

THE MAGAZINE I-Connect007

INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY



Plus: PCB
Design
in Europe

THE HUMAN SIDE OF AUTOMATION



PCB007
DESIGN007

SEPTEMBER 2026

Protection That Performs

THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...

UV Curable Conformal Coatings

Addressing Future Manufacturing Speed and Need



Brian J. Chislea
Cody Schoener, PhD
Dow, Inc.

I-007
BOOKS



Look Inside

"This book contributes useful clarity to an area of growing importance in high-reliability electronics manufacturing."

—Stanton Rak, PhD, SF Rak Company

I-007eBOOKS

PODCAST | SERIES 9



Engineering What's Next

with Jake Joo and
the expert team from Qnity

Explore the science behind the next generation of electronics. In this six-part series, Qnity experts examine the innovations advancing mission-critical reliability, flexible electronics, thermal management, and sustainable design.

Listen now to discover how better materials are enabling what's next.



Listen on
Apple Podcasts



Listen on
Spotify

LISTEN NOW!





From The Editor

- 8** **A World of Difference**
by Marcy LaRont

Shorts

- 91** ***Below the Surface***
What Engineers Find
Challenging About High
Frequency Signal Integrity
by Chandra Gupta

Pairing PCB Automation With Human Expertise

As AI and automation take on a greater role in PCB manufacturing, the knowledge of the people using them remains essential. This issue examines how employee experience strengthens process improvement, problem-solving, and decision-making. We also take you to Europe to explore its PCB design communities, events, awards, and standards work.

PCB007

Features

- The Right Approach*
- 12** **Old Philosophies, New Tools**
by Steve Williams
- The Chemical Connection*
- 26** **Employee and Operator Empowerment in the Age of AI**
by Don Ball
- 86** **The Factory of the Future Still Needs People**
by Michael Gleason
- 106** **Stop Inspecting AI and Start Understanding**
by Sean Patterson

Interviews

- 36** **Xscindo: No Data, No Landfill, No Waste**
with Aiden Neary
- 48** **SCHMID Pushes Toward the Submicron**
with Laurent Nicolet

Columns

- American Made Advocacy*
- 56** **Don't Build Our AI Economy on Foreign-sourced Components**
by Shane Whiteside
- Flex Connections*
- 64** **Material Selection in Nickel vs. Copper Tabs**
by Zack Schaner
- Trouble in Your Tank*
- 76** **HDI and UHDI: Building the Foundation of Advanced Packaging**
by Michael Carano
- Happy's Tech Talk #50*
- 98** **Start Your Digitalization Strategy With Automation**
by Happy Holden



WORLD-CLASS SUPPORT

Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, provides game-changing support to our customers.

CALL 949.587.3200 FOR MORE INFORMATION

Welcome to Our Newest Suppliers



ELITE MATERIAL CO., LTD

For All Your 370HR & 185HR Needs and more

EMC laminates are designed for very complex PCBs, utilizing advanced building blocks from ultra-thin prepregs to highly controlled cores. Examples of applications include "Any-layer" PCBs, mSAP, IC substrate, high layer count (HLC)/ high speed digital (HSD) and radio frequency (RF).

RESIN SYSTEMS FOR PCB LAMINATES

**EM-827(I)
EM-370(Z)
EM-528
EM-528K
EM-890K(A)**



ARLON

ELECTRONIC MATERIALS

P85HP - Tomorrow's Polyimide Today

Arlon specializes in Polyimide, Low Flow, and CTE-controlled PCB products.

Arlon is recognized for the highest quality standards in the industry and is positioned to support our customers most technological challenges through innovating technologies.

POLYIMIDES SYSTEMS FOR PCBS

**85N (High Temperature)
85HP (High Performance)
86HP (High Performance)**

PREPREGS FOR RIGID-FLEX & CAVITY PCBS

**37N (Low Flow Prepreg)
38N (2nd Gen Low-Flow
Polyimide Prepreg)
49N (Multifunctional Epoxy
Low-Flow Prepreg)
51N (Lead-Free Epoxy
Low-Flow Prepreg)**

- 28** Building Community,
Sharing Knowledge for
Designers in Italy
with Pietro Vergine



- 32** *Elementary, Mr. Watson*
The Rise and Fall of Europe's
PCB Community
by John Watson



In Every Issue

- 62 MilAero007 Highlights
- 112 Career Opportunities
- 120 Educational Resources
- 121 Advertiser Index

THE MAGAZINE
I-Connect007
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

DESIGN007

Features

- 20** Building Value for Europe Through Smart Standards and More
with Dominik Grau
- 72** PCB Design Takes Center Stage at FED's Design Awards
with Christoph Bornhorn and Erika Reel
- 82** Breaking Down the Barriers Between Design and Manufacturing in France
with Thomas Romont
- 94** Third Annual Pan-European Design Conference Important for Europe
with Susan Kayesar
- 104** An Open Invitation to Standards Development in Paris
by Marcy LaRont

Articles

- 16** Flex PCB Body Panels in CubeSats
by Anaya Vardya
- 68** When the World Becomes Your Design Team
by Stephen Chavez

Columns

- Powering the Future*
- 44** What Engineers Get Wrong About Designing for Reliability
by Brian Buyea
- Connect the Dots*
- 92** Getting the PCB Quote Right
by Matt Stevenson



Can't find your laminate? Need it quick?
WE GOT YOU COVERED!

QUICK TURN RF, MICROWAVE, ANTENNA EXPERTS



Accurate Circuit Engineering
3019 Kilson Drive
Santa Ana, CA 92707
(714) 546-2162 • sales@ace-pcb.com

For technical assistance call:
James Hofer
(714) 425-8950
www.ace-pcb.com

A World of Difference



BY MARCY LARONT, I-CONNECT007

I've spent a good deal of time in Asia over the years, both traveling and living there, so arriving in Thailand for THECA wasn't my first experience stepping into another culture or another way of doing business. Still, there is something about being there—moving through a different city, sitting down with people, listening to their conversations, and watching how they approach familiar challenges—that gets me thinking.

Then I come home. I return to my own routines, my own desk, and the people and places I know well. But every time I travel internationally, I return seeing things a little differently. It has a way of

reminding me that the way something is done in my corner of the world isn't necessarily the only way, or even always the best way. Sometimes you need to get outside your own space to see that. (Be sure to read Stephen Chavez's article in this issue about how working with a global team taught him some valuable communication skills.)

I've been thinking about this new perspective as we've put together this month's issue, because we're taking you outside some familiar spaces, too. In particular, we're looking to Europe, where PCB designers in Italy, France, Germany, and elsewhere are dealing with many of the same challenges

MicroCraft's Product Range Built for What's Next



Moving Probe Testers

Fast and reliable,
Emma testers are trusted worldwide
for precision and speed

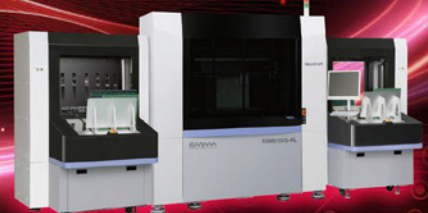


» **E8M6151G-AL**

8-Probe Loader & Unloader

» **E8S6151G**

8-Probe High Accuracy



CraftPix

On-Demand Inkjet Printers

Whether prototyping or
full production, the CraftPix lineup
offers inkjet systems to meet
your throughput needs



» **CPA8165ST**

Stacker Type Loader & Unloader

» **C8K7265T**

8-Head Inkjet Printer



Sarra

Automated TDR & VNA Testers

Built for today's high-speed designs,
Sarrra adds advanced impedance
and signal integrity testing
to your toolkit



» **S4V54**

4-Probe VNA Tester

» **H2P45**

2-Probe Variable Pitch TDR Tester



MicroCraft

designers face around the world, but they're also working within their own markets, cultures, organizations, and priorities. The articles in this issue on the PCB design communities, technical events, award ceremonies, and standards development in Europe provide real-time updates on how designers are approaching and overcoming these challenges.

Their experiences matter, particularly as our industry works toward standards and solutions intended to serve a global electronics community. Those ideas are also evident in my refreshing interview with Dominik Grau, the new vice president for Europe at the Global Electronics Association, whose impressive CV traces the evolution of standards in our fast-changing industry.

There's another idea running through this issue about the growing role of AI and automation in our work. What is the role of employees and empowerment? Our contributors aren't warning us about AI replacing people. Quite the opposite. They show how AI and automation are already being put to work on the shop floor, helping manufacturers analyze information, identify problems, improve processes, and do things faster and more consistently. Those are powerful capabilities.

But the technology becomes even more valuable when it's paired with the experience of the people using it. Someone who has spent years designing boards or working on a factory floor brings context to what the data is saying. We offer perspectives from Michael Gleason at GreenSource Fabrication, Sean Patterson, and columnists Steve Williams and Don Ball, demonstrating the importance of knowing which questions to ask, when something doesn't look right, and when a solution that works perfectly in one situation may not work in another.

In his column, Steve Williams reflects on some

of the great thinkers in quality and continuous improvement, suggesting that these leaders would have embraced AI. "None of these six would treat AI as a replacement for the discipline they spent their careers building. Each would use it to do what they already believed in, faster and at greater scale," he writes.

In other areas of interest, Anaya Vardya explores flexible circuits in nanosatellites, Michael Carano explains some of the challenges of producing reliable microvias in HDI builds, Shane Whiteside argues for American-made PCBs in American data centers, and Brian Buyea explores what designers get wrong about designing for reliability. Laurent Nicolet of SCHMID discusses the convergence between PCB fabrication and semiconductor processing. Finally, enjoy my interview with Aiden Neary, CEO of Xscindo, a company changing the way we approach circularity in electronics.

The lessons brought home from traveling can be applied to how we do our work each day. The more perspectives we bring to a problem, and the more willing we are to learn from them, the better our decisions become. The real opportunity comes from putting those different kinds of knowledge to work together. **I-CONNECT007**



Marcy LaRont is the managing editor of *I-Connect007*

Magazine and executive director of IPC Publishing Group. Marcy started her career in PCBs in 1993 and

brings a wide array of business

experience and perspective to

I-Connect007. To contact Marcy, [click here](#).

I-007e
Get smarter here.

SEAMLESS TRANSITION TO RELIABLE SUPPLY

Reliable Supply. Proven Performance. Global Support.

I-TERA MT40 is a reliable high-speed laminate designed for HDI and sequential lamination applications, I-Tera MT40 delivers stable electrical performance and robust processability.

TACHYON 100G is engineered for advanced high-speed digital designs, offering low loss and excellent stability through complex fabrication cycles.

TERRAGREEN 400G is an ultra-low-loss material optimized for the most demanding high-speed applications.

	Dk / Df @ 10 GHz	Dk / Df @ 20 GHz	Dk / Df @ 30 GHz
Industry Std Mid-Loss	3.40 / 0.0035	3.35 / 0.0035	3.33 / 0.0035
I-Tera MT40	3.30 / 0.0031	3.28 / 0.0031	3.27 / 0.0031
Industry Std Mid-Loss	3.18 / 0.0018	3.15 / 0.0018	3.12 / 0.0018
Tachyon 100G	3.10 / 0.0021	3.07 / 0.0021	3.04 / 0.0021
TerraGreen 400G	3.15 / 0.0017	3.10 / 0.0017	3.07 / 0.0017

Values shown are typical across 10-30 GHz and demonstrate comparable electrical performance.

Old Philosophies, New Tools

BY STEVE WILLIAMS, THE RIGHT APPROACH CONSULTING

About six years ago, I wrote a series on the Founding Fathers of Quality. With AI taking over the world, I wanted to posit how these quality gurus would apply this technology to their core quality management philosophies.

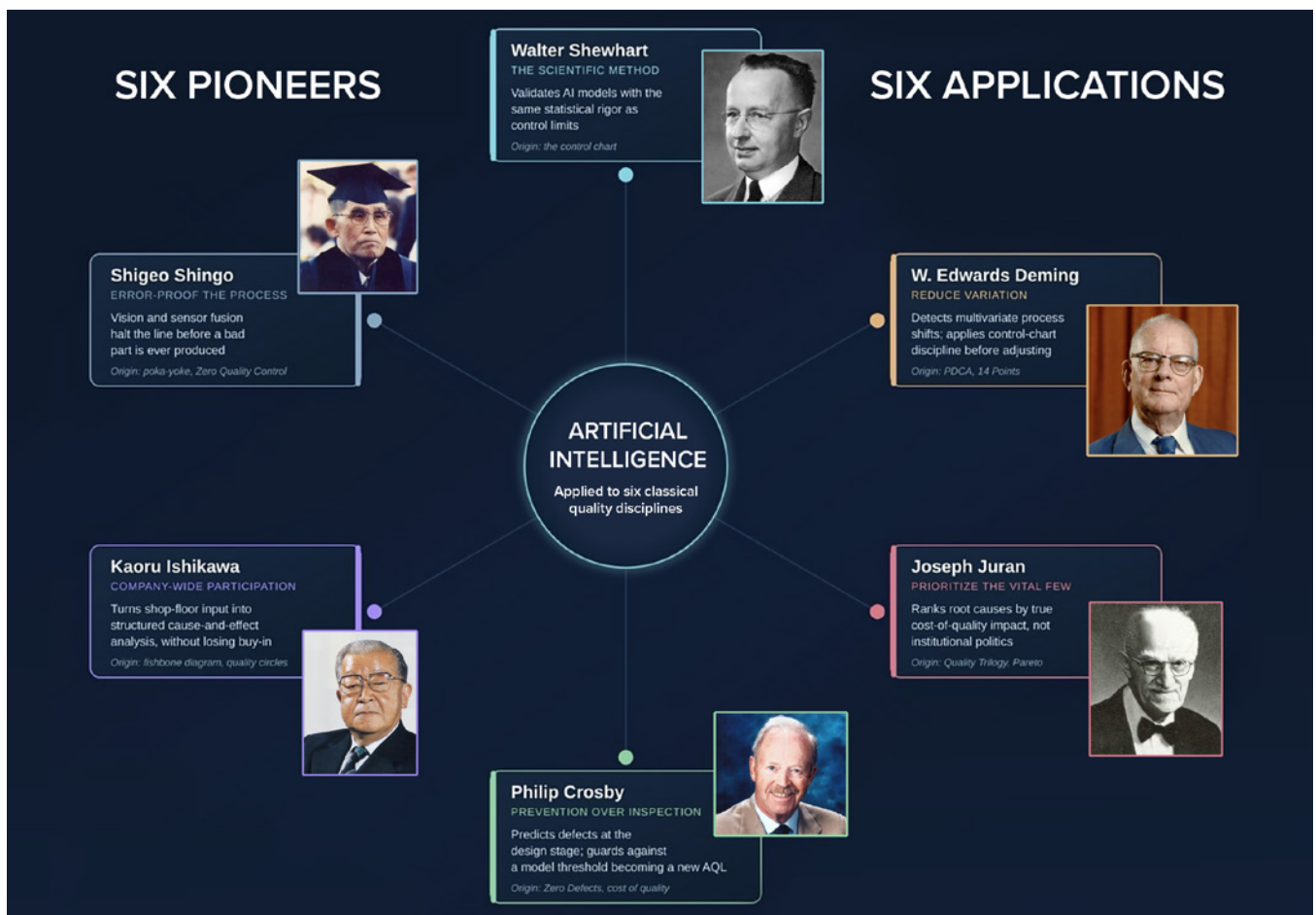
Quality management as a discipline was built by a handful of people who never touched a neural network, yet whose ideas map onto AI with almost eerie precision. Dr. Walter Shewhart, Dr. Edwards Deming, Dr. Joseph Juran, Philip Crosby, Dr. Kaoru Ishikawa, and Shigeo Shingo each built a philosophy around a specific lever for improving quality, including statistical rigor, reducing variation, priori-

tizing effort, eliminating defects, involving the workforce, and error-proofing the process itself.

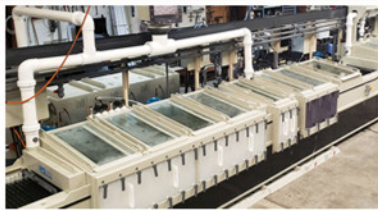
Rather than AI replacing any of those levers, AI sharpens each one, and imagining how each of these individuals would actually put it to work says as much about their original thinking as it does about the technology.

Shewhart: AI Bound by the Scientific Method

Shewhart's contribution predates all five of the figures below and quietly underlies each of their frameworks. He invented the control chart itself, and with it the



M A D E I N T H E U S A.



For more than 25 years, IPS has been your solution for the most state-of-the-art process equipment. All our systems are designed, tested and manufactured right here in America ...and we wouldn't have it any other way!



U.S. Coast to Coast • Parts and Service Fast
ips-vcm.com • sales@ips-vcm.com • 435-586-1188

statistical distinction between "chance cause" (common) variation and "assignable cause" (special) variation that Deming would later carry forward and popularize.

Shewhart's real achievement was insisting that any claim about a process being "in control" rests on a defensible statistical basis, not intuition or a manager's gut feel. He also originated the Plan-Do-Study-Act cycle that Deming would popularize as the PDCA cycle.

Shewhart would likely be the most rigorous skeptic of AI. A model that outputs an anomaly score is, in his terms, an assignable-cause detector, but unlike a control chart built on known statistical distributions, most machine learning models don't come with a transparent, auditable basis for why a given score crossed a threshold.

He would push hard for treating model validation with the same seriousness he brought to control limits: back-testing against historical data, quantifying false-positive and false-negative rates the way a control chart's three-sigma limits quantify the odds of a false alarm, and refusing to treat a model's output as authoritative until its statistical behavior is understood.

For Shewhart, the danger of AI wouldn't be the technology itself, but rather organizations trusting a black box the same way they once trusted an unproven chart without doing the scientific work to justify that trust. "Explainable AI," in his framing, is the modern equivalent of the statistical theory that made the control chart trustworthy in the first place.

Deming: AI as the Ultimate Variation Detector

Deming's entire framework rested on the insight that most quality problems come from variation in the system, not from the people working in it. His statistical process control charts existed to separate "common cause" variation, the noise inherent in any process, from "special cause" variation that signals something has actually changed. He was fiercely opposed to tampering, meaning adjusting a stable process in response to normal noise, which only makes performance worse.

Deming would almost certainly see machine learning as an extension of the control chart, not a replacement for it. A model trained on process data

can detect subtle multivariate shifts that a single-variable Shewhart chart would miss, with several parameters drifting together in a way no individual chart would flag.

But he would apply the same discipline to AI outputs that he applied to control limits: an anomaly score is not, by itself, permission to act. Any AI-flagged deviation still has to run through the same root-cause discipline before anyone touches the process, because a model that "detects" noise and triggers unnecessary adjustment is just tampering with better technology behind it. His 14 Points would likely gain a 15th in spirit: Understand your model as thoroughly as you understand your process, or you've automated the guesswork you were trying to eliminate.

Juran: AI as a Prioritization Engine

Juran's contribution was less about statistics and more about management focus, the Quality Trilogy of planning, control, and improvement, and his adaptation of the Pareto principle into the "vital few and useful many." For Juran, the central question was always where to spend limited attention, and which a handful of causes account for most of the defects, cost, and customer complaints.

AI is, in a very literal sense, a Pareto engine at scale. A model that ingests warranty claims, inspection records, supplier data, and field failures simultaneously can rank root causes by actual financial and quality impact far faster than a manual Pareto analysis, and continuously rather than as a quarterly exercise.

Juran would likely push organizations to use AI first for the unglamorous work of cost-of-quality analysis, accurately attributing the financial impact of failures so that improvement projects get funded on evidence rather than on institutional politics. He'd also warn that a prioritization engine is only as trustworthy as the data governance behind it; garbage inputs produce a beautifully ranked list of the wrong problems.

Crosby: AI in Service of Zero Defects

Crosby's "Zero Defects" philosophy was as much cultural as technical, and he is famous for his claim that quality is free because the cost of prevention is always lower than the combined cost of failure, appraisal, and rework. He was deeply skeptical of

“acceptable quality levels” that tolerated a planned percentage of defects, arguing that any tolerated defect rate becomes a self-fulfilling target.

Crosby would likely embrace AI-driven defect prediction enthusiastically, precisely because it shifts quality left, catching a likely failure during design or early production rather than paying for it in the field. Computer vision systems that flag a solder joint or a weld before it fails inspection fit his philosophy exactly: The earlier the catch, the closer the cost gets to zero.

But he'd push hard against a subtle risk of treating a model's confidence threshold as a new acceptable quality level, quietly tolerating a defect rate the algorithm doesn't flag rather than driving toward true prevention. For Crosby, the measure of quality was always the cost of nonconformance, and he'd insist AI systems report in those terms rather than abstract accuracy scores.

Ishikawa: AI as a Democratized Fishbone Diagram

Ishikawa built his reputation on making quality tools accessible to the entire workforce, not just engineers and statisticians. The fishbone (cause-and-effect) diagram and quality circles were both designed to provide frontline workers with a structured way to contribute to root cause analysis. His principle of company-wide quality control held that the people closest to the work often see problems that no dashboard can capture.

He would likely see generative AI's biggest contribution not in the algorithms themselves, but in translation: a natural-language interface that lets a machine operator describe a problem in plain terms and get back a structured cause-and-effect breakdown, or that turns scattered shop-floor observations into a properly categorized fishbone diagram without requiring a black belt to build it.

His skepticism would center on exactly what AI tends to erode if used carelessly: the participatory, cross-functional discussion that made his quality circles work. A model that outputs a root cause without a team ever discussing it together produces an answer, but not necessarily buy-in, and not the tacit knowledge that only comes from people arguing over a whiteboard.

Shingo: AI as Poka-Yoke at Scale

Shingo's most durable contribution, poka-yoke, was about making mistakes physically impossible rather than catching them after the fact: A part can only be inserted one way, or a jig won't close if a component is missing. His related concept of Zero Quality Control argued that source inspection, built into the process itself, should make sampling inspection obsolete.

AI-enabled vision systems and sensor fusion are close to a direct realization of that idea at a scale Shingo couldn't have built by hand: a camera that halts a line the instant a part is misaligned, a model that predicts a tool is about to drift out of tolerance, and stops the cycle before a bad part is even produced. Shingo would push hard for AI to be embedded at the point of creation rather than the point of inspection; an error-proofing device, not a dashboard reviewed after the shift ends.

The Common Thread

None of these six would treat AI as a replacement for the discipline they spent their careers building. Each would use it to do what they already believed in, faster and at greater scale, including grounding decisions in statistical evidence, reducing variation, focusing effort, preventing rather than detecting, involving the workforce, and error-proofing the process itself.

The risk they'd each flag, in their own language, is the same one quality management has always faced: mistaking a more sophisticated tool as a substitute for management commitment, root-cause thinking, and a culture that treats every defect as a problem worth understanding rather than a statistic worth tolerating. **I-CONNECT007**



Steve Williams is president of The Right Approach Consulting. He is also an independent certified coach, trainer, and speaker with the John Maxwell team. To read past columns, [click here](#).

Flex PCB Body Panels in CubeSats

A standard CubeSat (nanosatellite) leaves an engineer with almost no room for compromise. A 1U unit is a 10 cm cube; even a 6U or 12U bus measures its budget in grams and cubic centimeters, not the kilograms and racks available to a traditional satellite. Every wire harness, connector, and mounting bracket is mass and volume that could otherwise carry a sensor, a radio, or extra battery capacity. That pressure is what has pushed flexible printed circuits out of the wiring harness and onto the structure itself, turning the body panels that form a CubeSat's outer skin into functional electronic assemblies.

Instead of aluminum panels with cable harnesses routed along their inner face, a growing number of CubeSat programs now build the panel around a flex or rigid-flex circuit. The polyimide substrate is bonded directly to (or laminated onto) the structural

panel, carrying power distribution, data buses, and sensor traces across the full face of the spacecraft. The panel stops being passive structure with wiring attached to it and becomes the wiring itself.

Why Flex Makes Sense at CubeSat Scale

Three constraints make flex circuits a natural fit for this role.

1. **Mass and volume.** Discrete wire harnesses require connectors at every junction, strain relief, and routing clearance. A flex circuit integrated into the panel eliminates most of that hardware, replacing bundles of point-to-point wiring with a single, thin, conformal layer that adds negligible mass.
2. **Survivability under launch loads.** CubeSats ride to orbit as secondary payloads, often in a Poly-Picosatellite Orbital Deployer (P-POD)

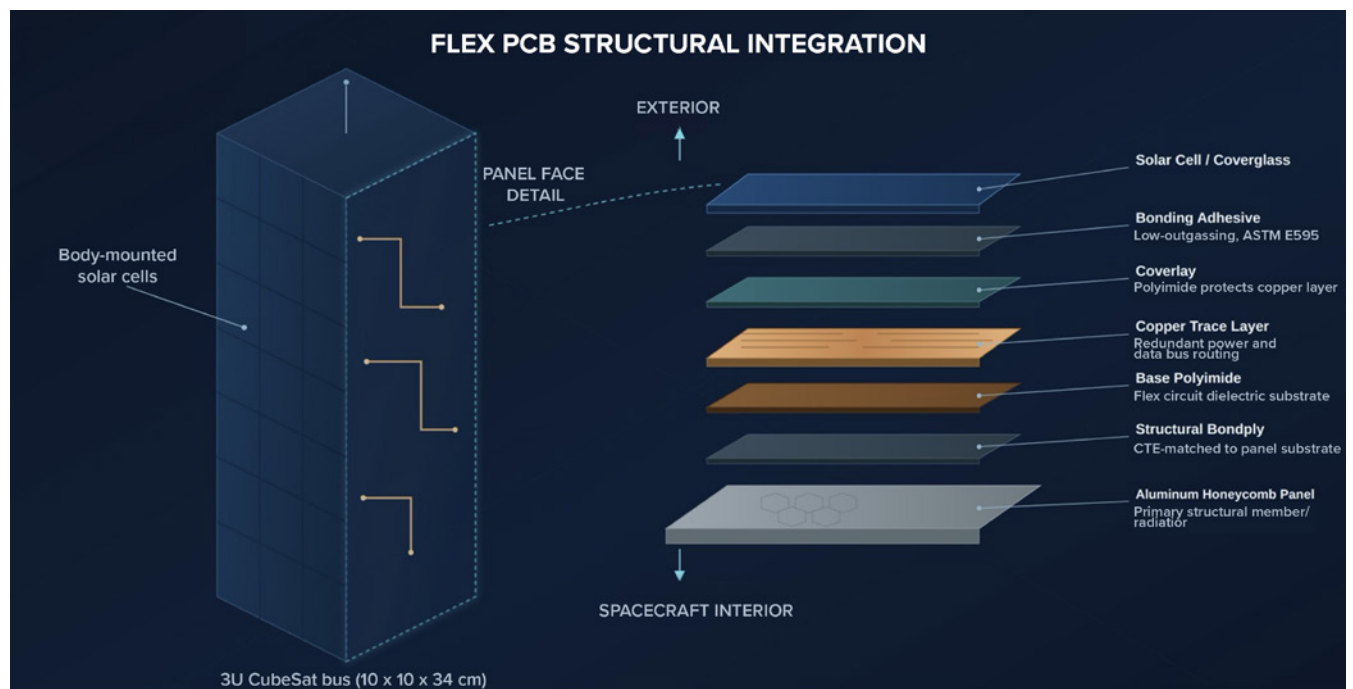
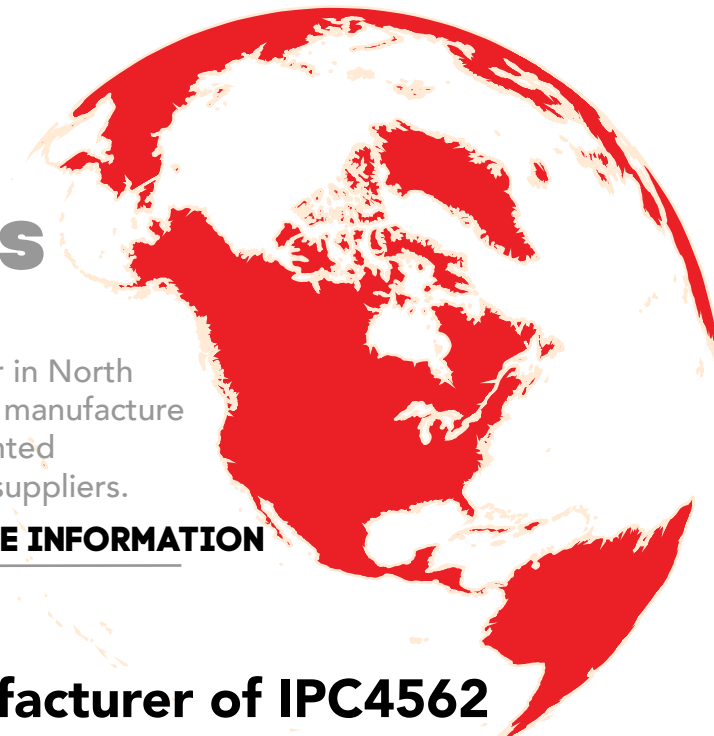


Figure 1: The body panel of a CubeSat.

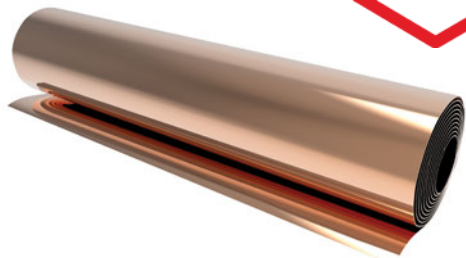
WORLD-CLASS SUPPLIERS

Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, salutes our premier suppliers.

CALL 949.587.3200 FOR MORE INFORMATION

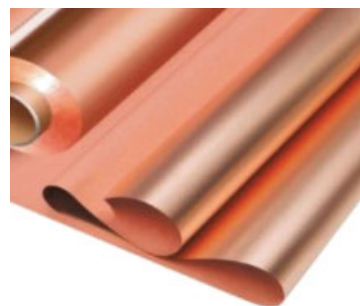


Manufacturer of IPC4562 B Grade HTE Copper Foils



CCP is a leader in the manufacture of high-quality electrodeposited copper for the printed circuit board industry. This privately held and Taiwanese owned has been

supplying foil since 1987. They produce a full range of grade 3 HTE foils in both matte side and shiny side (RTF) treatments for high performance markets covering HSD and RF with foil profile options as smooth as <1 micron.



Conformal Release Film



Pacothane provides innovative lamination processing solutions including revolutionary PACO•CLUTCH™: decouples PCB panel from lamination separators, simplifies layout, improves scaling predictability, and improves yield for cap lamination, thin core multilayer, and sub laminations.



FOCUSTECH™ was founded on the belief that chemicals and control systems must work together to provide the most consistent and efficient manufacturing processes and has always taken a process control approach when formulating chemical products.

FOCUSTECH™ is the only supplier to the PCB industry that designs and manufactures both process chemicals and control systems. Products include cleaners, etchants, developers, and strippers plus control systems.

A Complete Line of Chemistry & Controllers used in PCB Manufacture & Metal Finishing



Precision is our specialty!

or similar deployer, exposed to vibration and shock profiles that are punishing relative to the spacecraft's size. A flex circuit bonded across a rigid panel has no discrete wire segments to fatigue, no connector pins to back out, and no harness loops to resonate. The copper traces move with the structure rather than against it.

3. Real estate for solar cells and antennas.

Body-mounted solar arrays, patch antennas, and deployable mechanisms all compete for the same six faces of the bus. A flex circuit embedded in the panel can carry solar cell interconnects, antenna feed lines, and sensor traces on internal layers, freeing the exterior surface almost entirely for photovoltaics and RF elements.

Engineering the Panel-as-Circuit

Turning a structural panel into a circuit board is not a matter of gluing a flex cable to aluminum. It requires the same rigor applied to any spaceflight electronic assembly, with a few additional constraints layered on top.

Material selection starts with low-outgassing polyimide and adhesive systems that meet NASA ASTM E595 outgassing limits, critical when the circuit sits centimeters from an optical sensor or a deployable solar array that cannot tolerate contamination.

Thermal management becomes a structural problem as much as an electrical one. CubeSats swing through wide temperature extremes as they cross the terminator, and the panel itself is often the primary radiator. Copper coverage, thermal via placement, and CTE matching between the flex layer and the panel substrate must be engineered together to prevent repeated thermal cycling from delaminating the bond line or fracturing traces at flex-to-rigid transitions.

Radiation tolerance pushes designers toward simpler, more redundant trace routing rather than dense digital logic on the panel itself. The body panel typically carries power and low-speed sensor/telemetry lines, while more radiation-sensitive digital processing stays on conventional rigid boards inside the bus, shielded by the surrounding structure.

Mechanical integration must account for the panel's dual role. Mounting points, deployment hinges, and separation switches all interact with

the circuit layer, so via and trace keep-out zones need to be defined early, in collaboration with the mechanical design, rather than added after the fact.

What This Enables

The payoff is a spacecraft where structure and electronics are no longer separate disciplines competing for the same mass budget. Power and data can be distributed to every face of the bus without a single point-to-point harness. Solar cells can be interconnected directly through the panel they're mounted to. Sensors (magnetometers, sun sensors, temperature probes) can sit at the exact panel location an experiment requires, with their signal paths built into the surrounding structure rather than running to it afterward.

