flex is one segment that just keeps growing, and we'll be here to provide the flex design information that you can't find anywhere else.

FLEX007 ARTICLES:

FLEX007 COLUMNS:

DESIGN007 MAGAZINE



SHORTS:

HIGHLIGHTS:

DEPARTMENTS:

Reliability Is a Team Sport

The Shaughnessy Report
by Andy Shaughnessy, I-CONNECT007

My 2003 Mazda Tribute doesn't look very cool; it's classified as a "cute ute." This car was designed to spend a lot of time in an elementary school carpool lane, but it is about as reliable as any car can be. It just goes and goes, and it doesn't break down very often—especially for a car with over 200,000 miles on it (it helps that I telecommute and don't spend two hours each day driving back and forth to work).

Everything still looks new, inside and out. It's fun to drive, and it can haul four guitars and a pair of PA speakers with room to spare. It's been paid off for so long that I've been able to put more money away for my rapidly approaching golden years. This was no accident; as the saying goes, "Reliability isn't just an added feature." Every person involved in designing and building my car contributed to the car's long-term reliability.

Reliability has been in the news quite a bit lately, and as always, it's when something has proven to be unreliable. In 2018, IPC formed the IPC V-TSL-MVIA Weak Interface Microvia Failures Technology Solutions Committee to try to find the root causes of microvia interface failures that have been affecting the defense and aerospace segments. Members of that subcommittee spoke at the IPC High-Reliability Forum and Microvia Summit in Baltimore, Maryland, and I was fortunate enough to cover that conference.

Now in its third year, this conference has continued to grow with over 100 attendees. No one at the conference expressly blamed the designer for these failures, in case you were wondering. But there are steps that designers can take early in the design cycle to head off trouble later. Motorola's Jerry Magera and J.R. Strickland have been looking into these via failures for years. In their presentation, they offered a few tips for designers seeking to create the most reliable board; namely, avoid using stacking microvias. Try not to use stacked and staggered microvias in the same board. Keep the vias as large as possible, and the as-

Past issues of *Flex007 Magazine* are available for download here.

pect ratios as small as possible. Try to avoid mixing processes on the same board. Variation is the enemy.

This is a problem that will likely require competitors to share data and work together for the greater good. As Magera said, "OEMs can't let their fabricators go out of business." Incidentally, the IPC V-TSL-MVIA subcommittee needs more members who can share their company's data on microvia failures and put in the hard work needed to solve this problem. Talk about working as a team. I believe our industry will come together and work with each other on this problem, even if means a few uncomfortable moments of working with rivals.

This month, we kick off with an article by Greg Ziraldo, director of operations at Advanced Assembly. He explains how to design vias for maximum reliability, echoing the idea that, "When it comes to vias, bigger is better." Then, we have a feature interview with consultant and I-Connect007 columnist Dennis Fritz who discusses the difference between two terms that are often used interchangeably—"quality" and "reliability"—as well as a proposal to begin allowing the manufacture of DoD PCBs with lead-free solder. And we have a feature interview with Brook Sandy-Smith, IPC's technical education program manager, who gives a review of the IPC High-Reliability Forum and Microvia Summit and some of the reliability issues that technologists are facing today.

We also have columns from our regular contributors Barry Olney of iCD, Stephen Chavez with the IPC Designers Council, Bob Tise of Sunstone Circuits, consultant Tim Haag, and Phil Kinner of Electrolube. Further, we have a technical article by C.F. Yee of Keysight Technologies, titled "The Impact of Inductance on the Impedance of Decoupling Capacitors."

New Flex007 Section

Yes, it's official. Take a quick check of our Table of Contents, and you'll notice that we now have a Flex007 section. So much of the content of the quarterly *Flex007 Magazine* revolved around flex design that we've merged that publication with *Design007 Magazine*. Each month, you'll find cutting-edge flexible circuit design content in this new section of *Design007 Magazine*. Now, you won't have to wait three months to get the great flex technical articles and information that you need to stay ahead of the game. Flex is one segment that just keeps growing, and we'll be here to provide the flex design information that you can't find anywhere else.

See you next month! **DESIGN007**

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

PCB Reliability: Via Design

Feature by Greg Ziraldo
ADVANCED ASSEMBLY

When considering the long-term reliability of a PCB, you must take into account any vias on your board. While an invaluable and essential part of board design, vias introduce weaknesses and affect solderability. This article will discuss vias, the potential concerns that are introduced into your board through their implementation, and how to minimize those concerns to acceptable levels.

The first rule for via design is simple—bigger is better. Larger vias have greater mechanical strength as well as greater electrical and thermal conductivity. While space is always a consideration when it comes to PCB design, vias should have a minimum drill width of 20 mils with an annular ring of 7 mils and a minimum aspect ratio of 6:1. For many boards, this may be an unachievable goal; however, the basic premise of "bigger is better" stands true.

When a PCB is exposed to thermal changes in its processing or end working environment, the varying coefficient of thermal expansion (CTE) between the laminate and the copper can cause issues. PCBs are constrained through structural latticework to limit horizontal expansion but can expand and contract significantly in the vertical direction. As copper expands and contracts at slightly less than one-fourth of the rate of FR-4 laminate, vias are being pulled apart every time the board is heated. If the board is too thick and the copper in the via too thin, then the board will expand too much, and the copper will break, tearing the via apart. In the previous example, to get the appropriate aspect ratio with a drill width

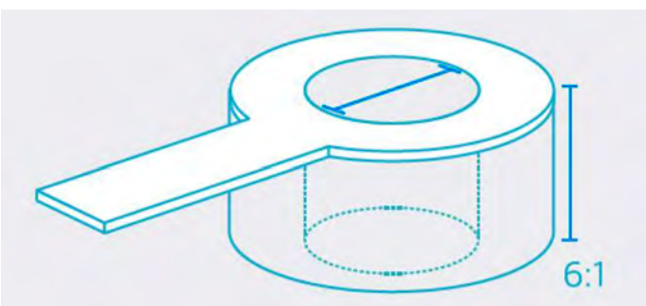


Illustration 1.

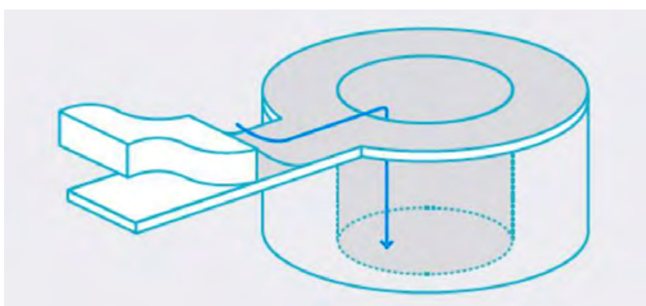


Illustration 2.

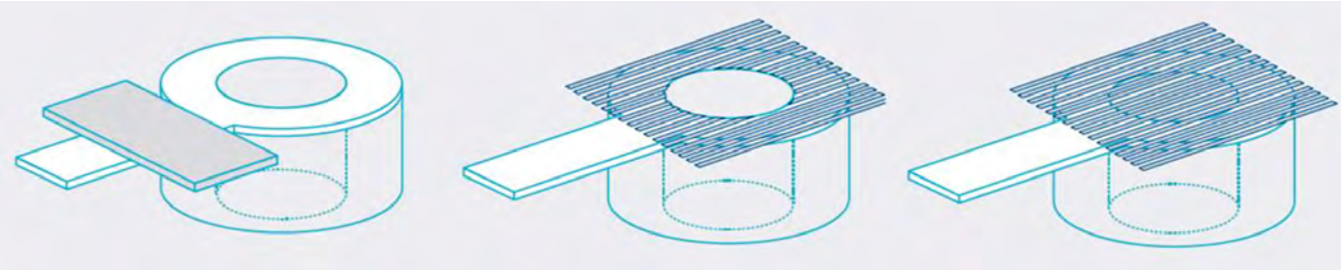


Illustration 3.

of 20 mils, this would result in a total pad diameter of 34 mils and allow a max board thickness of 120 mils.

Via size is important, but location is paramount. If a via is located close to a solder pad, a myriad of problems may arise, the foremost being the issue of solder wicking. As the via heats up, it pulls solder from the solder pad, through the via, and onto the other side of the board, leaving the pad either solder-deficient or completely solder-free. The larger the via, the more solder will likely wick away, making it less likely that you will have a solid mechanical and electrical joint. Fortunately, this concern can be fixed by any of three no-cost methods.

Providing a solder mask between the lead and the via creates a barrier to the movement of the solder. This is a simple yet effective method, though it does have its drawbacks. Due to the minimum width required for solder mask, this may require the via to be moved even farther from the lead. The distances required may seem minimal (in the 2–5-mil range). When space is at a premium or the board is carrying high-frequency signals, this may have a profound effect on your design. However, when these aren't issues, this is a great way to avoid solder wicking concerns.

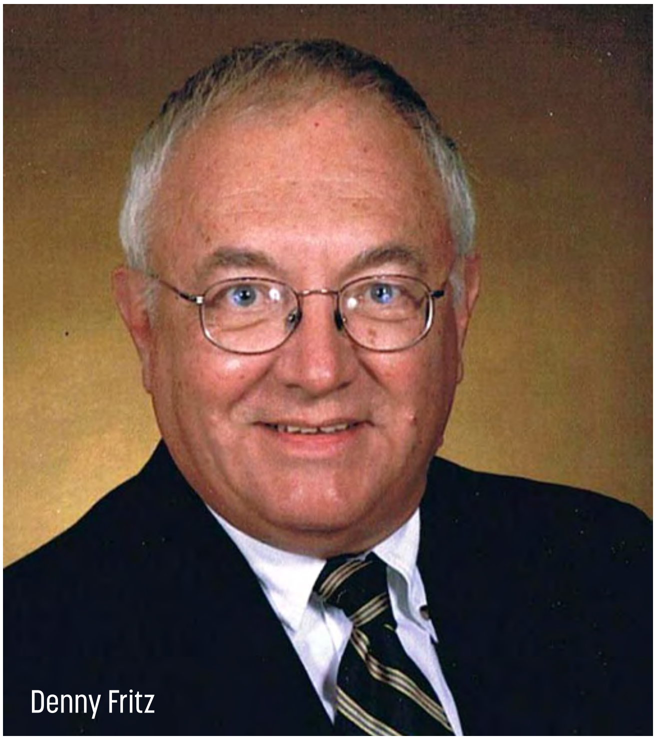
If there is no space to move the via and you need to minimize the via size, it is possible to use an encroached or tented via. By masking the via pad, you save space and also make it possible to silkscreen over the via. However, this makes it impossible to use the via as a test point as the copper will no longer be accessible. At this point, you need to decide whether an encroached or tented via is best. A tented via is completely sealed and will create a better surface for silkscreening as well as a better barrier against contamination. This barrier works both ways, though.

If a via is tented on both sides of the board, contamination can fill the void during fabrication. At elevated temperatures—such as when the PCB is being reflowed or wave soldered—the contamination can outgas and destroy the via and thus the board. When you tent a via, make certain you only do so on one side. An encroached via eliminates this issue by keeping the hole itself open and also has the added benefit versus a tented via of being possible no matter the via size. While tented vias need to be small enough for the solder mask to bridge the drill hole, the encroached via only covers the annular ring and can be as big or small as needed.

Filled vias are also an option; they provide increased strength and electrical/thermal conductivity while protecting the via from solder wicking and contamination. The main drawback to filled vias is that they can add significant cost to the board. But the other methods should have no impact on cost whatsoever.

Every design has its own requirements and constraints. However, when possible, utilize these tips by using the largest vias and the appropriate aspect ratio while thoughtfully choosing the solder masking style that suits your needs. This will help reduce the overall lifetime costs by increasing the reliability of your products. **DESIGN007**

Greg Ziraldo is director of operations for Advanced Assembly LLC.


Denny Fritz

The Difference Between Quality and Reliability

Feature Interview by Andy Shaughnessy
I-CONNECT007

I recently spoke with industry veteran (and I-Connect007 columnist) Denny Fritz about the relationship between quality and reliability—two terms that are unequal but often used interchangeably. We also discussed the current state of lead-free solders in the U.S. military and defense market as well as the microvia reliability issues Denny focused on at IPC's High-Reliability Forum and Microvia Summit in Baltimore, Maryland.

Andy Shaughnessy: It's good to see you again, Denny. What's the name of your company?

Denny Fritz: Fritz Technology Consulting. Send me something to do and I'll give you a price.

Shaughnessy: Good. Can you give us a wrap-up of what you've been up to lately? Everybody is fairly familiar with you, but what are you up to now?

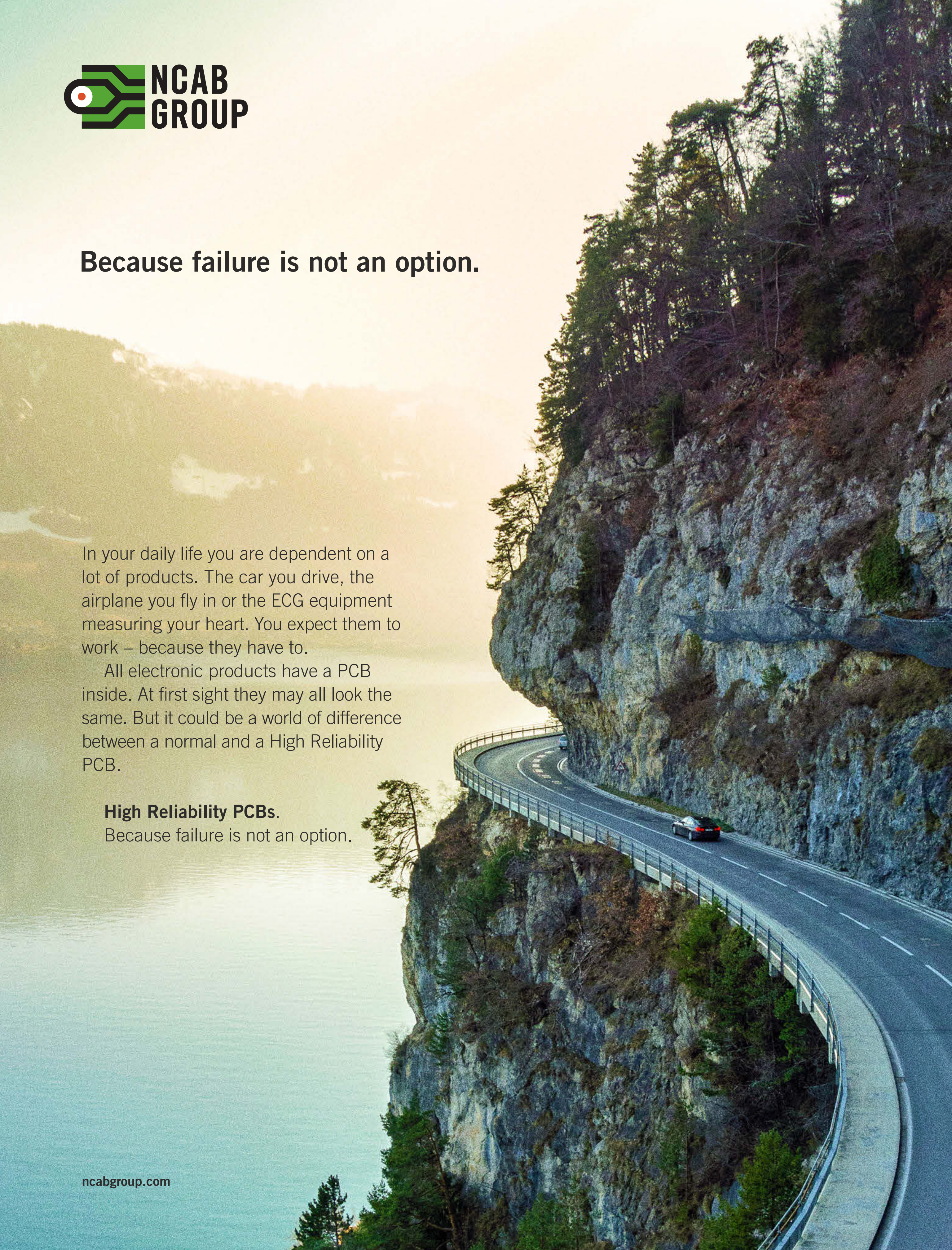
Fritz: Many people know me through IPC involvement, the IPC Hall of Fame, and 40 years of involvement in the PCB industry. Until July of 2018, I had worked about 12 years for SAIC, supporting the executive agent assignment at Navy Base Crane in Indiana. For many years—20 direct, 10 consulting, and a loose relationship today—I worked for MacDermid Inc. out of Waterbury, Connecticut, doing sales, technology, marketing, roadmaps, etc. Currently, if you have something that relates to technology forecasting and supply chain, I would consider consulting for you.

Shaughnessy: As a consultant, you're the expert from out of town.

Fritz: If you want to pay me, I'll be an expert.

Shaughnessy: Earlier, you mentioned the relationship between quality and reliability. To a lot of people, they're almost interchangeable, but you made some points that they're headed in the same direction but definitely aren't the same.

Fritz: Correct. They are not the same. Reliability is something that I've been very interested in for over 10 years now. It probably dates back to my involvement with a Navy Crane assignment and the reliability of lead-free solders


NCAB GROUP
Because failure is not an option.
In your daily life you are dependent on a lot of products. The car you drive, the airplane you fly in or the ECG equipment measuring your heart. You expect them to work – because they have to.
All electronic products have a PCB inside. At first sight they may all look the same. But it could be a world of difference between a normal and a High Reliability PCB.
High Reliability PCBs.
Because failure is not an option.
ncabgroup.com

in long-term defense system applications related to tin whiskers, etc. It has been impressed on me that we do a lot of inspection on our board fabrication—particularly board assemblies—to pass quality requirements, but those cannot completely ensure long-term reliability. Reliability is a matter of looking at data over the last 10–40 years when your product performed to the standards you expected. The buzz phrase I've heard is quality is care from birth to shipment, and reliability is care from cradle to grave. In my mind, that's the difference between the two.

Reliability is a matter of looking at data over the last 10–40 years when your product performed to the standards you expected.

Shaughnessy: You've spoken at these reliability conferences in the past, correct?

Fritz: I don't know how far back they go here at IPC, but I do know that I was involved with Werner Engelmaier and Dieter Bergman in 2008. It was a two-day conference in Massachusetts. That's the first time I became acquainted with organized reliability conferences through IPC. For the last three years, I've helped organize and participate annually. The first was held in 2017 in Rosemont, Illinois, and in 2018 and 2019, they were held here in Baltimore, Maryland.

Shaughnessy: You mentioned that the military is considering allowing lead-free. I know that with RoHS, military and aerospace had waivers, but now they're fine with lead-free?

Fritz: It's a complicated issue. The U.S. does not have a regulation that electronics have to be manufactured lead-free. In contrast, the European Union dictates that similar lead-free regulations have been adopted in Asia. Even in Europe, the defense industry has been able to get waivers to continue using leaded solder.

Now, in the United States, with the shift of components and board processes to lead-free, all of that is a tremendous market driver. I recently participated with the IPC Government Relations Committee proposing to Congress to form a consortium and have Defense funding for a five-year program. This would run tests that would make it a lot easier for Defense to convert new constructions to lead-free. This would help eliminate the "re-balling" or "re-tinning" of lead-free components when they come in to make them immediately assembly compatible.

If you mix eutectic, lead-containing solder with lead-free solder, you end up with an unknown alloy or a mixed solder joint, which sometimes is reliable and other times is not. If there's one thing Defense can't stand, it's unpredictability. We're looking to make sure that we could convert everything in new construction to lead-free. Defense has a small issue that legacy systems that were made with leaded solder will probably be field repaired for decades—maybe on a hand basis with leaded solder—but our proposal would convert new defense constructions over to lead-free.

Shaughnessy: That sounds like a good plan. When is this going to take place?

Fritz: It all depends on Defense budgets. It's all at the whim of our, some would say, dysfunctional Congress to get together to figure out how much is going to go to Defense priorities. Gradually, through the Pb-free Electronics Risk Management (PERM) Council under IPC, the industry has been solving some of these issues. But we've been at it now for 13 years, and the estimate is that maybe we've covered half of the information needed to establish lead-free. We think this five-year program, with government funding, would push lead-free to completion.

Shaughnessy: It sounds like a lot is happening.

Fritz: That's true. Some of this relates to overall reliability issues. For instance, I'm involved


NCAB GROUP
Because failure is not an option.
In your daily life you are dependent on a lot of products. The car you drive, the airplane you fly in or the ECG equipment measuring your heart. You expect them to work – because they have to.
All electronic products have a PCB inside. At first sight they may all look the same. But it could be a world of difference between a normal and a High Reliability PCB.
High Reliability PCBs.
Because failure is not an option.
ncabgroup.com

with this weak interface microvia reliability issue, and this is a nagging industry problem with multiple causes. We don't think there will be a single silver bullet. Most companies have thought that they alone had a microvia problem with three, four, or five stacked layers. We're in the process of trying to share information at IPC to see if we can avoid big test programs with hundreds of boards and multiple design iterations.

Shaughnessy: What are your plans at this conference?

Fritz: Here, I'm mostly involved with stacked microvias. We had a good meeting Monday afternoon with about 50 attendees of committee workers trying to sort this into sub-teams. We have seven sub-teams that address materials, laminates, processes, test methods, etc. One big item that needs help is gathering data and trying to organize that anonymously under IPC to get the answers as quickly as we can.

Shaughnessy: Is this whole thing related to the military board via failures?

Fritz: It can be, but it does not have to be. Military is a factor, but we have aerospace, medical, and high-complexity telecommunications people that are involved with weak microvias too. In the U.S., we utilize specific constructions with relatively thick layers that those industries use. Stacking saves space for the thousands of IOs from advanced semiconductor devices.

The military has been pretty conservative in design. They tend to use more standardized ICs, but microvia stacking is on the horizon. The people who are driving the hardest are the companies who would love to use AI, virtual reality, and IoT. Defense can see that they want the same features in their future weapon systems, needing to use high-density packaging, which includes stacked microvias.

Shaughnessy: But it's not happening with the staggered vias.

Fritz: If you don't make staggered microvias correctly, conductor breakage can occur. I think everybody has seen some single-layer microvias that weren't made well and failed. The thought here is that it's multiple levels of microvias, and the stacked microvias are showing the most problems, but it's not exclusive to stacked—predominantly, but not exclusively.

We anticipate that IPC will get involved in the specification for test methods to weed this out. I think testing will be a very active topic within IPC for the next year or two until these test methods are capable. The industry summary right now is that companies have mitigations in place that show that they are not shipping bad product; however, they don't know why they have to throw some suspicious panels away, and the root causes are unknown. And it's not economical to reject panels due to the mitigations that are in place.

Shaughnessy: It sounds like you're involved with a lot of things happening right now.

Fritz: Certainly enough to keep me busy.

Shaughnessy: I know you've tried to retire, but it's like Al Pacino, right? You keep trying to get out…

Fritz: But they keep pulling me back. Exactly.

Shaughnessy: What would you do if you weren't doing this?

Fritz: I live on old family land in Indiana that has been in my mother's family for 150 years and never been sold. I'm not exactly a hobby farmer, but I do rent some land out, raise a big garden, cut firewood, and other things like that. I have community activities, and I have worked half-time now for almost 20 years. I would just flip more into retirement.

Shaughnessy: Sounds like a great work-life balance. Thanks for talking to me, Denny.

Fritz: Thank you for covering this conference, Andy. **DESIGN007**


NCAB GROUP
Because failure is not an option.
In your daily life you are dependent on a lot of products. The car you drive, the airplane you fly in or the ECG equipment measuring your heart. You expect them to work – because they have to.
All electronic products have a PCB inside. At first sight they may all look the same. But it could be a world of difference between a normal and a High Reliability PCB.
High Reliability PCBs.
Because failure is not an option.
ncabgroup.com

IPC Reliability Forum Wrap-up


Brook Sandy-Smith

Feature Interview by Andy Shaughnessy
I-CONNECT007

I attended the recent IPC High-Reliability Forum and Microvia Summit in Baltimore, Maryland. The speakers and panelists focused on a variety of topics, but one issue that kept popping up was the failure of some microvias on military and aerospace PCBs. Fortunately, some smart technologists are focusing on determining the cause of these via failures. I asked Brook Sandy-Smith, IPC's technical education program manager, to give us a quick wrap-up of this event.

Andy Shaughnessy: Hi, Brook. It was great seeing you at the reliability conference in Baltimore. This conference has grown quite a bit in the last few years.

Brook Sandy-Smith: Yes, it has grown considerably. This was the third year for this conference, and I'm so pleased with the quality of the content and the number of attendees. It's a pretty big group, and yet it still feels intimate and special in this venue with lots of opportunities to interact and talk with other attendees. You'll be excited when you hear what we have planned for next year.

Shaughnessy: I talked to attendees who traveled from all over the country. Why do you think reliability is such a draw as a topic now? I know some of the attendees were concerned about military microvias failing.

Sandy-Smith: For several industries—such as defense, aerospace, and automotive—high reliability is the name of the game because our safety is in their hands, and the function of these devices is critical. In addition, companies that make other products look to these high-reliability industries to understand how to do things better. It's important to strive for a high level of quality and reliability even when making something less critical than a car or an airplane. This is why we keep seeing this "reliability" buzzword all over the industry.

You also mentioned the hot topic of weak microvia interfaces. This is an emerging topic because several companies across the industry have witnessed intermittent failures related to plating interfaces within microvias. This is a particularly difficult problem because the open

fracture may appear at higher temperatures but "heal" itself again upon cooling; nonetheless, this often results in field failures. It's insidious because the problem pops up after routine test procedures are successful. This issue is a big focus for this event because the team is working with IPC and the industry to define the problem, collect data to see how big of a problem it is, and find ways to mitigate it. Microvias have been—and will continue to be—a reliable and important feature in circuit boards, so it's important to figure out this concern quickly.

Shaughnessy: You all had a great line-up for the conference. What were some of the highlights in your opinion?

Sandy-Smith: It is difficult to single out a few moments because I enjoyed all of the presentations. I felt the keynote by John Bauer of Collins Aerospace was quite interesting. My favorite part is always the panel discussions because the interaction drives the conversation to the heart of the matter (both for the microvias panel and the surface reliability panel). We truly have some great experience among the speakers at IPC conferences, and they are so appreciated.

Shaughnessy: I noticed that some of the speakers said that PCB designers have a role in designing boards—vias in particular—to be more reliable. Reliability seems to involve everyone in the supply chain.

Sandy-Smith: Absolutely! Companies that make Class 3 electronics can only be successful when they are detailed about quality at every level of the supply chain—from design to incoming goods quality, monitoring their assembly processes, inspection, and testing of all the goods they produce. It is a major effort and one we all benefit from, for instance, when we're trusting the plane we're flying on to the next conference.

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

Sandy-Smith: Stay tuned for our next conference November 5–7 in Minneapolis, Minnesota—our first IPC Electronics Materials Forum. This conference will have a wide variety of topics, and it's perfect for people who are passionate about materials like I am. As always, we're also planning for IPC APEX EXPO 2020 where you'll see presentations from all facets of the electronics industry with the latest studies and novel strategies for circuit boards and assemblies.

Shaughnessy: Thanks for speaking with me, Brook.

Sandy-Smith: Thank you, Andy. **DESIGN007**

Phononic Devices Could Allow Smaller Mobile Devices

To make modern communications possible, today's mobile devices make use of components that use acoustic waves to filter or delay signals. However, current solutions have limited functionalities that prevent further miniaturization of mobile devices and constrain the available communication bandwidth.

Now, a research team led by Chiara Daraio, Caltech professor of mechanical engineering, has developed phononic devices that could find uses in new kinds of sensors, improved cellphone technologies applied physics, and quantum computing.

The phononic devices include parts that vibrate extremely fast, moving back and forth up to tens of millions of times per second. The team developed these devices by creating silicon nitride drums that are just 90-nm thick. The drums are arranged into grids with different grid patterns having different properties.

These findings open opportunities to design new devices–such as phononic transistors and radio-frequency isolators–based on phonons instead of electrons. Their findings appear in two papers published in *Nature Nanotechnology and Nature.* (Source: California Institute of Technology)

PCB007 Highlights

It's Only Common Sense: The Consultative Sale ▶

The best way to represent yourself and your company as a true expert is to produce and publish valuable content about your products through content marketing. The key word is valuable. The content has to be unbiased (it cannot be a flagrant sales pitch selling your specific product). Rather, it should be general information and best practice advice on your product.

