

DECEMBER 2022

iConnect007

DESIGN007

M A G A Z I N E

IPC APEX EXPO 2023
DESIGN CLASSES

Menu

Always fresh...
Always hot

DESIGN FOR

High 
Reliability

Assembly



Flip 
& Chips



Work-
flow



Scrap
Reduction



Rigid-
Flex

WEEKLY SPECIALS

✂ Sunday, January 22 - Thursday, January 26 ✂

Distinctly different.

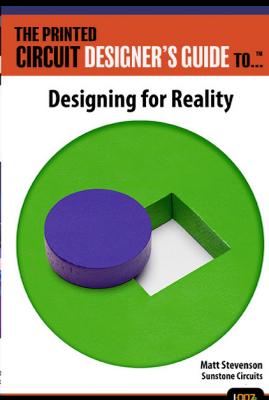
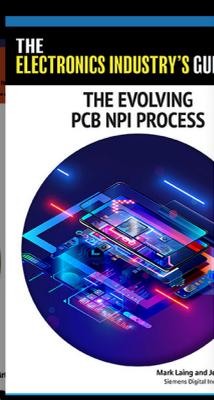
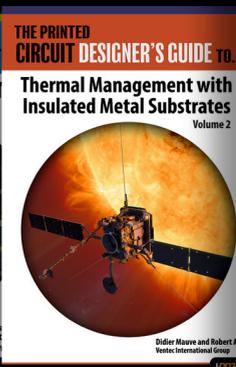
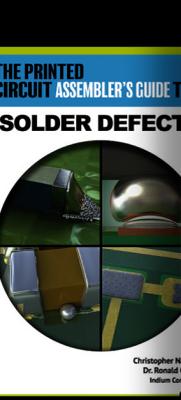
Our books are written by recognized industry experts. At around 8,000 words, they are unique in that they are able to be incredibly focused on a specific slice of technology.

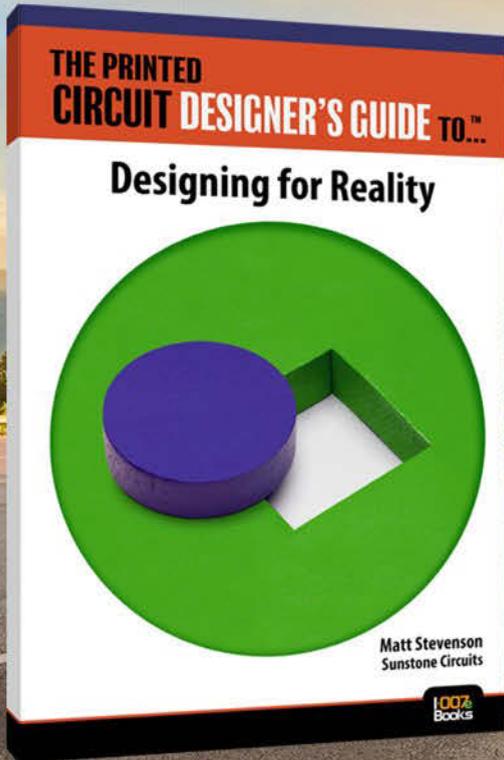
“I-007ebooks are like water in the desert ...it’s up to you to drink it in order to survive!”



Stephen V. Chavez
PCEA Chairman, MIT, CID+

[VISIT OUR LIBRARY](#)





Learn how to create realistic designs and advance your skills!

 [Download](#)

1007^e
Books

DESIGN007

M A G A Z I N E

A Show Within a Show

If you ask anyone in this industry to describe IPC APEX EXPO, they'll probably call it a PCB manufacturing show. But this year's event has more content than ever targeted at PCB designers and design engineers—it's becoming a show within a show. In this issue, IPC organizers and Professional Development instructors discuss their drive to provide more—and greater—curriculum for today's PCB designers and design engineers. Come check out the specials on the menu!

10 **FEATURE INTERVIEWS**
A New Gathering Place for Designers
with Kris Moyer



14 **The Battle of the Boards**
with Patrick Crawford

30 **Designing a Sustainable Future**
with Carlos Plaza



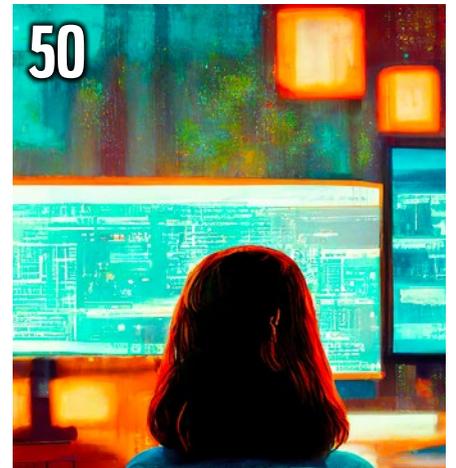
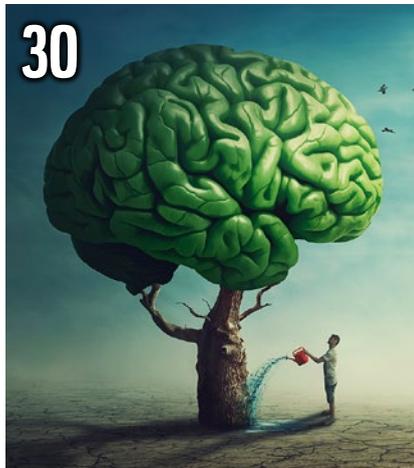
38 **A Sneak Peek With Jim van den Hogen**
Q&A with Jim van den Hogen

42 **FEATURE INTERVIEWS**
Workflow Challenges in Fabrication
Q&A with Paul Cooke

50 **Altium Focuses on Design Education**
Q&A with Rea Callendar



8 **FEATURE COLUMN**
Design at IPC APEX EXPO: A Show Within a Show
by Andy Shaughnessy



**FRUSTRATED
DEALING WITH
MULTIPLE SUPPLIERS?**

Hundreds of customers
have taken advantage of our
"One-Stop PCB Solution"

**QUICK TURN PCB
& ASSEMBLY**

All Under One Roof!

LEADING THE WAY SINCE 1998

Save **20%** on Turnkey PCB
Services *(when you
mention this ad)*

 **GET A QUICK QUOTE NOW!**

PRECISION ADVANTAGE

For the past 25 years, we have been helping PCB Designers, and Engineers avoid unintended design flaws and prevent costly errors. With our technically knowledgeable team and expertise in time sensitive, quick turn PCB Fabrication and Turnkey PCB Assembly services, we guarantee the highest quality and reliability of your products.

Let us help you get your products to market today!

STRATEGIC SOLUTIONS

- ✔ Avoid Communication Gaps
- ✔ Prevent Engineering Delays
- ✔ Consistent Quality Results
- ✔ Prototype and Production
- ✔ Streamlined Process - Fab to Assembly
- ✔ Single Point of Contact from Sales through Delivery



Precision Technologies is certified and registered:
UL94V-0, ISO-9001:2015, ROHS & ITAR,
CAGECode 7T6D9 DUNS 092237267



sales@eprotos.com
1-888-228-9440

Precision Technologies, Inc. USA.

Local: 847-439-5447 | Fax: 847-439-5448 | info@eprotos.com / www.eprotos.com

DESIGN007

M A G A Z I N E



54

SHORTS

- 12 Kulicke & Soffa Extends Advanced Packaging Leadership
- 22 New Stanford Chip-Scale Laser Isolator Could Transform Photonics
- 37 Lam Research Acquires SEMSYSCO to Advance Chip Packaging
- 53 IPC APEX EXPO Show Infographic
- 79 Changing Color of Quantum Light on an IC

DEPARTMENTS

- 83 Career Opportunities
- 94 Educational Resources
- 95 Advertiser Index & Masthead



HIGHLIGHTS

- 24 MilAero007
- 64 Flex007
- 80 Top Ten Editor's Picks



- 26 **COLUMNS**
The Recipe for Customer Service Success

by Matt Stevenson



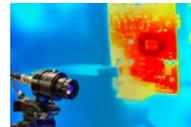
- 34 **Green Coats Are In**

by Saskia Hogan



- 46 **Optimum Thermal Stability Considerations**

by John Coonrod



- 66 **Flexible Circuits for In-line SMT Assembly Processing**

by Vern Solberg

- 70 **A Primer on Using Flexible Circuits in Printed Electronics**

by Happy Holden

- 76 **Flexible Circuits: A Road Less Traveled**

by Joe Fjelstad



ARTICLES & INTERVIEWS

- 18 **Where Exactly Is the Signal?**

by Douglas Brooks, PhD



- 54 **ICAPE: Solutions With No Limits**

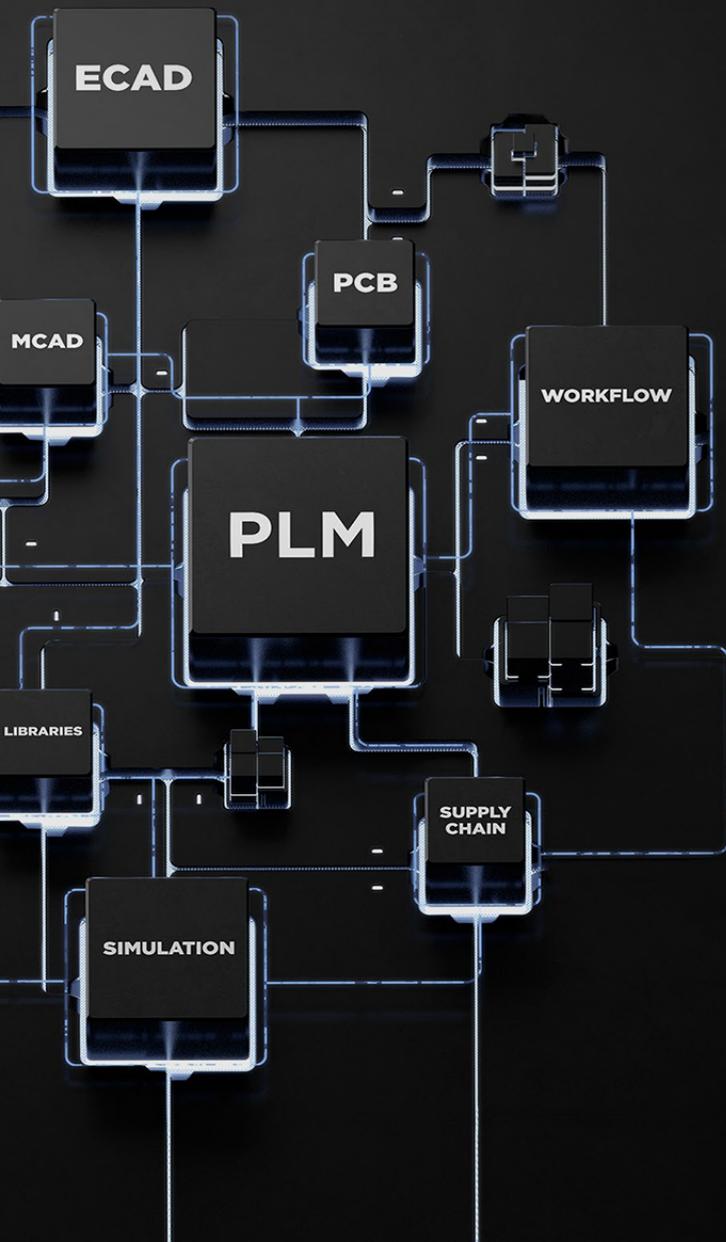
with Yann Duigou and Bingling Li Sellam

- 60 **DFM 101—Final Finishes: Electrolytic Nickel/Gold**

by Anaya Vardya



A comprehensive, digitally integrated electronic design solution for enterprises of all sizes.



- ▶ Digitally-connected ECAD to the rest of the enterprise
- ▶ Cross-domain and cross-discipline collaboration
- ▶ Digitally-managed workflows, processes and resources

As the most lightweight, easy-to-implement electronics design solution for enterprises, Altium NEXUS makes the last mile of digital transformation not only possible - but easy.

Design at IPC APEX EXPO 2023: A Show Within a Show

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

If you ask anyone in this industry to describe IPC APEX EXPO, they'll probably call it a PCB manufacturing show. They're not wrong, by any means; the show was created to serve the PCB fabrication and assembly markets.

But this year's event has quite a bit to offer PCB designers and design engineers. Is this event becoming a PCB design show as well—a show within a show?

Between the Professional Development Courses and Technical Conference, I counted 15 classes that focus on PCB design or design-related topics. There are also plenty of fabrication courses on the roster, which many designers would likely benefit from. Have you visited a board shop since “Cheers” went off the air?

The IPC Design Competition is returning for its second year. The first heat of the competition has already begun, with 47 contestants from around the world slugging it out. The survivors will face off in San Diego next month in a competition that will be live streamed on YouTube and Twitch.io.

There's certainly a thirst for knowledge among PCB designers. Every year, more of the “silverback” senior designers retire to the pickleball courts (seriously, pickleball?), taking with them decades of design knowledge and experience.

Many companies are facing a lack of subject matter experts in PCB design, and the problem is likely to get worse before it gets better. With



several PCB design positions available for each job-hunting designer, some companies are getting creative and developing PCB designers in-house. One manager told us he asks staff in other technical positions if they'd be interested in being a PCB designer. The company will pay for their continuing design education in exchange for their promise to stick around. I bet we'll be seeing more of this: If you can't find PCB designers, create them from the ground up.

Designers are aging out, but there is one big positive note: For the past few years, we've been seeing more young people enter the PCB design segment again. At this point, it's more of a trickle than a tidal wave, but young people seem to have rediscovered PCB design. Maybe we're all doing a better job of marketing this industry. I hope that all the headlines around the CHIPS Act are leading more young people to consider careers in the electronics industry.

All of which means that there's a premium on solid PCB design education. For this issue, we asked the IPC's show staff and Professional Development instructors to discuss their design content, as well as some of the reasons why PCB designers should consider adding this show to their continuing education schedule.

We start off with an interview with Kris Moyer, who answers the question that's on

the mind of designers: "Why should PCB designers attend IPC APEX EXPO? Isn't that a manufacturing show?" Patrick Crawford highlights the IPC Design Competition, including changes to the format and lessons learned during the inaugural event in January 2022. Carlos Plaza discusses the new PCB design curriculum, as well as IPC's drive to increase the PCB design offerings. Instructor Jim van den Hogen of Plexus provides details of his class on fabrication, and why designers who have never been to a board shop should attend. Ventec's Paul Cooke gives us a preview of his fab workflow class, created especially for designers, which illustrates the many factors affecting yield, reliability, and cost during manufacturing. Finally, Rea Callender of Altium discusses the company's education initiatives, which will be highlighted during the show.

We have columns from our regular contributors Matt Stevenson, Saskia Hogan, John Coonrod, Vern Solberg, Happy Holden, and Joe Fjelstad, as well as articles by Doug Brooks and Anaya Vardya.

I hope to see you in San Diego! **DESIGN007**



Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 23 years. To read past columns, [click here](#).

A promotional banner for JobCONNECT007. The background is dark blue with a textured, circuit-like pattern. On the left, the text "jobCONNECT007" is displayed in large, bold letters, with "job" in light blue, "CONNECT" in white, and "007" in yellow. Below this, the text "Companies seeking talent with circuit board industry experience post their jobs with us." is written in white. On the right side, there is a large, stylized magnifying glass graphic in light blue. Inside the lens of the magnifying glass, the website address "jobconnect007.com" is written in white.

A New Gathering Place for Designers



Feature Interview by Andy Shaughnessy

I-CONNECT007

IPC APEX EXPO started out as a show for the PCB manufacturing community, but it's grown beyond that. This year, there's more design curriculum at the show in San Diego than ever before. IPC instructor Kris Moyer has been instrumental in leading the organization's efforts around PCB design and design engineering curriculum. So, I asked him the million-dollar question.

Andy Shaughnessy: Kris, why should PCB designers attend IPC APEX EXPO this year?

Kris Moyer: Because it is the definitive gathering of the electronics manufacturing industry. It is an opportunity to network with peers, representatives from all sectors of the industry. These include fabricators, assemblers, material

suppliers, and equipment vendors to name a few. This makes the show a great opportunity to learn and grow knowledge in all aspects of the industry and how they might impact the designer. Additionally, the Professional Development Courses allow the designer access to training and knowledge direct from industry gurus.

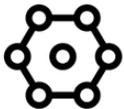
Shaughnessy: I've noticed that there are more PCB design classes set this year. IPC seems to be expanding the amount of design content at the show every year. Is this part of IPC's roadmap?

Moyer: Yes, it is. IPC is committed to supporting and growing the designers in this industry. We all know the challenges we are facing, with

A Total PCB Solution from a Single Manufacturing Partner



With an experienced staff and eight high-tech facilities, Summit Interconnect provides customer-focused services to move your PCBs from prototype through production.



As the largest, privately held PCB manufacturer in North America, we provide the best customer experience and a complete portfolio of high-mix, low-volume products including high-density rigid, flex and rigid-flex, RF/microwave, and semiconductor/ATE PCBs.

Quote your project today!

so many designers reaching retirement age and not as many new designers following in their footsteps. Because we are aware of this challenge, we are actively using our resources to improve the situation.

Shaughnessy: The IPC Design Competition returns this year. Will you be involved in that again?



Kris Moyer

Moyer: Yes, I am one of the judges as well as involved in the conception of the design projects. I've seen the number of competitors and it looks to be really interesting.

Shaughnessy: You teach a variety of design classes for IPC. What are the biggest challenges you see facing PCB designers today?

Moyer: As the speeds of designs increase, there is an increase in the need for designers to understand and grasp many engineering con-

cepts that were not previously part of the designer's skill set. These include signal integrity, power integrity, material properties, manufacturing processes pros and cons, assembly processes pros and cons, to name a few. Designs are only going to get more complex, and faster, and of course smaller.

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

Moyer: As a PCB designer who has been attending IPC APEX EXPO for almost 20 years, I can say I would not have been as successful in my career had it not been for the networking and knowledge I gained there.

Shaughnessy: Thanks for talking with me, Kris.

Moyer: Thank you, Andy.

Kulicke & Soffa Extends Advanced Packaging Leadership

Kulicke and Soffa Industries has inked multiple purchase orders for its thermo-compression solution and has also successfully shipped its first fluxless thermo-compression bonder (TCB) to a key customer.

To mitigate the growing challenges and decreasing benefits of two-dimensional node shrink, the semiconductor industry is aggressively pursuing more complex assembly approaches such as heterogeneous integration (HI) and system-in-package (SiP) for emerging logic, processor, mixed-signal, silicon photonics, and sensing applications. These new approaches are enabling more efficient, transistor-dense packages with higher levels of performance. Innovations by K&S, including its fluxless thermo-compression process are essential to the future of semiconductor packaging.

Through close engagements with industry-leading customers, the fluxless TCB approach is a market-ready solution for next-generation logic

assembly. Kulicke & Soffa's novel fluxless process eliminates contamination concerns while ensuring interconnect integrity through a unique and integrated fluxless delivery module.

Chan Pin Chong, K&S's executive vice president of products and solutions said, "TCB remains the most cost-effective micro-bump solution for the fine-pitch interconnect process. K&S's unique fluxless bonding brings strong product differentiation value for the Thermo-Compression process. With our extensive development efforts, innovations and engagements with several customers, we are well positioned to rapidly support the emerging industry needs for leading-edge logic production."

In addition to fluxless thermo-compression, the company's semiconductor advanced packaging portfolio and development initiatives also address wafer-level packaging, SiP, high-accuracy flip chip, lithography and hybrid bonding applications.

(Source: PRNewswire)

NEW!



IPC Community Magazine is an exciting, new, quarterly publication with a strong editorial focus on members' success. Coming in January!



The Battle of the Boards



Feature Interview by Andy Shaughnessy

I-CONNECT007

Last year, IPC held its first-ever design competition at IPC APEX EXPO in San Diego. PCB designers from around the world competed in a series of heats during the months before the show, culminating in a showdown on the show floor between the top three finalists. Rafal Przeslowski, now with AMD, took home the top prize last year.

This year, the competition is back for its sophomore year. I asked Patrick Crawford, manager of design standards and related programs for IPC, to “lay out” the details on the design contest, including lessons learned in 2022 and what’s new for the 2023 competition.

Andy Shaughnessy: Patrick, I see that the IPC Design Competition is returning for its second year. Give us the rundown on this exciting event.

Patrick Crawford: It is indeed, and we’re happy that it’s coming back. As with last year, the IPC Design Competition is a venue for PCB design engineers to test their mettle. We have two heats of challenge, with the first heat occurring in Fall 2022, and the final heat onsite in San Diego in January. The first heat involves a full board buildup, where designers receive an electrical schematic, a BOM, and a scope of work. They then must input the schematic into their respective design tools and, after they work their magic, output a Gerber file package, assembly, and fabrication drawing. We choose the best three designs and designers, then invite them to compete in the finals heat.

The finals heat is a bit different from the earlier stages: We provide a nearly complete Altium Designer file, so the designers will only be responsible for placement and routing. We

WE BRING MEDICAL ELECTRONICS TO LIFE

With medical electronics,
quality, reliability and
consistency is critical.

Our commitment to quality and
industry-leading technology
has served this demanding
market for more than 30 years.

How can we serve you?

[Check out our capabilities](#)



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits



Patrick Crawford

gave them exactly four hours in 2022, but for 2023, they get a little bit more wiggle room at five hours total, as we expect this year's final challenge will require denser and more difficult routing. This isn't a trivial ask; last year, while nobody took the full four hours, there was a lot of visible head-scratching and "Ah, yes, of course" facial expressions. We also had cast contestants' design tools live via a screen share, so the audience was able to see what they were doing in ECAD in real time, which led to some fun commentary on the floor.

That brings me to a key change for this year's finals. Instead of having the finals competition on the show floor like we did last year, we're moving it out into the conference area—the same place where we have the standards development committee meetings. This will give us a bit more space to play around and do some fun things, like live-stream the event to Twitch.io and YouTube without the need for a sound-cancelling mic setup, etc. This way, we can reach a much wider audience through the power of the internet. However, it will be exciting to be where the action is and accessing that space will be easier this year because it's free to attend our committee meetings.

Shaughnessy: That's a great idea. What sort of PCBs will the competitors be laying out this time?

Crawford: For the preliminary heat, we provided a schematic for a gamepad—basically a controller for a hypothetical video game console. There's some funky geometry to route around, and we've prescribed placement for some critical components, so they'll have some constraints to work with. It's a 10-layer board this year, and I think most people will go with a more traditional copper buildup, but we've given them the opportunity to use a 1+8+1 sequential buildup for microvias if they want. That's something we didn't do last year. The finals heat will also involve a game peripheral, but that's all I'll say for now. It's top secret.

Shaughnessy: What criteria are the judges considering?

Crawford: For the preliminary heat, the judges will look for designs with excellent electrical considerations: proper clearances, correct impedance structures, current via aspect ratios, and the other usual suspects. They'll also be checking to see that the designs are spec'd to the correct grade/modes, producibility levels, and IPC class. There are umbrella considerations like DFM, where a fabrication expert will take a step back and look for things like acid traps, width alterations, and other issues that would require front-end compensation. Maybe it's easier to just name the standards we'll be using as grading tools: IPC-2611, IPC-2612/2612-1, IPC-2614, IPC-2221B, IPC-2222B, IPC-2226, IPC-4761, IPC-2152, IPC-D-325, IPC-6012E, and more.

Shaughnessy: How many competitors are signed up this year, and what are their backgrounds?

Crawford: We had 49 competitors registered, but two of them have had to resign, so we're

looking at 47. I'm including that context because I think it's important to emphasize how flexible this activity is. If someone wants to compete next year but they aren't sure if they can commit yet, they can still sign up—there's zero obligation to participate. This is a fun activity intended to bring designers together in friendly competition. We won't hold anybody's feet to the fire.