For a class of spacecraft defined by doing more with less, that convergence of function is the point. A flex or rigid-flex circuit that also serves as structural skin doesn't just save mass on paper; it removes an entire category of failure points (connectors, harness chafe, loose fasteners) that have ended more than one CubeSat mission early. As constellations grow and mission timelines compress, the ability to fabricate a body panel that is simultaneously airframe and electronics is becoming less of a novelty and more of a baseline expectation for anyone designing at the small-satellite scale.

Conclusion

For PCB fabricators, this trend means treating rigid-flex capability, low-outgassing material qualification, and close mechanical collaboration not as niche services, but as core competencies for the next generation of space hardware. **I-CONNECT007**

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. He is the author of *Thermal Management: A Fabricator's Perspective*, *The Printed Circuit Designer's Guide to DFM Essentials*, and

The Companion Guide to Flex and Rigid-Flex Fundamentals. Visit I-007eBooks.com to download these and other free, educational titles.



WORLD-CLASS TOOLS

Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, provides game-changing precision cutting tools to our customers.

CALL 949.587.3200 FOR MORE INFORMATION

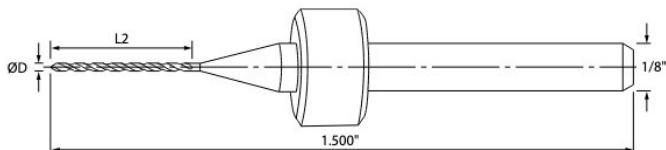


PRINTED CIRCUIT BOARD CUTTING TOOL SOLUTIONS

Kyocera designs and manufactures tight tolerance precision carbide cutting tools for PCB applications. Products include drills, routers, end mills, and specialized cutting tools ranging from: 0.05mm to 6.70mm diameters (0.0020" - 0.2638").

Drills, End Mills, Routers, Specialty Tools

Kyocera's renowned array of drills, routers, end mills, and specialty tools offer many benefits and advantages to customers for all their drilling and routing needs. Kyocera's tool reliability and design flexibility are key strengths of their products, along with a complete R&D facility in the US, new tool manufacturing capability, and local technical resources, all available to support customer needs.



4 Facet Point Geometry

REPOINTING will be a new service offered by Insulectro through Kyocera. The company has recently invested in automated, state-of-the-art equipment and all repointing will be done in Southern California.

Building Value for Europe Through Smart Standards *and More*



Europe's electronics industry is navigating major shifts, from AI and automation to workforce challenges, sustainability, and evolving industrial policy. As the Global Electronics Association's new vice president for Europe, Dominik Grau wants to understand what companies across the region need, and how the Association can help deliver it. In this interview, Dominik shares what he's hearing directly from European electronics companies, why smart standards will become increasingly important in an AI-driven industry, and

where he sees opportunities to bring together technology, workforce development, and the resources companies need to compete.

Marcy LaRont: Dominik, congratulations on your new role with the Association. Can you share a couple of highlights from your background?

Dominik Grau: For the past seven years, I've been working on standards, AI, business development, and the digital future for the German Institute of Standards (DIN), a close partner of the Global



pluritec® GROUP

Advanced Technology for Printed Circuits Boards



DRILLING - ROUTING
FLASH/BEVEL PANEL PREP
BACKDRILLING - FLIP DRILLING
X-RAY OPTIMIZER
FACTORY 4.0
AUTOMATION SYSTEMS



PUMICE SCRUB
CHEMICAL CLEAN
DEV-ETCH-STRIP
SHADOW
OXIDE/BONDFILM



SPRAY COATING
SOLDER & RESIST
FASTACK OVEN
TUNNEL OVENS
U.V. OVEN

X-CUT

FROM LAMINATION TO
DRILLING IN ONE STEP



D.E.S.

"SMART" ETCHING CONTROL
AND ULTRATHIN 1 MIL
MATERIAL HANDLING

SPRAY COATING OVEN

COAT AND TACK CURE READY FOR
IMAGING IN 20 MINUTES



Electronics Association. DIN Media, where I was the managing director, sells IPC standards in Germany.

How did you become aware of this new role with the Global Electronics Association?

A friend of mine said, "Look at this job opening. It sounds like you." I didn't have that on my radar, but I knew of the Global Electronics Association as I've been working with electronics companies up and down the supply chain as part of my smart standards efforts at ISO, CEN, and DIN. It seemed a natural fit.

Have you always worked in electronics and manufacturing?

I've been involved with the electronics industry since I started working for a computer magazine called *Chip* about 25 years ago. Eventually, I joined the German subsidiary of the parent company of *Computerworld* and *PC World*, International Data Group (IDG), which was based in Boston. I'm a trained journalist and business economist who started working as an editor and business de-



Dominik Grau

veloper. I moved into online and digital businesses, and then eventually into general management. As a managing director, I worked at tech, media, and standards companies in Europe and the U.S.

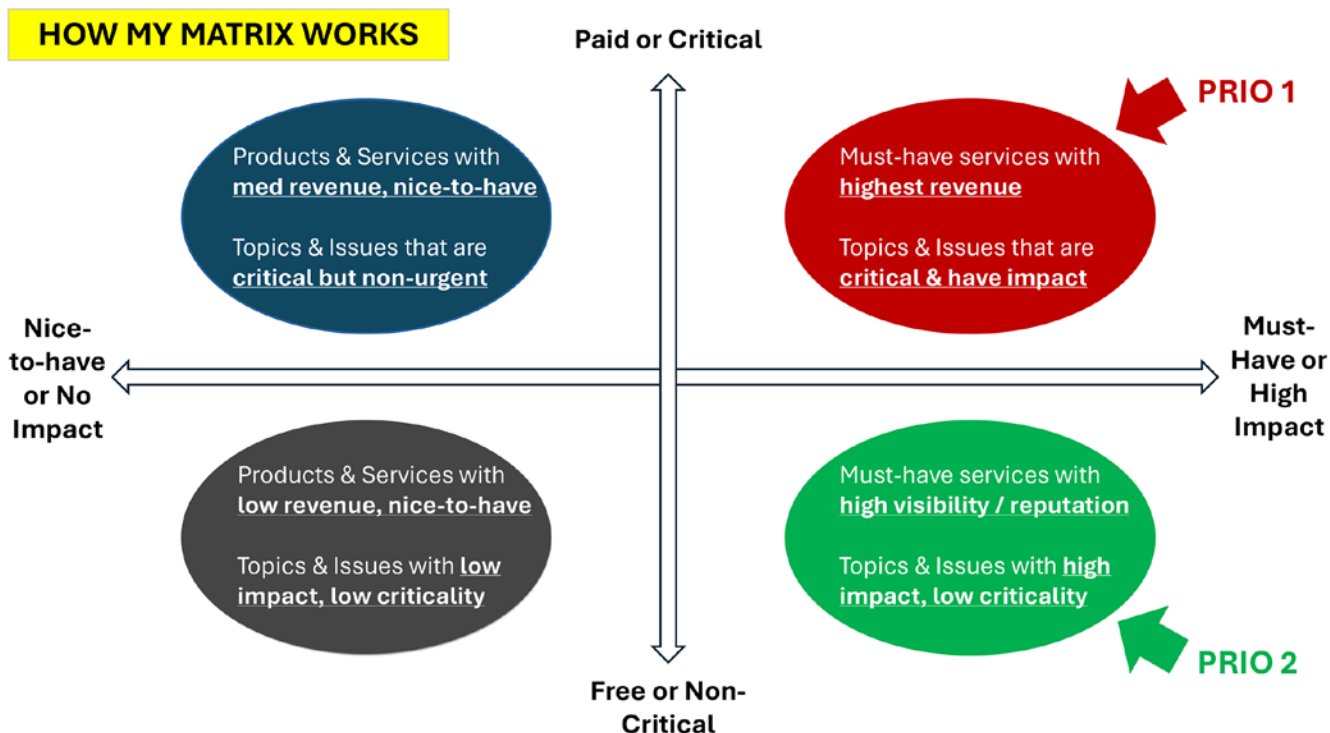
I was actually the reverse, where I started in manufacturing and found my way into publishing. It is interesting to look at where we start and where we end up.

I was a global publisher for a media company based in Manhattan, so I've had a strong network in the United States, which makes this role with the Global Electronics Association, with its headquarters in Chicago, a natural fit.

Dominik, tell me about your standards work in Germany.

I was brought on board by the German management of DIN, the Deutsches Institut für Normung (the German Institute for Standardization), for two main projects.

One project was to rebuild the company from a traditional standards publisher into a software busi-



ness for standards, serving German corporations and SMEs. My role was to rethink the way standards were being provided to industry there. I joined DIN Media as a managing director in 2019, and I said to myself, "They've been selling print standards for almost a century, and about 25 years as PDFs. It's time for something different. The industry has changed." They expect us to deliver it in a way that's more natural to what they're doing in terms of automation, digitalization, and later also robotics that are now just around the corner, in other words, all things smart standards.

Building on the existing tech teams at DIN Media, we built a user-centric software and AI company that eventually became an independent company in its own right. It's now called DIN Solutions, which is the high-tech innovation and AI arm of DIN. My main expertise is in industrial AI, data strategies, and business development, which I've been involved in for almost 10 years now, through software development and transforming businesses. That's one reason this job interested me. I do not develop the content of the standards; I focus on the standards' user side of our business. I come from the tech, software, and AI angles to service the community as well as possible.

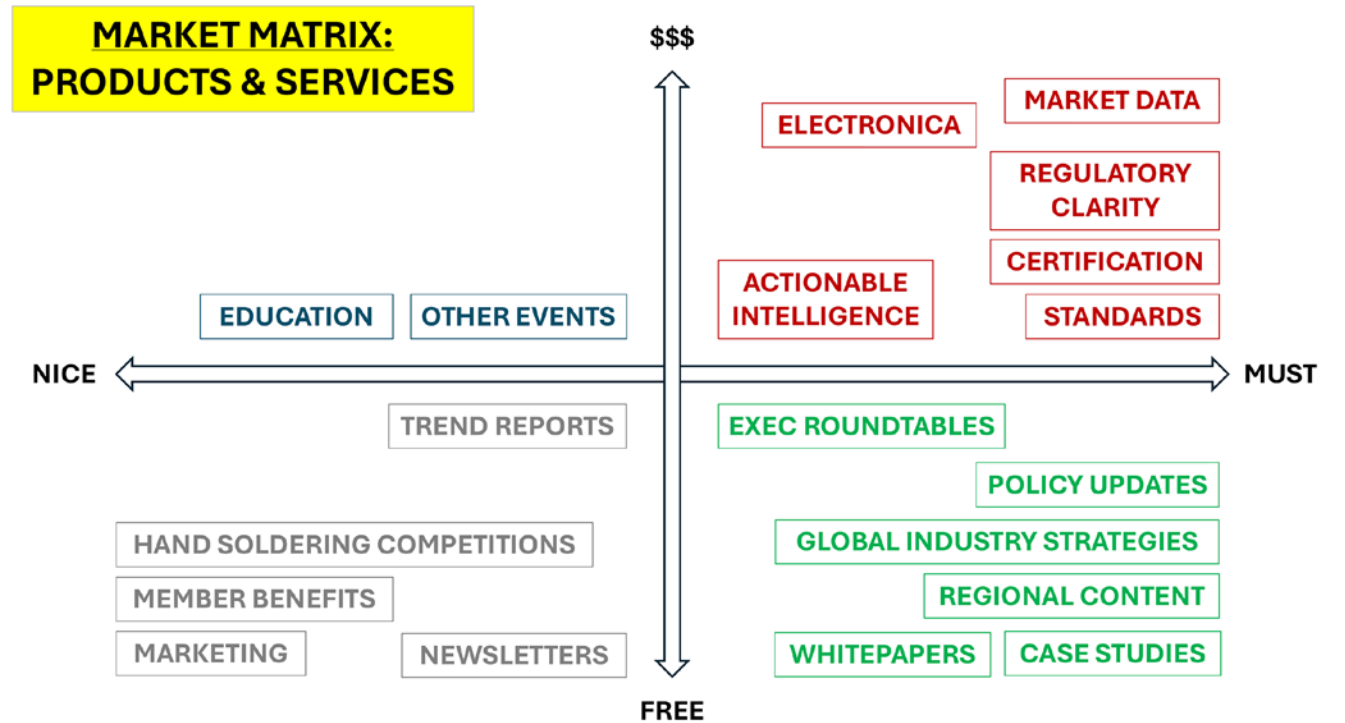
Given that DIN is a standards organization in and of itself, would you explain the relationship between DIN and the Global Electronics Association?

DIN sold IPC standards to our German customers. When I started working on smart standards with ISO and CEN, we reviewed many standards from around the world, including, of course, IPC standards. The electronics and chip industries work with IPC standards, and, in my mind, they sit at the very center of the European chip strategy. Smart standards are a gateway to the digital future of European industry, of which electronics are an integral part. That is why, in my opinion, IPC standards will play a key role in Europe's future digital landscape, particularly in smart standards and industrial AI.

What do you mean by "smart standards?"

Smart standards are machine-readable and applicable. Machines, robots, and any kind of device, whether in a manufacturing or industrial environment, can immediately use these standards.

Usually, they come in a format like XML or another digital format that you can push directly into a machine, computer, tablet, robot, or other manufacturing equipment in real time, and those devices can act on them immediately. In the not-so-



distant future, with smart standards, there will be no permanent need for a human in the loop feeding information into machines, taking the place of some quality assurance and other human operations. Machines will retrieve the standard from the web shop or another API system, load it, and act on it immediately.

Smart standards are configured in a way that the machine understands. “Smart” means machine-applicable, readable, and transferable across devices. But, of course, their application, the development of the underlying software and infrastructure, and the strategy behind that must be human-driven. This is also true for the quality management perspective and the certification part of our business.

In this new role for the Association, what do you hope to accomplish?

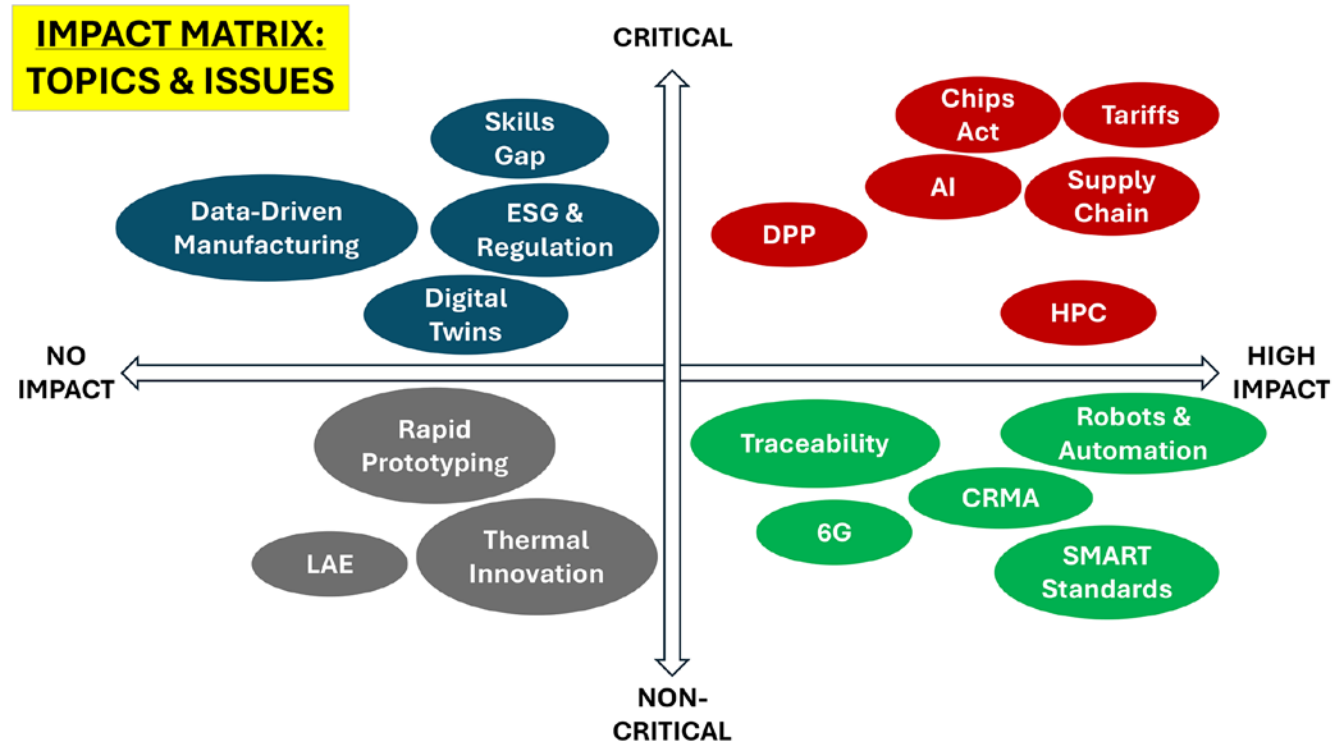
We are a member-driven organization that serves members and the industry as a whole. So, my primary goal is to work closely with current members while bringing in new members, where we

all recognize the value and benefit of being part of this network in Europe. My mission is to understand the priorities and needs of our members and potential members.

After that, I want to serve the industry as it works on its own transformation in terms of automation, especially industrial AI, robots, manufacturing of the future, smart standards, and critical raw material and sustainability efforts by the European Commission. How can the Global Electronics Association in Europe be of service and translate politics into executable information or actionable intelligence? Achieving these goals will help us be perfectly aligned with the market.

I understand you actually reached out to current members while you were interviewing for the position to receive feedback on what they found most valuable about Association membership. How did that help you?

Yes, I had great conversations with a handful of companies, both members and non-members, at their offices. I compiled the feedback into a matrix—



a tool I've used for almost 15 years in my managing director roles—that divides the company feedback into four categories. The top two boxes give you guidance on “critical” and “must-have” services for members, and the industry in general, and indicate where they are willing to invest in, indicating where they are willing to invest and identifying services they acknowledge are needed but don't feel they should invest in just yet.

I have a second matrix that lists the topics and issues that came up during these conversations. I ranked them based on the members' feedback to determine the most critical and high-impact issues. These are the ones that will get immediate focus.

What did you learn from this exercise?

Interestingly, in the past, the lower-priority issues were actually the things delivering membership growth, not typically the priority-one issues. Though important overall, those issues tend to be more expensive to address or involve. Members may be readily willing to pay for it, but nonmembers are not. Nonmembers need free or low-cost services that they perceive as high value to convince them to join. Once they have established the value of their membership, they are typically more ready to move toward priority-one initiatives and are willing to invest more in additional services. In Europe, we will focus on both areas from the outset.

Can you share some of the specific feedback you received?

The feedback we received indicated that industry companies, both Association members and nonmembers, expect executive roundtables, white papers, and case studies as part of membership in a professional organization. They have certain topics on their minds, such as AI, the skills gap in the workforce, and smart robots. A hand-soldering competition on its own is certainly fun and interactive, but it will not compel membership directly. If we want to leverage those areas, based on the members' feedback, we should do a live soldering competition with the hand-soldering world champion against a robot.

That would definitely get a lot of interest!

Yes, you get the idea. It's about combining topics with formats that industry expects will help bring these things together.

Another area of high interest is the roundtable discussion on topics top of mind for executives, including AI and the workforce skills gap. We believe there's value in discussing these two dimensions together because to use AI efficiently, you need people with the skills and experience to do so, and to develop those skills and experience, you need education and training. So, AI and workforce education go together. By the way, AI can accelerate the acquisition of new AI skills and make the whole educational experience more efficient. The best AI teacher sometimes is the AI.

In case studies, when I spoke with two of the largest EMS companies in Europe, both mentioned that they work with many robots. When I visited their factories and warehouses, I saw the robots running around the warehouse, delivering BOM items to the next machine in the process. That was really fun to watch 10 humans and 10 robots all working together almost at the same time. Both companies said, in essence, “We are willing to support a case study on the efficiency and operational savings of cobots and robotic manufacturing in EMS because we think that's a huge trend, one that will be coming to smaller companies.”

After those meetings, I asked myself, “What might a white paper look like based on the feedback I've gotten so far from these companies during these conversations?” In the past, at several organizations, I worked extensively on white papers. Three topics stood out: AI, rapid prototyping, and smart standards. They are intertwined. You cannot do prototyping in a rapid environment without AI, and you cannot do AI in the future of electronics without smart standards. So, this will be a very interesting intersection for the next four to five years. I am looking forward to a deep dive on that and feel it will be of great value to the industry.

Dominik, there is a lot happening in Europe, and I wish you great success. I look forward to catching up with you again soon. I-CONNECT007

Employee and Operator Empowerment in the Age of AI



BY DON BALL, CHEMCUT

In an ideal world, every operator would understand the process they are working on and be able to diagnose problems and suggest improvements. In the real world, however, we all know that these types of people are, if not rare, at least in the minority. Where do these people fit in when everyone is turning to automation and AI for quality and yield improvements and cost reduction? (Never lose sight of the fact that, in the end, it's always about the money.) In my opinion, these people are valuable resources, and every effort should be made to retain them, regardless of their education level or seniority.

During my career, I've had the privilege (mostly) of visiting dozens of PCB manufacturing sites, good, bad, or indifferent, and have formed the opinion that how these facilities treat their empowered operators is a sign of how good or bad that facility is. Here are some examples.

Examples of the 'Good'

Just a few months ago, I participated in a video conference with a smaller shop that was looking to add some automation to its cupric etch system.

They had a basic etcher that depended on an experienced operator to keep the system running. He was the type of guy who was continuously monitoring the specific gravity, ORP, and free acid levels, etc., to make sure they stayed within the optimal operating parameters and was trusted enough to do this with minimal supervision.

That operator was in the video conference along with the process engineer and plant manager, which, to me, was a good sign of a good shop. After we had come up with a system that met their needs and price range, I pointed to the operator and said, "Whatever you do, don't get rid of this guy. The automated system is good, but it still needs direct supervision." The manager said, "Oh no, the plan is that, since he doesn't need to be constantly monitoring the etch chemistry, we promote him to supervising the whole DES operation so our process engineer can spend less time there and work on other problems." That, in my opinion, is exactly how it should work.

Another time, I went to a facility to help with a process problem and was left to solve the problem

on my own while the process engineer went off to do something else. This happens far too often, but in this case, the etcher operator took an interest in what was going on and asked all kinds of questions about what I was doing and why. By the time the problem was resolved, he knew exactly what had happened and took responsibility to see that it never happened again.

I told his supervisor to give this guy whatever he wanted in terms of wages because lost time and production while trying to replace him (if they could find someone as good) would be more than any salary increase. The supervisor smiled and said he knew that and that it wouldn't be long before he was running the entire etching department. When I visited again six months later, that's exactly what had happened. Again, exactly how it should work.

An Example of the 'Bad'

However, the idea of using automation to free up time for a competent operator or process engineer to expand their responsibilities can be taken too far. Several years ago, I visited a rather large PCB shop on the West Coast to help with a process problem involving our equipment. I met their process engineer, who was responsible for the entire wet processing (DES, SES, plating, etc.) area across the whole facility.

The actual problem turned out to be a rather simple one with an easy fix, but he was constantly being called off to other sections of the plant to deal with other issues, so it took several extra hours to find and fix it. For months after that visit, I got phone calls and emails from him, time-stamped 10 p.m. or 3 a.m. and other ungodly hours. He was always at the plant for one emergency or another.

About a year and a half later, I got a call from a headhunter offering a wonderful job on the West Coast at a generous salary increase. As she described the job, I began to get suspicious and asked if the job was at this company I had just visited the year before. Well, yes, it was.

I politely turned down the opportunity since I wasn't eager for a job that required me to be on call 24 hours a day, seven days a week (that and the fact that the rent on an apartment a one-hour commute from the plant was more than the mortgage on my house, a 15-minute commute to the plant, in State

College, Pennsylvania). It isn't always about the money, you know. I later learned that the process engineer had burned out, suffered a nervous breakdown, and left the company to recover and seek sunnier climes.

So, using more automation to free up time for good employees to expand their responsibilities and retain their services is a good idea, but don't take it too far.

And What About AI? A History Lesson

Now we come to AI. I freely admit that I am uneasy about it—not so much about AI itself as about the motives of the people rushing to embrace it. It seems to me that the underlying (and unspoken) reason AI is so eagerly anticipated by some isn't that it can increase efficiency and quality (which it can), but for the potential to replace employees that need to be paid and thus maintain or increase profit levels. Granted, this will happen, but where do the reductions in personnel come from?

Back in the ancient times of the PCB industry (1980s), American PCB shops steadily lost business to Japan, which supplied high-quality circuit boards at very competitive prices. The Japanese had aggressively automated their production and run their plants with a few well-trained and well-paid employees. The response from many American shops was to lay off their highest-paid employees and hire more minimum-wage workers to cut costs and compete on price. Most of the shops that adopted that tactic were soon out of business, and the Japanese shops took over even more of the business.

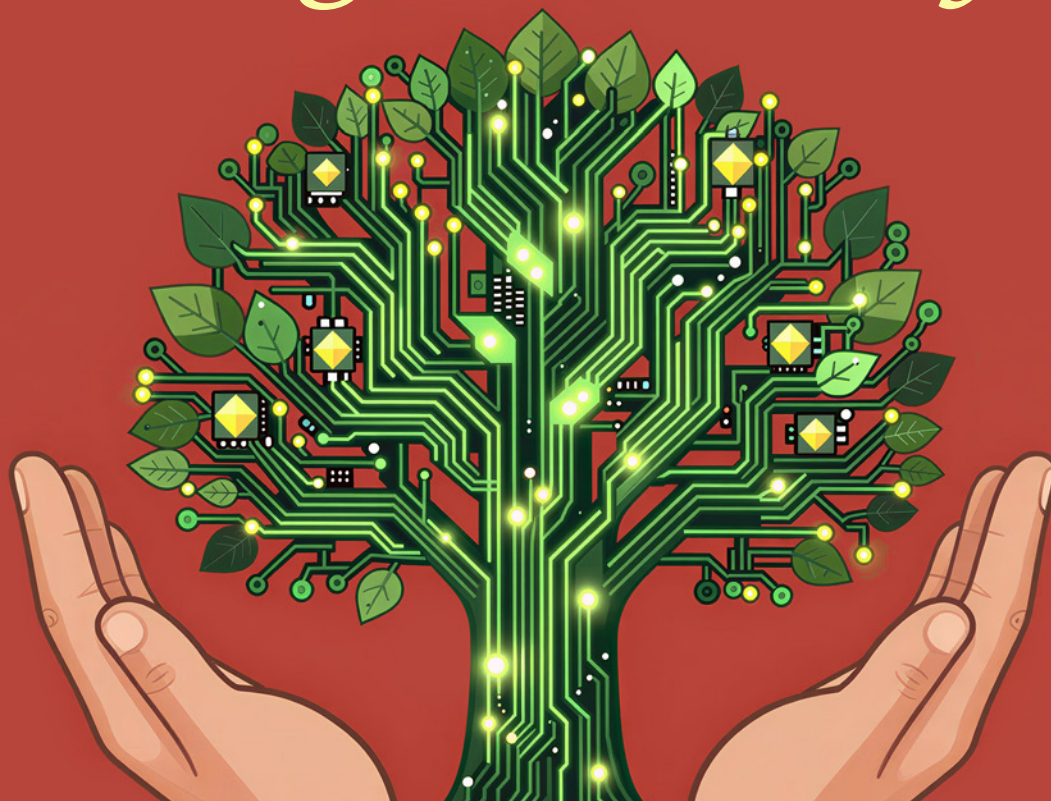
AI is coming, of that there is no doubt. My observation is that the good shops will use AI to increase the efficiency and productivity of their empowered employees, not to get rid of them. In my opinion, based on reports of experimental testing and real-world failures, no AI should be allowed to run without some human oversight and supervision.

I-CONNECT007



Don Ball is a process engineer at Chemcut. To read past columns or contact Ball, [click here](#).

Building Community, Sharing Knowledge *for* *Designers in Italy*



Building an industry community takes more than a good idea. It takes persistence, participation, and, perhaps most importantly, a willingness to speak up. Pietro Vergine, co-owner of Leading Edge and the first CID+ trainer in Italy, has spent years helping establish the IPC Designers Council Italy (IDCI) and participating in IPC international standards development. Pietro has become an advocate for greater knowledge sharing and a stronger European voice in PCB design. At the heart of it all is a simple idea: Give designers a place to connect, freely exchange knowledge, and make their voices heard.

***Marcy LaRont:** Pietro, it's nice to meet you. How did you first get connected with the Global Electronics Association?*

Pietro Vergine: When my partner, Nigel Woolaway, and I started Leading Edge in 2005, we proposed a European-level innovative solution for the electronics design domain at both the IC and PCB levels. That was our expertise, having worked at both Mentor Graphics (now Siemens EDA) and Synopsys. As we began, we reconnected with both companies to collaborate and teach classes on their behalf in the European market. That is how Leading Edge began.

Your Idea. Our Expertise. Complete Solutions.



From your idea to final inspection, our end-to-end services will transform your cutting-edge ideas into reality.

Our PCB production capabilities include:

- State-of-the-art 45,000 sq. ft. manufacturing facility
- Precision drilling and plating
- A variety of final finishes and lamination options
- Advanced technologies, including RF and HDI
- Testing and quality assurance

Our dedicated team is here to assist you throughout your PCB journey. Contact our team for quotes, technical assistance, or to discuss your specific PCB requirements.

OUR CAPABILITIES ►



UNITED ELECTRONICS
CORPORATION

In 2016, a customer asked me to lead an IPC CID certification course. I wasn't familiar with IPC (now the Global Electronics Association), but I learned about the organization, earned my CID, and took my CID+ certifications. I became Italy's first CID and CID+ trainer, which marked the beginning of our partnership with the Association. Today, we are its premier training provider in Italy for standards and CID/CID+ certification.



Pietro Vergine

ever shows up to a standards meeting is typically who gets a vote (though there is a mechanism for absentee votes to be submitted), but if there are only 10 people at a meeting in Europe and 100 in the U.S., you can see how the European input easily gets lost. The European committee members may all agree on something, but we will still be unable to implement what we felt was important at the main meetings.

Does your teaching expand beyond Italy?

As far as IPC certifications are concerned, we're also trainers for Advanced Rework Technology in the UK. As such, we release all available certifications today wherever there is an opportunity to do so. I always attend APEX EXPO to contribute my voice and be part of these standards committee meetings.

What are your thoughts on being involved in the standards process?

By design, it is a very elaborate and extremely important process. For those of us outside the U.S., it sometimes feels challenging to get our voices heard, which is especially important when word choice translations from English into other languages affect how things will be read and understood.

Unfortunately, there have traditionally been fewer Europeans represented at most of the big standards meetings. The process inherently lends itself to being overly U.S.-centric, though I believe the Global Electronics Association is trying to be more globally inclusive through its regional offices and by expanding meetings with regional events like this year's IPC Builds in Paris.

Yes, despite some efforts, standards activity is still largely U.S.-based. I believe IPC Builds and the Pan-European Design Conference are efforts to ensure greater international input into standards.

Unfortunately, I will be unable to attend IPC Builds (taking place in September 2026) due to my teaching schedule, but I look forward to hearing some positive feedback about it. In my experience, who-

How did IDCI get started, and how active is the Italian design community?

Ten years ago, I started talking to the Global Electronics Association about the need to establish a designers council in Italy, like the one that had originally existed in the United States. But I wanted to do something slightly different. I wanted to run the council, not as a forum, but as a place for stakeholders—PCB designers, electrical engineers, managers, or whoever could join from time to time—to discuss relevant issues, but without any real expectation. At its base, this would be about networking and sharing knowledge. I wanted it to be at no cost to participants and I would only ask for small contributions if we needed to pay for a high-level speaker.

Were you able to do as you hoped?

Yes, for the first chapter meeting, we invited industry expert Jan Pedersen of NCAB. He spoke about the need for Class 3 designs at the PCB level. It was free to attend. This was during COVID, so we met remotely, but it was a good knowledge exchange, and it was genuinely appreciated by all who attended.

Did making it so accessible spur better participation?

Yes, it did. I'm familiar with the Italian market. If I ask people to join a free meeting where I will give them valuable information, I will fill the room. If I start saying, "Yes, you come, but then you need to help me by working on X and Y," they will say, "Sorry, I have no time." At one point, we had more than 20 active members.

Right now, some things have not worked in our favor, so we are back to trying to get consistent

participation from past and new members. I am hopeful we can find a way to make this happen, that the Global Electronics Association will remain a good partner with us, understand some of the more unique aspects of our community, and that we will find a way to give our voices more influence in the overall process.

It's a delicate balance. You obviously speak in terms of Italy, but is that more or less true across Europe?

No. For instance, Italy is the second largest design chapter in Europe after France, which is much larger and more active. But French people think and work differently from Italians. Europeans work/think/ behave differently depending on the country they live in, so we cannot "standardize" how each chapter acts by doing things the same way as in other countries; that doesn't work well, at least, this is my experience.

Pietro, I understand that the CID curriculum is now out of date. What is being done to address that?

Yes, the CID and CID+ materials are 10 years old and are no longer adequate. So, I'm more challenged

during classes, as students ask me questions about new design technologies and methodologies that aren't covered in the current material. As a trainer, I want to add value, open the discussion, and share my experience. A few months ago, I sent the syllabus to a customer and informally asked him to study the guide because we would discuss it during the class. It's also on the examination. He called me, concerned, saying, "But the material in this guide is old." I had to say, "You're right, the material is not updated, and what's not in the guide will be discussed during the class."