Better to Light a Candle: Chapter Three—First-year Recap of the PCB Fabrication Course at MTU ▶

In the third installment of this series, Marc Carter acknowledges the many organizations and individuals that willingly and freely contributed their time, materials, and support to make this first "prototype" effort a success. This column also gives a sneak preview of some of the efforts underway to expand the efforts at MTU and to start similar grassroots, industry-academia supported programs elsewhere.

DuPont on Materials Challenges and New Opportunities ▶

John Andresakis, senior marketing technologist in the Interconnect Solutions (ICS) Group of DuPont, and Jonathan Weldon, RF applications engineer also in ICS at DuPont, spoke with the I-Connect007 editorial team about trends the company is seeing, what challenges their customers are facing with materials today, and future opportunities with new technologies, including 5G, electric cars, IoT, and more.

Rogers' Advanced Connectivity Solutions Adds N.A. Distributor ▶

Rogers Corporation's Advanced Connectivity Solutions (ACS) business unit announced today the introduction of a new distribution channel with the addition of International Electronic Components (IEC) to their sales and service team in the United States and Canada effective July 8, 2019.

The PCB Norsemen: Merging the Best of Both Worlds–Young Superheroes and Knowledgeable Wizards! ▶

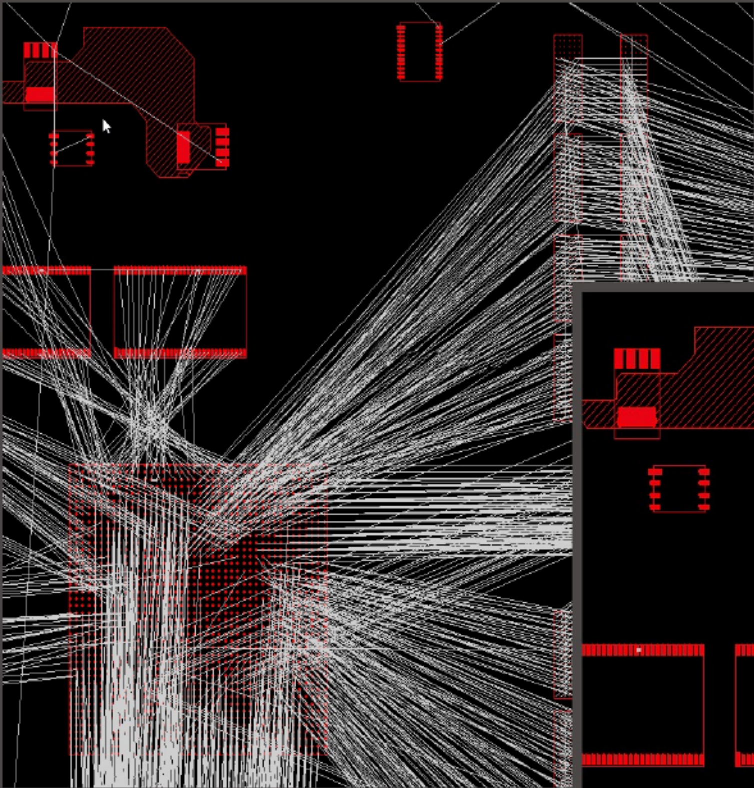
Companies that dare be true to themselves, trust their employees, and provide direction, freedom, and responsibility to their most important asset—namely, their employees—are more likely to succeed. However, we can all rattle behind these positive words and agree with these statements. The real question is, "How do you actually create and sustain an environment that motivates and attracts people—especially millennials—in the wave of Industry 4.0?"

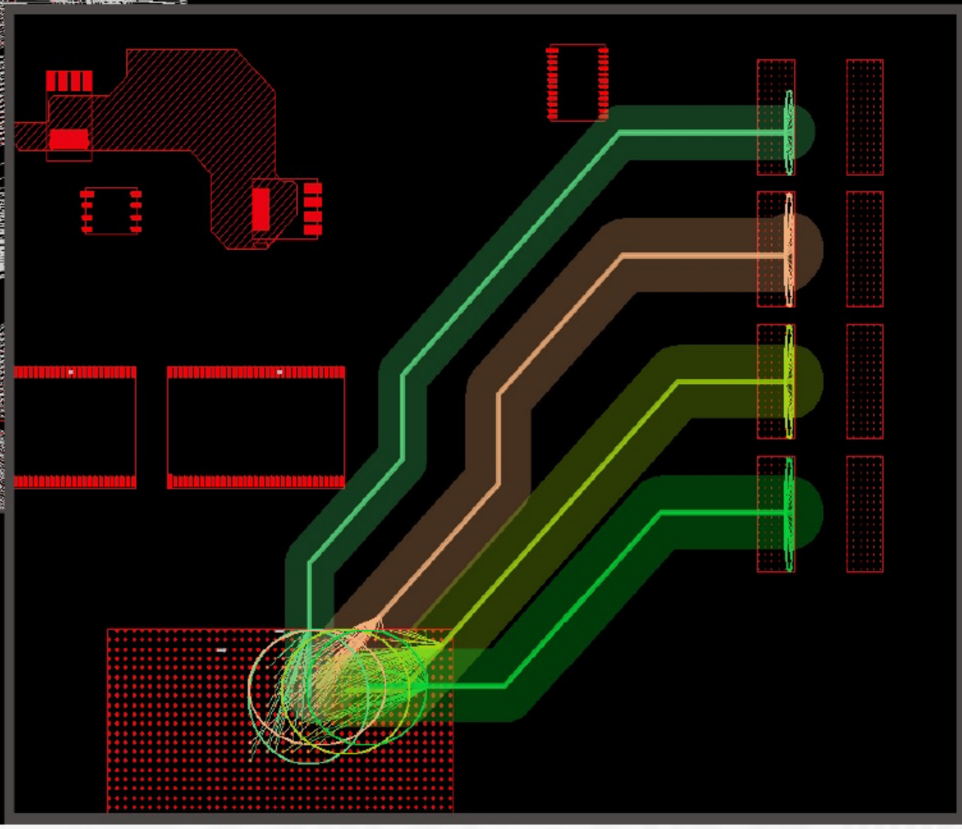
Orbotech Celebrates Success of Orbotech Diamond and Discusses Future Trends ▶

At the recent CPCA Show in Shanghai, Orbotech celebrated having over 100 of their Orbotech Diamond™ direct imaging machines in the marketplace. Barry Matties caught up with Meny Gantz—VP of marketing for Orbotech's PCB division—to talk about the drivers behind the success of Orbotech Diamond systems before turning the conversation toward the future and Industry 4.0.

Printed Circuits Upgrades Drilling With Pluritec ▶

Flex and rigid-flex circuit board manufacturer Printed Circuits has added to their drilling department with high-speed, fully automatic drill/routers from Pluritec.





Stackup Planning, Part 5

Beyond Design

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

Design methodologies change over time, particularly in the ways to simulate electromagnetic fields and return current paths. In my previous column series on stackup planning, I described the traditional stackup structures that use a combination of signal and power/ground planes. But to achieve the next level in stackup design, one needs to not only consider the placement of signal and plane layers in the stackup but also visualize the electromagnetic fields that propagate the signals through the substrate.

The four-part stackup planning column series was published over several months in 2015 in *The PCB Design Magazine:*

- In Part 1 [1], I looked at how the stackup is built, the materials used in construction, and the lamination process. And I set out some basic rules to follow for high-speed design. It is important to keep return paths, crosstalk, and EMI in mind during the design process.
- Part 2 [2] followed with definitions of basic stackups, starting with four and six layers. Of course, this methodology can be used for higher layer count boards (36 layers, 72 layers, and beyond). The virtual materials were replaced with items stocked by the PCB fabricator.
- Part 3 [3] examined higher layer count stackups as the four- and six-layer configurations are not the best choice for high-speed design. In particular, each signal layer should be adjacent to, and closely coupled to, an uninterrupted reference plane, which creates a clear return path and eliminates broadside crosstalk. As the layer count increases, these rules become easier to implement but decisions regarding return current paths become more challenging. More rules for HSD and EMI were also defined.
- In Part 4 [4], I elaborated on 10+ layer counts. The methodology I set out in previous columns can be used to con-

struct higher layer count boards. In general, these boards contain more planes, and therefore, the issues associated with split power planes can usually be avoided. Also, 10+ layers require very thin dielectrics to reduce the total board thickness. This naturally provides tight coupling between the adjacent signal and plane layers, reducing crosstalk and electromagnetic emissions. Additional rules for high-speed design were defined. The number one question about determining the required layer count was also addressed.

In this month's column, I will add to my stackup planning series with this final chapter (Part 5), covering all of the latest concepts in stackup design.

The speed of a digital signal does not depend intrinsically on the speed of electrons but rather on the speed of energy transfer between electronic components. The actual velocity of electrons through a conductor is very slow (~10 mm per second); however, the "knock-on" effect is very fast as it follows the electromagnetic field. The energy propagates between the signal trace and return path(s) as an electromagnetic wave (Figure 1). And the speed of this wave varies depending on the layer in the multilayer substrate and the surrounding dielectric materials. For instance, the wave will travel at approximately half the speed of light in a typical FR-4 material with a dielectric constant (Dk) of four.

The electromagnetic fields of a microstrip (outer) layer are shown in Figure 2. The fields tend to radiate outward, as there is only one solid plane beneath, which blocks the emissions. It is obvious that this configuration is not recommended for routing high-speed single-ended signals. The electromagnetic fields surrounding the microstrip exist partially within the dielectric material(s) and partially within the surrounding air. Since air has a dielectric constant of one, which is always lower than that of FR-4 (typically four), mixing a little air into the equation will speed up the signal propagation. You can see this in the kinks in the field lines as they travel into the air region from the dielectric material. Adding a solder mask (Dk of three) will put another kink in the fields.

A stripline is any trace sandwiched between reference planes on both sides (Figure 3). The electric fields (blue) of a stripline are totally contained between the two solid planes whereas; the magnetic fields (red) are also limited vertically by the planes. Radiation is reduced dramatically and limited to just the edge fringing fields due to the shielding effects of the planes. Signals that travel in the same dielectric material will couple. This may be good for a differential pair, but is not desirable for nonrelated signals as part of the aggressor signal will couple to the victim depending on the separation of traces.

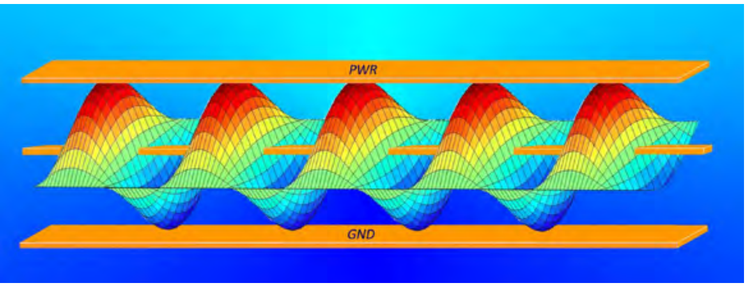


Figure 1: Digital signals travel as a wave of electromagnetic energy in a multilayer PCB.

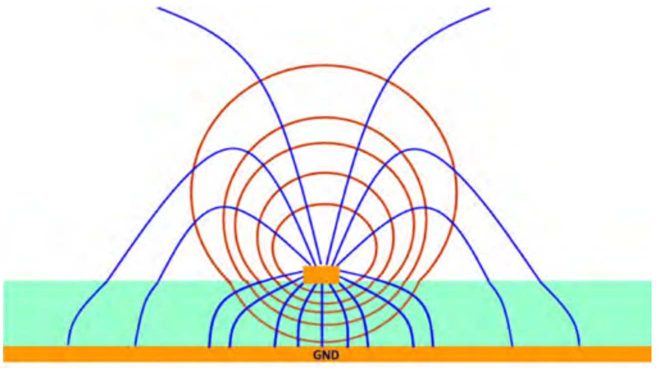


Figure 2: Microstrip electric (blue) and magnetic (red) fields.

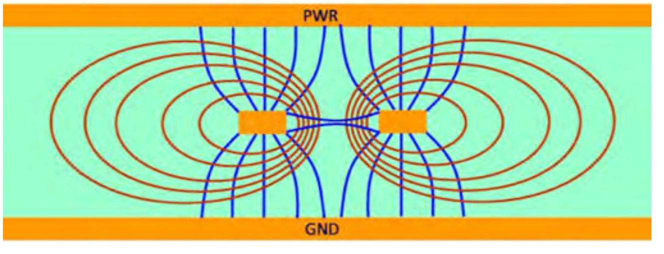


Figure 3: Stripline configuration, differential pair.

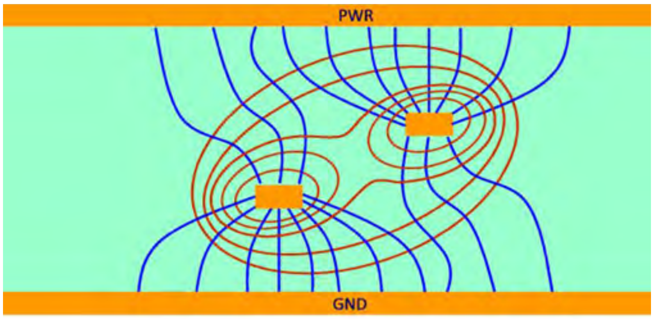


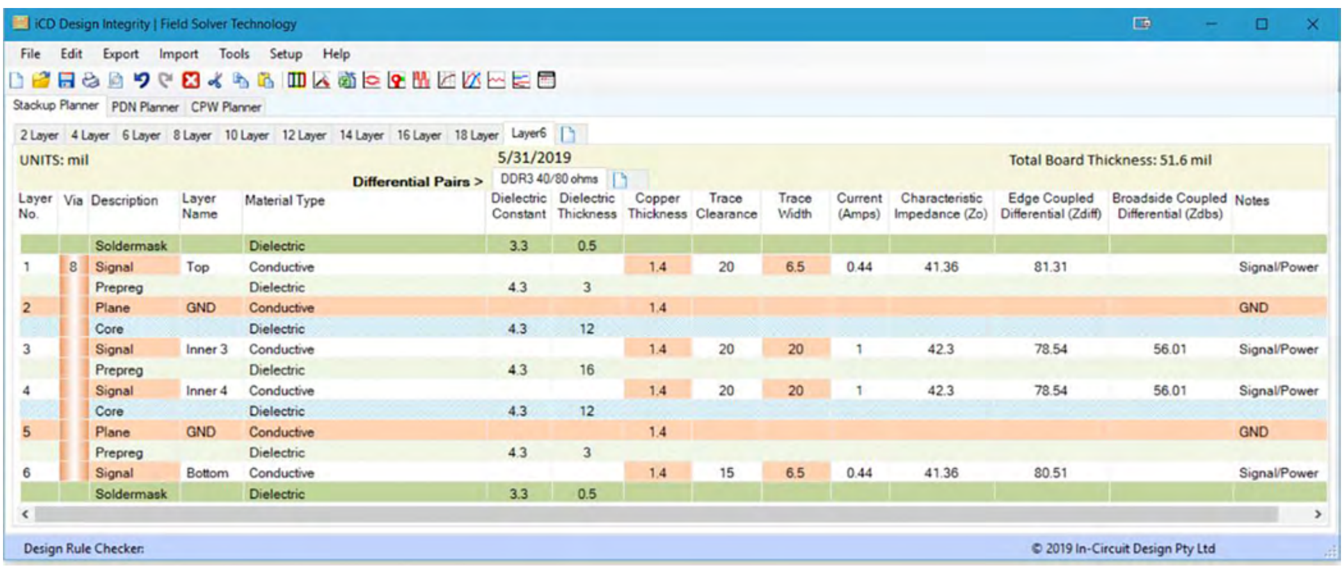
Figure 4: Dual stripline configuration (edge and broadside coupled).

Figure 4 illustrates a dual stripline configuration with a combination of edge and broadside coupling. This occurs when two signal layers are stacked between the planes. Again, this may create crosstalk depending on the trace separation. Crosstalk can be reduced by routing the signal traces orthogonally on adjacent layers reducing the couple to just a small area. However, as frequencies and rise times increase, this is not a good solution.

Fortunately, synchronous buses—as typically used for parallel data signal transfer—benefit from an extraordinary immunity to crosstalk. Crosstalk only occurs when the signals are being switched, and this crosstalk only has an impact within a small window around the moment of the clocking. Providing the receiver waits sufficiently long enough for the crosstalk to settle before sampling the bus, the crosstalk has no impact on the signal quality at the receiver. I typically use the dual-stripline configuration for the DDRx address, command, and control signals, which are far less critical than the data lanes. But generally, each signal layer should be adjacent to—and closely coupled to—an uninterrupted reference plane, which creates a clear return path and eliminates broadside crosstalk.

Figure 5 shows a typical six-layer PCB stackup, but here, I am using the signal layers for mixed-signal/plane pours to eliminate the impact of the electromagnetic fields coupling. These days, it is quite common to have 10 or more power supplies on a board. Rather than allocating one or two per plane, it is best to use the dual stripline layers to provide mixed-signal/power pours. It is a bit hard to visualize this in the spreadsheet format; however, Figure 6 illustrates the point clearly. Layers 3 and 4 can be used for critical signals but are separated by a power pour on either side (L–R: differential pair, broadside-coupled pair, coupling between unrelated signals). This gives the PCB designer flexibility, adds planar capacitance, and provides plenty of room for multiple supplies.

An advantage of using high layer count boards is that power and grounds planes can be placed closely together (2–5 mils) to provide high-frequency planar capacitance. Above



Layer No.	Via	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)	Notes
		Soldermask		Dielectric	3.3	0.5								
1	8	Signal	Top	Conductive			1.4	20	6.5	0.44	41.36	81.31		Signal/Power
		Prepreg		Dielectric	4.3	3								
2		Plane	GND	Conductive			1.4							GND
		Core		Dielectric	4.3	12								
3		Signal	Inner 3	Conductive			1.4	20	20	1	42.3	78.54	56.01	Signal/Power
		Prepreg		Dielectric	4.3	16								
4		Signal	Inner 4	Conductive			1.4	20	20	1	42.3	78.54	56.01	Signal/Power
		Core		Dielectric	4.3	12								
5		Plane	GND	Conductive			1.4							GND
		Prepreg		Dielectric	4.3	3								
6		Signal	Bottom	Conductive			1.4	15	6.5	0.44	41.36	80.51		Signal/Power
		Soldermask		Dielectric	3.3	0.5								

Figure 5: A six-layer stackup top/bottom microstrip and middle dual stripline. (Source: iCD Stackup Planner)

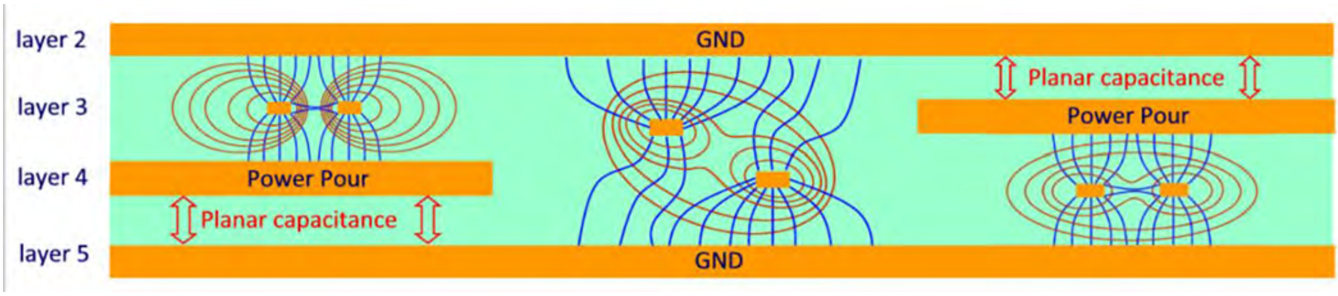


Figure 6: Mixed-signal/power pours on the inner stripline layers.

100 MHz, planar capacitance replaces individual decoupling capacitors. These plane pairs are best positioned directly under a BGA near the (top or bottom) surface rather than in the center of the symmetric stack to reduce via inductance.

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

When the return current flows through the impedance of a cavity between two planes, it generates voltage. Although quite small (typically in the order of 5 mV), the accumulated noise from simultaneous switching devices can become significant. This voltage, emanating from the vicinity of the signal via, injects a propagating wave into the cavity, which can excite the cavity resonances or any other parallel structure (for instance, between copper pours over planes). Other signal vias also passing through this cavity can pick up this transient voltage as crosstalk. Plane pairs should be coupled closely together to dampen this cavity resonance and to provide high planar capacitance.

To design the perfect stackup, one needs to understand how and where the electromagnetic fields propagate and where the current return paths flow through the substrate. Placing power supplies as copper pours allows the designer to make full use of the planar capacitance and to isolate critical signals within the substrate.

Key Points

- The speed of a digital signal does not depend intrinsically on the speed of electrons but rather on the speed of energy transfer between electronic components
- The energy propagates between the signal trace and the return path(s) as an electromagnetic wave
- Microstrip (outer) layer EM fields tend to radiate outward as there is only one solid plane beneath, which blocks the emissions
- The electric fields of a stripline are totally contained between the two solid planes whereas the magnetic fields are also limited vertically by the planes
- Radiation is reduced dramatically and is limited to just the edge fringing fields due to the shielding effects of the planes
- Stripline crosstalk can be reduced by routing the signal traces orthogonally on adjacent layers, reducing the couple to just a small area
- For synchronous buses, crosstalk only occurs when the signals are being switched and this crosstalk only has an impact within a small window around the moment of the clocking
- Each signal layer should be adjacent to—and closely coupled to—an uninterrupted reference plane, which creates a clear return path and eliminates broadside crosstalk
- Rather than allocating one or two supplies per plane, it is best to use the dual stripline layers to provide mixed-signal/power pours

- Power and ground planes can be placed closely together to provide high-frequency planar capacitance in high layer count boards
- Return current can also excite the parallel plate resonance mode, causing significant electromagnetic radiation from the fringing fields
- Plane pairs should be coupled closely together to dampen the cavity resonance and provide high planar capacitance

References

1. Olney, B. "Beyond Design: Stackup Planning, Part 1," *The PCB Design Magazine,* June 2015.
2. Olney, B. "Beyond Design: Stackup Planning, Part 2," *The PCB Design Magazine,* July 2015.
3. Olney, B. "Beyond Design: Stackup Planning, Part 3," *The PCB Design Magazine,* August 2015.
4. Olney, B. "Beyond Design: Stackup Planning, Part 4," *The PCB Design Magazine,* October 2015.

Further Reading

- Olney, B. "Beyond Design: Plane Cavity Resonance," *The PCB Design Magazine,* September 2017.
- Olney, B. "Beyond Design: Faster Than a Speeding Bullet," *The PCB Design Magazine,* February 2016.
- Olney, B. "Beyond Design: Signal Flight Time Variances in Multilayer PCBs," *The PCB Design Magazine,* December 2017.
- Olney, B. "Beyond Design: Crosstalk Margins," *Design007 Magazine,* July 2018.
- Olney, B. "Beyond Design: The Proximity Effect," *Design007 Magazine,* March 2019.
- Johnson, H. W., & Graham, M. *High-Speed Digital Design: A Handbook of Black Magic,* Prentice Hall, 1993.
- Ott, H. W. *Electromagnetic Compatibility Engineering,* Wiley, 2019.

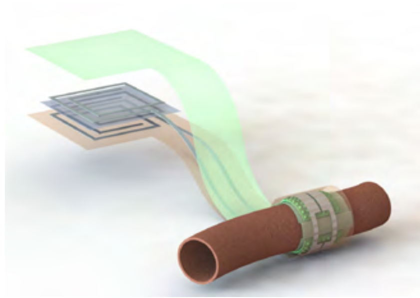
Editor's note: All numbered figures are attributed to Barry Olney.

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

Stanford Researchers Develop Wireless, Battery-free, Biodegradable Blood Flow Sensor

A new device developed by Stanford University researchers could make it easier for doctors to monitor the success of blood vessel surgery. The sensor, detailed in a paper published in *Nature Biomedical Engineering,* monitors the flow of blood through an artery. It is biodegradable, battery-free and wireless, so it is compact, doesn't need to be removed, and it can warn if there is a blockage.

"Measurement of blood flow is critical in many medical specialties, so a wireless biodegradable sensor could impact multiple fields including vascular, transplant, reconstructive and cardiac surgery," said Paige Fox, assistant professor of surgery and co-senior author of the paper.



Monitoring the success of surgery on blood vessels is challenging as the first sign of trouble often comes too late. By that time, the patient often needs additional surgery that carries risks similar to the original procedure. This new sensor could let doctors keep tabs on a healing vessel from afar, creating opportunities for earlier interventions.

The sensor wraps snugly around the healing vessel, where blood pulsing past pushes on its inner surface. As the shape of that surface changes, it alters the sensor's capacity to store electric charge, which doctors can detect remotely from a device located near the skin but outside the body. In the future, this device could come in the form of a stick-on patch or be integrated into a wearable device or smartphone.

(Source: Taylor Kubota, Stanford University)

New and Thriving Chapters in Mexico

The Digital Layout

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

This month's column highlights the Monterrey Designers Council Chapter located in Monterrey, Mexico. The chapter was established November 23, 2017, which makes it a younger chapter compared to several other global chapters currently in existence today. I also share news about a new chapter forming in Nogales, Mexico.

Luis Saracho is the president of the Monterrey Chapter and an IPC Certified Advanced PCB Designer (CID+) with a true passion for designing board layouts, not to mention leading and organizing chapter meetings. This chapter has yet to establish a solid leadership team, so Luis is basically running the show on his own there and has been doing an awesome job keeping this chapter active and successful. Their chapter meets quarterly (four times a year) with an average attendance of 12–18 attendees. Chapter meetings are typically held in the evening at a local university (ITESM Campus Monterrey), and most meetings consist of a technical presentation. The chapter is looking into getting government support through CANIETI—an organization that supports technical initiatives—but have had poor results to date.

Chapter Spotlight: Monterrey

by Luis Saracho, CID+
CHAPTER PRESIDENT

Our recent chapter meeting was held Thursday, May 30, and went well overall. There were around 20–25 attendees from different workplaces (several industry fields were represented, including automotive, industrial, lighting, manufacturing of electrical households, etc.) who shared their experiences with analog/digital/power grounding tech-

A past Monterrey Chapter meeting (Luis Saracho on the far left).

APCT

Passion | Commitment | Trust

Printed Circuit Board Solutions

There is always pressure to beat the competition to market.
Prove out the prototype; proceed quickly to production, assemble and ship.

APCT Global offers a solution to build and verify the prototype;
transfer the PCB recipe seamlessly offshore; and deliver a reliable product within weeks!
No pressure, just dependable execution.

Who better to manage your Offshore Programs than PCB experts?

Let Us Build You a Solution

Contact APCT Global Today | 203.284.1215

WHY CHOOSE APCT GLOBAL

ADVANTAGES
- Ease-of-Use
- Seamless Recipe Transfer
- Inventory Management
- Risk Mitigation

STRATEGIC REGIONS
- Japan
- South Korea
- Vietnam
- Taiwan
- Northern China
- Southern China

DISTRIBUTION SITES
- Austin, TX
- Hong Kong
- Wallingford, CT
- Santa Clara, CA
- Orange County, CA

APCT.com

APCT Santa Clara (Corporate 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

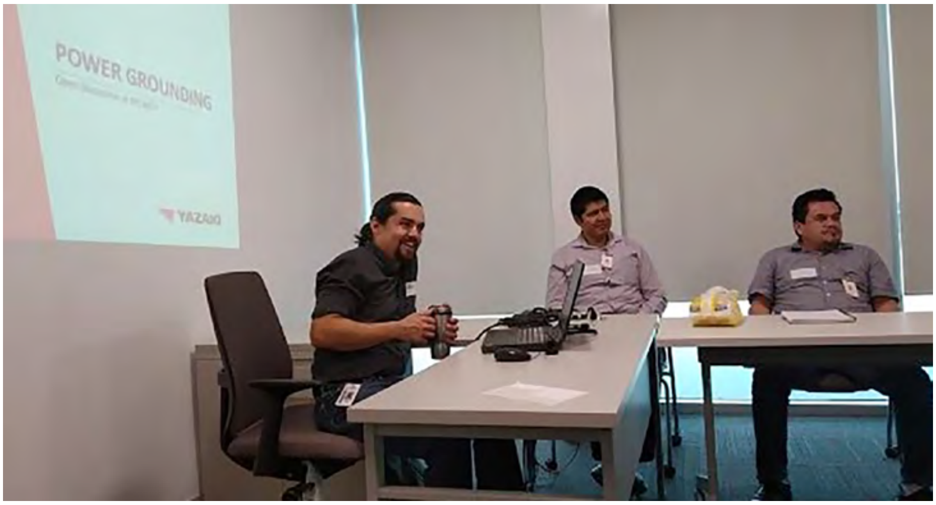


Figure 1: Technical discussion on power grounding.

niques on printed boards (Figure 1). Areas of conversation included:

- Keeping only one ground plane
- Splitting planes but keeping them connected at a single point
- Pouring with ground the signal layer as ground guards
- Keeping separated ground potentials

At the end of the meeting, the conclusion from the open technical discussion was that there is no recipe that fits for all of the product diversity that people work on. For a graphics controller, 10-layer, mid-power board design rules and considerations cannot be followed on a double-layer, high-power board design for industrial applications.