These competitors come from a wide area and represent diverse skillsets and experience. Geographically, we have folks from Australia, Austria, Canada, China, Egypt, France, Germany, Hungary, India, Italy, Japan, Nepal, Norway, Romania, Thailand, Turkey, the U.K., and 10 U.S. states competing now.

As far as experience is concerned, their average time in industry is about seven years; we do have some folks with as little as one year in industry, and some as much as 30. We judge the designs blind, so the judges don't know anything about the demographics of the competitors. At first glance, it might seem like those competitors with 30 years' experience under their belts would have the advantage over someone with five years, but last year proved that wasn't the case. In fact, we had a few veteran designers turn in incomplete, ungradable designs.

I can't name names, of course, but the diversity of companies represented is awesome. We have defense contractors, commercial aviation, a toy company—a big one, that's all I'll say—EMS companies, a few ECAD companies, a few automotive giants, medical device manufacturers—the list goes on.

However, in my opinion, the coolest group represented here are the hobbyists, who generally don't have a formal education in electronics, physics, or engineering, or any training in PCB design. But they have made themselves knowledgeable for their own projects. The biggest difference between those individuals and someone who is employed as a design engineer is the understanding of specifications and how requirements impact design. When you're try-

ing to automate your home lighting setup, you don't really care about producibility, right? Unless you're into high-altitude balloons, I doubt you care about elevation impacts. You don't need to design for wave soldering, unless you're sending it off to have boards built. Again, if last year is any evidence, prepare to be surprised, because these folks are crafty, and they get up to speed on specs pretty fast.

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

Crawford: Sure. I briefly alluded to this earlier, but we do allow for any tool to be used for the preliminary heat because you're just exporting engineering files, and the Altium Designer licenses are provided free of charge to install and use on whatever machine you would like to use for the finals. (Huge thanks to Altium, by the way.) We also provide complimentary tool training for Altium if you aren't already versed in it, and we're careful not to include anything in the finals heat that gives an Altium power user a huge advantage. So, don't let a lack of Altium experience stop you from signing up.

Finally, I want to address something that we've seen in our registrant demographics that is unfortunately indicative of a larger issue in the electronics manufacturing industry: There are currently no women registered. I know that there are many women working in our industry, and many of them are PCB designers—I would love to see them get involved in the design competition.

That's all I have. I'm looking forward to a fun event. The finalists will be announced in mid-December. I can't believe IPC APEX EXPO 2023 is just around the corner.

Shaughnessy: Well, thanks for your time, Patrick. I know you've put a lot of effort into the design competition.

Crawford: Thank you, Andy. DESIGN007



Article by Douglas Brooks

(Note: This article is adopted from my new publication, “The Physics of Electrical Engineering for PCB Designers,” available soon.)

When I first got involved in printed circuit board design in the early 1990s, fast rise/fall times were just starting to become an issue. Prior to that we had been pretty much a “connect-the-dots” kind of technology. But as rise times got faster, it became necessary to worry about (electromagnetic) fields. One manifestation of that was EMI, and the increasing need to pass FTC compliance testing.

So, a new type of engineer came on the scene: the electromagnetic compliance/compatibility engineer. Until that time, we understood that electrical current on a copper trace was the “flow” (movement) of electrons along the trace¹. In fact, the definition of an amp of current on a copper conductor is 6.25×10^{18} electrons crossing a surface in one second².

But this new breed of engineers came along and many of them started saying things like³:

- “No, current isn’t electron flow. Electrons can’t flow at the same speed signals flow.” (My response: But they can transfer energy between themselves at the speed of light, which is how they “flow.”)
- “Maxwell and Maxwell’s equations tell us that the signal is in the field around the trace, not on the trace itself.”
- Stop worrying about traces; ignore them. Just control the fields and you will be fine.”

So, before we answer the questions about where the current and signal truly are, let’s look at the fundamental principles behind Maxwell’s equations and see what they say⁴. The following discussion heavily paraphrases these principles and simplifies them for issues relative to this article. No, calculus will not be necessary.

Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

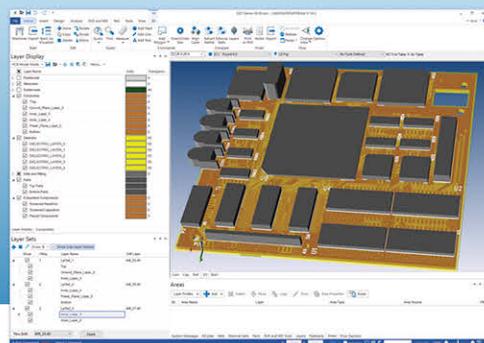
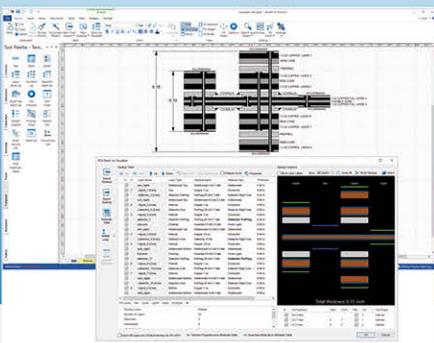
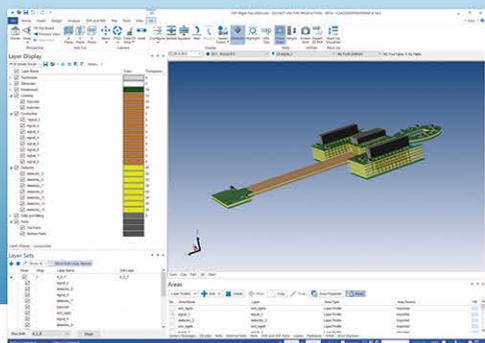


BluePrint-PCB®



CAM350®

- Import and Visualize Flex, Rigid-Flex and Embedded Component Designs
- 3D Visualization to Validate PCB Construction and Component Assembly
- Manage Variable Stackup Zones for Rigid-Flex Designs
- Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation



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

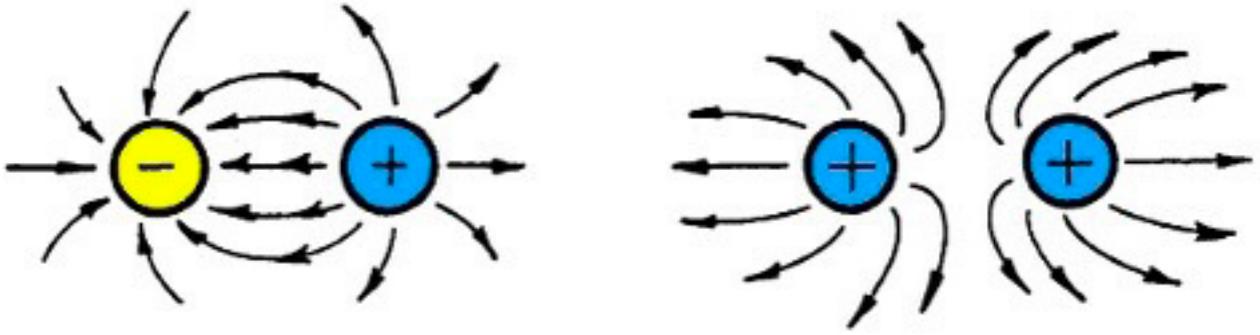


Figure 1: Electric lines of force (field) illustrate Coulomb's Law.

The Principles

Charles-Augustin de Coulomb (1736–1806) was a French physicist. He is best known (at least to us) for developing a couple of laws in the 1700s. One was:

“There are two types of charge, positive and negative. ‘Unlike’ charges attract and ‘like’ charges repel each other (Figure 1) with a force that is proportional to the product of their charge and inversely proportional to the square of the distance between them.”

Coulomb also gave us another law related to magnetism: Every magnetic pole is a dipole with an equal and opposite pole. That is the same thing as saying that a magnetic “north” pole cannot exist without there also being a magnetic “south” pole. Even if you cut a magnet in half (see Figure 2), the individual poles would not be preserved; new poles would appear to preserve the dipole nature of the magnet.

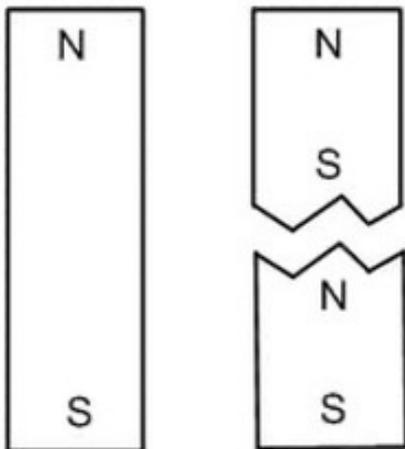


Figure 2: Every magnetic pole is part of a dipole.

This leads to a law similar to above:

“Magnetic force is a vector whose direction is a line along which the force acts (Figure 3). This magnetic force is inversely proportional to the square of the distance.”

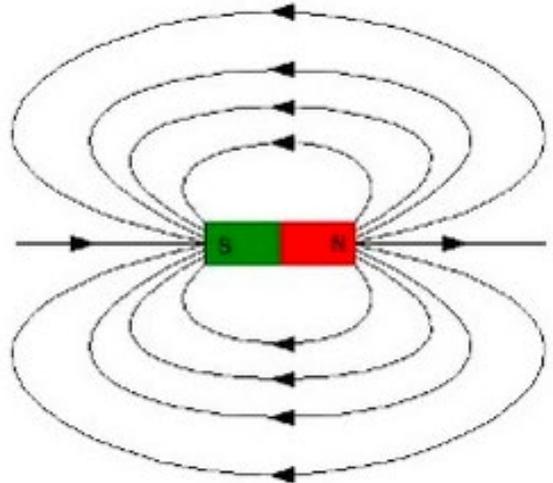


Figure 3: Magnetic lines of force (field).

Andre-Marie Ampere (1775-1836) was a French mathematician and physicist. He is credited with formulating Ampere's Law in 1825 (Figure 4):

“An electric current is accompanied by a magnetic field whose direction is at right angles to the current flow.”

There is an extension of Ampere's Law, credited to Maxwell:

“A changing electric current is accompanied by a changing magnetic field.”

Michael Faraday (1795-1867) had little formal training as a scientist. He was what we might refer to today as a “lab rat.” Most of what

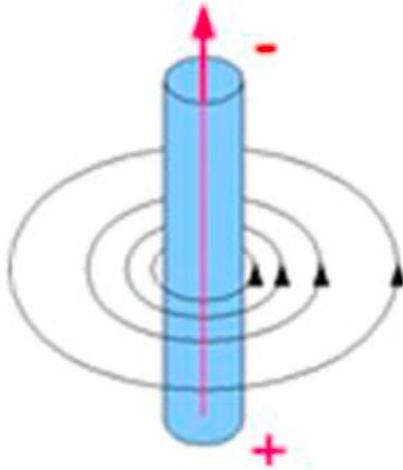


Figure 4: An example of Ampere's Law.

he discovered he did so empirically by experimenting in the lab. He is credited with developing Faraday's Law of Magnetic Induction⁵ in 1831:

"A changing magnetic field is accompanied by a changing electric field at right angles to the change of the magnetic field."

A Side Note on Magnetic Fields

We learned in EE101 that if we send a current down a trace, a magnetic field is formed around the trace. It takes some energy to create that field and that energy is stored in the field. But that is only a small portion of the energy flowing down the trace. We know that because the energy in the trace heats the trace due to the I^2R power dissipation in the trace. If we stop the current flow, the energy in the magnetic field collapses back around the trace and returns to the trace. In a lossless system, there is no energy lost in the field. Only the "real" term in the impedance expression (the resistance) can result in energy loss⁶. The energy that builds up around (ideal) inductors and capacitors is always returned back to the circuit.

The Implications

Think about this: We send a (changing) signal down a trace. This causes a "flow" of current (movement of electrons) along the trace.

The electrons, being charged particles, cause a changing electric field along the trace. The changing current, following Ampere's Law, causes a changing magnetic field along the trace. Thus, there is an electromagnetic field around the trace. So, where exactly is the signal?

This is not an either/or question. I have just described a system where the signal is inherent everywhere. It is in the current along the trace. It is in the voltage gradients along the trace. It is in the electromagnetic field around the trace. We know this because:

- We can measure the changing current along the trace with a probe
- We can measure the changing voltage gradients along the trace caused by IR drops (Ohm's Law)
- We can measure the EMI radiated from around the trace
- If we cut the trace, the electromagnetic field stops
- If we change a characteristic of the trace (physical size or resistance), the current and the electromagnetic field around the trace change
- If we change the terminating resistor at the end of the trace, changing the reflection coefficient, the current, signal, and electromagnetic field all change
- If we change the ϵ_r of the material around the trace (where the field is) the propagation speed of the (signal) current in the trace changes

We also know that these elements are not separable. The electric field cannot get out in front of the magnetic field, the magnetic field cannot get in front of the current, and the current cannot get in front of the electromagnetic field.

Conclusion

Board designers must pay attention to both the electronic circuit (trace, current, Ohm's Law, etc.) and the physics of the electromag-

netic fields around the trace. “Where exactly is the signal?” is a false choice. When we design a board, we must pay attention to all the elements of the system:

- The physical characteristics of the trace (including its impedance)
- How close the signal path is to the return path
- The material environment around the trace
- The electrical environment around the trace (proximity of other signal paths)

Do not pay too much attention to anyone who tells you that only one part of this system matters and the rest does not. **DESIGN007**

References

1. I have written about what current “is” several times. For example, see: *PCB Currents; How They Flow, How They React*, Section 1, Chapters 1-3, Prentice Hall, 2013, and “What Is This Thing Called ‘Current’, Electrons, Displacement, Light, Or What?” reprinted in *UltraCAD’s Best Articles and Applications Notes 2022*, Chapter 1.

2. Do a search on Google for the definition of electrical current and you will find an uncountable number of hits saying this or something close to this.

3. I will try not to let my bias show through here. But it has been my observation that many such engi-

neers think they are superior to all others because they think that they understand Maxwell’s equations and that you don’t. Nonsense. As you will see, there is no choice of “either/or” here. And the dirty little secret is that most of them can’t solve a set of Maxwell’s equations either.

4. *Maxwell’s Equations Without the Calculus*, by Doug Brooks. This booklet describes how these principles are the basis for Maxwell’s equations. Side note: James Clerk Maxwell was a mathematician, not an engineer or physicist. His contribution to all of us was recognizing that these same laws could be combined into a “closed” system and he wrote the equations for them. An outstanding biography related to Faraday’s work as well as Maxwell’s contribution is *Faraday, Maxwell, and the Electromagnetic Field: How Two Men Revolutionized Physics*, by Nancy Forbes and Basil Mahon, 2019.

5. Ampere’s Law and Faraday’s law combined together explain the principles behind motors and generators.

6. The situation is slightly different in electronic transmission (as from an antenna) where we account for energy loss through a virtual radiation resistance.



Douglas Brooks, PhD, is a veteran signal integrity instructor and the founder of UltraCAD Design in Issaquah, Washington.

New Stanford Chip-Scale Laser Isolator Could Transform Photonics

Researchers at Stanford University say they have created a simple, effective chip-scale isolator that can be laid down in a layer of semiconductor-based material hundreds of times thinner than a sheet of paper.

“Chip-scale isolation is one of the great open challenges in photonics,” said Jelena Vučković, a professor of electrical engineering at Stanford and senior author of the study.

The nanoscale isolator is promising for several reasons. First, this isolator is “passive.” It requires no external inputs, complicated electronics, or magnetics. The new isolator is made from common semiconductor-based material and can be manufactured using existing technologies.

The new isolator is shaped like a ring. It is made of silicon nitride, a material based on silicon. The strong primary laser beam enters the ring and the photons begin to spin around the ring in a clockwise direction. At the same time, a back-reflected beam would be sent back into the ring in the opposite direction.

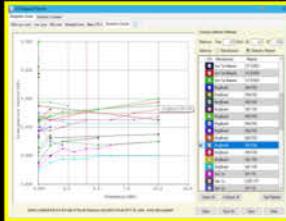
The primary laser then exits the ring and is “isolated” in the desired direction. Vučković and team have built a prototype as a proof of concept and were able to couple two ring isolators in a cascade to achieve better performance.

(Source: Stanford University)

Simulation & Analysis Avoid Costly Re-spins

Now just \$5,000*

Analyzed by Design007 Columnist Barry Olney

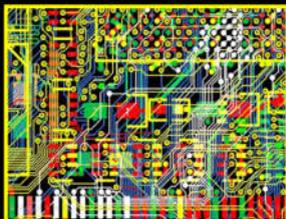


A Comprehensive Report Includes:

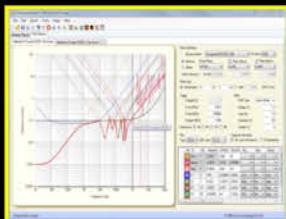
Material Selection for Cost/Performance to Required Frequency and Bandwidth, Design Constraint Review



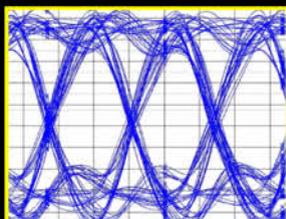
Stackup Impedance Analysis, Single-ended, Differential Pairs and CPW Blind and Buried Via Definition, Reference Plane Assignment Validation



Critical Placement and Routing, Plane Pour Definitions, Return Current Paths, Plane Cross-overs and Broadside Coupling Review



PDN Analysis - Minimizes AC Impedance, Decap Selection, Mounting Inductance Analysis, Plane Resonance Dampening



Critical Net Simulation, EMC Analysis to FCC Class B, Timing Measurement, Termination Review, Crosstalk Analysis

Ensure your next design works!

iCD
In-Circuit Design Pty Ltd

[Discover More](#)

* conditions apply

www.icd.com.au



MilAero007 Highlights



Collins Aerospace, AVIC LETRI Receive FAA TSO Certification for COMAC C919 Weather Radar Antenna ▶

Collins Aerospace and AVIC Leihua Electronic Technology Research Institute (LETRI) announced that they have received TSO certification for COMAC's C919 Weather Radar Flat Plate Antenna (WFA), designed by LETRI, from the U.S. Federal Aviation Administration (FAA).

PCBAA Names Executive Director ▶

The Printed Circuit Board Association of America named David Schild as the association's first executive director. PCBAA is a consortium of U.S.-based companies that support U.S. domestic production of printed circuit boards or PCBs. The association focuses on reshoring American high-tech manufacturing and building a resilient and secure domestic supply chain.

FTG Inks Definitive Agreement to Acquire IMI Inc. ▶

Firan Technology Group Corporation has entered into an agreement to acquire IMI, Inc. based in Haverhill, Massachusetts, north of Boston. IMI is a manufacturer of specialty RF circuit boards focused on the aerospace and defense markets.

Intelsat Chooses SWISSto12 to Build Intelsat 45 ▶

Intelsat announced that scale-up manufacturer SWISSto12 will produce the Intelsat 45 satellite (IS-45). With the order, Intelsat becomes the first commercial customer for the innova-

tive HummingSat geostationary (GEO) telecommunications product.

Ground Control Joins Inmarsat ELEVATE Program to Focus on Satellite IoT Opportunities ▶

Inmarsat, a world leader in global, mobile satellite communications, has launched a new IoT service plan to drive growth and generate long term business for its distribution partners.

American-Made Advocacy: The CHIPS Act is Just the Beginning ▶

Now that the CHIPS and Science Act has been signed into law, the work to secure the entire microelectronics ecosystem must begin. We have a long way to go in restoring balance and resilience in our critical supply chains.

Purdue Defense Research Focuses on Expanding Knowledge of Nontoxic Alternatives for Circuit Boards ▶

Carol Handwerker's life's work is aimed at increasing reliability in everything from the largest Navy carrier to the family car. She does it by researching probably the most inconspicuous material: solder.

Raytheon Technologies Collaborates with Microsoft ▶

Raytheon Technologies is deepening its collaboration with Microsoft Corp. to equip the aerospace and defense company's employees with cloud-based tools, technologies, and platforms to enhance collaboration, optimize operations and unlock intelligent insights that drive greater value for customers.

www.prototron.com



Serious About Speed

For over 30 years Prototron Circuits has led the pack when it comes to providing quality circuit boards FAST.

Be it Class 3, AS9100, ITAR or MIL-PRF-31032, Prototron has the speed you need.



Prototron Circuits

America's Board Source

The Recipe for Customer Service Success

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

Sunstone is 100% committed to both keeping our employees safe and doing well as a company, but we all deal with life struggles and changes differently. We concentrate on the current best practices for keeping our employees safe while keeping quality work as a top priority, and providing a remote work option for our employees has been one such effective strategy. It brings me great joy to report that, so far, we have effectively maintained both the health and employment of all our employees. As I was recently reflecting on this, I decided to ask Al Secchi, our global customer support and sales manager, what he has learned professionally these past two years.

Matt Stevenson: Al, thank you for joining the conversation. What have you and the team

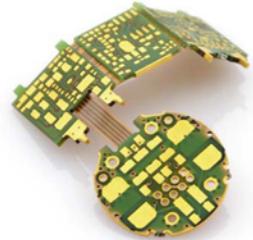
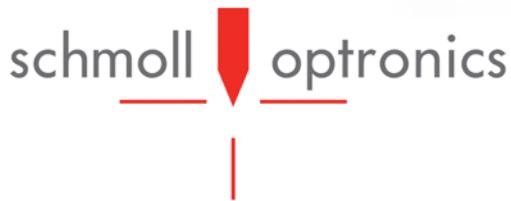
learned over the past few years on how to serve our customers?

Al Secchi: Thank you Matt, it's great to be able to share the ideas my team came up with to answer that very question—how to best serve our customers in this time of unprecedented challenges.

We all got through a pandemic, but we also saw how fragile our global supply chains can be. As time went on, a “new normal” set in that redefined our industry, the country, and the world. Businesses closed, jobs were lost, and essential workers were put into a new environment—one whose war cry centered on being “safe and healthy.” Those who were required to go into the office had to wear masks, had their temperatures taken when they walked in the



Focused on Flex



Laser μ Flex series

The versatility and precision of laser processing makes it the ideal tool for machining a wide range of materials.

Typical applications:

- Polymer cover foil
- Covered polymer laminates
- Thin, rigid-flex materials
- Inner layers and prepregs
- Separation or decap from tracks
- Structuring cavities
- Micro-via and through-hole drilling



For more information call us: +1 (336) 660-2701

burkleamerica.com



Al Secchi

door and throughout their day, had to wash and disinfect continuously, and had to maintain a safe six-foot distance from each other. Workers who could work remotely did so.

Stevenson: How did Sunstone create a customer-centric environment in this type of situation?

Secchi: With so many changes and unknowns, the critical part of creating a customer-centric work environment was put at risk. Some businesses needed to reduce work hours, cut staff, or simply couldn't deliver the product or service that they had promised to their customers. Company buyers and consumers alike were forced to search for new, reliable sources. Without a reliable source, businesses struggled to maintain production, causing a domino effect further down the supply chain—no materials, parts, product, sales, revenue and ultimately, no jobs. Customer service people had their work cut out for them; they had to ensure ongoing business while providing reassurance and peace of mind to both their existing customer base and new customer prospects.

Stevenson: What does customer service look like now, or rather, what does it need to look like to not only meet but exceed the needs of customers so that it's memorable enough to create loyal customer advocates? How do we create this exceptional level of service in today's world?

Secchi: Simply put, it takes a team. Not just any team, but a team of committed customer service professionals who understand both the challenges of the company they work for as well as the needs and expectations of the customers that they also work for. To be able to support a customer and provide the highest level of service, you must first know what you can commit to and how best that commitment fits into the customer's supply chain—both in product and delivery. After all, isn't that what we all want when we're looking for a product or service—a company whose employees understand your needs and are willing to do whatever possible to meet those needs?