So, to provide a valuable education to students, we have to teach around the curriculum. Luckily, at the end of the day, even though the material is not updated, the process is still there, and the idea is to learn the process and the challenges designers face. I know the Association is aware of it. But this is something that needs to be addressed, and quickly.

Thank you, Pietro. I wish you continued success with the designers council and working to get European voices heard.

Thank you, Marcy. **I-CONNECT007**

Insight, Delivered

Subscribe to our daily and weekly newsletters, tailored to your interests, and stay connected to what matters most.



SUBSCRIBE NOW →



I-Connect007
The Global Electronics Resource



The Rise and Fall of Europe's PCB Design Community

BY JOHN WATSON, PALOMAR COLLEGE

One of the most famous lines associated with the attack on Pearl Harbor was attributed to Japanese Admiral Isoroku Yamamoto. After the horrific attack of Dec. 7, 1941, he was reported to declare, "I fear all we have done is to awaken a sleeping giant." That image has endured because it captured the moment when something enormous, and perhaps underestimated, began to move.

Today, another giant is awakening in Europe, and it may not be what or where you expect. It's not from one country, company, or organization, but from an entire region that has built a deep engineering heritage and a long history in electronics. At the center is Europe's PCB design community.

A few years ago, EIPC released an excellent, detailed report about the challenges facing the

We are dedicated
to excellence through

innovation, technology and most importantly, service.



COMMERCIAL • MILITARY • MEDICAL • BAREBOARD



U.S.CIRCUIT

www.uscircuit.com



European PCB industry.¹ It warned that Europe's PCB industry had declined sharply, and manufacturing capacity and its supporting materials supply chain were shifting to Asia. This growing dependence, combined with high energy costs and limited domestic capacity, posed increasing risks to Europe's industrial, aerospace, and defense sectors.

The report stated that in 2000, Europe accounted for about 16% of worldwide PCB production. By 2022, that share had fallen to just 2.3%, while the number of European PCB manufacturers dropped from 555 to fewer than 200.

This decline reflects a broader trend across several key industries in Europe, the United States, and elsewhere, where significant portions of manufacturing have shifted overseas. Those are not small changes. EIPC reported that Europe's share of global PCB production fell by roughly 86%, while more than 60% of its PCB manufacturers disappeared, consolidated, or left the market entirely.

But although the industry in Europe shifted, it didn't stop designing electronics. Far from it. Europe has

remained a major force in automotive, aerospace, industrial electronics, medical systems, telecommunications, defense, and advanced technology. True, the manufacturing footprint may have shrunk, but the engineering capability remained.

So, why would I be bold enough to say we are awakening a giant? Because I've been personally involved with the European PCB design community for the past 10 years. It started with productronica, the Altium Live conferences, and now the Pan-European Electronic Design Conference (PEDC). I've interacted with FED, TechWorks, and IMEC, and as a technical advisor to the Delft Hyperloop team in the Netherlands.

Along the way, I've met many great designers, fabricators, suppliers, and engineers. It's always been obvious that Europe has the expertise. There are world-class engineers, strong universities, major technology companies, and some of the most demanding electronics industries. The challenge, as in the United States, is ensuring there are enough skilled designers to carry the work forward.

Another challenge is how that talent is organized. For years, much of Europe's PCB design strength existed in separate regional pockets. For example:

- Germany developed deep proficiency in automotive, industrial electronics, and manufacturing
- France built strength in aerospace, defense, and telecommunications
- The UK developed strong semiconductor, embedded-system, and hardware-design communities
- Scandinavia became closely associated with communications, wireless technology, and advanced electronics
- The Benelux region built a powerful semiconductor, research, and technology ecosystem
- Italy and Central and Eastern Europe continued to build strong PCB and electronics manufacturing capabilities of their own

Different languages, national organizations, conferences, industries, and engineering cultures



The giant is not being created; it is waking up. As Europe's designers begin to connect, collaborate, and recognize their collective strength, we may be seeing only the beginning of what this community can become.”

naturally kept many designers focused within their own countries or regions. A PCB designer in Germany could be solving many of the same problems as a designer in France, Italy, Sweden, or Poland without necessarily feeling part of one larger European design community.

I fully believe that is changing as designers are connecting more effectively across borders, technical conferences are becoming more international, and industry organizations are placing greater emphasis on design. Suppliers, fabricators, educators, researchers, and designers are showing up in the same rooms and having the same conversations.

PEDC, for example, deliberately brings together Europe's electronics design and research communities rather than focusing on a single country or part of the electronics supply chain.

In recent years, the European Union (EU) has also steadily increased its focus on electronics and semiconductor capability. The European Chips Act entered into force in September 2023, aiming to strengthen Europe's semiconductor research, manufacturing, and supply-chain resilience. In April 2025, the EU launched its AI Continent Action Plan, followed by major investments in AI factories and computing infrastructure across Europe.

Then, in June 2026, the European Commission proposed Chips Act 2.0, aimed at further reducing strategic dependencies and expanding both advanced and mainstream chip design and production within Europe.

At the same time, Europe has been increasing its focus on defense manufacturing. The EU introduced its first European Defense Industrial Strategy in March 2024, with longer-term goals tied to European defense readiness through 2030.

These initiatives show that Europe's renewed emphasis on electronics did not appear overnight. Since 2023, the pace has clearly accelerated, with semiconductors, AI, defense, advanced manufacturing, and supply-chain security being treated as strategic capabilities.

This creates an interesting situation because while Europe's PCB manufacturing base has shrunk dramatically over the past two decades, the need has grown for more sophisticated electronics.

On both sides of the pond, as they say, we are recognizing that allowing large portions of Europe's electronics manufacturing capability to migrate elsewhere has come with major consequences. But rebuilding an industry takes much more than factories. It takes people who know how to turn complex technology into something that can be built. Processors, memory, sensors, RF devices, power systems, connectors, advanced packages, and thousands of other components still must come together somewhere. That somewhere is the PCB, brought together by the PCB designer.

So, when I call Europe's PCB design community an awakening giant, I am not suggesting that the capability suddenly appeared. The capability, experience, and engineering strength were already there. The connection between those communities, the investment surrounding them, their growing visibility, and the recognition that PCB design will play a critical role in Europe's technological future is what has changed.

That future is getting more exciting by the day. I believe the push for a stronger European electronics supply chain will demand more capable designers, closer collaboration, and an entirely new generation of PCB professionals.

The giant is not being created; it is waking up. As Europe's designers begin to connect, collaborate, and recognize their collective strength, we may be seeing only the beginning of what this community can become. **I-CONNECT007**

References

1. "Challenges Facing the European PCB Industry," EIPC, March 2024.



John Watson is a professor at Palomar College, San Marcos, California. To read past columns, [click here](#).

Xscindo

No Data, No Landfill, No Waste

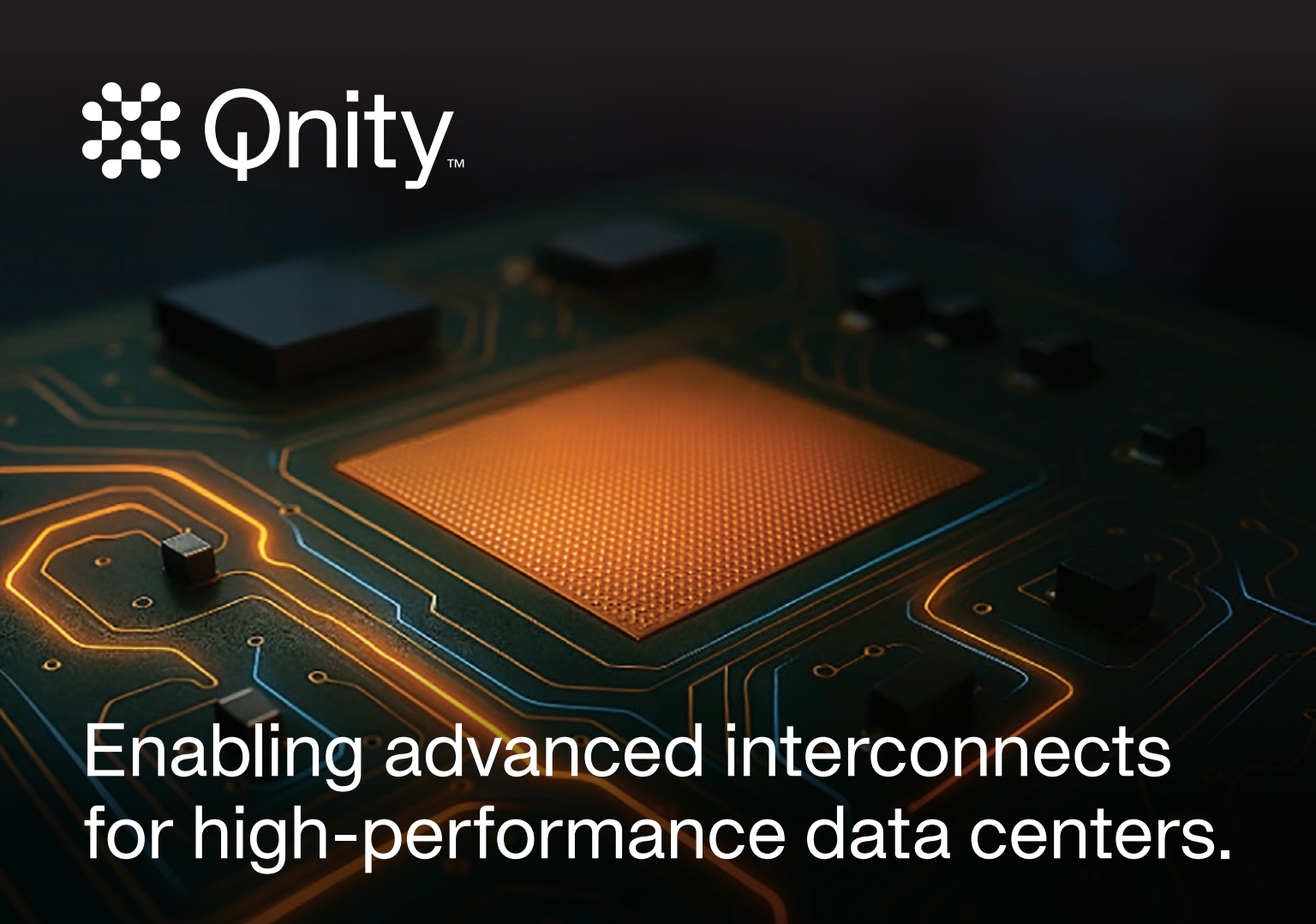


Xscindo is a start-up recycling company developing an alternative approach to conventional e-waste processing, focusing on transforming discarded electronics into clean, recoverable critical and precious metals rather than losing them to traditional disposal and recycling streams. Xscindo's work sits at the intersection of circularity, resource security, and a rapidly expanding need for critical materials, and speaks to a much larger challenge facing the electronics industry:

finding better ways to keep valuable resources in circulation rather than exporting them. In this interview, Aiden Neary, founder and CEO, tells us more about this important work.

Marcy LaRont: Aiden, it is great to meet you. You've had an interesting career, so please introduce yourself and Xscindo. How did Xscindo begin?

Aiden Neary: My background is investment banking, not data security or retired electronics. I was



Enabling advanced interconnects
for high-performance data centers.

Qnity powering AI at scale.

AI and hyperscale data centers are redefining PCB requirements—demanding higher speeds, greater density, and uncompromising reliability. Qnity enables **diversified PCB designs**, from flex and rigid-flex architectures to high-density interconnects.

Kapton® polyimide films and Pyralux® flex laminates circuits from Qnity are engineered for superior signal integrity, thermal resilience, and design flexibility—enabling the performance and durability needed to scale AI and data center innovation.

Scan to learn
more about
Kapton®



Scan to learn
more about
Pyralux®



hacked some years ago, which had made me conscious of data security. So several years ago, I began looking much more closely at what was happening in the electronics and data-security markets, and it became apparent to me that a gap was developing.

The amount of sensitive information we carry on electronic devices was increasing enormously, while at the same time, bad actors and data-recovery techniques were becoming more sophisticated. Yet the way we dealt with those devices at the end of their lives had not fundamentally kept pace.

That led me to look beyond data security and ask a much broader question: What actually happens to an electronic device at the end of its life?

Once you follow that journey, you quickly encounter not just data security, but landfill, plastics and PFAS, overseas exports, critical-material recovery, and a general lack of transparency. Xscindo grew out of trying to address those issues together rather than treating each one as a separate problem.

What does Xscindo mean?

It is derived from the Latin word for “destroy.” The principle behind Xscindo is fairly simple: If an electronic device has reached the end of its useful life, we want to eliminate the liabilities associated with it, particularly the data and non-metal components, while recovering the metals and returning them to the domestic circular economy rather than allowing the device to end up in landfill.

How does Xscindo’s focus on e-waste and materials recovery play into data security?

They’re much more closely connected than they might initially appear. An end-of-life electronic device has both assets and liabilities. The metals are the assets, so we want to recover those. The liabilities include sensitive data, plastics, PFAS, and the environmental consequences of improper disposal.

What interested me was whether you could design an end-of-life process around addressing both sides



*Aiden
Neary*

of that equation simultaneously. Instead of asking only, “How much metal can we recover?” we started asking, “Can we absolutely destroy the data, address the environmental liabilities, and still preserve the metals for domestic reuse?”

That became the basis of our approach.

The world is in a very different place than even three years ago about this issue. In the area of product circularity, what has been done well, in your opinion?

Awareness has changed dramatically. A growing number of states restrict electronic waste from landfills, manufacturers have developed take-back programs, and consumers understand that electronics shouldn’t simply be thrown away.

Where I think the next evolution needs to occur is what we mean by recycling. Collecting a device is only the beginning. We need to understand where it ultimately goes, what happens to the data, the plastics, and other non-metal components, and whether the metals genuinely return to the circular economy.

In PCB manufacturing, metals recovery is part of the business model. There is a direct financial benefit for a company to do it. It’s true that we don’t want heavy metals going into the environment. It is regulated, so that tried-and-true business initiative happens to also have environmental benefits, to your point.

Absolutely. Metals recovery is an important part of the equation, both economically and environmentally. The broader challenge is that only a relatively small proportion of global e-waste currently enters documented recycling streams, and even the word “recycling” can mean very different things depending on what ultimately happens to the material.

For me, the important question is not simply whether something was sent for recycling, but what actually happened to it afterward. Were the metals recovered and returned to the circular economy? What happened to the plastics and other non-metal components? Was the data securely destroyed, and

Innovation

+ Precision

+ Reliability



In-house lab available for customer testing

Wet processing equipment for all of your circuit board manufacturing needs, such as:

- Cleaning
- Developing
- Etching
- Stripping
- Conveyorized Plating

To learn more about our lab, contact
Chemcut at sales@chemcut.net



CHEMCUT

BOUNDLESS INNOVATION | UNBEATABLE PRECISION

can the original owner verify those outcomes?

That distinction between recycling as an intention and recycling as a verifiable outcome is really where Xscindo came from.

What did you learn in your market research?

One of the biggest things we discovered was that visibility often diminishes very quickly once a device leaves the original owner's control. An organization may quite legitimately say, "We send our electronics for recycling," but when you ask what happens two, three, or four steps downstream, the answer often becomes much less clear.

A device may pass through a data-destruction company, processor, broker, refiner, or other intermediary. The original owner can therefore find it extremely difficult to establish the final destination of every component.

That led us to ask organizations what they would change about the existing system if they could.

How did companies respond?

When we spoke to organizations about what they would change if they could, four themes came up consistently.

The first was data security: How can I verify that the data has actually been destroyed?

The second was the environmental impact of the non-metal components, particularly plastics and forever chemicals. The third was metals: Where do they ultimately go, and can we demonstrate that they have genuinely been returned to the circular economy?

The fourth really encompassed all three: transparency. Organizations wanted a better chain of custody and greater visibility into what actually happened to their equipment after it left their possession.

That is a big leap of faith to send out your e-waste and believe it's being disposed of properly. The lack of transparency around the current system is disturbing.

That's really the issue: trust without verification. Organizations aren't only responsible for their own sensitive information; in many cases, they're entrusted with customer, employee, and client data as

well. In the U.S. in 2025, 3,300 data compromises were reported; many more likely went unreported. Many large organizations are regularly being fined for data breaches, so data protection is a key concern for most, if not all, organizations.

Once a device leaves their control, they need confidence not only that the data has been irreversibly destroyed, but also that the device itself has been handled as they were told it would be. That's why I think transparency and verifiable outcomes are becoming increasingly important. It's no longer enough to say something was recycled; organizations increasingly want to know what actually happened to it.

After years of education on recycling in the U.S., we now have the realization that much of it has ended up in landfills, particularly overseas or in the ocean. Recently, the U.S. seem to have halted some its environmentally proactive measures. That is not the case in Europe. What's happening there? Europe and the UK have generally moved further toward requiring organizations to understand and report the environmental impact of their activities and supply chains, but the U.S. is moving in that direction as well.

California's SB 253, for example, is part of a broader movement toward greater emissions and supply-chain accountability. This requires companies with over \$1 billion in annual revenue to provide Scope 1 (emissions caused by the company directly), Scope 2 (additional emissions associated with purchased energy) carbon footprint reporting this year, and Scope 3 (indirect carbon emissions through a company's value chain) reporting in 2027.

What I find particularly relevant is the increasing focus on Scope 3, the indirect impact occurring throughout an organization's value chain. That changes the question from "Did we send it for recycling?" to "Can we demonstrate what actually happened to it?"

That's an important distinction. Whenever we create targets, there is a risk of focusing so heavily on the measurement that we lose sight of the outcome we're actually trying to achieve. History is full of examples of well-intentioned policies producing unintended consequences. A relevant

Electronics U

by Global Electronics Association

Train Smarter.

Build a Future-Ready Engineering Team.

All-in-one training, certifications, and expert-led learning with Engineering Pro.

ENGINEERING PRO SUBSCRIPTION

COURSES | INDUSTRY EXPERTS

- Train engineers faster with expert-led, real-world electronics courses
- Keep your team aligned with evolving standards and technologies
- Standardize training across teams and locations

120+ HOURS OF ROLE-BASED TRAINING

Essentials for New Engineers, Manufacturing Engineering, Quality & Acceptability Standards, and More!

**UP TO
75% off**



Start Training Your Teams

current example is many company policies on the proper disposal of laptop computers, where the success of the policy is measured by the number of computers collected and not by their actual recycling or proper disposal, which, as it turns out, typically does not occur.

The peril of unintended consequences!

Exactly. There's an economic principle known as Goodhart's Law: When a measure becomes a target, people naturally begin optimizing for the measure rather than the underlying outcome.

I think that's very relevant to electronics recycling. The next evolution should be moving from measuring what was collected or what was designated for recycling toward verifying what actually happened to it.

For us, the important question isn't simply whether a laptop was recorded as recycled. It's whether the data was irreversibly destroyed, whether the material avoided landfill and unnecessary export, what happened to the non-metal components, and whether the recovered metals genuinely returned to the circular economy.

Are there some organizations doing valuable work in this space?

Organizations such as the Basel Action Network (BAN) have done important investigative work in highlighting what can happen to electronic waste once it enters complex downstream recycling channels.

Their Brokers of Shame investigation, for example, used electronic tracking devices to follow material through the recycling chain and documented significant quantities of U.S. electronic waste being exported overseas.

What I find particularly important about work like that is that the original owner may have acted entirely responsibly. They handed their equipment to someone believing it would be properly recycled. The problem is that once material passes through multiple intermediaries, visibility can quickly disappear.

That brings us back to the distinction we discussed earlier between intention and outcome. An organization can have every intention of recycling responsibly and still have very little ability to verify the ultimate outcome.

Have these revelations affected trust with the average citizen who wants to dispose of e-waste properly?

They inevitably undermine confidence. Most people want to do the right thing; the problem is knowing whether what they were promised actually happened.

I think the way you rebuild that confidence is through transparency, traceability, and verifiable outcomes. People shouldn't simply have to take our word for it. They should be able to understand what happened to their device and where the resulting materials ultimately went.

What might that look like?

That's really what we've tried to build at Xscindo.

We operate a closed-loop process designed around a zero-landfill objective. The electronics we receive are processed at our facility, data-bearing components are irreversibly destroyed, and the non-metal fraction is thermally treated. The recovered metals are then directed into domestic refining and the U.S. circular economy. Everything else is basically disintegrated into a non-harmful gas—completely gone.

Just as importantly, the process is documented. We want customers to have a clear chain of custody and evidence of the final outcome: proof rather than promise.

Who does Xscindo partner with?

We work with companies and institutions that need a secure and responsible solution for their end-of-life electronics. We also operate a community outreach program, through which we allocate a portion of our processing capacity to community e-waste recovery.

What's interesting is that we are increasingly working across the entire lifecycle, from organizations looking for secure disposition of their equipment to landfill operators trying to prevent these materials from being buried in the first place.

I am glad to hear Xscindo is actively working with landfills.

Electronic waste accounts for 2% of landfill mass, yet it's responsible for over 70% of landfill toxicity.

So, landfill avoidance has become an important part of what we do.

Electronic waste presents particular challenges for landfill operators. Lithium-ion batteries can create significant fire risks, while electronics also contain plastics, persistent chemicals and metals that simply don't belong buried in the ground if there is a viable alternative.

So rather than dealing with those risks after the material has been buried, we're working with landfill operators to intercept the electronics before they enter the landfill and provide another pathway for them.

How much e-waste do you estimate ends up in landfills?

The exact numbers vary depending on how electronic waste is defined and measured, but we know that significant quantities still fail to enter appropriate recycling streams.

For us, the more important question is what we can practically do about it. We can work directly with landfill operators and communities to divert those electronics before burial, process them securely, address the non-metal components, and return the recovered metals to the domestic circular economy.

That changes something that would otherwise have been a landfill liability into a new domestic source of materials.

Xscindo is still a young company, but what are you most proud of accomplishing?

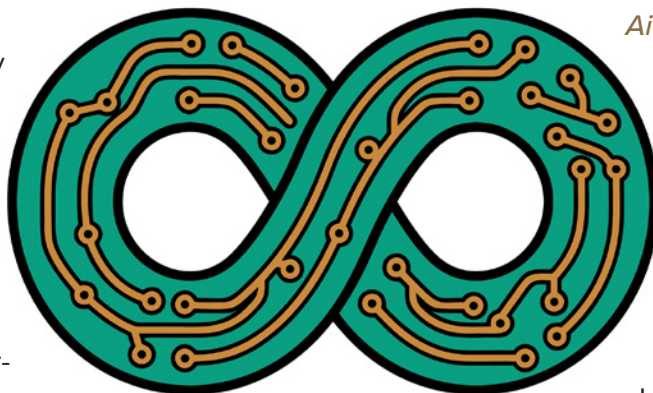
Probably seeing material that would otherwise have been buried actually being diverted from landfill and returned to productive use.

But I think there's a larger opportunity emerging as well. The United States, and in fact most countries, needs secure and resilient sources of critical and strategic materials. At the same time, enormous quantities of those materials already exist inside electronics that have reached the end of

their useful lives.

If we can securely release those devices, eliminate the data and environmental liabilities associated with them, and return their metals to the domestic supply chain, we're effectively opening an additional source of critical and precious materials from resources that already exist here.

I'm also particularly proud of the trust we're beginning to build. Organizations entrust us with devices containing extremely sensitive information. That carries a significant responsibility, and our philosophy is that trust should be supported by transparency and evidence rather than simply asking customers to believe us.



Aiden, can people visit your facility to see what you do?

Absolutely. We actively encourage it. We've welcomed customers, government officials, landfill operators, and other stakeholders to our Wyoming facility to see the process firsthand.

I think your own eyes are often the best form of transparency. If someone is entrusting us with sensitive electronics and their data, they should be able to see exactly what we're doing with them.

More broadly, I think the market itself is changing. Awareness of data security, landfill risk, PFAS, overseas exports and critical-material dependency is increasing rapidly. As those risks become better understood, I expect customers, regulators, and the recycling industry will demand more secure, transparent, and verifiable end-of-life solutions.

We're positioning Xscindo for where we believe that market is going.

It has been so interesting to learn about Xscindo. It gives me hope that we might be able to get this e-waste thing under control. Thank you.

Thank you, Marcy. I really appreciate the opportunity to tell our story and, more importantly, to bring greater attention to an issue I think we're only beginning to fully understand. **I-CONNECT007**



What Engineers Get Wrong About Designing for *Reliability*

BY BRIAN BUYEA, REMTEC

One question I hear more than almost any other is: How do I design electronics for high reliability? It's a fair question. In industries like aerospace, defense, EV, and industrial power, reliability isn't a feature; it's the product. When things fail, they don't just fail quietly. They fail catastrophically, expensively, and sometimes dangerously.

The truth is that most reliability problems are not

discovered in testing. They are designed in from the very beginning, and that's where engineers—good engineers—get it wrong.

The Biggest Misconception About Reliability

The most common mistake is thinking reliability is something you validate at the end. It's not. Reliability is not a test plan or a qualification cycle. It's not

DIRECT METALLIZATION

The Next Evolution in PCB Technology

As advanced PCB and IC substrate technologies continue to evolve, direct metallization is emerging as a critical process for improving reliability, enabling higher-density interconnects, and supporting sustainability goals.

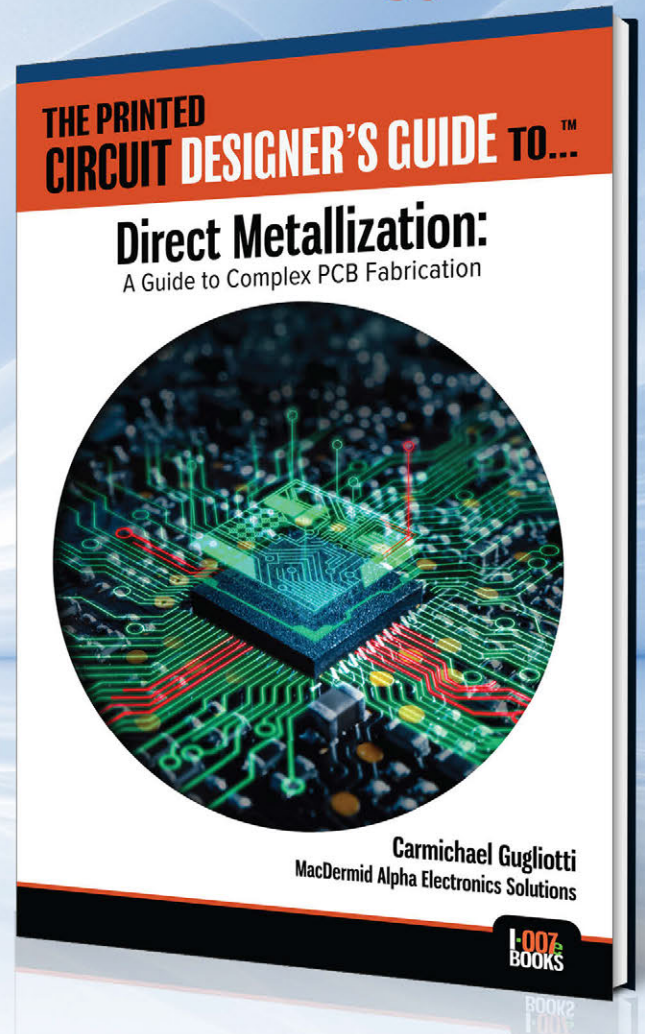
Featuring insights relevant to PCB designers, fabricators, OEMs, and process engineers, this book provides the technical foundation needed to understand, evaluate, and implement direct metallization technologies in next-generation manufacturing environments.

"An essential resource for fabricators and OEMs."

—Venkat Raja, Advint, Inc.

"A clear and practical perspective on direct metallization."

—Jerry Magera, RTX



"This book provides the definitive view on the use of direct metallization for advanced printed circuit boards and IC substrates."

—Mike Carano, Global Electronics Association



AVAILABLE NOW AT I-007EBOOKS.COM



I-007eBOOKS

PUBLISHED BY I-CONNECT007



High-reliability design begins by asking a simple question: What environment must this survive—not just initially, but over time? Answer that honestly, and your material decision becomes clear.”

something you “check” once the design is complete. Reliability is a design discipline.

Too often, teams move fast to get a design functional, assuming they can “harden it later.” But by the time you reach testing, the fundamental decisions—materials, structure, thermal paths—are already locked in. At that point, you’re not designing for reliability. You’re reacting to its absence.

Material Choice Is the First Decision, Not the Last

Let’s get very practical: Reliability starts with materials. Not layout, routing, or even architecture. Yet material selection is often treated as a secondary decision constrained by cost targets or legacy preferences.

Take substrate choice. Engineers default to familiar materials because they’ve “always worked.” But what worked at lower power densities, lower temperatures, and slower switching speeds doesn’t always hold up in today’s applications.

If your material can’t handle the thermal load, mechanical stress, or environmental exposure, no amount of clever design will save you. High-reliability design begins by asking a simple question: What environment must this survive—not just initially, but over time? Answer that honestly, and your material decision becomes clear.

The Hidden Risk of Thermal Expansion Mismatch

One of the most overlooked failure drivers is the CTE mismatch. Different materials expand and contract at different rates as temperatures change. When those materials are bonded together, as they always are in electronics, you create stress. Over time, that stress leads to fatigue, cracking, delamination, and eventual failure.

Here’s the problem: These failures don’t show up immediately. They show up after hundreds or thousands of thermal cycles, long after the product is in the field.

Engineers often focus on peak temperature. But reliability is rarely about peak conditions. It’s about repeated cycling. If your substrate, die, and interconnect materials aren’t aligned in their expansion behavior, you’re building in a slow failure mechanism, and you won’t see it until it’s too late.

Mechanical Stress: The Silent Killer in High-Power Systems

Electrical engineers love to think in terms of signals, currents, and voltages, but in high-power electronics, mechanical forces, such as vibration, shock, mounting stress, and assembly-induced strain, are just as important. These forces don’t care how elegant your circuit design is.

In power modules, for example, thermal cycling creates expansion and contraction. That movement introduces mechanical fatigue into solder joints, wire bonds, and interfaces.

If your design doesn’t account for that movement—if it constrains it instead of managing it—you create stress concentrations, and that leads to failure.

Reliability means designing not just for electrical performance, but for physical durability.

The Role of Substrate Technology in Long-term Performance

This is where substrate technology becomes critical because the substrate is not just a platform. It is the foundation of your system’s reliability. It dictates thermal performance, mechanical stability, and how well your design handles environmental stress.

Traditional PCB materials can work well in many

applications. But in high-power, high-temperature, or high-reliability environments, they often reach their limits. Advanced substrate technologies, particularly ceramic-based systems, offer significant advantages:

- Superior thermal conductivity
- Better dimensional stability across temperature ranges
- Improved resistance to environmental stress

But the key is not simply choosing a “better” material. It’s choosing the right material for the application. Reliability comes from alignment between material properties and real-world conditions.

Design for Manufacturability (DFM) vs. Design for Reliability (DFR)

Every engineer knows DFM. It’s important and ensures your product can be built efficiently and consistently. But the issue is that DFM is not the same as design for reliability (DFR). In fact, they can sometimes be in tension.

A design that is easy to manufacture may not be optimized for long-term durability. Conversely, a design that maximizes reliability may require tighter controls, more advanced processes, or different materials. Too often, teams prioritize manufacturability because it’s immediately visible in yield, cost, and throughput.

Reliability, on the other hand, is deferred. It shows up later in the field. The best designs balance both, but they never sacrifice reliability for short-term manufacturing convenience because field failures are always more expensive than factory challenges.

Early Collaboration Is a Competitive Edge

One of the simplest and most underutilized ways to improve reliability is to involve your manufacturing partner early. Manufacturers bring a different perspective. They see patterns in designs, materials, and failure modes that individual engineering teams may not.

They understand how materials behave in real production environments and know where processes introduce variability. They’ve seen what

works—and what doesn’t—over time.

Early collaboration turns that knowledge into a design advantage, because instead of discovering problems in testing, you prevent them in design. That’s not just better engineering. That’s faster time to market, lower cost, and higher confidence.

Reducing Field Failures Before They Happen

Ultimately, DFR is about eliminating surprises. You can’t control every variable in the field, but you can control how your design responds to them.

That means anticipating thermal cycles, not just peak temperatures, managing mechanical stress, not just electrical performance, selecting materials based on lifetime behavior, not initial cost, designing interfaces that tolerate movement, not resist it, and most importantly, it means thinking beyond the first success.

A prototype that works is not a reliable product, and a product that survives testing is not necessarily a durable one. True reliability is proven over time, but it is created at the start.

Final Thought: Reliability Is a Choice

Reliability is not an accident. It’s not a byproduct of good engineering, and it’s certainly not something you can bolt on at the end. It is a series of deliberate decisions about materials, design, collaboration, and priorities.

Engineers don’t fail because they lack skill, but because they underestimate how early reliability decisions must be made. If you want to design for high reliability, start sooner. Choose materials with intent, design for the real world, not just the schematic. Collaborate early, and never assume you can fix it later.

In high-reliability electronics, later is usually too late, and the future you’re powering depends on getting it right the first time. **I-CONNECT007**



Brian Buyea is president of Remtec Inc. To read past columns, [click here](#).