The only recommendations agreed upon by everyone were to keep the loop between power and its return as small as your layout allows and avoid crossing loops, referring to a wide range of design options. But considering the diverse audience, it was very interesting to hear where they had designs flaw on EMC/EMI and what the engineers did to correct those flaws. You can't apply the same specific rules to different application and technology designs. The magic relies on identifying the aggressive and sensitive loops, and if simulation or testing shows a flaw, understanding the root cause of it. This is not an easy task for printed board engineers.

If you want to contact me, email luis.saracho@gmail.com. Learn more about their chapter here.

Figure 2: May 2019 Monterrey Chapter meeting.

A New Designers Council Chapter

The Monterrey DC Chapter will no longer be the only chapter located in Mexico. In a recent IPC CID certification class that I taught at the Chamberlain Group located in Nogales, Mexico, the spirit of comradery, success, and knowledge sharing was truly overwhelming—so much so that the entire group made up of EEs wanted more than what one would typically get from a four-day IPC CID certification class. They wanted to get involved" and become part of the industry IPC collective, especially regarding PCBs/CCAs. They also wanted to represent their country in the best possible way as the next wave of EEs.

The group is led by Heriberto (Heri) Alanis who is an extremely knowledgeable EE and very strong industry veteran with over 30 years of experience. Heri is also an excellent mentor for many of the young engineers within Chamberlain Group. He will have a strong support network to establish this new chapter. His core members, who will form the leadership of this new chapter and are all recent achievers of their IPC CID certification, are as follow:

- Rodrigo Martin del Campo Alcocer
- Roberto Ivan Villalba Gonzalez
- Jacob Ivan Castellanos Juarez
- Carlos Arturo Garcia Gamez
- Rafael Antonio Gomez Sandoval
- Marcos Eduardo Chavez Manon

This new chapter from Nogales, Mexico, will be the spotlight for next month's column as they establish their presence in the global IPC Designers Council community of other local chapters.

2019 Training and Certification Schedule

IPC Certified Interconnect Designer (CID)

- August 6–9: Baltimore, MD
- August 26–29: Markham, ON
- September 6–9: Santa Clara, CA
- September 19–22: Schaumburg, IL
- October 8-11: Carmel, IN
- October 21–24: Anaheim, CA
- November 2–5: Raleigh, NC
- November 5–8: Dallas, TX

IPC Advanced Certified Interconnect Designer CID+

- September 6–9: Santa Clara, CA
- September 10–13: Kirkland, WA
- September 17–20: Schaumburg, IL
- October 21–24: Anaheim, CA
- November 2–5: Raleigh, NC
- December 3–6: Manchester, NH

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

PCB Design Events

PCB West 2019

- September 9–11: Santa Clara, CA

AltiumLive 2019

- October 9–11: San Diego, CA

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

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

Five Best Practices to Ensure Manufacturability

Connect the Dots

by Bob Tise, SUNSTONE CIRCUITS

Optimizing your board using design for manufacturability (DFM) techniques just makes sense. But without formal training, this isn't as straightforward as one would hope. Every day, our team receives designs that are too complex, too thick, incomplete, with components placed too close together, among other things.

When something like this happens, rejection of your design is a best-case scenario because it means your manufacturing partner has your back. If your design is just dropped into an automated queue without concern for manufacturability, quality, or functionality, the output can be a batch of unusable boards.

This doesn't have to happen because you are in control of your vendor choice. While not a replacement for good design practice, carefully picking a manufacturing partner and service level that meet your needs will help you achieve a quality result.

Now, let's examine five design best practices that will help reduce costs, increase yield, and improve manufacturability.

1. Always Double Check Your Design

It will save you time on the front end. Regardless of which partner or service level you choose, I strongly encourage you to be sure to review your design, taking a second look for issues, such as insufficient power trace widths or blind vias. Ditto if components are laid out too close together. It is easy to accidentally cut holes or route slots too close to pads or traces.

Copper thickness is another biggie. Higher current may require thicker copper and narrow traces probably should be thinner, so be specific about what you need to ensure your design will not fail. Check land patterns against the part supplier's manufacturer data sheets (MDS) as well. Often, that's all it takes to avoid common design issues.

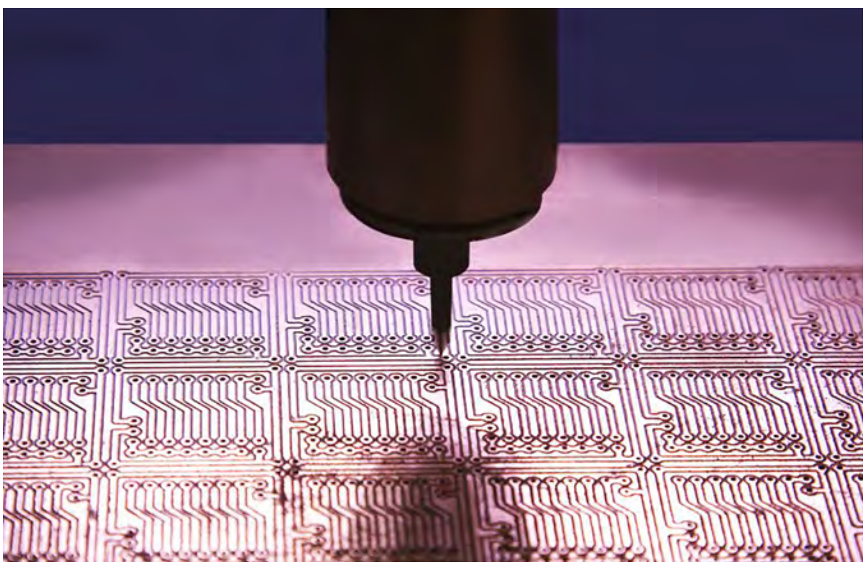


Figure 1: PCB production.

2. Always Consider Your Environment

When you send your design for manufacturing, your partner does not know what type of device the board will be part of nor the conditions in which it will have to perform. It's common for harsh environments or exposure to mess up a board's performance. If you call out materials that will not tolerate the end-product's operating environment, bad things can happen—such as a smoking board,

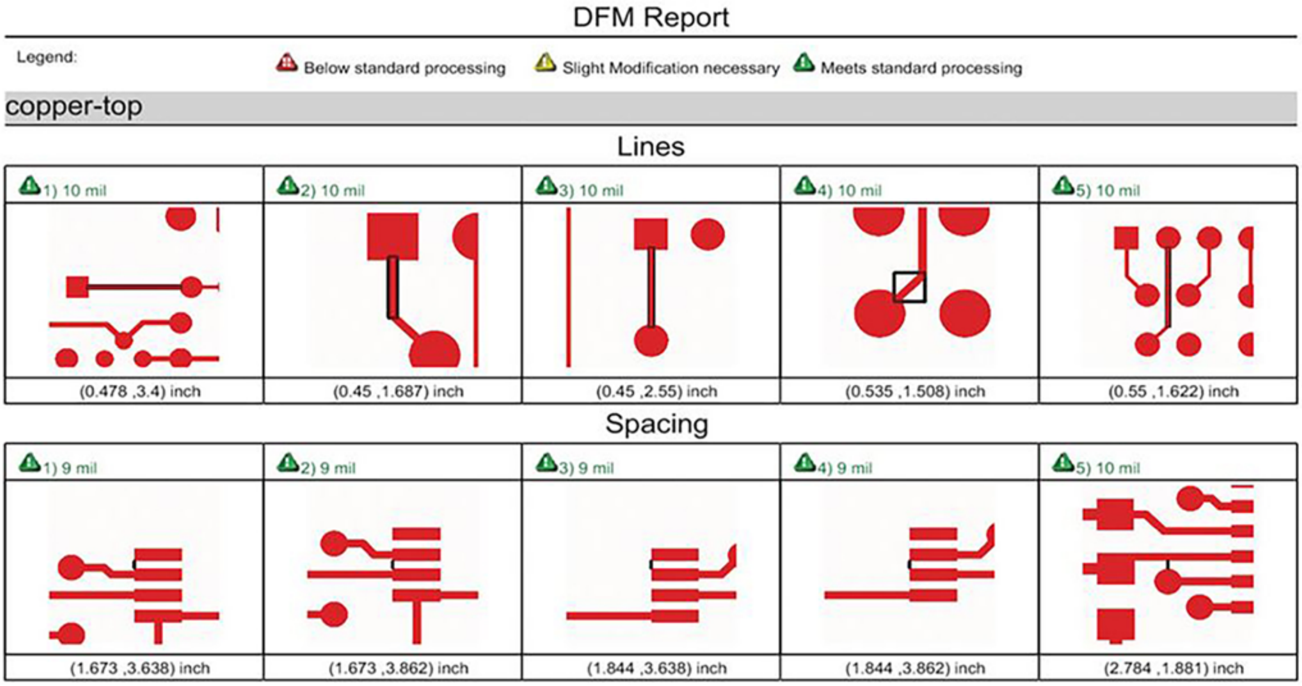


Figure 2: DFM errors.

for example. Be sure your board can tolerate thermal stress or solder joints risk breaking and damaging components.

3. Pay Attention to the Design Rules

Not every manufacturer prioritizes making DFM rule sets for popular design tools available, cost-effective, and up to date. Working with a manufacturer who does will save time and money in the end. Using design rule checks (DRC) alone will not guarantee manufacturability, but performing them is a great way to ensure that you have created as close to an error-free design as possible.

An error-free design is like a unicorn—it's something you can see out of the corner of your eye, but never quite catch. The goal should be a design that lowers the risk of defects and can then be manufactured and assembled. The DRC will help you identify conflicts—such as issues with insufficient clearances between electrical conductors—that will affect electrical functions and create potential manufacturability issues.

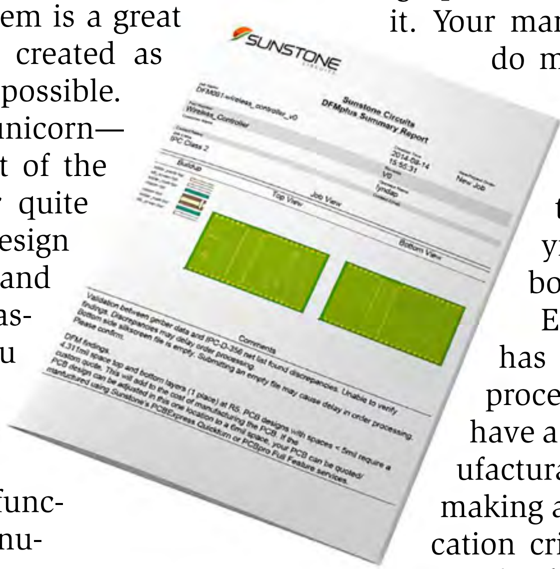


Figure 3: DFMplus report.

4. Adhere to the Principles of DFM

I recommend integrating all of the above into a design process that looks forward to the manufacturing process. Finding issues earlier makes them cheaper and easier to fix. From the outset, perform all design tasks while considering yield and other manufacturing issues that affect cost and quality.

A key element of DFM is communicating with your manufacturing partner at the beginning of the design process and regularly throughout it. Your manufacturing partners can do more than just find problems. Each has valuable insight into how design decisions can impact the manufacturability, yield, and quality of the boards.

Each manufacturer also has unique capabilities and process requirements that can have a large impact on the manufacturability of your design, making an open line of communication critical. For example, your supplier(s) can confirm that you

are using up-to-date footprint files throughout the design process.

Highly manufacturable designs exist within a range of specifications. Good manufacturers can help you tweak your design to make the best use of range or wiggle room. This kind of designer-manufacturer communication about DFM can help save time, reduce costs, and even improve the functionality of your board.

5. Make Sure Help Is There When You Need It

Choosing a manufacturing partner with readily available support staff can help you solve manufacturability puzzles during the design process rather than after submission. It's useful to ask a few questions about any potential manufacturing partner before deciding, including:

- Do they make it easy to get competent technical support?
- How quickly will they respond to your needs?
- How can you get support for time-sensitive issues other than by email?
- Are they available when you need them?
- What is their escalation policy for difficult support requests?
- Is support free, or is it pay-per-incident?

Conclusion

The real key to optimizing the manufacturability of your PCB designs is to choose tools, processes, and a manufacturing partner that meet your needs. Treat your parts suppliers, PCB manufacturer, and PCB fabricator as members of your design team, prioritizing open and persistent collaboration with each. This—along with adherence to your own DFM-focused process—will ensure board quality, manufacturability, and cost-effectiveness. **DESIGN007**

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

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

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

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

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

Why is this method able to produce stable zigzag nanographene? The explanation lies in the fact that the product crystallizes directly even during synthesis. In their solid state, the molecules are not in contact with oxygen. In solution, however, oxidation causes the structures to disintegrate quickly.

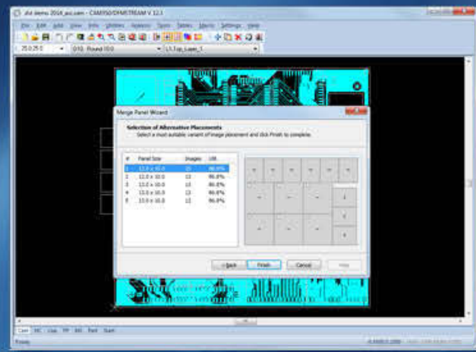
This approach allows scientists to produce large pieces of graphene while maintaining control over their shape and periphery. This breakthrough in graphene research means that scientists should soon be able to produce and research a variety of interesting nanographene structures—a crucial step towards finally being able to use the material in nanoelectronic components.

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

Solutions to Ensure PCB Manufacturing Success!

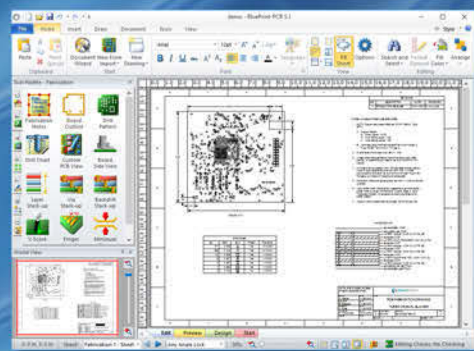
CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing.



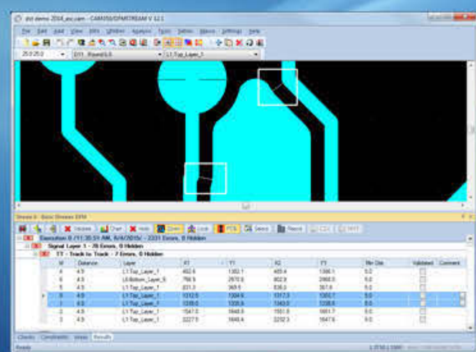
BluePrint·PCB

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



DFMStream™

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



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

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

DownStream Technologies

Customer Support Is People Support

Tim's Takeaways

by Tim Haag, CONSULTANT

A couple of months ago, I ordered a T-shirt with a silly phrase for my son Peter's birthday. Since my son lives 2,000 miles away, we often order gifts and ship them to each other. In this particular case, however, I noticed that after filling out Peter's home address for shipping, there wasn't anywhere to add a billing address. To make sure that there wasn't any confusion between the shipping and billing addresses, I sent them a quick email to clarify the situation. I told them who I was, what I was doing, and the order number.

Within a couple of hours, I received a response that said, "Hello, Peter." It went on to thank me for my order without answering my billing address question. Their confusion over my name gave me some real concern now that the order was mixed up, so I wrote back again and told them very clearly, "This is Tim, not Peter. I'm the one who ordered this gift, and I'm trying to send it to Peter." Once again, they responded with, "Hello, Peter. We apologize for the trouble you had placing your order." At this point, I gave up and let Peter know that there may or may not be a gift delivered to him. Fortunately, it all worked out. In a few days, Peter received the gift, my credit card was charged the correct amount, and all was well.

I found the whole thing to be kind of funny, and my wife got a kick out of referring to me as "Peter" for a while. As amusing as it was, though, it did make me realize once again how important good customer service is for productivity in business. In the case of the confused-name T-shirt order mystery, my email was probably routed through a system that included templates for generic responses. Whoever handled it probably didn't even read my message where I said that I wasn't Peter, and instead, replied with a standard automated response. For all I know, my emails may not even have been handled by the same company that

ships the shirts. Companies like this deal with scores of messages about problems every day, and I was probably just another one of those in a large customer complaint pile. And that's the focal point of this whole story.

Even though the shirt was delivered as promised, the thing about this transaction that I am going to remember the most is that they couldn't get my name right. It's not their product, their price, or their service, but instead, it will be calling me by the wrong name that will stay in my memory. You may be thinking at this moment that I'm making a mountain out of a molehill, and I would agree, but there's more here to my line of thinking than a $20 shirt.

What if instead of a shirt I was waiting for an updated board outline from my mechanical design department? And what if instead of calling me by the wrong name, they neglected to notify me that the new data was ready to be worked with? When it comes to getting the PCB design done correctly and out the door on time, how we work with and support those around us becomes much more important than a wrong name on a T-shirt order.

All of us are in the business of customer support no matter what we do. Even if you aren't actively operating a customer support line or answering emails, someone is still depending on you for their success. Do you have deliverables to finish on time so that someone else can do their job? Do you have responsibilities to fulfill so that your department functions smoothly? Are you part of a team that provides a product or service for your company? If so, you're in customer support. Whatever it is that we do, we are usually working with others, and their success depends on how well we do our own jobs. Although we may not be providing support in the traditional sense of customer support, we are still providing support—even if the customer is a co-worker sitting right next to us.

Not communicating clearly, following through, or even using the wrong name can send a very subtle message to a co-worker that we don't place too much value on them. This perception of decreased value can then tend to reduce the amount of trust that our co-workers have in us, and on it goes, becoming worse and worse. Once this cycle starts, interdepartmental communication will break down, leading to even more problems. The breakdown of value and trust in our work relationships is like adding resistance to a circuit; both communication and the flow of energy will be choked off. I'm not saying that just because we forget someone's name that the company is doomed to failure, but I believe that for success, we should all look for better ways to support each other in our efforts.

Here are four ideas to provide better support, build trust, and communicate with our co-workers that they do have value and are important to us.

1. Listen

Our co-workers and the people around us all have great ideas, some of which might be vitally important to the success of the project that we are working on. Yet it is so easy to get fixed on what we want to say that we don't listen to what these other ideas are. Try focusing your complete attention on your co-worker, and listen all the way through before responding.

2. Reply Promptly

When you receive a message—whether it is a voicemail, email, chat, or text—respond back as soon as you can. Not responding quickly to a message sends a very clear signal to the sender that their attempt to reach out wasn't important to you. Now, here's the trick; it's okay if your answer is "no," or "let me get back to you when I know more." The important thing is to get back to them with some kind of response so that they know you aren't ignoring them.

3. Be Encouraging

Our society today is very used to the sharp and biting comments that are prevalent on social media. People are becoming so used to being knocked down for their ideas that in return, they are starting to keep their ideas bottled up. Start encouraging your co-workers as a way to reverse this trend. You don't want to go overboard and sound phony, but finding something simple that we can encourage each other with will build up self-esteem and go a long ways towards improving communication.

4. Use the Right Name

I just couldn't resist, but there is more to this than you might think. I've been in situations where someone's name has been mispronounced or a nickname used that grates on that person. I've seen those people close up and not respond, and a seed of bitterness gets planted that has a lasting impact. Our names are important; they matter.

Building a healthy work environment with all of our co-workers is every bit as important as our capabilities and skills in PCB design. I've been in work situations where I've been isolated either because of distance or work culture. On the other hand, I've been in places that have pushed me to excel by building me up with trust and support. I can tell you honestly that I've gotten a lot more accomplished in the latter work environments than I ever have in the former. Because of this, I'm a big advocate of always looking for ways to support and build each other up. The end results of these efforts promote better communication and clear the way for the open development of new ideas and concepts that are vital to our success.

Remember, we are all in the business of supporting our customers. Whether that means actual customers who purchase the products we design, our management who signs our paychecks, or our co-workers who depend on us to supply them with what they need to do their jobs. We should all keep looking for better ways to build up and support those that we work with. And yes, it can start with something as simple as using someone's correct name.

Until next time then, keep on supporting and designing. **DESIGN007**

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

A Flexible Future

Xiaojing Hao, associate professor and Scientia Fellow in the School of Photovoltaic and Renewable Energy Engineering at the University of New South Wales (UNSW), and her team are building thin, flexible solar panels made of kesterite photovoltaics. Containing copper, zinc, tin, and sulfur, they are cheaper to make and more environmentally friendly than other thin-film varieties on the market, some of which contain toxic materials, such as cadmium, or rare elements, such as indium and tellurium.

There are significant industrial applications for flexible, thin photovoltaic cells. But a key challenge has been their limited capacity to generate energy with comparable efficiency and cost to that of conventional silicon solar panels.

In 2017, Hao's team built a kesterite photovoltaic cell that attained 11% efficiency. This set a world record—a fourth for the team—and was the first time that the 10% efficiency threshold was broken for this type of cell. This work was published in *Nature Energy*.

"For each efficiency change, we need a step-change technology to make it happen," says Hao. She hopes to improve the efficiency of kesterite photovoltaic cells to close to 20% in the next five years. To do this, her team needs to work out how to prevent the common defects in the lattice of the kesterite material. The reasons for these defects are not yet well understood. However, once corrected, the electrical quality of the material will improve.

Exciting findings from 2018 point to some new directions to control the defects and improve the performance of kesterite cells. The plan is to apply for Australian Research Council funding to delve deeper and publish some of these results this year.

(Source: UNSW)

Conformal Coatings: How to Design Out Production Problems

Sensible Design

by Phil Kinner, ELECTROLUBE

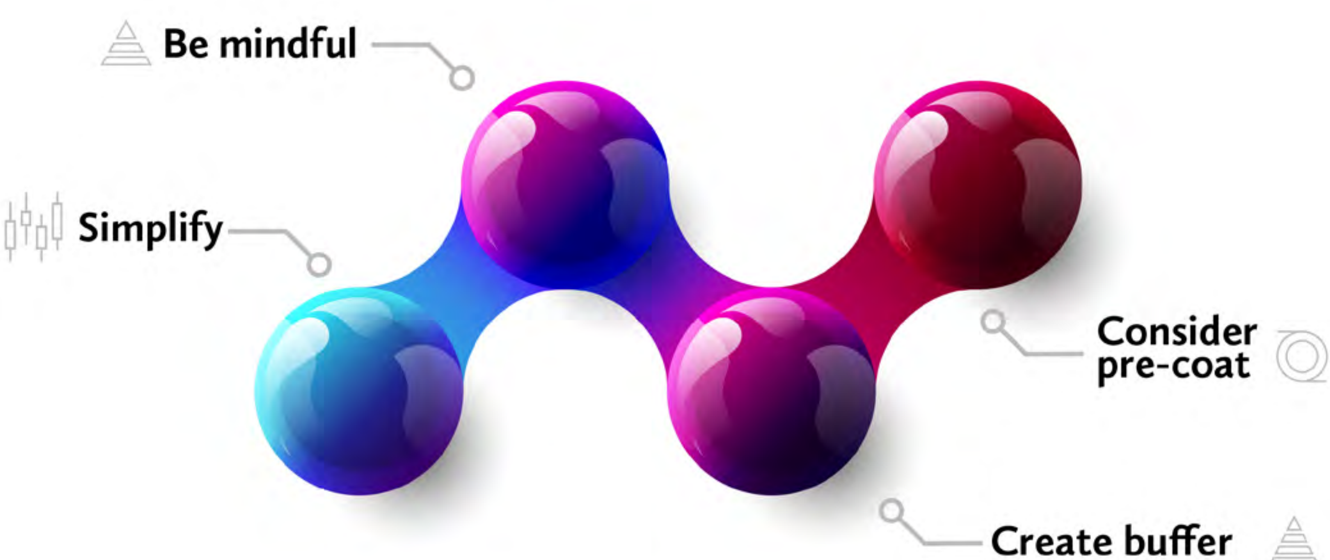
In my last few columns, I've covered quite a bit of ground regarding the important considerations for conformal coating selection and performance, and the suitability of conformal coatings for LEDs and protecting circuitry from the harshest environments. I hope these columns have provided plenty of food for thought as well as given you a basic understanding of coatings and their benefits and limitations. In this column, I'm going to look at the different angles that design engineers and purchasing professionals come from and explore how these can sometimes conflict when selecting conformal coatings.

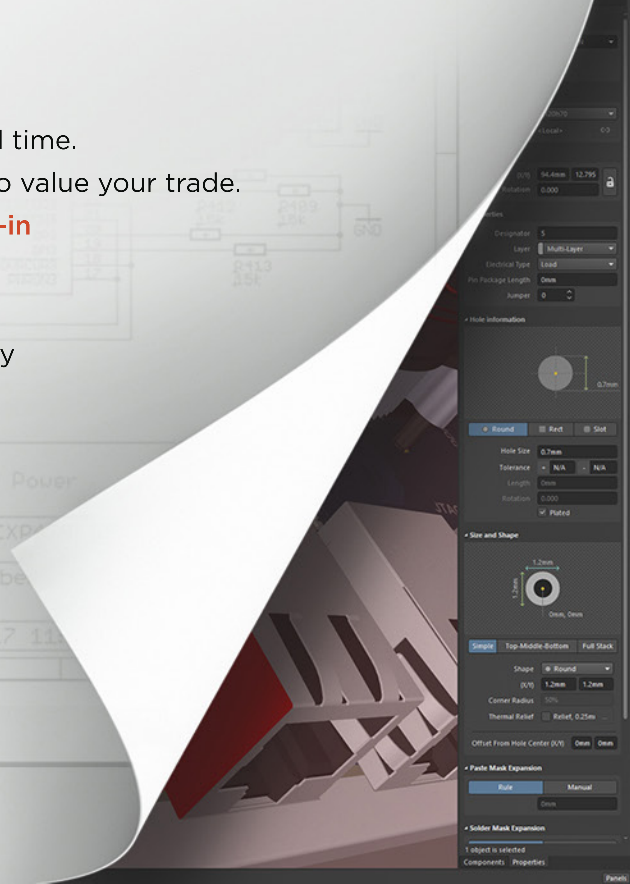
It is an area we are often confronted with at Electrolube. However, if experienced production personnel are reviewing the designer's suggestions at an early enough stage, they could potentially prevent the perpetuation of some common problems. Thoughtful design will always pay huge dividends down the line, and designers will have friends for life among your production colleagues if you make their jobs just that little bit easier!

In an ideal world, the design engineer would work closely with either production engineers or the industrial engineers who oversee the coating process in the factory. Identifying potential production problems at the design stage will always be preferable and far easier than trying to fix problems or concerns following finalization of the engineering drawings. Here are four factors to consider when designing out production issues with coatings:

1. Simplify the Board Layout

By the simple act of placing connectors and components that must not be coated along one edge of the assembly, the conformal coating application process will be simplified. This might





allow dip coating to be explored as a potential alternative methodology, speeding up application times, and reducing costs. Also, avoid large arrays of discrete components, which can pose a huge coating challenge due to the high levels of capillary forces present. The net result is often areas of no coverage/protection on the board as well as areas of excessive thickness prone to stress-cracking, delamination, and other coating defects. Similarly, tall components present challenges of their own by the creation of shadowed or hard-to-reach areas. Splashing is another associated problem. The trick is to avoid placing tall components next to must-coat components to avoid this eventuality.

2. Be Mindful of the Processes That Can Impact the Coating

The designer should ideally be aware of what kinds of manufacturing practices may occur following the application and cure of the coating because other materials, such as thermal greases/putties and rework/repair chemicals, can all have an impact on the integrity and overall performance of a coating. Also, when selecting adhesives for assemblies, care should always be taken to ensure that they are compatible with the selected coating materials and processes. Adhesives that are not compatible can have a detrimental effect on the overall performance of the coating.