Stevenson: What do you think customers are looking for?

Secchi: With all the uncertainty, the one shining light is how a business focuses on its customers and helps them be successful. Sure, everything may be a little different, but the customer's need is still there, and meeting that need is the most important thing. Customers are looking for a "reliable partner" that shows an understanding of their needs; one that gains their confidence and builds a level of trust that they can feel good about. Along with this, customers want to be able to reach you easily in a way they're comfortable with, so you need to provide multiple communication channels via technology—phone, chat, email, and text—so customers can feel confident about you and your work. They also want you to be proactive in contacting them with both the good news and the not so good news. The important thing is that you proactively reach out and be that

CUSTOMER SERVICE RECIPE

Start with a high-quality product or service that you believe in and can stand behind. Add in a thorough understanding of your customer—who they are, what they do, and what they need. Show that you understand them and their needs, and that you care. Then, if (only if) you can meet that need do you share with them how you can help them be successful. Only make realistic commitments that you can stand behind. Be timely in all your communications as you mix in a good portion of proactiveness and work, as any good partner would. Now combine and add an unexpected positive—more than, or faster than expected.

Ingredients:

- 1 part high-quality product/service
- 3 parts understanding customer needs and demonstrating this with understanding and care
- 2 parts sharing how you can help the customer be successful
- 3 parts promises you can confidently commit to
- 2 parts timely and proactive communications
- 5 parts exceeding your customer's expectations

Mix these time-tested, critical ingredients with an easy, multi-channel way of communicating with your customers that includes self-service FAQs where they can easily and quickly find answers to their questions. The result? An award-winning customer service dish.

Lastly, always remember that you can only be as successful as you can make your customers. Good luck in the New Year and remember, be the best customer service master chef that you can be.



honest, helpful partner they are looking for. Don't make them have to contact you; you need to reach out to them in an honest, helpful, and timely fashion.

Stevenson: Can you give an example of how you do this?

Secchi: I tell my customer service team to consider themselves master chefs with “customer service” as a menu option and a main dish that everyone wants to be tasty and satisfying.

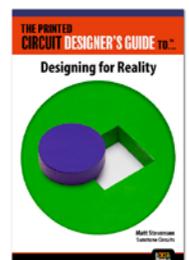
Stevenson: Thank you Al, it's great that your team strives to give customers peace of mind and improve their experience. I agree that it does take a team. A big shout out to you and

your team for their hard work. I love how each member thinks of themselves as a master chef, allowing them to cater and personalize the experience for each customer in each evolving situation. **DESIGN007**



Matt Stevenson is vice president at Sunstone Circuits. To read past columns, [click here](#).

Download *The Printed Circuit Designer's Guide to... Designing for Reality* by Matt Stevenson.





Designing a Sustainable Future

Feature Interview by Andy Shaughnessy

I-CONNECT007

If you've had the chance to peruse the schedule for IPC APEX EXPO 2023, you may have noticed something surprising: A good chunk of the classes cover PCB design this year.

While it may be traditionally considered a PCB fab and assembly show, it's quickly becoming a destination for PCB designers and design engineers. Flipping through this year's schedule, I counted 15 Professional Development and Technical Conference classes that focus on PCB design, as well as several fabrication classes that many of you should probably take.

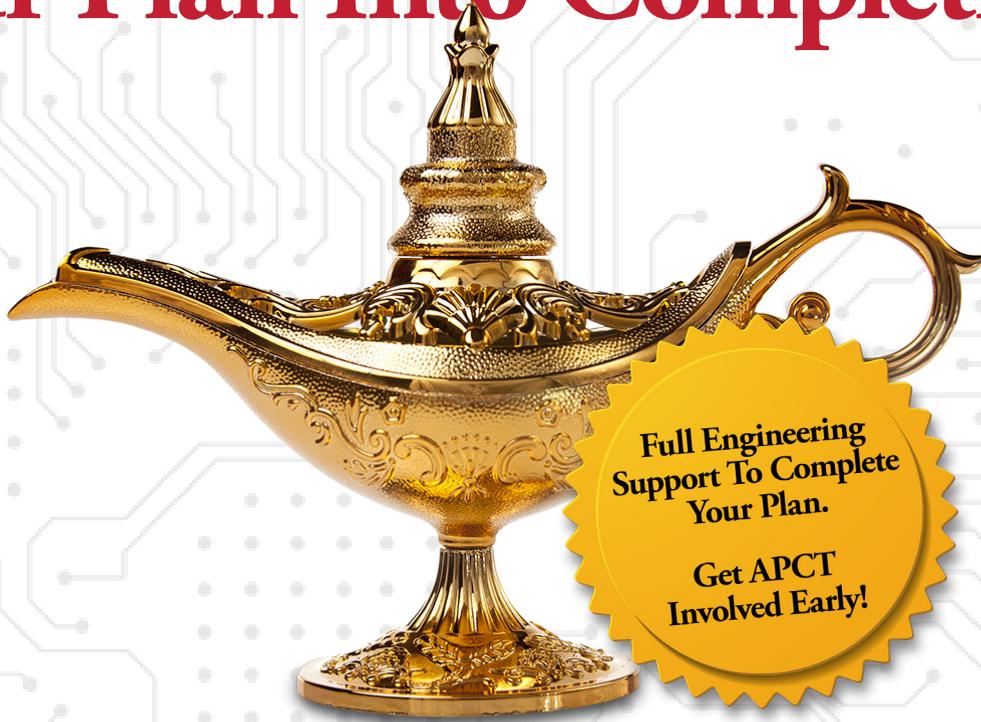
I checked in with Carlos Plaza, IPC's senior director of educational development, to discuss the organization's drive to present more PCB design curriculum at the upcoming show and how the show can give designers the tools they need to overcome the many challenges currently facing the industry.

Andy Shaughnessy: Carlos, I notice that there are more PCB design classes on the docket this year, as well as a bigger focus on design overall at the show. Was that one of your objectives for 2023?

Carlos Plaza: Yes. To accommodate the large number of registrants we've had this year, we've expanded the availability of our existing courses so that participants now have more opportunities to take high-demand courses like PCB Design I & II, PCB Design for Military Applications, and Advanced Troubleshooting and Defect Analysis. We have also responded to industry members' requests for additional courses on highly topical subjects. Look for new courses on design for excellence, troubleshooting for PCB assembly, implementing Industry 4.0 technologies, and more.

Printed Circuit Board Solutions

You Just Can't Wish Your Plan Into Completion



Full Engineering
Support To Complete
Your Plan.

Get APCT
Involved Early!

We're Ready For The Challenge

HDI Technology | RF Circuit Design | Miniaturization

APCT Is The Solution

Markets Served

- Defense & Aerospace
- Industrial
- Automotive
- Telecommunications
- Semiconductor
- Medical



APCT

Passion | Commitment | Trust

APCT.com

APCT Santa Clara HQ

408.727.6442

APCT Anaheim

714.921.0860

APCT Orange County

714.993.0270

APCT Wallingford

203.269.3311

APCT Global

203.284.1215

Shaughnessy: I understand that you made some improvements to the design classes this year. Tell me about that.

Plaza: Improvement is an integral part of our process. Ultimately, we want our curriculum to help designers and engineers do their jobs more effectively. To achieve that goal, we make a point of following up with course participants about their experience and meet with subject matter experts to identify topic areas of interest to industry professionals. The information obtained from these conversations, as well as the feedback obtained from our end-of-course surveys, are then used to update content and tailor the delivery of each course on a regular basis. Recent improvements, for example, have ranged from tweaking class starting times to adding new modules that address emerging technologies and methods.

Shaughnessy: You're in touch with a lot of these designers and design engineers. What are their biggest challenges right now?

Plaza: At an organizational level, the retirement of large numbers of baby boomers poses an ongoing challenge. From now until 2030, approximately 10,000 boomers will retire every day. The pandemic exacerbated this pattern, as



Carlos Plaza

many older workers decided to retire early rather than risk infection. This unprecedented brain drain means that organizations are forced to constantly onboard new workers and upskill new and current staff members to both preserve decades of institutional knowledge and keep up with the lightning-fast pace of technological innovations, not to mention the concomitant evolution of industry standards and customer requirements. The shortage of skilled candidates also means that many current staff members are being asked to quickly learn new skills to fill the breach.

Shaughnessy: Why should PCB designers attend IPC APEX EXPO and take some courses?

Plaza: The Professional Development program brings together the most experienced industry experts in every area of PCB design. It's the ideal opportunity to learn from those who have already made the big mistakes and developed processes that will help you maximize your effectiveness and efficiency.

Shaughnessy: Thanks, Carlos.

Plaza: Thank you, Andy. DESIGN007

I.007e
WORKSHOP

American Standard
Circuits

On Demand: Free Training Video Series

FLEXIBLE CIRCUIT TECHNOLOGY

with Expert Joe Fjelstad

WATCH NOW

Integrated Tools to Process PCB Designs into Physical PCBs



Visualize

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



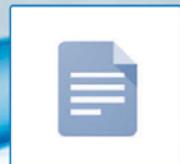
Verify

Ensure that manufacturing data is accurate for PCB construction.



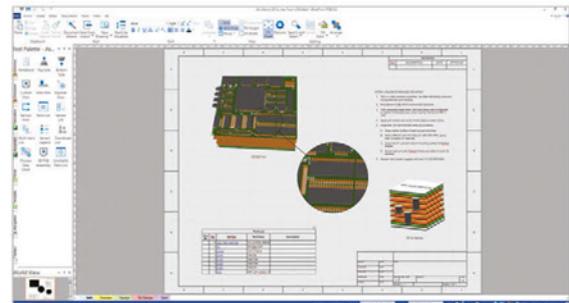
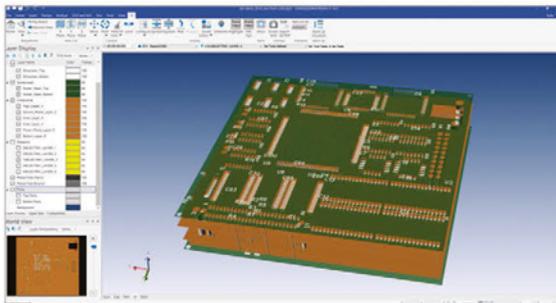
Panelize

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



Document

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



CAM350

Verify and Optimize PCB Designs for Successful Manufacturing



BluePrint-PCB

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

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



800-535-3226 | downstreamtech.com

©2019 DownStream Technologies, Inc. All rights reserved.

NOW WITH **3D**

Green Coats Are In

Sensible Design

by Saskia Hogan, ELECTROLUBE

It goes without saying that bio-based coatings have significantly less impact on the environment and are better able to address the ethical and sustainability requirements of manufacturers and end users than conventional solvent-based coatings. However, the most surprising benefits observed during the development of many new bio-based conformal coatings have been the significant improvements to performance and reliability.

The bio-based material we've developed, for example, has shown improved condensation resistance, thermal stability, flexibility, and adhesion compared to many petrochemical-derived materials. This bio-coating has distinguished itself as a top performer with the

environmental credentials to match, making it a win-win solution. Making the switch to bio-coatings can also help manufacturers "future proof" their processes. For example, with all the new climate-change targets currently being suggested, including some environmentally conscious measures requiring manufacturers to make drastic changes in just a couple years, those who have already transitioned to bio-based proportional alternatives will have a significant leg up on the competition.

Let's examine why these conformal coatings, based on organic, renewable ingredients, can help manufacturers stay one step ahead without having to compromise performance. Here are two questions and answers to consider.



LEADING

MANUFACTURING OF UNRIVALED PCB TECHNOLOGIES

Mike

Expertly processes panels to build complex internal via structures.

Sarah

Ensures that surfaces and surface finishes are free of defects and compliant.

John

Manages and moves inventory and materials throughout the facility.

Brian

Copper plating, Strip-Etch-Strip, and AOI expert, trainer, and problem solver.

Danielle

Inspects PCB VIAs in process for plating or etching defects.

Rich

Maintains a safe and beautiful corporate campus.

Trey

Mechanical CNC routing and scoring expert.



 **Calumet**
ELECTRONICS CORPORATION

DOMESTIC
MANUFACTURING
READY


100%
MADE IN
AMERICA

ONSHORE
TRUSTED
FOR DOD AND
COMMERCIAL
OEM ELECTRONICS
MANUFACTURING

AMERICA'S ONLY
HUBZONE
PCB MANUFACTURER

calumetelectronics.com

Do you see the future of coatings as “greener,” or will there always be a demand for the more traditional solvent-based coatings?

Solvent-based conformal coatings are the most used conformal coatings in the electronics market, and there are many reasons why: They are easy to use, cost-effective, and there is a wide selection available. Whilst solvent-based conformal coatings are incredibly popular and will surely stay with us for quite some time, we are currently seeing a shift to a more careful attitude regarding our finite resources and a desire for more environmentally friendly and sustainable conformal coating products.

Specifically with younger generations on the electronics consumer side, but also in the workplace with product designers, electronic design engineers, chemical engineers, marketers, and

Nature is providing us with better solutions that are as sustainable as they are efficient...

others, there is a growing interest in sustainability and environmentally friendly solutions, as well as a willingness to create products that really make a difference. This attitude shift has been filtering through the supply chain, creating pressure to meet this demand for a greener approach to manufacturing and, ultimately, a greener future overall.

Alongside this environmentally conscious perspective, a demand for protective materials that will allow electronics to withstand harsher environments has also materialized in the industry. Such requirements are becoming more stringent every day, and the pressure is on manufacturers to meet this evolving need. As it turns out, our research has shown that natural materials—including materials from

food waste, for instance—can provide similar or better properties and protection than conventional options. With such promising characteristics, it seems only logical to dig into these renewables for coating product development. Nature is providing us with better solutions that are as sustainable as they are efficient, so there appears to be a definite future for “greener” conformal coatings.

What is the benefit of a secondary chemical cure in UV curing conformal coatings?

When we look at the traditional UV-cure conformal coatings, we find materials that have a UV primary cure and a secondary moisture cure. This means that wherever the correct UV light wavelength reaches the coating, it will cure the material within a matter of seconds. However, the issues lie within the areas that are not reached by the UV light. Between high components or under component parts, we still want the conformal coating to cure to ensure even protection. For those areas, we will need the secondary cure.

Many common UV-cure coatings use atmospheric moisture for their secondary cure, which can release by-products that produce a strong odor. In applications where the PCB is placed in an airtight enclosure, the available moisture can be limited, potentially resulting in an insufficient cure and limited protection of the electronics. Therefore, it is important to leave the coated PCB until the secondary cure has fully taken place. Depending on the UV coating, the secondary cure can take days, weeks, and, in some cases, even months.

However, with a chemical cure as the secondary cure, the chemical cure will happen with the conformal coating. This is because, rather than relying on a single component material, this process uses a material that consists of two parts, part “A” and part “B,” that will react when mixed to complete the cure. So, after the UV primary cure has taken place, the material will go into the secondary cure phase without the need for any environmental input.

The secondary cure will also happen much faster, depending on the type of coating, the coating thickness, and the temperature; normally, a secondary cure can be completed in 16 to 24 hours. The coated PCB can be handled and, in most cases, even assembled into an enclosure straight away. As there is no need for moisture, the enclosure can be closed without the worry that some areas have failed to cure.

This is a huge advantage for customers, as it reduces or even diminishes the need for excessive buffer storage between production and assembly. This, together with a faster production process, means huge cost savings in production. With general manufacturing costs going through the roof, saving costs wherever possible has become a priority for many.

Regardless of the cost-saving benefits, real progress means looking at how we can ensure that our products are environmentally friendly; the degree of performance improvement achieved with these innovative new coatings is only a small part of what makes them so exciting. Whilst the development focus for such materials has been very much geared toward

meeting the demanding requirements of automotive and aerospace applications, the high-performance levels of these materials make them a great choice for any application. These products really come into their own whenever thermal shock cycling and condensation resistance are at play, though these are by no means the only failure mechanisms.

As we all continue thinking about how we can revolutionize our processes to reach that “greener” future, consider how bio-coatings might benefit both your business and the environment—these materials may just be the solution you’ve been looking for. **DESIGN007**



Saskia Hogan is global product manager, conformal coatings, at Electrolube. To read past columns from Electrolube, [click here](#). Download your free copy of Electrolube’s book, *The Printed Circuit Assembler’s*

Guide to... Conformal Coatings for Harsh Environments, and watch the micro webinar series “Coatings Uncoated!”

Lam Research Acquires SEMSYSCO to Advance Chip Packaging

Lam Research Corp. has completed the acquisition of SEMSYSCO GmbH, a global provider of wet processing semiconductor equipment from Gruenwald Equity and other investors. The acquisition of SEMSYSCO broadens Lam’s packaging offerings, bringing a portfolio of innovative cleaning and plating capabilities for chiplet-to-chiplet or chiplet-to-substrate heterogeneous integration. This includes support of fan-out panel-level packaging, a game-changing process in which chips or chiplets are cut from a large, rectangular substrate sheet several times the size of a traditional silicon wafer. This approach enables chipmakers to significantly increase yield and reduce waste.

“Packaging plays an important role in extending Moore’s Law and enabling future leadership products with higher levels of system in package integra-

tion. New substrate-based panel-level approaches are vital to cost-effectively realizing the high-performance chiplet-based solutions needed for the digital world,” said Keyvan Esfarjani, chief global operations officer at Intel Corporation.

“The strategic acquisition of SEMSYSCO furthers our commitment to help chipmakers address their emerging technology challenges, adding deep capabilities in advanced substrates and packaging processes,” said Tim Archer, president and chief executive officer at Lam Research. “With innovative offerings and leading-edge research and development in packaging, Lam is well-positioned to support our customers as they scale to future chiplet-based technologies.”

(Source: Lam Research)



A Sneak Peek With Jim van den Hogen

Q&A with Jim van den Hogen

Jim van den Hogen has been teaching PCB designers and design engineers about fabrication processes for decades. Twenty years ago, I had the opportunity to see a class of his at PCB West; even back then, the room was jammed with designers eager to learn more about DFM techniques. Now Jim is bringing his teaching expertise to IPC APEX EXPO 2023 this January with a similar class directed at PCB designers. I asked Jim to give us a sneak peek into his curriculum and to share what he hopes attendees will take away from his class, as well as his thoughts on how to best bridge the gap between design and fabrication.

Q Jim, could you tell us a little about your Professional Development class at IPC APEX EXPO 2023?

A The class walks you through a basic multilayer fabrication process from front to back. It includes discussions on material, stack-ups, via metrics, aspect ratios, solder mask, surface finish, and various other fabrication-related topics. I have several sample sets that I'll pass around for students to inspect. Each sample set has over 25 individual pieces, all of them taken at various steps of a multilayer fabrication process, so students will be able to visualize the concepts we'll be discussing in the class. We will also cover the typical requirements for a PCB fabrication drawing. The final portion of the class will cover array or manufacturing panel design, and ways to maximize cost savings when working with these processes.



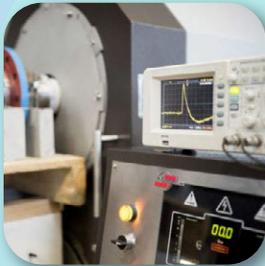


Candor

Circuit Boards

Innovative solutions for complex circuit board manufacturing

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



RF/Microwave
PCBs



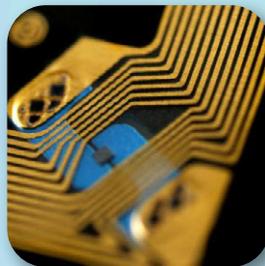
Power
Electronics



High-Speed
Data



Automotive
Electronics



Lamination



Startups

Our technology produces what our customers dream!

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



Jim van den Hogen with Mayim Bialik from IPC APEX EXPO 2017.

Q Why should someone attend your class?

A Students in my class will gain a strong foundation in the basics of PCB fabrication and will get to go in-depth on the complexities of the process. You will learn how your design and specification thereof can affect the fabrication process and, in turn, how the delivered PCB can affect the assembly process.

Q It's a fabrication class, but your target audience includes both PCB designers and design engineers. I know many designers who haven't been in a board shop in decades, if at all; does that seem to be the case with your classes?

A Yes, there are many designers and engineers that have never set foot in a board shop. It is difficult to understand the complexity of board fabrication without firsthand experience. While the PCB Fabrication Basics course presentation will get you close, a "behind the glass" board shop tour can tie it all together.

Q What is the most important piece of advice you give your attendees?

A Engage with your suppliers up front. This becomes increasingly more important as your designs become more complex. Work with your supplier to define a fabrication stack-up that includes material selection, copper weights, plating thicknesses, and via/trace/space metrics. You will want to discuss what qualification specification(s) the finished product will need to meet. This may be an IPC specification (typically IPC-6012 and IPC-4101) or a corporate specification. This should be done as early in the design process as possible, preferably before you start placing parts in your design.

Q Is there anything else you'd like to add?

A This class has been presented before and most of the attendee feedback comments mentioned how the sample sets that are passed out during class make it easier to grasp many of the topics presented.

Also, while at the conference, you should plan on attending a standards development committee meeting. If you disagree with what is written or would like to help improve or update current specifications, that is where you'll have your chance to make your voice heard. **DESIGN007**

BENDING THE POSSIBILITIES



BY TAIYO

AVAILABLE IN TWO COLORS



AMBER



BLACK

- 🏆 Excellent bendability
- 🏆 High Resolution by photolithography
- 🏆 High Reliability
- 🏆 Simplified process
- 🏆 UL94 VTM-0 certified
- 🏆 Low spring back force
- 🏆 Resolution Line/Space=50/50 μ m
- 🏆 Solder mask defined opening (30 μ m diameter)

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.



Follow us on social media





Workflow Challenges in Fabrication

Q&A with Paul Cooke

The job of the PCB designer is becoming more complicated every day. Designers are now acting as what amounts to project managers for the PCB, and they must be familiar with the many trade-offs that happen during manufacturing, many of which they can control during the design cycle.

Paul Cooke, the senior director of business development for Ventec International Group, is presenting a class at IPC APEX EXPO 2023 that looks at workflow challenges in fabrication, and the myriad drivers that can affect yield, reliability, and cost. Here he discusses the details of this Professional Development course, what he hopes attendees will take away, and why designers and design engineers would benefit from this class.



Q Paul, tell us a little about the Professional Development Course on process flow and defects you are teaching at the show.

A I put this course together to give designers, engineers, etc., a more in-depth look at what happens to their design as it goes through the fabrication process. It looks at how the fabricator adjusts processes to meet the design intent and what happens when it goes wrong; it's like a [IPC] 2221, 6012, 600, and

PREMIER GLOBAL SUPPLIER of **FLEXIBLE CIRCUITS**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Superior design consultation & support

Flexible Circuits

Rigid Flex

Flexible Heaters

Flex/Rigid Flex Assemblies

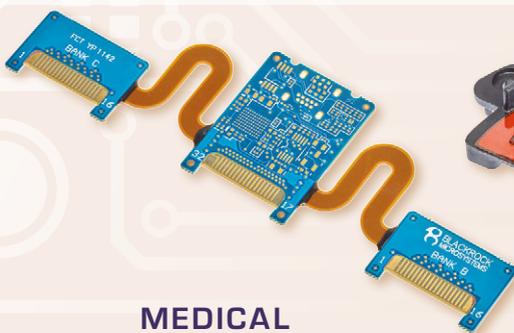
Membrane Switches

Plastic Moldings

Specialized EMS/Assembly

Product Module to Complete Box Builds

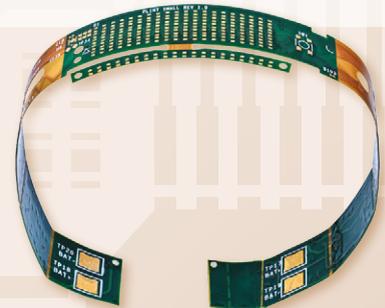
**EXPLORE the
POSSIBILITIES!**



MEDICAL



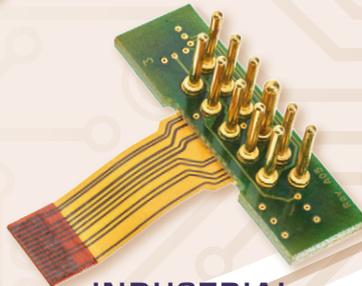
AUTOMOTIVE



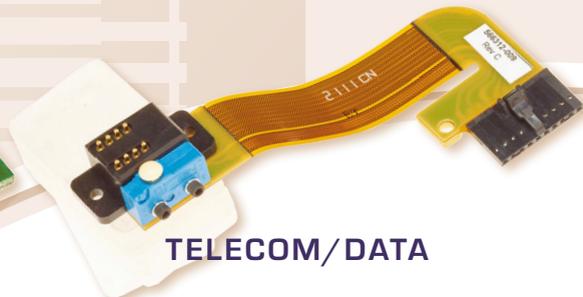
CONSUMER PRODUCTS



WEARABLES



INDUSTRIAL



TELECOM/DATA

FLEXIBLECIRCUIT
Technologies[®]

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

9121 course all rolled into one. It shows the complexity of the process and how designs can affect the process flow, yield, quality, etc. We talk about reliability, for example, of how something like via size can be affected by aspect ratio, layer count, drill parameters, desmear, and material selection and how they would all interact with each other to affect yield, cost, and reliability.