SCHMID Pushes Toward the Submicron

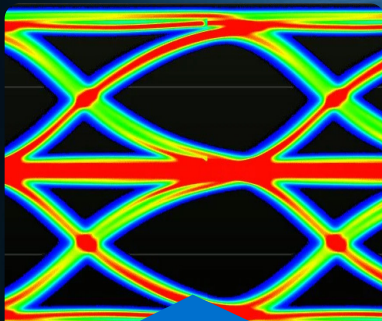
Fabricators are confronting process requirements today that would have seemed foreign to the PCB shop not long ago, presenting both a promise and a significant manufacturing challenge. SCHMID is one company rising to it in big ways. Laurent Nicolet, VP for the Business Unit PCB, talks

about the convergence between PCB fabrication and semiconductor processing as feature sizes shrink, the emergence of glass as an attractive substrate and interposer material, and SCHMID'S embedded trace technology and the work toward moving that technology into high-volume manufacturing.

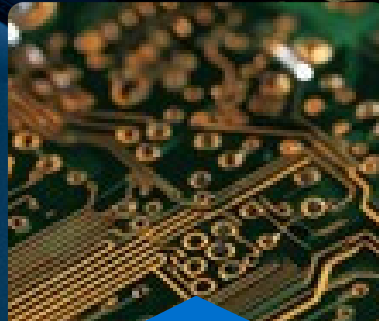
Introducing EM-896K2 and EM-896K3

Halogen-free, supreme low-loss materials
developed for AI servers and 1.6T switching networks

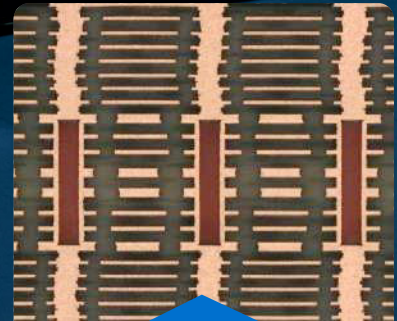
- Excellent Dk/Df performance for high-speed signal integrity (< 1.0 dB/inch @ 53GHz)
- Low X/Y-axis CTE for improved dimensional stability includes CoWoP applications
- Proven multi-lamination reliability (≥ 12 x laminations)



Excellent Dk/Df
performance



Ultra-low
X/Y-axis CTE



Proven multi-
lamination reliability

Marcy LaRont: Laurent, tell me about the work SCHMID is doing in advanced manufacturing. What are you working on?

Laurent Nicolet: There are two significant things that are worth talking about. One is our embedded trace process for PCBs, though we are still quite early in the game in terms of industry readiness for adoption.

The other has to do with the tremendous amount of interest we are seeing in redistribution layers (RDL), the microscopic network of fine copper wiring built directly onto a glass substrate or glass core, and photo-imageable polyimide (PID) material for panel leveling, which is easier to proceed through a lithography process than ABF, although it has some material issues compared to ABF.

Our InfinityLine C+ is a vertical rotating wet-processing cluster system designed specifically for high density panel-level packaging (PLP), in that it treats both sides of HDI PCB or substrate simultaneously. We are going close to the submicron level: 0.8-micron vias and 1-micron line-and-space. That is also quite a significant development for the industry right now.

What are some of the technical nuances specific to panel-level packaging and working with glass substrates?

When working with these glass-core substrates, you are ultimately looking at a thickness of 0.8 to 1.10 millimeters. Because it is glass, the handling is more delicate. For PLP you have material thicknesses up to 6 millimeters, including carrier systems, and weights up to 4 or 5 kilograms, with warpage up to 4–6 millimeters. Because of these intense physical dimensions, handling the material safely through the machine becomes incredibly critical.

The working environment for PLP is approaching semiconductor-grade levels. These process technologies are beginning to show similarities.

This also means we are now playing in an ISO Class 5 clean environment. The filtration of the solu-



tions, liquid, in use in these tools needs to have filtration down to 0.05 μm . Our machines will be modified to meet ISO Class 5 standards.

We have also created a new product called QuantumLine, a manufacturing equipment platform designed specifically for producing next-generation glass-core substrates for advanced semiconductor packaging. It is built to run in a Class 5 cleanroom and is aimed at manufacturing quantum-level computing.

Where are you primarily seeing this activity and interest?

There is significant demand in Asia. We recently delivered lines for panel-level packaging: one InfinityLine C+ rotating with six chambers that can handle five different processes, one InfinityLine C+ with four chambers handling four different processes, and two InfinityLine H+ horizontal lines for stripping and cleaning processes.

We just delivered nine lines in Texas for panel-level packaging, along with customized features for what this company wants to do, but they are not yet at cleanroom level 5, only ISO 6. By contrast, the projects we have in Asia are for cleanroom environments and are all for panel processing, which requires high levels of cleaning at every process step.

We have also been working with the chemistry suppliers for the InfinityLine C+ to adapt the chemistry to this type of tool.

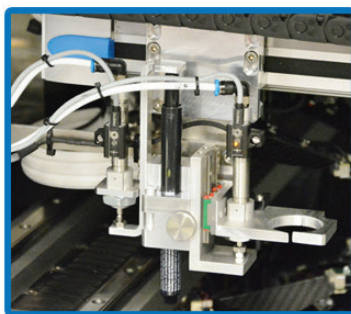
As you said, worlds are coming together as technology is now pushing us toward semiconductor manufacturing processes. Is this where glass comes in?

Yes, panel processing and panel leveling are an important focus point right now. The next most important area is the glass substrate. We are investing in developing tools for processing glass substrates and glass interposers. We are also seeing a development in the mindset around it. At first, people thought it would be quite easy to laser drill

Unrivalled Test Speed with Full Automation



Introducing the newly designed atg A9a with 8 test probes and a new high speed “lights out” automation for unrivalled throughput.



Highlights:

- Small footprint (6 square meters)
- Dual shuttle pick & place automation
- High accuracy combined with high test speed
- Pen or label marking option

Watch video

Get more info

atg Luther & Maelzer GmbH

Zum Schlag 3 • 97877 Wertheim • Germany
Phone +49-9342-291-0 • Fax +49-9342-395 10
sales@atg-lm.com • www.atg-lm.com





Figure 1: Cross-sections of through-glass vias (TGV) filled with bottom-top technology with different densities that show no voids.

the glass, but due to microbreakage, which typically doesn't appear until after the holes are filled, that is not the case. Germany and Korea have strong players in this field.

The cleanliness of the glass prior to lasering is also a significant challenge because every particle on the glass can affect the laser process, creating altered or poor hole shapes that can compromise the etching process and make it difficult to meet hole tolerances.

Is the cleaning process for glass similar to other such processes?

There are similarities. We are currently investigating the particles on the surface of the glass and determining which particles come from the glass's production. We are working with glass suppliers who are well-renowned in materials science and glass technology.

Most people etch glass using an alkaline-based immersion process, which involves dipping. It typically takes 10 hours or more, which is a very long time. At SCHMID, we prefer to etch with hydrofluoric acid (HF) solution. So, we have, as an option, a first stage with alkaline etching on a horizontal machine, followed by a second stage with HF, which etches faster.

But in a dipping process, which uses a basket, it's easy to get non-uniform etching because the solution cannot flow well through all the holes and the surface. Another problem is creating micro-cracks simply from panel placement in the basket.

In our horizontal etching lines, we have a flooding

system for better uniformity when processing full panels. This produces greater uniformity during the etching of the glass.

Given that microcracks in glass are a critical issue, how are you gauging the quality of glass?

Some companies have a system that allows you to detect micro-cracks on the surface at the entrance of the holes. But detecting this type of microbreakage is difficult. Microcracks are usually an issue after thermal processing, which is primarily when they are detected.

The real challenge is the metallization of the through-glass vias (TGVs). There are two ways to fill a glass via. One is with a seed layer, which can be deposited by sputtering or by a metal oxide. Sputtering can have an aspect ratio of no more than 1:6.

For metallization with a metal oxide, the ratio can go up to 1:20, but beyond that, filling the holes will be very difficult. In our work, we see the optimum process at an aspect ratio of 1:6 to 1:8. You get voids beyond that. Given the difficulty of filling the TGV, some companies are now providing an acceptable tolerance for voids in glass vias.

You mentioned there is a lengthy processing time for etching glass. What does the plating process look like for glass?

Plating time with conformal plating, bridging, and filling can be lengthy as well. Some say four to six hours, but it's more like 12 to 20 hours of total plating time. We have the "bottom/top plating," meaning we apply special copper layers on one side and

Visit Ventec at PCB West
Booth 209 - 30 September 2026



ventec
INTERNATIONAL GROUP

pro-bond & thermal-bond

Glass-Free Materials: Next-Generation Performance.

Ventec's pro-bond series marks a significant advancement in dielectric material technology, offering high-performance, glass-free laminates engineered for the most demanding electronic and thermal management applications.

By eliminating woven glass reinforcement, pro-bond eliminates fiber weave effects, delivering superior signal integrity, uniformity, and reliability. This innovation is essential for high-speed and high-frequency designs in aerospace, automotive, and next-generation communication systems.

By eliminating woven glass in thermal-bond, the thermal insulating effect of glass is removed from the dielectric, optimizing thermal performance and heat dissipation in MiB and embedded component multi-layer build up structures and component packaging applications.

Wherever **technology** takes you, Ventec delivers.

www.ventec laminates.com

**JOIN VENTEC'S
GLASS FREE
REVOLUTION**



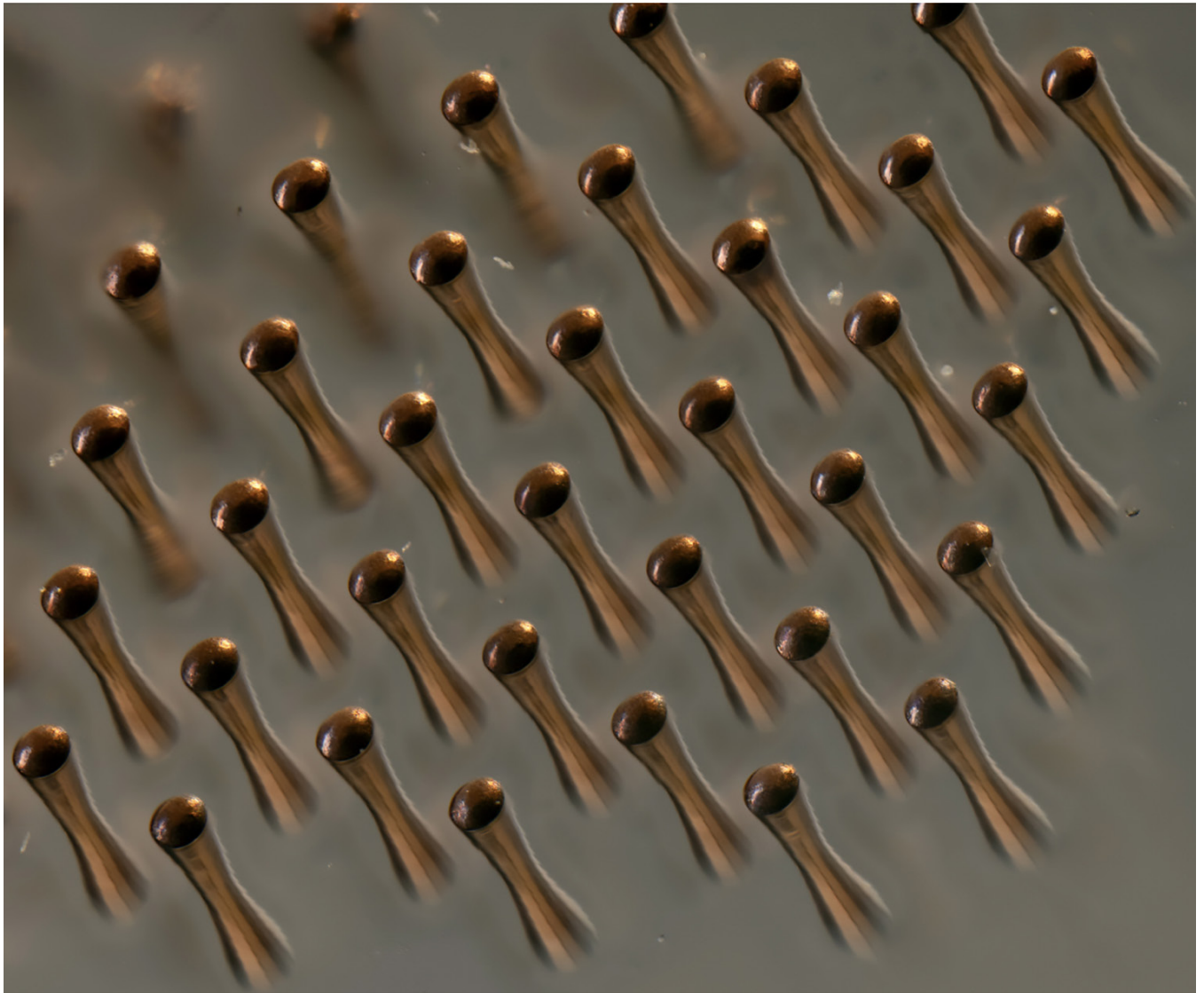


Figure 2: How the surface looks after the plating process and before chemical mechanical planarization (CMP).

fill the holes without a seed layer. Then we planarize the bottom and the top, and it's finished. We are currently running panels for a customer in China, but for our lab on a scale of a dozen panels. Scaling is always a great challenge, but we will get there soon.

Where is SCHMID in the development of this advanced manufacturing equipment with customers?

We are in the final stages of developing these new tools, and we continue to work on making them fully automated. Our lab is full of samples, some of which are 100% void-free in the glass vias. It is very promising.

What other partnerships are involved?

We are experimenting with some companies on plating chemistries to reduce the required plating time, currently estimated at 24–30 hours for holes with a 1:50 aspect ratio.

Wow, 1:50 aspect ratio is difficult to imagine.

Now we are working to scale this bottom-top plater. The design is defined, and the first equipment will be delivered during the second quarter of 2027.

When it comes to materials, and other than glass, are there other alternatives to ABF that you are aware of?

Yes, other players are coming into that market



Some of these boards have four million microvias per layer, so you can imagine the time required in laser drilling. This can be processed much more quickly with plasma etching.”

with whom we have tried to collaborate on testing their material, but it's clear that ABF is still the primary player.

Can you explain how your embedded trace process goes hand-in-hand with the demand for substrates on glass?

Our embedded trace process meets the need for very dense circuits and novel advanced packaging solutions. We have several customers that want to test with us. The challenge is that our lab's exposure system is limited to a six-micron line-and-space, we do not have AOI, and we have a small photoresist laminator. We are not a PCB or substrate manufacturer, so some of this testing cannot be done solely in our lab, and you can't have the panels go back and forth with customers. It's not good for a technology that requires a highly controlled, clean environment.

But we are in active testing with a Japanese customer. Some of these boards have four million microvias per layer, so you can imagine the time required in laser drilling. This can be processed much more quickly with plasma etching. The market is not quite ready yet, but we have achieved good performance with different materials. We currently have three different customer projects in process.

In your opinion, what does the global demand look like?

Global demand is currently focused on PCBs for AI applications, mainly AI server boards. This type of board requires inner layers and some build-up layers, which demand an mSAP process. A few years ago, there was a lot of discussion about mSAP in the U.S. But other than Calumet and GreenSource, nothing has really happened in the U.S., which was surprising.

Those companies, along with TTM, received government assistance to gain private investment, which helped add such processes to their fabs. What about for smaller fabricators?

They have not been able to get the assistance, and that's a problem. It's too bad, because the engineering expertise is still huge in the U.S. and, right now, still relevant. It is unfortunate that investment has not kept up with technology. It is the same in Europe, where, other than AT&S, the investment tends to be in the traditional tools and standard technologies. In Asia, the main investments are focused on high tech. There is high demand in Asia. An important customer in China just purchased 55 lines for high-volume manufacturing, for example. Theoretically, we should be seeing this in the U.S. as well, given the growing need for AI server board capacity. We will continue to watch that market with interest.

Laurent, thank you.

Great to talk with you, Marcy. I-CONNECT007

Don't Build Our AI Economy on Foreign-sourced Components



BY SHANE WHITESIDE, PCBAA

Hyperscale data centers are now all over the U.S., from large cities to small towns, and building out the burgeoning AI economy is on the minds of policymakers and legislators at all levels of government. According to Forbes, there are approximately 580 operational hyperscale data centers in the U.S., out of roughly 1,360 worldwide.

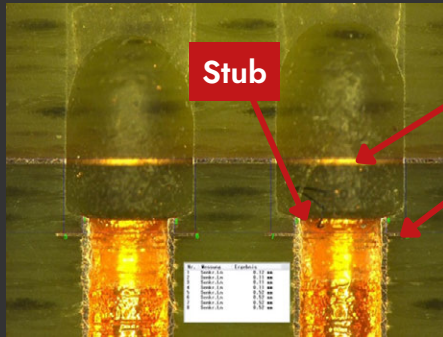
These types of AI data centers have been a boon for electrical and construction workers and public utilities. Unfortunately, despite being full of elec-

tronics, the American microelectronics industry has not benefited from the massive investments made by the world's largest companies. Instead, the servers for U.S. data centers are sourced where they are most easily obtained. The irony is that most hyperscale data center developers design these servers in America but source them from China or Taiwan.

Throughout the debate about where to put data centers and how many will be needed, it's clear that an American company building a new data

EAGLE EXY

HIGH-END EQUIPMENT FOR **BACKDRILLING**
AND **DEPTH DRILLING** APPLICATIONS

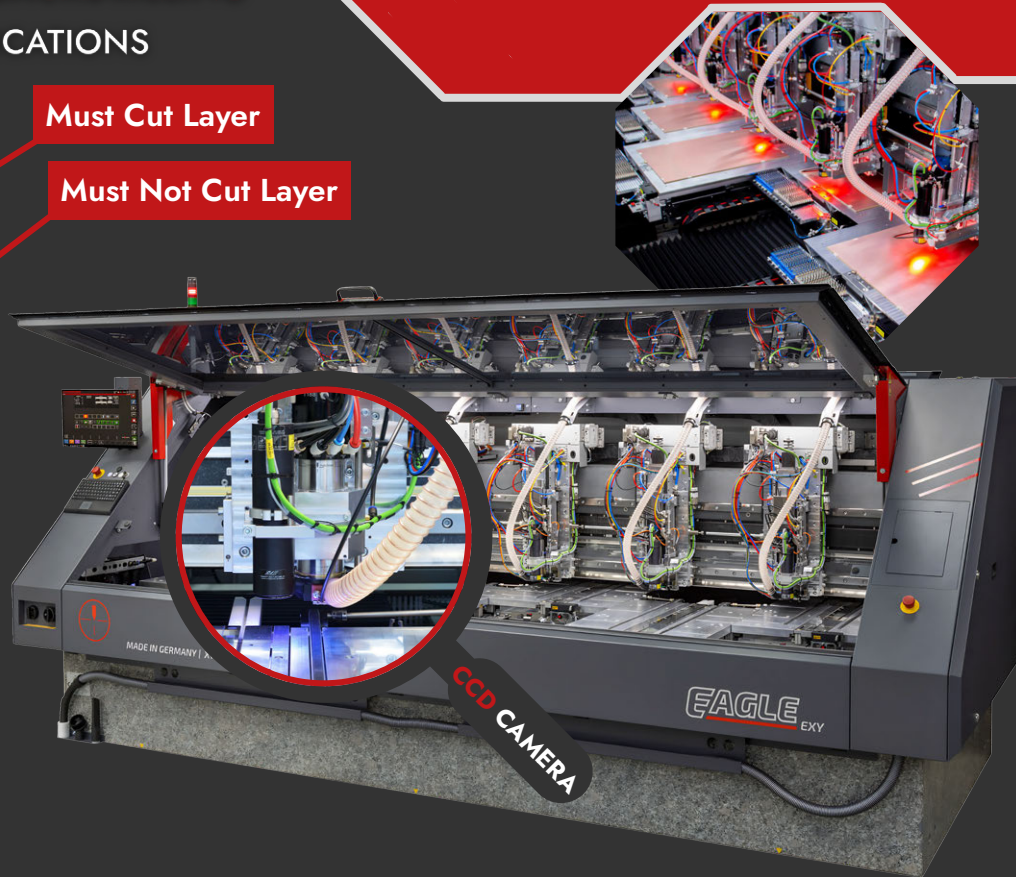


Back Drilling: stop of depth drilled hole before must-not cut layer within stub length tolerance

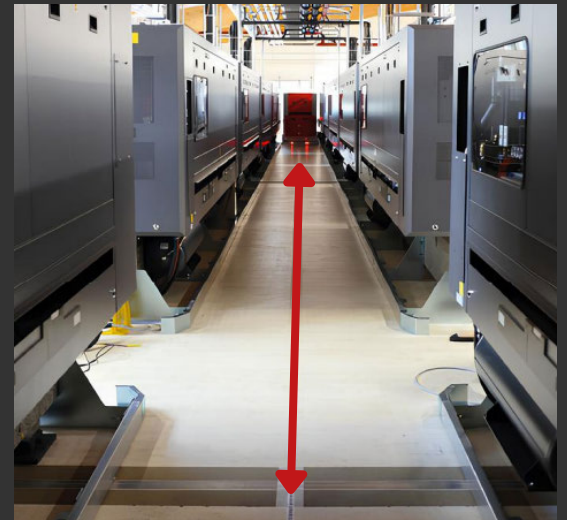


Depth Drilling: precise and reliable drilling to defined inner layers, enabling high-quality blind vias.

AUTOMATED COLLECT CLEANING (ACC)



**AUTO LINE SHUTTLE (ALS)
AUTOMATION**



center in the U.S. and just painting an American flag on the building cannot disguise the fact that the tens of thousands of semiconductors, PCBs, and servers needed to operate them are from Asia, and predominantly from China.

One popular solution is to remove China from supply chains through a China Plus One approach, but that is not necessarily the answer. Behind many of the new PCB operations in friendly Asian countries, including Thailand, Vietnam, and Malaysia, Chinese-backed enterprises are often the investors. This moves the risk geographically but does nothing to reduce the potential negative consequences of Chinese influence.

While the U.S. debates policy, the rest of the world is deploying capital at unprecedented levels. But if the U.S. continues to run behind, Taipei, not Topeka, and Phuket, not Philadelphia, will benefit from the rapid rise of AI and the data centers that power them.

Policymakers in Washington should push for reshoring and the restoration of the PCB industry. When people doubt our ability to bring back American microelectronics manufacturing, I remind

them it wasn't that long ago that America was dependent on foreign sources of energy.

Because of innovation and smart industrial policy, we became a net exporter of energy and the engineering prowess to build out infrastructure.

With the right industrial policy and government incentives, we can change our dependency on microelectronics made by our largest global competitor.

The Printed Circuit Board Association of America is working with Congress and government agencies to provide the policy and legislative support that will ensure the American flag on data centers truly represents what's inside. **I-CONNECT007**



Shane Whiteside is chair of the Printed Circuit Board Association of America and president and CEO of Summit Interconnect. To read past columns, [click here](#).

Educational Resource Center

Podcasts 

Books 

Roundtables 

White Papers 

Webinars 

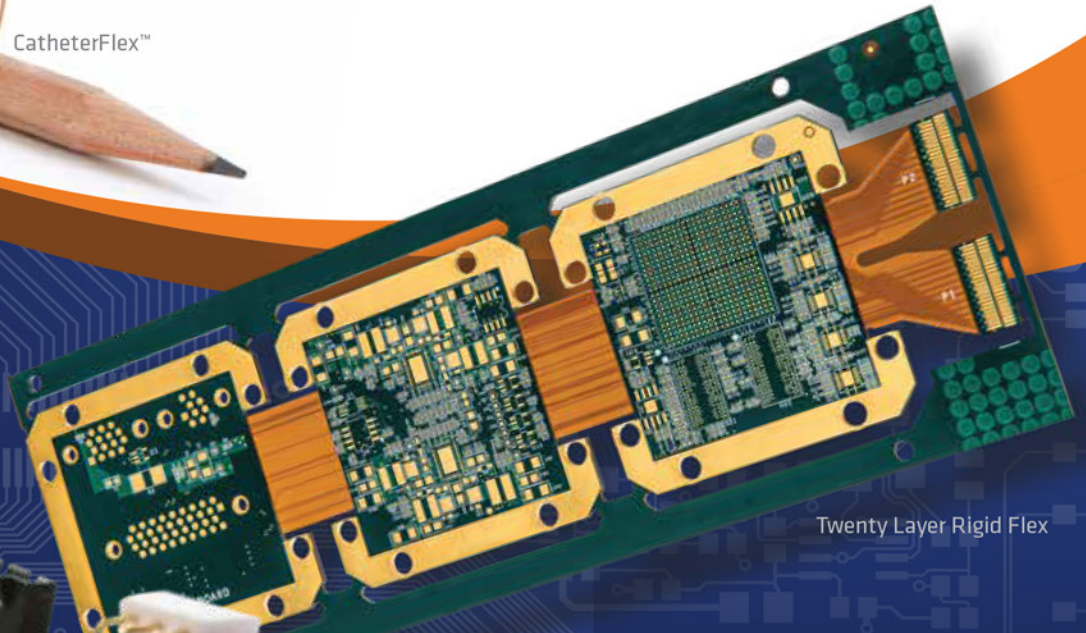
Subscribe 

I-Connect007

ALL YOUR FLEXIBLE SOLUTIONS IN ONE PLACE



CatheterFlex™



Twenty Layer Rigid Flex



And, Assembly Tool!

- Flexible circuit solutions
- Rigid flex solutions
- CatheterFlex® solutions
- Flexible heater solutions
- And assembly solutions too!

All Flex Solutions

1601 Cannon Lane
Northfield, MN 55057

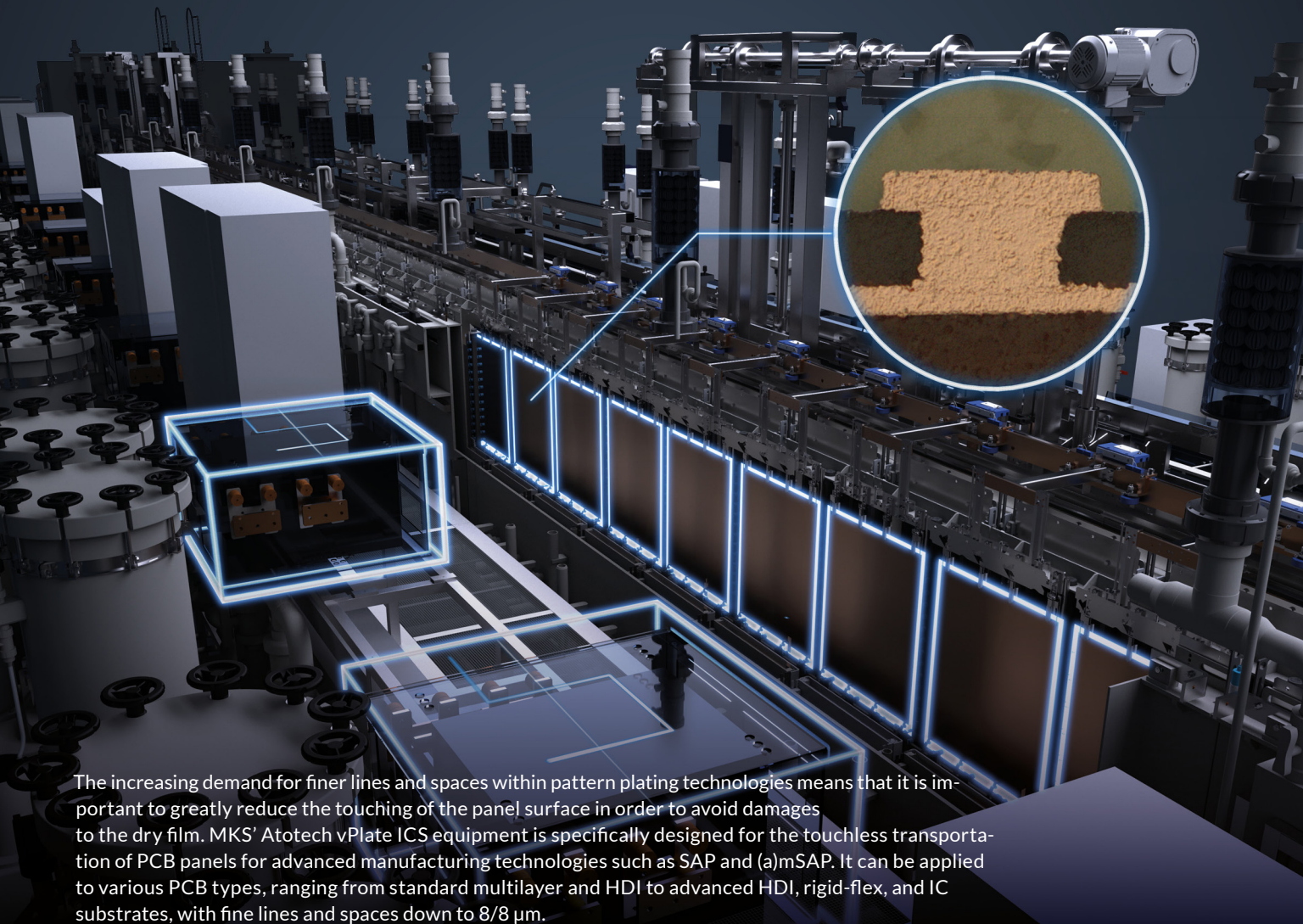
(507) 663-7162
AllFlexInc.com



SCAN ME



vPlate[®] ICS – Vertical systems solution for IC substrates and advanced HDI



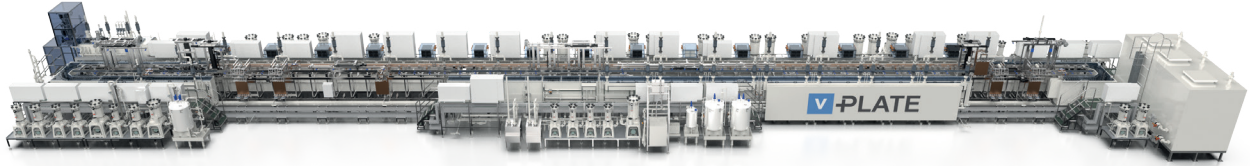
The increasing demand for finer lines and spaces within pattern plating technologies means that it is important to greatly reduce the touching of the panel surface in order to avoid damages to the dry film. MKS' Atotech vPlate ICS equipment is specifically designed for the touchless transportation of PCB panels for advanced manufacturing technologies such as SAP and (a)mSAP. It can be applied to various PCB types, ranging from standard multilayer and HDI to advanced HDI, rigid-flex, and IC substrates, with fine lines and spaces down to 8/8 μm .

The vertical continuous copper plating equipment is fully automated and compatible with our InPro[®] chemical process series for TH and BMV filling for advanced HDI, mSAP, and IC substrates, as well as our conformal plating processes such as InPro[®] VLF and InPro[®] Flex 2. For HAR and HHD applications we provide a combination of our Cuprapulse[®] IN process with vPlate ICS pulse plating capability. With vPlate ICS, we provide a one-stop-shop solution for effective equipment, software and process solutions.

To find out more about vPlate, scan the QR Code to the right.



vPlate® ICS – Vertical copper plating for finer features and more demanding substrates



As advanced HDI and package substrate designs move toward finer lines and spaces, copper plating equipment must handle increasingly delicate panels while maintaining consistent deposition across complex feature geometries. In pattern-plating processes, unnecessary contact with the panel surface can damage the dry film and compromise fine-feature production.

The manufacturing challenge

Future-oriented substrate production requires more than a reduction in line and space. Manufacturers must process ultra-thin panels, fill blind microvias and through-holes, manage variations in panel design, and achieve uniform copper distribution at production-relevant current densities. At the same time, thicker substrates and high-aspect-ratio through-holes place additional demands on mass transport and throwing power.

Equipment design therefore becomes an integral part of process performance. Anode configuration, shielding, panel alignment, jig condition, and solution exchange influence the distribution of copper across the panel and within the features.

A systems approach to vertical plating

MKS' Atotech developed vPlate® ICS as a vertical continuous copper plating platform for advanced HDI, SAP, (a)mSAP, and package substrate production. Its touchless transportation concept supports panels down to 36 µm core thickness with 2 × 2 µm copper cladding and design rules down to 8/8 µm line and space.

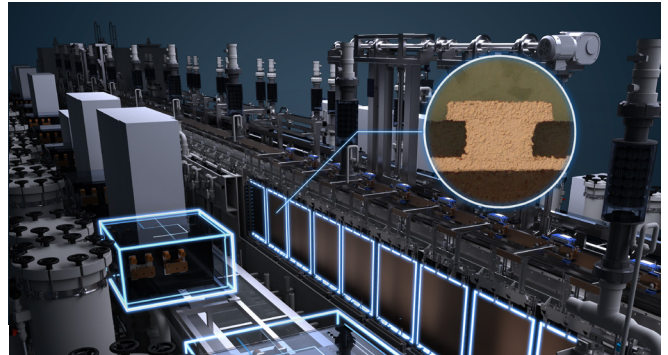
The platform combines insoluble anodes with adjustable anode and cathode shielding to control copper distribution. Specially designed jigs and precise panel alignment help maintain a consistent inter-panel gap and limit edge effects. Automatic jig monitoring checks critical jig conditions, supporting process stability and increasing yield while reducing the operator workload associated with manual inspection.

vPlate ICS supports both direct-current and reverse-pulse plating. Depending on the application, it can be combined with MKS' Atotech InPro® processes for blind-microvia and through-hole filling or with Cuprapulse® IN for conformal plating of high-aspect-ratio through-holes. Current uniformity of ±7% and densities of up to 5 A/dm² are supported.