3. Consider the Importance of Pre-coat Cleaning

It is worth bearing in mind that choosing the most appropriate conformal coating is merely the first stage in the protective process. The condition of the assembly before coating is an often-overlooked part of the protective process. The designer should always consider the process steps leading up to conformal coating, and a key element of this is cleaning. The overall cleanliness of the substrate or the potential presence of residues on the substrate (such as no-clean flux residues) can have a critical impact on coating performance. If the substrate is not adequately clean, the residues present may interfere with the curing mechanism, lead to poor adhesion of the coating to the substrate, and trap conductive/ionic materials under the coating.

Without meticulous attention to preparation or pre-coat cleaning regimes, corrosive residues bridging the PCB's conducting tracks can cause failures over time. Whilst the coating may delay failure for many years, at some point, failure is still inevitable. Contaminants present on the surface before conformal coating will be sealed in by the operation and may cause long-term problems. Such contaminants might include fingerprints, flux residues and moisture, and other atmospheric pollutants. Boards should always be cleaned and dried before conformal coating to obtain optimum long-term performance. Even when using so-called no-clean fluxes, cleaning boards before coating will usually improve long-term performance and reliability.

4. Create a Buffer to Make Coverage Easier for Production

Conformal coatings are usually liquid when applied and will flow with a combination of gravity and the capillary forces present. Conformal coating materials will simply go where they are placed with some naturally occurring wetting/spreading to a greater or lesser extent, which is a function of the material's surface tension, thixotropic index, and drying time. Most materials will tend to slump away from sharp edges of the component, leads, and solder joints due to gravity. This behaviour can be made worse by longer dry times and also by baking if the initial viscosity drop is greater than the increase due to solvent evaporation.

Understanding and controlling this behaviour, and its effect on the conformal coating coverage, will be key to the performance of coated assemblies operating in harsh environments. Whether the process specifies masking or reliance upon the process of selective conformal coating, a production team will be greatly relieved if you leave a buffer (or a don't-care area) of at least 3 mm clear between the areas that must be coated and those that must not. This small buffer will make the production process easier and prevent future issues in production.

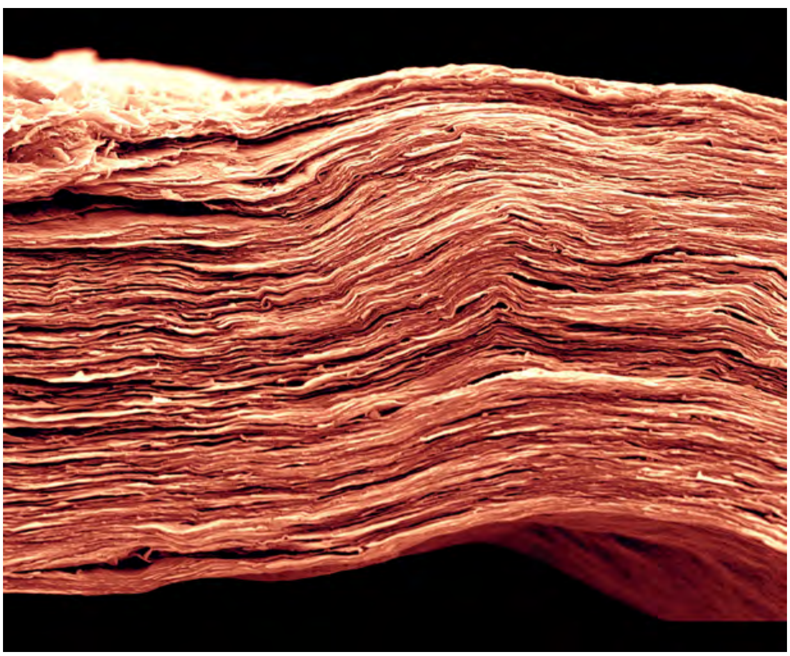
Having started the conversation about the importance of making sound early-stage design decisions, it is important to understand "what effects what" on the surface of the board to ensure successful conformal coating is achieved. Implementing these lessons will prevent potential production disasters not only with conformal coatings but also in other areas of production. In a nutshell, choose the right material for the protection required, apply it appropriately, and cure it well. Check for interactions with other process chemistries, thoroughly clean and dry the assembly before coating, and ultimately, you will establish a solid and reliable electronic assembly process. **DESIGN007**

Phil Kinner is the global business and technical director of conformal coatings at Electrolube. To read past columns or contact Kinner, click here. Kinner is also the author of *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*. Visit I-007eBooks.com to download this and other free educational titles.

Right Electrolyte Doubles Material's Ability to Store Energy

Scientists at the Department of Energy's Oak Ridge National Laboratory, Drexel University, and their partners have discovered a way to improve the energy density of promising energy-storage materials with conductive, two-dimensional ceramics called MXenes. The findings are published in *Nature Energy*.

Today's batteries offer high energy-storage capacity, but slow charging speeds limit their application in consumer electronics and electric vehicles. Tomorrow's energy-storage mainstays may be supercapacitors, which store charge at the surface of their electrode material for fast charging and discharging. However, they currently lack the energy density of batteries.



"The energy storage community is conservative, using the same few electrolyte solvents for all supercapacitors," said principal investigator Yury Gogotsi, a Drexel University professor who planned the study with his postdoctoral researcher Xuehang Wang. "New electrode materials like MXenes require electrolyte solvents that match their chemistry and properties."

The surfaces of different MXenes can be covered with diverse terminal groups, including oxygen, fluorine, or hydroxyl species, which interact strongly and specifically with different solvents and dissolved salts in the electrolyte. A good electrolyte solvent-electrode match may then increase charging speed or boost storage capacity.

"Our study showed that the energy density of supercapacitors based on two-dimensional MXene materials can be significantly increased by choosing the appropriate solvent for the electrolyte," added co-author Lukas Vlcek of the University of Tennessee who conducts research in UT and ORNL's Joint Institute for Computational Sciences. "By simply changing the solvent, we can double the charge storage."

The work was part of the Fluid Interface Reactions, Structures, and Transport (FIRST) Center—an Energy Frontier Research Center led by ORNL and supported by the DOE Office of Science.

(Source: Oak Ridge National Laboratory)

MilAero007 Highlights

Standard of Excellence: Three Ways to Face the Future With Your PCB Suppliers

Once you have established a solid, trusting relationship with your PCB vendor, you can start working together developing new products, technologies, and in some cases, services. There are even times when your PCB supplier will bring projects to you. Here are three ways that you can work with your PCB suppliers to face the future.

What Is Reliability Without Traceability?

High reliability and compliance are hot topics at conferences all over the world. If you are a supplier to industries like defense, automotive, medical, and aerospace/space, high-reliability and regulatory compliance are strict demands for electronic device manufacturers.

NextFlex Launches $10.5M Funding Round for Anti-counterfeiting, Flex Batteries, and Hypersonics Innovation

The PC 5.0 total project value is expected to exceed $10.5 million (project value/investment figures include cost sharing), bringing the total anticipated investment in advancing flexible hybrid electronics since NextFlex's formation to over $83.5M.

U.S. House Approves Measure to PromoteLead-free R&D in Milaero, Automotive, Medical

IPC is applauding the U.S. House of Representatives for approving a measure that would promote research and development into the performance of lead-free electronics in high-reliability sectors, such as aerospace, defense, automotive, and medical equipment.

Second Lockheed Martin-Built GPS III Satellite Ready for July 25 Liftoff

The GPS satellite constellation is about to get its next heathy dose of new technology and more advanced capabilities. The second next-generation, Lockheed Martin-built GPS III satellite—nick-named "Magellan" by the U.S. Air Force—is sealed up and ready for its planned July 25 launch.

Rogers Launches TC350 Plus for Higher-power Microwave, Industrial Heating

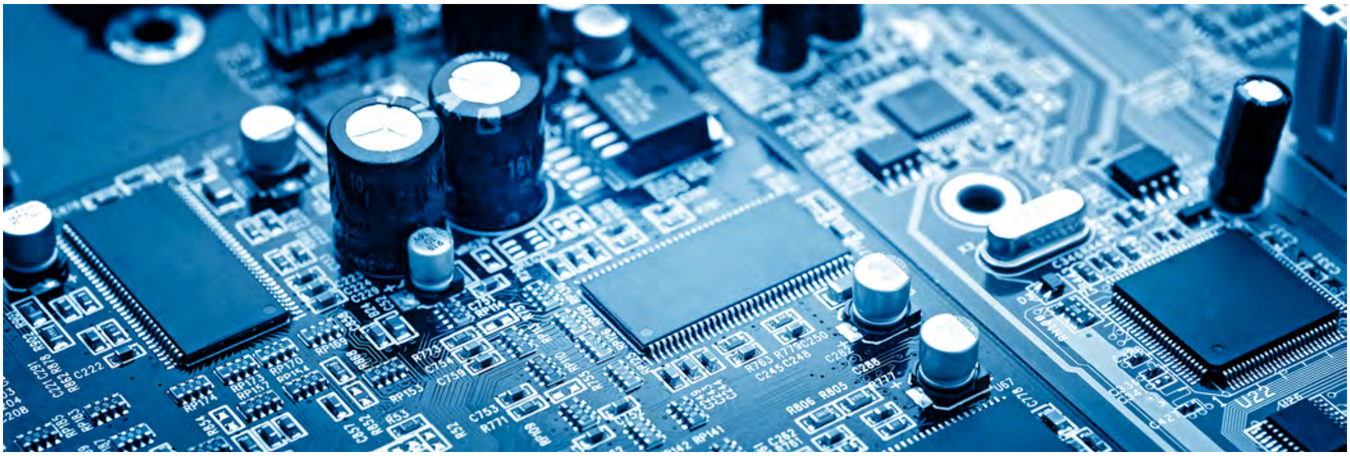
Rogers Corporation introduces TC350 Plus laminates. TC350 Plus laminates are ceramic-filled, PTFE-based, woven glass-reinforced composite materials providing a cost-effective, high-performance, thermally enhanced material for the circuit designer.

Angeles Equity Partners Acquires APCT

Angeles Equity Partners LLC, a private investment firm focused on value creation through operational transformation, announced the acquisition of APCT Inc., a leading manufacturer of highly reliable, quick-turn printed circuit board (PCB) prototypes with global production management capabilities.

Growing Military Demand to Boost SecurityRobots Market at 8.9% CAGR Until 2025

As per the Transparency Market Research report, the global security robots market is expected to be worth nearly $3.9 billion by the end of 2025. The market stood at $1.9 billion in 2016. It also states that the market shall exhibit an impressive 8.9% CAGR during the forecast period of 2017 to 2025.



The Impact of Inductance on Impedance of Decoupling Capacitors

Article by Chang Fei Yee
KEYSIGHT TECHNOLOGIES

This article discusses the impact of interconnection inductance on the impedance of the decoupling capacitor, which influences the power integrity of the PCB. The investigation is performed with 3DEM simulation by varying the trace length and height of stitching vias that connect the decoupling capacitor across the power rail and ground.

On a PCB, a power distribution network (PDN) with low impedance across the wideband is required to transfer power with low switching noise and high stability from the supply to the digital and analog ICs. Each decoupling capacitor—together with its interconnection inductance—are the major factors that contribute to the impedance of the PDN on a PCB. As shown in the cross-sectional view of the PCB depicted in Figure 1, interconnection inductance is formed by the traces and stitching vias hooking up the decoupling capacitor across the power rail and ground (e.g., Loop 1, Loop 2, and Loop 3). This parasitic inductance is directly proportional to the stitching via height and trace length, as governed by Equations 1 and 2, respectively.

Furthermore, referring to the directly proportional relationship between impedance and in-

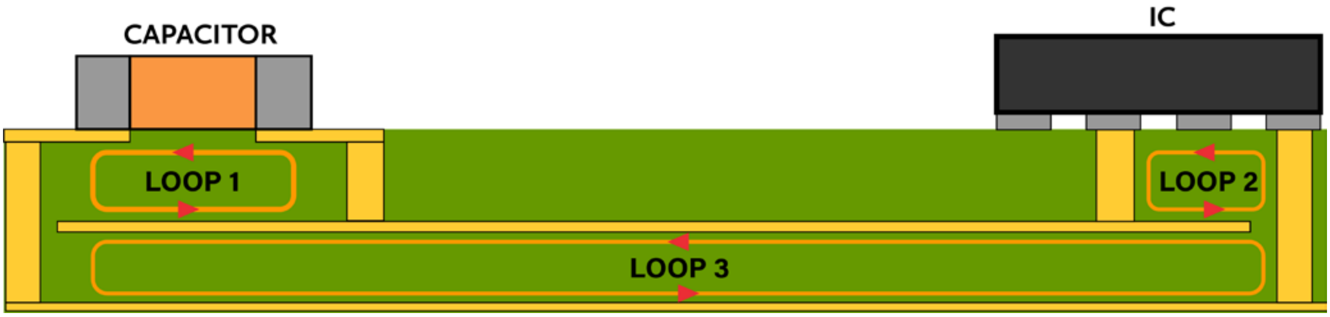


Figure 1: Interconnection inductance associated with a decoupling capacitor.

IT-968 / IT-968 SE

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

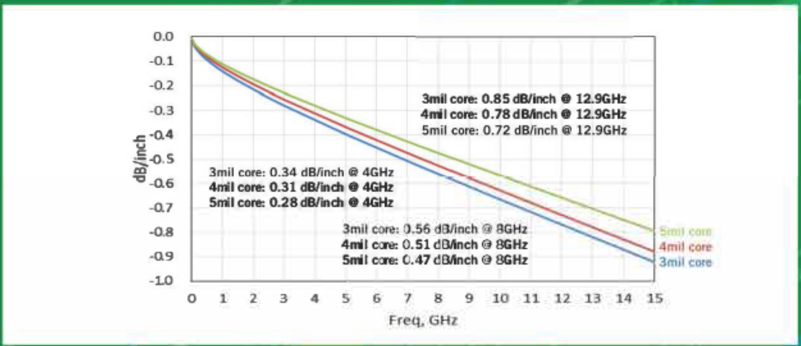
IT-968	IT-968 SE
Dk @ 10 GHz - 3.85	Dk @ 10 GHz - 3.30
Df @ 10 GHz - 0.0043	Df @ 10 GHz - 0.0019

Insertion Loss - Measured

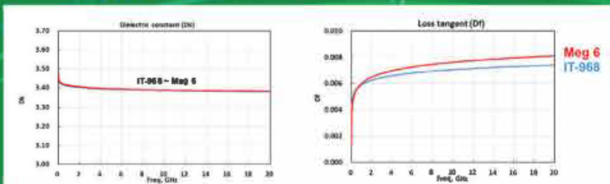
Industry Test Vehicle

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

IT-968 Loss Performance



IT-968 SPP Comparison



GHz	10		12.9		16		20	
Parameter	Df	Dk	Df	Dk	Df	Dk	Df	Dk
IT-968	0.0071	3.39	0.0072	3.39	0.0073	3.38	0.0074	3.38
M6	0.0076	3.39	0.0078	3.39	0.0079	3.38	0.0081	3.38
%	7.95	0.04	8.52	0.04	8.98	0.03	9.45	0.02

Ultra Low Loss IT-968 SE

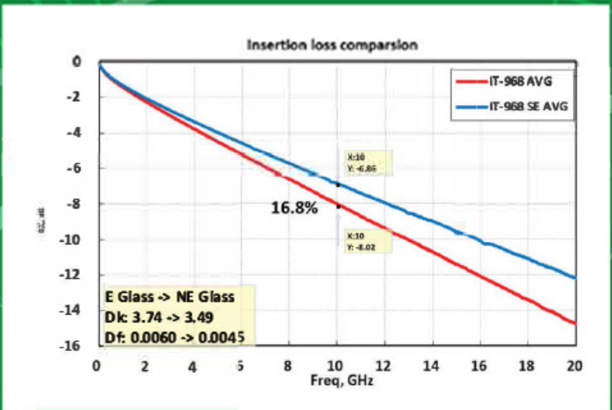
Features
High Tg, low CTE, High Thermal Reliability with Ultra Low Df < .005. Designed for High Speed Applications.

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

Feature *High Speed , 25Gbs/path Solution*

Property	Test Method	IT-968	IT-968 SE
Tg (℃)	DSC	185	185
T-288 (w/ 1 Oz Cu, min)	TMA	120+	120+
Td-5%(℃)	TGA 5% loss	390+	390+
CTE (%), 50-260℃	TMA	2.2	2.2
Peel strength (lb/inch)	1 oz	6	6
Water Absorption	D-24/23	< 0.1	< 0.1
Dk, 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	3.4 3.4-3.7	3.4 3.4-3.7
Df, 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	0.0032 0.0036-0.0050	0.0028 0.0031-0.0040

E-Glass vs Low Dk Glass



ITEQ

terconnection inductance in Equations 3 and 4, it is crucial to keep the interconnection inductance low to minimize the impedance of the PDN, which is achievable by reducing trace length and stitching via height.

$$L \approx \frac{h}{5}[1 + \ln(\frac{4h}{d})]$$ Equation 1:

- L = parasitic inductance of stitching via (unit nH)
- h = stitching via height (unit mm)
- d = via hole diameter (unit mm)

$$L \approx 2\,l\,[ln(\frac{5.98\,h}{0.8\,w+t})]$$ Equation 2:

- L = parasitic inductance of interconnection trace (unit nH)
- l = trace length (cm)
- w = trace width (cm)
- h = dielectric substrate thickness (cm)
- t = trace thickness (cm)

$$X_L = 2\,\pi\,f\,L$$ Equation 3:

$$Z \sim X_L$$ Equation 4:

- X_L = inductive reactance (unit ohm)
- Z = impedance (unit ohm)
- f = frequency (unit GHz)
- L = interconnection inductance (nH)

The analysis of the impact of interconnection inductance due to stitching via height and trace length is discussed in section A and B, respectively, in this article.

A. Impact of Stitching Via Height

To investigate the impact of interconnection inductance due to stitching via height on the impedance of decoupling capacitor, three simulation models are constructed using Keysight EMPro. In model 1A (Figure 2), a 0201 package decoupling capacitor with two terminal pads separated 10 mils apart (10 mils by 5 mils per pad) is constructed. This 0.1-uF capacitor is set to have an intrinsic parasitic ESR of 0.005 ohms and an ESL of 0.5 nH. One of its terminal pads is connected to the middle of a 50-mil fixture trace (representing the power rail) with trace length, l, of 20 mils. Meanwhile, its second terminal pad is hooked up to the ground plane on layer 2 of the PCB with trace length, l, of 20 mils and stitching via height, h, of 2.65 mils. The 1-oz. top layer of copper is separated from the ground plane on layer 2 using low-loss dielectric material also with a height, h, of 2.65 mils. Subsequently, two-port shunt probing points are positioned

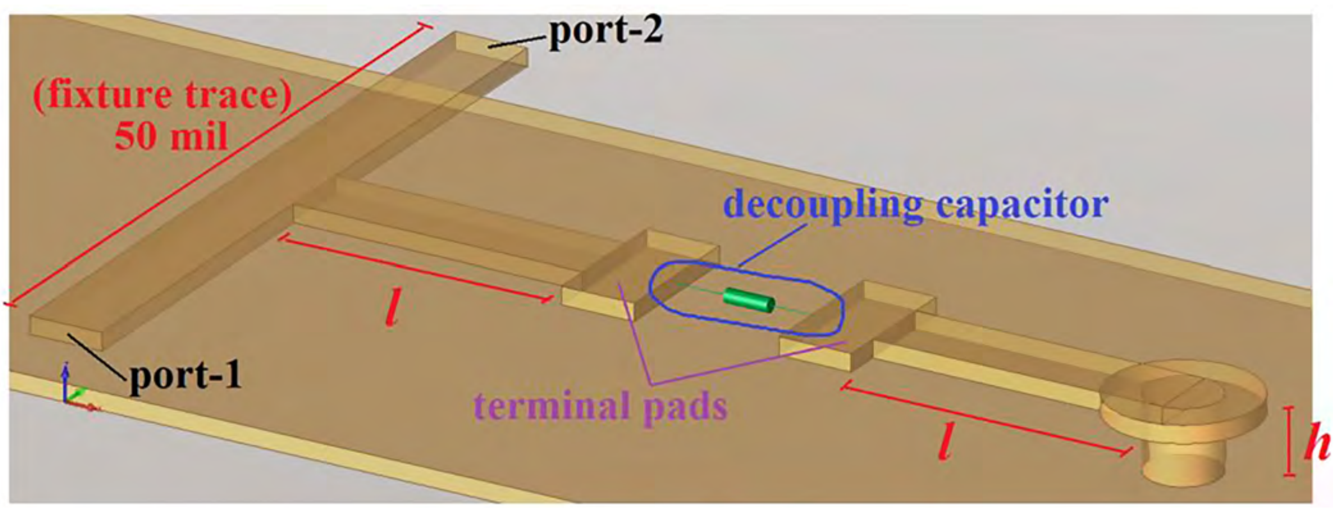


Figure 2: 3DEM structure of model 1A together with 50-mil fixture trace.

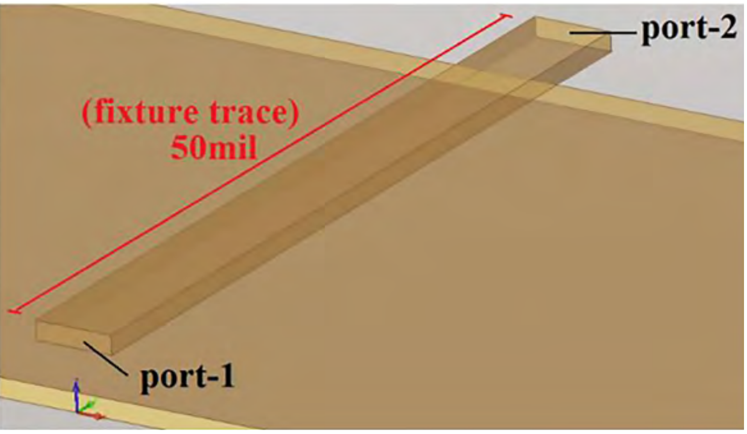


Figure 3: 3DEM structure of the 50-mil fixture trace to be de-embedded.

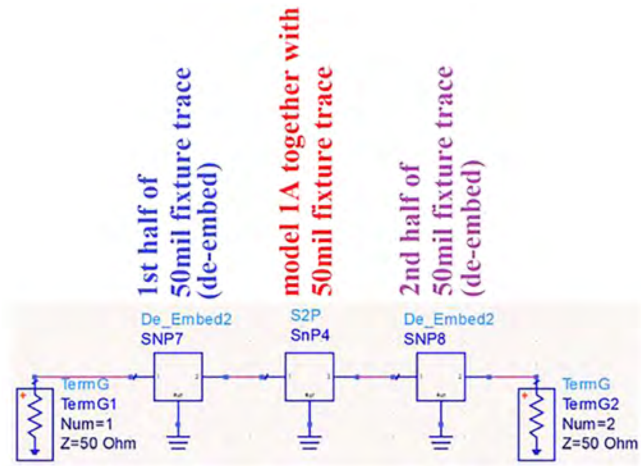


Figure 5: Topology to de-embed fixture trace in Keysight ADS.

at both ends of the 50-mil fixture trace to generate the S21 parameter.

The S21 parameter of model 1A consists of the effect of interconnection inductance (i.e., point of interest) together with the 50-mil fixture trace (for the purpose of two-port shunt probing). To de-embed or remove the effect of the fixture trace, the standalone trace depicted in Figure 3 with the same dimension and height as in Figure 2 is modeled to generate its S21 parameter. The S21 parameter of the fixture trace is then split into two halves using the automatic fixture removal (AFR) as shown in Figure 4. Subsequently, S21 parameters of the two halves generated by PLTS-AFR and model 1A are imported to Keysight ADS to perform de-embedding (Figure 5). The de-embedded S21 parameter that comprises the effect of decoupling capacitor and its interconnection inductance only is converted to impedance using Equation 5.

$$Z = 25 * S21$$ Equation 5:

(Z = impedance of decoupling capacitor and interconnection inductance)

The modeling and de-embedding procedures are repeated for model 1B and 1C with stitching via height, h, of 7.95 mils and 13.95 mils, respectively. With reference to impedance plots shown in Figure 6, the three models experience the same resonance frequency of 103 MHz. Across the wideband range, model 1A

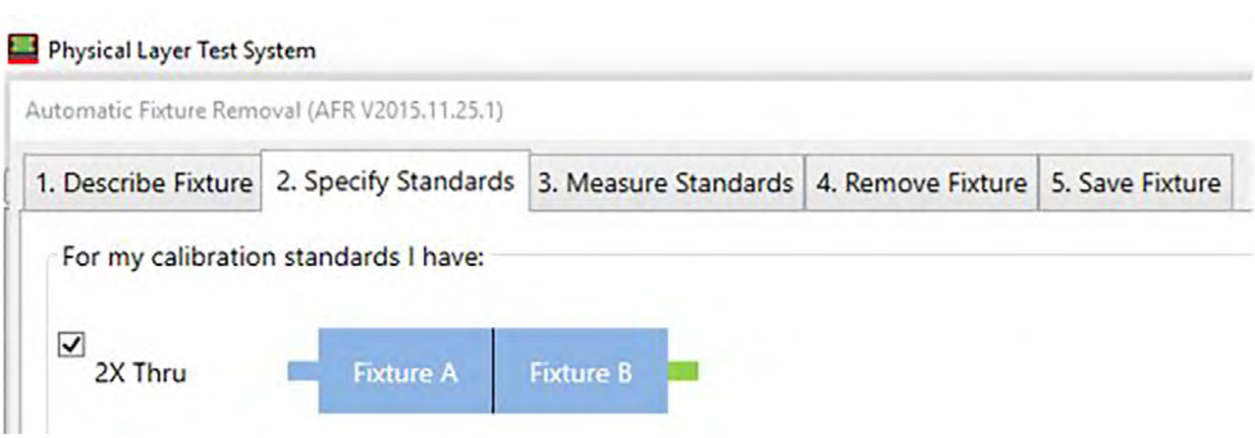


Figure 4. The S21 of the fixture trace is shown split into two halves.

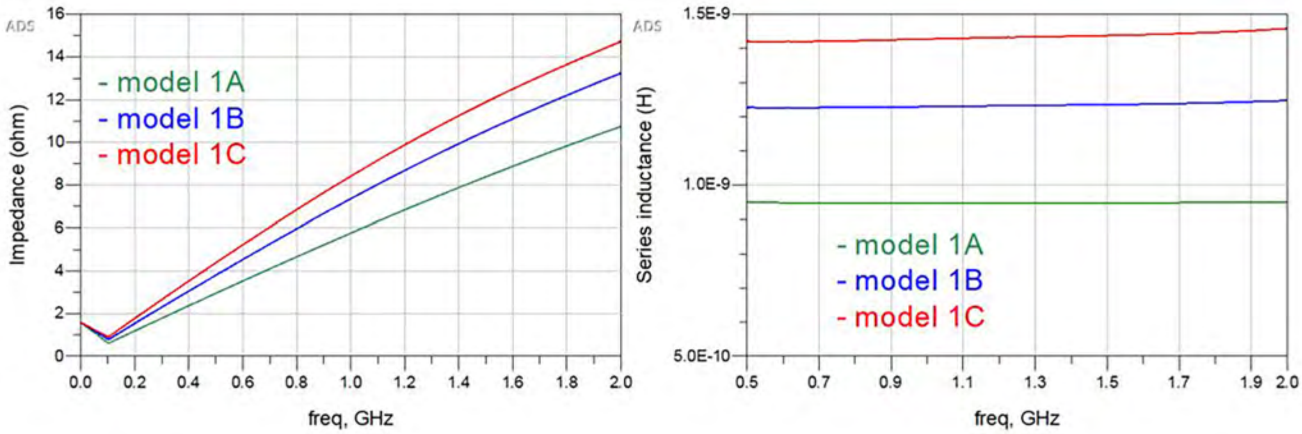


Figure 6: Plots of impedance (L) and series inductance (R) for models with varying stitching via heights after de-embedding.

has the shortest via height (2.65 mils)—thus the lowest interconnection inductance (a total series inductance of 0.95 nH)—and incurs the lowest impedance. At 1 GHz, a further increase of via height to 7.95 mils in model 1B worsens the impedance by 1.5ohm versus model 1A. Model 1C with the highest via height of 13.95 mils and largest interconnection inductance (a total series inductance of 1.42 nH) incurs additional 1-ohm impedance at 1 GHz compared to the other two models.