Paul Cooke

Q Why should someone attend your class?

A It's for anyone wanting to increase their knowledge of the fabrication process. It is aimed at designers and process engineers at any level. The course is designed to be interactive with a lot of questions and hopefully 30 years of knowledge transfer. Time permitting, we may get to cover some emerging technologies.

Q This sounds like a great class for PCB designers. What are some of the trade-offs designers should be aware of that can affect yield and reliability?

A There are many factors affecting yield, reliability, and ultimately cost. We will look at each part of the process, discussing how each can be affected, which attributes of the design have the most impact on yield improvement, and how the fabricator achieves the desired level of release.

Q What is the most important piece of advice you would give your attendees?

A Attend the course with a lot of questions and take a lot of notes as we will cover a mountain of information. By the end I hope they have a comprehensive knowledge of the process and learn to always talk to a field application engineer prior to setting out on a new type of design, to eliminate any changes later due to cost, capability, yield, etc.

Q Is there anything else you'd like to add?

A I hope the attendees all enjoy the course, and I am always looking for feedback, good or bad, so that I can adapt and improve for future classes. **DESIGN007**



Get smarter here.



MEETINGS
& COURSES Jan. 21-26

CONFERENCE
& EXHIBITION Jan. 24-26
SAN DIEGO CONVENTION CENTER | CA

ADVANCE IN A NEW ERA

IPC EMS Leadership Summit

Monday | January 23 | 8:00 AM-5:00 PM
6:00 PM Networking Dinner at Osteria Panevino

EMS Leaders of Today and Tomorrow Helping Each Other and the Industry to Prosper

The IPC EMS Leadership Summit brings together current and future industry leaders to solve problems, build business networks, and share insights to doing business better. Focused on high level topics which drive business growth and financial success, leaders gather insights from experts and discuss their own and potential new best practices.

This meeting of minds inspires action and builds resources for participants, future leaders, and the greater EMS industry.

IPCAPEXEXPO.ORG

#IPCAPEXEXPO

ALL EMS LEADERSHIP SUMMIT REGISTRANTS QUALIFY FOR ONE FREE LICENSE FOR IPC'S ELECTRONICS ASSEMBLY FOR ENGINEERS. ONCE REGISTERED OUR TEAM WILL CONTACT YOU TO PROVIDE ACCESS TO THE COURSE.

Optimum Thermal Stability Considerations

Lightning Speed Laminates

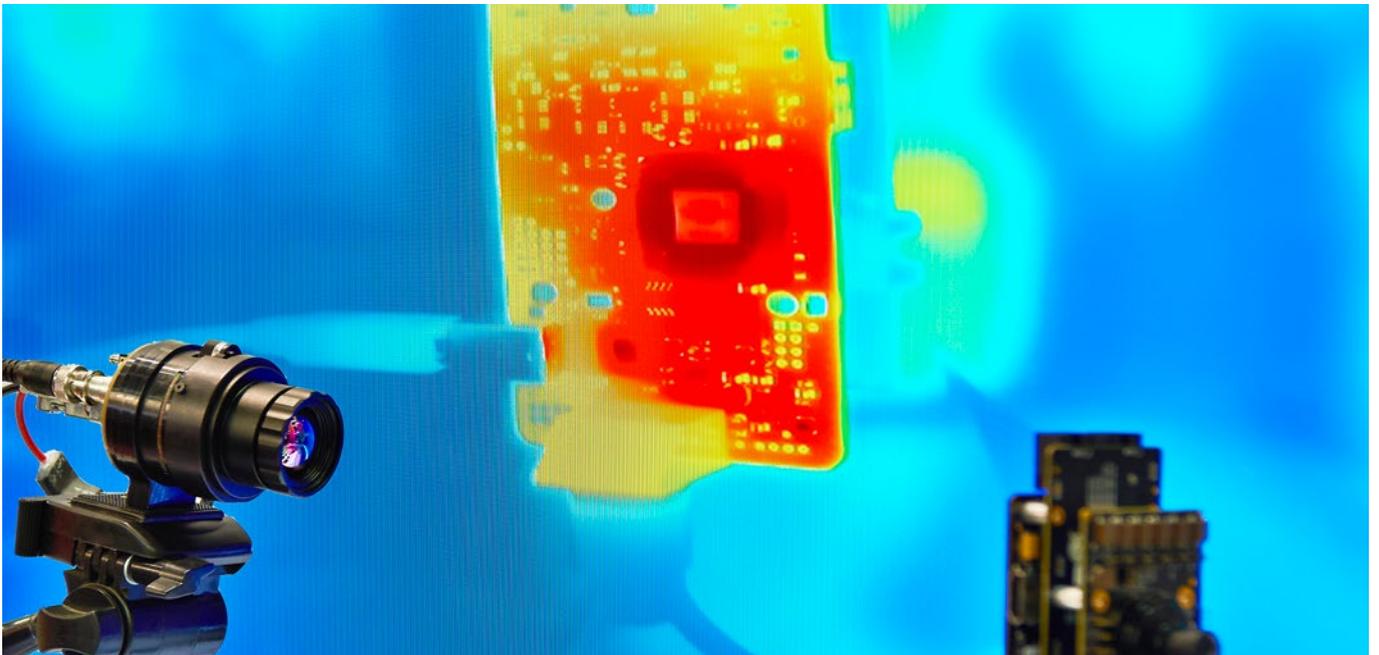
by John Coonrod, ROGERS CORPORATION

As technology advances, thermal expectations are becoming more critical, and designers need to be aware of the many aspects that can alter the performance of a circuit due to thermal issues. Thermal concerns can be different for digital applications vs. RF applications, and when DC or AC power is included, they can further complicate the matter.

Thermal management is a very broad term for many potential thermal concerns for PCBs. Probably the most common reference to thermal management is related to keeping the circuit temperature below a critical limit by considering the many variables that can impact circuit heating. However, there are other thermal issues which can impact the RF or digital performance of a circuit. Thermal

coefficient of dielectric constant (TC DK) is a property that all materials have, and it is characterized by how much the Dk of the material will change given a change in temperature. There are also related properties such as thermal coefficient of dissipation factor (TC Df) and thermal coefficient of insertion loss (TC IL).

The TC Df and TC IL relate to how much the Df or the insertion loss can change given a change in temperature, respectively. Additionally, a circuit performance change due to aging can be significantly impacted by the thermal behavior of the circuit and its operating environment. Finally, moisture absorption is normally not considered as part of thermal management concerns, but it can be.



We are dedicated
to excellence through

innovation, technology and most importantly, service.



COMMERCIAL • MILITARY • MEDICAL • BAREBOARD



U.S.CIRCUIT

www.uscircuit.com



As a general statement, the amount of power that can be applied to a PCB is related to the temperature rise it will cause; this is critical to ensure the circuit temperature does not violate the maximum operating temperature (MOT). The MOT is the maximum temperature at which the circuit can operate without degradation to critical properties of the circuit. There are many things that will impact the heating of a circuit, including the removal of the heat through good heat flow management and the use of heat sink technology.

There are many things that will impact the heating of a circuit, including the removal of the heat through good heat flow management and the use of heat sink technology.

A typical circuit thermal management example is to have heat generated on the top signal plane and have a heat sink attached to the ground plane on the bottom of the circuit. The heat flow path will originate at the signal plane, migrate through the substrate, and terminate at the ground plane below. The ground plane is at the same thermal potential as the heat sink, which is attached to that plane.

Basic heat flow concepts suggest that using a thinner substrate will shorten the heat flow path and move the heat more effectively from the signal plane to the heat sink, which enables a lower circuit temperature. Another option for optimizing heat flow is to increase the size of copper areas. Because copper is an extremely good thermal conductor, the larger the copper area between the signal

plane and the ground plane, the wider the heat flow path(s), which assists in maintaining cooler circuit temperatures. Additionally, since the heat flow path is going through the substrate in this example, the thermal conductivity of the substrate can also be significant. A circuit material with higher thermal conductivity will increase the heat flow and aid in lower circuit temperatures. A simple rule of thumb is that a thermal conductivity of 0.5 W/m·K or greater is considered good for a circuit material.

To complicate circuit heating matters further, due to RF power being applied to the circuit, insertion loss is a major concern. Basically, an increase in insertion loss will give an increase to the heat generated for the circuit. In some cases, designers will ignore the heat flow concerns previously mentioned and consider using very low loss circuit material to generate less heat. As thermal management has become increasingly demanding, this simple approach is often not adequate.

Material choices may result in undesirable consequences due to interactions between the multiple thermal management properties. For example, using a thinner circuit material to increase heat flow typically means higher insertion loss and more heat generated. For this thermal management tradeoff and many others, a good thermal simulation model is needed to understand the different relationships at play. That aside, if a thinner substrate is used, which has very low loss (low Df), using copper with a smooth surface (which gives lower conductor loss) with high thermal conductivity creates an optimum scenario for good heat flow and minimizing circuit temperature. The thin substrate gives a short heat flow path and its low Df gives lower insertion loss (i.e., generates less heat); by combining this with smooth copper, which also gives lower insertion loss, and high thermal conductivity, the heat generated is moved to the heat sink very effectively. Rogers has a material formulated with these optimal thermal management properties. This

laminates has a Df value of 0.0017 when tested at 10 GHz, a variety of substrate thicknesses, choices of copper type with different surface roughness, and very high thermal conductivity of 1.24 W/m·K.

TCDk is another related thermal stability issue that is very often considered for circuits operating in environments with changing temperatures, or a circuit that will change temperature significantly due to changing duty cycles. As a general rule of thumb, a good TCDk for circuit material is 50 ppm/°C. The ideal TCDk would be 0 ppm/°C, but few circuit materials have numbers in that range.

Moisture absorption is typically not considered with thermal stability issues, but in some cases it should be.

The TCDf issue is more difficult to accurately measure due to the influence of heating on different mechanisms which are part of the test method, thereby affecting the results. TCIL is a better measure in this regard because it is typically more accurate and has the benefit of being a real-world measurement. TCDf considers the change of Df, which is related to dielectric losses only. However, TCIL shows performance change in terms of the overall loss of a circuit given a change in temperature.

The TCDk, TCDf and TCIL are parameters which show instantaneous changes in certain properties due to a temperature change. However, circuits and their materials can have changes in properties due to long-term aging. Long-term aging at room temperature is typically a minor issue, but if aged at elevated temperatures, circuit and material

properties can change in shorter timeframes. A higher temperature will cause the circuit/material to change aging properties quicker. As another general rule, circuits made with thermoplastic materials will have better long-term thermal aging performance than thermoset materials. However, there are some thermoset materials which are formulated to be very well-behaved for long-term thermal aging.

Moisture absorption is typically not considered with thermal stability issues, but in some cases it should be. A circuit material with a high moisture absorption property can change Dk and Df due to the circuit being exposed to changing humidity levels in the operating environment. This effect can be exaggerated with higher temperatures and should be considered. However, a possible exception for this issue is a circuit operating at or above water evaporation temperature. Even if the material has a high moisture absorption property, if the temperature is too high to allow water vapor to accumulate within the circuit material, the performance difference due to moisture uptake will be minimal.

Thermal stability of a PCB can be affected by many different circuit material properties. Also, a circuit's design and construction will impact the thermal performance of the PCB in the end-use application. It is highly recommended to contact your material supplier when working with thermal management issues. **DESIGN007**



John Coonrod is technical marketing manager at Rogers Corporation. To read past columns, [click here](#).



Q&A with Rea Callender

Altium keeps its eyes on the designers of the future. The company has been working with colleges and universities for years, providing free seats of Altium Designer for the next generation of PCB designers and design engineers. At IPC APEX EXPO 2023, Altium will be providing software for the finalists in the IPC Design Competition just as it did last year. They offer a variety of other educational programs as well, including Upverter classes and a design competition that aims to address environmental change.

Here, Rea Callender, Altium's VP of education, discusses its educational programs and plans for the week of the show.



Q You head up your education programs. Tell us about some of the company's educational efforts.

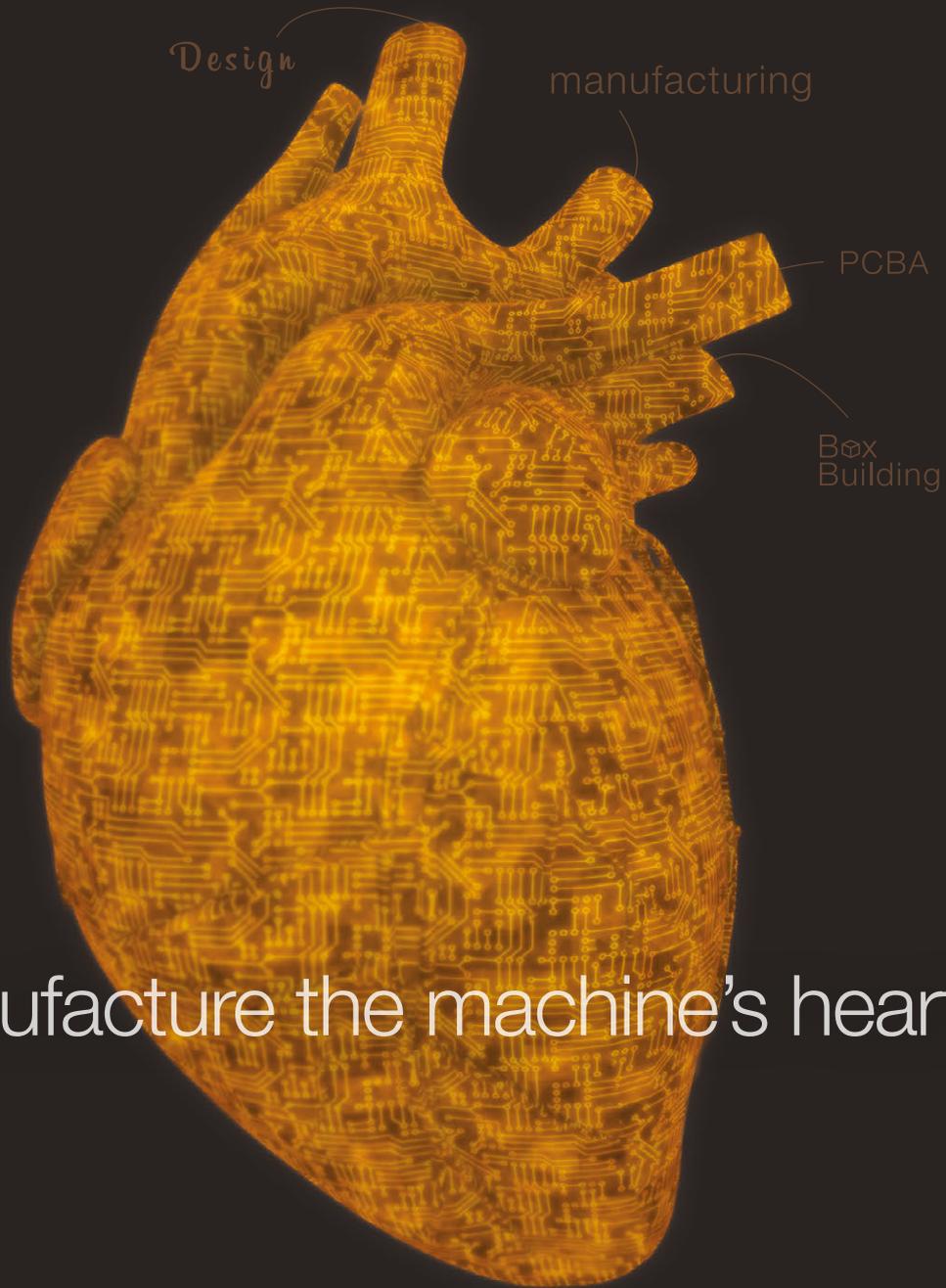
A Our mission is to attract talent to the field of electronics design, and equip students

with the proper skills so that they can enter the workforce and drive the next generation of technology and innovation to a higher level.

In addition to Upverter Education, our most recent initiative, Altium Education, prepares college students to advance to industry-level printed circuit board design and manufacturing careers. It offers a free Altium Designer student license and curriculum for college and university students interested in engineering and electronics design. Our courses take students step-by-step from learning the basics of electronics to designing their first printed circuit boards.

We have taken a unique approach by developing the curriculum needed to teach PCB design and providing free Altium Designer licenses. Educators are using the curriculum to introduce PCB design in multiple ways in universities globally, ranging from a prerequisite for EE courses at the University of Pennsylvania to a standalone PCB design course to be launched by MIT.

Our mission is to provide equal access to all, with specific outreach to women in electronics design to expand the pool of talent. Today, women comprise only 28% of the workforce in STEM fields. To address this concern, we are



Manufacture the machine's heart ...

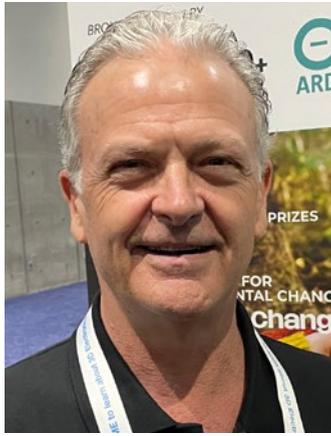
We offer an all-in-one solution through our experience, integrity, and clear communication. Get on board for our joint journey into the future.

www.PCB-technologies.com

A L L I N
ONE

 **PCB TECHNOLOGIES**
leading the way

launching a scholarship program for women enrolled in universities and colleges. The scholarship provides free Altium Professional Training and Altium Designer licenses. Currently, 50 women are enrolled in the pilot program, from countries including Brazil, India, and Pakistan. The first year goal is to provide over \$1 million in grants to 175 women students worldwide.



Rea Callender

Q For the second year in a row, you are providing software for the finalists in the IPC Design Competition, while also being involved in the PCBTheChange Global Design Competition. This contest is co-sponsored by Altium, the IPC Education Foundation, and Arduino. Tell us about it.

A Yes, this is our second annual PCB design contest focused on Innovation for Environmental Change. Last year we had an amazing turnout with 87 teams from 17 countries. It was fascinating to see the projects and designs coming from high school and college students from around the world.

We are delighted to support emerging engineers in this way and we are looking forward to an incredible turnout this year as well. We launched the contest in August and competition winners will be announced mid-December.

There are two categories—high school and college—with first prize of \$1,500, second prize of \$750, and third prize of \$500. Additionally, Arduino will supply a free Arduino Portenat H7, and we will manufacture the winning Upverter designs for free, an estimated value of \$2,500. The winners' designs and finished products will be displayed at the Altium/IPC Design booth at IPC APEX EXPO 2023.

Our engineers and staff will be on hand to answer technical and design questions as the students finalize their design submissions and

prepare their team presentations. We will present the results jointly with the IPC Education Foundation. We're expecting over 250 students to attend.

Q Are your classes primarily online, in person, or a mixture?

A The education curriculum, including an Altium Designer student license, is available to students free and online at Altium Education. Universities and college instructors are also integrating the curriculum into their course syllabuses as well.

Q What are the biggest technical challenges for PCB designers today?

A The biggest challenge for emerging engineers and designers is access to professional tools and, more importantly, training on the basics of PCB design and the electronics design process from concept to manufacturing. Many students graduate with engineering degrees but no hands-on experience or knowledge of actual PCB design. Those who are able to learn PCB design skills and understand the collaboration needed throughout the design process will be at a big advantage as they search for a job and launch their careers.

Q Any closing thoughts?

A We have been recognized for our ability to equip educators and students with an innovative virtual electronic design curriculum and engineering learning solution. This unique approach to education has been celebrated for its ability to empower educators and students alike. Altium received the Gold Stevie American Business Award in 2021 and 2022, as well as the EdTech Cool Tool 2022 and EdTech Breakthrough Awards in 2021. **DESIGN007**

THE WORLD OF ELECTRONICS MANUFACTURING ALL AT ONE EVENT

Connect, learn, and advance at the largest international gathering of industry professionals.

THE LARGEST INTERNATIONAL GATHERING

of electronics professionals in North America with representation from 61 countries



ALL SKILL LEVELS OF ATTENDEES



Beginner



Intermediate



Advanced

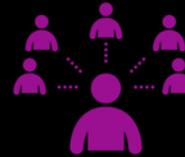
TOP 3 REASONS TO ATTEND

NEW!



Exposure to
New Products

(The largest exhibition in North America featuring the industry's leading equipment manufacturers and suppliers)

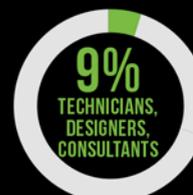
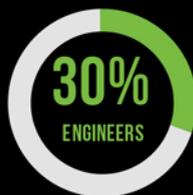


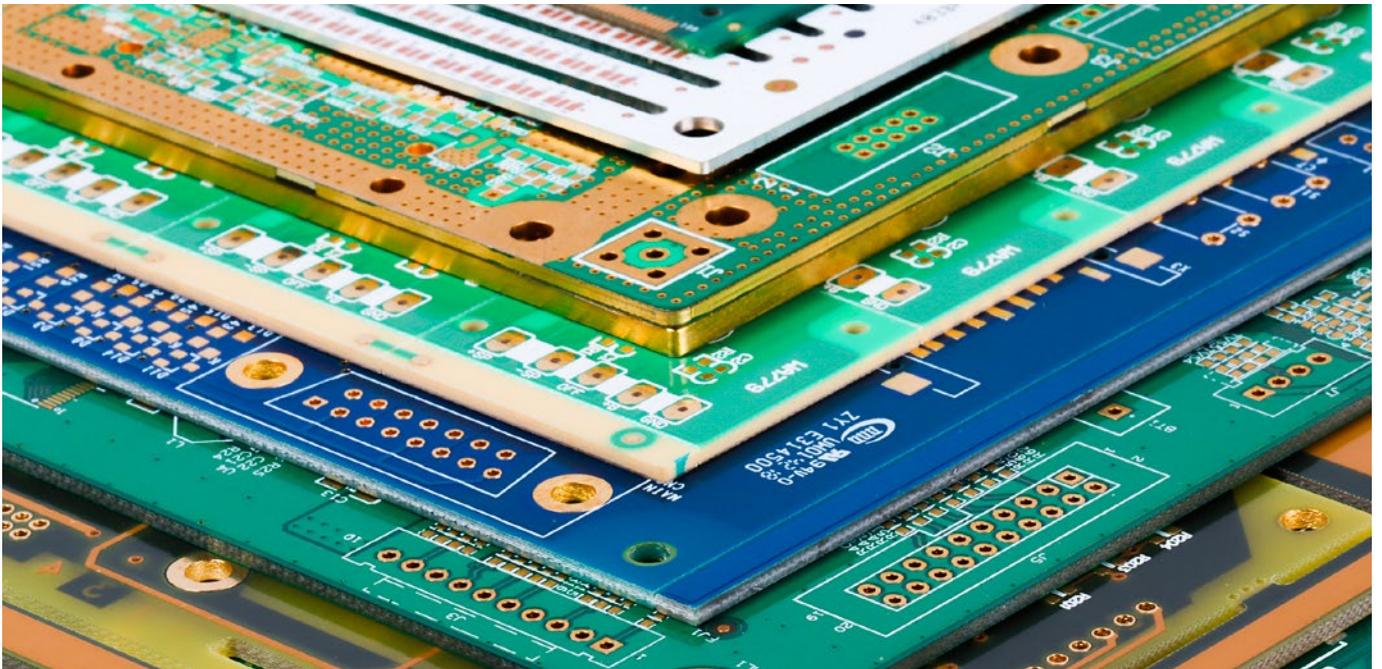
Network
with Industry
Colleagues, and
Technical Experts



Training
and Education

WHO'S ATTENDING





ICAPE: Solutions With No Limits

Interview by Nolan Johnson

I-CONNECT007

Over the past five years, ICAPE Group has strongly focused on analyzing the complex needs of the market to provide worldwide solutions for customers whose needs range from simple to very complex technologies. The services provider has done this by putting offices in key strategic locations—particularly China—with staff that understand the nuances of language and culture. These types of decisions make an important difference when it comes to getting customers exactly what, and when, they need it. Yann Duigou, CMO, and Bingling Li Sellam, VP of Northern Europe, share their secrets for success.