From equipment to process control

Automated loading and unloading, recipe management, parameter monitoring, RFID-based tracking, and automated grease supply contribute to repeatable operation. By bringing together equipment, chemistry, software, and process expertise, the systems approach allows the plating line and electrolyte to be configured around the specific panel and feature requirements.

Beyond installation, MKS' Atotech supports the equipment lifecycle through its global After Sales services. The Digital Factory Suite, or DFS, an IIoT software platform for equipment monitoring, traceability, notifications, and troubleshooting, complements this support. Together, these capabilities help manufacturers maintain transparency over equipment and process performance during production.



In short

As substrate geometries become more demanding, plating performance depends on the interaction between chemistry, equipment, automation, and panel handling. vPlate ICS provides a scalable vertical platform designed to support fine features, thin-panel processing, via filling, and high-aspect-ratio applications within one integrated production environment.



Dirk Ruess

Global Product Manager Equipment
at MKS' Atotech
dirk.ruess@mks.com
+49 9128 72 56 06

Lockheed Martin Conducts Second Successful Test of New JR3 Rocket ▶

Lockheed Martin has successfully tested its new Joint Reduced Range Rocket (JR3) solution for the second time, an affordable training round with a reusable launch pod designed to replace the current Reduced Range Practice Rocket for the U.S. Army and global partners.

Kratos Wins \$20M Contract for Mobile SATCOM Gateways ▶

Kratos Defense & Security Solutions, Inc., a technology company in defense, national security and global markets, announced that it has been awarded a contract valued at more than \$20 million to deliver mobile satellite communication (SATCOM) gateways for a defense customer in Asia.

NOTE Secures SEK 160 Million Security & Defense Contract ▶

NOTE has been awarded a contract by a European customer in the Security & Defense sector, with an estimated order value of approximately SEK 160 million over a three-year period. Order intake to date indicates a higher delivery rate than initially planned. The contract strengthens NOTE's position within the segment and confirms the growing demand for European electronics manufacturing.

Defense Speak Interpreted: Will Defense Become a Totally New Supplier for Drone Electronics? ▶

When I read recently that Tobyhanna Army Depot would begin manufacturing up to one million circuit cards annually for military drones, I immediately wondered whether this opens a whole new chapter of volume electronics manufacturing within Defense. Historically, Defense has had access to some aspect of printed circuit production.

American Made Advocacy: American PCBs in Big Demand and Short Supply ▶

The PCB industry faces challenges that are only now being recognized by government agencies. Against the backdrop of conflicts in Europe and the Middle East and the ongoing threat of Chinese business and military aggression, the microelectronics industry overall has been upended. American PCB production has been hit especially hard.

QTREX Quantum AME Technology Deployed by Major Israeli Defense Company ▶

QTREX Quantum Ltd., a company focused on advancing Additively Manufactured Electronics (AME) for quantum computing infrastructure, announced that one of Israel's three largest defense companies and one of the world's leading aerospace and defense organizations has begun deploying QTREX's AME technology for a joint development program focused on strategic systems operating at the physical limits of precision and performance.

Battalion Advanced Technology, Penn State Launch National Security Research Collaboration ▶

Penn State and Battalion Advanced Technology Ltd. have entered into two research agreements worth up to \$6 million to develop advanced semiconductor platforms and high temperature materials that aim to enable next-generation energy, transportation, aerospace, and industrial systems while also providing critical capabilities for national security.

Everspin, Teledyne HiRel Partner to Accelerate MRAM Adoption in Aerospace and Defense ▶

Everspin Technologies, Inc., the world's leading developer and manufacturer of MRAM solutions, and Teledyne HiRel Semiconductors, a business unit of Teledyne Technologies Incorporated and a leading supplier of high-reliability semiconductor solutions, announced a strategic partnership to accelerate adoption of Everspin's MRAM in aerospace, defense and other demanding systems.

Teledyne Surpasses 1,000 Space Imaging Sensors with NASA Roman Telescope Launch ▶

Teledyne Space Imaging, a leading supplier of space-qualified imaging sensors, focal plane arrays, and integrated camera systems, celebrated a major milestone following the successful launch of NASA's Nancy Grace Roman Space Telescope, which lifted off on Aug. 30 aboard a SpaceX Falcon Heavy rocket from Kennedy Space Center in Florida.

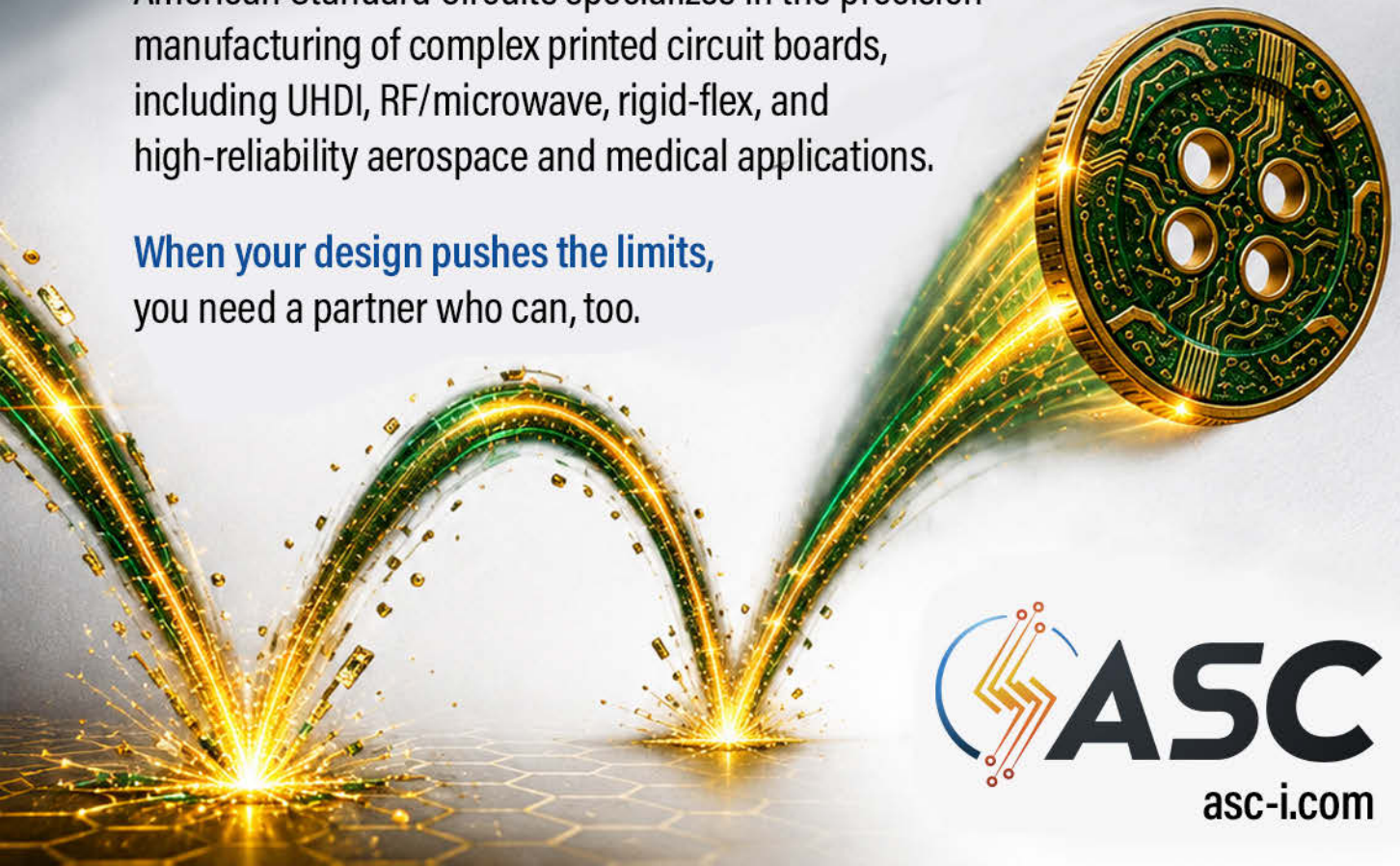
Stressed By **TIGHT** *Tolerances?*

RELAX. *Your specs won't
have us bursting at the seams.*

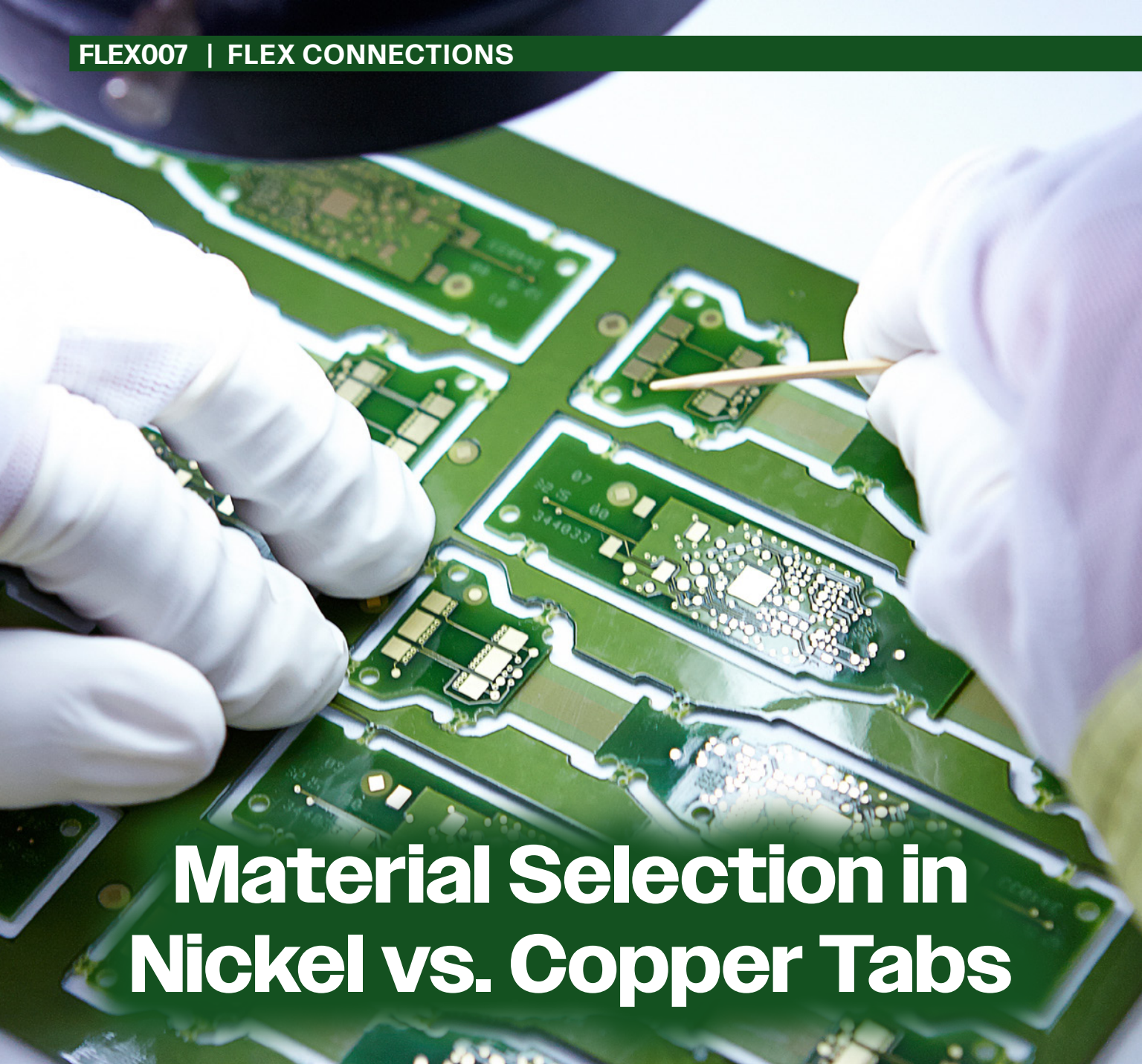


American Standard Circuits specializes in the precision manufacturing of complex printed circuit boards, including UHDl, RF/microwave, rigid-flex, and high-reliability aerospace and medical applications.

When your design pushes the limits,
you need a partner who can, too.



asc-i.com



Material Selection in Nickel vs. Copper Tabs

BY ZACK SCHANER, FLEXIBLE CIRCUIT TECHNOLOGIES

Flexible printed circuits are being designed to do more than route signals. In many applications, the flex circuit must also provide robust electrical interfaces, mechanical reinforcement, and reliable transitions between the circuit and surrounding assemblies. One effective way to accomplish this is by integrating copper or nickel tabs directly into a flexible printed circuit.

These tabs can serve as termination points,

battery contacts, grounding features, wear surfaces, or mechanical attachment points. For each application, choosing the right tab material and thickness is critical, because the electrical, mechanical, and environmental demands of the end application can vary significantly. Copper and nickel offer unique advantages, and selecting the correct material and thickness can improve performance, simplify assembly, and increase long-term reliability.

TiTAN Hybrid: UV LED + Laser Technology Direct Imaging for Dry Film & Soldermask



1 Dual Light Source
UV LED and laser hybrid technology

2 Exceptional Precision
 $3\sigma \leq 10\mu$ alignment accuracy

3 Cost Effective
Advanced automation minimizes labor costs

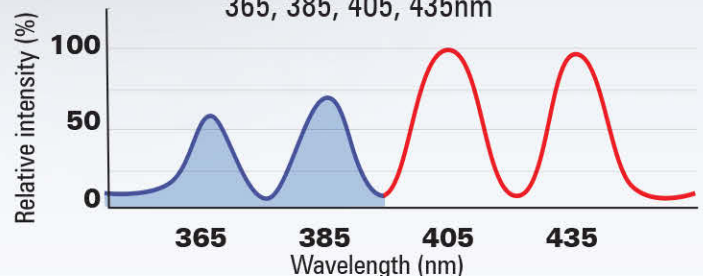
4 Smart Automation
Seamless integration with MES systems

5 Excellent Product Traceability
Multiple tracking methods to enhance QC

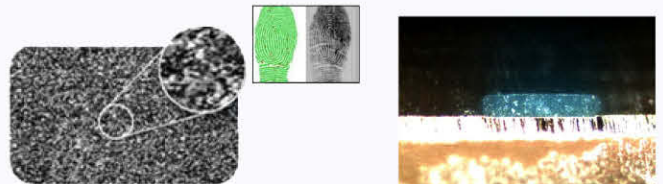
6 User-friendly Interface
Supports industry-standard data formats

Hybrid quad-wavelength with adjustable settings

UV LED + Laser integration featuring wavelengths:
365, 385, 405, 435nm



Hole-free patented technology



TECHNICA, U.S.A.

1-800-909-8697 • www.technica.com

Copper Tabs: High Conductivity and Versatile Integration

Copper tabs are commonly integrated into flexible circuits because they combine excellent electrical conductivity with compatibility across standard assembly processes such as SMT and reflow. As the primary conductor used in copper-clad laminates, copper can be incorporated into tab features efficiently and with minimal complexity.

Advantages of Copper Tabs

- **Excellent electrical conductivity:** Copper offers very low resistance, making it ideal for power delivery, grounding, and signal transfer applications where electrical efficiency matters.
- **Good solderability and termination flexibility:** Copper tabs work well for soldered terminations, crimping, and mechanical attachment. Surface finishes such as tin, ENIG, or electrolytic nickel/gold can be added as well.
- **Cost-effective:** Copper is generally more economical than nickel, especially in high-volume production.

Considerations With Copper Tabs

Copper is relatively soft compared to nickel and, depending on thickness and temper, can deform more easily under repeated mechanical contact or insertion cycles. It can also oxidize over time if left unprotected. Understanding how the tabs will be used and the environment they will be in can assist in surface finish selection.

Common Applications for Copper Tabs

- Power distribution and battery interconnects
- Voltage sense and temperature sense circuits
- Ground tabs and shielding connections
- Solderable terminals
- Medical devices requiring compact electrical terminations

- Industrial assemblies where conductivity is critical.

Nickel Tabs: Strength, Wear Resistance, and Environmental Durability

Nickel tabs are often selected when the application requires more mechanical durability or environmental resistance than copper alone can provide. Nickel offers excellent strength and stiffness while maintaining reasonable conductivity.

Advantages of Nickel Tabs

- **High mechanical strength:** Nickel resists bending and deformation better than copper, making it well-suited for applications involving repeated contact or higher insertion forces.
- **Improved wear resistance:** Nickel performs well in contact surfaces where abrasion or friction is present.
- **Excellent corrosion resistance:** Nickel maintains performance in demanding environments involving humidity, chemicals, or temperature cycling.
- **Dimensional stability:** Nickel retains shape well, which is especially valuable when utilizing preformed nickel tabs.

Considerations With Nickel Tabs

Nickel has significantly higher electrical resistance than copper, so it is generally not the preferred option where maximum conductivity is required. It can also require more specialized surface preparation depending on thickness and attachment method.

Common Applications for Nickel Tabs

- Battery tabs and pack interconnects
- Wear-resistant terminals
- Voltage and temperature sensing
- Medical devices requiring durable repeated-use contact surfaces
- Harsh-environment industrial electronics

ATTRIBUTE	COPPER TABS	NICKEL TABS
Electrical conductivity	Excellent	Moderate
Mechanical strength	Moderate	High
Wear resistance	Moderate	Excellent
Corrosion resistance	Good with finish	Excellent
Ease of integration	Excellent	Good
Cost	Lower	Higher
Best suited for	Power and signal	Mechanical contact and durability

Design Considerations for Tabs in Flexible Circuits

When integrating copper or nickel tabs into a flex circuit, material selection is only one part of the design equation. Engineers also need to consider:

- **Mechanical transition:** The interface between rigid metal tabs and flexible polyimide must be designed carefully to manage stress and prevent fatigue over time.
- **Plating and finish requirements:** Depending on the electrical or mechanical function, finishes such as tin, nickel, or gold may be needed.
- **Forming requirements:** Some tabs may require bending or pre-forming after fabrication.
- **Assembly environment:** Temperature exposure, chemicals, sterilization requirements, and repeated-use cycles can all influence material selection.

I've found that material selection becomes much easier once the application is clearly defined.

Engineers naturally gravitate toward conductivity, strength, or cost, but the best choice usually comes from balancing all three while considering how the finished product will actually be used.

That's also why I encourage designers to involve their flex circuit manufacturer early in the design process. A short conversation before the design is finalized often uncovers considerations that are difficult to identify from a drawing alone.

Copper and nickel have each earned their place in flexible circuit design. Understanding where each performs best helps create products that are easier to manufacture, more reliable in the field, and less likely to require design changes later. **I-CONNECT007**



Zack Schaner is a senior applications engineer for Flexible Circuit Technologies. To read past columns, [click here](#).



When the World Becomes Your Design Team

More than a decade ago, I stood in front of an IPC APEX EXPO Design Forum audience and talked about something that was becoming more important in electronics design: successful communication with an international engineering team.

I had been working with a global PCB design orga-

nization that included a talented team in Bangalore, India. However, I was based in Arizona, where there was a 12-hour time difference separating us. With different work schedules, cultures, languages, and communication styles, there were plenty of reasons that made collaboration difficult.



inTFlex
The EMS Group of Flexible Circuit Technologies

FLEX SOLUTIONS ENGINEERED FOR INNOVATION

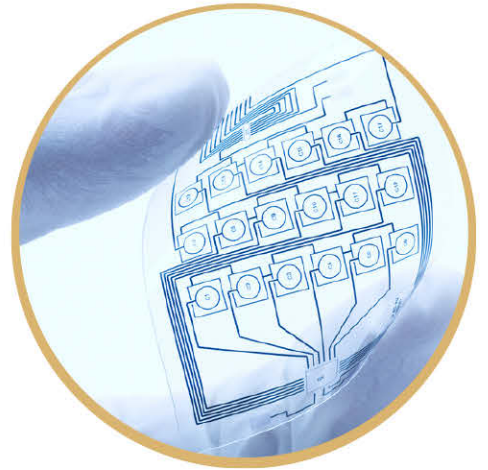
Industry-leading flex design services, premier in-house manufacturing of flexible circuits and EMS services



Flexible Circuits



Integrated Solutions



Printed Electronics

Superior Design Support
From Prototypes to Production

OUR PRODUCTS

- Flexible Printed Circuits
- Catheter Flexible Circuits
- Flexible Heaters
- Flat Flex
- Printed Electronics
- Circuit Assemblies
- Plastic Moldings
- Product Box Builds

**SINCE
2000**

OUR SERVICES

- Design Consultation & Optimization
- Prototyping
- Manufacturing & Assembly
- Testing & Traceability
- Inventory Stocking
- Program Management
- Turnkey Services
- Advance Manufacturing

sales@flexiblecircuit.com 763.545.3333 flexiblecircuit.com

Instead, the experience taught me that these differences did not need to be a barrier. If a global team is structured and managed correctly, geographic and other cultural separations can become an advantage. Communication, trust, accountability, respect, and knowledge sharing must always remain the foundation of any successful global collaboration.

This is equally true for global engineering teams, and now, with improvements in our technology and communication tools, distributed engineering has become integral to the development of complex electronic products.

The Time Difference Can Become an Advantage

One of the most obvious challenges of an international team is working across extended time zones. When there is a 12-hour time difference, meetings are more difficult to schedule, and questions and problems may have to wait for many hours to be answered or resolved.

But I found that the time difference became an engineering advantage. The engineering day effectively follows the sun, so if the work is properly organized, one team can continue where another team left off. A PCB designer in one location completes a portion of the design, documents the work, identifies open questions, and hands it off to another designer who is just beginning their workday. That creates engineering continuity.

However, continuity doesn't happen simply because two teams are in different time zones. It still requires disciplined communication and effective handoffs. In my experience, the receiving team needs to know what was completed, the decisions made, the remaining issues, and what needs to happen next. More than simply transferring files, a handoff is transferring context.

Communication Is Still the Infrastructure

Looking back at the lessons learned from working with that India team, I would still put the importance of effective communication at the top. At the time,

we used phone and conference calls, WebEx sessions, email, instant messaging, daily handoffs, weekly meetings, and one-on-one conversations to accomplish our work.

Today, we use even more sophisticated collaboration technologies, but we still face another fundamental communication challenge: A distributed engineering organization can have every collaboration tool available and still operate as a collection of individuals working independently rather than as a team.

Effective communication is not just about the tools. It requires enough shared context for engineers to

make good decisions even when the person who made the original decision is halfway around the world. You must communicate early, document important decisions, ask questions, provide feedback, and make sure people know when they are expected to be available.

It also means the willingness to make some personal sacrifices. Sometimes, I worked outside my normal schedule to accommodate the workday in India, because I viewed the team as part of the engineering organization rather than an

external resource. I wanted to be available to them and adapt my meeting schedules when necessary. That distinction mattered.

Building Trust Through Results

Every global engineering team eventually faces the same question: How do we build trust among people who may be thousands of miles apart? In the organization I worked with, gaining the confidence of the local engineering team required demonstrating good-quality work and on-time delivery. The same principle applied to customers. Geographic separation could not serve as an excuse for uncertainty about the quality of the work or about whether commitments would be met.

Once a global team consistently demonstrates that it can deliver, our conversation changes from "Can they do this?" to "What can we accomplish



Technology makes it easy to forget that global collaboration is fundamentally about people."

together?” That’s when you see a distributed team become a true engineering organization.

Don't Let Distance Create Distance

Technology makes it easy to forget that global collaboration is fundamentally about people. It’s important to get to know your team members personally. I believed strongly in spending time with the team in India rather than attempting to manage the relationship entirely through technology. I regularly traveled there, so I could get to know my team members, helping me approach the relationship with respect and positive assumptions. I feel that personal connection made a difference for them and for me.

When engineers know and trust one another and understand their working styles, communication becomes easier. We are more comfortable asking questions, challenging assumptions, sharing ideas, and acknowledging when we need help. This becomes particularly important when cultural differences are involved.

A different communication style does not necessarily mean a lack of engagement. A question that seems obvious to one person may not be obvious to someone working within a different cultural or organizational context. The answer is not to assume the worst. The answer is to ask, listen, and seek to understand, not just to be heard.

Knowledge Grows Through People

A global engineering organization has access to a tremendous amount of collective experience. The challenge is to make that knowledge accessible to everyone and to create an environment where engineers learn from one another.

One of the most valuable initiatives we developed within the India organization was an internal IPC club. As designers, we met frequently to discuss PCB design experiences, IPC knowledge, complex design challenges, tools, processes, and lessons learned. Our objective was to improve both individual designers and the team’s overall capability, and it worked.

That concept remains just as relevant today. Design reviews, technical forums, lessons-learned sessions, training, supplier interactions, and industry engagement can turn individual experience into organizational capability.

However, knowledge sharing is only part of the equation. Organizations should invest in the people expected to apply that knowledge by enabling them to develop new skills and pursue new learning opportunities. When that happens, I’ve seen the entire organization become stronger. Rather than remain with one person, project, or location, that experience becomes part of the team’s collective capability.

This is especially important in PCB design, where much of what separates a good design from a great one comes from experience. Understanding why something happens in a design can be just as valuable as the design data itself, and the best global teams make both knowledge and experience portable. More importantly, they recognize that knowledge does not just develop on its own.

One Team, Not Two

The most important lesson from my experience with the India team was that global collaboration fails when an organization creates an artificial distinction between “the local team” and “the global team.” There should be one engineering team.

It doesn’t mean every location performs the same work or has the same responsibilities, but that we all understand the common objectives, share accountability for the outcome, and have a voice in solving problems.

When a challenge arises, it should be handled the same as if the engineer were sitting in the cubicle next to you: address it immediately, discuss the options openly, listen to suggestions, and solve it together.

Effective global collaboration enables engineering organizations to improve quality, control costs, manage manufacturability, and get products to market faster. When that happens, the world truly can become your design team. **I-CONNECT007**



Stephen V. Chavez is principal technical product marketing manager at Siemens EDA and chair of PCEA.

PCB Design Takes Center Stage at FED's Design Awards



PCB designers often work behind the scenes, but their decisions can determine whether an electronic product succeeds in manufacturing. At the FED Annual Conference in Germany on Sept. 23, the PCB Design Award 2026 will put that work in the spotlight. Marcy LaRont interviewed FED Managing Director Christoph Bornhorn and board member Erika Reel about what makes a winning design, and how the profession is changing as AI, automation, and other technologies reshape electronics development.

Marcy LaRont: Christoph and Erika, before we get into the Design Awards, please tell us briefly about your roles with FED.

Christoph Bornhorn: I am responsible for the day-to-day operations of the association and for ensuring that FED remains a leading platform for the electronics community. My responsibility is to continuously expand the association's reach and membership, ensure we provide excellent service to our members and customers, and organize training sessions, events, and conferences.

Candor: Relentless Innovation

Our commitment to innovation ensures that our products not only meet the highest standards, but also lead the way in environmentally-friendly practices.

Our innovations include:

- Landless Vias
- Liquid Organic Photoresist
- Mechanical Drilling Blind/Buried Vias
- Graphite Metalization
- Panel Plating
- Induction Lamination

EXPLORE OUR INNOVATIONS



Candor
Circuit Boards



www.candorind.com

Erika Reel: As a longtime member of FED and now a member of the board, I help shape the strategic direction of the association. I am responsible for PCB design as well as training and continuing education. One of my projects has been the introduction and development of the Certified Electronics Designer (ZED), a comprehensive professional qualification program for PCB designers in Europe.



I'm sure you're looking forward to your annual conference. What can you tell me about it?

Bornhorn: This is Germany's leading industry conference for electronic design and manufacturing and serves as a platform for the entire electronics value chain, bringing together PCB designers, electronics manufacturers, EMS providers, OEMs, technology suppliers, and applied research institutions to exchange practical knowledge and discuss the latest developments in electronic hardware design and production. Each year, more than 300 industry professionals participate in 45 expert presentations, panel discussions, keynote speeches, and a dedicated technology exhibition.

What makes the FED Conference particularly important is its strong focus on real-world industrial challenges and solutions. The conference promotes the exchange of best practices and highlights emerging technologies such as AI, automation, advanced packaging, and digitalization.

For many participants, the FED Conference is not only a source of technical expertise but also an important networking event in the German-speaking electronics industry. It creates opportunities for collaboration, innovation, and knowledge transfer that ultimately drive the future of electronics development and manufacturing.

What do the PCB Design Awards recognize?

Reel: The awards were established in 2012 to recognize outstanding

achievements in PCB design and the increasingly important role of PCB designers. Our primary goal is to give visibility to the engineers, designers, and development teams whose work forms the foundation of every successful electronic product. While end products often receive the attention, the complex engineering behind PCB design frequently remains unseen. We want to change that!

PCB designers serve as the central interface between development and manufacturing by coordinating requirements from circuit design, production, assembly, and testing. They have a significant influence on the costs and quality of the subsequent manufacturing process. Therefore, the award is intended to raise public awareness of the profession.

How are the awards categorized?

Reel: There are four categories: 3D, High Power, High Density, and Simply Ingenious. Among the numerous award winners are actually two of our current board members: Michael Schleicher, who is also very active at the Global Electronics Association, and Michael Matthes, who oversees design matters on the board. What all the award winners have in common is that they have successfully implemented exceptionally clever solutions to complex challenges in PCB design.

What makes an award-winning design?

Bornhorn: An award-winning design combines technical excellence with practical manufacturability and clear documentation. It demonstrates outstanding engineering skills in overcoming design constraints, whether it was maximizing performance in extremely limited space, managing high currents and controlled-impedance signals, or achieving exceptional density and reliability for advanced computing applications. In short, an award-winning design translates com-



plex requirements into elegant, efficient, and manufacturable solutions.

What separates a very good PCB design from an award-winning one?

Bornhorn: A very good PCB design successfully fulfills its technical requirements. An award-winning design goes beyond that by solving particularly difficult challenges in an innovative and elegant way. The winning projects in the past demonstrated, for example, exceptional optimization of space, power density, thermal management, signal integrity, reliability, and manufacturability, often under severe constraints. They stood out not only because they worked, but because they achieved outstanding results with remarkable precision, creativity, and engineering discipline while maintaining excellent documentation and production readiness.

What are some of the judges' criteria?

Bornhorn: The jury evaluates submissions against a range of technical and practical criteria that reflect both engineering excellence and real-world manufacturability. Key aspects include the overall symmetry and clarity of the design, the effective use of available board space, and a placement strategy that achieves high component density without compromising functionality or reliability.

Particular attention is paid to whether functional groups can be clearly identified within the layout, demonstrating a logical and well-structured design approach. The stackup concept is also an important evaluation factor, which is why entrants are required to provide a layer stackup sketch as part of their documentation.

In addition, the judges assess DFM considerations to ensure that the design can be produced efficiently and economically. Finally, a strong submission clearly describes the technical challenges involved and convincingly explains how these challenges were addressed through the design.

What are you most looking forward to at this year's PCB Design Awards, and what can attendees expect?

Reel: I am looking forward to the opportunity to

put PCB designers and their achievements in the spotlight. As I mentioned earlier, these professionals often work behind the scenes, yet they play a critical role in turning innovative ideas into reliable, manufacturable electronic products.

Attendees can look forward to seeing some truly outstanding and innovative designs. The award finalists showcase what is possible when engineering excellence meets creativity, and they provide valuable inspiration for the entire industry. It is always fascinating to learn how design teams have solved challenging technical problems and transformed ambitious concepts into successful products.

In addition, attendees can benefit from the accompanying FED Conference, which covers current trends, best practices, and emerging technologies in electronics design and manufacturing.

How has the role of the PCB designer changed, particularly as AI and digital tools become part of the process?

Bornhorn: One of their biggest challenges is the rapid transformation driven by AI and digitalization. PCB designers see a fundamental change in how products are conceived, developed, and optimized.

AI can support tasks such as component selection, design verification, layout optimization, signal integrity analysis, and documentation. That creates tremendous opportunities for greater productivity and faster development cycles, but it is changing the role of the PCB designer. Their routine activities are being automated, while the demand for system-level thinking, engineering judgment, creativity, and interdisciplinary collaboration is growing.

The profession is therefore evolving from a focus primarily on PCB layout to a key contributor to the overall product development process. Continuous learning has never been more important as designers need to understand electronics, manufacturing, reliability, sustainability, data management, and the intelligent tools that support these activities.

Thank you, Christoph and Erika. I-CONNECT007

HDI and UHDI

Building the Foundation of Advanced Packaging

BY MICHAEL CARANO, GLOBAL ELECTRONICS ASSOCIATION

The electronics industry ecosystem has never been more robust or dynamic. AI is a major factor in this rapid growth, fueling advances in high performance computing (HPC), advanced memory, high speed networking, and optical interconnects. In addition, advanced semiconductor packaging and data center build-out are pushing HDI and UHDI fabrication to circuit densities not seen previously.

The Semiconductor Industry Association Roadmap is driving wafer transistor dimensions smaller, enabling faster rise times, and creating the need for finer-pitch interconnects. TSMC is advancing nodes toward 2 nm. Yet despite these remarkable semiconductor breakthroughs, “chips don’t float.”