B. Impact of Trace Length

To investigate the effect of interconnection inductance due to trace length on the impedance of decoupling capacitors, the modeling and de-embedding steps are repeated for model 1D and 1E with trace length, l, at each end of the capacitor set as 10 mils and 0 mils, respectively. Models 1A, 1D, and 1E have the same stitching via height, h, of 2.65 mils. A 3DEM structure of model 1E with a 50-mil fixture trace is shown in Figure 7.

With reference to impedance plots shown in Figure 8, the three models experience the same resonance frequency—103 MHz. Across the wideband range, model 1E with no interconnect trace but with via-in-pad and thus the lowest interconnection inductance (total series inductance of 0.68 nH) experiences the low-

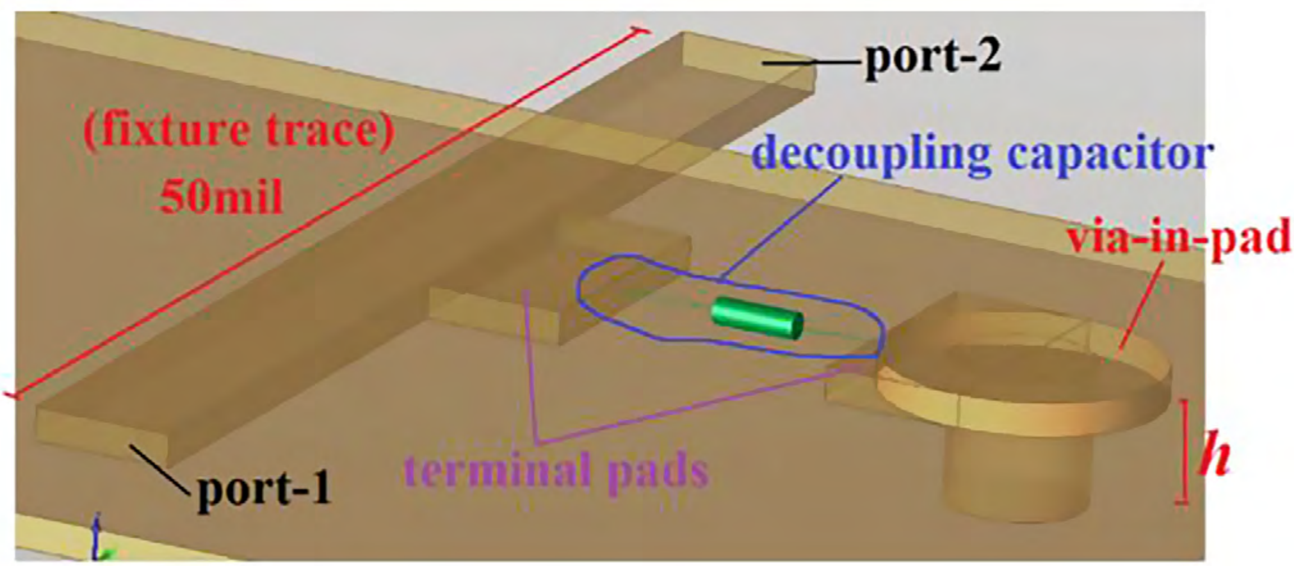


Figure 7: A 3DEM structure of model 1E together with a 50-mil fixture trace.

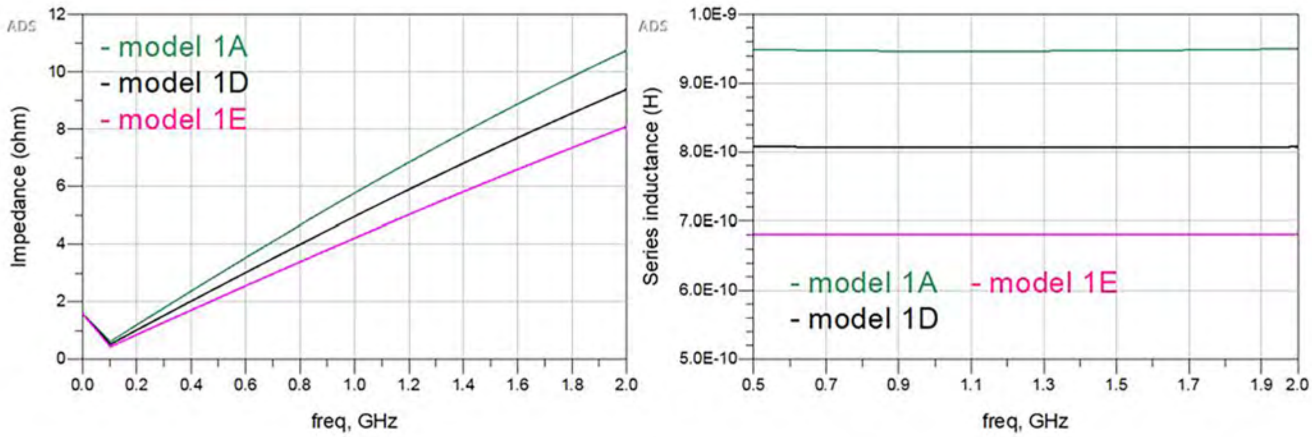


Figure 8. Plots of impedance (L) and series inductance (R) for models with varying trace length after de-embedding.

est impedance. At 1 GHz, a further increase of trace length to 10 mils in model 1D worsens the impedance by 1 ohm versus model 1E. Model 1A with the longest trace length of 20 mils and largest interconnection inductance (a total series inductance of 0.95nH) experiences additional 1-ohm impedance at 1 GHz compared to the other two models.

Summary

It is crucial to minimize interconnection inductance to achieve minimal impedance in the decoupling capacitor and the PDN. Via-in-pad shall be applied as an interconnection medium to nullify the effects of trace length. Furthermore, power and ground planes shall be assigned closer to outer PCB layers that feature decoupling capacitors to reduce stitching via height. **DESIGN007**

References

1. "High-speed PCB Layout Techniques," Texas Instruments.

2. F. Carrio, V. Gonzalez, & E. Sanchis, "Basic Concepts of Power Distribution Network Design for High-speed Transmission," *The Open Optics Journal*, 5, pp. 51-61, 2011.

3. R. Keim, "PCB Layout Tips and Tricks: Minimizing Decoupling Inductance," All About Circuits, July 26, 2018.

4. "Estimating the Connection Inductance of a Decoupling Capacitor," Learn EMC.

5. "Ultra-Low Impedance Measurements Using 2-Port Measurements," Keysight Technologies.

6. "Auto Fixture Removal (AFR)," Keysight Technologies.

Chang Fei Yee is a hardware engineer with Keysight Technologies. His responsibilities include embedded system hardware development, and signal and power integrity analysis.

Wrap-around Sensors for the Grid

Scientists at Oak Ridge National Laboratory have developed a low-cost, printed, flexible sensor that can wrap around power cables to precisely monitor electrical loads from household appliances to support grid operations.

Using an inkjet printer, researchers deposited wires on a flexible plastic substrate, then wove in a magnetic strip to channel the flux produced by an electric current, making the sensor suitable to install in tight spaces. When tested on conductors in the lab and on a building HVAC unit, the sensor measured responses of up to 90A of electrical current and is expected to exceed 500A in larger applications.

"These inexpensive sensors provide crucial, real-time usage data needed to monitor and control devices, such as smart HVAC and water heaters for better power grid efficiency and resilience," said Pooran Joshi, a senior scientist in the Materials Science and Technology Division at ORNL.

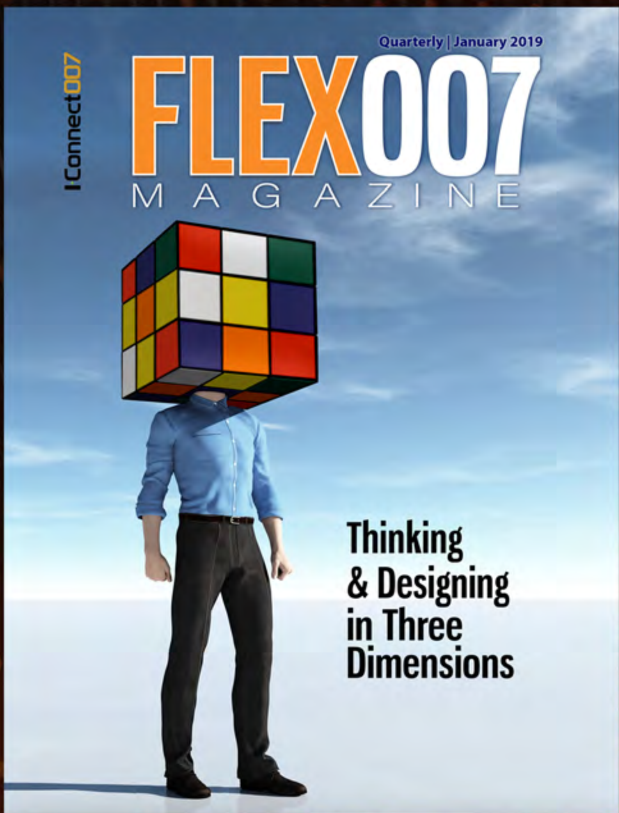
The team is currently testing new materials, electronics, and packaging to increase the sensor's range and applications while keeping costs low.

(Source: Oak Ridge National Laboratory)

NEW! DESIGN007 MAGAZINE SECTION

FLEX007

CLICK COVERS TO VIEW PAST ISSUES OF FLEX007 MAGAZINE



What the Flex?

by Andy Shaughnessy, I-CONNECT007

Welcome to the new Flex007 section in *Design007 Magazine.* Have a look around. I think you'll enjoy this new format.

As most of you know, our quarterly *Flex007 Magazine* has proven to be pretty popular. With columns by flex veterans—such as Joe Fjelstad, Tara Dunn, and Bob Burns—as well as feature articles by some of the top flex experts in the industry, *Flex007 Magazine* has become a must-read.

It's no wonder: the flex segment continues to expand. More and more OEMs are finding themselves using flex now, often because rigid boards will not fit in the form factors of new, cutting-edge products. Some OEMs now choose to use flex for better reliability. Rigid board designers and engineers often have to get up to speed on flex in a hurry, and they don't know where to turn for up-to-date flex design information.

Flex007 Magazine was loaded with flex design content, and now we've merged that publication with *Design007 Magazine.* Now, you'll find flexible circuit design content in each issue of *Design007 Magazine;* you won't have to wait three months to find great flexible circuit design content. Our expert contributors will continue to bring readers everything they need to know about flex and rigid-flex design to stay ahead of the game.

It's a good fit. Most of the really cool flex innovations are happening at the design stage, and most EDA tools are now optimized for flex design, eliminating the need for "paper dolls" and other workarounds from the Stone Age, relatively speaking.

This is a brave new world for flexible circuits. Flex was a tiny sub-category of PCBs in the not-too-distant past, but flex is on fire now; witness the number of fabricators moving into flex. Indeed, fabricators have found that they can make a pretty penny on flexible circuits, particularly rigid-flex.

But for all the growth in the flex segment, there seems to be a lack of available design information. Search online for yourself. How many flex design instructors can you name? There is a definite need for more information about flex and rigid-flex design tips and techniques, and we are pleased to be able to do our part.

We open our new section with a feature interview with Nikolay Ponomarenko, Altium's director of product management, who discusses the company's move into printed electronic circuits design capabilities and how PEC design differs from rigid and flexible circuit design. In our next feature interview, David Wiens—product marketing manager with Mentor, a Siemens Business—details their tools' latest flex and rigid-flex design capabilities, and some of the challenges that designers face with rigid-flex circuits.

We also have a technical paper by Weifeng Liu, et al., where the authors describe a process for creating flexible hybrid electronics, which combines traditional manufacturing techniques with today's precision ink technologies. Next, an interview with Philip Johnston, managing director of Trackwise Designs, covers their patented length-unlimited printed circuits, including flexible circuits that measure 26 meters in length.

Further, we have columns by our regular contributors, including an update from I-Connect007's Patty Goldman. This month, Joe Fjelstad offers details on some recent efforts to make flexible circuits more stretchable. Next, Steve Williams brings us an interview with Prototron's Van Chiem, who has been instrumental in developing flex and rigid-flex processes at the company's Tucson, Arizona facility.

Take a look around our new section, and let me know what you think. 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.

Changing Seasons

Patty's Perspective

by Patty Goldman, I-CONNECT007

If you have already read Andy's column, then you know we've decided to incorporate content related to flex and rigid-flex into *Design007 Magazine* rather than wait for a quarterly publication. It's all about design anyhow, right? Am I preaching to the choir here or what?

So, this made a perfect opportunity for me to back a little further away from a particular magazine and do what I love, which is soliciting, reading, and editing technical articles for I-Connect007. Right now, we are focused on some of the technical papers from IPC APEX EXPO 2019 that we have permission to publish. You'll find some of these in the upcoming issues of all three magazines over the next few months.

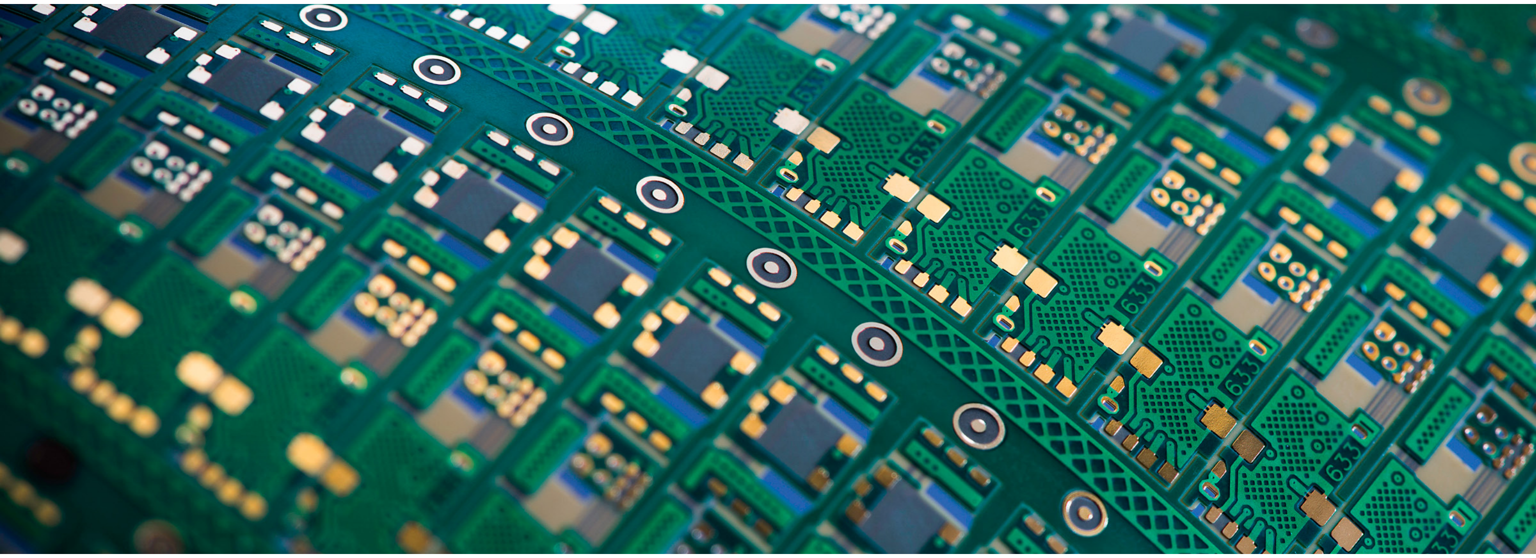
Dinnerplate hibiscus in 2018.

Of course, we all know that in this industry, you can't just retire and move on—you have to back off slowly and probably never leave completely. It's in your blood and DNA! We love technology and being a part of the industry that is changing the world more rapidly than ever. I am of the firm opinion that technology will bring the world together, solve famine and disease, and keep us from getting toasted as the planet warms—if only the politicians would stay the heck out of it. If you read the Electronic Industry News items in our daily newsletter (click here to receive it), you know there is a lot of positive stuff going on.

Coneflowers, bee balm, and other pollinator-friendly flowers.

In the meantime, though, I am in that back-off mode. And most importantly, from my perspective, I now have more time for my personal interests, which include gardening and involvement with some local non-profits. We all have our passions, and I hope yours extend beyond your day-to-day work too. I'm still with I-Connect007 for the foreseeable future, so enjoy, and I'll see you around at IPC APEX EXPO for sure. **FLEX007**

Patricia Goldman is managing editor of *Flex007 Magazine.* To contact Goldman, click here.




109C

Altium Designer 19.0 Features Printed Electronics Design Functions

Flex007 Feature by Andy Shaughnessy
I-CONNECT007

The newest version of Altium Designer—revision 19.0—includes functionality for designing printed electronic circuits. We wanted to get the scoop on Altium's PEC tools, so we asked Nikolay Ponomarenko, Altium's director of product management, to give us a tour of the new functions.

Andy Shaughnessy: Nikolay, tell us about the printed electronics design capabilities in the newest version of Altium Designer. I've heard good things about it.

Nikolay Ponomarenko: Starting with AD 19.0, we have added a set of capabilities targeting the design of non-conventional boards. One of these is printed with conductive ink in a semi-additive style without traditional etching of the solid copper planes.

Altium Designer users can now design a stackup with all of the specifics of the processes of printed electronics, which takes into consideration the passes of the conductive or non-conductive material deposition; internally, we refer to this as layerless stackup (Figure 1).

As part of that process, designers can define explicit dielectric layers (i.e., the layers where dielectric might be represented by some ge-

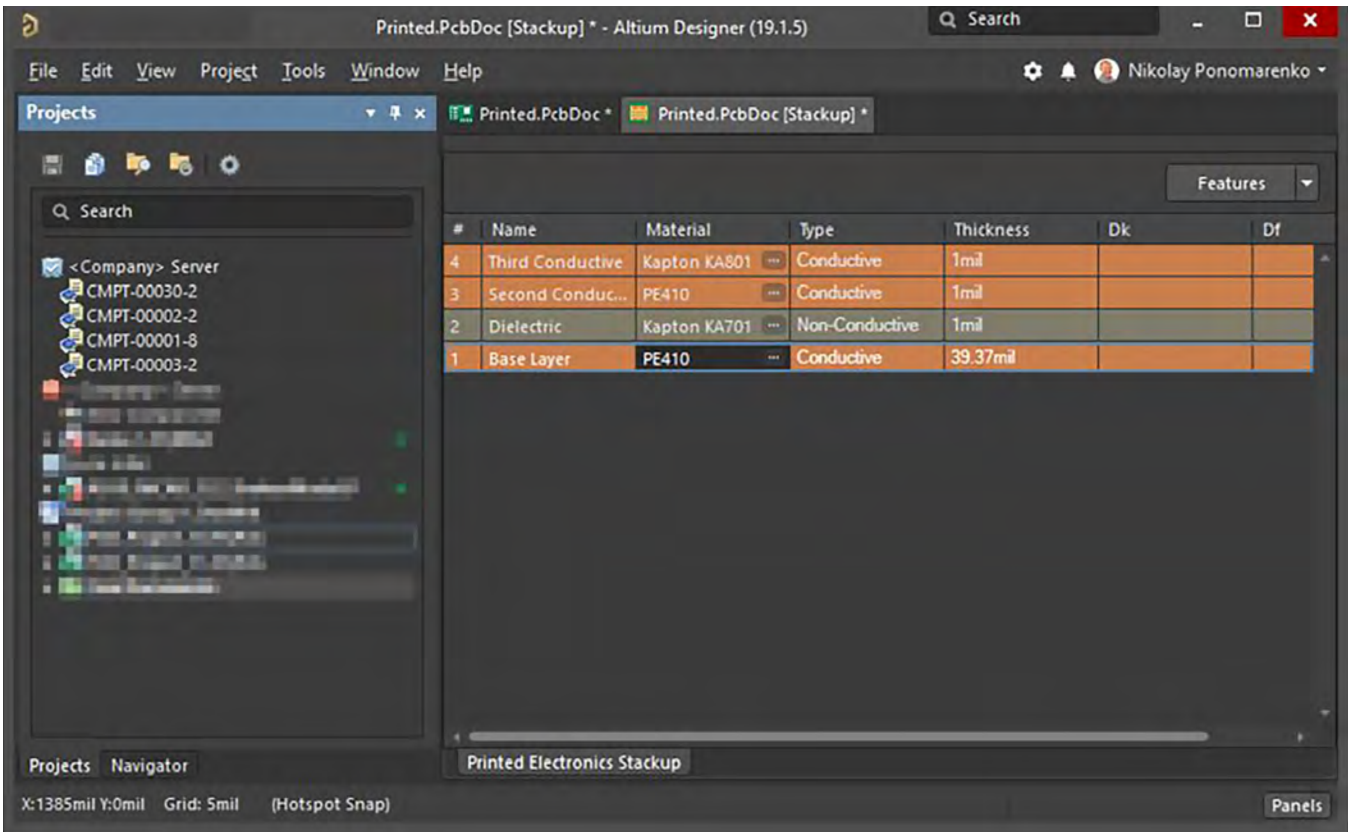


Figure 1: The layerless stackup feature found in Altium's PEC design capabilities.

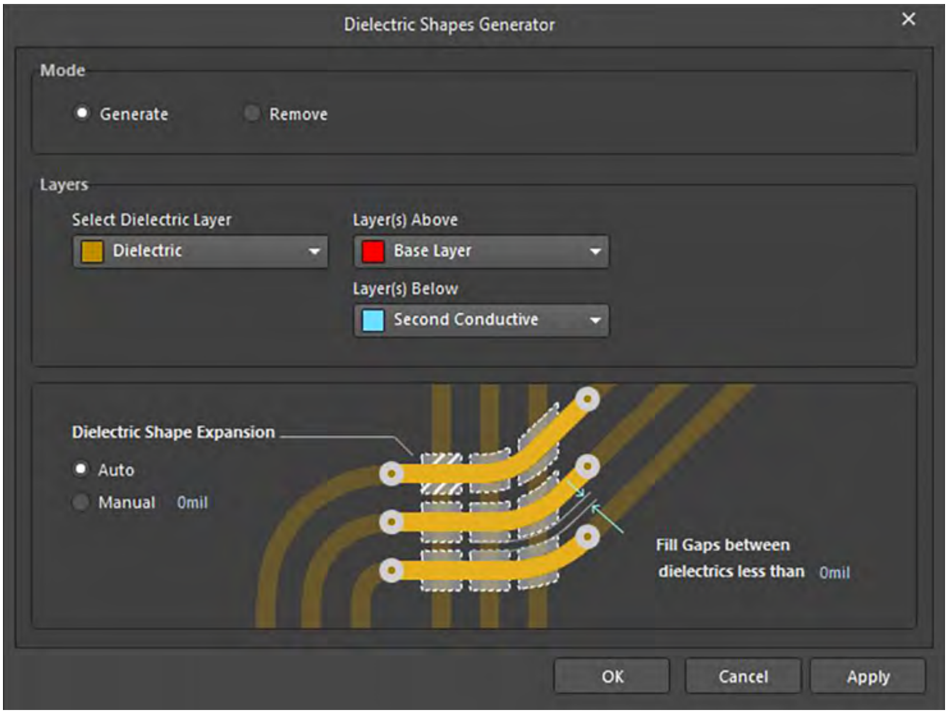


Figure 2: A dielectric shapes generator helps designers with unique issues related to printed electronic circuits.

ometry) instead of the assumed solid as seen in traditional boards. For each of the printing passes ("layers"), you can explicitly define the ink material.

Connectivity between conductive layers assumes the physical connection of the printed materials in Z-axis. For example, part of the trace might consist of segments printed with different inks or it takes multiple passes to reach the desired thickness of the feature.

When such connectivity is not desirable, the designer can simply build the dielectric crossovers, isolating different nets when they need to jump over each other using a manual or automated approach (Figure 2).

To control the shorts and connectivity, the rules and constraints system was adjusted and acknowledges the uniqueness of the Z-axis connectivity of the printed designs.

Shaughnessy: What were some of the drivers that led you all to develop PEC design functions?

Ponomarenko: Altium has always been a leader of innovation in the EDA software industry, and we see printed electronics as one of the emerging niches in the market. We want to ensure our users can stay up to date and are not limited by the tool to move forward with their products—that was the key motivation. We have been working with an excellent partner, Tactotek, to create design capabilities and parameters for printed electronics. They make production scale injection molded structural electronics (IMSE).

Shaughnessy: What sort of applications are your customers producing that feature printed electronics? Do they include wearables?

Ponomarenko: We don't always have these insights, but some applications we know about are capacitive touch displays, automotive, consumer electronics, medical, and even some wearables.

Shaughnessy: What are some of the biggest challenges related to designing printed electronics?

Ponomarenko: Maturation of the technology remains the biggest hurdle. However, if we just talk about the design aspect, establishing the continuous and scalable flow is a big one. This includes the basics of how the outputs will be generated. Since it is just printing, traditional outputs, such as Gerbers, are not always the best. This shift in the paradigm of the design has its toll. PCB designers need to shift not only their design habits but also the perception of the board. In some situations, it becomes a structural element rather than just the traditional computational element.

Shaughnessy: What trends do you all see in the PEC market? Do you think printed electronics is finally gaining acceptance?

Nikolay Ponomarenko

Ponomarenko: Significant investment in materials development, as well as big names getting behind the technology, are indicators to us that printed electronics is gaining its place in the market. We don't see this trend as replacing traditional approaches, but extending the reach of electronics into new domains.

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

Ponomarenko: For us, printed electronics is part of the larger landscape of structural electronics. If you are moving in that direction technologically but experience challenges at the design stage, Altium would be like to hear from you to explore possible solutions. You can email me directly at nikolay.ponomarenko@altium.com.

Shaughnessy: Thanks for your time, Nikolay.

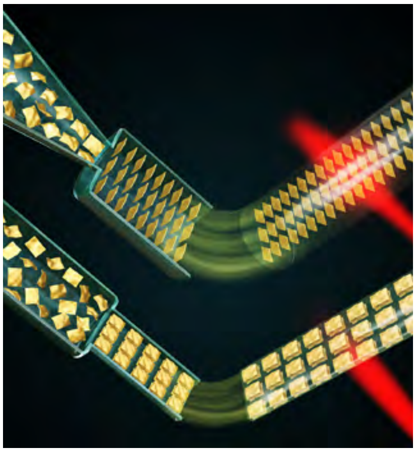
Ponomarenko: Thank you.

Visit I-007eBooks.com to download your copy of Altium's book *The Printed Circuit Designer's Guide to... Design for Manufacturing* as well as other free, educational titles. FLEX007

Researchers Pioneer Microfluidics-enabled Manufacturing of Macroscopic Graphene Fibers

A team of researchers at Rensselaer Polytechnic Institute has developed a new microfluidics-assisted technique for developing high-performance macroscopic graphene fibers. A recently discovered member of the carbon fiber family, graphene fiber has potential applications in diverse technological areas, from energy storage to electronics, optics, electromagnetics, thermal conductor and management, and structural applications. Their findings are published in *Nature Nanotechnology*.

It has historically been difficult to simultaneously optimize both the thermal, electrical, and mechanical properties of graphene fibers. However, the Rensselaer team has demonstrated their ability to do both.



Macroscopic graphene fibers can be manufactured by fluidics-enabled assembly from 2D graphene oxide sheets dispersed in aqueous solutions forming lyotropic liquid crystal. Strong shape and size confinements are demonstrated for fine control of the graphene sheet alignment and orientation, which is critical for realizing graphene fibers with high thermal, electrical, and mechanical properties. This microfluidics-enabled assembly method also provides the flexibility to tailor the microstructures of the graphene fibers by controlling flow patterns.

"The control of different flow patterns offers a unique opportunity and flexibility in tailoring macroscopic graphene structures from perfectly aligned graphene fibers and tubes to 3D open architecture with vertically aligned graphene sheet arrangement," said Jie Lian, a professor in the Rensselaer Department of Mechanical, Aerospace, and Nuclear Engineering (MANE) and the lead author on the article.

"This research paves the way for new sciences to optimize the fiber assembly and microstructure to develop high-performance graphene fibers," said Lian.