Nolan Johnson: Yann and Bingling, what dynamics do you see in the market?

Yann Duigou: We see that the electronics market is growing by about 12% per year, which is confirmed by the statistics we read in industry publications. The war in Ukraine has cre-

ated a lot of issues with sea and air transport. This makes the market very difficult. There's also a lack of components, as you know, and customers are always working on where to buy them. In fact, we just met with some customers today, trying to find brokers for components. All these elements are disrupting the markets.

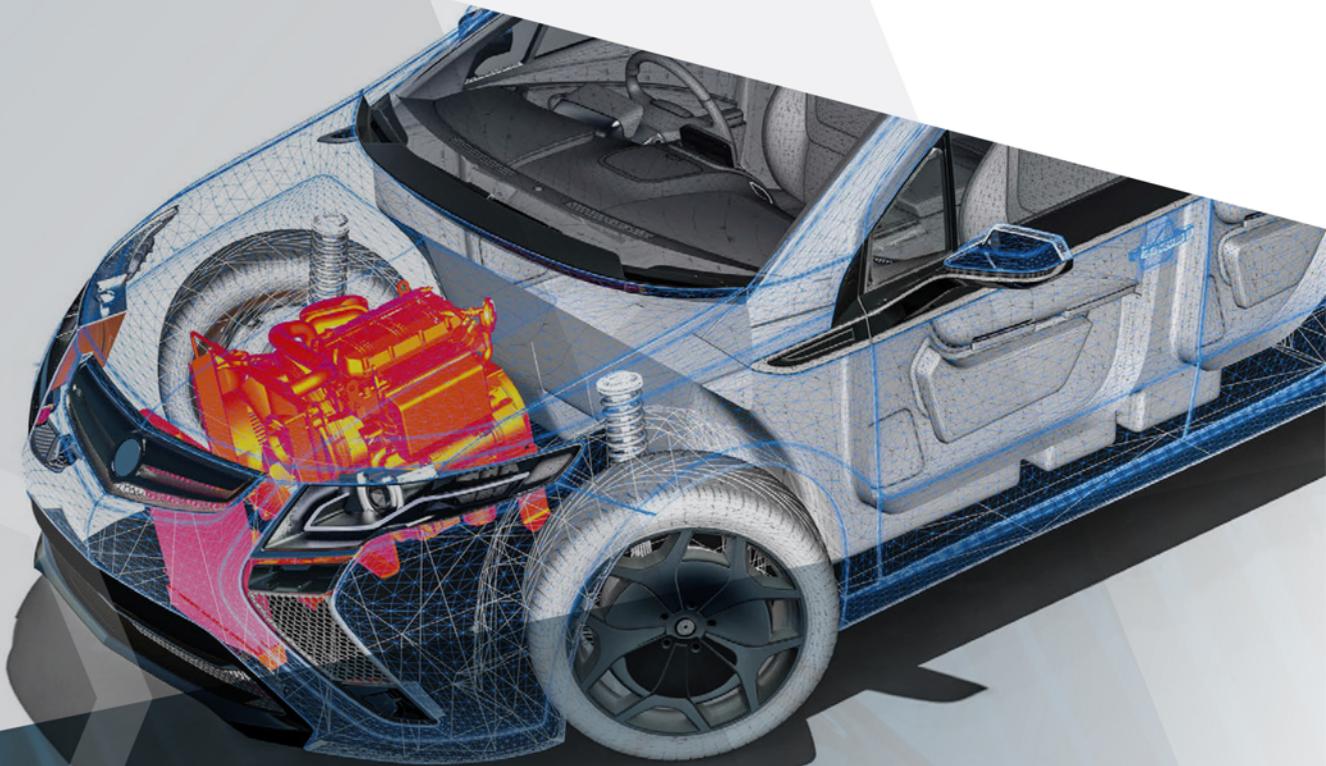
By the end of June, we saw our revenue increase by 50%. This organic growth was 15% higher because we bought some of our competitors, especially in Europe. Because we are seeing increasingly higher technologies, two years ago we created a small team especially for customers who ask for very complex technologies you can't find in most factories. This team works with the customers, the factories, to find a technical solution.

That's the real advantage of ICAPE Group. We know how to manage that for various types of technologies because customers ask us first to help them to buy in Asia. It's very difficult to buy in Asia and especially in China. Why is



ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam 

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



ventecclaminates.com



Yann Duigou

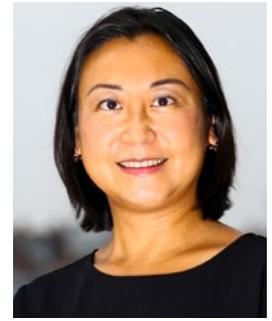
that? Because there are a limited number of people who speak English. We have more than 200 people in our China offices with Chinese, English and French speakers who work between Europe, the U.S., and Asia. They bring added value in terms

of logistical skills and the ability to answer technical questions from customers. Also, we bring the value of working between the suppliers, manufacturers, and customers.

COVID restrictions make it very difficult to go to China, but because we have our office there, we are more able to help our customers. For example, we have done audits, even for big corporations that already have their own department for buying directly in China. They asked us to make the audit for them because it was not possible for their quality inspectors to go there. This just adds more value to the market due to the pandemic and other global situations.

Johnson: This is more than just being a broker, this is advocating. It's not just connecting a job to a manufacturer but helping to solve problems technically and solve issues administratively; audits are an administrative function. It's really becoming a part of the process for your OEM customers, isn't it?

Bingling Li Sellam: Yes, exactly. We see ourselves as a service provider, not a broker. You are absolutely right: We are here to offer added value to the customer, to make them feel secure in their supply chain. It's not just about placing orders.



Bingling Li Sellam

Johnson: You must account for so much uncertainty in the supply chain and the economy, yet you are able to give peace of mind to your customers. Also, if they're trying to accomplish something in their design that they can't quite figure out, you have the expertise in manufacturing to do that. That's also peace of mind for the customer.

Sellam: Exactly. That's why I continue to enlarge our field application engineer (FAE) team worldwide, so that in different countries and different markets we have dedicated engineers working with customers not only in the design phase but also in manufacturing, helping them optimize their project from the very beginning.

Johnson: How early does the field application engineer team get involved with the customer?

Sellam: It depends on the customer's needs. When the customer comes to our engineers, most of them have already started the first version of the design phase. The first specification is out, and our customers have already consulted our engineers. We may even go to their side and hold a technical conference to discuss the project together. Our goal with this team is to help the customer have a better, more optimized design; afterward, of course, the customer comes to us most of the time for prototyping, implementation, and more. Sometimes it's even before the design phase. That means that the customer has a rough idea of



what they need, they explain their idea to us, and we propose a technical solution.

Duigou: We now have 30 business units around the world, and we feel it's important that the customer can speak in their own language. We know that we are the bridge between China and the rest of the world on a daily basis. Here, for example, we know that we will meet a lot of Germans. They will talk to our German team, and then we create the bridge to China. For them, it's very comfortable. There can be a lot of misunderstandings with China because of the language and culture. At ICAPE Group, we have a mix of cultures, and the same in our China office. We have many French employees living in China and who are married to Chinese people. This may seem like a small detail, but not for us because we understand each other, and we know how each one works. For business, it's quite important. For example, if I have an issue with a Chinese partner factory or our warehouse in Hong Kong, I can go directly to our teams in China. I can tell Bingling that we have an issue, and she can make a call in Chinese. It's the same for our customers. That's the bridge, the gateway.

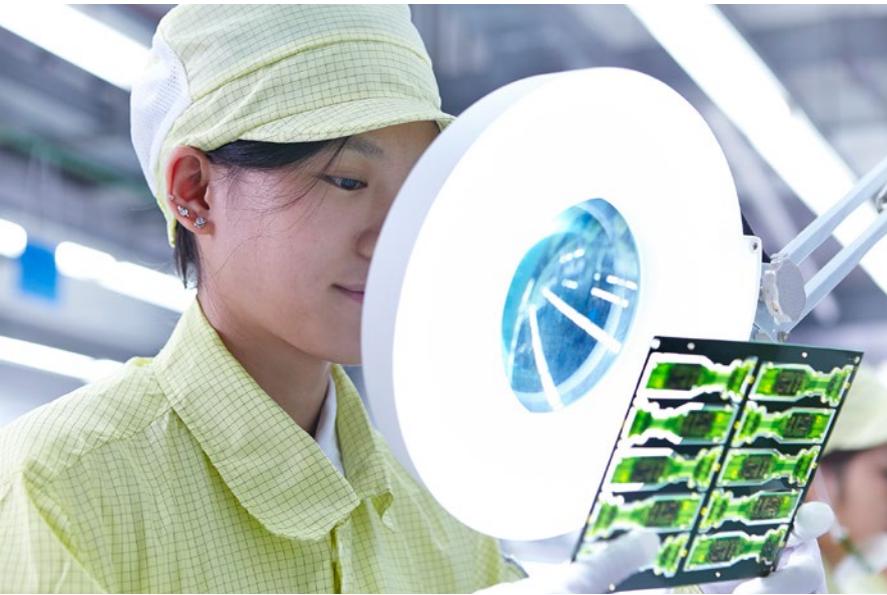
Johnson: That is a value add because that can often be an issue when you're outsourcing globally, regardless of whether it's China. Any cultural difference can create a communications barrier.

Duigou: Yes, and it's more complicated now. You can buy directly in China, but you must manage your own technical, quality, and logistic issues, and that's been a huge problem this year. Our customers don't have to deal with these issues; we do it for them.

Johnson: Those are the hidden costs in outsourcing—the admin, oversight, and follow-up to make that happen. The price may be different, and may be lower, but then you have the added costs of oversight and staffing to make sure that the outsourcing works to your quality specifications.

Duigou: This year we will make \$240 million in purchases. This means we have real buying power. For our small- or medium-sized customers, we can offer prices they would not get if they went to China on their own. It's a win-win for both parties.





they're shifting down into that sort of space, what are the specific sorts of technologies that your customers are looking for?

Duigou: They ask a lot for flex-rigid, more than 10 layers, or HDI. They are also looking for quality. There are always issues with delivery time. They are not satisfied with what they have come to find in the European market, so they try to find solutions outside of it. We are trying to find some solutions for them, but it's for very high technologies as many manufacturers in Europe are overloaded

because of the situation in Ukraine; for example, orders are increasing but they don't have enough production capacity.

Johnson: Which markets are driving the advanced technology demand the most? Where are the hot applications?

Duigou: Automotive, defense, aerospace, medical, and IoT.

Sellam: It can be also smartwatches that have very fine pieces.

Johnson: Right, most of the customers are trying to work in more advanced technologies for you. You're not seeing much of the standard consumer-grade sort of work.

Duigou: We already have our organic and acquisition growth target. For this year, it's good. I think we will hit the target, but we will see about next year.

Sellam: It's going to be a little bit more difficult, at least at the beginning of next year. Just this morning, my customers were talking about component prices, which seem like they will improve in the second half of next year. This is already good news. Several customers have

Johnson: You mentioned some growth for ICAPE Group outside of China as well. How much of that can we talk about?

Duigou: Yes, that's a good question. Our goal is to manufacture 20% of our projects outside of China in the next five or six years. That's our goal because more and more customers are asking us to buy outside of China. It's a very ambitious goal, but we are on the right track. For example, we bought a PCB factory in South Africa two years ago. We have another target in Europe, but it's too early to tell, and we also bought a small PCBA plant in the United States because a certain number of customers requested it. We are also looking at suppliers able to manufacture complex technologies outside of China like in Taiwan, Malaysia, Singapore, and so on.

Johnson: Let's talk about complex technologies. Obviously, these are a market driver. Customers are moving in that direction. What are your customers telling you about complex technologies? What are they asking for? Where do you see ICAPE Group going on the roadmap to provide what the customers are asking for regarding advanced technologies? Are they moving to a smaller space and trace, or maybe additive for certain applications? As

confirmed this to me; they all see a positive market in Europe from the second part of the year onward, even though we have the energy crisis in Europe and the war continues. I think the first part of next year might be a bit difficult, but I hope we'll catch up in the second part.

Johnson: Things are looking very positive for ICAPE Group.

Duigou: Our sales team is very positive. A few weeks ago, we were budgeting with our sales team for next year and they were so positive; we thought the figures were too high for us and told them to please check their figures. Despite what we heard from our customers about component prices this year, our sales team is very positive about next year. Overall, I think 2023 will be a good year for us. We have a lot of projects coming up with new acquisitions.

Johnson: Any closing thoughts?

Duigou: I'm very positive because just today, at our booth at electronica in Munich, we had two visits from markets where we didn't have customers before. They came to us and said, "We need your help." I believe there are no markets today where we cannot work and provide



solutions. For all our customers in every market, there is no limit. Five years ago, we may not have been able to help our customers with very complex technologies, but today, we have some special demands for high tech solutions. We have manufacturers who say there is no solution on the planet. Yet we tell them, "Let's get together and try to find a solution with you and with the client to meet in the middle." We always come up with solutions. We were not able to do that five years ago.

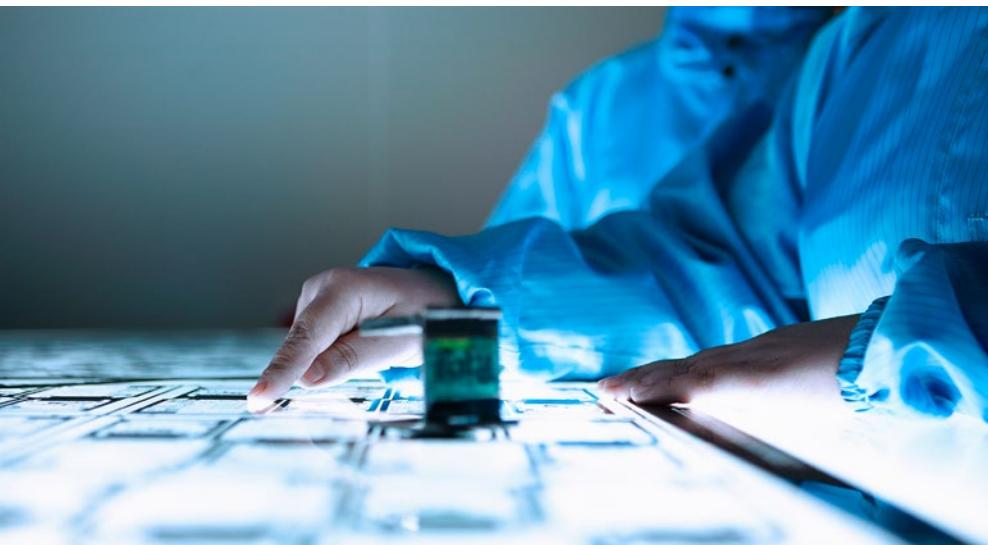
Johnson: You're beyond just delivering on customer requirements; you're innovating along with the customer to deliver what they need.

Duigou: Yes. But you know what? The danger with that is that a lot of clients end up thinking that there is no limit. They think we can find a solution for everything, which is not always true. We have limits (laughs).

Johnson: Excellent, I think that's a good summary. Thank you. **DESIGN007**

Sponsored links:

- [icape-group.com](https://www.icape-group.com)
- [linkedin.com/company/icapegroup](https://www.linkedin.com/company/icapegroup)



DFM 101

Final Finishes: Electrolytic Nickel/Gold

Article by Anaya Vardya

AMERICAN STANDARD CIRCUITS

Introduction

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. This article is the latest in a series that will discuss these cost drivers (from the PCB manufacturer's perspective) and the design decisions that will impact product reliability.

Final Finishes

Final finishes provide a surface for the component assembler to solder, wire bond, or conductively attach a component pad or lead to a pad, hole, or area of a PCB. The other use for a final finish is to provide a known contact resistance and life cycle for connectors, keys, or switches. The primary purpose of a final finish is to create electrical and thermal continuity with a surface of the PCB.

There are several final finishes in use in the industry today. These include:

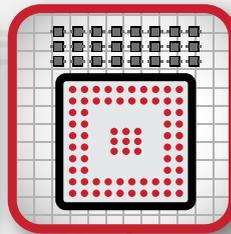
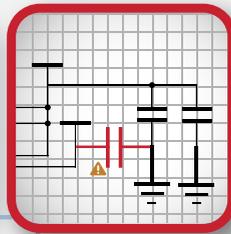
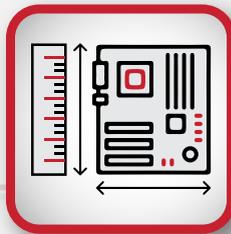
- ENIG (electroless nickel, immersion gold)
- ENIPIG (electroless nickel, immersion palladium, immersion gold)
- ENEPIG (electroless nickel, electroless palladium, immersion gold)
- ImmAg (immersion silver)
- ImmSn (immersion tin)
- Sulfamate nickel/hard or soft gold (electrolytic nickel/gold)
- HASL (hot air solder leveling)
 - > SnPb (63/37 tin/lead)
 - > LF (lead-free)
- OSP (organic solderability preservative)

Final finishes are primarily application driven, so there are several considerations that



Introducing **EMA Support +**

CircuitFit • Highlight Part • Find In Design • CIS Template-Based BOM • PCB Clustering



Get more value for your maintenance with exclusive benefits from the PCB Design experts at EMA

Improve your productivity, solve problems faster, and work more effectively with access to EMA experts, training, and technology to optimize your Cadence design experience. All available for FREE to active maintenance EMA customers.

Learn More >>

should be part of any decision to choose a final finish:

- Lead-tolerant or lead-free (LF) process
- Shelf life
- Flatness
- Lead or ball pitch
- Wire bondability
- Lead insertion
- Solder joint integrity
- Corrosion resistance
- Potential problems
- Cost



Lead-free Finishes

Lead-free finishes are considered “RoHS” compliant (< 0.1% BW of finish, for Pb, Hg, or Cd) with the single exception of tin/lead HASL. The RoHS compliant finishes include the following:

- ENIG
- ENIPIG
- ENEPIG
- ImmAg
- ImmSn
- Electrolytic nickel/gold
- LF HASL
- OSP

Lead-free PCBs require that the standard HASL surface finish cannot be used. There is still a significant amount of discussion on what the long-term surface finishes will be. Currently, the immersion silver and OSP surface finishes are the most prevalently specified surface finishes for solderable PCBs. Immersion tin is the prevalent surface finish for press-fit

backplanes. Please contact your PCB fabricator for current information on where industry specifications are heading.

Final Finishes

Electrolytic Nickel/Gold (Electrolytic nickel and hard or soft gold plating)

Electrolytic nickel/gold is a two-layer metallic plating of a varied thickness of either hard or soft electroplated gold over a varied thickness of “low stress” electroplated sulfamate nickel.*

Electrolytic nickel/gold has the best shelf life and environmental resistance, as well as a low contact resistance. Selective finishes are possible with electrolytic nickel/gold (< 15 µin soldering, > 30 µin contacts and wire-bondable soft gold).

Gold Wire Bondable

- Most expensive (thicker gold = \$\$\$)
- Selective finishes more expensive (filled vias required in some cases)
- Cannot be reworked
- Moderately lossy RF (nickel)

*Per IPC-4556, nominal pad size of 0.060" x 0.060" (1.5mm x 1.5mm).

Understanding the cost drivers in PCB fabrication and early engagement between the designer and the fabricator are crucial elements that lead to cost-effective design success. Following your fabricator’s DFM guidelines is the first place to start. **DESIGN007**



Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer’s Guide to... Fundamentals of RF/Microwave PCBs and Flex and Rigid-Flex Fundamentals*; and author of *Thermal Management: A Fabricator’s Perspective*. Visit I-007eBooks.com to download these and other educational titles. He also co-authored “Fundamentals of Printed Circuit Board Technologies” and provides a discussion of flex and rigid-flex PCBs at RealTime with... American Standard Circuits.

Show planning starts NOW!

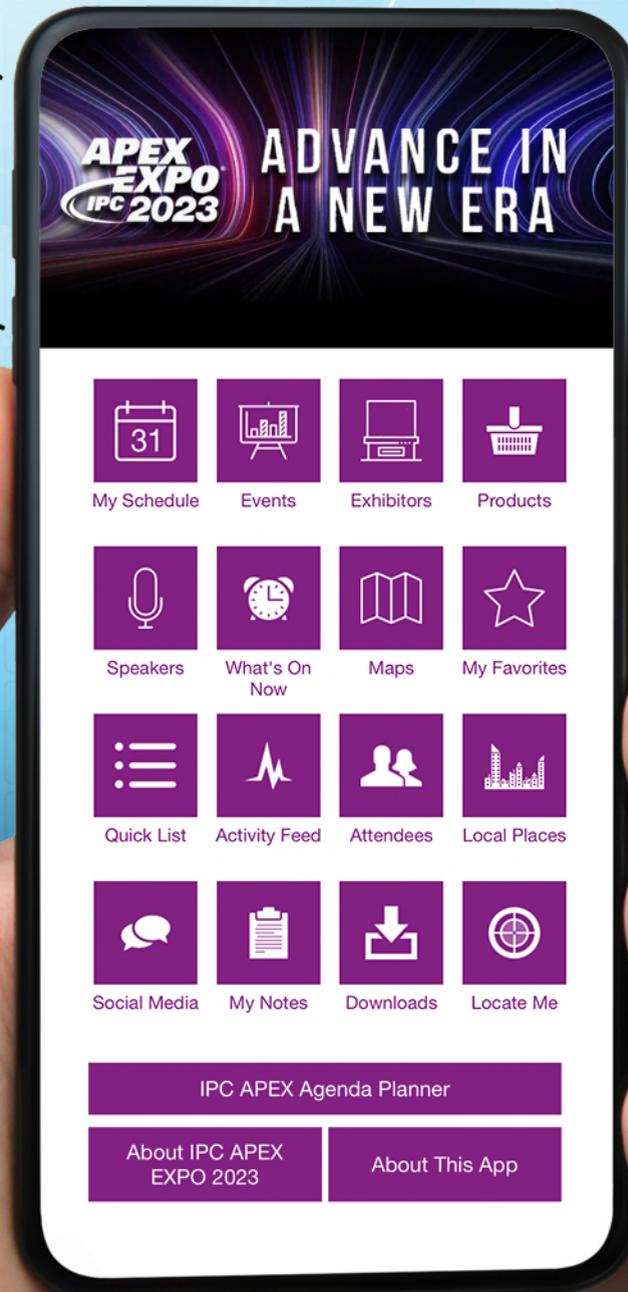
Get up-to-the-minute info on the conference, courses, and exhibits

Access and manage your schedule

Connect with and locate all exhibitors

Stay up to speed with the latest schedule updates

Receive important event alerts





Flex007 Highlights



Flexible Thinking: The Chameleon of Interconnection Technologies ▶

Flexible circuits are arguably the first instantiation of electronic interconnections. A flexible interconnection structure was first disclosed in patent literature by Albert Hansen—unearthed by gifted researcher, innovator, and self-described technology generalist Dr. Ken Gilileo. The roots of flexible circuits, as determined by the patent Gilileo uncovered, date back to a 1903 British patent.