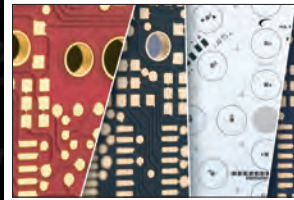
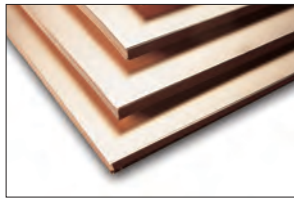
Silicon still requires a mounting platform, as well as printed circuits and organic IC substrates needed to create functional products.

In North America, there is a growing need for greater HDI capability, yet the adoption of HDI board fabrication has been slow. Three obstacles (or chokepoints) may include:

- Defining density needs
- Process controls for fabricating reliable microvias
- Bare board/in-circuit test (ICT) solutions once HDI is assembled

Panasonic

Multi-layer & Flex
Laminates

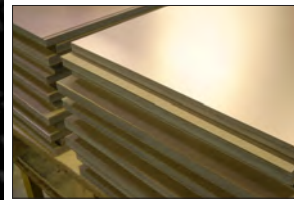


peters

Printed Circuit
Coatings



Electro-deposited
Copper Foils



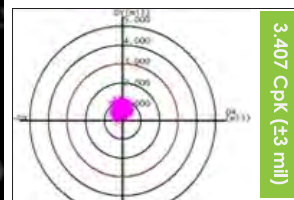
KB

KINGBOARD

Rigid &
Multi-layer CCL



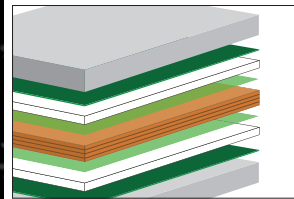
Drills & Routers



Drills Entry & Backup



PCB Press Plates
& Tooling



PCB Lamination
Assist Products



Dry Film Resist
Lamination



PCB Automation
Solutions

and more....



MATRIX
CIRCUIT BOARD MATERIALS

Supplier to the North American
Circuit Board Industry since 1977

www.matrixusa.us
1-800-668-5447

TORONTO • SANTA ANA • SANTA CLARA • CHICAGO • MINNESOTA • OSAKA

Defining Density Needs

Industry veteran Happy Holden once explained the interrelationship between component packaging, SMT assembly, and printed wiring boards:

- **Assembly complexity:** A measure of the difficulty to assemble surface mounted components, measured in parts per square inch and leads per square inch.
- **Component packaging:** The degree of sophistication of a component, measured by its average leads (I/Os) per part.
- **Printed wiring board density:** The amount of density (or complexity) of a PCB as measured by the average length of traces per square inch of that board, including all signal layers. The metric is inches per square inch.

Wiring density is defined as the amount of electrical interconnection routed within a given board or IC substrate footprint. The greater the wiring density, the smaller the footprint. This allows for miniaturization of the package and board.

Microvias used in HDI and UHDI designs increase density and functionality, while reducing layer counts and the overall footprint. Where are we today? It wasn't too long ago that the industry was thrilled to get down to 50-micron lines and spaces and 100-micron blind vias.

Today, with advanced packaging, heterogeneous integration and the use of chiplets, 15–20-micron lines and spaces, and 45–60-micron diameter blind vias are at the leading edge. Major advances in photolithography, metallization, and etching process, laser via formation, unique registration systems, and online process controls have enabled

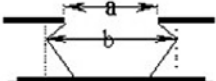
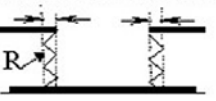
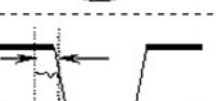
Item		Spec	Measurement	Sample Size	Control limit
1. Overhead		Class A: $a/b \geq 0.95$ Class B: $a/b \geq 0.90, a/b < 0.95$ Class C: $a/b < 0.90$	Cross section inspected by microscope	1. first piece of Per Part 2. one of per 10 lots.	$a/b = 0$ $a/b \leq 0.9$
2. Roughness		Class A: $R < 0.1$ mil Class B: $R \geq 0.1, P \leq 0.5$ mil Class C: $R > 0.5$ mil	Cross section inspected by microscope	1. first piece of Per Part 2. one of per 10 lots.	$R = 0$ $R \leq 0.5$ mil
3. Taper angle		Class A: 5° Class B: 10° Class C: 15°	1. Cross section inspected by microscope 2. Z-check	1. first piece of Per Part 2. one of per 10 lots.	$\theta = 0^\circ$ $\theta \leq 20^\circ$
4. Voids		Class A: No Voids Class B: Voids	Cross section inspected by microscope	1. first piece of Per Part 2. one of per 10 lots.	No Voids
5. Wicking		Class A: $L \geq 0, L \leq 0.1$ mil Class B: $L > 0.1, L \leq 0.5$ mil Class C: $L > 0.5$ mil	Cross section inspected by microscope	1. first piece of Per Part 2. one of per 10 lots.	$L = 0$ $L \leq 0.5$ mil
6. Resin remains		Class A: $t \geq 0, \tau \leq 0.2$ mil Class B: $> 0.2, t \leq 0.5$ mil Class C: $t > 0.5$ mil	Cross section inspected by microscope	1. first piece of Per Part 2. one of per 10 lots.	$t = 0$ $t \leq 0.5$ mil
7. Cracks		Class A: No Cracks Class B: Cracks	Cross section inspected by microscope	1. first piece of Per Part 2. one of per 10 lots.	No Cracks

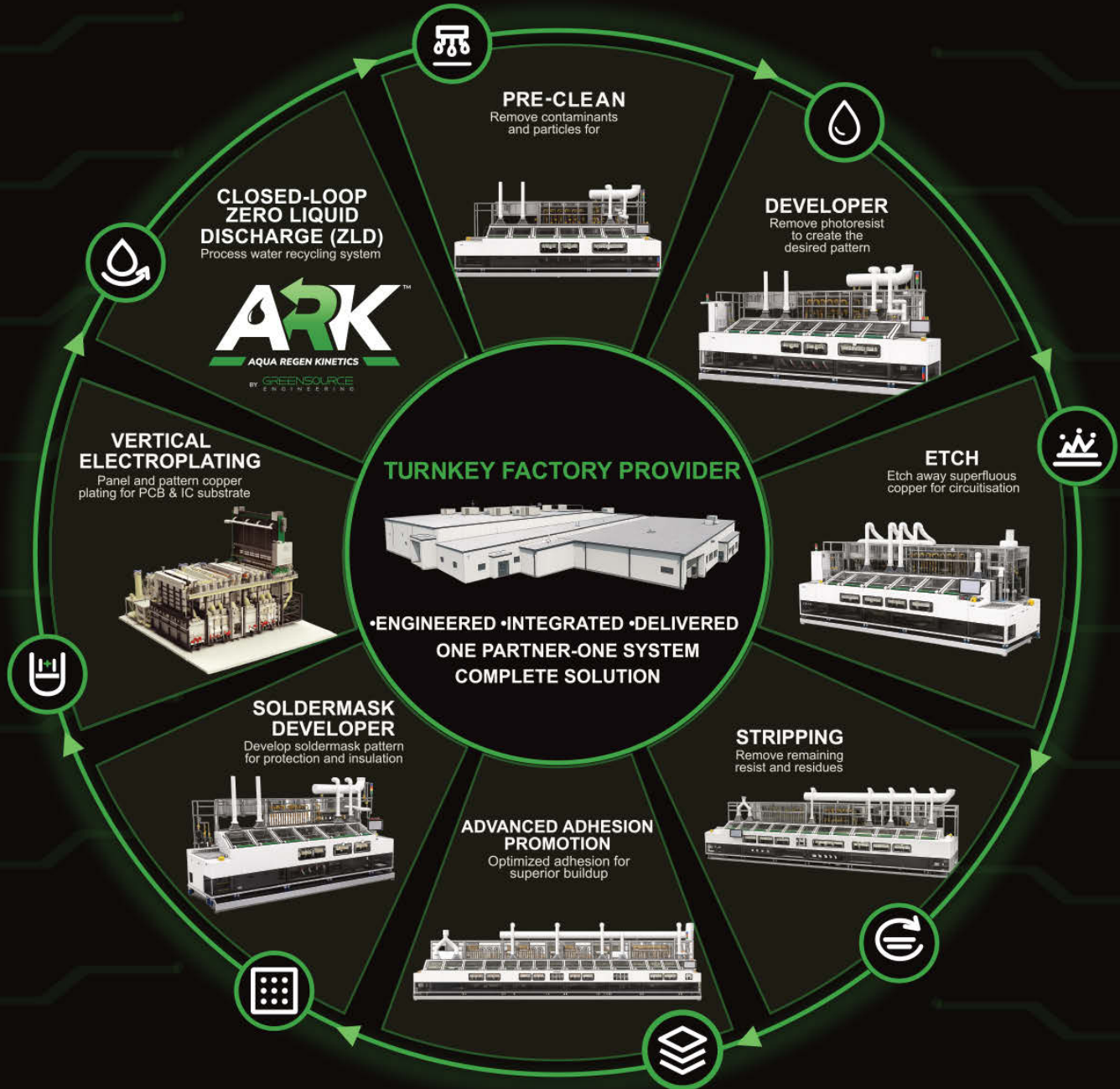
Figure 1: Blind via defects. (Source: Happy Holden)



GREENSOURCE
ENGINEERING

COMPLETE SOLUTIONS FOR
PCB & IC SUBSTRATE

ADVANCED TECHNOLOGY · END-TO-END SOLUTIONS



BUILT FOR PERFORMANCE. ENGINEERED FOR THE FUTURE.



PCB



HDI



IC SUBSTRATE



SPECIALIZED EQUIPMENT
DESIGNED ON DEMAND



AUTOMATION



ENVIRONMENTAL
SOLUTIONS

the technology to create very high-density PCBs and substrates. Of course, this sounds easier than it really is. With increased complexity, yields are challenged. At the end of the day, it is all about yields.

Process Controls for Fabricating Reliable Microvias

The small blind vias, known as microvias, pose distinct challenges for PCB fabricators. Unlike through-holes, microvias are far more difficult to drill, desmear, metallize, and plate. Laser drilling adds further complexity compared with mechanical through-hole drilling.

As shown in Figure 1, there are seven fundamental quality concerns, all of which can only be verified through microsectioning. Therefore, dedicated panel coupons are essential for confirming reliable microvia fabrication. This is more reason for extreme diligence in the via formation opera-

tion. With a poorly formed via, even the best plating chemistries will not improve the situation. In a future column, we will address the issues shown in Figure 1 in more detail.

The testing of HDI PCB prototypes can be accomplished with flying probe electrical testers, but volume production is still relegated to bed-of-nails-type custom fixturing. Multiple head flying probe testers do have more throughput, but even those models with 12 heads have only limited production capacity. It is the bed-of-nails fixture that will have to handle the production load. Unfortunately, the very high-density nature of HDI boards makes custom fixtures very expensive.

HDI boards have test points that, depending on the application, average 100–2,000+ pins per square inch. As an example, HDI server board may require 150–250 pins per square inch. Semiconductor package substrates exceed 1,000 pins per

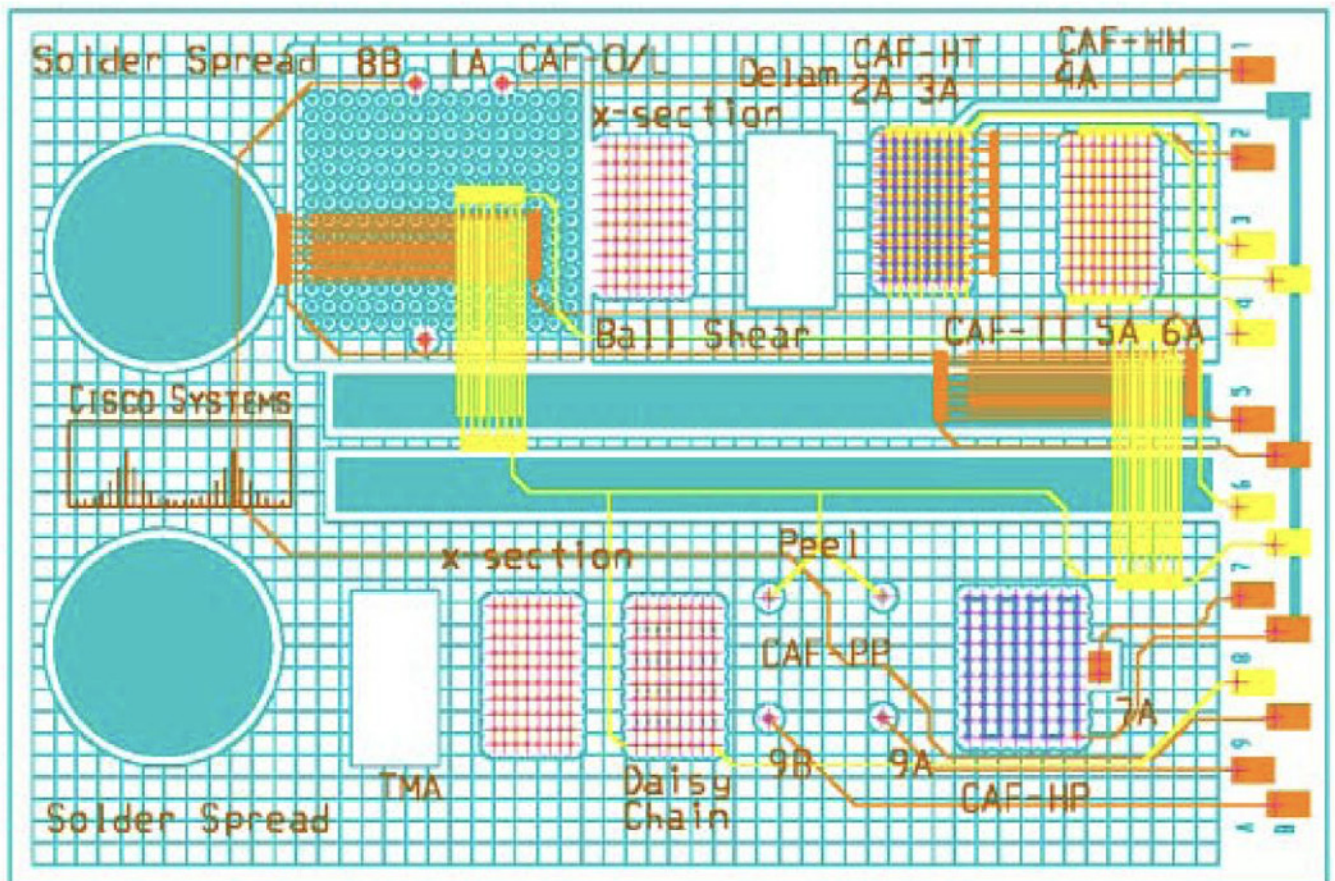


Figure 2: Multipurpose test vehicle. (Source: Happy Holden)

square inch. This fact requires the test equipment to navigate fine pitch HDI boards below 50-micron lines and spacing and 0.4mm pitch.

The good news is that traditional flying probe and bed-of-nails test equipment are being supplemented with a multitude of advanced electrical test technologies. These include very high speed flying probe testers that not only are capable of testing fine pitch designs, but are able to perform impedance measurements and detect intermittent opens and current leakage.

Panel Coupons

Special artwork designed into the panel edges or in unused space on the panel is essential. For additional traceability, if the artwork shares common features on the assembly sub-panel, there is continuity for whoever performs the SMT assembly. Figure 2 shows a small coupon design that can be used on fabrication panels and assembly sub-panel carriers.

The coupons are detachable and can be tested by production personnel during the fabrication process. These coupons test for:

- Desmear-metallization-plating quality (10 immersions of 10 sec in 288°C solder pot inspection)
- Layer-to-layer registration and movement directions (continuity and cross-section)

- Daisy chain electrical reliability (four immersions at 288°C of 10 seconds followed by 4-wire Kelvin measure)
- First drilled hole of multiple diameters/last drilled hole of multiple diameters (inspection if present)
- Layer-order windows (inspection if present)
- CAF reliability (CAF test chamber)

HDI and UHDI fabrication will continue to push the limits of PCB manufacturing as circuit densities increase and feature sizes shrink. For the fabricator, success will depend on much more than having the equipment to produce smaller features. Via formation, process control, testing, and the proper use of panel coupons all become important as tolerances tighten.

In the next installment, I'll take a closer look at process control for HDI and UHDI fabrication.

I-CONNECT007



Michael Carano brings over 40 years of electronics industry experience with special expertise in manufacturing, performance chemicals, metals, semiconductors, medical devices, and advanced packaging. To read past columns, [click here](#).

**Industry news. Real time.
On LinkedIn.**

FOLLOW US

I-Connect007
The Global Electronics Resource



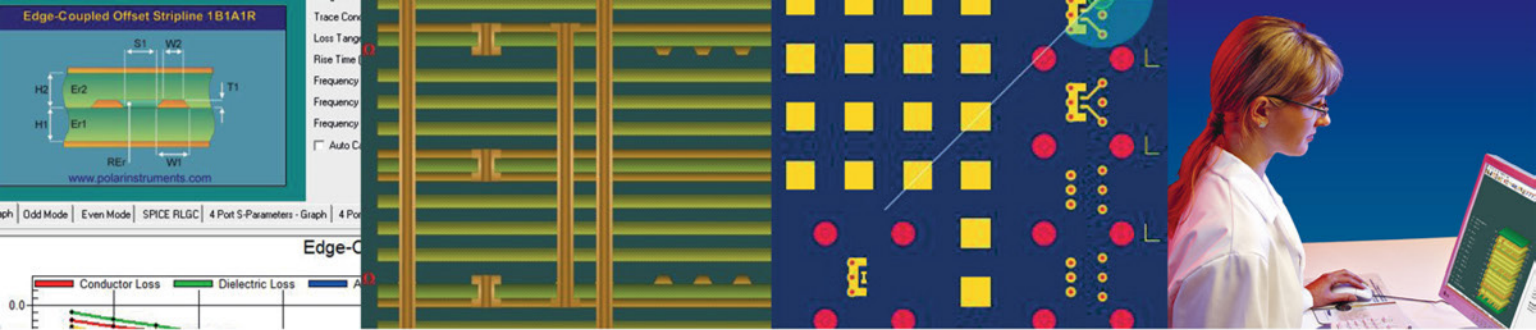
Breaking Down the Barriers Between Design and Manufacturing in France

The IPC Designers Council France has nearly doubled its reach since I-Connect007 last reached out to Thomas Romont, a manager at WWEC who leads the council. But growth is only part of the story. Through working groups, practical design tools, and an annual conference, the council is creating opportunities for designers, engineers, fabricators, EMS providers, and other stakeholders to solve problems together. Thomas explains how the group is addressing gaps in design guidance, improving communication between design and manufacturing, and encouraging the

kind of collaboration increasingly required across the electronics industry.

***Marcy LaRont:** Thomas, it's been a couple of years since we last caught up with you and the IPC Designers Council France. Bring us up to date.*

Thomas Romont: When we last spoke, about 250 people were involved in the council, representing not only PCB design but also electrical engineering, CAD, PCB fabrication, EMS, and OEMs. Today, nearly 500 people follow our work, but the origins of the council go back 10 years.



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

PCB Signal integrity tools for design & fabrication

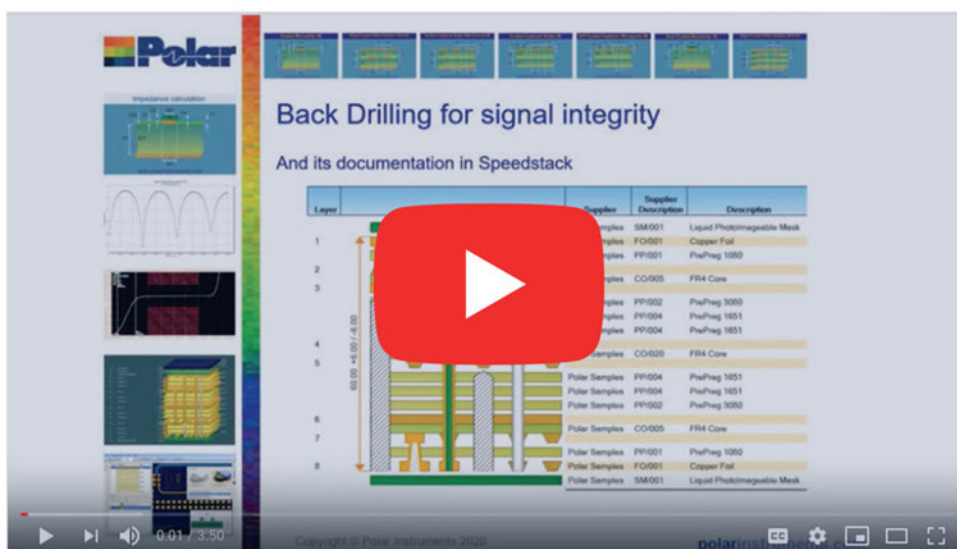
- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of SI topics

SUBSCRIBE



Add to your digital library:
*The Printed Circuit Designer's
Guide to... More Secrets of
High-Speed PCBs*

DOWNLOAD BOOK



polarinstruments.com

How did the council get started, and how has it evolved since then?

The origins of IPC Designers Council France (IDCF) go back to when I joined the Executive Board of the IPC Design Council. There, I had the opportunity to exchange ideas with many American design experts, including Gary Ferrari, Mike Creeden, and Rick Hartley. I also discovered the numerous Designers Council chapters throughout the United States, creating a network of designers on a large scale while still being available locally.

I was immediately convinced this format could appeal to French designers, who are spread across many small and medium-sized companies and are often one of only a few designers, or even the only designer, in their organization.

It didn't take much persuasion to convince Pierre-Jean Albrieux, CEO of IFTEC, that it would benefit the French electronics industry to have its own Designers Council. IFTEC, a long-standing player in the French electronics industry and partner of the Global Electronics Association, was well-positioned to initiate the group.

In February 2018, IFTEC and IPC Europe organized the first IPC Design Day in Paris, and I was responsible for the technical content. That event marked the official birth of IDCF and the formation of its board. Since then, IDCF has offered working groups throughout the year and an annual conference open to the whole industry. Our most recent conference in June attracted 135 attendees.

As the council has grown, has its mission changed, or are you still addressing the same fundamental needs within the design community?

Our mission remains to address gaps perceived by the industry by fostering exchanges among all design stakeholders, from electronics engineers to manufacturers. Our approach is to highlight IPC rules and recommendations when they exist and to propose additional guidance when they do not.

How important are IPC standards and your partnership with the Global Electronics Association to



Thomas Romont

that work? Is the Association the council's primary focus?

The partnership works on two levels. Technically, IPC standards and certifications are a great source of good practices that engineers and designers often don't know about or may have forgotten.

Logistically, the Global Electronics Association provides great support with registration and reservations, conference room bookings, welcoming and catering for participants, and the technical organization of broadcasts for remote attendees.

IDCF members believe IPC is the standards organization most dedicated to our industry, but we may also need to consult standards from organizations such as JEDEC, IEC, ISO, and UL for complementary recommendations.

When we last spoke, the council had several working groups developing practical resources for designers. What has come out of that work, and what are your groups focused on now?

We still have several working groups in progress. Their aim is to produce practical and efficient tools that are available to the whole industry. Although French is used in the working groups, we are all aware that electronics is now a global industry. That is why our tools are offered in English and can be used outside France. This is probably also why we have received requests for them from far beyond French borders.

The tools developed so far include:

- A Design Desk Reference, published by the Global Electronics Association as IPC-DR-DES-2022, which we are hoping to review in 2026
- A Master Drawing Template with instructions
- A DFM tool for bare PCBs that summarizes cost-sensitive parameters and returns an assessment based on IPC producibility levels to judge manufacturability and, indirectly, PCB cost

- A Layout Instructions Checklist
- An Assembly Drawing Checklist

Except for IPC-DR-DES-2022, these resources are available for free by request through the [IPC Designers Council France LinkedIn page](#).

How do the working groups operate?

The working groups are the council's day-to-day activity. Each has its own operating rules, but groups generally meet for about an hour every two weeks. We propose topics after each conference, and people join the groups that interest them.

Currently, two groups are active, working on design for testability and design for selective wave assembly. We also have several pending topics, including design for conformal coating, design of experiments to optimize PCB baking parameters, PCBA cleanliness and cleaning, the influence of glass fabric selection on material properties, high-voltage clearance and creepage distances for airborne applications, and design of experiments on via plating.

Participation in these groups continues to grow. At the very beginning of the council, groups had three to five people. Today, we regularly have 15 to 20 people attending each meeting.

Communication and documentation between design and manufacturing remain persistent challenges. How is the council working to improve that connection?

This is still an issue because electronic applications are so diverse that it is impossible to have a single document covering all of them.

Our solution is a document that covers the common requirements. Designers should not consider it a definitive tool, but a basic document that helps them avoid forgetting information and gives structure to a requirements list. It often needs to be customized for their specific requirements.

It is also a good opportunity to encourage designers to use standardized definitions of requirements—the IPC definitions, for sure—to avoid misunderstandings between designers and manufacturers.

I would also add that simply participating in the working groups is often a source of professional development. The very direct exchanges between those who draft documents and those who use them can highlight misunderstandings caused by certain wording and help identify both superfluous and missing information.

The industry is talking more about co-design, and the council already brings many of those stakeholders to the same table. How does IDCDF encourage greater collaboration across design and manufacturing?

DFX is definitely a co-design approach because it takes into account all requirements, from the electrical engineer to the manufacturers.

Our council was born from the observation that the industry has heavily segmented the different players involved in designing and manufacturing products according to their specific roles. This structuring into a customer-supplier chain has stifled communication among these stakeholders, and our aim is to restore a space for dialogue free from commercial or competitive connotations.

How do you see co-design?

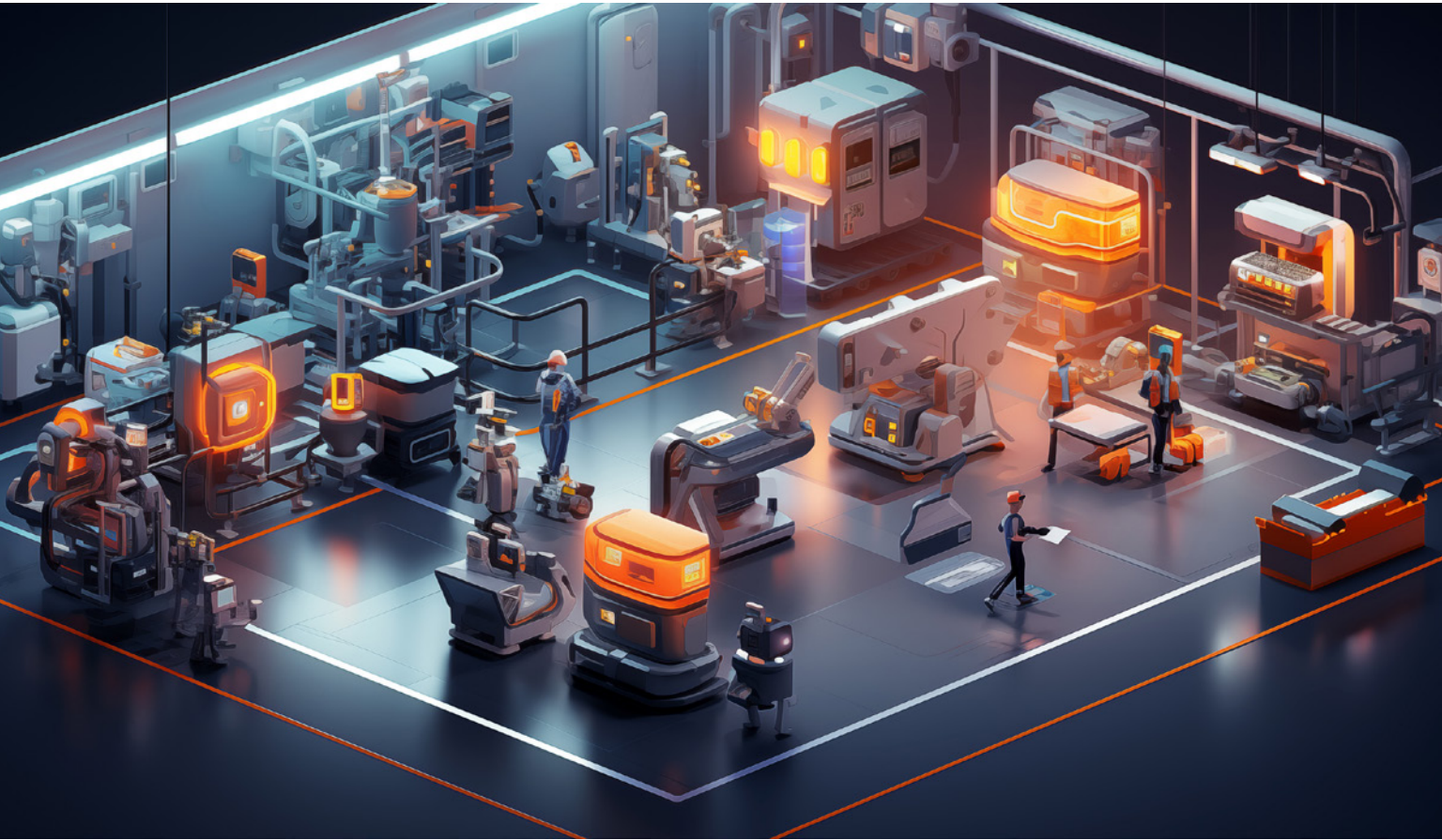
The term “co-design” has now become more of a synonym for integrating hardware into component design, or vice versa, and there is still a barrier between the component world and the hardware world.

We are in contact with other organizations involved in component and package design, such as IMAPS, but these exchanges remain sporadic. Each “world” focuses on its own specific issues without necessarily considering the impact its solutions will have on the other.

So, misunderstandings persist, and it is difficult to envision pooling efforts in the short term. Even if some of our pending working groups have elements of co-design behind them, I fear it will be the industrial reality that forces us to work together.

Thank you, Thomas. Keep us informed as the council moves ahead.

Yes, will do, Marcy. Thank you. **I-CONNECT007**



The Factory of the Future Still Needs People

Modern PCB factories can incorporate sophisticated automation, advanced process controls, machine vision, extensive traceability, real-time data collection, predictive analytics, and AI. These technologies are enabling manufacturers to see, measure, and identify opportunities that once depended almost entirely on human observation.

They also create an opportunity to think differently about the manufacturing workforce. The factory of the future brings technology and human expertise together, allowing each to contribute where it creates the greatest value.

So how can our intelligent factories help us create more knowledgeable, capable, and empowered people?

What Automation Makes Possible

PCB manufacturing is particularly well-suited to automation because it combines repetitive operations with demanding precision requirements, complex chemical processes, extensive material movement, tight process windows, inspection requirements, and enormous amounts of manufacturing data. Automated systems can perform many repetitive functions with exceptional consistency and can reduce unnecessary handling and variability while allowing manufacturers to build more repeatable and controlled processes.

However, as processes become more automated, we need people to provide an understanding of why a process exists and which variables matter. They



ElectraJet® EMJ110 Inkjet Soldermask

PCB Manufacturing with Proven Performance

ElectraJet® EMJ110 is the market-leading inkjet soldermask, delivering unmatched quality, robustness and reliability. Designed for modern PCB production, EMJ110 provides a streamlined, sustainable, and cost-effective solution for high-performance applications.

Available in Four Vibrant Colours:

- Green
- Blue
- Black
- Red



We Also Offer Flexible Soldermask:

- Green
- Amber



Key Features/Highlights

High Performance

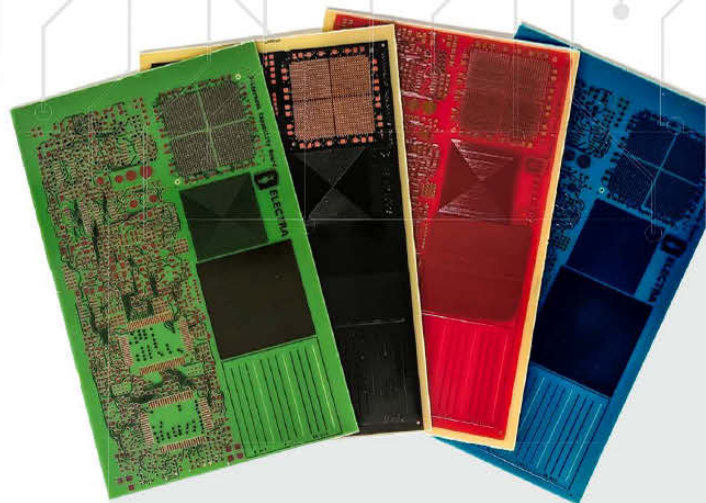
- EMJ110 designed to meet all key performance criteria
 - IPC SM840, UL-94V-0, no SVHCs, thermal cycling and heat storage

Robust Formulation

- Ink stable under a range of temperatures and environmental conditions
- No change in ink viscosity or particle size during use or shelf life of product
- Ink robustness critical for ensuring consistent printing with minimal problems
- Approved for use on leading inkjet printheads:
 - Konica Minolta – KM1024iMHE
 - Konica Minolta – KM1024iSHE
 - Konica Minolta – KM1800i
 - FUJIFILM Dimatix – Samba®

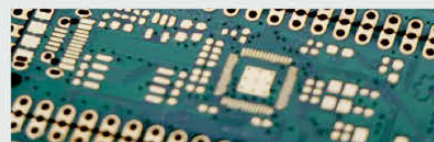
Reliability

- Waveform designed to reduce print quality defects
- Formulated to ensure high reliability jetting
- Minimal tool downtime during operation



Why Choose ElectraJet® EMJ110?