(Source: Rensselaer Polytechnic Institute)

Mentor Tools: Optimized for Flex and Rigid-flex Design

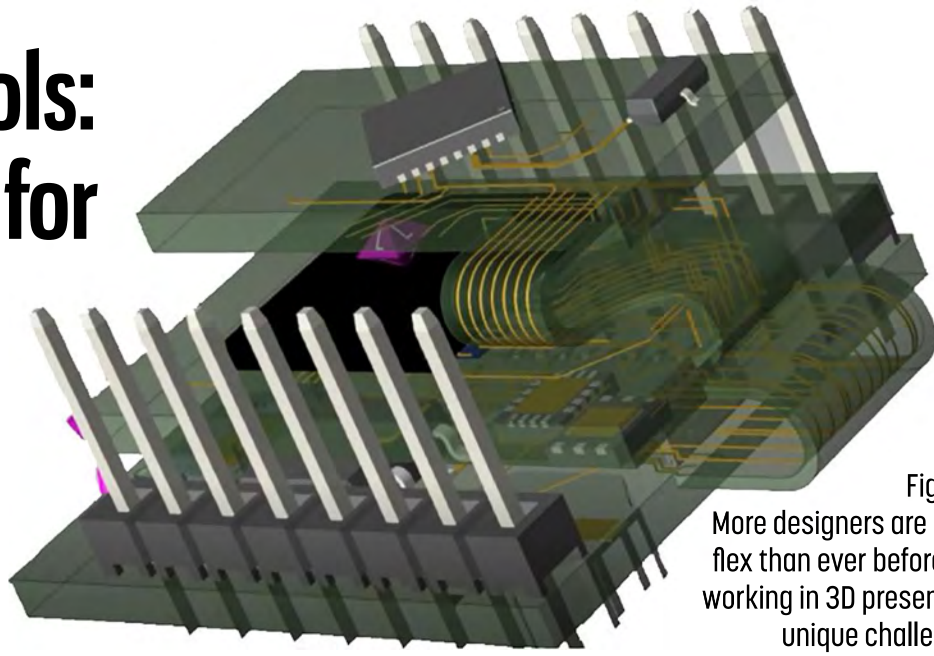


Figure 1: More designers are using flex than ever before, but working in 3D presents its unique challenges.

Flex007 Feature by Andy Shaughnessy
I-CONNECT007

With the launch of the new Flex007 section in *Design007 Magazine,* we asked David Wiens, product marketing manager with Mentor, a Siemens Business, to tell us about their tools' flex and rigid-flex design capabilities. As David explains, today's higher-end design software tools are optimized for flex design, making workarounds a thing of the past.

Andy Shaughnessy: What are your customers' biggest challenges in designing rigid-flex?

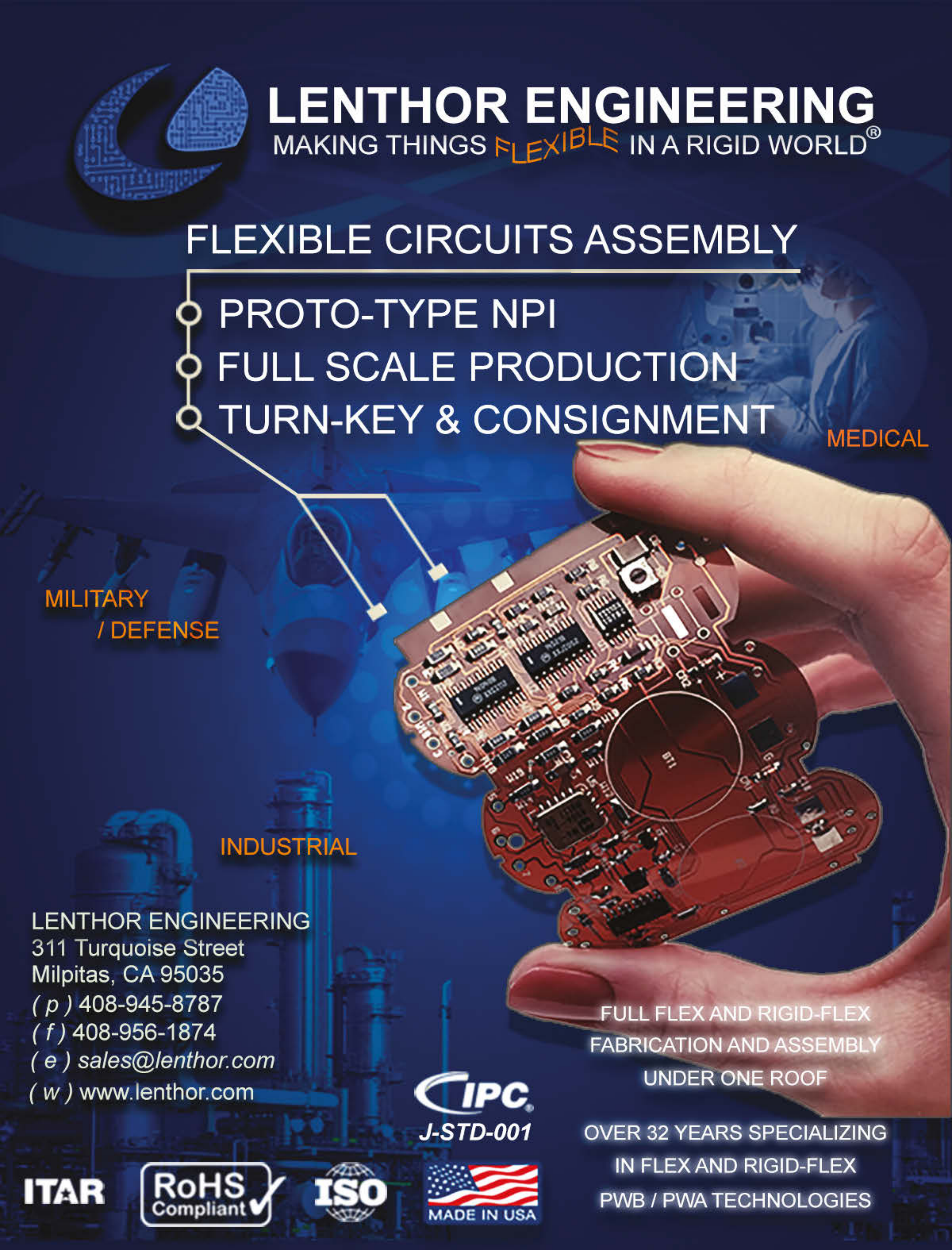
David Wiens: Engineering teams have designed advanced rigid-flex products for years using a series of workarounds to their EDA tools, often verifying with paper dolls. Rigid-flex designs require advanced stackup constructs (e.g., multiple outlines, each with its own stackup, and new materials). There are also additional rules that need to be applied, including bend/fold control with collision clearances, curve routing with arcs and teardrops, hatched plane fill shapes, component placement limits in flex areas, and fabrication rules around board stiffeners and coverlays. The workarounds naturally take longer to implement and often result in errors because the design must be checked manually. This can lead to a non-optimized product because once something is designed, nobody wants to go back and make ECOs. Some errors, such as copper micro-cracks, create long-term product reliability issues. Manufacturing is also a challenge. Design teams must align with their manufacturer to understand the costs of different rigid-flex structures—costs can go up quickly—and optimize the hand-off from design to manufacturing.

Shaughnessy: Tell us about the rigid-flex design capabilities in the latest versions of Mentor's tools.

David Wiens

Wiens: Our solution supports flex, rigid, or rigid-flex with a common set of functionality. Native support for flex/rigid-flex extends across the flow, from initial stackup definition through design validation and manufacturing outputs, eliminating time-consuming workarounds.

It starts with an independent stackup for each rigid or flex element; these can easily be modified or overlapped. This approach limits the


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

board outline and stackup modifications necessary when the shape of the board changes. With flex stackups, there are additional materials and layer types to model, such as cover layers, stiffeners, and adhesives. These materials are intelligent and are understood at design verification as well as the hand-off to manufacturing. Control of where bends occur is critical, so a bend area object defines the location, radius, angle, and origin. Attributes also define placement, routing (e.g., via utilization, trace corners, trace width changes, etc.) and plane metal (e.g., hatch/cross-hatch) rules in the area.

For place and route, each rigid-flex area has its own external/internal layers, so parts can be placed on any external layer (including flex regions and/or in cavities) with appropriate pads and openings automatically handled. During routing, true arcs are utilized to minimize stress fractures in flex regions, and they adhere to the constraint-driven, correct-by-design methodology for which we're known. Curved teardrops are automatically generated and maintained dynamically. Due to the automation throughout layout, design changes are easy and safe.

Regarding 3D design and MCAD collaboration, the complexity of rigid-flex structures requires full 3D design and verification, not just 3D viewing. For example, parts can be placed on bent or flat surfaces in 3D. Also, 3D DRC checks will identify any interference when the circuit is bent. An enclosure can be imported from MCAD and mated with the design to visualize and check alignment. In addition, rigid-flex structures (and their bend attributes) can be passed to MCAD for further modeling and analysis.

With signal/power integrity analysis, as traces transfer between flex and rigid areas, their impedance and propagation speed changes. Our analysis tools accurately model distinct stackup areas to accurately model this effect. They also recognize the unique material types, such as adhesives, copper foils, and stiffeners.

And for manufacturability verification and hand-off, DFM is critical to successful fabrication of rigid-flex. An extensive set of flex/rigid-flex DFM checks is available during design. A few examples include conductors parallel to bend area, plated holes too close to a stiffener, and rigid area copper close to interface area. Valor technology is integrated directly within Xpedition's layout editor to enable checks at any time during design with er-

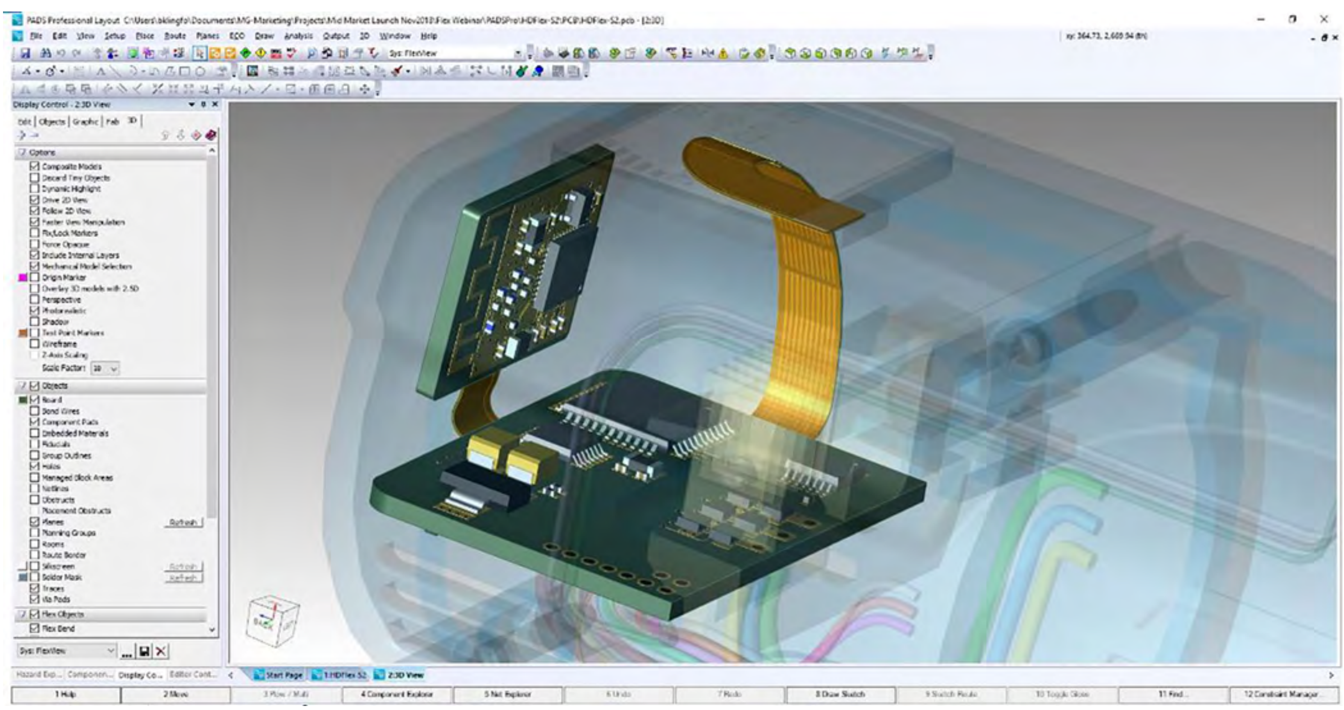


Figure 2: Today's enterprise-level EDA tools are often designed for flex and rigid-flex circuits.

rors shown within standard reports. To convey all rigid-flex information to the fabricator in a completely unambiguous way, an intelligent product model is required. ODB++ has specific format constructs for rigid-flex to clearly communicate intent and mitigate any ambiguities, and it's planned for IPC-2581.

Shaughnessy: What trends do you all see in the flex market both here and overseas?

Wiens: The increased product stability and performance now possible with rigid-flex have driven higher adoption, particularly in consumer and medical wearables and IoT devices. With tighter form factors, flex/rigid-flex is a strong option for dense electro-mechanical structures.

Shaughnessy: What's next for Mentor's flex design tools?

Wiens: Tighter integration between Xpedition and NX has enabled more intelligent modeling of rigid-flex structures, including bend previews and improved ECAD/MCAD collaboration and change management of rigid-flex designs. We're also adding extended electrical rule checks specific to rigid-flex structures.

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

Wiens: As noted at the beginning, teams have been designing rigid-flex for quite some time. By augmenting our tool flows with flex-aware elements and automation, designers can cut cycle times, improve reliability, mitigate risk, and optimize for a lower cost.

Shaughnessy: Always a pleasure, Dave. Thanks for your time.

Wiens: Thank you, Andy.

Visit I-007eBooks.com to download your copies of *The Printed Circuit Designer's Guide to... Signal and Power Integrity by Example* from Mentor, a Siemens Business, as well as other free educational titles. **FLEX007**

Nanophysicists Develop High-Performance Organic Phototransistor

Researchers from the Physical Institute and Center for Nanotechnology (CeNTech) at the University of Münster, headed by Prof. Dr. Harald Fuchs, together with colleagues from China, have developed a novel thin-film organic phototransistor (OPT) arrays based on a small-molecule (2, 6-diphenylanthracene—DPA), which has a strong fluorescence anthracene as the semiconducting core and phenyl groups at 2 and 6 positions of anthracene to balance the mobility and optoelectronic properties.

The fabricated small-molecule OPT device shows high photosensitivity, photoresponsivity and detectivity.

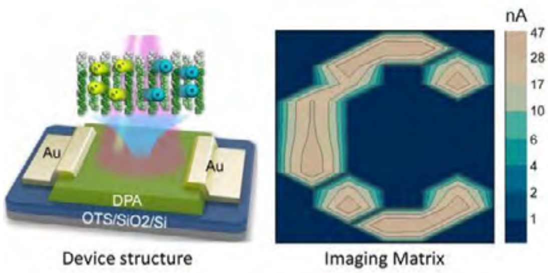
"The reported values are all superior to state-of-the-art OPTs and among the best results of all previously reported phototransistors to date. At the same time, our DPA-based OPTs also show high stability in the air," says Dr. Deyang Li.

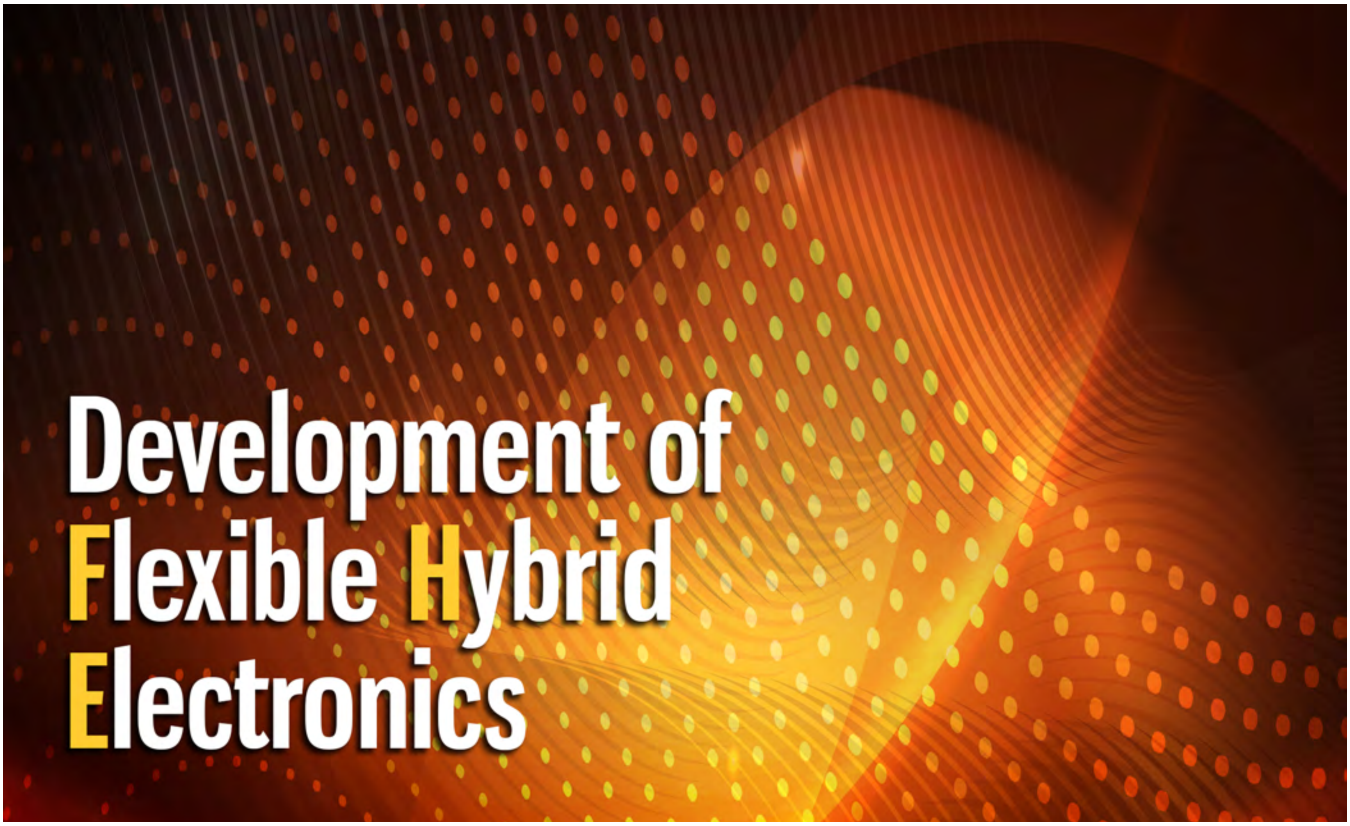
Dr. Saeed Amirjalayer adds, "By combining our experimental data with atomistic simulation, we are, in addition, able to explain the high performance of our device, which is important for a rational development of these devices."

The WWU researchers believe that, therefore, DPA offers great opportunity towards high-performance OPTs for both fundamental research and practical applications such as sensor technology or data transfer.

The research was published in the latest issue of *Nature Communications*. The work was funded by the German Research Foundation.

(Source: University of Münster)





Development of Flexible Hybrid Electronics

Flex007 Feature by Weifeng Liu, Ph.D., Alex Chan, William Uy, Dennis Willie, Dongkai Shangguan, Ph.D.
FLEX, MILPITAS, CA

Introduction

Flexible hybrid electronics (FHE) refers to a category of flexible electronics that are made through a combination of traditional assembly processes of electronic components with high-precision ink printing technologies [1]. By integrating silicon components with printed inks and flexible substrates, FHE will revolutionize the IoT and wearable industries. With FHE, designers can create a heterogeneous electronic system that can be fully integrated with different sensors, lighter in weight, more cost effective, more flexible and conforming to the curves of a human body or even stretchable across the shape of an object or structure—all while preserving the full functionality of traditional electronic systems.

The FHE industry is still in the early stages of development, and a variety of design, material, assembly and reliability issues need to be addressed. For example, electrical interconnections formed with conductive adhesives may not be as conductive or reliable as compared to conventional solder assembly. Typical polymer-based conductive inks are not as conductive as the etched copper used to make circuit boards and they are mostly not readily solderable. Additionally, commercially available stretchable thermoplastic-based film substrates have relatively low heat resistance and cannot withstand the current lead-free reflow process temperatures.

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

Experimental

This study is to convert a rigid multilayer wearable development platform into a flexible one using printed conductive ink and flexible substrate. The current rigid platform contains two active semiconductor components

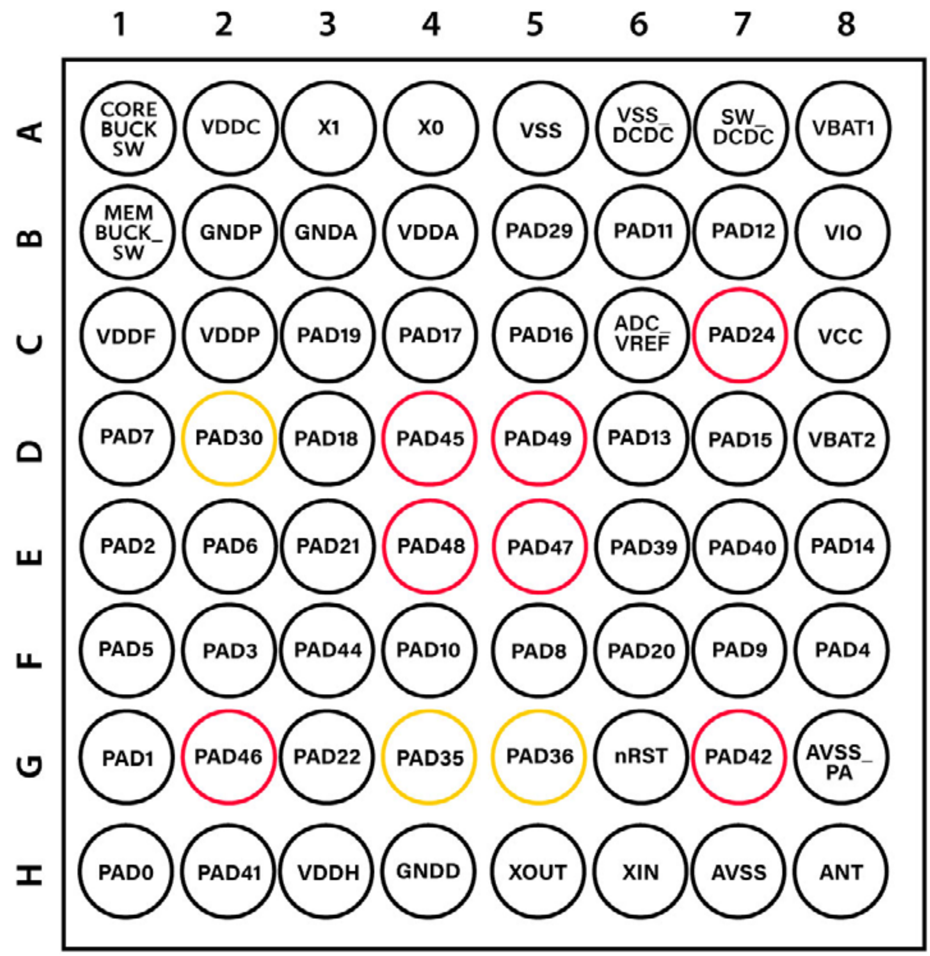


Figure 1: Footprint of the LGA package.

and dozens of passive devices. The largest component is the microprocessor chip. Figure 1 shows the footprint for the microprocessor chip. It is an LGA package with 8X8 array and 64 I/Os. The pad size is 250 µm and the pitch is 400 µm, which leaves the space between pads at 150 µm.

One of the biggest challenges is how to escape route the traces under this processor chip. Different options were considered for the escape routing. The initial proposal from the designer was to have a 1:1 conversion. The original rigid board is a six-layer board; conversion to a six-layer FHE system requires at least 12 screen printing processes, including six for the conductive ink and six for the dielectric ink, which makes the process development very expensive. Other challenges to be dealt with for a six-layer substrate are higher thickness, less flexibility, and misalignment of conductive ink and dielectric ink layer to layer.

The other option is to use a single layer of conductive ink to route all the processor pads. However, this requires fine-line ink printing, considering the space between the pads is only 150 µm. If one trace passes through two neighboring pads, 50 µm lines and spaces are needed; if two traces pass through, 30 µm line/space are needed. In the previous project, we have demonstrated our capability by printing conductive ink in 50 µm line and space in the lab environment. However, printing 30 µm lines/spaces with consistent quality still requires significant process development. Printed conductive ink typically shows much higher electrical resistance as compared to the conventional etched copper trace. Printed ink in the fine line will make the electrical resistance even higher, which makes it very challenging to satisfy the high-frequency system requirement.

Working with the design team, we eliminated some of the pinouts of the LGA package for escape routing by simplifying certain functionalities of the system. We were able to use the two-layer structure of the printed ink. For this, we also have two options. One is to build up the interconnect structure layer by layer, by printing conductive ink first, then dielectric ink, followed by conductive ink then dielectric ink. The other option is to print conductive ink on the top and bottom of the flexible substrate and use filled microvias to connect the top and bottom inks. We have projects ongoing to pursue both approaches, but for this article, only the latter one (microvia) is presented.

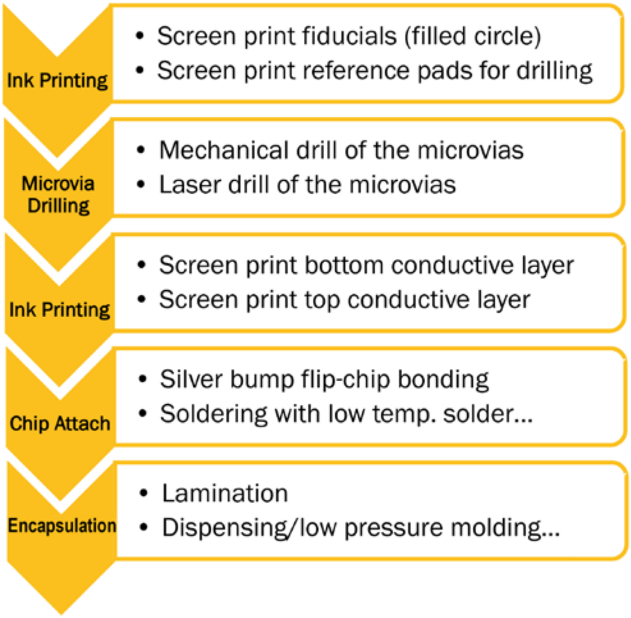


Figure 2: Process flow to make a two-layer interconnect structure.

Figure 2 shows the process flow to make the two-layer printed ink interconnect structure on PET substrate using filled microvias. The first step was to screen print fiducials for later fabrication processes, including via drilling and screen printing. Reference or anchor pads were also printed at this step. By using the reference pads, we can easily check the sizes and positions of the drilled microvias with regard to the reference pads.

For microvia drilling, mechanical and laser drilling were evaluated initially. Mechanical drilling yielded more consistent results in terms of the via diameter and position with sharp via walls. Accordingly, mechanical drilling was used to fabricate the microvias for this project with two hole sizes: 150 µm and 250 µm, using a precision drilling system. The printed ink fiducials were used for the alignment of the microvias during drilling. After drilling, positions and diameters of the microvias relative to the reference pads were inspected under optical microscope.

After microvia drilling, conductive ink was printed on the bottom of the substrate using the fiducials made in the first step. The PET substrates are precoated for better printing quality. A polyester screen was created with a mesh count of 305, thread diameter of 34 µm and thread angle of 45 degrees. Screen printing parameters were as follows:

- Pressure 14–16 Kg
- Speed: 35 mm/second
- Gap: 4 mm

After printing, the ink was then cured at the elevated temperature (120°C for 10 minutes). The substrate was afterwards flipped over and the conductive ink was printed on the top side. The ink was then cured at 120°C for another 10 minutes. During the screen printing process, the conductive ink was expected to be automatically filled through the microvias, making connection between the top and bottom inks. For this initial trial, the dielectric ink was not printed.

The next step is to attach the electronics components on the printed substrates. Several approaches were evaluated in parallel: anisotropic conductive paste (ACP), thermocompression flip-chip bonding with silver bumps and non-conductive paste (NCP), soldering reflow process, and electrically conductive adhesive (ECA). ECA is mainly used for attaching the passive devices, while the other processes can be used for attaching both passive and active devices. Only the development work on the reflow process is presented and discussed in this article. One big advantage of the reflow process is that it is compatible with the conventional PCB assembly process, so we can fully utilize current manufacturing infrastructure that is already in-house.