Global Flexible PCB Market Report 2022: Increasing Demand from Automotive Industry Driving Growth ▶

According to official industrial sources, around 1.5 billion smartphones were sold in the year 2019, due to the rise in disposable income and increasing mobile data connectivity. Countries such as China, India, and the United States are expected to play a significant role in the market growth in the coming years.

The Printed Electronics Roundtable, Part 3 ▶

In this third and final installment of the roundtable, experts discuss some of the differences and similarities between PEC and traditional PCB processes, the future of printed electronic circuits, and why the best way to learn about this technology is through networking with veterans of this segment who are eager to share their expertise with the next generation.

Ibiden 2Q Sales Up 9% YoY ▶

Japanese PCB maker Ibiden Co. Ltd has posted a net sales of ¥213.58 billion (\$1.4 billion at

\$1:¥148.53) for the second quarter ended September 30, 2022, up by 9% from the same period in the previous fiscal year.

Nan Ya PCB Jan-Oct Revenue Up 24% YoY ▶

Nan Ya Printed Circuit Board Corp. (Nan Ya PCB), a Taiwan-based manufacturer of single-sided PCBs, HDI PCBs, and rigid-flex PCBs, has posted unaudited sales of NT\$6 billion (\$187.76 million at \$1=NT\$31.95) in October 2022, up by 1.66% from the previous month and by 17.7% compared to October of last year.

DownStream Flexes in Rigid-Flex ▶

DownStream Technologies co-founder Joe Clark and Senior Product Marketing Manager Mark Gallant discuss some of their latest tool updates, including a greater focus on bringing post-processing functionality, such as inter-layer analysis capability, to rigid-flex circuits.

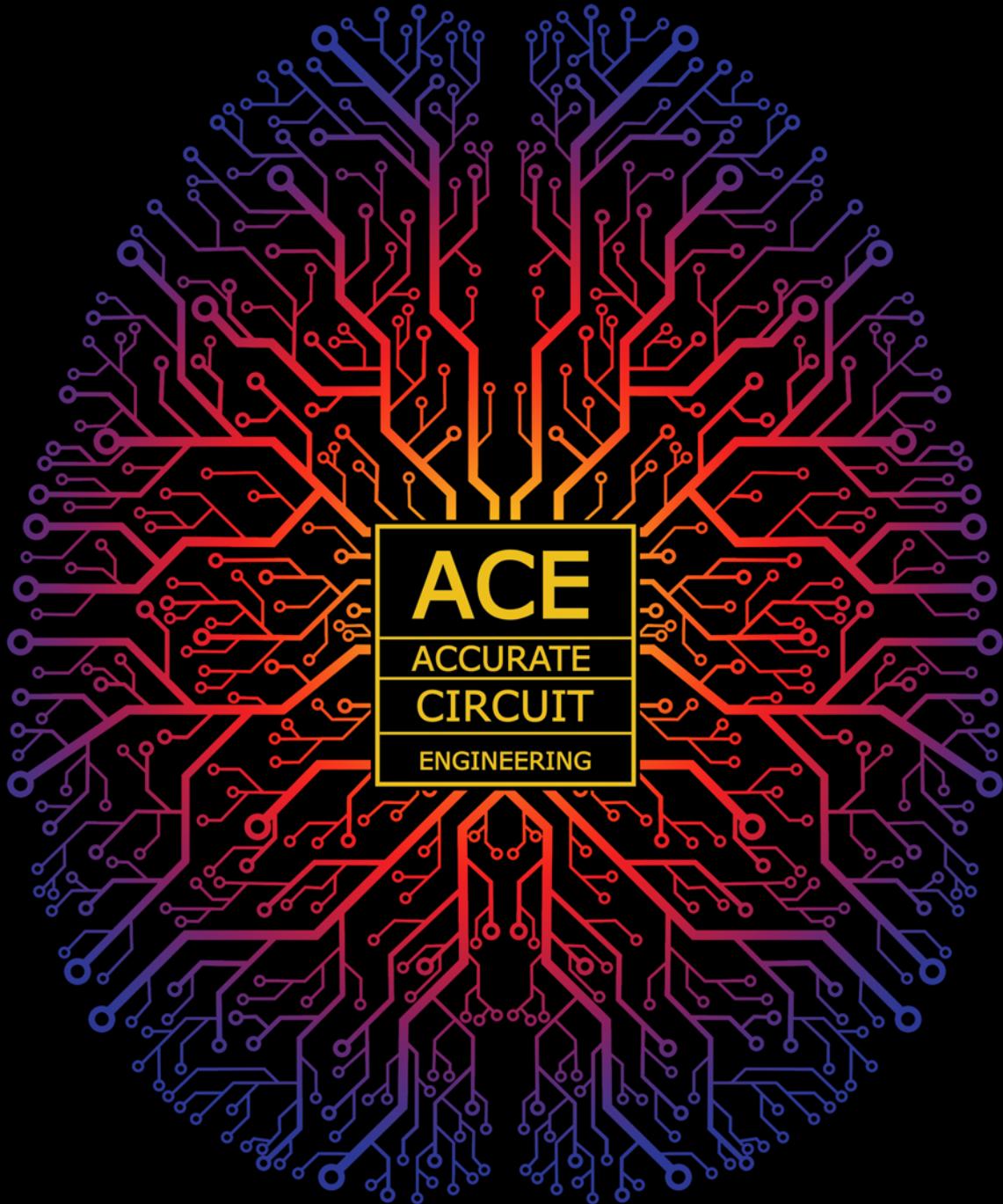
American Standard Circuits Earns Its IATF 16949 Automotive Re-Certification ▶

Anaya Vardya, president and CEO of American Standard Circuits, recently announced that his company has earned their IATF 16949 Automotive re-certification by passing their recent audit.

Skin-like Electronics Could Monitor Your Health Continuously ▶

Flexible, wearable electronics are making their way into everyday use, and their full potential is still to be realized. Soon this technology could be used for precision medical sensors attached to the skin, designed to perform health-monitoring and diagnosis.

Don't Overthink Your Prototype Builds



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

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS



Accurate Circuit Engineering

3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:

James Hofer
(714) 227-9540

www.ace-pcb.com

Flexible Circuits and In-line SMT Assembly Processing

Designers Notebook

by Vern Solberg, CONSULTANT

Incorporating surface mount components directly onto a flexible circuit's etched copper land patterns is not unlike the assembly process used for rigid circuit boards. To maximize robotic assembly efficiency and increase throughput of the flexible circuit, however, the circuit design engineer will need to provide a format that includes all features required for in-line assembly processing. There are three primary process stages for surface mount assembly: solder paste deposition, component placement, and reflow-solder processing. To maximize manufacturing efficiency, automated systems developed for performing these functions are each designed to accommodate the in-line conveyor transfer between one system to another (Figure 1).

Conveyor systems, although adjustable, rely on at least two parallel edges to support and transport the circuit from one machine to another. While rigid circuit boards are easily configured for conveyor processing, handling flexible material in a production environment will require a provision for the support of the thin, film-based circuit throughout each stage

of the assembly process. For the low- and medium-assembly volume applications, a conveyor-compliant pallet fixture can be prepared to retain the individual flexible circuits during each stage of the assembly; however, products requiring high-volume assembly processing require a more efficient solution.

To better accommodate conveyor handling, flexible circuits can be furnished from the circuit fabricator with a temporary, rigid carrier panel backing. Figure 2 exhibits a single flexible circuit assembly with a rather complex out-

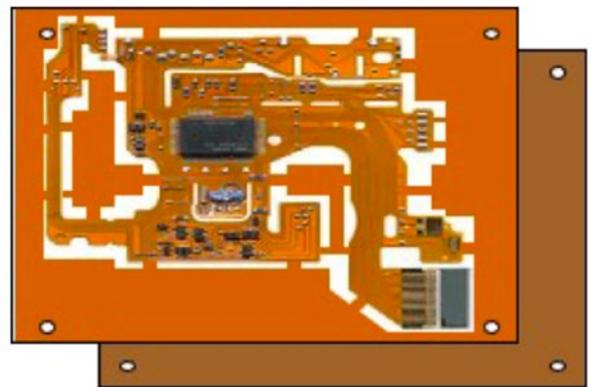


Figure 2: Flexible circuit retained in a uniform panel format.

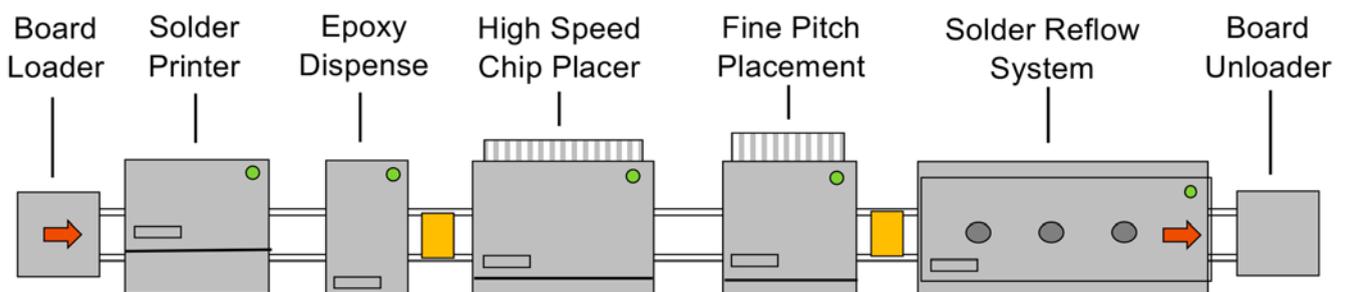
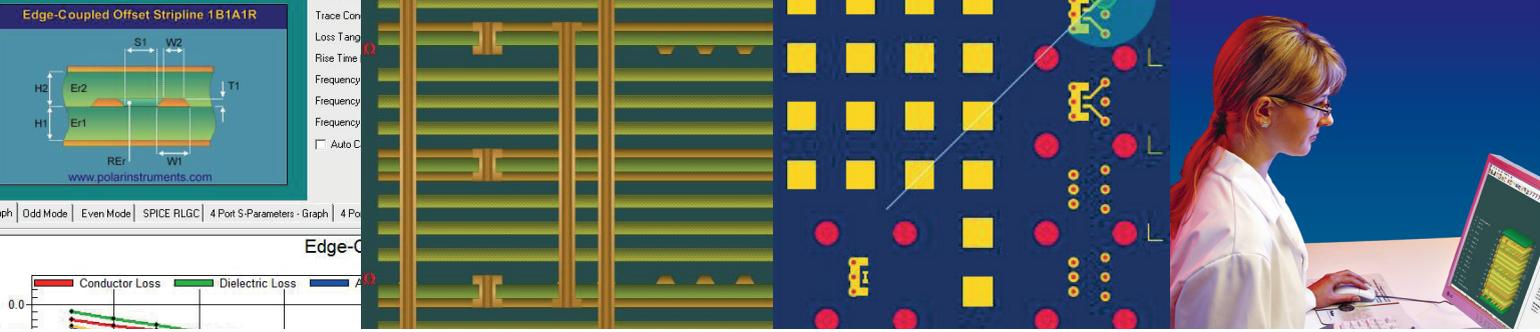


Figure 1: Basic conveyorized SMT assembly line configuration.

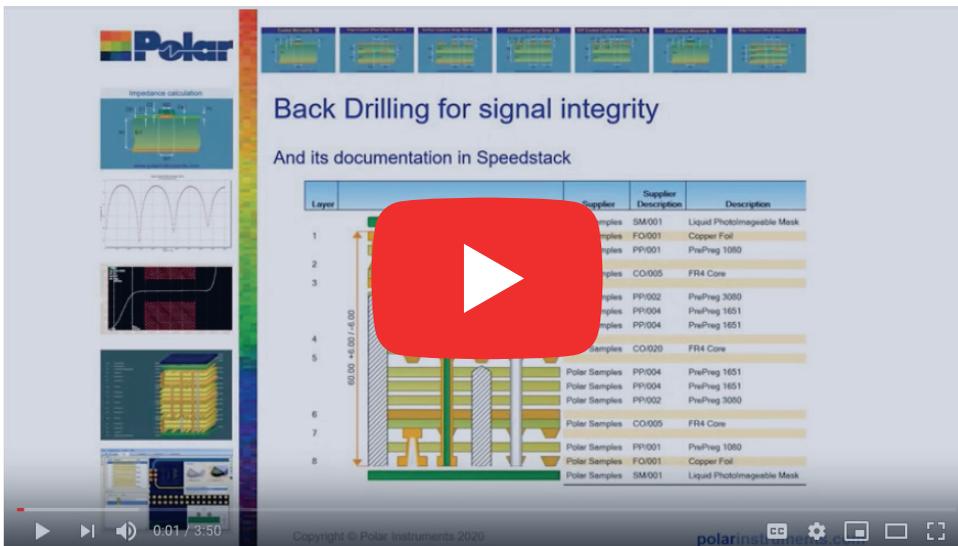


SUBSCRIBE

Subscribe to the Polar Instruments YouTube channel for helpful impedance and stackup videos

PCB Signal integrity tools for design & fabrication

- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics



line. Without the benefit of the rigid backing material and a dedicated, product-defined carrier pallet fixture, precise solder paste deposition and accurate component placement would not be possible.

Fabrication companies have developed a broad range of solutions for panel layout to both encourage efficient handling and support large and small form factor circuits through the assembly process:

- Single unit format (medium and large circuits)
- Multiple unit array format (row and column layout)
- Nested array format (maximizing area utilization)

When processing the smaller form factor flexible circuits, the multiple unit array format has proved both efficient and economical.

Palletizing the Flexible Circuit

The carrier panel developed for flexible circuit applications is designed to provide the uniform, rigid-board outline needed for conveyor transfer and to physically support the flexible circuit through each assembly process sequence. To keep the single- and multiple-unit flexible circuits in place, the designer will

need to provide small tab-like connecting features. To provide an area for the adhesion of the flexible material to the carrier panel, the flexible base material is simply extended outward to match the carrier panel outline. The adhesive film joining the flexible material to the rigid carrier panel must remain clear of the flexible circuit outline.

The palletized, multi-unit example shown in Figure 3 represents a grouping designed to maximize material utilization for small, irregularly shaped flexible circuit units. This example represents flexible circuit units that are arranged in the opposing orientation or “nested” array format.

Key Features Required for Automated Assembly

The supporting panel must include several key attributes:

1. Edge clearance of the carrier panel must allow for unobstructed access to the conveyor support belt.
2. Two or more tooling holes are required, located outside the flexible circuit units to secure the panel during the post assembly separation procedure.
3. An equal number of “global fiducial

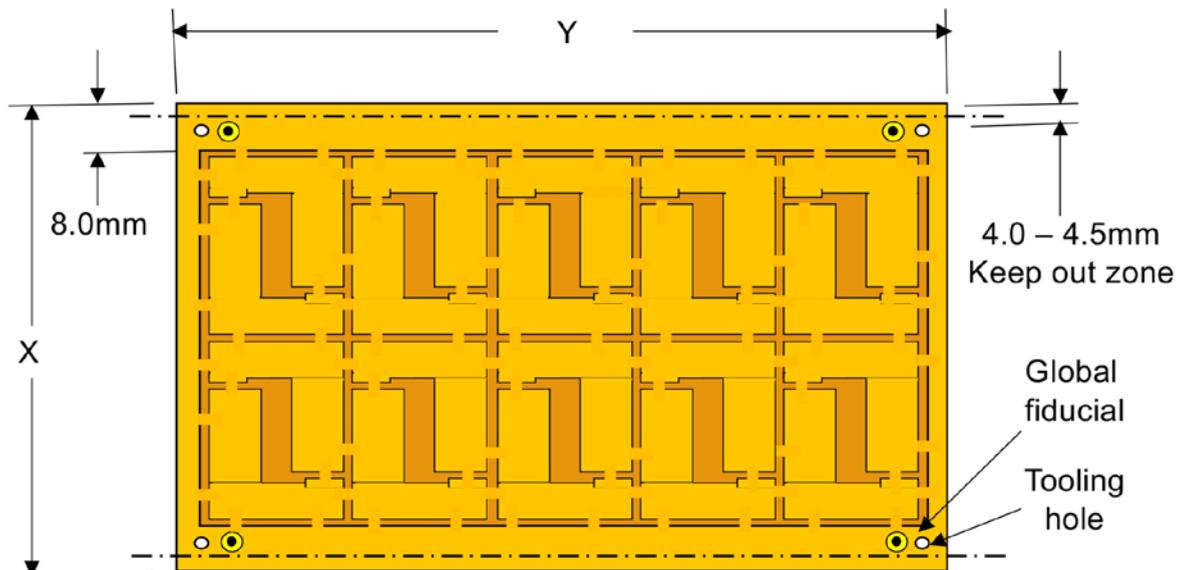


Figure 3. Opposing orientation array format.

targets” need to be located near the panel edge area to assist panel alignment during the solder deposition process. Solder deposition systems use cameras to pinpoint the fiducial targets, enabling precise alignment of the solder stencil to the land pattern features on the individual circuit units within the panel’s central area.

In addition to the globally located fiducial features, two or more fiducials will be required within the component mounting zone to facilitate precise automated component placement. Using multiple fiducial datum features within the SMT component placement area will minimize the effects of variable shrinkage or any process distortion in the flexible materials.

The illustration in Figure 4 is an example of a flexible circuit with SMT components mounted within two zones that are separated by a narrow interconnect section. Multiple fiducial locations will provide a tighter tolerance within each datum zone or termination area while relaxing the need to maintain a constricted tolerance of the flexible interface section between other component termination areas.

Fiducial Target Design Specification

The optimum fiducial target is simply a solid fetched copper circle that is clear of surface coating or cover layer material. Coatings and cover layer film material openings surrounding the fiducial must be adjusted to provide enough clearance around the fiducial’s perimeter to ensure that it does not overlap onto the fiducial target features during the cover layer-to-base circuit lamination process.

- The optimum fiducial is a solid circular land pattern that is 0.25 to 0.50 mm (~0.010" to 0.020") in diameter.
- To enable visual access for locating the fiducial targets, the solder mask or cover coat must provide a clearance 2 x R of fiducial R.
- Fiducial location must be clear of the panel or circuit outline edge by a minimum

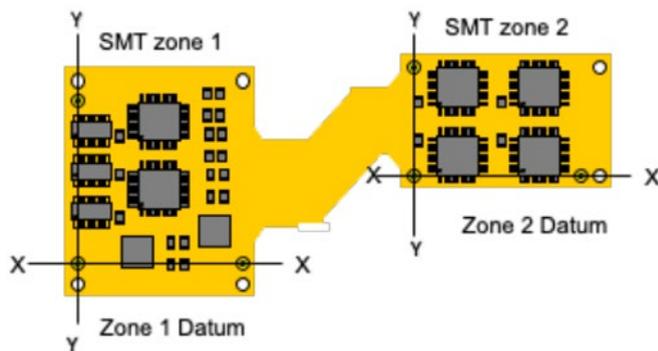


Figure 4. Fiducial target location for two component mounting zones.

of 4.75 mm (~0.187") and provide a consistent high contrast.

No plating is required on the etched copper fiducial surface, but if a secondary alloy plating over the base copper is specified in the control document, the designer must ensure that flatness of the fiducial surface is maintained within 0.015 mm (0.006").

Final Comments

When planning multiple unit circuits for panel processing, the flexible circuit designer should attempt to coordinate the final panel size and unit configuration between both those responsible for the assembly process and the fabricator designated for manufacturing the circuit.

Technical Course

Vern Solberg will be presenting a half-day technical course, “Flexible and Rigid-flex Circuit Design for Manufacturing,” on Sunday, Jan. 22, at IPC APEX EXPO 2023. The course will focus on SMT design, fabrication, and assembly process principles. **DESIGN007**



Vern Solberg is an independent technical consultant, specializing in SMT and microelectronics design and manufacturing technology. To read past columns, [click here](#).

A Primer on Flexible Circuits in Printed Electronics

Happy's Tech Talk #15

by Happy Holden, I-CONNECT007

Introduction

The printed electronics sector is presently an area of great interest to many in the electronics manufacturing industry. Because of their incredible utility, printed electronics (PE) are poised to generate tens of billions of dollars in the coming years. Recent research indicates that the total market for printed, flexible and organic electronics will grow from \$41.2 billion in 2020 to \$74 billion in 2030¹.

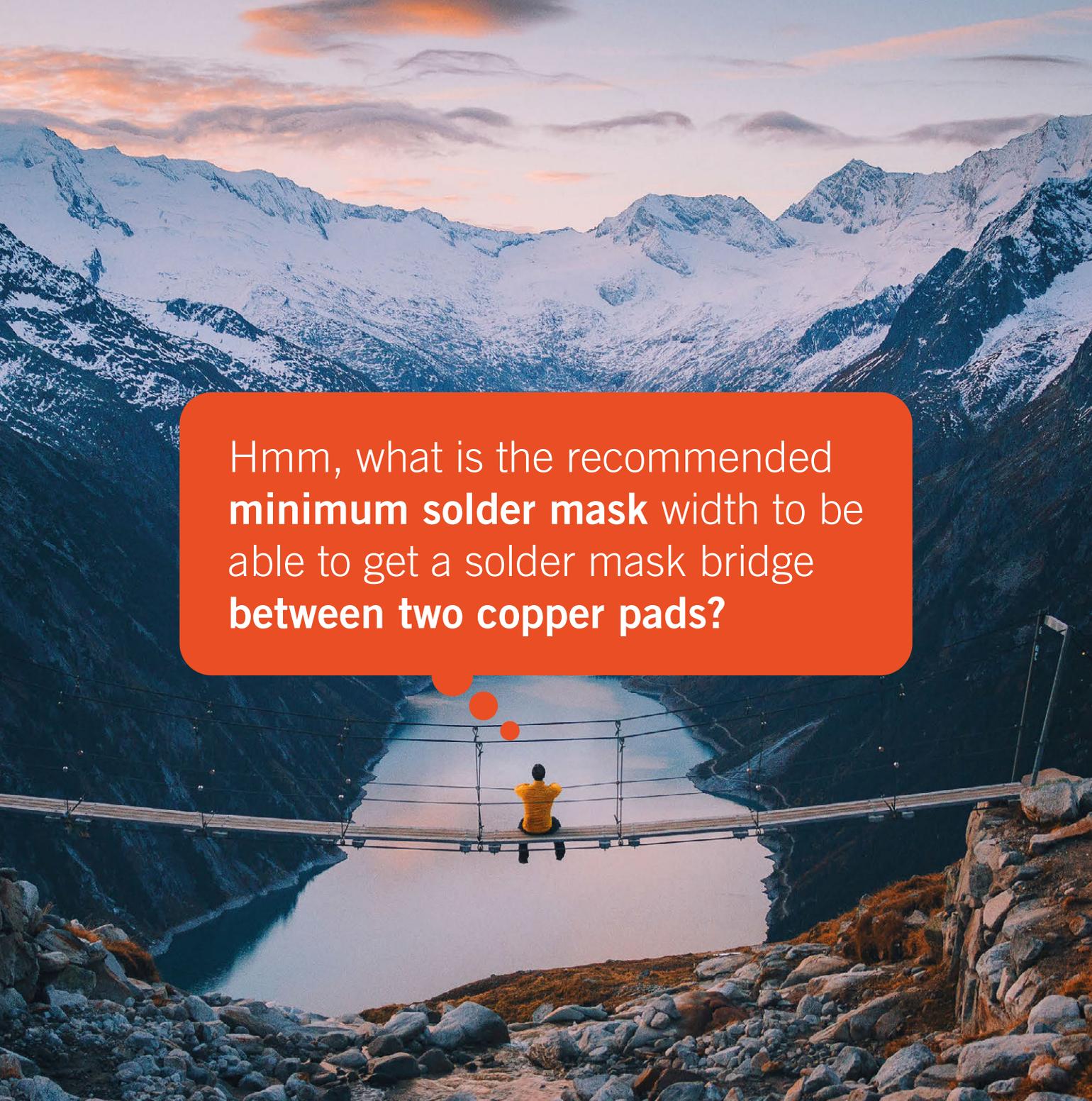
The term “printed electronics,” with its relatively recent appearance in the industry’s lexicon, may seem to refer to electronic circuits comprised of conductors and various active and passive components (i.e., transistors, resistors, capacitors) printed directly onto a substrate of some type. According to this definition, printed electronics are clearly a significant departure from traditional approaches to

electronics manufacturing. However, marketers are trying to position a host of manufacturing solutions under the “printed electronics” umbrella to take advantage of the growing buzz surrounding the technology, which has created confusion around the term.