- **Proven Production Formulation** – Commercialised since 2019, extensively validated in real-world manufacturing.
- **Market Leader** – Electra is the global leader in inkjet soldermask, trusted by top PCB manufacturers worldwide.
- **Largest supplier** – Largest supplier of inkjet soldermask globally.
- **High-Performance & Reliable** – Meets strict automotive specifications, including high-temperature storage and thermal cycling.
- **Culmination of over 10 years of Innovation** – Developed through a decade of dedicated R&D in inkjet soldermask technology.
- **Eco-Friendly Solution** – Supports sustainable PCB manufacturing with reduced waste and material efficiency.



Join the Future of Soldermask Technology

Experience the benefits of ElectraJet® EMJ110 and take your PCB manufacturing to the next level.

Contact us today to learn more!

can recognize when a process is drifting, investigate unusual results, interpret data, understand the relationship between one operation and the next, and determine when intervention can improve an outcome.

Most importantly, people can question what is happening, understand the context, and determine what should happen next. That remains one of the most valuable capabilities in manufacturing.

AI Finds Patterns While People Provide Context

AI introduces another level of capability into automation due to the extraordinary volume of information generated in modern manufacturing environments. Consider the data that can accumulate across every area of the shop floor: plating thickness, chemistry, imaging, registration, lamination, drilling, inspection, electrical testing, environmental conditions, equipment performance, material lots, product types, and final yields. Those relationships can be extraordinarily difficult for a person to identify on their own. AI and machine learning help manufacturers detect anomalies, recognize developing process trends, support predictive maintenance, improve visual inspection, and identify correlations between process conditions and manufacturing outcomes.

Human experience adds context to what the data reveals. For example, an experienced technician standing beside a piece of equipment may notice that a pump sounds slightly different or may recognize that a material is behaving differently than usual. An engineer may understand that an unusual result reflects a particular customer requirement. A veteran employee may remember encountering a similar condition years earlier and know where to begin looking for the cause.

When AI can identify a pattern, experience can help explain what that pattern means, leading to powerful manufacturing decisions, and the conversation moves beyond AI toward augmented intelligence: technology that expands what people are able to see, understand, and act upon.

The Technician's Role Is Evolving

As technology changes the factory, so does workforce development. As machines perform more

repetitive physical tasks, the people overseeing those systems can develop a deeper understanding of the processes surrounding them. The PCB technician of the future will increasingly understand the entire process surrounding an operation: what happens before, during, and after, which variables matter, what the data means, how quality is affected, and when intervention is needed.

Consider an automated plating operation. An employee overseeing that process can develop knowledge that extends from basic electrochemistry and current density to solution control, plating distribution, material effects, equipment maintenance, process limits, statistical process control, defect recognition, upstream influences, downstream consequences, and troubleshooting. In essence, your skilled operator who no longer needs to manually operate a line can become more like an engineer in how he or she views the process and analyzes data.

Automation creates an opportunity to upskill employees and deepen the expertise present on the manufacturing floor. As PCB technology grows more sophisticated and manufacturing environments become more interconnected, that knowledge becomes a valuable part of the manufacturing system.

Building a Continuous Learning System

Manufacturers have long embraced continuous improvement for equipment and processes. Applying that same philosophy to people creates an opportunity for training. Employees can expand their understanding of processes, materials, quality, automation, data, troubleshooting, maintenance, customer requirements, and adjacent operations.

Cross-training is particularly valuable because manufacturing processes are deeply interconnected. A condition identified during drilling may have a relationship to an earlier process. A plating defect may reveal an upstream influence. An electrical failure discovered near the end of production may provide insight into something that occurred several processes earlier.

Turning Experience Into Institutional Knowledge

The PCB industry has an extraordinary amount of

The Answers Behind Better PCB Design

Build practical PCB design skills.
Solve complex design challenges.
Create better products.

PCB design challenges look different at every stage, from building a strong foundation to improving signal performance, manufacturability, and readiness for emerging technologies.

Electronics U's live online PCB courses help designers strengthen practical skills and make better design decisions with confidence.

How do I build a strong foundation in PCB design?

PCB Design I

Oct 12 – Nov 18

How do I design with manufacturability in mind?

PCB Design for Manufacturability

Oct 13 – Oct 29

How can I take my PCB design skills to the next level?

PCB Design II

Oct 12 – Dec 3

How can I prepare for emerging PCB technologies?

PCB Design for Emerging Design Technologies

Oct 12 – Nov 18

How can I design for stronger signal performance?

PCB Design for Signal Integrity

Oct 12 – Nov 4



FIND THE ANSWER TO YOUR NEXT PCB DESIGN CHALLENGE.

Need training for your team? Private instructor-led courses are available for organizations.
Contact us to discuss a custom training solution.



Register today at
[ElectronicsU.org](https://www.ElectronicsU.org)

Electronics U
by Global Electronics Association

expertise concentrated among professionals who have spent decades learning how materials, equipment, chemistry, and processes behave, and much of that expertise extends beyond what can be found in a manual. Ask a veteran PCB professional how they know a process is beginning to drift and the answer may be, “You can just tell.” What sounds like intuition can represent 30 years of accumulated pattern recognition.

This presents an important opportunity to pass it on that expertise to the next generation. AI and digital knowledge systems can provide the tools to capture that knowledge as employees document troubleshooting methods, unusual failure modes, process relationships, lessons learned, customer-specific challenges, and other insights that rarely appear in an SOP. AI and digital systems can then help organize and retrieve that knowledge.

Empowering People to Improve the Process

The person closest to the process often sees opportunities for improvement first, from technicians who notice unnecessary movement and recognize recurring alarms to maintenance personnel who see components that repeatedly require attention.

These observations represent an enormous continuous-improvement resource. Technology gives us more data, and people give that information context, ask questions, identify opportunities, and turn what they learn into action.

Creating an environment where employees are encouraged to identify problems, question inefficient practices, recommend changes, and act when something requires attention strengthens the entire manufacturing system. It is, in essence, the Toyota way—the company that first embraced Edward Deming’s principals and showed the world how to go about making continuous improvement part of one’s manufacturing culture. The more sophisticated the factory and products being built become, the more valuable that engagement.

What This Looks Like at GreenSource

At GreenSource Fabrication, we have seen this evolution firsthand. Our manufacturing environment incorporates a high degree of automation, including loaders and unloaders, touchless centering, turn

stations, fault and flexible buffers, automated material handling, automated electrical testing, and other purpose-built systems developed in partnership with our sister company, GreenSource Engineering. These investments are designed to create greater consistency, reduce unnecessary handling, generate more useful process information, and allow our people to focus their expertise where human judgment creates the greatest value.

Our workforce philosophy reflects that same approach. Experienced industry professionals bring decades of manufacturing knowledge, while rising talent brings new perspectives and an ability to grow alongside sophisticated technologies. Continuous learning, cross-training, and knowledge sharing create opportunities for employees to progress from foundational knowledge toward deeper expertise.

We’ve found that when automation, technology, and workforce development advance together, people spend more time solving problems, improving processes, developing new technologies, mentoring colleagues, learning new skills, and making informed decisions. The result is a manufacturing environment where both the technology and the people operating it become more capable.

Building the Human-centered PCB Factory

The next generation of PCB factories will include more automation, AI, robotics, machine vision, predictive analytics, digital traceability, and interconnected manufacturing systems and become even more important as customer expectations rise and deliverables become more complex.

Building the factory of the future means investing in both technology and people. Every major automation investment creates an opportunity to ask what new skills our people can develop. AI implementation creates opportunities to teach employees how to understand and apply the insights these systems produce. More manufacturing data creates opportunities to strengthen employees’ ability to interpret it. And critically important, decades of experience across the workforce create opportunities to capture and transfer knowledge to the next generation.

The Best Factory Is a Learning Factory

Manufacturers that bring these sophisticated capabilities together with human expertise create environments where technology generates insight, processes continuously improve, and people continually become more knowledgeable.

At GreenSource Fabrication and GreenSource Engineering, we see the factory of the future as one where technology allows people to do more valuable work, where automation handles repetition, and people apply judgment. AI finds patterns, people ask better questions, experienced professionals share their knowledge, and the next generation brings new perspectives to

established processes. Continuous improvement applies equally to machines, processes, systems, and people.

Ultimately, the smartest factory is the one that gets smarter because of its people. **I-CONNECT007**



Michael Gleason is director of product development and strategic growth at GreenSource Fabrication.

BELOW THE SURFACE

What Engineers Find Challenging About High Frequency Signal Integrity

BY CHANDRA GUPTA, REMTEC

How do materials affect RF signal performance? It's a fair question, and most will assume they already understand the answer. But this misunderstanding is exactly why so many high frequency designs underperform, require rework, or worse, fail in the field.

Below the surface, where signal integrity is either preserved or destroyed, the truth is that at high frequencies, material selection matters, and that's where most engineers get it wrong.

The Myth of Layout Dominance

PCB design has been driven by geometry in the form of trace widths, spacing, stackups, controlled impedance calculations, differential pairs, return paths, and digital and power. It all matters, but the problem is that at RF and microwave frequencies, geometry is no longer the dominant variable. It's ruled by material behavior.

You can design the cleanest layout in the world—with perfect impedance control—and still lose signal integrity if the material itself is unsta-

ble because, at high frequencies, the material defines the playing field long before layout begins.

Why Dielectric Stability Matters More Than You Think

Most engineers know that Dk affects impedance and signal propagation speed, but what's often missed is not that the value of Dk that matters, but how stable that value is. In many traditional PCB materials, especially FR-4, Dk is not constant. It varies with frequency, temperature, resin content, and moisture absorption.

So, what happens? You design for 50 ohms and differential lines of 100 ohms based on a nominal Dk value, but in real-world conditions, that Dk shifts, which also shifts impedance. Then, reflections increase, losses rise, and signal integrity degrades. Your layout wasn't wrong, but your material properties were changing with conditions and frequency.

[Continue reading this column here.](#)



Getting the PCB Quote Right

BY MATT STEVENSON, ASC SUNSTONE CIRCUITS

Once the PCB design moves from the designer's hands to the fabricator, the quote process becomes much more than a matter of price. It's where we determine what it will take to build your board successfully, from materials and capabilities to cost and production requirements. The more closely designers and fabricators work together during this stage, the better the opportunity to identify potential problems before they reach production.

Clear and Open Communication Lights the Way

In many cases, quote requests are straightforward and include everything the fabricator requires to produce quality boards that function as intended. But sometimes more is needed and it takes a healthy back-and-forth to make the design manufacturable.

After the fab team evaluates a PCB design, they often come back with questions for the designer because:

- Information in the quote request does not provide the manufacturer with everything needed to build the board. This is often due to vague or incomplete component information in the bill of materials (BOM).
- The design is too complex to complete before the requested deadline. Complex designs take more time to quote, and back-and-forth is often necessary to ensure accuracy. Designers should build it into their project timeline.
- Specs can be incomplete, omitting details such as surface finish and impedance control requirements.

- Specs can also conflict, such as when the drawing and the Gerber files don't match, or when stated clearances are too tight.

Simply filling out the quote checklist isn't always enough for the fabricator to provide a quote, because standard quote request forms do not fully capture complex requirements such as multiple-via structures or rigid-flex callouts.

This is especially problematic for more complex boards, including UHDI PCB designs. Quote requests should include complete Gerber files, a full BOM, and fab drawings. With those in hand, instead of asking for more information, fabricators can request clarifications and make suggestions for improvement.

Collaboration: The Key to Success

When designers and fabricators work more closely together during the quote process, they align the layout with the manufacturer's specific capabilities, increasing the likelihood of a smooth manufacturing process. Early collaboration can also reduce the chances of delays caused by board redesigns and even lower overall production costs.

Designers learn more about how their fabricator operates. Many PCB designers have never been on the shop floor, so they may not have a complete working knowledge of how a board is built. When designers collaborate with their manufacturing partners, they can better understand manufacturing rules and the manufacturer's capabilities, ensuring their design is more functional, manufacturable, and economical.

Consult with your manufacturer about laminates, as long lead times and higher costs are ongoing issues. To save time and money, your manufacturer should guide you through the laminate selection process and alert you when one laminate type can be substituted for another.

Trust your manufacturing partner. PCB manufacturers often do not know what type of electronic device the boards will power. It makes sense to have a signed NDA between the designer and the fabricator. That way, you can let them know what type of device the board will be part of and what the PCBs need to do for it to function properly.

Key best practice: Don't outsource the quote request

With more complex designs, frequent back-and-forth communication during the quoting stage is essential for success, so the designer should make the quote request and be available to the fab team. When third parties insert themselves into the quote process, collaboration becomes more difficult because those individuals often lack the technical acumen necessary.

Teamwork Translates Into Value

To master the quote process, establish and foster open communication between the designer and the manufacturer. Even if it seems like you're throwing "everything and the kitchen sink" at the fabricator, more information eliminates guesswork. Collaborative efforts turn into design improvement opportunities rather than hasty measures to fill in blanks.

Early collaboration between designers and fabricators is the foundation of total value manufacturing. A total value manufacturer committed to quality and innovative production will offer more options to increase yield and avoid costly rework.

Designing with the quote in mind is how designers can unlock their manufacturing partner's full potential. Working together as a team during the quote process results in higher quality output, produced cost-effectively and on time. **I-CONNECT007**



Matt Stevenson is vice president and general manager of ASC Sunstone Circuits. To read past columns, [click here](#). Read Matt's book, *The Printed Circuit Designer's Guide to ... Designing for Reality* and listen to his podcast series [here](#).

Third Annual Pan-European Design Conference

Important for Europe

The Pan-European Design Conference has quickly established itself as a premier event for the European electronics design community. Now in its third year, PEDC, set for Jan. 27–28, 2027, in Amsterdam, has already attracted a record number of technical paper submissions.

In this interview, Susan Kayesar, a product manager at Siemens who is an active participant in IPC design standards, discusses the upcoming conference and the trends driving its continued growth.

Marcy LaRont: Susan, it's always good to catch up with you. Can you start with some background on how PEDC got started?

Susan Kayesar: European professionals face standards, customer requirements, and industry challenges that are distinct to the region. Many do not have the budget to attend conferences in the United States, and there was no dedicated forum where they could participate in a strong technical program while networking with peers. Before PEDC, the Global Electronics Association hosted

WORLD-CLASS SUPPLIERS

Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, salutes our premier suppliers.

CALL 949.587.3200 FOR MORE INFORMATION



REVOLUTIONARY LAMINATING & BONDING TECHNOLOGY



InduBond® has long been respected for tight accuracy in registration, its lamination presses, automation (automatic loading/unloading), and its eco-friendly best practices.

InduBond® X-PRESS lamination presses utilize a different way of producing the heat required for lamination using standard pressing methods.

The innovation of this technology is that it produces heat on the laminated material, and only the laminated material, of each layer of the press pack at the same time, at the same temperature level, without any thermal conduction.

X-PRESS - Inductive heating is the revolutionary new way of PCB laminates

RFX - High-precision (<10 microns) layer-to-layer registration

230N - Next-generation inductive bonding machine



Proven Performance Partner with InduBond® Presses!
COPPER-ALUMINUM-COPPER & COPPER-STEEL-COPPER
Contact us for more information.

Wintercom, an event focused primarily on standards. The final Wintercom took place in Barcelona in 2023.

The Association now co-hosts PEDC with FED. It was created to bring together a broader cross-section of European professionals and draw on their expertise across disciplines, and was intentionally designed as a pan-European event that would also engage people who might not attend the German-language conference hosted by FED. Our first two events were held in Vienna and Prague, and this year we are moving to Amsterdam. The conference offers an excellent opportunity for European professionals across our industry, and the response has been very positive. PEDC has grown significantly since its inaugural event in 2024.

What is the intent of PEDC? What can attendees expect?

PEDC is designed as a technical deep dive into the challenges shaping Europe's electronics industry. The program explores innovation and new product design, along with the legislative, regulatory, and geopolitical issues affecting the industry, particularly where those challenges are unique to Europe.

We intentionally keep the conference non-commercial and firmly focused on technical content. We are strict about this so attendees can engage in meaningful discussions without being continually subjected to sales pitches. Sessions center on ideas, solutions, trends, and issues relevant to the European community, with the goal of maintaining an academic tone.

Networking is an equally important aspect at PEDC, and it offers distinctive value by connecting professionals from across Europe.

Is there still any focus on standards?

PEDC is not a standards-focused conference. For example, there are no committee meetings, and no standards development work takes place. Of course, the industry naturally revolves around the



Susan
Kayesar

standards, and it comes up in conversation, but they are not an explicit focus of the program content or presentations. Unlike IPC Builds, which revolves entirely around standards building, PEDC emphasizes the exchange of ideas and offers exceptional networking opportunities for professionals across Europe.

What have you learned after your first two years of hosting PEDC? What has changed?

One of our key lessons was the importance of creating a more integrated and intimate expo experience. In the first year, exhibitors were placed in a separate room, which we found limited interaction, so we redesigned the conference layout to allow exhibitors to share the main conference space with attendees.

Last year, exhibitors were located in a central area that attendees naturally entered during session breaks. This encouraged significantly more interaction and created a stronger connection between the technical program and the expo. Attendees could discuss ideas from a session, encounter a relevant booth, and continue the conversation with a supplier—sometimes over coffee with someone who had attended the same session. The arrangement worked very well, and I hope we can use a similar setup this year.

The tech conference sounds very comprehensive. Have you received enough submissions?

Participation has been really good. Submissions were a little slow at first, but we know folks often submit at the last minute. We extended the deadline by a few weeks, and now we have an abundance of submissions, far more than is needed. Since we have only so many slots, we will be curating submissions for our final tech program lineup. Some of them will, naturally, be a better fit for this conference than others, but we are confident we can use all of them somewhere, maybe pushing them over to APEX EXPO.

What's the range of things being covered? How much are you covering system design as opposed to PCB design?

There is a lot of coverage around system design, which includes, by definition, all aspects of design. But the scope of the topics covered is not just, "You should use this trick when you're laying out your copper traces." The discussion goes far beyond that to discuss supply chain, lead times, geopolitical considerations, qualifying new suppliers, single source of truth, and so on. On the production end, we will discuss communication up and down the supply chain as products move from concept to production.

We also touch on design considerations for end-of-life, a requirement in Europe. Those topics might include when products are considered obsolete, how they can be disposed of, and what the cost considerations are. Cost is obviously a significant factor in most product designs, but what does that mean? Are you choosing different suppliers, starting out with a different bill of materials, or are you considering different substrate materials or components?

Designing for end-of-life is not something designers had to address two or three decades ago.

No, certainly not, and now they do. Now, part of designing a system is considering what will happen to it once it has finished its role. Additionally, we discuss designing for different risk factors, IP security for defense contractors, and how to measure compliance against the new standards and technologies.

When will you be releasing more information about the conference and the topics themselves?

The first review period concluded in mid-August, during which we gave an initial response to people who had submitted abstracts. Once we have the final list of abstracts, we will work with those people to ensure their final content meets all our quality criteria. So far, some of the abstracts are really interesting, and they provide insight into the innovation within their companies that nobody is talking about in the general community.

You're right, and innovation in design is always so interesting.

Much of the innovation being showcased is not only about innovative design techniques, but the whole system, becoming more efficient, and how that contributes at a higher level. How are we getting better, and how do we avoid getting worse? How can we leverage the crazy new tools presented by AI?

What are the implications of PEDC for new designers who have workforce concerns?

This conference is very friendly to beginners. It's a great group for newer designers to be a part of, learn from, and meet people. Because PEDC is not huge and anonymous, you see a lot of informal mentoring, and that's nice. Those who are newer to the design community can get a lot more out of this conference than they would at a larger one.

That is good to hear. To wrap up, how do you feel about where electronic design is headed?

We've reached the point where collaboration on information and on design is imperative. We're on the cusp of needing more powerful tools for collaborative design. How do we achieve that, and how do we make it relatively easy for people? Who will provide the most robust and best customer experience? There will always be a trade-off, but the amount of design on every square inch of a board is growing exponentially, so there's a need to involve more than just one product stakeholder. You can't accomplish some of these things by yourself.

The other thing I see is that product and materials lead times will really affect the designs and designers. Lead times are huge right now, so if you make a mistake, you are in trouble. There's a lot of pressure to get it right. The best way to get something right the first time, or in as few revisions as possible, is to get input on how others are doing it, and how it might best be done. Then, maybe, check it six times throughout the design workflow instead of just once at the very end.

That's the perfect place to end. Susan, thank you for providing an update on PEDC. It sounds like a great conference. I-CONNECT007

Start Your Digitalization Strategy With Automation

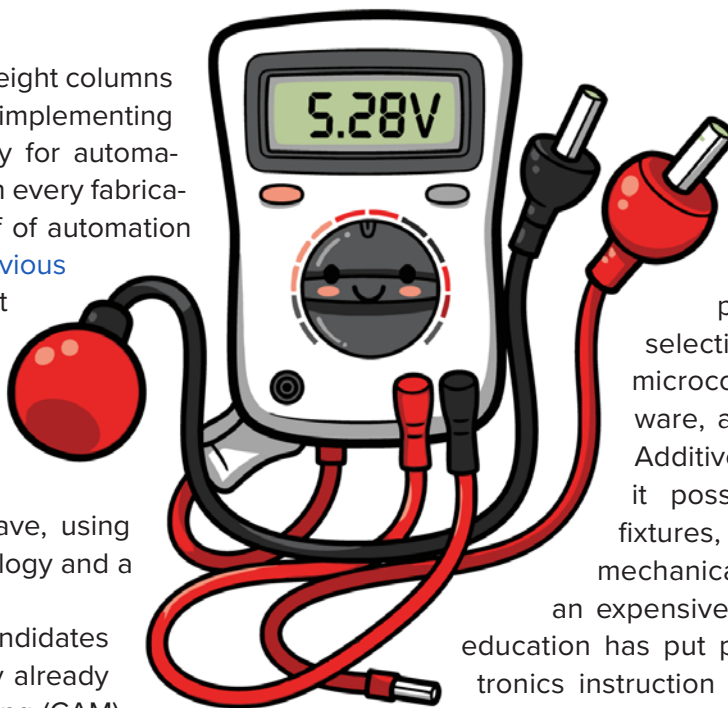
BY HAPPY HOLDEN, I-CONNECT007

This is the first of eight columns on creating and implementing a digital strategy for automation. This does not mean every fabricator needs to hire a staff of automation engineers. In my previous column, I proposed that much of the automation and digitalization needed in PCB manufacturing can be accomplished by employees you already have, using readily available technology and a do-it-yourself approach.

Some of the best candidates for doing this work may already be working in your tooling (CAM) and maintenance departments. They know your processes, understand the equipment, and know where the problems are. With some additional training, they can begin developing the skills needed to automate and improve those processes.

Automation Has Become More Accessible

Automation has come a long way in the past few decades. Unfortunately, much of the commercial equipment that aids in this endeavor is designed for manufacturing high-value products and comes with a price to match. PCB fabricators operate in a different world. A multilayer PCB can sell, on a square-inch basis, for less than doughnuts, although hopefully with better margins.



Fortunately, several developments have made lower-cost automation much more practical. The maker community has produced an enormous selection of inexpensive microcontrollers, sensors, software, and development tools. Additive manufacturing makes it possible to build custom fixtures, housings, brackets, and mechanical components without an expensive machine shop. Online education has put programming and electronics instruction within reach of almost anyone willing to learn.

Put these together, and a PCB fabricator can build some surprisingly capable automation tools without spending a fortune.

Over the next seven columns, I will look at practical DIY projects that can be added to a manufacturing operation. We will begin with sensors and specific gravity measurement, then move into colorimeters for measuring metals, ampere-hour monitoring for plating, conductivity sensors, 3D confocal imaging, low-cost panel transporters, and finally the use of collaborative robotic arms.

The objective is not automation for automation's sake. Each project should solve a manufacturing problem, improve process control, collect useful data, and/or reduce repetitive manual work.

5G

FERMAT

PLASMA SYSTEM

Engineered for
Next Generation PCBs

CLEANER | STRONGER | HIGHER RELIABILITY



5G
HIGHER
FREQUENCY
GREATER
POSSIBILITIES

ADVANCED
PLASMA SOLUTIONS
FOR HIGH-SPEED
COMMUNICATIONS,
AUTOMOTIVE,
AI SERVERS AND
HIGH-RELIABILITY
ELECTRONICS.



LARGER DATA
VOLUME



HIGHER
FREQUENCY



BETTER HEAT
DISSIPATION



IMPROVED
RELIABILITY



HIGHER
PRODUCTIVITY

5G FERMAT KEY SPECIFICATIONS

Electrode Effective Working Area	24" × 44"
Chamber Size	1480W × 1443D × 1010H mm
Footprint (W × D × H)	2610W × 1534D × 2263H mm
RF Power	10 kW
Mass Flow Control	5 Pcs
Vacuum Pump	2200 m ³ /h
Power	380V 33kVA
Dry Air Pressure	0.2 – 0.5 Mpa
Dew Point	< 10°C

OPTIMIZED FOR 5G MATERIALS

- Excellent Uniformity
- Rapid temperature rise
- Processes PTFE, low loss materials, high-speed laminates and more
- Enhances adhesion and reliability
- Supports advanced HDI and high-layer-count PCBs

5G
AI
AUTOMOTIVE
DATA CENTER
NETWORKING
BEYOND

P15V

VERTICAL PLASMA SYSTEM

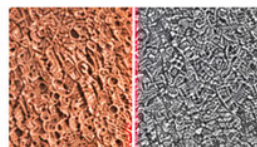
- Vertical electrode design
- Desmear, cleaning and activation
- For PCB, FPC and HDI applications
- Stable process control
- Flexible for multiple materials
- Compact footprint



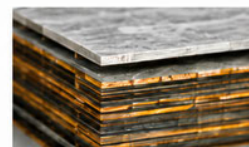
PROVEN PROCESS RESULTS



Via Desmear
& Carbon Removal



Copper Surface
Before / After Treatment



Improved Adhesion
for Advanced Materials

KEY APPLICATIONS



HDI &
Multi-layer PCBs



Automotive
Electronics



AI Servers
& Data Centers



5G/6G
Infrastructure



Semiconductors
& Advanced Packaging

QUALITY. PROCESS CONTROL.
LOCAL SUPPORT. REAL FLEXIBILITY.



Process
Development
Support



Local
Technical Support



Flexible for Your
Applications

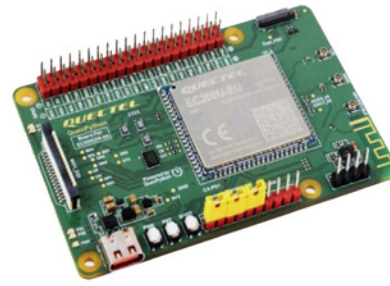


Scalable for
Your Production
Needs

all4-PCB®
NORTH AMERICA



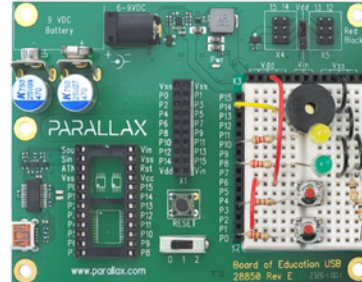
Arduino DUA



QuePython w/network



Arduino UNO



BASIC stamp



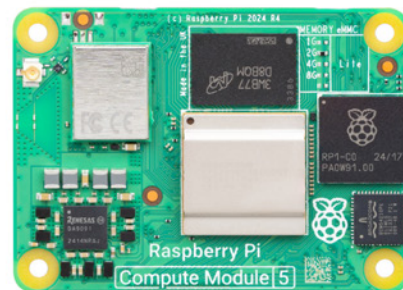
Arduino Industrial Portenta



Raspberry Pi 4 w/network



BBC micro:bit v2 ARM 32bit



Raspberry Pi 4 Compute

Figure 1: A comparison of microcontrollers.

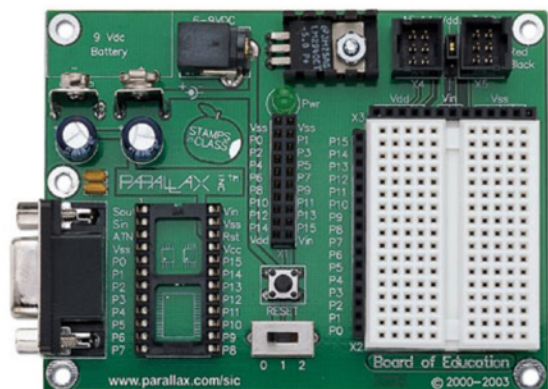
Start With the People You Have

In my experience, the best team to begin this effort probably already works for you.

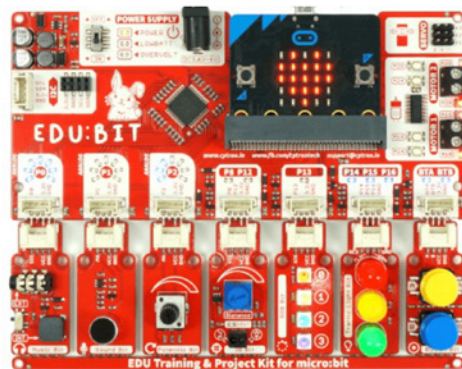
Maintenance personnel understand how the equipment behaves and where it fails. CAM and tooling personnel understand the product, the process requirements, and much of the digital infor-

mation that drives manufacturing. Most importantly, these people already understand printed circuit fabrication.

That experience has considerable value. It can be easier to teach an enthusiastic maintenance or CAM employee some programming and electronics than to hire a new engineer and teach that person



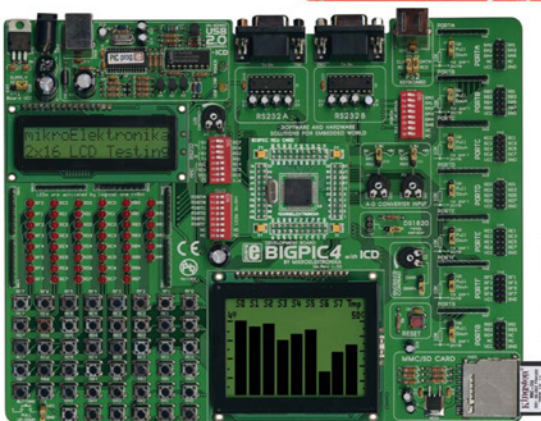
Parallax's Board of Education training of BASICStamp for 14+



EDU:BIT training using BBC Micro:bit, for 11+ Middle School, includes micro:bit V2



Raspberry Pi 4 Starter Kit for Computer Programming Beginner (32GB Card)



User training 'BIGPIC4' for PIC microcomputers and PBASIC.

Figure 2: Training kits provide a hands-on way to learn programming and the fundamentals of working with microcontrollers and sensors.

the complexities of PCB manufacturing.

Look for people who like to solve problems and are curious about how things work. Those are often your best candidates.

Give the Team the Skills It Needs

Once you have identified the team, education comes first. The skills needed for this type of program are achievable, but don't expect everyone to learn everything at once.

The goal is to develop enough knowledge of automation planning, sensors, data acquisition, programming, PLCs, and statistics that the team can begin experimenting. I have included a preparation checklist at the end of this column with the training and reference materials I recommend.

Programming often makes people nervous, but it shouldn't. Your team does not have to become a group of professional software developers. Someone does, however, need to become comfortable enough with programming to customize a microcontroller, collect information from sensors,

and eventually use that information to control a process.

This is where the maker community has changed the game. Inexpensive microcontrollers such as the BBC micro:bit, Arduino, Raspberry Pi, BASIC Stamp, and others give beginners a practical way to learn. They are supported by extensive educational material, software examples, and projects. The important thing is to start using them.

Build by Doing

Learning automation should not become an endless training exercise. At some point, you need to build something. A small training setup allows employees to experiment with a controller, connect sensors, collect data, and see what their programs actually do. As their skills grow, they can add data acquisition equipment, PLCs, and other devices that more closely resemble what they will encounter on the manufacturing floor.

Additive manufacturing provides another useful tool. A small 3D printer can produce sensor



DATAQ DI-1100 DAQ – 4 Ch Analog



DATAQ DAQ – 8 Ch Analog Ethernet



1000V DAQ – 8 Ch Analog Ethernet



Keysight DAQ – 6 digits - 20 Channel



MCC DAQ HATs For Raspberry Pi

Figure 3: Data acquisition (DAQ) systems range from simple, low-cost units and Raspberry Pi HATs to more sophisticated multichannel systems.

holders, brackets, enclosures, guides, and other custom parts needed for automation projects. Instead of waiting for a part to be machined, or deciding an idea is too expensive to pursue, the team can design, print, test, and modify it themselves.

This is one reason I believe a DIY approach can work so well in PCB fabrication. The technology has become inexpensive enough that employees can experiment without every project becoming a major capital request.

Start Small

Don't give in to the temptation with automation to think too big. Don't start by trying to automate an entire production line. Look for a manufacturing problem that is well understood, measurable, and small enough for the team to complete successfully.

A good first project teaches several things at once. The team learns how to select and connect a sensor, write or modify a program, collect data, calibrate the instrument, and determine whether the information is useful for controlling the process.

Just as importantly, employees gain confidence that they can actually build something that works. That first success makes the second project much easier.