From our previous work, we have demonstrated the solderability of a specialty formulated conductive ink using low-temperature solder Sn42/Bi57.6/Ag0.4 with a melting temperature of 138°C. By using low temperature solder, we can minimize the damage to the PET substrates. For this initial feasibility study using low-temperature solder and printed conductive ink, only the microprocessor packages were attached and evaluated. Figure 3 shows the reflow profile for the assembly process with a peak temperature of 155°C.

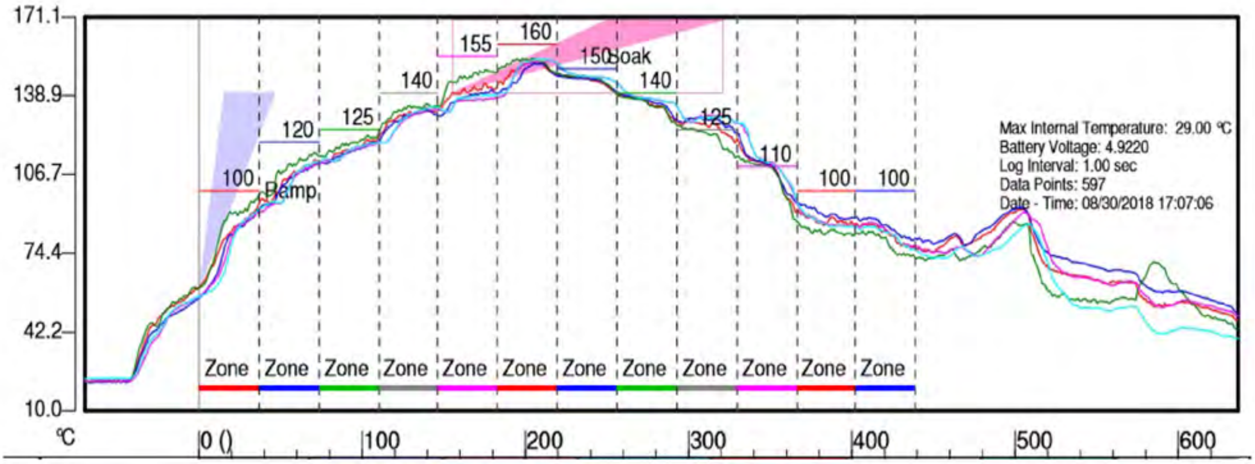


Figure 3: Reflow profile of low-temperature soldering process.

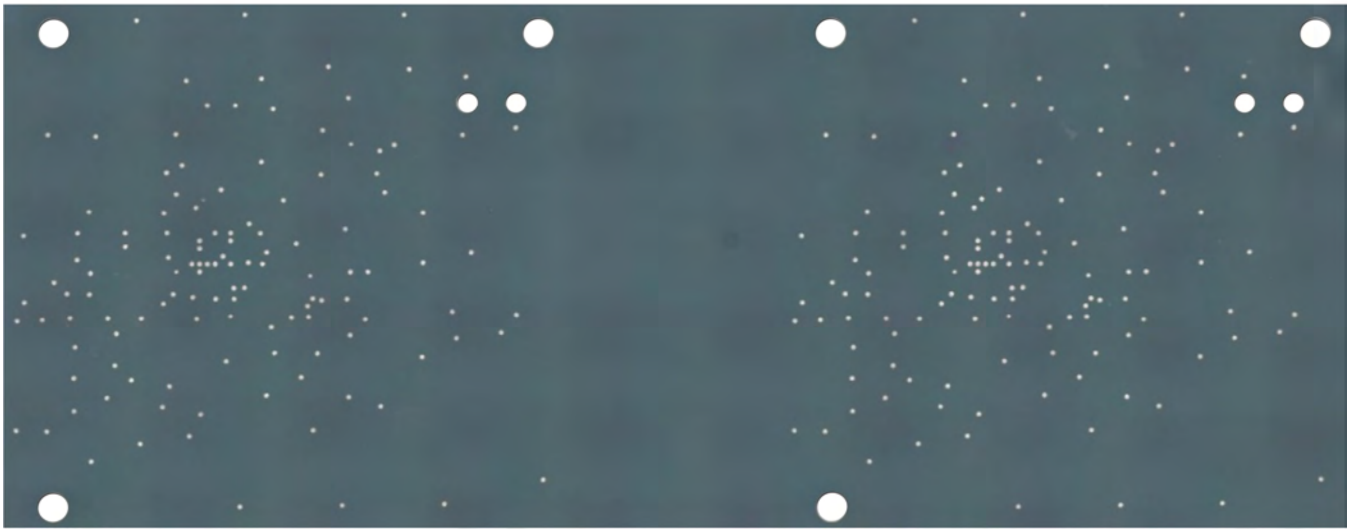


Figure 4: Screen-printed fiducials and reference pads.

Results and Discussions

Figure 4 shows the first screen print results: fiducials for the microvia drilling and reference pads. The reference pads are mainly used to facilitate inspection of the dimensions and positions of the drilled microvias. Without these reference pads, the actual measurement has to be done to understand the actual tolerances of the drilled vias. With these reference pads, the quality of the drilled vias can be quickly obtained under the microscope by comparing to the reference pads. The automatic inspection process can be quickly developed and employed.

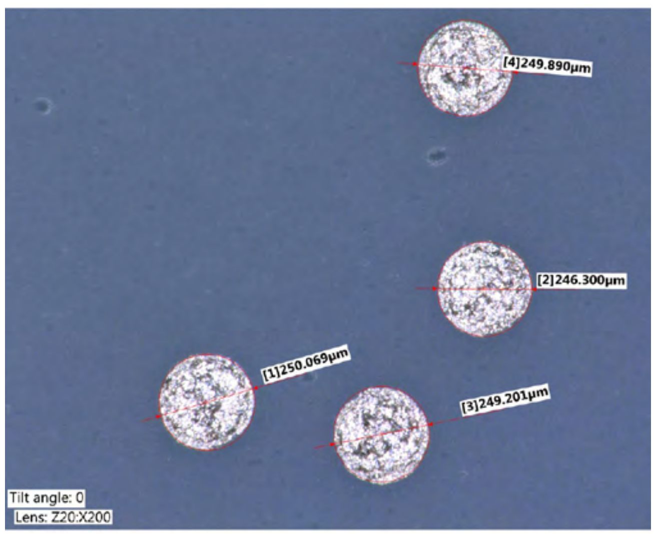


Figure 5: Measurements of the reference pads.

Figure 5 shows the actual measurements of the reference pads. The target diameter of the

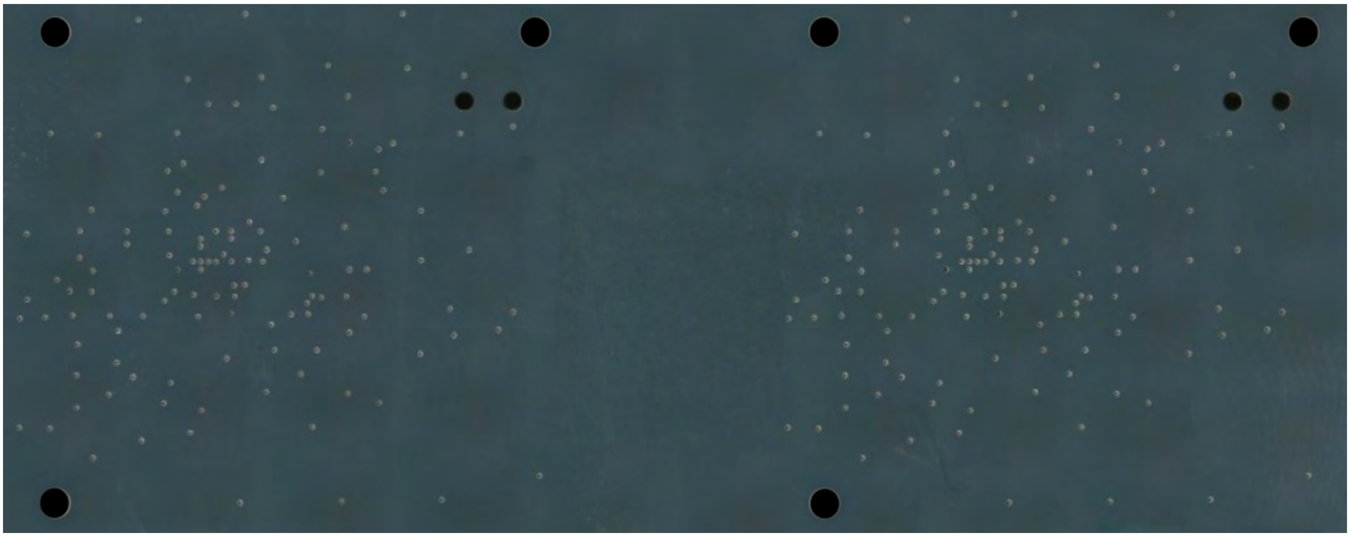


Figure 6: Drilled microvias in 150-µm diameter.

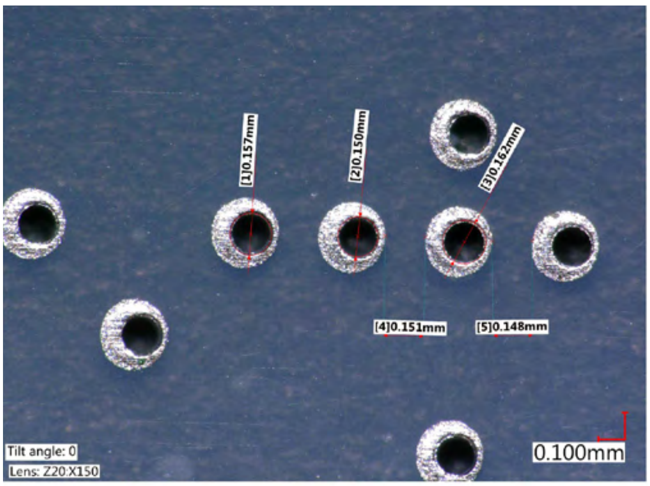


Figure 7: Measurements of the drilled microvias (150 µm).

reference pads is 250 µm; from the measurement, the actual diameters of the reference pads are well within the tolerance target ±10 µm.

Two sizes of microvias were drilled using mechanical drill bits: 150 µm and 250 µm. Figure 6 shows the 150 µm microvias. Figure 7 shows an enlarged view of the microvias and actual measurement. With printed pads as reference, the quality of the drilled microvias can be readily examined.

Figure 8 shows the drilled microvias 250 µm in diameter and Figure 9 shows an enlarged view of the microvias and actual measurement. With printed pads as a reference, the

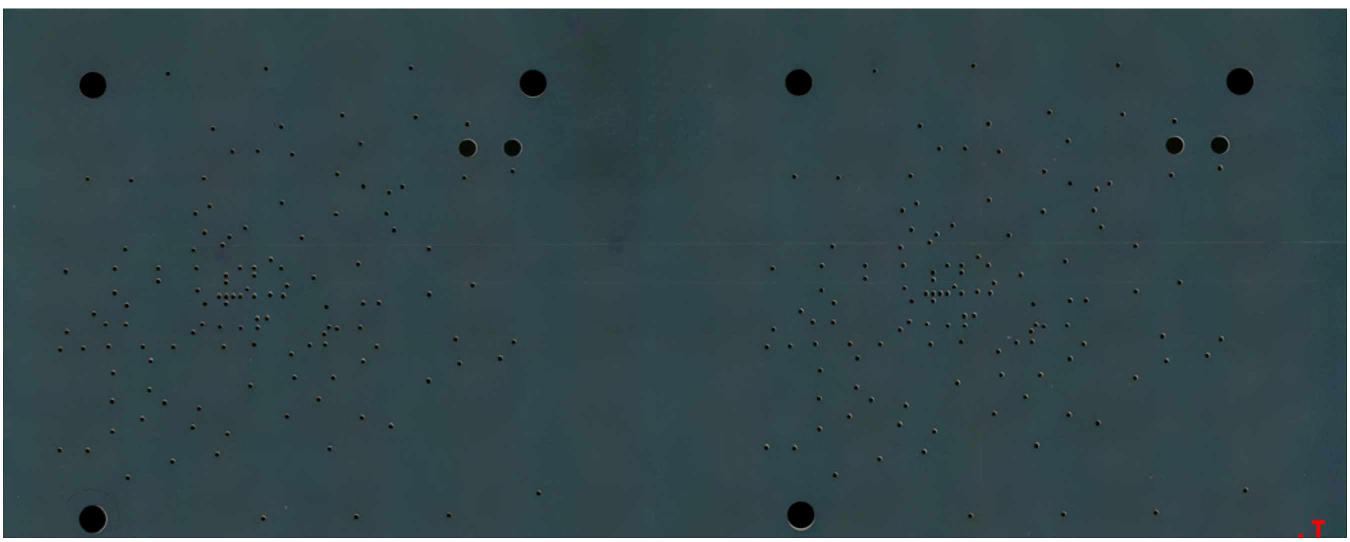


Figure 8: Drilled microvias 250 µm in diameter.

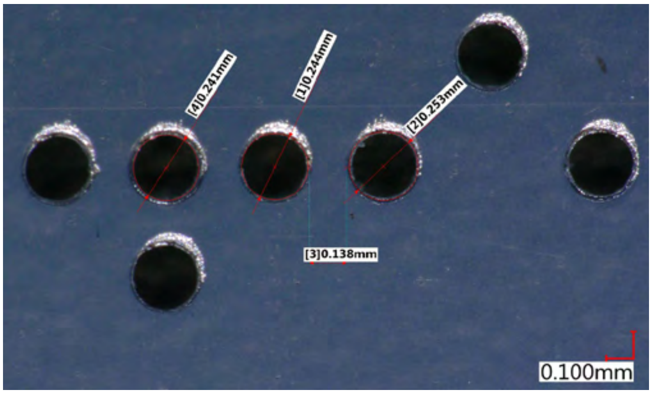


Figure 9: Measurements of the drilled microvias (250 μm).

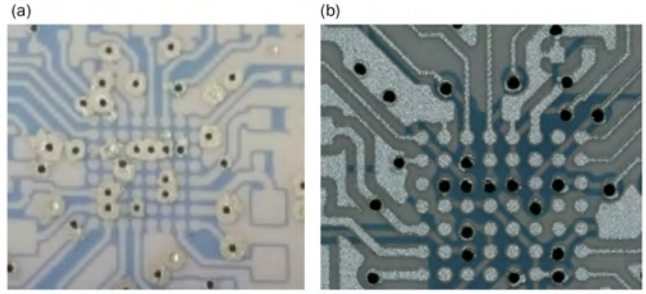


Figure 10: Ink splattering on the bottom of substrate (a) and unfilled microvias (a/b).

quality of the drilled microvias can be readily examined. Since this was just the first phase feasibility study, no statistical study was performed on the dimensional and positional tolerances of the drilled microvias.

The next step is to print the conductive ink traces on the top and bottom layers of the substrates. It was expected that, by printing the ink traces, the microvias would be automatically filled by the conductive ink. However, the results did not turn out as expected. Figure 10 shows the unfilled microvias after ink printing. When a liner was not used beneath the substrate, the vacuum sucked the conductive ink out of the microvias and the conductive ink formed a splattering pattern around the via holes on the bottom layer of the substrate (Figure 10a). Even when a liner was used under the substrate, microvias were not filled either (Figure 10b). This phenomenon occurred for both 250 μm and 150 μm microvias.

A number of trials were performed in order to fill the microvias and understand the root causes for the unfilled microvias. By analyzing the characteristics of the conductive ink used in this study and trial results, we gradually formed a list of potential root causes. The viscosity of the conductive ink used to print the traces, 20,000 cps, is too low for microvia filling. For via filling in the PCB industry, a typical value for the viscosity of the conductive ink is in the range 40,000 cps. The low viscosity of the ink makes it difficult to stay inside the via holes. Since we use a screen to print the conductive ink, the fine wires of the screen may drag the ink out of the holes during the release of the screen after ink printing. Some ink may stay on the liner if there is a certain adhesion between the ink and the liner when the liner is separated from the substrate. Other causes may include via hole size effect and substrate material/thickness effect.

Based on this analysis, we defined our plan to address the unfilled microvia issue. At first, we planned to evaluate conductive adhesive with higher viscosity for hole filling using stencil printing instead of screen printing. Secondly, we planned to evaluate a new liner with weak adhesion to the conductive ink. Silicone film has been shown to have weak bonding with typical conductive ink. We also planned to work with our partners to evaluate different holes sizes from 500 μm down to 100 μm on another substrate material, such as TPU.

After several trials, we successfully plugged the microvias using stencil printing conductive adhesives. Figure 11 shows the filled microvias 150 μm in diameter on the top and bottom sides.

After filling the microvias, ink traces were printed on the top and bottom sides of the substrates. Figure 12 shows the printed substrate sample with printed circuitry on the top and bottom sides.

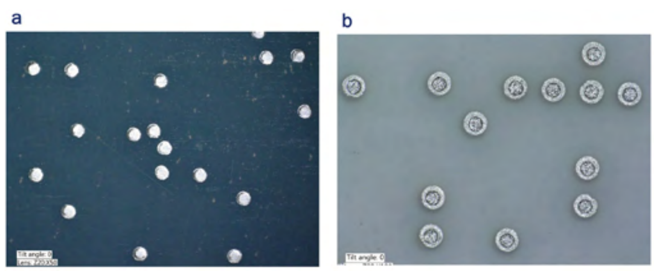


Figure 11: Filled microvias: (a) top side, (b) bottom side.

(a) (b)

Figure 12: Printed substrate: (a) top side, (b) bottom side.

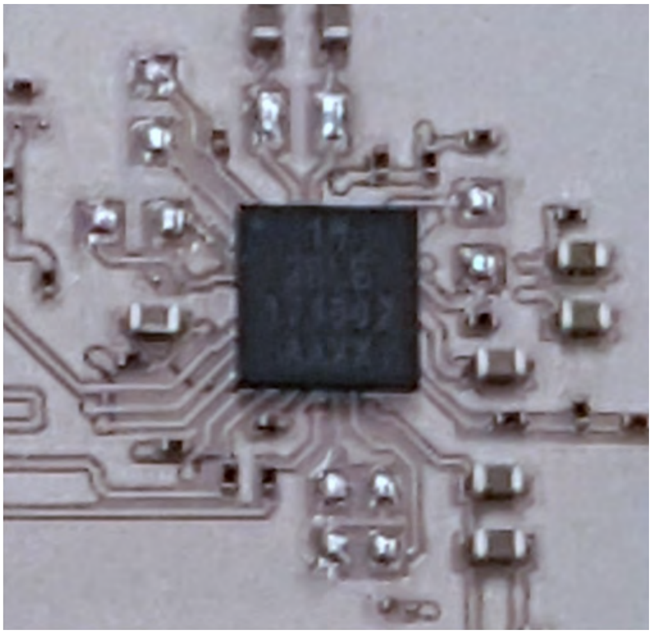


Figure 13: Printed substrate with components attached using low-temperature soldering process.

The microprocessor LGA package was attached to the printed substrate using the Sn42/Bi57.6/Ag0.4 low-temperature solder reflow process with a peak temperature at 155°C. Figure 13 shows a printed substrate with the LGA package attached. Further work is underway to evaluate the quality of the solder joints and further optimize the attach process.

Summary and Conclusions

Development of a flexible hybrid system was presented in this article. A two-layer interconnect structure was developed to accommodate the high-density, fine-pitch routing for the microprocessor package, through screen printing of conductive ink and filled microvias. Different approaches to attach the components to the printed substrates are being explored. Initial feasibility of using low temperature solder Sn42/Bi57.6/Ag0.4 was demonstrated to solder the LGA packages to the substrates printed with solderable ink. Further development and evaluation are underway. **FLEX007**

Acknowledgments

The authors would like to acknowledge the contributions from Robert Penning, Dr. Jie Lian and Dr. Francoise Sarrazin of the company Advanced Manufacturing Engineering (AME) Lab, and Lipika Bhattacharya, Glenn Henriquez, Hien Tran, Arnold Andreas, Bruce Thompson, and Raj Dhesikan of the company Innovation Lab.

References

1. E. Forsythe, B.J. Leever, "NextFlex: Flexible Hybrid Electronics Manufacturing," Defense AT&L, pp. 32-36, September-October 2016.

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

Weifeng Liu, Ph.D., is engineering manager, Advanced Manufacturing Engineering Group, Flex, Milpitas, California.

William Uy is program manager, Advanced Manufacturing Engineering Group, Flex, Milpitas, California.

Dennis Willie is director of engineering, Advanced Manufacturing Engineering Group, Flex, Milpitas, California.

Dongkai Shangguan, Ph.D., is vice president , Flex, Milpitas, California.

Alex Chan is no longer with the company.

Making Flexible Circuits Stretchable

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Flexible circuits have seen increased application and use since their introduction several decades ago. They are replacing or augmenting rigid circuit technology in a wide range of product markets from simple consumer items to incredibly complex military and aerospace applications. The limitations of the technology are truly boundless as it makes an appearance in the never-ending parade of new and improved electronic products.

While flexible circuits continue their forward march, over the past decade, there has been increased interest and use of stretchable circuits for a variety of new applications where circuits have the ability to deform and/or elongate elastically and return to their original size. This upstart variation of the venerable flexible circuit has been given the title of "stretchable circuit," and it appears that they are going to be an increasingly important subset of future flexible circuit designs.

It is my opinion that the initial driving impetus for the development of stretchable circuits was a bit different than normal, meaning that military and aerospace have traditionally driven the development of arcane electronic interconnection technologies as they did with the development of both flexible and rigid-flex circuits. In contrast, it was a consumer-driven market that appears to have been the gate opener in the form of wearable electronics. The roll-out products that drove imaginations were fanciful and eye-catching, led by fashionware with LED lighting as an integral element

of their construction. The European Union took point in helping to grow the early market by making a substantial investment of euros to fund research and jump-start the technology. They also sponsored a series of bi-annual conferences over six years (I had the honor and privilege of delivering one of the keynotes at the inaugural 2007 conference held in Leuven, Belgium, at the IMEC Research Center). The key challenge was and has always been not so much one of metallizing an elastic substrate but in making metal circuits that could be stretched.

Flexible circuits have arguably been stretched for a few decades when they have been folded in an accordion fashion to allow for servicing electronic systems contained in drawers as they are opened and closed while maintaining a connection. This method was one of the approaches used for the metal circuits but where the circuit elements are created with "humps" on stretchable materials. Thus, when the material is stretched, the circuit humps are flattened. Fundamentally the same principle as used in the original but on a much smaller scale. However, many applications were thinking more along the lines of a traditional planar construction.

Designers figured out that stretching of circuits could be accomplished by creating serpentine circuit patterns.

Designers figured out that stretching of circuits could be accomplished by creating serpentine circuit patterns. Other researchers sought ways to make circuits that did stretch without relying on in- or out-of-plane reshaping of the circuits, focusing on how to make circuits stretch reliably without breaking. What they came up with were polymer conductors that were resilient. Mixtures of elastic materials infused or mixed with conductive powders comprised of microscopic metals such as silver, copper, and gold. Others demonstrated the ability to contain liquid metals in channels formed in an elastic material. There has clearly been a great deal of cerate thinking applied to serve the growing interest and need.

Since the organized roll-out of stretchable circuits in the EU a little over a decade ago, the areas of actual or prospective applications have grown. Stretchable circuits are still being used in fashion, but the applications have been (if you can forgive the pun) stretched considerably as well. Many of the applications still center on interfaces with humans in the form of electronic diagnostic and monitoring devices with integral sensors (think mobile heart and breathing monitors and the like).

While stretchable circuits clearly have a certain appeal, they usually are not as cheap as traditional flexible circuits owing to the more limited manufacturing infrastructure. There are other ways to achieve the objectives if one thinks the problem through. As I have suggested in my flexible circuit seminars over the years, the most "flexible" interconnection "medium" is the air through which virtually all of the information in the world can now be accessed on a smart device. With the continuing growth and advancement of wireless technologies and near-field communication devices, such as IoT tags and smart devices, wireless solutions may well provide a simple answer to an otherwise complex interconnection construction. It is up to you to make the decisions based on the needs of the product in consideration.

One must be flexible in one's thinking first before setting out to making a new design, either flexible or stretchable. Enjoy your journey. **FLEX007**

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

A Conversation with Prototron's Van Chiem

The Right Approach

by Steve Williams, THE RIGHT APPROACH CONSULTING

I recently spoke with Van Chiem, a process engineer with Prototron Circuits, about developing in-house flex and rigid-flex processes and capabilities at their facility in Tucson, Arizona.

Steve Williams: Van, you have a very diverse technical background in PCBs. Can you tell us a little bit about your past experience and what brought you to Prototron?

Van Chiem: I have over 25 years of experience in this field. I started as a process engineer at Unisys Corporation where we made computer chips and advanced component packaging for interconnections like wire bonding, flip chip, and tab bonding. This is my ninth job in the PCB industry. Another job I had was with Enthone in Connecticut at the time, which is now MacDermid Alpha Electronics Solutions. I was in the research group working on the formulation of solder mask to try and convert it to photodefinable dielectric material for microvia. Instead of drilling, we used a photo process to create a microvia, and then the electroless copper followed by plating the copper to form the microvia with smart circuitry. That was my second job in the industry. I have had seven more jobs, but that might be too much to talk about.

Williams: How about we save the other seven for the next interview (laughs). So, you've seen a lot of technology changes in the last 25 years in the industry and were actually one of the original technologists.

Chiem: Well, I don't like to talk about myself, but I guess you could say that.

Williams: How long have you been at Prototron?

Chiem: About a year and a half.

Williams: And you have already accomplished quite a bit here. Prototron brought you on board for the specific purpose of upgrading their technology in a number of different areas, right?

Chiem: Yes, Kim O'Neil, the general manager here, brought me on especially to develop flex and rigid-flex processes.

Williams: So, you were kind of the R&D lead on developing that technology and getting it to become a viable technology here in Tucson. What was that process like, and what were the challenges?

Chiem: From a technology standpoint, there weren't any major challenges, as this technology is 20–30 years old and very mature. I used to work at Honeywell back in the '90s where we built flex as a chip carrier every day and talked about 3-mil lines and spaces 20 years ago.

Williams: It may be new to Prototron, but it's old to you?

Chiem: Exactly. We were doing some crazy PCB technology 20 years ago in Costa Mesa, California.

Williams: And did you have to do anything here from an equipment set standpoint, including changes to processes, or was it just a matter of using different raw materials?

Chiem: Of course, flex and rigid-flex require a different material set, but mainly, we just needed to develop custom frames, or material handling fixtures, to process the thin materials through Prototron's standard, rigid PCB conveyorized equipment. The Tucson shop was set up for standard, rigid multilayer production, and the equipment set reflected that.

Williams: Were there any obstacles you had to overcome because the shop had never built flex here before? Did you have to re-educate the workers or develop new processes and retrain people on how to handle this type of material?

Chiem: We also had to modify our processes and procedures to adapt to flex processing. Then, it was just a matter of blending the current chemistry and equipment with the new frames to make flex a reality here at Prototron. As we've progressed, we've gained knowledge on how to develop the in-house flex capability. I also needed to train people, write specifications, upgrade the procedures, and make sure everybody was familiarized with the new processes.

Williams: Are you also looking at doing rigid-flex?

Chiem: We have developed the processes and ran test orders successfully, so I am confident we can build rigid-flex here right now. It's not difficult.

Williams: I love your confidence. I understand you're currently working on another project with some very small mechanically drilled holes also.

Chiem: Correct. Kim wanted me to work on mechanically drilling 4-mil microvias using our current Schmoll drilling equipment.

Williams: And that would be controlled depth drilling of the microvias?

Chiem: Yes, 4-mil diameter by 5 mils deep.

Williams: Earlier, you mentioned that mechanically drilling 4-mil holes is not that big of a deal; it's the fluid dynamics of getting the chemistry through a blind hole of that size hole.

Chiem: Therefore, we need to put in some new, different chemistry, as the current conventional chemistry that we have right now is not meant for this application.

Williams: So, it's pretty standard microvia stuff, but you are doing it mechanically instead of laser drilling. Excellent. What else is on your to-do list for Prototron in advancing their technology levels?

Chiem: I think we are open to doing R&D collaboration with other companies on advanced technology. For instance, I am working with an organization in Toronto, Canada, to develop a very advanced PCB design. In that way, we have a mutual interest between the two companies. We can co-develop this PCB technology and process that can be applied at both companies.