This confusion may be a factor in the sector’s slow growth rate. While there is presently a market for these devices, the growth rate predicted by exuberant market forecasters has not yet been realized. Barriers to exponential growth seem to be related to the current inability of suppliers to reduce costs to enable large-scale manufacturing of printed electronics. Every disruptive technology that has sought to surpass existing, highly competitive, and/or traditional technologies has faced this challenge. So far, printed electronics have replaced some existing applications but have made little



Figure 1: Some of the major potential applications of printed electronics².

A person in a yellow shirt is sitting on a suspension bridge that spans across a deep valley. The bridge is made of wooden planks and is supported by cables. Below the bridge is a calm lake that reflects the surrounding mountains. The mountains are covered in snow and are very high. The sky is a mix of blue and orange, suggesting a sunset or sunrise. The overall scene is peaceful and scenic.

Hmm, what is the recommended **minimum solder mask** width to be able to get a solder mask bridge **between two copper pads**?

PCBs are complex products which demand a significant amount of time, knowledge and effort to become reliable. As it should be, because they are used in products that we all rely on in our daily life. And we expect them to work. But how do they become reliable? And what determines reliability? Is it the copper thickness, or the IPC Class that decides?

Reliable answers. **Reliable** PCBs.

Every day we get questions like those. And we love it. We have more than 500 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/pcb-design-mistakes/



progress in creating new applications outside of radio-frequency identification (RFID) and organic light-emitting diodes (OLEDs). The latter is a true printed electronics solution that has successfully established a solid market presence. Other than that, no real “killer” app or paradigm shift technology has emerged from the current fray. The simple reality is that it’s especially challenging to displace incumbent products with entirely new design solutions that must first be developed, understood, and implemented.

Applications

Disposable electronics that can compete with paper are a compelling and intriguing prospect to marketers. However, before fully adopting this technology, we must consider the environmental impact of throw-away electronics and adequately prepare for end-of-life matters. Assuming that these factors are taken into consideration, there are a variety of intriguing prospects and processes being suggested for the next generation of printed electronics.

Further, thanks to its ability to rapidly print multiple conductive, insulating, and semiconductor layers to form electronic circuits, printed electronics technology offers the ability to produce a simpler and larger integrated circuit (IC) at a much lower cost than using conventional fabrication methods, albeit with less functionality per unit area.

The number of potential applications for printed electronics technology is growing:

Printing Methods

Deciding which printing method to use is determined by both the requirements of the printed layers and the properties of the printed materials, as well as the product’s economic and technical considerations. Presently, printing technologies are divided between sheet-based and roll-to-roll-

based approaches. Sheet-based techniques, such as inkjet and screen printing, are best for low-volume, high-precision work. Gravure, offset, and flexographic printing are more common for high-volume production, such as for solar cells, reaching 10,000 square meters per hour (m^2/h). While offset and flexographic printing are mainly used for inorganic and organic conductors (the latter also for dielectrics), gravure printing is especially suitable for quality-sensitive layers, like organic semiconductors and semiconductor/dielectric-interfaces in transistors, due to its high layer image registration quality. Gravure printing is also suitable for inorganic and organic conductors in terms of high resolution. Organic field-effect transistors and integrated circuits can be wholly prepared using mass-printing methods.

Altogether, there are presently eight key manufacturing image transfer technologies that are either being employed or are proposed for printed electronics production (Figure 2).

Traditional printing technologies:

- Screen printing
- Rotogravure printing
- Flexographic printing
- Offset printing

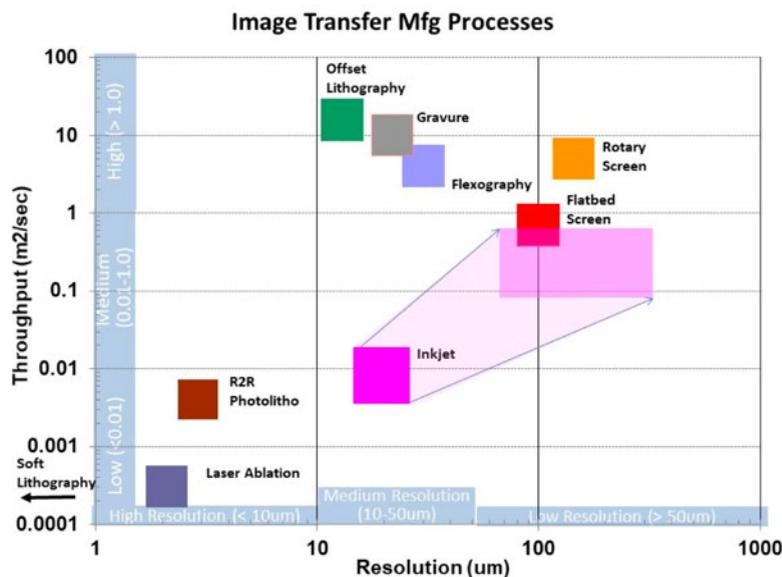


Figure 2: Printing techniques compared by resolution versus throughput. Arrow indicates soft lithography as a photoimageable technique that is off the scale to the left³.

New wave printing technologies:

- Inkjet printing
- R2R photolithographic
- Plasma printing
- Direct laser imaging or ablation

Of these enabling technologies, screen printing has been around the longest and is still widely used in production today. Membrane switch technology, part of the printed electronics family, heavily relies on screen printing to make products. Technical details are shown in Table 1 for speed, resolution, film thickness, and ink viscosity.

Table 1: Popular printing methods for printed electronics and their technical attributes²

Printing Method	Speed (m/min)	Resolution (um)	Film Thickness (um)	Viscosity (Pas)
Flexo	50 - 80	20 - 50	0.5 - 2	0.05 - 0.5
Gravure	20 - 100	20 - 50	0.5 - 2	0.05 - 0.2
Offset	15 - 100	15	0.5 - 2	30 - 100
Screen	10 - 100	80 - 100	5 - 25	0.5 - 50
Inkjet	.5 - 5	20	100 - 500	0.002-0.025
Imprint/stamp	50 - 100	1 - 15	0.5 - 200	0.05 - 0.5
Plasma	.5 - 3	20 - 100	0.5 - 200	Post Metallize
Laser ablation	.1 - 1	.1 - 5	0.5 - 2	0.05 - 0.2
R2R Photolitho	5 - 50	1 - 100	0.5 - 5	0.05 - 0.2

Base Materials and Inks

Printed electronics uses flexible substrates, which lowers production cost and allows fabrication of mechanically flexible circuits. While inkjet and screen printing typically imprint rigid substrates like glass and silicon, mass printing methods almost exclusively use flexible foil and paper. Additional substrate alternatives include polyethylene terephthalate (PET), a common choice due to its low cost and higher temperature stability; polyethylene naphthalate (PEN); polyether ether ketone (PEEK), a colorless organic polymer thermoplastic used in engi-

neering applications; and polyimide (PI) foil. Paper's low cost and manifold applications make it an attractive substrate, but its roughness and absorbency make it problematic for electronics. Low roughness and suitable wettability, which can be tuned pre-treatment (coating, corona), are important criteria for substrates (Table 2).

Organic and inorganic materials are both used for printed electronics. Ink materials must be available in liquid form for solution, dispersion, or suspension. They must function as conductors, semiconductors, dielectrics, or insulators. Material costs must be appropriate for the application.

Most of the attention given to materials has appropriately centered on conductive inks, especially silver. Silver is the most conductive commonly used metal for making circuit conductors. Membrane switch circuits, which operate at relatively high voltages and low currents, have been printed onto polyester base materials using silver inks for more than a few decades. The challenge has been getting these circuits to have the bulk conductivity associated with copper. Common inks have conductivities that hover around 10% of copper and are not generally suitable for higher performance applications that operate at lower voltages or may require more power. Several

Table 2: Base materials for printed electronics and their suitability²

BASE MATERIALS	Glass	Metal	Paper	PET	PEN	PC	PI	PEI	PVF
Smoothness	++	++	O	+	+	+	O	O	O
Temperature Resistance	++	++	O	+	+	O	++	++	++
Flexibility	--	++	++	++	++	++	++	++	++
Optical Transmittance	++	+	--	+	+	+	+	+	+
Barrier	++	+	-	+	+	+	+	+	+
Price	--	+	++	+	O	O	--	-	--
Score:	not suitable: -- - O + ++ :favorable								

Table 3: Suitability and characteristics of various PE inks²

Materials & Inks	Costs	Conductivity (S/cm ²)	Flexibility	Printability
Gold Ink	++++	+++	++	+++
Silver inks	+++	+++	++	+++
Copper inks	++	++	++	++
Tin inks	++	++	++	++
Graphite Inks	+	+++	+++	+++
Catalyst Inks	+	+++	+++	++
CNT Inks	++	+++	+++	++
Thin film Cu	+	++++	++++	-
Thin film Ni	++	++++	++++	-
Or-Semiconductors	+++	+	++	++
Indium Tin Oxide	+++	++	++++	+++
PEDOT:PSS	++	+	++++	++

PEDOT:PSS= Poly(3,4-ethylenedioxythiophene) poly(styrene-sulfonate)
 CNT=Carbon NanoTube

suppliers have attempted to address this problem by using a combination of new formulations of binders in the ink and nanoparticle silver; so far, results are promising. Table 3 shows materials and inks used in printed electronics.

Conclusions

Flexible materials are a key characteristic of printed electronics. Many products traditionally utilize glass to protect the active layers. To replace glass, the flexible substrates need to be an effective barrier against oxygen and water vapor, be sufficiently strong not to rip or tear and, if a cover, transparent to visible light. Many plastics, such as Mylar®, polyimide, PET, and ORMOCER® have these characteristics. Substrates can even be papers and paper hybrids.

Table 4 shows the suitability of various base materials to printed electronics applications. Important to the growing ranks of suitable substrate materials is “photonic soldering,” a new technology pioneered by NovaCentrix in their PulseForge® product line, which allows standard lead-free solder pastes to be soldered without heating the flexible base materials; this unique ability enables metallic

Table 4: Current suitability of base material substrates for printed electronics applications¹

MATERIALS & APPLICATIONS	Glass	Metal	Paper	PET	PEN	PC	PI	PEI	PVF
RFID (antenna only)			+	+	(+)	(+)			
RDIF (incl. circuits)				+	(+)	(+)			
Printed Memory				+	(+)				
oTFT/Display	+	+		+	+			+	
OLED/Display	+		(+)	+	+	+			
OLED/EL Lighting	+		(+)	(+)					
Printed Battery		+		(+)	(+)				
Organic Solar Cell	(+)	(+)		+	+				+
Sensors			+	+	(+)	(+)		+	
Clothing				+	(+)				
Signage	+	+	(+)	(+)	(+)	(+)	(+)	(+)	(+)
Score	Not Recommended: 'blank'		suitable +			Preferred (+)			

inks to be sintered/cured without damaging the substrates². DESIGN007

References

1. “Flexible, Printed and Organic Electronics 2020–2030: Forecasts, Technologies, Markets,” IDTechEx.com.
2. “Chapter 11: Printed Electronics,” by Joe Fjeldstad, *Flexible Circuit Technology, 4th Edition*.
3. “Photonic Soldering,” by Happy Holden, *PCB007 Magazine*, Dec. 20, 2021.

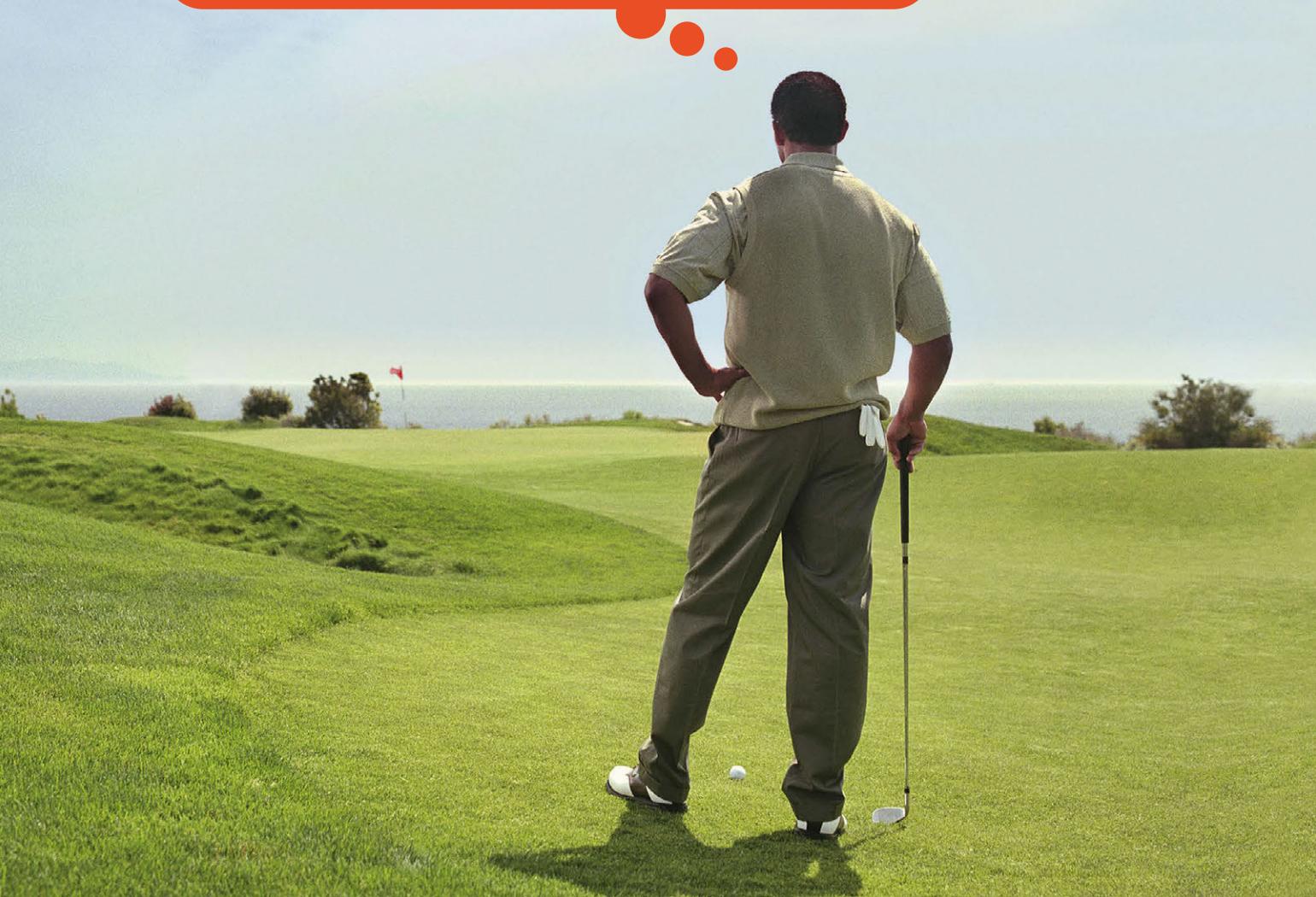
Resources

See “A Review of the Opportunities and Processes for Printed Electronics (Part 1),” by Happy Holden, a white paper originally published for SMTA International.



Happy Holden has worked in printed circuit technology since 1970 with Hewlett-Packard, NanYa Westwood, Merix, Foxconn, and Gen-tex. He is currently a contributing technical editor with I-Connect007, and the author of *Automation and Advanced Procedures in PCB Fabrication*, and *24 Essential Skills for Engineers*. To read past columns, [click here](#).

Hmm, what is recommended
**minimum distance for
copper to board edge?**



PCBs are complex products which demand a significant amount of time, knowledge and effort to become reliable. As it should be, because they are used in products that we all rely on in our daily life. And we expect them to work. But how do they become reliable? And what determines reliability? Is it the copper thickness, or the IPC Class that decides?

Reliable answers. **Reliable** PCBs.

Every day we get questions like those. And we love it. We have more than 500 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question, contact us!**

What's your PCB question?

www.ncabgroup.com/pcb-design-mistakes/



Flexible Circuits: A Road Less Traveled

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Referencing a famous poem may seem an odd way to start a technology column, but I find it fitting; my long engagement with flexible circuit technology has taken me places, both mentally and physically, that I would never have seen or experienced had I not developed an interest in what was once a marginal interconnection technology. Flexible circuits were a road less traveled when I first encountered

them but taking that path has made all the difference in my career.

For the last couple of decades, this column—as well as the musings of other flexible circuit advocates in the industry—have extolled the myriad advantages flexible circuits offer as an interconnection medium. As flexible circuit aficionados, we have collectively stressed the unique abilities of flexible circuits to make

three-dimensional interconnections, which free the designer to create a never-ending variety of electronic products. We have continued to point to the versatility flexible circuits offer as a means of interconnecting electronic elements that must move relative to each other, such as is required for inkjet print heads, the read-write heads of disc drives, the hinges of laptop computers, and many others. We have also described how effective flexible circuits can be as a medium for controlling the transmission of electronic signals from point to point. With the massive number of opportunities residing in these domains,



Choose wisely.

With today's material performance requirements, it's more important than ever to understand and choose your materials wisely.



Download Now

1.007e
Books

it might seem quite unnecessary to explore those that lay beyond; yet today, that is exactly where I intend to take you.

A Vast Highway of Opportunity

Heater Circuits

There are countless design opportunities that lie outside the generally perceived realm of flexible circuits, but far too often we simply don't see them. For example, consider the flexible heater circuit. Thin flexible heaters are used in numerous invisible applications. These heater circuits are often found in such items as fog and frost-free mirrors. They are a simple, yet very practical flexible circuit application that can be found anywhere—from the automobile to the bathroom—where relatively low temperatures suffice to accomplish the objectives. These low-power circuits can be made relatively inexpensively using materials such as polyester films and polymer thick film inks. However, higher temperature flexible circuit heaters are also possible. Such high-temperature heaters can be made using highly resistive metal films or foils such as nickel-chrome, Iconel, or stainless steel—in combination with high temperature materials, such as polyimide, that are capable of withstanding and putting out significant wattage and heat.

Smart Cards

Another area where flexible circuit technologies have played a significant role over the years is in the manufacture of smart cards. This is not a new application for flexible circuits, but it is clearly an emergent technology as we press on into the age of the internet, where nearly every individual item ever manufactured can have a unique, trackable identity. Smart cards have become the backbone of the internet of things (IoT). Such devices generally have a chip attached to provide their identity. Metal coils etched into the copper

foil of the flex circuit material are typical features on such devices, used to both send and/or receive wireless data. Because of the huge and growing volume of such circuits, roll-to-roll processing—a process that flexible circuits are naturally suited to—is generally employed to manufacture these increasingly ubiquitous circuits.

Wearable Electronics

Yet another area of gathering potential for flex circuits is wearable electronics, an industry that has exploded in recent years due to the growing interest in personal health monitoring. Typically, these devices include both electronic devices with thinned integrated circuit chips and passive devices that conform to the anatomical features of the wearer. Such devices can monitor many bodily functions, including heart rate, respiration, electrolytes, blood sugar, and many others; this data can then be transmitted to a smartphone, for example. These expanded capabilities have inspired developers to label these more integrated assemblies “flexible hybrid electronics” (FHE). They have been integrated into another area of investigation: stretchable circuits, which use elastic or elastomeric substrates; this technology opens doors to even more practical and fanciful electronic product innovations.

This brief column may have fallen short of covering the usefulness of flexible circuits in its entirety, but for those not yet familiar, it has hopefully provided a taste. The simple reality is that flexible circuits offer an unending range of opportunities to solve problems. This column has been an exercise in trying to inspire you to consider flexible circuits as an alternative “road” to your next electronic design destination and implementation.

Circling back to where I started this column, I'd like to share the poem that has inspired me many times over my life as a courtesy to those who may never had an opportunity to read and enjoy it.

The Road Not Taken

by Robert Frost

*Two roads diverged in a yellow wood,
And sorry I could not travel both
And be one traveler, long I stood
And looked down one as far as I could
To where it bent in the undergrowth;*

*Then took the other, as just as fair,
And having perhaps the better claim,
Because it was grassy and wanted wear;
Though as for that the passing there
Had worn them really about the same,*

*And both that morning equally lay
In leaves no step had trodden black.
Oh, I kept the first for another day!
Yet knowing how way leads on to way,
I doubted if I should ever come back.*

*I shall be telling this with a sigh
Somewhere ages and ages hence:
Two roads diverged in a wood, and I—
I took the one less traveled by,
And that has made all the difference.*

DESIGN007



Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than

185 patents issued or pending. To read past columns or contact Fjelstad, [click here](#). Download your copy of Fjelstad's book *Flexible Circuit Technology, 4th Edition*, and watch his in-depth workshop series "Flexible Circuit Technology."

Changing Color of Quantum Light on an IC

Optical photons are ideal carriers of quantum information. But to work together in a quantum computer or network, they need to have the same color and bandwidth. Changing a photon's frequency requires altering its energy, which is particularly challenging on integrated photonic chips.

Recently, researchers from the Harvard John A. Paulson School of Engineering and Applied Sciences (SEAS) developed an integrated electro-optic modulator that can efficiently change the frequency and bandwidth of single photons.

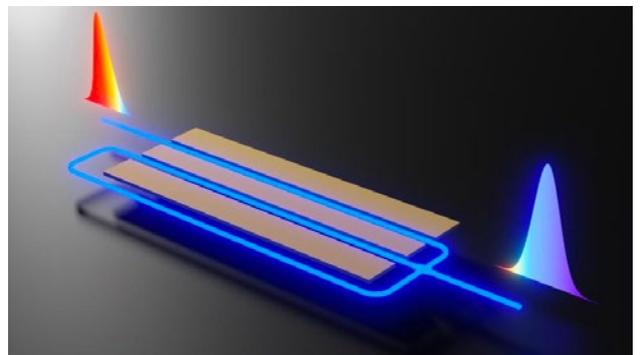
Converting a photon from one color to another is usually done by sending the photon into a crystal with a strong laser shining through it, a process that tends to be inefficient and noisy. Phase modulation, in which photon wave's oscillation is accelerated or slowed down to change the photon's frequency, offers a more efficient method, but the device required for such a process, an electro-optic phase modulator, has proven difficult to integrate on a chip.

One material may be uniquely suited for such an application: thin-film lithium niobate.

"In our work, we adopted a new modulator design on thin-film lithium niobate that significantly improved the device performance," said Marko Lončar, the Tiantai Lin Professor of Electrical Engineering at SEAS and senior author of the study. "With this integrated modulator, we achieved record-high terahertz frequency shifts of single photons."

Next, the team aims to use the device to control the frequency and bandwidth of quantum emitters for applications in quantum networks.

(Source: Harvard John A. Paulson School of Engineering and Applied Sciences)



PCB Carolina Breaks Attendance Record

PCB Carolina has been growing consistently for the past decade, and this year the show reached a milestone: more than 1,000 attendees. The show was the busiest I've ever seen it; I almost had to park off the NC State University campus. Fortunately, I'm an expert at "the parking lot game." I waited until a guy was leaving and then I sniped his spot. Bam!



Designers Notebook: Ultra High-Density Circuit Board Design

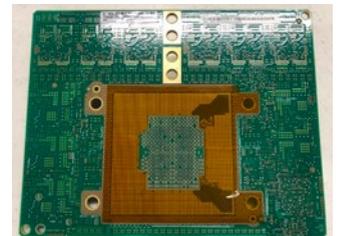


To facilitate new generations of high I/O semiconductor packaging, circuit board technology is

undergoing significant refinement in both fabrication process methods and base materials selected. Many of the new high-function semiconductor package families require significantly more terminals than their predecessors. Interconnecting these very fine-pitch, high I/O semiconductors can dramatically affect the procedures used in both circuit board design and assembly processing.

Quiet Power: Noise Mitigation in Power Planes

Inductive kick has been a well-known phenomenon in the electronic industry from very early on. First associated with motors, AC-mains transformers and mechanical relays, people noticed large voltage spikes when the current-carrying circuit was opened. Later, as more sophisticated electronic circuits emerged, the same thing was noticed any time current was changing through an inductor, or for that matter, through any inductance.

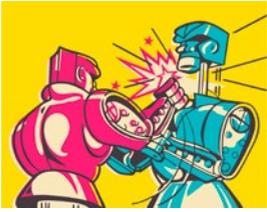


Elementary, Mr. Watson: Is Your Bathroom in the Kitchen?

Several years ago, a report came out of St. Louis of a strange apartment on the market. It was in the community of Central West End. With a small floor plan of only 200 square feet, the entire bathroom was placed right in the middle of the kitchen. It gives new meaning to the studio apartment. In real estate, it's pretty uncommon to find the bathroom in the kitchen; but metaphorically speaking, it's done all the time in a PCB design (ouch).



The Shaughnessy Report: It's All About the Physics—or Is It?



Lately, we've heard quite a few design experts say, "PCB design is all about the physics. Designers should focus more on understanding the laws of physics and less on circuit theory." While putting this issue together, we investigated potential cover ideas. "What if we had James Maxwell and Gordon Moore boxing on the cover, in a Faraday cage match? Let's get ready to rumble!"

Fresh PCB Concepts: Copper Coin for Dissipating Heat

Are you trying to fit more high-power parts on your board, but can't find the proper heat management technique? With more high-power components there is a need for increased thermal management. When you've tried everything else and are still having trouble keeping your parts within proper operating temperature, copper coin is one of the best ways to dissipate heat throughout your board.

All Systems Go: Auto-Detecting Over- and Under-Derated Parts

There are many reasons why a design may fail in the field. How painful would it be if it turned out that a catastrophic failure was caused by a one-cent part, such as an innocuous resistor that was stressed beyond its specified operational parameters?

The Physics of PCB Design

In this wide-ranging interview, Dr. Eric Bogatin discusses the relationship between physics and electrical theory, and why it's critical for designers and design engineers to understand the laws of physics. As he points out, the math is important, but designers shouldn't let the principles of physics "hide behind the math." Eric discusses some points of physics that designers need to understand, the physics resources available, and why it's so important to have some understanding of Maxwell's equations, even if you don't have a strong math background.



Tim's Takeaways: What More Do We Need to Know?

Although it's been more years than I care to admit, I still remember very clearly the class on careers that I was required to take in junior high school. On a table in the front of the class were several boxes filled with all sorts of different job cards that a student would search through to learn about different professions.



My Experience With Maxwell

I was first introduced to James Maxwell in 1967 as a college student. I had to decide whether I would take the Maxwell fields course or the switching and coding course. Being a chemical engineering major with a co-major in control theory, I had heard about the trials and tribulations of the infamous Maxwell fields course.



For the latest news and information, visit PCBDesign007.com

Career Opportunities



Find industry-experienced candidates at jobConnect007.

For just \$750, your 200-word, full-column ad will appear in the Career Opportunities section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, suppliers and the academic community.

In addition, your ad will:

- be featured in at least one of our newsletters
- appear on our jobConnect007.com board, which is promoted in every newsletter
- appear in our monthly [Careers Guide](#), emailed to 26,000 potential candidates

Potential candidates can click on your ad and submit a resume directly to the email address you provide, or be directed to the URL of your choice.

No contract required. Just send over your copy and company logo and we'll do the rest!

Contact barb@iconnect007.com to get your ad posted today!

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY



Career Opportunities



Test Engineer, Electronics Engineer, and Senior PCB Designer

Keytronic is a dynamic, team-based contract manufacturer with facilities worldwide. Innovation defines us. Come join us in Spokane, Washington! We invite you to bring your engineering expertise and passion for excellence. In turn, we provide meaningful opportunities for you to implement these attributes to their fullest while working together to bring our customer's high-tech automotive, aerospace, medical and commercial products to full production.

We encourage you to apply to one of our open positions below if you enjoy being challenged, working in a dynamic work setting and being a part of a team creating products to improve our world.

- **Test Engineer**—You will assist in conducting electrical test engineering support involving automation, assembly, maintenance, and data collection.
- **Electronics Engineer**—You will work on a team creating electronic circuitry, writing firmware for microprocessors and interfacing with customer development teams producing a wide array of products.
- **Senior PCB Designer**—You will perform PCB layout and documentation of complex printed circuit assembly products as part of a project team including procurement, electrical & mechanical engineering, PCB fabrication, Assembly and Test engineering stakeholders.

To learn more and apply for any of these openings please visit keytronic.com/join-us or email your resume to: llitsheim@keytronic.com.

apply now



Technical Sales Representative

We are an established distributor that represents manufacturing equipment and specialty consumables for the PCB manufacturing industry as well as other markets. All4-PCB represents products from suppliers in both Asia and Europe.

The objective of the position is to maintain and further develop the manufacturing consumable product business in the PCB industry. Excellent and well-organized communication flow between our principles and the customer base is required. We are looking for a dynamic, results-orientated sales personality with a technical background, capable of understanding the technical applications of the products.

A generous commission structure is available on top of solid base salary.

Responsibilities

- Grow existing accounts by maintaining relationships with clients
- Manage operation of accounts through responding to customers, forecasting, inventory management
- Generate new leads and tackle existing leads to contribute to business growth
- Attend trade shows and relevant conferences
- Supporting sales network in North America. Travel is required.

Qualifications

- A technical background in chemistry or engineering is beneficial. Min. 2-year degree.
- Proficient in Microsoft Office
- Strong organizational, communication and analytical skills
- Strong understanding of full sales process
- Experience utilizing customer relationship management software
- US citizenship or green card is needed and a valid driver's license

Apply to: Torsten.Reckert@all4-pcb.us.

apply now

Career Opportunities



Formulating Chemist

Taiyo is the world leader in solder mask products and has extensive worldwide R&D resources to further offerings that include the latest in inkjet technology, specialty dielectric inks and via filling inks for use with microvia and build-up technologies, as well as thermal-cure and UV-cure solder masks.

PRIMARY FUNCTION

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

ESSENTIAL DUTIES

1. Prepare design of experiments (DOE) to aid in the development of new products related to the solar energy industry, printed electronics, ink jet technologies, specialty coatings and additives, nanotechnologies, and applications.
2. Compile feasibility studies for bringing new products and emerging technologies through manufacturing to the marketplace.
3. Provide product and manufacturing support.
4. Provide product quality control and support.
5. Participate in multifunctional teams.

REQUIRED EDUCATION/EXPERIENCE

1. Minimum 4-year college degree in engineering or chemistry.
2. Preferred: Minimum 4-years' experience in chemical technologies and polymer science.
3. Knowledge of ink jet technologies, advanced materials and emerging technologies, including nano technologies.

[apply now](#)



Application Engineer

Flexible Circuit Technologies (FCT) is a global supplier providing design, prototyping and production of flexible circuits, rigid flex circuits, flexible heaters and full assembly services.

Responsibilities

- Gain understanding for customer/specific project requirements
- Review customer files, analyze - application, design, stack up, materials, mechanical requirements; develop cost-effective design to meet requirements
- Quote and follow-up to secure business
- Work with CAD: finalize files, attain customer approval prior to build
- Track timeline/provide customers with updates
- Follow up on prototype, assist with design changes (if needed), and push forward to production
- Work as the lead technician/program manager or as part of FCT team working with an assigned application engineer
- Help customer understand FCT's assembly, testing, and box build services
- Understand manufacturing and build process for flexible and rigid-flex circuits

Qualifications

- Demonstrated experience: flex circuit/rigid-flex design including design rules, IPC; flex heater design +
- Ability to work in fast-paced environment, broad range of projects, maintain sense of urgency
- Ability to work as a team player
- Excellent written and verbal communication skills
- Willing to travel for sales support and customer support activities if needed

Competitive salary, bonus program, and benefits package. Preferred location Minneapolis, MN area.

[apply now](#)

Career Opportunities



Electrical Engineer

Located in State College, Pennsylvania, Chemcut, a world leader in wet processing equipment for the manufacture of printed circuit boards and chemical etching of various metals, is seeking an electrical engineer.

Objectives:

The electrical/controls engineer will not only work with other engineers, but interface with all departments (manufacturing, sales, service, process, and purchasing). The engineer will design customer systems, creating electrical and control packages, while focusing on customer requirements.

Responsibilities:

- Process customer orders (create schematics, BOMs, PLC programs, relay logic controls, etc.)
- Startup and debug customer equipment on production floor
- Interface with engineering colleagues and other departments, providing input & direction
- Provide electrical/control support to customer service
- May require occasional travel and overtime

Qualifications:

- Bachelor's degree in electrical engineering or an EMET degree
- Machine control design experience a plus
- Good communication skills working in a team environment
- Strong ability to work independently with minimal supervision
- PLC and HMI experience a plus (ex. Studio 5000 Logix Designer, Factory Talk)
- Experience with AutoCAD, Microsoft Word, and Excel

Chemcut benefits include: Medical, dental and vision Insurance, life and disability insurance, paid vacation and holidays, sick leave accrual, and 401K with company match.

To apply, please submit a cover letter and resume to hr@chemcut.net.

[apply now](#)



Technical Marketing Engineer

EMA Design Automation, a leader in product development solutions, is in search of a detail-oriented individual who can apply their knowledge of electrical design and CAD software to assist marketing in the creation of videos, training materials, blog posts, and more. This Technical Marketing Engineer role is ideal for analytical problem-solvers who enjoy educating and teaching others.

Requirements:

- Bachelor's degree in electrical engineering or related field with a basic understanding of engineering theories and terminology required
- Basic knowledge of schematic design, PCB design, and simulation with experience in OrCAD or Allegro preferred
- Candidates must possess excellent writing skills with an understanding of sentence structure and grammar
- Basic knowledge of video editing and experience using Camtasia or Adobe Premiere Pro is preferred but not required
- Must be able to collaborate well with others and have excellent written and verbal communication skills for this remote position

EMA Design Automation is a small, family-owned company that fosters a flexible, collaborative environment and promotes professional growth.

Send Resumes to: resumes@ema-eda.com

[apply now](#)

Career Opportunities



Technical Service & Applications Engineer Full-Time — Midwest (WI, IL, MI)

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement-based inspection technology for electronics manufacturing. Located in Duluth, GA, Koh Young America has been serving its partners since 2010 and is expanding the team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience preferred
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, and specifications

Benefits

- Health/Dental/Vision/Life Insurance with no employee premium (including dependent coverage)
- 401K retirement plan
- Generous PTO and paid holidays

apply now



Regional Manager Midwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the States of KS, MO, NE, and AR. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deployment and management of sales reps. Identifies objectives, strategies and action plans to improve short- and long-term sales and earnings for all product lines.

DETAILS OF FUNCTION:

- Develops and maintains strategic partner relationships
- Manages and develops sales reps:
 - Reviews progress of sales performance
 - Provides quarterly results assessments of sales reps' performance
 - Works with sales reps to identify and contact decision-makers
 - Setting growth targets for sales reps
 - Educates sales reps by conducting programs/ seminars in the needed areas of knowledge
- Collects customer feedback and market research (products and competitors)
- Coordinates with other company departments to provide superior customer service

QUALIFICATIONS:

- 5-7+ years of related experience in the manufacturing sector or equivalent combination of formal education and experience
- Excellent oral and written communication skills
- Business-to-business sales experience a plus
- Good working knowledge of Microsoft Office Suite and common smart phone apps
- Valid driver's license
- 75-80% regional travel required

To apply, please submit a COVER LETTER and RESUME to: Fernando Rueda, Americas Manager

fernando_rueda@kyzen.com

apply now

Career Opportunities



MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, Ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and X-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport, and major credit card for travel.

Must be able to travel extensively.

[apply now](#)



ventec
INTERNATIONAL GROUP
騰輝電子

European Product Manager

Taiyo Inks, Germany

We are looking for a European product manager to serve as the primary point of contact for product technical sales activities specifically for Taiyo Inks in Europe.

Duties include:

- Business development & sales growth in Europe
- Subject matter expert for Taiyo ink solutions
- Frequent travel to targeted strategic customers/OEMs in Europe
- Technical support to customers to solve application issues
- Liaising with operational and supply chain teams to support customer service

Skills and abilities required:

- Extensive sales, product management, product application experience
- European citizenship (or authorization to work in Europe/Germany)
- Fluency in English language (spoken & written)
- Good written & verbal communications skills
- Printed circuit board industry experience an advantage
- Ability to work well both independently and as part of a team
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

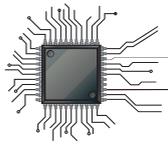
What's on offer:

- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits. Please forward your resume to jobs@ventec-europe.com.

[apply now](#)

Career Opportunities



MivaTek

Global

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now



Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- **Engineering**
- **Quality**
- **Various Manufacturing**

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

Arlon is a major manufacturer of specialty high-performance laminate and prepreg materials for use in a wide variety of printed circuit board applications. Arlon specializes in thermoset resin technology, including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems. These resin systems are available on a variety of substrates, including woven glass and non-woven aramid. Typical applications for these materials include advanced commercial and military electronics such as avionics, semiconductor testing, heat sink bonding, High Density Interconnect (HDI) and microvia PCBs (i.e. in mobile communication products).

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information please visit our website at www.arlonemd.com

apply now

Career Opportunities

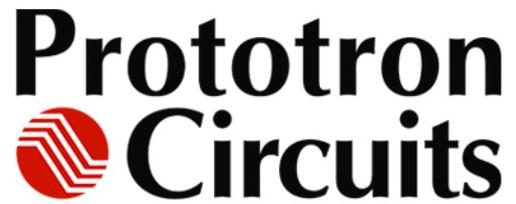


Are You Our Next Superstar?!

Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers build better boards faster. Our nationwide network provides many opportunities for a rewarding career within our company.

We are looking for talent with solid background in the PCB or PE industry and proven sales experience with a drive and attitude that match our company culture. This is a great opportunity to join an industry leader in the PCB and PE world and work with a terrific team driven to be vital in the design and manufacture of future circuits.

[apply now](#)



Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the Oregon, and Northern California territories. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:

- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer focused team

Interested? Please contact Russ Adams
at (206) 351-0281
or russa@prototron.com.

[apply now](#)

Career Opportunities



Rewarding Careers

Take advantage of the opportunities we are offering for careers with a growing test engineering firm. We currently have several openings at every stage of our operation.

The Test Connection, Inc. is a test engineering firm. We are family owned and operated with solid growth goals and strategies. We have an established workforce with seasoned professionals who are committed to meeting the demands of high-quality, low-cost and fast delivery.

TTCI is an Equal Opportunity Employer. We offer careers that include skills-based compensation. We are always looking for talented, experienced test engineers, test technicians, quote technicians, electronics interns, and front office staff to further our customer-oriented mission.

Associate Electronics Technician/Engineer (ATE-MD)

TTCI is adding electronics technician/engineer to our team for production test support.

- Candidates would operate the test systems and inspect circuit card assemblies (CCA) and will work under the direction of engineering staff, following established procedures to accomplish assigned tasks.
- Test, troubleshoot, repair, and modify developmental and production electronics.
- Working knowledge of theories of electronics, electrical circuitry, engineering mathematics, electronic and electrical testing desired.
- Advancement opportunities available.
- Must be a US citizen or resident.

[apply now](#)

Test Engineer (TE-MD)

In this role, you will specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly HP) and/or Teradyne (formerly GenRad) TestStation/228X test systems.

- Candidates must have at least three years of experience with in-circuit test equipment. A candidate would develop and debug our test systems and install in-circuit test sets remotely online or at customer's manufactur-

ing locations nationwide.

- Candidates would also help support production testing and implement Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks.
- Some travel required and these positions are available in the Hunt Valley, Md., office.

[apply now](#)

Sr. Test Engineer (STE-MD)

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

We proudly serve customers nationwide and around the world.

TTCI is an ITAR registered and JCP DD2345 certified company that is NIST 800-171 compliant.

[apply now](#)

Career Opportunities



eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now



BLACKFOX
Premier Training & Certification

IPC Instructor Longmont, CO; Phoenix, AZ; U.S.-based remote

*Independent contractor,
possible full-time employment*

Job Description

This position is responsible for delivering effective electronics manufacturing training, including IPC Certification, to students from the electronics manufacturing industry. IPC instructors primarily train and certify operators, inspectors, engineers, and other trainers to one of six IPC Certification Programs: IPC-A-600, IPC-A-610, IPC/WHMA-A-620, IPC J-STD-001, IPC 7711/7721, and IPC-6012.

IPC instructors will conduct training at one of our public training centers or will travel directly to the customer's facility. A candidate's close proximity to Longmont, CO, or Phoenix, AZ, is a plus. Several IPC Certification Courses can be taught remotely and require no travel.

Qualifications

Candidates must have a minimum of five years of electronics manufacturing experience. This experience can include printed circuit board fabrication, circuit board assembly, and/or wire and cable harness assembly. Soldering experience of through-hole and/or surface-mount components is highly preferred.

Candidate must have IPC training experience, either currently or in the past. A current and valid certified IPC trainer certificate holder is highly preferred.

Applicants must have the ability to work with little to no supervision and make appropriate and professional decisions.

Send resumes to Sharon Montana-Beard at
sharonm@blackfox.com.

apply now

Career Opportunities



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

The CAD/CAM engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creating manufacturing data, programs, and tools required for the manufacture of PCB.

Essential Duties and Responsibilities

- Import customer data into various CAM systems.
- Perform design rule checks and edit data to comply with manufacturing guidelines.
- Create array configurations, route, and test programs, panelization and output data for production use.
- Work with process engineers to evaluate and provide strategy for advanced processing as needed.
- Itemize and correspond to design issues with customers.
- Other duties as assigned.

Organizational Relationship

Reports to the engineering manager. Coordinates activities with all departments, especially manufacturing.

Qualifications

- A college degree or 5 years' experience is required. Good communication skills and the ability to work well with people is essential.
- Printed circuit board manufacturing knowledge.
- Experience using CAM tooling software, Orbotech GenFlex®.

Physical Demands

Ability to communicate verbally with management and coworkers is crucial. Regular use of the telephone and e-mail for communication is essential. Sitting for extended periods is common. Hearing and vision within normal ranges is helpful for normal conversations, to receive ordinary information and to prepare documents.

[apply now](#)



U.S. CIRCUIT

Plating Supervisor

Escondido, California-based PCB fabricator U.S. Circuit is now hiring for the position of plating supervisor. Candidate must have a minimum of five years' experience working in a wet process environment. Must have good communication skills, bilingual is a plus. Must have working knowledge of a plating lab and hands-on experience running an electrolytic plating line. Responsibilities include, but are not limited to, scheduling work, enforcing safety rules, scheduling/maintaining equipment and maintenance of records.

Competitive benefits package.
Pay will be commensurate
with experience.

Mail to:
mfariba@uscircuit.com

[apply now](#)

Career Opportunities



APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

[apply now](#)



YOUR
JOB
AD
HERE

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

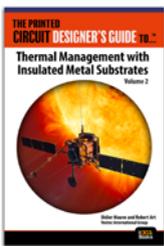
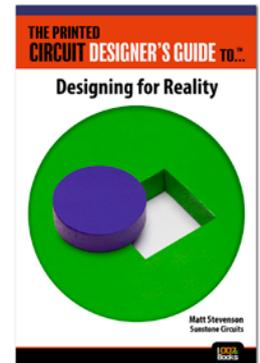
I-Connect007
GOOD FOR THE INDUSTRY

I-007eBooks The Printed Circuit Designer's Guide to...

NEW BOOK!

Designing for Reality by Matt Stevenson, Sunstone Circuits

Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process that often lead to variations in manufacturability, reliability, and cost of the board. Speaking of making better decisions, [read it now!](#)



Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples and focusing on specific solutions and enhanced properties of IMS. [Add this essential book to your library.](#)



High Performance Materials

by Michael Gay, Isola

This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. [Get your copy now!](#)



Stackups: The Design within the Design

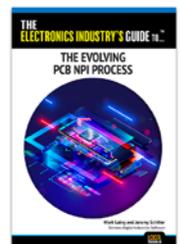
by Bill Hargin, Z-zero

Finally, a book about stackups! From material selection and understanding laminate data-sheets, to impedance planning, glass weave skew and rigid-flex materials, topic expert Bill Hargin has written a unique book on PCB stackups. [Get yours now!](#)

THE ELECTRONICS INDUSTRY'S GUIDE TO... *The Evolving PCB NPI Process*

by Mark Laing and Jeremy Schitter, Siemens Digital Industries Software

The authors of this book take a look at how market changes in the past 15 years, coupled with the current slowdown of production and delivery of materials and components, has affected the process for new product introduction (NPI) in the global marketplace. As a result, companies may need to adapt and take a new direction to navigate and thrive in an uncertain and rapidly evolving future. Learn how to streamline the NPI process and better manage the supply chain. [Get it Now!](#)



Our library is open 24/7/365. Visit us at: I-007eBooks.com

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

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

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

EDITOR | COLUMNIST COORDINATOR: **MICHELLE TE**
michelle@iconnect007.com

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

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 365-1727; barb@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@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 DESIGN: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © KRAPHIX**

DESIGN007

M A G A Z I N E

DESIGN007 MAGAZINE®

is published by IPC Publishing Group, Inc.
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

© 2022 IPC Publishing Group, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

December 2022, Volume 11, Number 12
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Accurate Circuit Engineering.....	65
Altium.....	7
American Standard Circuits.....	15
APCT.....	31
Burkle.....	27
Calumet.....	35
Candor Industries.....	39
Downstream Technologies.....	19, 33
EMA Design Automation.....	61
Flexible Circuit Technologies.....	43
I-Connect007 eBooks.....	2, 3, 77
In-Circuit Design Pty Ltd.....	23
IPC Community	13
IPC.....	45, 63
NCAB Group.....	71, 75
PCB Technologies.....	51
Polar Instruments.....	67
Precision Technologies.....	5
Prototron Circuits.....	25
Summit Interconnect.....	11
Taiyo America.....	41
US Circuit.....	47
Ventec International Group.....	55

Problems solved!



SUBSCRIBE NOW