In the next column, we will look at sensors and instruments useful for advanced PCB processes, and begin our first project: measuring specific gravity.

PCB fabricators face growing demands for more complex multilayers, HDI, higher yields, and greater reliability. Better process control will be essential to meeting those demands. You don't have to automate everything tomorrow. Start with your people, give them the opportunity to learn, and solve one problem at a time.

See Happy's tips
to get started 

Get Ready for Your First Automation Project

Before beginning the first DIY sensor project:

- **Identify your automation team.** Look first to interested employees in maintenance, tooling/CAM, and other areas who already understand your processes and equipment
- **Download these three I-Connect007 books on automation:** [24 Essential Skills for Engineers](#), [Automation and Advanced Procedures in PCB Fabrication](#), and [The Printed Circuit Assembler's Guide to... Advanced Manufacturing in the Digital Age](#)
- **Complete the ICT “Automation Planning and Essential Skills” training**, consisting of nine one-hour webinars
- **Gather three additional reference books:** a PLC ladder logic manual from AutomationDirect, the *NIST Engineering Statistics Handbook*, and Keysight's *Fundamentals of Sensor Measurements*
- **Choose someone to begin learning programming.** BLOCKS, BASIC, JavaScript, Python, or C++ can be used. For beginners, the BBC micro:bit and Parallax BASIC Stamp provide approachable training platforms
- **Select a microcontroller platform and training kit.** Options include micro:bit, Raspberry Pi, Arduino, BASIC Stamp, PIC-based systems, and QuePython
- **Begin looking at data acquisition equipment** that can interface sensors and existing equipment with your controller
- **Consider adding a 3D printer and basic PLC trainer** as your team's skills and projects develop
- **Get ready for Project No. 1:** Specific-gravity measurement



Happy Holden has worked in printed circuit technology since 1970 with Hewlett-Packard, NanYa Westwood, Merix, Foxconn, and Gentex. He is currently a contributing technical editor with I-Connect007. To read past columns, [click here](#).

An Open Invitation to

STANDARDS DEVELOPMENT

in Paris

The Global Electronics Association is doing something a little different around standards this year. IPC Builds, Sept. 19–24, in Paris, is a series of standards development meetings open to both Association members and nonmembers, offered at no charge, with lunch provided.

Sponsoring an event like this at no charge caught my attention. Removing any registration cost clearly removes a barrier to participation at a time when global input into standards development is needed more than ever. An open forum naturally makes it easier for someone who is curious about standards work or has wanted to get involved but hasn't yet taken the first step to experience the process firsthand, with no risk or expectation.

The most established involvement continues to be in the U.S., where standards were initially created, and the Association has its global headquarters, however, this opportunity is critical for gathering a diversity of voices needed to make global standards legitimately address and serve global needs.

But how much should regionalism matter? In one sense, perhaps it should not. A standard, by definition, speaks to a sameness in content and execution, whatever that may be. But in a global business, different regions, business cultures, and local laws and regulations present unique realities.

There is a distinctiveness to the experiences of European PCB fabricators, assemblers, designers, suppliers, OEMs, and other technical experts, who may have experiences quite different from their counterparts in North America or Asia. There may be differences in the supply chain, customer

requirements, and manufacturing challenges.

Beyond that, translating IPC standards from English into many other languages is important, and the nuances of translation and appropriate word choice and phraseology are areas where the European community will provide important insight for the many languages that region represents. If those perspectives aren't represented in the committee rooms, they are much harder to account for in the final standards, which can lead to unintended negative consequences that must be resolved later.

This meeting will focus on advanced manufacturing guidelines, with agenda topics that include working groups on CO₂e reporting, materials and process traceability, automotive requirements, additive electronics, CFX and Hermes, and high speed/high frequency applications.

It is a misconception that one needs to be an expert in standards to participate in the standards development process. The standards process works better when the people affected by those standards are part of the conversation. Value comes from regular industry professionals who know their field well and are willing to share their knowledge. IPC Builds in Paris is a great opportunity to bring the European community together and provide a forum for their voices to be heard. **I-CONNECT007**

[Click here for more information and to register.](#)



Get the **CapStone™** Advantage



INCREASE THROUGHPUT UP TO **2X!**
REDUCE OVERALL PROCESSING COSTS BY UP TO **30%.**

CapStone™

ACCURACY

MAXIMUM AVERAGE VELOCITY

MAXIMUM DRILLING VELOCITY

POWER CONTROL

$\pm 15 \mu\text{m}$ |M|+3s over entire panel area

>10 mm/s

10,000 mm/s

Precision™ Pulse real-time

Discover the CapStone advantage – www.ESI.com





Stop Inspecting AI and Start Understanding

If you are new to a PCB shop, there is a good chance your first real job is at the back end of the line. Maybe you run the AOI station or pull boards at final inspection and flag the ones that do not pass.

I want you to notice something: Every board you reject has already been built. The copper was plated and the layers pressed before you ever saw it. Your job is to catch the mistake, not to stop it from happening. That is a problem, but not with you. Unfortunately, it has always been the way most shops go about achieving quality.

The Guy Who Figured This Out in the 1950s

Much of what we call Lean and Six Sigma grew out of the work of quality expert W. Edwards Deming. He had a rule: You cannot inspect quality into a

product. By this, he meant that by the time a board reaches inspection, the quality is already decided. Inspection can sort good from bad, but it cannot make a bad board good. If your shop is producing defects, the cause lies upstream in the plating bath, the press, or maybe in the way one shift sets up a machine differently than the other.

Deming's wisdom taught manufacturers to shift their energy toward understanding the process and fixing its gaps and imperfections. If you do that, you'll have fewer defects, and need less inspection.

- **Bad approach:** Make the board, inspect it, find the defect, throw it away, repeat
- **Better approach:** Figure out why the defect happens, change the process, stop making the defect

The road to American manufacturing is paved with PCBs.

PCBs make modern life possible. Every semiconductor needs a printed circuit board and IC substrate to function.

Over the past 20 years, PCB production has moved away from the United States at an alarming rate. PCBAA is the industry's voice in Washington, DC, leading a bipartisan coalition to advocate for investments in this critical industry.

JOIN US TODAY

Click to Watch
**The New
Frontier**
The Documentary

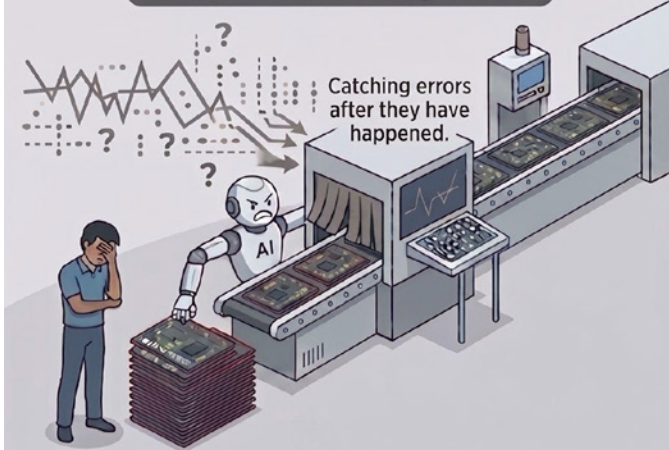


PCBAA.ORG



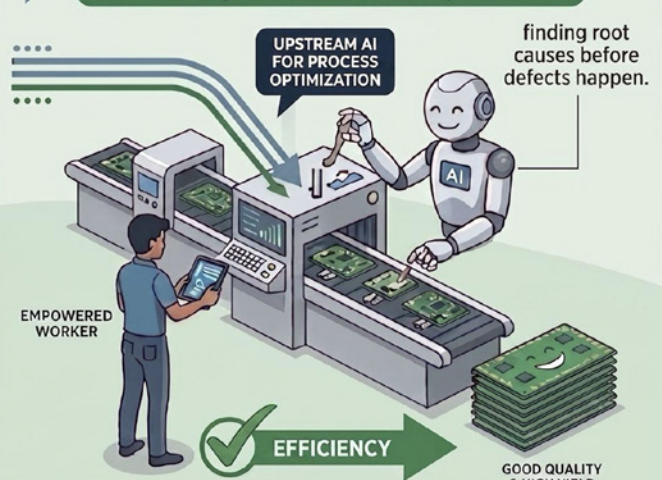
TRANSFORMING PCB QUALITY: Move AI Upstream

CURRENT REALITY:
Focus: End-of-Line Inspection



CURRENT REALITY: End-of-Line Inspection

FUTURE VISION:
Proactive Upstream Process Improvement



FUTURE VISION: Upstream Process Improvement

Everyone agrees to this on paper. Yet almost no one does it in practice, because understanding is hard, and inspection is easy.

Where AI Shows Up

I spent a big chunk of my career on PCB shop floors, and now I spend my days teaching small manufacturers how to use AI to their greatest benefit. When I walk into a shop that says it is "using AI," nine times out of 10, they mean AI-powered inspection. It's a camera with a smart model behind it that can spot a short or a void faster and more accurately than a person with an eye loupe. That is real, and it is good. If you can catch more defects with fewer false calls, you should.

But that machine is simply a better inspector, sitting at the end of the line, finding problems after they have been created. It is Deming's bad approach, done faster. The more valuable use of AI is further upstream, and it does not look like a camera.

The Real Problem Is Not the Defect

Defect data lives in multiple places. The AOI system logs one thing. ERP has its own specific scrap code. First shift keeps a spreadsheet. The second shift may keep a different one, and the plating lead keeps a notebook that never leaves the plating room. One writes "misregistration," another writes "layer shift." A third writes "registration out" while a

fourth writes "reg" and a number. A fifth does not write anything because the traveler was wet. These are four "names" for what is almost certainly the same problem.

However, your computer only sees four separate defects with small counts, and none of them look like a big deal, so nobody chases it. Meanwhile, the total might be the biggest quality problem in the building, which remains hidden because four different people used different words.

A language model earns its keep because it's very good at reading messy human text and saying, in effect, that these four terms appear to describe the same thing. The model can read the logs, spreadsheets, and notebooks once someone photographs the pages, and puts it all into one table with one set of names—a consistent data reference.

Nothing magical happens in that step. No robot fixes the press. But for the first time, an engineer can see the whole pattern and ask the right question: Why does X happen only on the eight-layer jobs, and why did its occurrence double in March? "Seeing the whole system" means getting all the evidence into one place where a human can look at it.

AI Organizes, Humans Make the Call

The engineer decides the root cause, not AI. There was once a case in which researchers used AI to read a pile of engineering notes and failure reports.

You should be in this picture.



When it comes to revitalizing American-made PCB manufacturing, there is strength in numbers, and your voice matters.

Join PCBAA's growing team to help educate, advocate and champion policies and legislation that supports a brighter future for our industry.

JOIN US TODAY



PCBAA.ORG





Who Should Be Holding the Tool

A second lesson from manufacturing history comes from Toyota, where the big idea was not a machine but that the people doing the work should be the ones improving it. Any operator could stop the line. Knowledge stayed on the floor instead of being locked in an office.

Most shops are about to get this wrong with AI. They will either park it in IT or buy it from a vendor that controls it from three time zones away, and unfortunately, the people who actually touch the boards will never get their hands on it.

Think about how a problem travels in a shop like that. An operator notices something odd. She tells an engineer, who

puts in a request to the data team, and three weeks later, there is a report. The operator eventually gets an answer, if she still remembers the question.

Now think about it another way. The operator notices something odd, opens a tool, and pastes in the last two weeks of shift notes. She asks it to pull every mention of that symptom and line them up by date and job number. She takes the result to the engineer, who checks it, and the process gets fixed that week.

Same people. Same problem. The difference is who was trusted with the tool.

The Plater With 20 Years of Experience

Every shop has that person who has run a process so long they can hear when it is going wrong. If the bath is drifting before the meter does, this operator just knows. When that person retires, the shop loses 20 years of knowledge in one afternoon. Everyone says, "We should really write that down," but nobody does. The plating employee does not have time to type out a work instruction, which he hates doing anyway.

This is not a knowledge gap. It is a format gap.

The knowledge is all there, but it's in a form nobody else can use, which is something that AI is good at. You can sit with that plater and have the AI tool interview him: "What do you listen for? What changes when the bath starts behaving badly? What do new people always miss? What do you

They had a strict rule that the model was not allowed to make the engineering decision. Its only job was to organize the information and show where each piece came from. The engineers worked through problems in a fraction of the time, and every claim in their write-up pointed back to a specific record.

That last part matters more than the speed. The word for it is "provenance," which means "the source or origin of something"—basically, can you prove the basis or root of your conclusion?

- **Weak root cause:** "The plating bath caused the failure."
- **Strong root cause:** "The plating bath is the likely cause. These three lots had the same chemistry reading outside the window, and the operator notes on those same lots mention the bath running hot. The thickness measurements shifted at the same time, and so did the rinse conductivity."

The second statement can be verified or refuted by a human who can pull and examine those lots. It used to take a week to find and connect those three pieces of evidence. Now it can take an afternoon, but the engineer is still the critical expertise and decision point behind the AI efficiency. Ask the tool to organize the evidence, but don't ask it to discern the problem or conclusion.

check first when a lot comes back bad?”

The operator is now having a conversation. It is not a writing assignment. The tool takes the answers and organizes them into a draft that could become a work instruction or a training page. He corrects what is wrong, and for the first time, what is in his head is on paper.

Are You New to the Factory?

If you are the 21-year-old who just got hired, I am talking to you. You are probably the least experienced person in the building, and not the one who will make the engineering call. However, you are also the best-positioned person in the building to start this work. I mean that literally.

The senior engineers already know everything, and for years, they’ve been saying that the defect data is a mess and somebody should clean it up and write down what the plater knows. They are right, but they are also buried. They have a customer escape and a quote due Friday. The press is down again, and the cleanup never gets to the top of the list.

You have two things they do not: time that is not yet fully claimed, and a relative comfortability with such tools, having “grown up” with them. You already know how to talk to an AI. But you don’t yet have the process knowledge of the shop. That’s fine, because the plater has it and he will talk to you if you ask with respect. You are the person who can gather the evidence in one place, utilizing the person with the required expertise.

How to Start Now

Do not start with a big project. Take one problem that has been an issue for a long time, like the defect that comes and goes, that nobody has ever nailed down. Ask a supervisor which one it is, and they will (should) know immediately.

Then go collect every place that the problem has been written about: scrap codes, spreadsheets, shift notes, photos of the notebook pages, and the emails from the last time it got escalated. Do not clean anything yet, but gather all of it.

Now, hand it all to an AI tool and ask it to do two things. First, find all the different words people used for this problem and group them. Second, have it tell you what is missing. Where are the gaps in the

dates? Which lots have a scrap code but no note? The gaps are often where the answer is hiding.

Again, do not ask the AI tool about the root cause, though you will be tempted to. Ask it only to organize the evidence and show where each piece came from. Take that output to the person who has run that process the longest (the expert). Show them the pattern. What do they notice that never made it into a log? Interview them, using the tool to help capture what they say.

Then bring the whole package to an engineer and have them make the call. When they reach a conclusion, make sure it points back to the records. If it does not, it is a guess with a nice font.

That is the whole method: Gather it, put it in one set of names, find the gaps, interview the expert, let the engineer decide, and keep the provenance. Repeat on the next “stuck” problem.

One More Time

Inspection is an important part of the overall quality process. Keep your AOI and catch what you can at the end of the line.

But do not confuse better inspection with better quality. A machine that finds a bad board faster is useful. A person, with a tool in their hands, who figures out why you keep making that bad board in the first place is worth far more.

Deming said it 70 years ago, and the industry is still catching up: “Quality comes from understanding and improving the system, not from catching mistakes at the end.” AI finally gives a normal person on a normal shop floor a way to better understand the system. The only question is whether your shop creates the right tools (with AI) and puts the tool in the right hands.

If you are new, those hands just might be yours.

I-CONNECT007

Sean Patterson is the founder of CrossGen AI. He previously served as COO and CTO at Summit Interconnect, held senior multisite roles at TTM Technologies, and was CRO of Nano Dimension, a former Amazon executive, and U.S. Navy submarine veteran.



Find Industry-experienced Candidates at jobConnect007



For just \$975, 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:

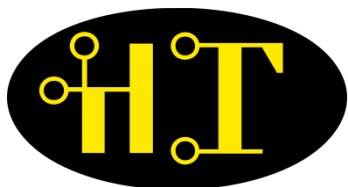
- be featured in at least one of our newsletters
- appear on our jobConnect007.com board, which is promoted in every newsletter
- appear in our monthly [Careers Guide](#), emailed to 26,000 potential candidates

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!

CAREER OPPORTUNITIES



GLOBAL CIRCUITS

Join HT Global Circuits—Build the Future of Advanced PCB Manufacturing

HT Global Circuits is a leading American manufacturer specializing in high-reliability Printed Circuit Board (PCB) fabrication. With domestic facilities in **Milwaukee, WI** and **St. Petersburg, FL**, we deliver mission-critical solutions to the aerospace, defense, medical, automotive, and industrial sectors.

At HTG, we believe our people are our greatest asset. We foster a culture of technical excellence, continuous improvement, and accountability—offering our team the opportunity to work on advanced technologies, solve complex manufacturing challenges, and grow long-term careers within a family-owned business with over 35 years of history.

Open Positions

Milwaukee, WI

- Plant Manager
- Quality Manager
- Quality Engineer
- CAM Engineer
- Maintenance Technician

St. Petersburg, FL

- Quality Engineer
- Process Engineer
- CAM Engineer

How to Apply

Please direct all inquiries and submit your resume to candy.duff@htgpcb.com. Include the position title and preferred location in your subject line.

We look forward to welcoming talented professionals who are passionate about quality, precision, and manufacturing excellence.

Apply Now!



Join Pioneers Systems—Shape the Future of Electronics Manufacturing

Pioneers Systems is a leading Saudi manufacturer specializing in Printed Circuit Board (PCB) fabrication and Printed Circuit Board Assembly (PCBA). Based in Jeddah, Saudi Arabia, we deliver advanced electronic manufacturing solutions to industries including defense, aviation, medical, automotive, industrial, IoT, fintech, and commercial sectors.

At Pioneers Systems, we believe our people are our greatest strength. We foster a culture of innovation, collaboration, continuous improvement, and technical excellence, providing employees with opportunities to work on advanced technologies, solve complex challenges, and grow their careers in a dynamic manufacturing environment.

Open Positions

- Senior PCB Production Manager
- Senior PCB Quality Engineer
- Senior PCB Process Quality Engineer
- Senior PCBA Process Quality Engineer
- Senior Process Engineer – Dry Process
- Senior Process Engineer – Wet Process

How to Apply

Please submit your application through our Career Portal.

We look forward to welcoming talented professionals who are passionate about quality, innovation, and manufacturing excellence.

Apply Now!

CAREER OPPORTUNITIES



FAE Wanted

San Francisco Bay Area, United States

PCB Recruitment Ltd—the **global PCB and electronics industries recruitment specialists**—are working in partnership with our client to source a world-class FAE in the Bay Area.

Outstanding Field Application Engineers in the San Francisco Bay Area – we have an exceptional opportunity that's well worth your attention.

Join one of the world's leading manufacturers of high-performance server PCBs at a pivotal stage of growth.

We're looking for someone with **specialist expertise** in high-speed signal PCBs and experience supporting AI infrastructure, GPUs, 800G–1.6T networking, or similar high-performance computing applications.

Salary: \$250,000–\$300,000 plus an excellent, industry leading benefits package.

For a confidential, no-obligation conversation, contact william@pcbrecruitment.co.uk.

This opportunity is not to be missed and could be an immense step in your career.

Apply Now!



Multiple Positions

At Arlon EMD, we support aerospace, defense, and next-gen technology. We're looking for people who show up, take ownership, and get the job done on the production floor and beyond. No fluff. Just real work, real impact, real opportunity.

Now hiring for roles like:

- Field Application Engineer
- Technical Sales Manager
- Maintenance Technician
- Machine Operator
- Customer Service Representative
- Accounting Manager
- Senior Accountant
- Entry-level Sales Engineer

Arlon is a major manufacturer of specialty high performance laminate and prepreg materials for use in a wide variety of PCB (printed circuit board) applications. Arlon specializes in thermoset resin technology including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems.

Apply Now!

CAREER OPPORTUNITIES



Buyer/Purchaser

At NCAB Group, USA, we are hiring for an Electronic/Component **Buyer/Purchaser**. If you have experience in EMS, Component and Printed Circuit Board Assemblies (PCBA) then this is the role for you! Do you have a strong history with negotiating with distributors? Do you have strong communication skills? Join our growing team in our **San Jose, CA** office and help us become the *Number 1 PCB Producer—wherever we are*. Apply today!!

You will have the following experience

- Purchasing experience in the EMS sectors
- Quoting experience (BOM)
- Sourcing experience
- Experience with ERP system management
- Knowledge of most prevalent vendors
- Working knowledge of VLOOKUP (Excel experience)

What you will bring to us:

- Bachelor's degree in Supply Chain Management, preferred but not required in lieu of experience
- Proven experience in purchasing, particularly with a focus on PCBA components
- Strong negotiation skills and the ability to build lasting supplier relationships
- Familiarity with quality standards and specifications for PCBA components
- Excellent organizational and communication skills.
- Team player, you have the ability to work with a variety of teams
- Likes to have fun and WIN!

Apply Now!



Inside Sales to Electronics and Semiconductor Manufacturers

MacDermid Alpha Electronics Solutions is seeking an inside salesperson based in New Jersey, Chicago, or the Bay Area to sell to circuit board assembly and semiconductor package manufacturers. Our materials include soldering pastes, die attach adhesives, polymers, chemicals, and other engineered assembly products.

This is not a customer service position. We are seeking a hunter who is driven to proactively drive new business with technical buyers.

Job Responsibilities

- Service and grow our business with existing customers
- Proactively seek business at new companies with little or no engagement history
- Lead management

Requirements & Qualifications

- Bachelor's degree required
- Experience in using a CRM tool is required, preferably Salesforce
- 5 years of business development experience
- Preferred locations: Our offices in Itasca, IL, Santa Clara, CA, or Piscataway, NJ

Applicants may also reach out directly to:

Keith Furr, Commercial Director, at
Keith.furr@macdermidalpha.com.

EEO Statement

Element Solutions Inc group of companies are E-Verify Companies and provide reasonable accommodation for qualified individuals with disabilities and disabled veterans in job applicant procedures. Equal Opportunity Employer: Minority/Female/Veteran/Disabled/Gender Identity/Sexual Orientation.

Apply Now!

CAREER OPPORTUNITIES



ventec
INTERNATIONAL GROUP

Director – OEM Sales and Business Development Full-time, USA

Ventec International Group is a global leader in high-performance PCB materials, serving aerospace, defense, AI, automotive, industrial, medical, and high-speed digital markets.

We're seeking an accomplished sales leader to drive OEM business across the United States. This is a business development role focused on winning design-ins, building executive relationships, and securing long-term strategic partnerships with leading electronics companies.

Key Responsibilities

- Win new OEM opportunities across Ventec's product portfolio
- Develop relationships with OEMs, PCB manufacturers, and technology partners
- Drive specification wins in advanced electronics applications
- Lead and develop our manufacturer representative network
- Partner with technical teams to deliver customer solutions
- Contribute to commercial strategy, forecasting, and market intelligence

About You

You have at least five years' experience selling to OEMs within the electronics or PCB industry, with a proven record of winning new business and delivering sustained growth. You're commercially driven, confident engaging senior decision-makers, skilled at solution selling, and able to understand technical products while working independently and within global teams.

Why Ventec?

You'll work alongside senior leadership, influence major global programmes, and help shape the future of advanced electronics while making a direct commercial impact.

Apply Now!



Senior Thick Film & Microelectronics Quality Engineer

Remtec is growing and has an exciting opportunity at our dynamic, team-oriented tech-manufacturing company, based in our newly renovated headquarters in Canton, MA, just south of Boston.

The Senior Thick Film & Microelectronics Quality Engineer ensures product quality and AS9100 compliance across thick film fabrication and microelectronics assembly, including printing, firing, laser trimming, plating, wire bonding, soldering, assembly, testing, and inspection.

Responsibilities include:

- managing nonconforming materials
- leading root cause investigations
- conducting audits
- developing control plans, PFMEAs, and inspection procedures to ensure customer and regulatory requirements are met

The role partners with production and suppliers to establish quality standards and improve supplier performance and yield, reduce waste, and support continuous improvement overall.

The ideal candidate has a bachelor's degree in engineering or a related technical discipline, at least three years of quality engineering experience in microelectronics, semiconductor, or PCB manufacturing, and hands-on knowledge of thick film processes, quality systems, and problem-solving.

Preferred skills include AS9100/ISO 9001 systems; nickel, gold, and silver plating and surface finishes; GEA and MIL testing and inspection standards; and statistical software.

**Join Remtec and contribute to a collaborative
culture focused on quality, integrity, and innovation.
Learn more and apply here:**

Apply Now!

CAREER OPPORTUNITIES



TECHNICA, U.S.A.

Various Positions

For 40 years, Technica USA has been providing products, equipment, and services to the printed circuit board fabrication and assembly markets.

Working with our worldwide partners, we offer our customers solutions through best-in-class product lines.

Technica has offices in San Jose, Calif, and Rancho Cucamonga, Calif.

We are expanding and looking for highly qualified Business Development/Account Managers for both the PCB and PCBA markets.

We are adding to our growing national equipment service coverage and looking for experienced Equipment Service Technician/Engineers.

Are you a PCBA equipment applications expert with experience in component placement and inspection? We are looking for Equipment Product Specialists to work within our San Jose, Calif., PCBA Equipment Demo center.

Please visit www.technica.com/careers to learn more about these positions and submit your resume today!

Apply Now!



eptac
TRAIN. WORK SMARTER. SUCCEED.

Instructor – Electronics Manufacturing

EPTAC is seeking experienced professionals from the electronics manufacturing industry to join us as Instructors. This role offers flexible part-time or full-time opportunities. Ideal candidates will have expertise in soldering, quality control, PCB design, or related fields.

Responsibilities:

- Teach IPC Certification and electronics manufacturing related courses
- Travel as needed to teach at EPTAC locations or customer sites across the U.S.
- Contribute to the development of new training programs and enhance existing courses
- Stay current with industry trends, emerging technologies, and best practices to refine instructional techniques
- Provide hands-on demonstrations and supervise use of industry tools
- Foster a positive learning environment, enforcing classroom and safety rules

Requirements:

- 10+ years experience in the electronics manufacturing industry
- Current or past IPC Certification
- Bachelor's degree or equivalent
- Excellent communication, instructional, and organizational skills
- Practical, quick, and hands-on approach to problems
- Unrelenting drive to understand and meet student needs

EPTAC is the leader in Electronics Manufacturing Training and Certification in North America with 24 locations. Our headquarters is in Salem, NH. We offer a modern, positive environment with benefits including health care, paid time off, retirement savings, and professional development.

Apply Now!

CAREER OPPORTUNITIES



Independent Sales Representatives Wanted

P. Kay Metal / MS2 Technologies, headquartered in the United States, is seeking established **independent sales representative firms** to add a proven, patented product to their portfolio.

We manufacture and supply **MS2® Molten Solder Surfactant**, a patented solution designed for both **Tin/Lead and Lead-Free soldering processes**, delivering measurable performance and cost benefits to electronics manufacturers.

MS2® already has successful customer installations throughout the United States, as well as a growing international footprint with active customers in Mexico, El Salvador, Honduras, Ecuador, and the Dominican Republic. We ship globally and support customers worldwide.

This opportunity is ideal for rep firms currently calling on electronics manufacturing, PCB assembly, soldering, or process engineering accounts who are looking to expand their offering with a differentiated, high-value consumable.

Strong technical products, international growth potential, and factory support provided.

To learn more about our company and technology, visit **www.pkaymetal.com**.

Interested firms are encouraged to reach out for additional details to Gladys Flores at gladysflores@pkaymetal.com.

Apply Now!



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.

TTCL 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)

TTCL 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!

CAREER OPPORTUNITIES



Quality Assurance Specialist—Bare Board PCB Manufacturing

Accurate Circuit Engineering seeks an organized, technically proficient Quality Assurance Specialist dedicated to bare board PCB production. You will champion compliance with industry standards, lead internal audits, manage certifications, and drive continuous improvement based on product performance data and customer feedback.

Key Responsibilities:

- **Standards Compliance & Certification:** Enforce IPCA600, IPC6012 (CIS/CIT preferred), and ISO 9001 quality standards throughout fabrication
- **Internal Process Auditing:** Conduct scheduled and ad hoc audits of incoming materials, fabrication stages, testing protocols (etest, AOI), and documentation traceability
- **Employee Training & Development:** Create and deliver training programs for inspectors and production staff on IPC standards, quality procedures, and inspection tools to maintain certification
- **Failure Analysis & Corrective Actions:** Investigate nonconforming boards—including internal findings and customer returns/RMAs—analyze root causes, and lead corrective/preventive actions (8D/CAPA)
- **Procedure Optimization:** Analyze quality trends and RMA data to update processes, inspection checklists, and control plans

Qualifications:

- Associate degree or equivalent experience in electronics manufacturing
- 3+ years in bare board PCB QA, with IPCA600/CIS and IPC6012 certification
- Strong auditing, training, documentation, and cross-functional collaboration skills
- Proficient in rootcause failure analysis

Join us to ensure rigorous compliance, elevate fabrication quality, and continuously improve manufacturing standards.

Contact brandon@ace-pcb.com and James@ace-pcb.com to apply.

Apply Now!



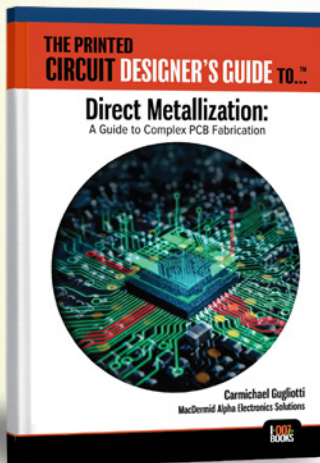
Are You Our Next Superstar?!

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 to 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!

EDUCATIONAL RESOURCES



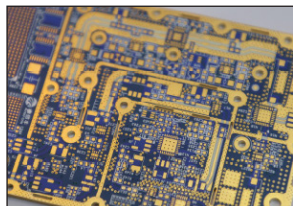
BOOKS

“A clear and practical perspective on direct metallization for today’s innovators working to advance PCB technology.”

— Jerry Magera, Sr. Principal Engineer, RTX

[VIEW BOOK](#)

WHITE PAPERS



Can ENIG & ENEPIG Process Be Sustainable?

by MacDermid Alpha Electronics Solutions

Gold reduction, precise XRF measurement, longer bath life, and lower-temperature processing can make ENIG and ENEPIG finishes significantly more economical.

[↓ DOWNLOAD](#)



PCB Cost Drivers

by NCAB Group

Learn how smarter design, material choices, sustainability, and collaboration can reduce costs and waste without sacrificing reliability.

[↓ DOWNLOAD](#)

PODCASTS



Engineering What's Next

with Jake Joo, Qnity

[LISTEN NOW >](#)



Meet the Author: UV Conformal Coatings

with Brian Chislea and Cody Schoener, Dow

[LISTEN NOW >](#)



PCB Materials: The Backbone and Future of Electronics

with Sean Mirshafiel, Isola

[LISTEN NOW >](#)

I-Connect007

The Global Electronics Resource

EXECUTIVE DIRECTOR | MANAGING EDITOR

MARCY LARONT

marcy@iconnect007.com • (480) 280-5229

Associate Editor

MICHELLE TE

michelle@iconnect007.com

Technical Editor

PATTY GOLDMAN

Contributing Technical Editor

HAPPY HOLDEN

happy@iconnect007.com

Business Development Manager

BARB HOCKADAY

barb@iconnect007.com

Marketing Services Manager

TOBEY MARSICOVETERE

tobey@iconnect007.com

Digital Marketing Manager

BRITTANY MARTIN

brittany@iconnect007.com

Art Director | Production Manager

SHELLY STEIN

Magazine Layout

TYLER ZAPOLUCH

Ad Design

TOBEY MARSICOVETERE

Director of Digital Operations

and Media Systems

BRYSON MATTIES

Cover Design

SHELLY STEIN

Cover Image

ADOBE STOCK © ADRIFERRER

I-CONNECT007 MAGAZINE®

is published by IPC Publishing Group, Inc.,

3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

© 2026 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.

September 2026, Volume 1, Number 9

I-CONNECT007 MAGAZINE is published monthly, by IPC Publishing Group, Inc., dba I-Connect007.

ADVERTISER INDEX

Accurate Circuit Engineering	7
All Flex Solutions	59
all4-PCB	99
ASC Sunstone.....	63
atg Mycronic	51
Candor Industries	73
Chemcut	39
Electra Polymers	87
Elite Materials.....	49
Flexible Circuit Technology.....	69
Global Electronics Association	41, 89
GreenSource Engineering	79
I-007eBook	2, 45
Insulectro	5, 17, 19, 95
IPS	13
Isola	11
Matrix USA	77
MKS' Atotech.....	60, 61
MKS' ESI	105
On the Line with.....	3
PCBAA	107, 109
Pluritec	21
Polar Instruments	83
Qnity	37
Schmoll America	57
Technica USA	65
United Electronics Corporation.....	29
US Circuit.....	33
USA Microcraft.....	9
Ventec International Group.....	53



Problems solved!