Williams: Wow! As a fellow old board rat, I find this really interesting. What do you think the timeline is?

Chiem: All I can tell you is that it is an ongoing iterative process as design adjustments are made after each batch of PCBs are built and tested until the process is perfected.

Williams: It's exciting to work with you. I know you're doing a lot of great things here, and I'm looking forward to what you come up with next. Anyone who needs high-quality flex and rigid-flex PCBs can contact Van and the team at Prototron, and they will take care of you. Thanks for taking the time today to talk.

Chiem: Of course. I am always available to help. **FLEX007**

Steve Williams is the president of The Right Approach Consulting. To read past columns or contact Williams, click here.

People With Mobility Issues Set to Benefit From Wearable Devices

The lives of thousands of people with mobility issues could be transformed thanks to ground-breaking research by scientists at the University of Bristol. The FREEHAB project will develop soft, wearable rehabilitative devices with a view of helping elderly and disabled people walk and move from sitting to a standing position in comfort and safety.

Led by University of Bristol Professor of Robotics Jonathan Rossiter, FREEHAB builds on discoveries from his previous The Right Trousers project where his team developed new soft materials that could be used as artificial muscles.

Professor Rossiter said, "There are over 10.8 million disabled people living in the U.K. today. Nearly 6.5 million have mobility impairments. These numbers are growing as the median population age increases and age-related mobility issues due to conditions such as arthritis and stroke become more prevalent."

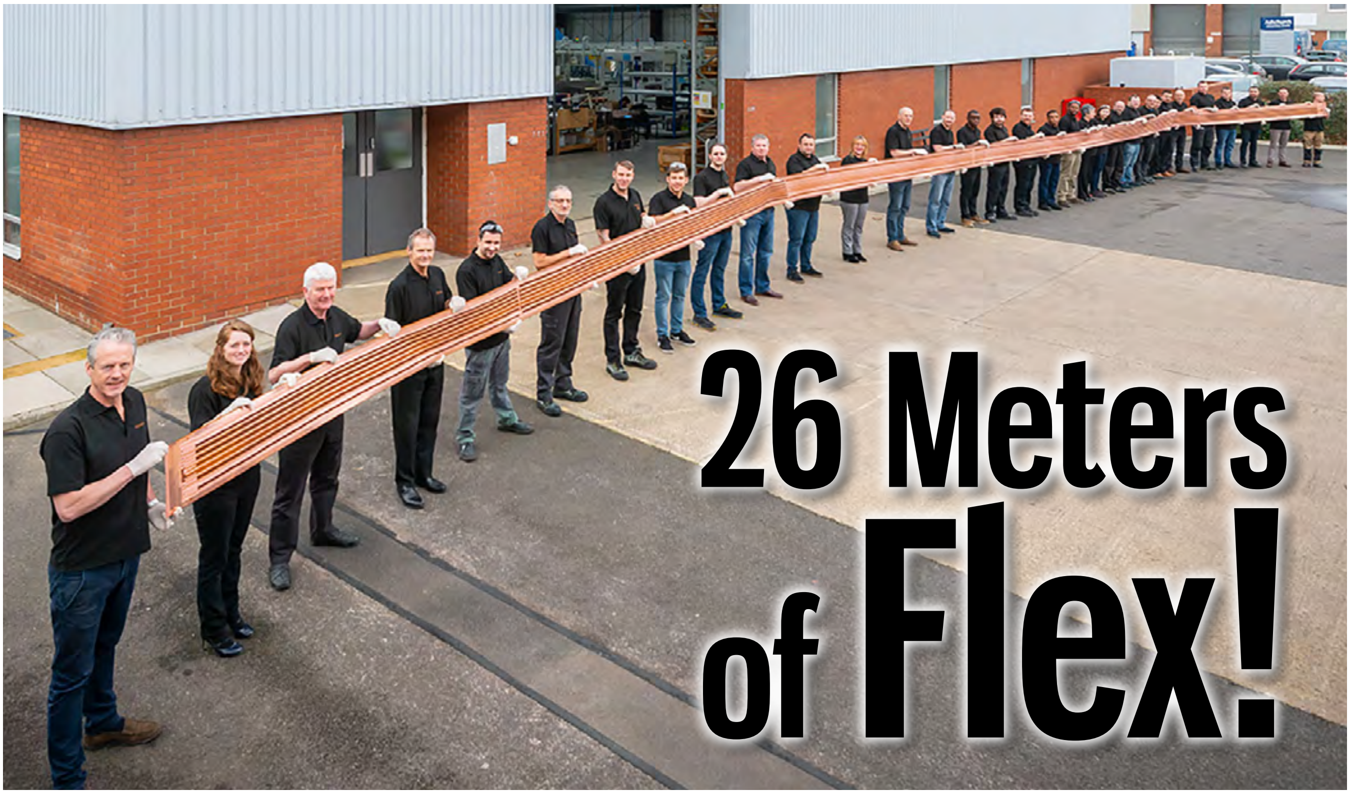
Rehabilitation is vital for patients, but according to Professor Rossiter, outcomes are hampered by a lack of easy-to-use dynamic tools to help therapists accurately analyse mobility performance and devise effective programmes; and as rehabilitation increasingly takes place in patients' homes in the absence of a therapist, better ways to support in-home mobility and training are needed.

The materials from which the artificial muscles are made include 3D-printable electroactive gel materials, and soft but strong pneumatic chains that change shape when inflated and can exert considerable force.

Professor Rossiter said, "Together with integrated sensing technology, we will make devices that physiotherapists can use to accurately pinpoint limitations in their patients' movements, thus enabling them to plan personalised training programmes. We will also make simpler devices that the patient can use to enhance their mobility activities and exercise with confidence when a therapist is not with them."

To develop the project, the researchers will work with physiotherapists in the NHS and private practice and people who have undergone physiotherapy for their mobility problems. Following R&D, the aim is to conduct clinical trials and then bring the devices into the supply chain once the project is over.

(Source: University of Bristol)



26 Meters of Flex!

Flex007 Feature by Barry Matties
I-CONNECT007

I recently spoke with Philip Johnston, managing director of Trackwise Designs, about the company's patented length-unlimited multilayer printed circuits aimed at replacing conventional wire harnesses. Originally created for the aerospace industry, Trackwise has since seen growing interest from a number of different industries. Jake Kelly, managing director and chairman of Viking Test Ltd., also joined the conversation to discuss the importance of having a flexible equipment supplier when dealing with such a unique technology.

Barry Matties: For our readers, can you give us an overview of Trackwise Designs?

Philip Johnston: Trackwise Designs was formed in May 1989. We just celebrated our first 30 years this month. It was started as a PCB design bureau, hence the name. Soon after, we moved into manufacturing, and in the mid-1990s, we were asked to make a nine-foot-long PCB, which was one of the early mobile phone base station antennas. The company developed a means of making long PCBs, and that was originally the main USP.

Now, as the frequency of the mobile telephony has gone up, by the laws of physics, the size of the circuits has come down. Our original USP has morphed into the manufacturing of antennas using printed circuit technology, and we export these all over the world—about 70% of our product goes to export. We've retained our large format manufacturing capability, which is still needed for some niche applications like aircraft guidance radars and long, linear scales for silicon chip manufacturing machines. A few years ago, a large U.K. aerospace engine manufacturer came to Trackwise and said they were looking to replace the wire harness inside of their aerospace engine with flexible PCBs to save weight and space. And because aerospace engines are big, these PCBs were going to be big. They knew that we manufactured these big boards, so they came to see us.

One of the requirements was that the roughly eight-meter-long run should be manufactured as a single piece rather than a daisy chain of in-

terconnected sections. The rest of the world's manufacturing communities said, "No, that can't be done." We had to think about it, and we came up with a means of making length-unlimited multilayer flexible printed circuits. That was the innovation that we first produced a proof of concept sample for and have continued to develop. This patented technology, which we call Improved Harness Technology™, has been granted in the U.K., the U.S., and China, and is in the process of being granted in the EU, Brazil, and Canada—all of the main airframe manufacturing locations. The technology has been focused on the aerospace industry primarily because of its ability to save weight and space.

The technology has been focused on the aerospace industry primarily because of its ability to save weight and space.

This is not a new technology. Flex PCBs have been around for the best part of 100 years, and the reason we can get a cellphone in your top pocket is that all of the wire inside has been replaced by flexible printed circuits. It's now a multibillion-dollar worldwide industry usually used inside of a phone, laptop, or an avionics box. Trackwise is now able to offer box-to-box wiring, so those proven benefits of weight, space, reliability, and precision can now be achieved at the fuselage or wing level instead of just inside of a box.

Our original approach/intent was for the aerospace industry, but I think we are seeing an interesting macrotrend. In all industries—including industrial, aerospace, automotive, and medical—everybody wants more functionality into either the same or less weight or space. If you are trying to do that with wires, it's impossible.

A large business jet manufacturer has been in touch with Trackwise too. They've given us an analysis that shows that the wire content in their business jets is increasing by 25% every five years because more and more functionality is desired. Operators want to fly jets with this greater functionality with as many passengers as they can; therefore, the issues of reducing harness weight and space are becoming ever more significant. As I said earlier, we are seeing tremendous demand from across the industrial spectrum for this new technology, and we're all about length-unlimited multilayer flexible printed circuits.

Matties: A single- or double-sided length-unlimited PCB might be simple, but when you start adding in multilayer, that sounds like quite a challenge.

Johnston: The longest we've made to date is 26 meters long with shielded power cables. The challenges have been many and varied. One of the challenges is to find equipment and machining manufacturers willing and able to modify their equipment for the particular needs of making length-unlimited. Viking Test Ltd. has been a big part of that journey for Trackwise.

Matties: You must have an equipment partner willing to pay attention to these details and make it happen.

Johnston: Yes. In spite of this innovation, Trackwise is still a small company, and we need suppliers who buy into the vision and are willing and prepared to listen to our story and adapt the equipment. If you want to make a 25-meter long roll into a 60-meter long circuit, we don't have a 60-meter long factory, so it has to be done roll-to-roll. Some of the PCB manufacturing steps are effectively static in nature, so we've had to turn those static ones into a dynamic process for Improved Harness Technology™ by looking outside of the industry for technologies that can be applied to the manufacturing process. Otherwise, we've had to find equally innovative and adaptive equipment suppliers, such as Viking, to contribute and help us meet this challenge.

Matties: You started as a design business. Are you still providing design services?

Johnston: I bought the business out in 2000 and focused solely on manufacturing antenna PCBs, so we rebranded the company as Trackwise. We are AS9100 accredited for the manufacturing of products using printed circuit technology. Generally, the first conversation with a new client is, "Here's a wire harness. Tell us what it would look like in flex." Our design requirements and needs have come back to full, so we have readopted the name Trackwise Designs and are changing our AS9100 accreditation to include design and manufacturing.

Matties: What is the primary motivation for customers to come here?

Johnston: Either to save weight and space from their current wire offering or to get more functionality into the same space. Sometimes, those two trends combine.

Matties: But it's more on a large-scale format; you're not looking at building small flex.

Johnston: We are deliberately declining any of what we would consider to be standard.

Matties: You can get that anywhere. What you are offering is unique. Are you the only shop offering this with your patented process?

Johnston: Yes.

Matties: There must be an incredibly large market for you.

Johnston: This has been one of both challenges and opportunities. In theory, the total addressable market is wherever wire is used. We have smart buildings putting sensors into walls, medical devices, and electric vehicles, which is a very interesting growing market for us as well as industrial and resistive and conductive circuits for heating various aerospace and industrial applications.

Matties: Does a typical customer look for design services, or do they already have a design?

Johnston: The initial conversation is to hand us a wire harness and tell them what it would look like in flex. But because a wire harness is so linked up with the system, touches lots of different equipment, and is a 3D geometry, etc., ultimately, the design will be done by the OEM rather than us. But certainly, the initial phase and convincing of the benefits start with the design being done by Trackwise.

Matties: And the benefits are obvious. Are they already looking to eliminate weight or add functionality?

Johnston: Yes, and if you imagine a bundle of wires with 30 conductors, you know approximately where in the circumference each individual wire is. However, in a PCB, you know to the nearest few microns where each conductive track is, that precision and repeatability has great ENC performance and predictability benefits, amongst many others.

Matties: So you were doing a lot of antennas initially. We see antennas being printed on almost any surface these days. How are you addressing that?

Johnston: Our main market is still on Teflon and PTFE boards because that remains the industry-leading substrate for low-loss and very tightly controlled Dk, but we always say we're etchers of copper. We've etched antennas on polyimide, polyester, and a lot of things. We're etchers rather than additive, so that is one of the confusions these days with the PCB. It was probably a good description when it was first coined, but it's now a little confusing with the arrival of additive manufacturing, which is a printed circuit approach.

One of the other advantages of a PCB is that it can be flexible, which, coupled with its planar nature, makes it entirely suitable to being embedded into structures, such as into the bumper of a car or onto the glass of a windscreen. The other very interesting facet that's going to be the killer distinction is that a wire harness is an entirely passive interconnect. Our initial discussion concerns the replacement of a passive wire harness with a passive flex. But ultimately, a flex PCB is a PCB; therefore, the ability to add components to make it a smart harness rather than a passive harness is possibly a second-generation discussion, but that's where the real benefit of the technology is.

Matties: That's exactly where the market is heading—more and more sensors everywhere. I saw some prediction of sensors from where we are today to where we're going to be in a few years, and it was staggering.

Johnston: There was a very interesting discussion yesterday about heating electric vehicles because the energy capacity available for heating electric vehicles is much less than internal combustion. Thus, you have to run at high humidity and closer to the dewpoint, so accurate temperature control is going to be very important. You are absolutely right. I sound like an enthusiastic entrepreneur, but many macrotrends are encouraging the use of flex versus wire harness. Our to offer to the market

I sound like an enthusiastic entrepreneur, but many macrotrends are encouraging the use of flex versus wire harness.

length-unlimited multilayer flex will open up completely new avenues that had previously been closed off due to traditional supply chain manufacturing limitations. There's a discussion as to whether it's the chicken or the egg—has there been no large-scale supply because there's been no demand, or has there been no demand because there has been no supply? Ultimately, we are now going to answer that question.

Matties: Those two are going to meet in the middle somewhere.

Johnston: We think so, and we are seeing that happen.

Matties: You've built a business plan around it, so you must believe that.

Johnston: That's correct. We do believe that.

Matties: This is new in the minds of the marketplace. Where do you see your business in five years?

Johnston: The intent is always that this facility will be a new product development (NPI) type of facility. One of the reasons that we follow the patterns around the world is because we expect that when this is picked up by an OEM, it's likely that the mass-scale production will be wherever in the world is suitable for the customer's needs. We anticipate having a scale-up facility in Timbuktu or wherever it is.

Matties: Is this something that you are going to license out to other fabricators, or is this something you are going to scale internally?

Johnston: We don't need to make that decision at this point in time, but we're having early-stage discussions. We floated on the London Stock Exchange last year to accelerate our development capability.

Matties: How did that work out?

Johnston: We were oversubscribed. Now, we have to walk the walk having talked a good story. It brings some new pressures, but one of the other benefits is that when Trackwise previously talked to large OEMs, it represented a fairly undercapitalized small business with an associated risk profile. Today, that PLC status

Jake Kelly and Philip Johnston.

after our name and a much better finance balance sheet is hopefully a big risk reduction for them. In early discussions with large global interconnect companies about scale-up, having access to capital allows us to say "We'd like to build a self-funded scale-up facility."

Matties: We know that smart factories are an important part of the manufacturing sector. On the manufacturing side of your business, what sort of investments are you making? Where are you at in that process?

Johnston: We are at a relatively early stage of thinking about it, but we're about to kick off a program in that respect. One of our key markets is aerospace, and traceability is a key requirement of the aerospace industry. One of our smart projects is that we would like to be able to have a full product history of each individual product that is going through the whole manufacturing process. We'll be able to call out machine settings, chemistry readouts, etc., into an embedded history of each individual part. The other facet of it is while we have to make a 60-meter-long circuit on a roll-to-roll basis, it also allows us to make two three-meter-long medical devices in roll-to-roll form. The roll-to-roll process allows us to produce a very machine-intensive manufacturing process, which is another feature of smart factories.

Matties: And there's not necessarily an off-the-shelf smart factory product that you are going to buy and plug in because you have to rely on equipment suppliers and software. So, what's your approach to this?

Johnston: We still need to do some research. Part of our program is to see what the off-the-shelf solution is, but we recently bought a new, smart plating line from Jake. We've had discussions around the smarts we're going to be able to extract from that machine as processing is going through.

Matties: Jake, from your point of view as an equipment supplier to an organization like that, can you talk about how working with somebody who's doing something unique has played into your thinking?

Jake Kelly: It's exciting working with a company like Trackwise and to have Phillip get involved and find out what the challenges are, how we can help, and how we can make a specialist tool for a specialist product. Our day-to-day business is generally the same type of equipment, and we like to work with a challenge; it keeps it fresh and real.

And the equipment that we've just supplied here at Trackwise is, as Phillip was saying, a very specialized plating line with wheel-to-wheel, some automation, and other advanced features—a lot of which we can't go into in detail—but it shows that Viking, in particular, is a thinking company. We're adaptable and flexible. We listen to what people want and are prepared to go do it. Sadly, a lot of companies don't do that today. It's more, "This is what it is. Do you want it or not?" or, "Do you want it? If so, how many do you want?" You have to go out of your way, and it's challenging, but you must be versatile. That's where Viking is succeeding today where others aren't doing so well.

Johnston: I would thoroughly endorse that. We have a limited production space here. The plating line would generally be 40 meters long, and we have approximately 12.2 meters. I wouldn't say it was a battle, but we cooperated to fit that functionality into that footprint. And as Jake said, there aren't many other suppliers who are willing to do that, or we haven't found them. It's a big part of what we've been doing.

Matties: It makes a big difference having a cooperating partner that buys into the vision. I completely agree with what you're saying.

Johnston: And particularly if we are doing something no one else in the world is doing.

Matties: I saw a photo with your team holding a flag that was 26 meters or something like that. There's a lot of pride in this, isn't there?

Johnston: Yes, that was funny. It was a freezing cold day and we were all in our company-branded polo shirts outside, and the photographer was on the crane with his thick, wooly coat and hat, and we were freezing. But yes, there's a lot of work that goes in from the team, the suppliers, and everybody to produce that, and the customer is rather shy about allowing us to publicize what they're doing, although it's an absolutely amazing craft. We are proud of what we are doing.

Matties: How many people do you have working here?

Johnston: 50.

Matties: Coming out of the design background and moving into fabrication, that's not a path that we see a lot of the design bureaus taking. What was the thinking behind that?

Johnston: This was 30 years ago, but if you see a design, why not make it as well? But that was before my day.

Matties: Even 30 years ago, there was a move from design to manufacturing.

Johnston: It's not a trivial step. Today, there's more specialism. One generation ago, there was probably more general services.

Matties: Let's talk about designers for a few minutes. We know that in America, there's an aging population with a lot of tribal knowledge that the designers of the future are now entering into. You are still doing some design so you must be hiring some designers.

Johnston: Yes. We've hired graduate R&D engineers who are three extremely bright and very capable people. Considering they're in their first jobs, I'm absolutely staggered by their knowledge. We're all on a learning curve. It was a few years ago, but I looked around and there was nobody under 30 in the business, so we took a purposeful decision to start to train new people. We've had four intakes of apprentices, and training folks is a time-consuming and challenging process. We've had to lose one or two on the way, but we have some real talent out of that. We've been able to because we've introduced new technology and we learn as we develop. Even in my role, I'm learning as we're going along as well.

Matties: For a designer coming into your training program, is there a predefined path or course that you're taking them through, or is there a flavor of the day based on workflow?

Johnston: No, we're starting to produce that. As I said, it's early days for the design side, but we are still primarily and predominantly a PCB manufacturing company. The vast majority of our products are built to print.

Matties: But bringing design back into the mix, you're faced with the same challenges as everybody else. Is your approach to solving it to train them yourself?

Johnston: Yes, it is.

Matties: Is this something that you are finding people are interested in or do you have to convince somebody to become a designer?

Johnston: We haven't struggled with that so far.

Matties: There are a lot of different career paths for people in technology, and layout design may not be on the top end of that. It's just interesting to see what's going on over here. Phillip, thanks for your input today. We really appreciate it.

Johnston: Thank you. We appreciate your time.

Matties: Thank you, Jake.

Kelly: You're welcome. **FLEX007**

Washable, Wearable Battery-like Devices Could Be Woven Directly Into Clothes

Researchers at the University of Cambridge have developed washable, wearable batteries based on cheap, safe, and environmentally friendly inks and can be woven directly into fabrics. The devices could be used for flexible circuits, healthcare monitoring, energy conversion, and other applications.

Working in collaboration with colleagues at Jiangnan University in China, the Cambridge researchers have shown how graphene and other related materials can be directly incorporated into fabrics to produce charge storage elements, such as capacitors, paving the way to textile-based power supplies, which are washable, flexible, and comfortable to wear.

The research demonstrates that graphene inks can be used in textiles to store electrical charge and release it when required. The new textile electronic devices are based on low-cost, sustainable, and scalable dyeing of polyester fabric. The inks are produced by standard solution processing techniques. The research is published in *Nanoscale*.

Most other wearable electronics rely on rigid electronic components mounted on plastic or textiles. These offer limited compatibility with the skin in many circumstances, are damaged when washed, and are uncomfortable to wear because they are not breathable.

The research was supported by the Engineering and Physical Science Research Council, the Newton Trust, the National Natural Science Foundation of China, and the Ministry of Science and Technology of China. The technology is being commercialised by Cambridge Enterprise, the University's commercialisation arm.

(Source: University of Cambridge)

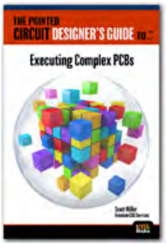
Editor Picks from PCBDesign007

1 Mentor's EDA Perspective on Managing Design Rules ▸

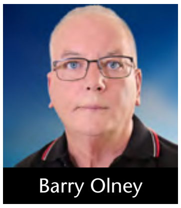
I recently spoke with Dave Wiens, product manager, and Mike Santarini, EDA content director of corporate marketing, both of Mentor, a Siemens Business, about design rules and constraints, and what their customers want regarding design rules. They explained how EDA companies like Mentor help designers constrain for performance while avoiding over-constraining and increasing the cost of the board and also being manufacturing-aware.

2 Freedom CAD Book Excerpt: *The Printed Circuit Designer's Guide to… Executing Complex PCBs* ▸

The following is an excerpt from Chapter 5 on "Design Rules, Simulations, and Analyses" from The Printed Circuit Designer's Guide to… Executing Complex PCBs written by Scott Miller of Freedom CAD Services.



3 Beyond Design: High-speed PCB Design Constraints ▸


Barry Olney

Digital design has entered a new realm. Modern high-speed design (HSD) not only requires the designer to continuously break new ground on a technical level but also requires the designer to account for significantly more variables associated with higher frequencies, faster transition times, and higher bandwidths.

4 Altium's Craig Arcuri on Design Rules: Past, Present, and Future ▸


Craig Arcuri

I-Connect007 recently spoke with Altium's Craig Arcuri about his views on design and manufacturing rules. Craig details some of the challenges with setting and managing hundreds of often divergent design and manufacturing rules, and how both design and manufacturing constraints need to evolve.

5 Wild River, eSilicon, and Samtec Team up for 112-Gbps Test Vehicle ▸


Al Neves and Matt Burns

During DesignCon, Andy Shaughnessy sat down for an interview with Tim Horel from eSilicon, Al Neves of Wild River Technology, and Matt Burns from Samtec. They've recently teamed up to create a 112-Gbps test vehicle that may be the first of its kind of test fixture.

6 Design for the Unknown ▸

Our industry loves DFX, also known as the "design fors." As PCB designers, we not only design for manufacturability (DFM) but we design for assembly, reliability, cost, test, and many more factors which we like to lump together as DFX. But now, I think it's time we embrace a new DFX: Design for the unknown, or DFU.

7 Sensible Design: Thermal Management—Why It Should Be High on Your Circuit Protection Agenda ▸


Jade Bridges

In a previous column, Jade Bridges highlighted a few cautionary notes on the pain points associated with thermal management products, particularly the choices that you will be confronted with, such as which material or product type (i.e., pad or paste) is best suited to your application. In this column, Bridges underlines the importance of getting it right and explains the consequences if you don't.

8 The Bare (Board) Truth: Fabrication Starts With Solid Design Practices ▸


Rick Hartley

It's a fact: Great board design is the key to a great PCB. I'm even more certain of this after spending two days in a wonderful class presented by Rick Hartley titled "Control of Noise, EMI, and Signal Integrity in High-speed Circuits and PCBs." Several times during Rick's presentation, I wanted to slap myself in the forehead and say, "I should have had a V-8!"

9 Words of Advice: What Feature Would You Like to See in Your CAD Tool? ▸

In a recent survey, we asked the following question: What feature would you like to see in your CAD tool? Here are a few of the answers, edited slightly for clarity.

10 The Digital Layout: Spotlight on the Orange County Chapter ▸


Scott McCurdy

This month's column highlights the Orange County Chapter, which is the largest IPC Designers Council (DC) chapter and one of the most active and thriving. Scott McCurdy, Orange County Chapter president, describes the successful format the chapter follows to reach local designers and PCB professionals. You'll also find an update from the IPC DC Executive Board as their collaboration with the new IPC Education Foundation continues.

Career Opportunities

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

Technical Support/Sales Engineer, U.K.

We are looking to expand our U.K. technical support team. As a technical support/sales engineer (home office/Leamington Spa), you will assist potential customers and current customers in appreciating the benefits of using–and optimizing the use of–Ventec materials in their printed circuit board manufacturing processes, enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. You will provide a two-way channel of technical communication between Ventec's production facilities and U.K./European customers.

Skills and Abilities Required for the Role

- HNC, HND, degree, or equivalent in a technical/scientific discipline
- Scientific/technical educational background
- Printed circuit board industry experience an advantage
- Good written and verbal communications skills
- Ability to work in an organized proactive and enthusiastic way
- Ability to work well both in a team as well as an individual
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's Being Offered

- Excellent salary and benefits commensurate with experience

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

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

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

Analyst Programmer, Hong Kong

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

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

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

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

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

apply now

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

Career Opportunities

Field Service Engineer: Multiple U.S. Locations

Reporting to a regional service manager, these customer-focused engineers will uphold the Koh Young culture while delivering professional technical services for our award-winning portfolio of inspection solutions. The role will enthusiastically visit our growing list of customers for installations, training, and evaluations, as well as technical support and maintenance.

We are looking for candidates with a technical degree or equivalent plus three or more years in a production environment with relevant experience. Given our growing customer base, the position will require extensive travel, including some internationally, as well as a collaborative attitude that drives success.

Koh Young is the leading 3D measurement-based inspection equipment and solutions provider. We perform quality control and process optimization across a growing set of industries including PCBA, machining, final assembly, process manufacturing, and semiconductors. In addition to our corporate office in Seoul, our international sales and support offices help us maintain a close relationship with our customers and provide access to a vast network of inspection experts.

Join the industry's leading provider of true 3D inspection solutions. Forward your resume to Michelle.Hayes@KohYoung.com.

apply now

Vision and Machine Learning R&D Engineer

Atlanta, GA or San Diego, CA

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

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

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

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

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

apply now

Career Opportunities

Career Opportunities

Career Opportunities

Gardien

Sales Personnel, Japan

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

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

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

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

Kindly note only shortlisted candidates will be notified.

apply now

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

Career Opportunities

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
+1 916.608.0660 (-7 GMT)

I-Connect007
GOOD FOR THE INDUSTRY

Events Calendar

NEPCON South China 2019 ▶
August 28–30, 2019
Shenzhen, China

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

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

productronica and electronica India 2019 ▶
September 25–27, 2019
Delhi NCR, India

52nd International Symposium on Microelectronics ▶
September 29–October 3, 2019
Boston, Massachusetts, USA

Altium Live—San Diego ▶
October 9–11, 2019
San Diego, California, USA

Altium Live—Frankfurt ▶
October 21–23, 2019
Frankfurt, Germany

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

Additional Event Calendars

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

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

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

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

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

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 PHOTOS BY © GRAFIKPLUSFOTO & © MAT277**

DESIGN007 MAGAZINE

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

July 2019, Volume 8, Number 7
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX