October 2026

THE MAGAZINE I-Connect007

INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY



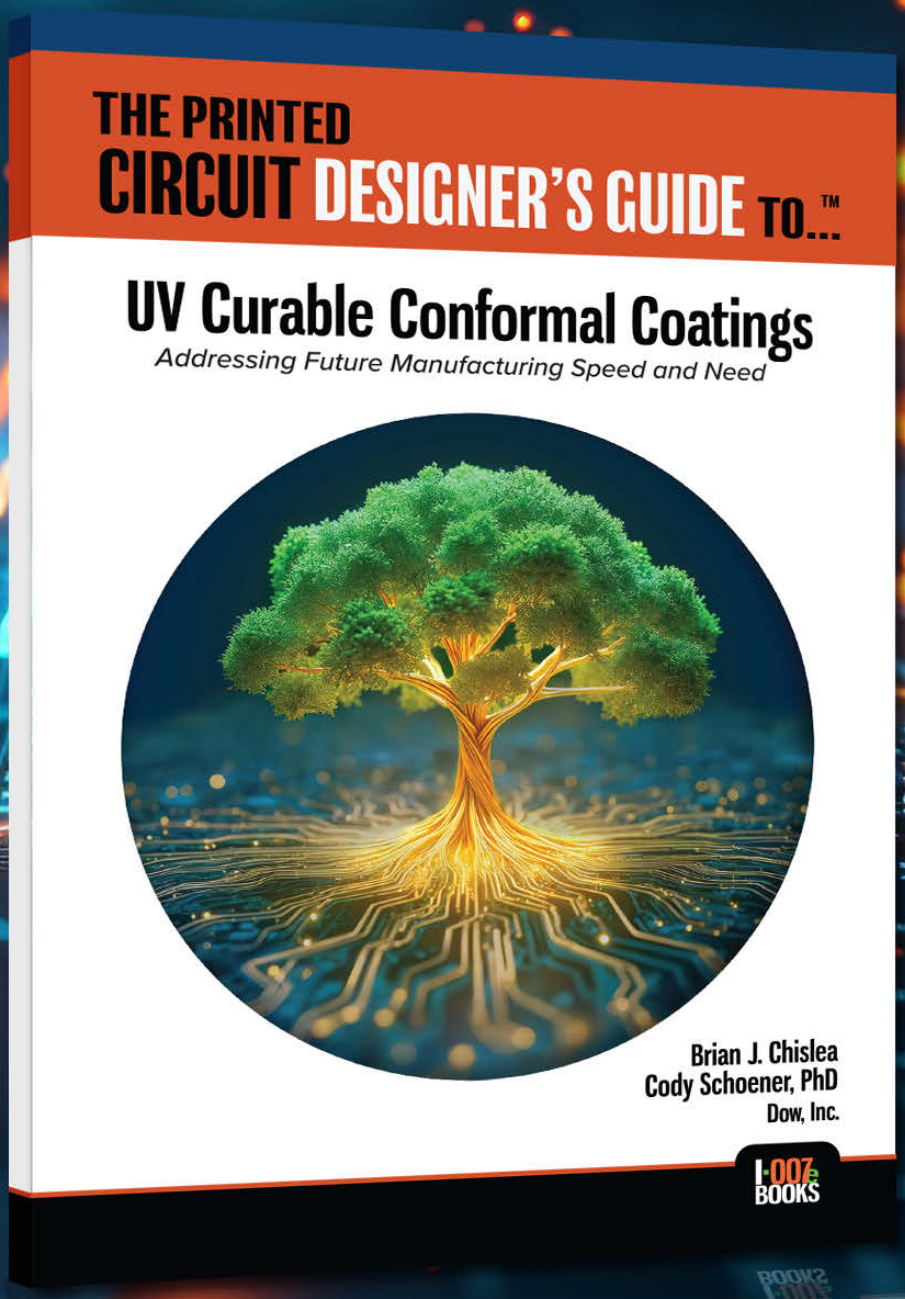
Enabling the Interconnect Technologies That Transform Our World

MKS Inc.

Plus: Q&A with MKS Inc. CEO and President, John T.C. Lee

PCB007
DESIGN007

Protection That Performs



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.

ON THE LINE WITH... I·Connect007

Listen on Apple Podcasts

Listen on Spotify

LISTEN NOW!



Inside MKS Inc. and the Challenges Ahead

The October issue spotlights the work of MKS Inc., Thailand's factory expansion, and practical guidance on design and manufacturing challenges, from signal integrity and ceramic substrates to putting AI into factory operations. Our columnists have some fun by talking about what haunts designers and manufacturers.

Spotlight

12 **Enabling the Interconnect Technologies That Transform Our World**
MKS Has a Strategic Role in Powering the Future of AI and HPC
by Dr. John T.C. Lee, et al.

16 **Q&A: Optimizing the Interconnect with Dr. John T. C. Lee**

From the Editor

08 *Marcy's Musings*
A Frighteningly Good October Issue
by Marcy LaRont

I-Connect007 THE MAGAZINE
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

PCB007

Features

The Right Approach
46 **The Top 5 Scariest Things About AI in PCB Fabrication**
by Steve Williams

The Chemical Connection
94 **A Few Things That Still Scare Me**
by Don Ball

Articles

28 **The Supplier's Job Has Changed: Turning PCB Supply Chain Uncertainty into Action**
by Dan Thau

102 **Turn Your AI Into Action: Interview Your Plater**
by Sean Patterson

Interviews

40 **The Need for Speed in Mil/Aero Manufacturing**
with James Hofer

Columns

American Made Advocacy
56 **While Congress Debates, America's PCB Competitors Invest Billions**
by David Schild

Trouble in Your Tank
96 **Are PCB Fabricators Ready for the Next Technology Curve?**
by Michael Carano

INSULECTRO



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

EMC
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 Temperture)**
- **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)**

Thailand Factory Series

by Tulip Gu and Marcy LaRont



72 FSQuality in Thailand: From Automotive and Industrial Controls to AI and Optical Modules



80 Sundell Technology Laying the Groundwork for a 'Certain' Future



88 Delton Turns Its Thailand Location Into a Rules-driven Smart Factory



24 *Elementary, Mr. Watson*
A Nightmare on PCB Street
by John Watson

In Every Issue

54 MilAero007 Highlights
112 Career Opportunities
120 Educational Resources
121 Advertiser Index

I-Connect007 THE MAGAZINE
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

DESIGN007

Articles

50 Smart Dental Retainers With Flex PCBs
by Anaya Vardya

108 IPC Standards Released for Q3 2026
by Global Electronics Association

Columns

Beyond Design
34 Inductance: The Silent Architect Driving SI and PI
by Barry Olney

NEW *Built for What's Next*
20 The End of Flat Thinking
by Michael Gleason

Flex Connections
60 Is a Printed Silver Ink Circuit Right for Your Application?
by Terrill Schmidt

Powering the Future
64 Why Engineers Are Switching to Ceramic Substrates
by Brian Buyea

Connect the Dots
70 Design With the Quote in Mind
by Kelly Atay



QUICK TURN RF, MICROWAVE, ANTENNA EXPERTS

ACE
ACCURATE CIRCUIT ENGINEERING

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 *Frighteningly* Good October Issue



BY MARCY LARONT, I-CONNECT007

This month, we spotlight MKS Inc., which is expanding its role as an essential supplier to the PCB, assembly, and semiconductor supply chains. In addition to my interview with President and CEO John T.C. Lee, we examine the company's efforts to develop technologies that bring high-end solutions to real-world manufacturing challenges. Together, they offer a powerful look inside an important industry company.

I also want to share some thoughts on my summer travels. In June, I went to Lithuania for the EIPC Summer Conference and to visit Teltonika's impressive manufacturing operation. In August, I attended the Thailand Electronics Circuit Asia (THECA) show, where I spoke with the leaders of several PCB companies through interviews and tours.

These are very different geographies and markets, but their energy is palpable as they tackle

CBTech®

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
3σ ≤10μ alignment accuracy

3 Cost Effective
Advanced automation minimizes labor costs

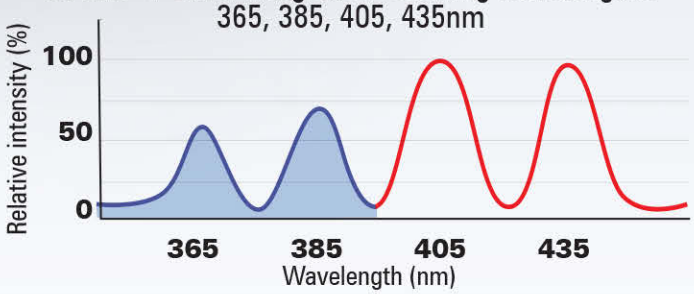
4 Smart Automation
Seamless integration with MES systems

5 Excellent Product Traceability
Multiple tracking methods to enhance QC

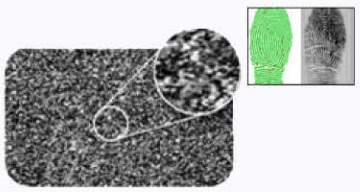
6 User-friendly Interface
Supports industry-standard data formats

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

the challenges in front of them. In Europe, that has meant strengthening the domestic supply base. In Thailand, companies are positioning themselves strategically within an increasingly complicated geopolitical and import/export landscape.

I came home from both trips energized and impressed by their enthusiasm, vision, and optimism, and what they have accomplished in a short time.

Read more about my visit to Lithuania in the July issue of I-Connect007 Magazine. In this issue, learn about three Chinese companies investing in Thailand as they execute remarkably fast factory ramp-ups and move toward profitable operations. And stay tuned for coverage on Thailand's considerable skilled-workforce shortage, looking at a strategic partnership through the prestigious Thai university MUT, with THPCA, and the Thailand Board of Investment.

Of course, October also means Halloween, and our columnists got into the spirit.

John Watson has a "nightmare" story about what PCB designers fear most—or perhaps should fear. Don Ball offers some real talk about the industry "fears" that I suspect many of us are feeling right now. Steve Williams writes about the scarier side of implementing AI in our factories, identifying five things manufacturers should keep in mind as we head down the seemingly primrose path of AI. His reminder: Quality systems still demand rigor.

On the technical side, Barry Olney goes deep into inductance and its effects on signal and power integrity, arguing that understanding these relationships can mean the difference between a board that merely functions and one that performs reliably at multi-gigabit speeds. New columnist Kelly Atay of ASC Sunstone helps PCB designers understand why they must keep the quote in mind at the beginning of the process.

Anaya Vardya writes about a topic I didn't see coming: dental retainers. These aren't the ones you wore in middle school; these are "smart retainers" that incorporate tiny flexible sensors capable of monitoring insertion time, bite force, inflammation, infection, and even biomarkers in saliva. Serious cool factor.

In "Powering the Future," Brian Buyea gets down to brass tacks about the ability of ceramic substrates to provide connectivity while managing the heat generated by increasingly powerful electronics. On the U.S. manufacturing front, PCBAA points to real-world examples of prominent American PCB manufacturers succeeding with government support, and makes the case that much more is needed. Michael Carano challenges PCB fabricators to examine whether their processes keep them ahead of the technology curve, and we welcome new columnist Michael Gleason from GreenSource Fabrication, who will help engineers design better by thinking differently and planning for future possibilities in PCB fabrication.

Also, I catch up with Accurate Circuit Engineering's James Hofer, whose enthusiasm and expertise in PCB prototype fabrication are matched only by his insights into the evolving demands of the U.S. high-reliability defense and aerospace markets.

We're continuing our series from Sean Patterson about how to put AI into action on the shop floor. Dan Thau of Millennium Circuits writes about a type of supply chain "insurance policy" called SupplyShield.

As you can see, October has more to offer than pumpkin spice and indulging in your favorite seasonal scare. Our industry is operating at 110% at all times, a constant reminder of how exciting it is to be here and of the privilege of telling some of your stories. **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.



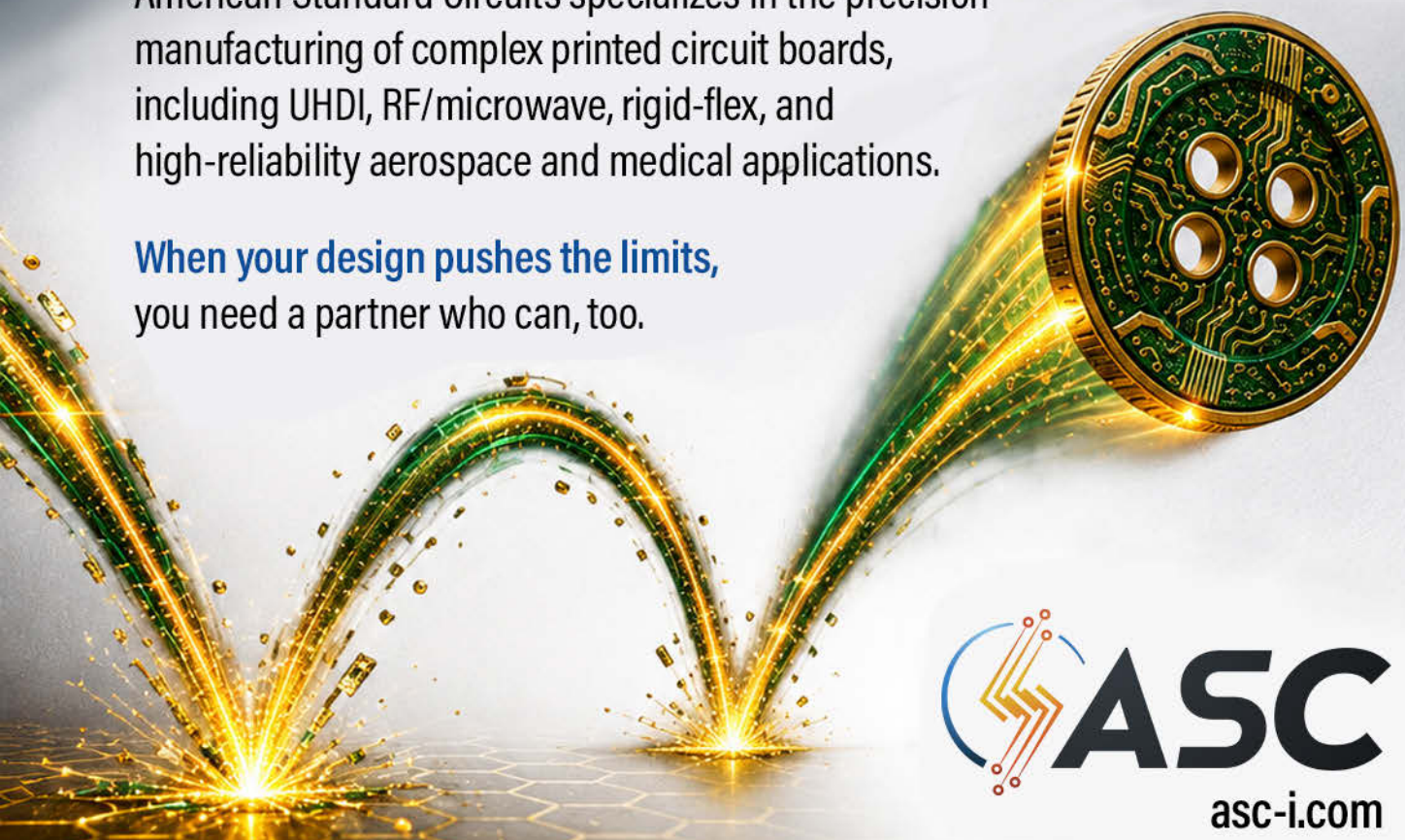
Stressed By TIGHT Tolerances?



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 UHDI, 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
asc-i.com

Enabling the Interconnect Technologies That Transform Our World

MKS Has a Strategic Role in Powering the Future of AI and HPC

BY JOHN T.C. LEE, MKS PRESIDENT AND CHIEF EXECUTIVE OFFICER
WITH WON LEE (VSD), MARC TRICARD (PSD), AND KULDIP JOHAL (MSD), CHIEF TECHNOLOGY OFFICERS

AI IS RESHAPING INDUSTRIES, ECONOMIES, AND DAILY LIFE at an unprecedented speed. From generative AI and autonomous systems to high-performance computing (HPC) applications in science, defense, and advanced manufacturing, the demand for computational power is accelerating exponentially. But while algorithms capture the spotlight, the true enabler of this transformation lies deeper in the physical infrastructure that makes AI possible.

Performance in the AI era is no longer defined by transistor shrink alone. It is increasingly defined by how efficient systems are integrated, interconnected, cooled, and manufactured at scale. As Moore's Law slows, innovation has shifted from simple transistor scaling to advanced packaging, heterogeneous integration, and precision interconnect engineering. In this new paradigm, packaging is performance.

At the center of this transformation is the interconnect stack, from high layer count PCBs and HDI routing to package substrates, silicon interposers, hybrid bonding, and high bandwidth memory (HBM). Every layer must operate in perfect harmony. Signal integrity, thermal stability, yield, dimensional control, contamination mitigation, and atomic-level surface precision all determine whether AI systems scale efficiently or stall under complexity.

MKS stands uniquely positioned at this intersection of materials science, photonics, and vacuum engineering, enabling the foundational technologies that power next-generation AI and HPC infrastructure.

The Shift: From Transistor Scaling to System Integration

For decades, performance gains were driven primarily by transistor miniaturization. Today, the scaling frontier has moved outward. Advanced packaging now integrates multiple specialized dies—logic processors, graphics processing units (GPUs), AI accelerators, and stacked HBM—into tightly coupled system-in-package architectures.

These architectures demand:

- Ultra-fine redistribution layers
- High aspect ratio microvias and through-silicon vias (TSVs)
- Sub-5-micron line and space geometries
- Hybrid bonding for wafer-to-wafer and die-to-wafer connections at near-atomic precision
- Controlled copper deposition with minimal stress
- Extreme thermal and power density management

The performance of AI hardware increasingly

depends on how precisely these elements are manufactured. A single micron of misalignment, a microscopic void in copper, or slight warpage in a substrate can have a significant impact, e.g., compromised yield, efficiency, and reliability.

This is where MKS plays a strategic role, not just as a supplier of critical subsystems or materials, but as an integrated enabler for holistic solutions across the interconnect ecosystem.

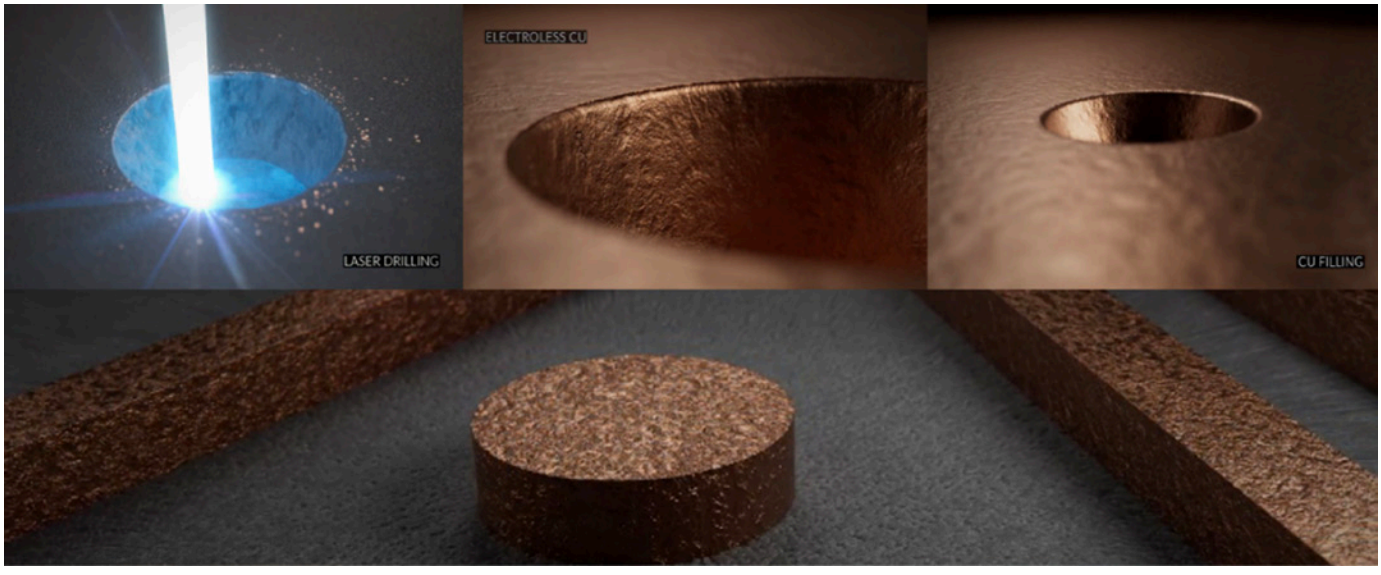
Building the AI Stack From the Ground Up

AI infrastructure begins at the foundation: the high layer count multilayer PCB. These boards must route massive power and high-speed signals across dozens or even hundreds of copper interconnects. As AI workloads intensify, current densities rise, and signal margins shrink. Extremely high precision laser drilling, high uniformity copper plating, and advanced process control become essential to ensure reliability at scale.

Above this foundation lies HDI technology, where routing compresses into microscopic geometries. Microvias shrink to dimensions smaller than a human hair. Even the smallest defect can impact bandwidth or long-term reliability. MKS advanced surface treatment, laser drilling, metallization, and plating chemistries and systems enable small features, high accuracy, selective material removal, ultra-pure copper deposition, consistent via filling, and uniform grain structure, which is critical for maintaining electrical performance under extreme conditions.

The package substrate represents the next critical bridge connecting board-level routing to silicon-level precision. As I/O counts increase and HBM stacks grow taller, overlay accuracy and warpage control become defining challenges. Sub-5-micron geometries require highly controlled plating environments, precision laser-via formation, and contamination-free surface preparation. Through its integrated surface treatment and plating, laser processing, and vacuum-enabled plasma technologies, MKS supports scalable substrate manufacturing for AI and HPC applications.

At the heart of AI bandwidth scaling sits the silicon interposer. Acting as the high-density bridge between GPU logic and stacked memory, interposers demand ultra-fine redistribution layers and deep via formation. As the industry transitions from traditional microbumps to Cu-Cu hybrid bonding, process and systems precision intensifies. Hybrid bonding requires atomically clean, ultra-flat copper surfaces and tightly controlled deposition environments. The MKS high-precision hybrid bonding system, advanced copper plating technologies, and precision vacuum instrumentation enable robust, low-resistance interconnects at the highest



densities, unlocking bandwidth performance critical to AI training and inference, while driving throughput.

Finally, at the top of the stack reside the GPU die and HBM towers, where power density peaks and data throughput explodes. Thermal management becomes mission-critical. Uniform deposition, precise thin-film control, and contamination-free processing directly impact efficiency and long-term reliability. Here again, MKS has combined capabilities across vacuum control, photonics, and advanced materials enable stable, repeatable manufacturing processes that withstand the escalating demands of AI hardware.

Integration as Strategic Differentiation

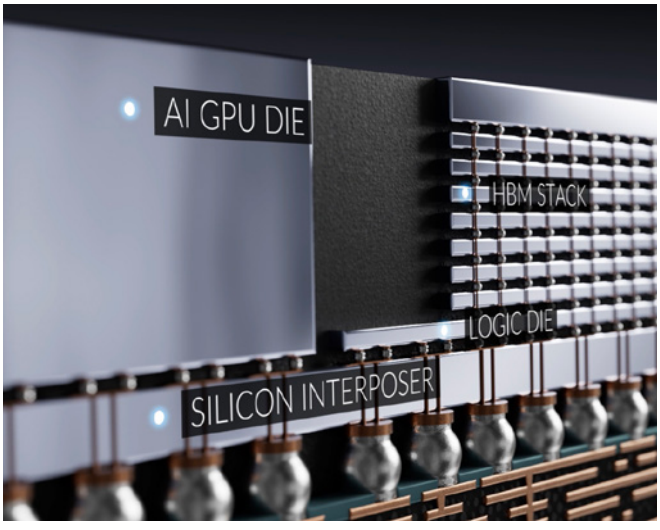
Many companies contribute individual technologies to AI infrastructure with production equipment/subsystems, specialty chemistries, laser technologies, vacuum solutions, process control and analytical instruments, or metrology systems. The MKS strategic advantage lies in its ability to integrate and optimize across these domains.

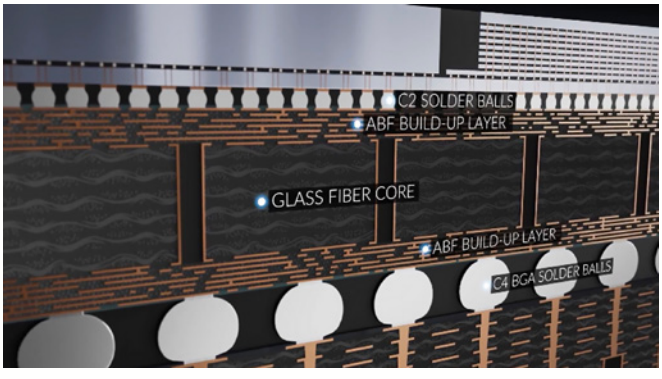
Through its advanced materials portfolio, MKS enables ultra-fine copper interconnect formation, diffusion barrier control, and surface preparation optimized for hybrid bonding and high density routing. Through its photonics technologies, including precision laser systems and optical components, MKS supports accurate via formation, material structuring, and high-volume manufacturing scalability. Through its vacuum solutions division, MKS ensures stable pressure control, gas flow precision, and contamination mitigation essential for plasma activation, thin-film deposition, and advanced packaging processes.

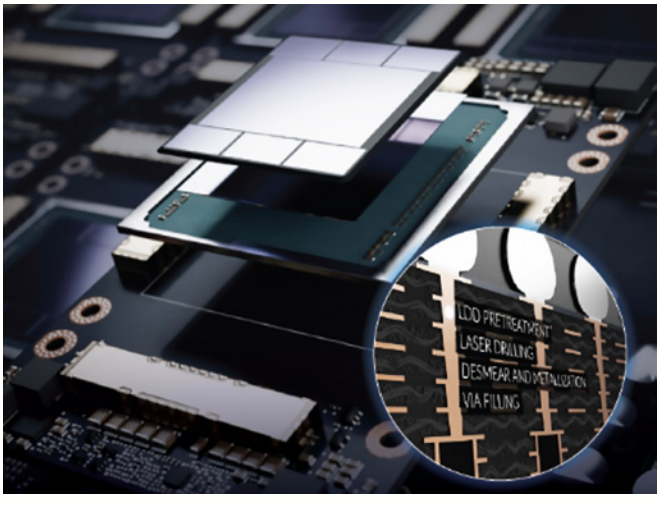
This integration allows manufacturers to address complexity holistically rather than in isolated steps. As AI architecture grows more intricate, yield and performance depend on the seamless coordination of multiple process technologies. The MKS Optimize the Interconnect® approach reflects this system-level philosophy, aligning materials, equipment, and process control into unified manufacturing solutions.

Enabling the Future of AI Infrastructure

The next decade of AI growth will be defined by increasing data center density, higher HBM stack counts, more advanced packaging architectures,







and growing energy efficiency pressures. Thermal management and power delivery will become even more critical. Glass substrates and panel-level packaging are already introducing new materials challenges. Photonics-electronics integration is already redefining interconnect strategies altogether.

In each of these transitions, precision at scale will determine competitive advantage.

At MKS, the strategic role is to enable that precision across every layer of the AI hardware stack. By combining materials science, laser and photonics innovation, and atomic-level vacuum control, MKS supports manufacturers in overcoming escalating density, contamination, thermal, and reliability challenges.

AI does not scale on algorithms alone. It scales on infrastructure—on the invisible layers of copper, silicon, surface chemistry, laser structuring, and vacuum-controlled environments that connect and cool the system.

As AI continues to transform industries and societies, MKS stands at the foundation of that transformation—enabling the interconnect technologies that power the infrastructure of intelligence. **I-CONNECT007**

About the Authors



John T.C. Lee
President and Chief Executive Officer, MKS

John T.C. Lee leads leads the MKS global strategy and vision, guiding the company's role in enabling critical technologies that power next-generation innovation. His perspective spans the entire technology ecosystem, from semiconductor manufacturing and advanced electronics to industrial and AI-driven applications, providing a unique view of the market forces shaping the future.



Marc Tricard
Vice President and Chief Technology Officer, Photonics Solutions Division (PSD)

Marc Tricard oversees technology innovation across across the MKS Photonics Solutions Division, including laser-based manufacturing, photonics, precision optics, and advanced process solutions. His perspective highlights how laser and photonics technologies accelerate semiconductor scaling, advanced packaging, and the high-performance computing systems that drive AI adoption.



Won Lee
Vice President, Power Solutions and Chief Technology Officer, Vacuum Solutions Division (VSD)

Won Lee leads the Power Solutions Business Unit and technology strategy and innovation for the MKS Vacuum Solutions Division, with expertise spanning vacuum technologies, power solutions, and the critical subsystems that enable advanced semiconductor manufacturing. His insights focus on the infrastructure technologies required to support increasingly complex and performance-intensive AI device production.



Kuldip Johal
Chief Technology Officer and Vice President, Business Development, Materials Solutions Division (MSD)

Kuldip Johal leads technology strategy and business development for the MKS Materials Solutions Division, focusing on advanced packaging, specialty chemistries and equipment, and interconnect solutions. His expertise provides key insights into the innovations required to enable advanced semiconductor packaging, high-density interconnects, and the next generation of AI scaling.

Optimizing the Interconnect

With Dr. John T.C. Lee

When Dr. John T. C. Lee talks about the evolution of electronics, he keeps coming back to the interconnect. As President and CEO of MKS Inc., he saw growing complexity in the connections between increasingly powerful devices as both a manufacturing challenge and a business opportunity.

That thinking helped drive the MKS acquisition of Atotech in 2022, bringing process chemistry and electroplating together with MKS' existing strengths in lasers, optics, and precision motion, and giving substance to a strategy the company calls Optimize the Interconnect®. Several years later, AI, advanced packaging, and high-performance computing are putting unprecedented demands on those interconnections.

We talked with Dr. Lee about what he saw coming, how that vision has shaped MKS' Atotech, and its strong position to address the technology challenges being faced.



When MKS announced the Atotech acquisition in 2021, you described the interconnect as the "next frontier for miniaturization and complexity." What were you seeing at that time?

Even before AI became the dominant conversation in our industry, we could see a fundamental shift underway. For decades, semiconductor performance advanced primarily through transistor scaling. However, as Moore's Law began to slow, the industry increasingly relied on system-level innovation to continue delivering performance improvements.

We observed that future gains would depend less on individual transistors and more on how efficiently chips, memory, interposers, package substrates, and advanced circuit boards are integrated and interconnected. The interconnect was becoming the critical link between all these technologies.

That insight was one of the driving forces behind the Atotech acquisition. Bringing together MKS' expertise in lasers, optics, motion control, vacuum technologies, and photonics with Atotech's leadership in process chemistry and electroplating allowed us to address manufacturing challenges across the entire interconnect stack.

Looking back, AI and high-performance computing have accelerated this trend even faster than many expected. Today, packaging and interconnect technologies are among the most important determinants of system

mks
Atotech

AI server PCBs and advanced packaging require highest copper reliability



From microvia geometry to direct Cu-to-Cu integration

AI computing is accelerating the adoption of advanced packaging and high-density interconnect (HDI) architectures in PCBs, package substrates, and heterogeneous integration. Copper recrystallization behavior and nanovoid formation can become significant factors in everything from stacked blind microvia junctions (capture pad / electroless copper / electrolytic copper) to the fine-grained interfaces needed for hybrid copper-to-copper bonding. MKS' Atotech enables innovation by linking process controls to microstructure outcomes and quantifiable reliability indicators. This is supported by predictive metallographic screening using Normalized Crack Length (NCL) and is made possible through its unique combination of process chemistries and equipment platforms for MLB/HDI (mSAP), package substrates, and advanced packaging.

Download our White Paper on this topic by scanning the QR code to the right.

info-atotech@mks.com

www.atotech.com



performance, bandwidth, power efficiency, and manufacturability. In many ways, the opportunity has unfolded exactly as we anticipated, but the pace has exceeded expectations.

What can MKS solve today because these capabilities sit under one roof?

Many companies provide individual technologies. Few can address the broader manufacturing challenge across multiple levels of the interconnect ecosystem.

Today, MKS combines advanced materials, electroplating chemistry and equipment, laser processing, optics, photonics, vacuum control, motion systems, and process expertise. This enables us to approach customer challenges from a system perspective rather than as isolated process steps.

For example, forming an advanced microvia or preparing a surface for hybrid bonding is not simply a chemistry challenge or a laser challenge. Yield, reliability, throughput, and manufacturability depend on the interaction of multiple process technologies.

By bringing these capabilities together, we can help customers optimize entire manufacturing flows, improve yields, shorten development cycles, and accelerate adoption of next-generation architectures. That is the essence of our Optimize the Interconnect strategy.

What is AI infrastructure demanding from PCBs and package substrates today that manufacturers weren't being asked to deliver even three or four years ago?

The requirements have increased dramatically AI systems demand substantially higher bandwidth, greater power density, lower latency, and more efficient thermal management. As a result, manufacturers are being asked to produce increasingly complex PCB, substrate, and packaging structures with tighter tolerances than ever before.

We're seeing demands for higher layer counts, finer geometries, smaller microvias, more advanced routing, tighter overlay accuracy, improved warpage control, and greater consistency across extremely large production volumes.

At the package level, technologies such as silicon interposers, advanced redistribution layers, HBM integration, and hybrid bonding are pushing manufacturing precision to entirely new levels. Features measured in only a few microns are becoming increasingly common.

The challenge is no longer simply manufacturing smaller structures. The challenge is manufacturing them repeatedly, reliably, and economically at scale.

What does the next chapter of MKS look like?

The next chapter for MKS will be defined by increasing complexity across the electronics ecosystem.

Over the next five years, I expect to see continued growth in advanced packaging, heterogeneous integration, high-bandwidth memory, photonics-electronics convergence, panel-level packaging, and next-generation interconnect architectures.

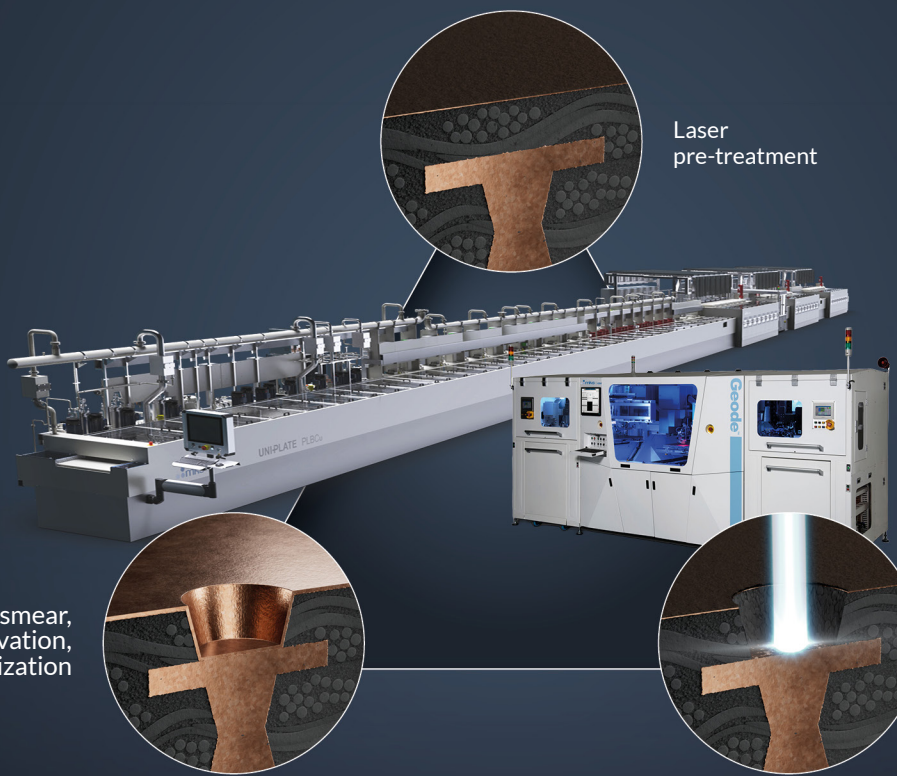
These trends all point toward one conclusion: Manufacturing precision will become even more important.

Our focus remains on enabling precision across the entire hardware stack. We will continue investing in the technologies that help customers address challenges related to connectivity, bandwidth, thermal management, yield, contamination control, and scalability.

If I were placing a bet, it would be on the continued importance of system-level integration. The future leaders in electronics will not simply have the best individual technologies. They will have the best ability to integrate multiple technologies into reliable, scalable manufacturing solutions. That's exactly where MKS is uniquely positioned to create value. AI may be driven by software, but it scales through physical infrastructure, and MKS intends to remain at the center of enabling that infrastructure.

I-CONNECT007

Next frontier of PCB manufacturing – Optimize the Interconnect®



Atotech plating equipment and process chemistries combined with ESI laser drilling systems

We are committed to driving the next level of technology development in our industries and accelerating time to market through our combined capabilities. This allows us to look at problems holistically and provide customers and OEMs with faster development cycles for new products and materials that enable next-generation technologies. In PCB manufacturing, where precision is paramount, MKS delivers the winning combination for superior yields. Our strategic fusion of Atotech leading plating systems (Horizon® / Uniplate® / vPlate®) and process chemistries for LDD pre-treatment (BondFilm® LDD), laser drilling splash removal (BondFilm® LDD SR), desmear (Securiganth®), ionic activation (Neoganth®), electroless Cu metallization (Printoganth® P2) and electrolytic Cu filling (Inpulse®/ InPro®) with ESI Geode™ Via Drilling System for optimized laser drilling creates a breakthrough integrated approach. By addressing the complexities of intricate PCB designs and challenging materials, MKS ensures you stay ahead of the competition.

For more information on our company or products, just scan the QR-code.

info-atotech@mks.com

www.atotech.com

The End of FLAT THINKING



BY MICHAEL GLEASON, GREENSOURCE FABRICATION

Editor's note: We're excited to introduce a new column by Michael Gleason, director of product development and strategic growth at GreenSource Fabrication. Michael will challenge engineers to think beyond today's design conventions, explore new possibilities in PCB fabrication, and rethink how they approach designing for the future. The column will appear monthly in I-Connect007 Magazine.

Advanced PCB design increasingly brings electrical performance, thermal behavior, mechanical requirements, material selection, and manufacturability into the same conversation. The board sits within a larger physical system, and each decision can influence how that system performs. Looking at those relationships together creates more opportunity to optimize the design as a whole.

The key is understanding where each design constraint comes from. Physics, reliability, and process capability establish some boundaries, while others may carry forward from an earlier design or a manufacturing process that has since evolved. Understanding the reason behind a rule gives designers and fabricators a stronger foundation for determining the best path forward.

We see this with customer-designed stackups. These can come in with overly complex via structures, unbalanced constructions, suboptimal material selections, and copper thicknesses that are inappropriate. Taken at face value, they can be unmanufacturable. Working with the customer to understand the design intent, key attributes, performance expectations, etc., can, more often than not, provide a path forward to deliver a manufacturable stackup that meets the customer's desired outcome.

electronica

Visit us at Booth 207 | A1
10 - 13 November 2026

ventec
INTERNATIONAL GROUP



Halogen Free:
FR4.1 & FR15.1 materials

Engineered for high-voltage and demanding environments, featuring Class 0 CTI and CAF resistant grades.

Ventec's range of halogen free FR4.1 and FR15.1 laminates and prepregs for clean high-tech products, offering excellent environmental credentials without compromizing thermal or electrical performance. Properties include high Tg, Maximum Operating Temperature (MOT) up to 150°C, low CTE, and high Comparative Tracking Index (CTI) for safety in high-voltage applications.

Comprehensive supply chain solutions available from fully stocked warehouses across Europe and the UK. Contact your local sales representative today.

venteclaminates.com

Wherever technology takes you, Ventec delivers

Designing the Physical System

That thinking extends across the physical system. Heat generation and heat flow influence component placement, materials, copper distribution, and mechanical design. Embedded components can free surface area and shorten electrical paths while introducing additional fabrication and reliability considerations. Enclosures, airflow, connectors, and mounting points also shape what is practical at the board level. Routing, impedance, stackups, materials, and signal integrity remain fundamental, with each decision contributing to the performance of the complete system.

Manufacturing capability is an important part of that equation. Fabrication rules reflect what a process can produce repeatedly and reliably, while continued advances in processes and materials expand the available design space. A spacing requirement, construction method, or feature size tied to one process may have additional options as capabilities evolve.

One example that comes to mind is annular ring. Due to our exceptional registration capabilities, we can support smaller pads for a given via size, affording the designer greater routing density.

At the edge of established PCB capability, that distinction creates valuable opportunities. A fabricator may be able to address a challenging requirement through a different process, construction, or material approach. Understanding the intent behind the requirement gives both sides more room to identify the solution that best supports performance and manufacturability.

Manufacturing Must Be Part of the Conversation

For complex designs, manufacturability is most valuable when it is part of the process from the beginning. Early collaboration gives designers access to a broader range of materials, fabrication processes, and construction options while the architecture is still taking shape. It also allows performance, reliability, manufacturability, and cost to be considered together.

The number one thing we tell our customers is to bring us in as early as possible on complex designs. We've worked on multiple projects where we have recommended an mSAP manufacturing process for a given layer rather than traditional subtractive etch. The immediate advantage is greater routing density through finer features, but the benefits can extend beyond density. When RF performance is the driver, the straighter sidewalls and tighter feature tolerances achieved through mSAP can offer significantly improved performance over subtractively etched features.

Engaging the fabricator early gives the design team an opportunity to identify those options while there is still flexibility in the design, helping avoid costly tear-ups, reroutes, and fabrication respins later in the process.

Advanced fabrication brings together an understanding of what the design is trying to accomplish with the materials, processes, and manufacturing capabilities that can help achieve it. The earlier that conversation happens, the more opportunity there is to optimize performance, manufacturability, and reliability together.

Beyond the Board Boundary

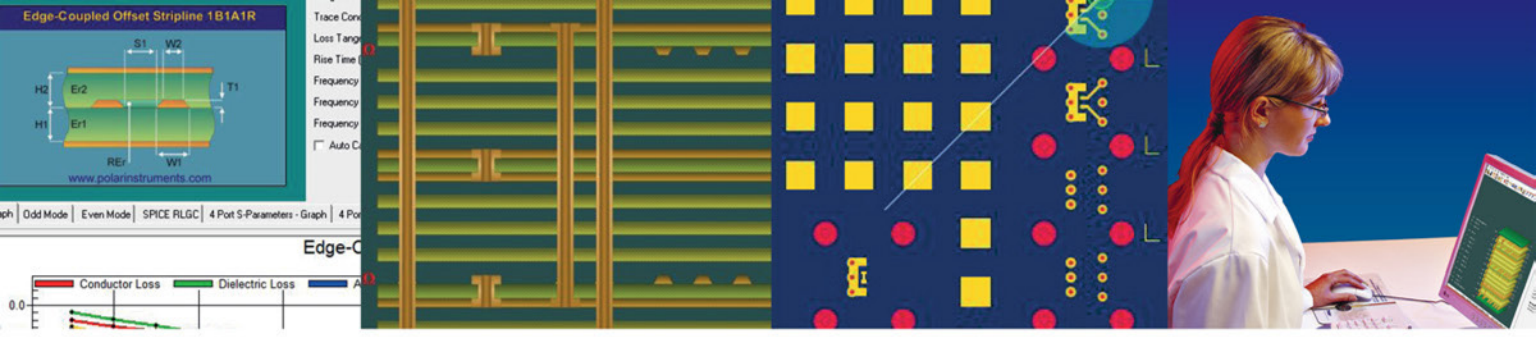
This is also why electrical, mechanical, and thermal engineering come together at the point where the product takes shape. Heat moves through materials and structures. Mechanical stress reaches components and interconnections. Geometry affects electrical behavior. Fabrication choices influence all of them, and the PCB often sits at the intersection. Recognizing when a board-level decision reaches beyond the board creates an opportunity to bring the right expertise into the conversation early and make those disciplines work together.

As designs become more complex, the opportunity lies in looking beyond individual requirements and understanding how each decision connects to the larger system. That means bringing design, engineering, and manufacturing expertise together earlier, asking what the design is ultimately trying to achieve, and using the full range of capabilities available to get there.

The future of advanced PCB design will be shaped by how well we connect those pieces, and how early we start the conversation.



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



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

PCB Signal integrity tools for design & fabrication

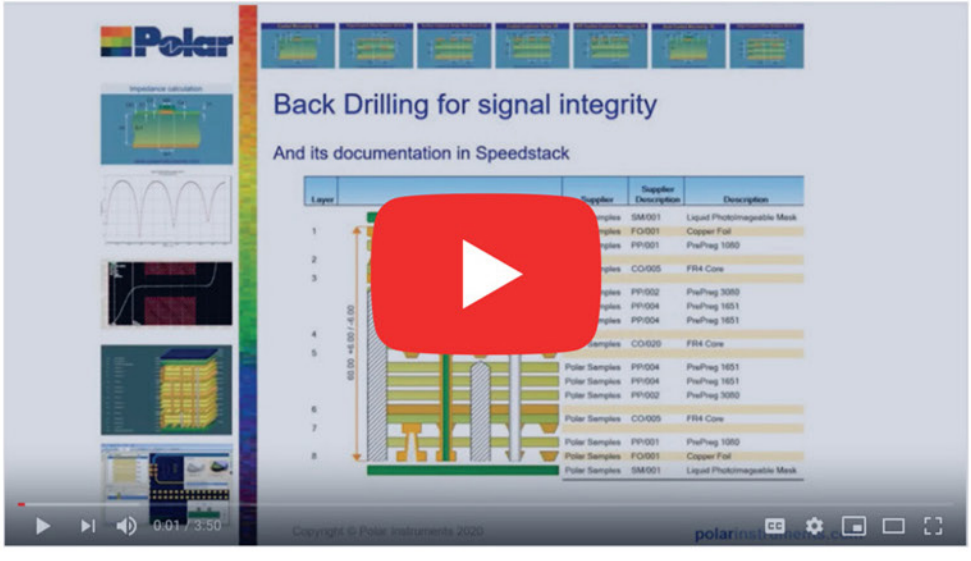
- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics

SUBSCRIBE



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

DOWNLOAD BOOK



polarinstruments.com



A Nightmare on PCB Street

BY JOHN WATSON, PALOMAR COLLEGE

Can you feel it? One of the best times of the year is finally here. The kids are back in school (finally), the weather is cooling, the days are getting shorter, football is back, and fall is showing up everywhere. Of course, beginning Sept. 1, two groups immediately emerge: those ready to enjoy fall, and those already dragging out the Christmas decorations like October and November no longer exist.

Then there is pumpkin spice: Coffee, cookies, candles, soap, air freshener—the list goes on. Is anything left that someone hasn't tried to turn into pumpkin spice?

In fact, I thought I had seen it all until I was offered

Flexible Circuit Technologies | inTFlex

The EMS Group of Flexible Circuit Technologies

FLEX SOLUTIONS
ENGINEERED FOR INNOVATION

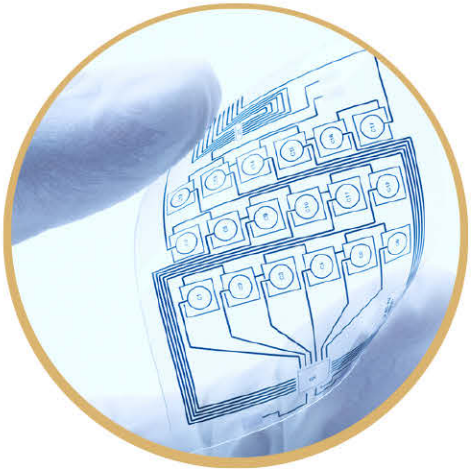
Industry-leading flex design services, premier in-house manufacturing of flexible circuits and EMS services



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

a pumpkin spice oil change. Seriously? I can tolerate the candles and hand soap, but pumpkin spice motor oil is taking it too far. What's next? Pumpkin spice brake fluid? Cinnamon-nutmeg transmission fluid? Limited-edition autumn coolant with just a hint of clove?

The Scary Stuff

Personally, I don't rush straight into Christmas as some people do. I like to enjoy October, and, of course, Halloween. It's the one time of year when we actually enjoy being scared. With haunted houses, scary movies, and creepy decorations, we willingly walk straight toward the things designed to make us uncomfortable.

At my age, getting scared has become a lot easier. I really don't need a haunted house anymore. Some mornings, just getting out of bed provides enough suspense. If everything bends, straightens, and starts moving in the right direction without making any new sounds, I already consider the day a major success.

PCB designers also don't need Halloween to know what true terror feels like. This profession has enough built-in “horror” to keep us awake all year long. That probably explains why so many senior PCB designers and engineers eventually fall into one of two categories: gray hair or no hair at all. After enough impossible schedules, last-minute changes, obsolete components, and "quick” revisions, your hair basically decides for you.

The Nightmare on PCB Street

So, in the spirit of Halloween, let’s imagine a typical street lined with typical houses in an ordinary neighborhood in a quiet town. This is PCB Street. At the very end sits one house that looks like all the others. But it isn't.

This is the PCB Fright House. There are no ghosts, chainsaws, or creepy clowns. You’re going to find out that inside this house is much, much worse, especially for PCB designers who like everything just right.

As you approach the front door, it creaks open on its own. At first, everything looks perfectly normal—maybe a little too normal. Then you step inside. The lights are dimmed, but you can see a darkened hallway, and you feel that something scary is waiting in that darkness.

Since this is a typical house on a typical street, logically, you have no need to fear. You start down the hallway. On the first door hangs a small sign: “The Room of Moving Walls.” You reach for the handle, push the door open, and step inside, thinking you know exactly what this room will look like: four straight walls. But you were very wrong. It is, in fact, just like the sign on the door—the walls shift in front of your eyes.

This dimly lit room represents one of the most familiar and frustrating fears in PCB design: changing requirements. You approach the design believing it is finally stable: The board outline is set, the connectors are placed, the stackup is defined, and the major decisions have been made.

Then, without warning, the requirements begin to move. For a moment, you wonder, "Are the “walls” of this design actually moving? Are the requirements changing again?"

Yes, they are. You notice that the enclosure changes, a connector shifts, the stackup gets revised, or a component is replaced. Then you hear those terrifying words that send chills down every PCB designer's spine: "Let's add just one more feature to the design."

Now, individually, each change looks manageable. That’s what makes this room of shifting walls so dangerous. In PCB design, very few changes stay isolated. If you move one connector, the placement changes. Placement changes routing, which can affect signal integrity, power distribution, clearances, thermal performance, and manufacturability.

What looked like a simple adjustment can quietly ripple across the entire design, and before long, the room in this analogy looks nothing like it did when you first walked in.

Don’t Be Afraid of Change

The real fear isn't that things change. Change is part of every design. The danger comes when those changes happen late, when their impact isn't understood, or when they aren't communicated to everyone involved.

In this imaginary house of horrors, you turn to leave, but the door is no longer where you came

This is the PCB Fright House. There are no ghosts, chainsaws, or creepy clowns. You're going to find out that inside this house is much, much worse, especially for PCB designers who like everything just right."

in. The walls have shifted one last time. You begin to panic, but you search around, find the door just barely open, and you narrowly slip through just before it closes.

You gather your thoughts and continue down the hallway. It grows darker and quieter, but at least the walls have stopped moving. Another door comes into view, and you can make out a battered sign hanging crookedly across the front: "The Unmapped Room."

Just like the first door, you reach for the handle. Again, the door creaks open on its own. It's completely dark inside. You stretch out your hands in front of you, feeling for whatever might be waiting just beyond your reach.

For PCB designers, that is a familiar feeling. Emerging technologies push us into territory where the path forward exists, but it might not be clearly marked or even visible. UHDI, SAP and mSAP, advanced materials, embedded technologies, finer geometries, and AI-assisted design are moving faster than the standards, manufacturing experience, and proven design rules behind them.

We feel uncomfortable. We still have to decide on trace widths, spacing, materials, stackups, via structures, impedance, and fabrication tolerances. Those decisions eventually become something physical that must be built and must work. But what if the answers are still evolving?

As you feel your way through the dark, stepping more fully into the room, you hear voices and recognize them as fabricators: "It can be done," says one. "That's too aggressive," says another.

In PCB design, this could be referring to a promising new material, but there isn't enough field history to know what surprises may be waiting. It may be a new process that promises incredible density, but the design guidelines are still being written.

You hear more voices, with different answers and opinions, and you still have to make the design decision. Walking blindly into something new can result in more than just a quick scare. It can mean a failed prototype, fabrication problem, costly respin, or weeks lost from the schedule.

That is what makes this "unmapped" room so frightening. It feels like the scariest room in the house, although it probably isn't. There's something even more sinister afoot.

In the Unmapped Room, you feel along the walls until you find another doorway and, oddly enough, you're back into the hallway. For a moment, there is relief, until you squint in the low light and see one more door at the far end of the hall. Unlike the others, this one is already open. It has no flickering lights, moving walls, or strange sounds.

There's just nothing, and somehow, that makes it worse.

You step into the doorway and peer into the dimly lit room. You can make out that a desk, a few old drawings, a stack of notes, and one empty chair. You stand there for a moment, wondering if someone will appear, but nobody does.

Now, a real fear starts to envelop your body. Something happened in this room. Someone used to be here, and now they're gone.

To be continued... **I-CONNECT007**



John Watson is a professor at Palomar College, San Marcos, California. To read past columns, click here.

BY DAN THAU, MILLENNIUM CIRCUITS LIMITED

The Supplier's Job Has Changed:

Turning PCB Supply Chain Uncertainty into Action



Information about what's happening in the PCB supply chain is abundant. Customers know materials are constrained and that lead times can change quickly. They know tariffs, geopolitical uncertainty, shifting manufacturing capacity, and growing demand from emerging technologies are creating pressure throughout the electronics supply chain. Another market update won't solve their problem.

What customers need is someone who can tell them what it all means for their program, and what they should do about it. I believe that's where the role of PCB suppliers is changing.

For years, suppliers have differentiated themselves primarily around price, quality, technology, and delivery. Those things remain essential, but in an unpredictable market, simply quoting and delivering boards isn't enough.

A strong supply chain partner helps customers make better decisions before disruption reaches their production schedules by anticipating risk rather than simply reporting it. They create sourcing options before they're needed, identify engineering

INSULECTRO



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

InduBond®

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

CAC

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

issues before they become production problems, secures material before availability becomes critical, and when conditions change, they have the relationships, experience, and financial strength to act.

It's not simply about navigating the PCB supply chain, but making the PCB supply chain easier for our customers to navigate.

Information Is Valuable, Action Is Better

One of the most important lessons I've learned through periods of supply chain disruption is that information has a relatively short shelf life. While knowing that a **material is becoming constrained** is useful, knowing what to do about it is much more valuable.

If lead times are beginning to move, ask good questions: What does this mean for upcoming builds? Is there material that should be secured now? Should a release be pulled forward? Is there an alternate material or manufacturing path worth evaluating? Would a stocking strategy reduce risk?

That's the difference between communicating a problem and helping manage it. Customers shouldn't need to become experts in laminate availability, global factory capacity, tariffs, logistics, and PCB manufacturing just to keep their programs on schedule. A good partner should absorb some of that complexity and turn it into clear choices.

Resilience Happens at the Program Level

The PCB industry talks a lot about supply chain resilience. It's an important conversation, but resilience can become an abstract concept.

Having more factories doesn't automatically make a supply chain more resilient. Every PCB has its own technology curve, quality requirements, volumes, certifications, cost pressures, and timeline. What matters is whether you have the right manufacturing option for that specific program, and whether you've done the work to understand those options before they're needed.

I've learned that this distinction becomes especially important when markets tighten, and I start asking important questions: Does the manufacturing partner's core competency align with the technology? Does it have the appropriate certifications? How has it performed on comparable work? Are quality and communication expectations already established? If material or capacity becomes constrained, what alternatives exist?

The goal isn't simply to have more suppliers, but to have the right options before you need them.

Risk Often Starts Before the Purchase Order

Supply chain risk isn't created only by material shortages or factory capacity. Sometimes it's already sitting in the design. A stackup issue, tolerance requirement, material selection, or manufacturability concern discovered after a board reaches production can quickly become a schedule problem. That's why I believe PCB suppliers need to become involved earlier.

We've seen firsthand how an early DFM conversation can eliminate a problem that otherwise might not surface until weeks later in production. The earlier that engineering and manufacturing requirements are aligned, the more opportunity there is to address manufacturability, reliability, material availability, cost, and lead time.

In a stable environment, catching something early saves time and money. In a constrained environment, it can protect an entire production schedule. The cheapest disruption to solve is the one that never makes it into production.

Speed Without Chaos Matters

When conditions change quickly, the natural reaction is to move faster. But speed without coordination can create an entirely different set of problems. A rushed sourcing decision that creates a quality issue isn't a solution. Neither is moving a board to a factory that isn't well-suited for the technology simply because capacity is available.

The goal should be speed without chaos. Supplier relationships need to occur before capacity gets tight, technical capabilities need to be understood before an alternate source is required, and quality expectations need to be established. Communication channels need to be open.

You can't build those things in the middle of an emergency. When customers need an answer, they need accurate information quickly. When action is required, their supply chain partner needs to be positioned to move.

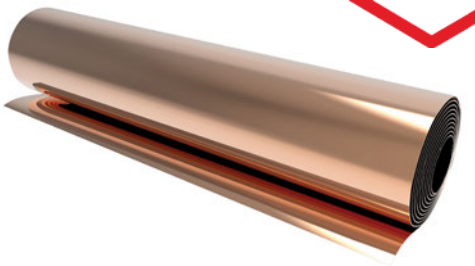
INSULECTRO

WORLD-CLASS SUPPLIERS

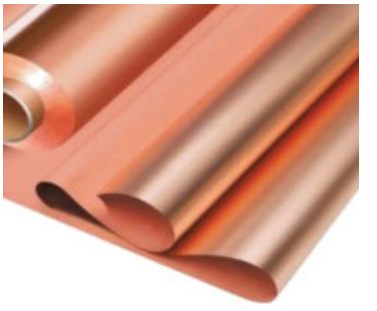
Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, salutes our premier suppliers.

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.



PACOTHANE TECHNOLOGIES



Conformal Release Film

MADE IN USA

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™

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!

INSULECTRO 20362 Windrow Drive, Lake Forest CA 92630 • 949.587.3200 • insulectro.com

The Customer Shouldn't Have to Manage the Supplier


Dan Thau

We've ended up transferring a lot of that complexity to the customer. This is a consequence of complicated supply chains that I don't think we talk about enough, where the buyer follows up on delivery, engineering answers manufacturability questions, quality chases documentation, and supply chain evaluates risk. Before long, the customer is coordinating the supplier instead of the supplier coordinating the program.

I think we've got that backward. A PCB partner should create a layer of simplicity between the customer and a more complicated global manufacturing environment. Engineering, quality, sourcing, manufacturing, and logistics may all be involved behind the scenes, but the customer shouldn't have to coordinate those functions individually.

That's the thinking behind the Customer Concierge model we've built at MCL: one accountable point of contact responsible for coordinating execution across the program.

The larger principle is simply that a good PCB partner shouldn't add another thing for your team to manage. It should take work off your team's plate.

Forecasting Should Create More Than Visibility

The current material environment has also made me think differently about forecasting. Suppliers have always asked customers for better forecasts because visibility helps us plan. But if a customer can give us reasonable visibility into future demand, we should be asking how we can turn that visibility into greater stability for the customer.

If a customer already knows what they are likely to need six months from now, waiting until each purchase order comes due may unnecessarily expose that program to changes in material pricing, availability, capacity, or lead time.

Earlier commitment creates options, and that's what led us to develop SupplyShield at MCL. It is designed for customers with known or reasonably predictable PCB requirements. Customers commit earlier against forecasted demand, giving us greater ability to plan with our manufacturing partners and secure material and capacity when appropriate. In return, we can provide price stability for a defined six-month period.

We've seen what that can mean in practice. One recent customer committed approximately $750,000 in forecasted PCB requirements through this program, and that visibility allowed us to secure material and capacity months earlier than would otherwise have been possible.

In the months that followed, multiple suppliers implemented price increases, while lead times on several materials increased. Because the customer had committed to known demand in advance, pricing remained fixed for the duration of the agreement, and production schedules remained intact despite the broader market volatility. Based on the affected requirements, we estimate that early commitment helped the customer avoid approximately $250,000 in additional costs.

But the financial benefit goes beyond the avoided price increase. The customer didn't have to purchase and take possession of the entire $750,000 requirement at once to gain the benefits of earlier planning. That allowed the business to preserve working capital for other priorities while creating greater predictability around future PCB requirements. The value wasn't simply locking in a PCB price. It was reducing price, availability, and schedule risk before market conditions changed.

SupplyShield isn't the right answer for every program. For another customer, the answer might be a blanket or stocking strategy, an alternate manufacturing geography, a material substitution, or a design change. The solution should follow the customer's risk, not the supplier's product catalog.

Uncertainty Tests Supplier Relationships

Periods of disruption have a way of exposing weaknesses that aren't as visible when everything is running smoothly. Customers quickly find out who communicates well before there's a problem and

who waits to call until after the delivery date moves. They find out which suppliers understand their programs and which simply process purchase orders. They find out who has meaningful manufacturing relationships, who has options, who has the financial strength to act, and who takes ownership when something goes wrong.

That can change supplier relationships quickly. It's also why periods of uncertainty can create significant shifts in market share. Customers aren't necessarily looking for different PCBs. They're reconsidering what they expect from the companies supplying them.

Price will always matter. So will quality and delivery. But transparency, technical expertise, optionality, financial stability, proactive communication, and accountability are becoming important parts of the equation.

The Supplier's Job Has Changed

None of us can predict the next disruption.

Today's challenge might be material availability. Tomorrow, it might be capacity, tariffs, logistics, geopolitics, or something none of us are discussing yet. Trying to predict every disruption isn't realistic. Building an organization capable of responding to them is. That requires strong engineering, the right manufacturing relationships, financial stability, good information, and the ability to move quickly without sacrificing quality.

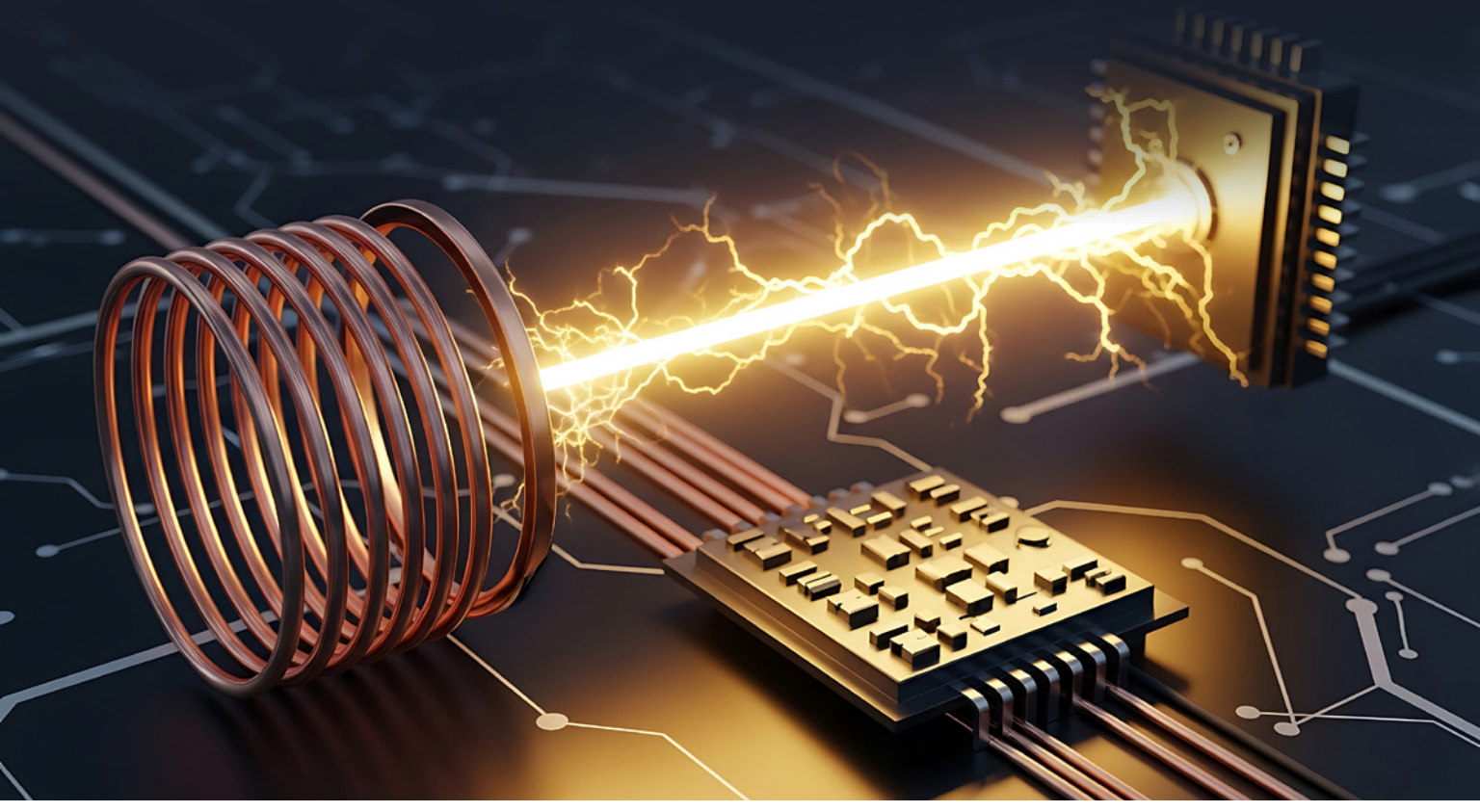
But it also requires a change in mindset. The PCB supplier can't simply be the company that delivers the board. We have to be willing to help customers anticipate what's coming, understand their options, reduce complexity, and execute when circumstances change.

At MCL, we describe that commitment as where demanding PCB programs meet relentless execution. But that isn't about making the supplier the hero. The customer is the one with a product to launch, a production schedule to protect, and a commitment to keep. Our role is to give them the engineering, information, options, and execution they need to succeed. Because when requirements are demanding and the market is uncertain, customers don't just need boards. They need a partner who executes. **I-CONNECT007**

Dan Thau is CEO of Millennium Circuits Limited (MCL), a PCB supply chain partner helping engineering, procurement, quality, and manufacturing teams execute demanding PCB programs.



Inductance:
The Silent Architect Driving SI and PI



BY BARRY OLNEY, IN-CIRCUIT DESIGN PTY LTD | AUSTRALIA

Inductance is the quiet saboteur of high speed design. It doesn't reveal itself the way impedance discontinuities do, nor does it leave the obvious clues that poor routing or inadequate stackups produce. Yet it shapes nearly every facet of signal integrity (SI) and power integrity (PI), influencing how energy moves, how noise couples, how timing shifts, and how the power distribution network (PDN) responds under dynamic load.

In multilayer PCBs, electric and magnetic fields interact to transfer energy between components. Whenever energy moves, inductance is present. Whenever inductance is present, SI and PI are affected. Understanding this relationship is the difference between a board that merely functions and one that performs reliably at multi-gigabit speeds.

Electric and magnetic fields play equal roles in the transport of energy in a multilayer PCB. The presence of voltage implies an electric field, and changing that field creates a magnetic field. What may not be appreciated is that moving a voltage between two components requires moving energy

Millennium Circuits Limited

DON'T GET STUCK IN LINE.

LOCK PRICING. SECURE MATERIAL. PROTECT PRODUCTION.

Material shortages are widening. Lead times and pricing remain under pressure. MCL's **SupplyShield** locks your pricing for an established period and secures material and capacity before the market tightens further.

- ✓ **Lock pricing** for an established period
- ✓ **Secure material** and capacity earlier
- ✓ **Protect production** from market disruptions

REAL CUSTOMER IMPACT

CUSTOMER EXAMPLE

$250,000 annual PCB spend

BUYING MONTHLY	WITH SUPPLYSHIELD™
Re-priced higher every order. Constant interruption.	Price locked across the holding period.

POTENTIAL COST AVOIDANCE **25–50%**

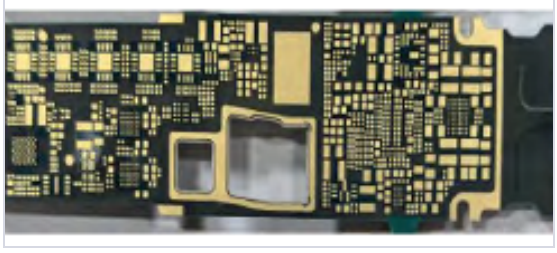
Estimated savings over the holding period. Actual results vary by volume, material, and market conditions.

THE TIME TO ACT IS NOW

See what SupplyShield could save you. **Contact MCL for a PCB Supply-Risk Review.**

WHEN CONVENTIONAL PCB FABRICATION CAN'T KEEP UP.

ADVANCED PCB TECHNOLOGY



400G	800G	1.6T
NETWORKING	DATA CENTER	NEXT-GEN DESIGNS

MCL is expanding its advanced capabilities with **Modified Semi-Additive Processing (MSAP)**. MSAP enables finer traces and spacing for increasingly dense, high-performance designs.

- ✓ Greater routing density
- ✓ Improved signal integrity
- ✓ Potential layer count reduction

BUILT FOR NEXT-GENERATION AI, DATA CENTER AND HIGH-SPEED APPLICATIONS.

Millennium Circuits Limited

AS9100 CERTIFIED | ISO 9001 CERTIFIED | ITAR REGISTERED | CLASS II & III CAPABILITIES

www.mclpcb.com

717-558-597

(not a signal), which requires the existence of both electric and magnetic fields. When energy is not moving, the magnetic field is zero.

Inductance and Signal Integrity: The Geometry of Crosstalk, Noise, and Delay

When current flows through a conductor, a magnetic field is produced. When a second conductor carrying current is brought into proximity, a force acts between them. If both currents flow in the same direction, then they are additive (think of two parallel trace segments). When the currents flow in opposite directions (think of a trace over a return plane), the currents cancel. This implies that two individual traces should be kept well apart to reduce crosstalk, whilst a signal trace should be tightly coupled to its return path (plane) to increase coupling and reduce inductance.

Figure 1 illustrates the near-end (NEXT) and far-end (FEXT) crosstalk on victim traces adjacent to a 1.5 V, 1 GHz aggressor. The traces are 4 mils wide, spaced 4 mils apart, and designed for 40 Ω impedance. As each victim trace moves farther from the aggressor, the crosstalk naturally diminishes. The self-inductance field loops represent the magnetic field generated by a trace's own current, whereas the mutual-inductance loops are the field lines encircling a trace due to current in a neighboring conductor. These are responsible for crosstalk. The resulting coupling produces ringing, which in turn generates electromagnetic radiation.

When current flows in a conductor, a magnetic field forms around it. Bring another conductor close, and the two magnetic fields interact. This is mutual inductance, which is the root cause of crosstalk, switching noise, and timing uncertainty.

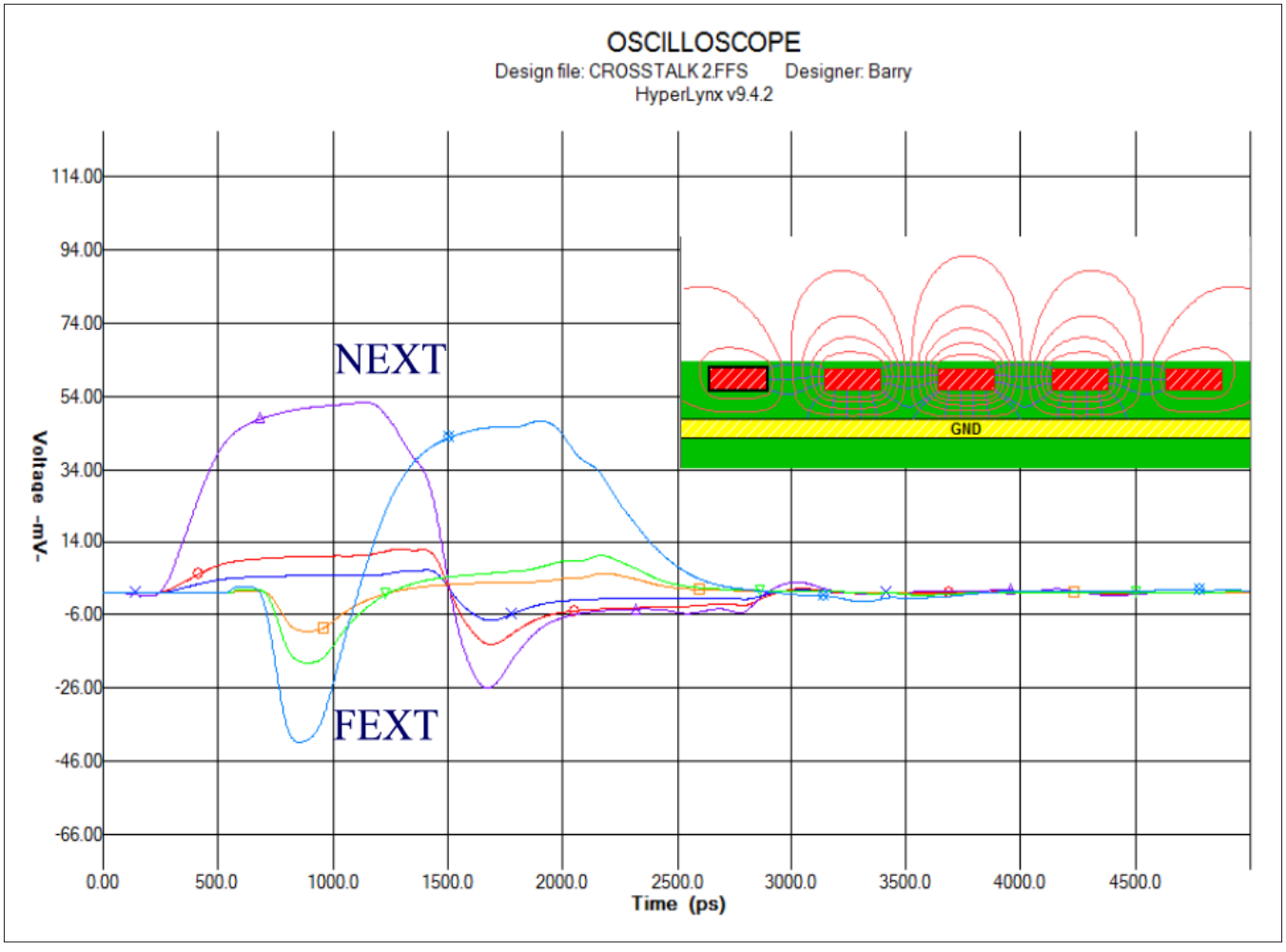


Figure 1: Near and far end crosstalk for microstrip (simulated in HyperLynx).

Two simple geometric truths govern SI:

1. Traces close together increase mutual inductance, meaning more crosstalk. Two individual traces should be kept well apart to reduce crosstalk.

2. Traces tightly coupled to their reference plane reduce loop inductance, meaning cleaner signals. A signal trace should be tightly coupled to its return path (plane) to increase coupling and reduce inductance.

The PDN'S High-frequency Ceiling

A PDN is far more than a set of power traces and scattered capacitors; it behaves as a distributed AC-impedance system that must simultaneously deliver high transient current with minimal voltage noise, maintain impedance below the target level across nine decades of frequency, provide a clean and continuous return path for high-speed signals, and suppress cavity resonance, ground bounce, and EMI radiation.

In the time domain, a decoupling capacitor acts as a local energy reservoir, supplying instantaneous current to the processor during fast load transients until the voltage regulator can respond. However, in the frequency domain, that same capacitor reduces the PDN's AC impedance around its self-resonant frequency, enabling more high-frequency current to reach the IC with minimal voltage noise.

By lowering impedance and providing instantaneous charge, decoupling capacitors suppress dI/dt noise, simultaneous switching noise (SSN), rail collapse during burst activity, and jitter induced by supply modulation. Plus, a decoupling capacitor provides a local displacement-current loop between power and ground, reducing loop inductance, switching noise, common-mode radiation, and ground bounce.

As operating frequencies and rise times increase, the PDN's AC impedance climbs because the inductance of the bypass and decoupling capacitors becomes the limiting factor. Every capacitor includes an equivalent series inductance (ESL) component, which causes its impedance to rise at high frequencies. Bulk capacitors work well up to roughly 10 MHz, and ceramic capacitors extend coverage into the hundreds of megahertz. Above that, only the PCB's plane capacitance and on-die capacitors can meaningfully suppress PDN impedance.

A single capacitor covers only a narrow band. A network of capacitors—bulk, mid-frequency, high-frequency—creates a stair-step impedance profile that keeps PDN impedance below target from kHz to hundreds of MHz. This is why "one capacitor per pin" is a myth. You need a distribution of values and ESLs. However, the power/ground plane pair acts almost like a perfect capacitor, with negligible inductance (no leads) and very low equivalent series resistance (ESR), making it ideal for ultra high-frequency noise suppression. Tight plane spacing (<5 mil) further boosts this high-frequency capacitance that lowers PDN impedance, reduces switching noise, supports fast transient currents, and improves overall PI stability.

Figure 2 illustrates the relationship between capacitance, resistance, and inductance of a capacitor. If the capacitance is increased, then so is the frequency response. However, it is inevitably the mounting and spreading inductance that pushes the response down.

There is, however, a leverage point that designers can use to minimize mounting inductance. The loop area formed by the capacitor, its vias, and the planes determines mounting inductance. Reduce the loop area, and PI improves dramatically. So, we need to place the vias closer to the component lands.

The best practice hierarchy for via fanout:

1. Via in pad (lowest inductance)
2. Dual vias adjacent to pad
3. Single via close to pad
4. End via fanout (worst case)

Never use thermal reliefs on PDN vias; they add inductance and sabotage high frequency performance.

Return Path Discontinuities (RPDs): The Hidden SI Killer

Designers obsess over trace length but often ignore the return path. If the return displacement current must detour around a void, split, or poorly placed

via, the loop area increases and so does inductance. Inductance slows the flow of energy, and in digital logic, timing is nothing more than the speed at which energy can be delivered to and from charge and discharge nodes.

When inductance increases, several timing-critical effects appear, including delayed edge transitions, propagation delay, and mode conversion. Signals do not propagate along traces; they propagate through EM fields, and if the return current must detour around a split or void, the loop area increases, then inductance increases, causing the electromagnetic wave to slow down. The consequences are severe:

- Channel-to-channel skew
- Reflections and jitter
- Mode conversion in differential pairs
- Increased common-mode radiation
- In modern buses running at tens of gigabits, even small RPDs create measurable skew and channel-to-channel noise.

Designing for Low Inductance

Across both SI and PI, the same principles apply:

- Separate signal traces to reduce mutual inductance and crosstalk.
- Couple signals tightly to reference planes to minimize loop area.
- Maintain continuous return paths—no splits, voids, or detours.
- Use low-inductance capacitor mounting (via-in-pad or dual-via).
- Place decoupling capacitors on the same side as the BGA to reduce loop area.
- Tightly couple power and ground planes to create high-frequency capacitance.

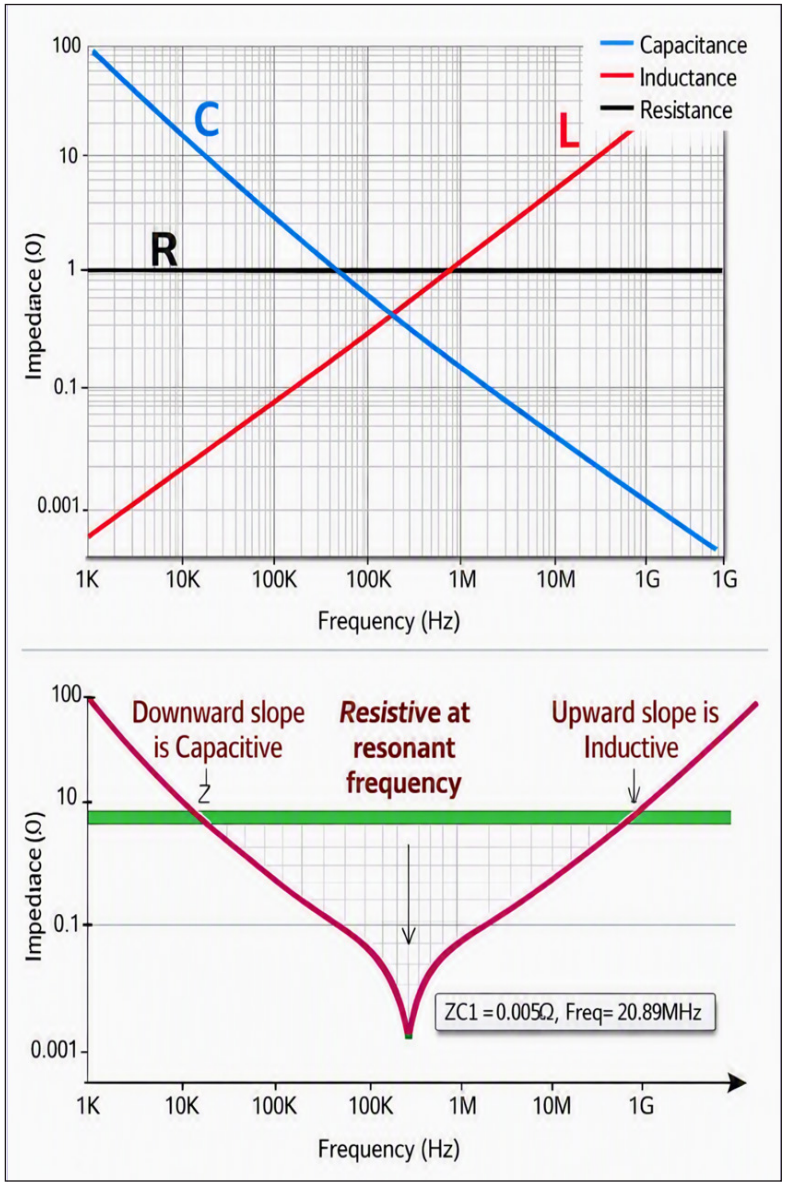


Figure 2: A capacitor has series capacitance, resistance, and inductance.

These practices are not optional in modern designs. They are foundational.

Inductance is the common thread linking SI and PI. It shapes crosstalk, return-path behavior, PDN impedance, capacitor effectiveness, and electromagnetic emissions. By treating inductance as a first-order design parameter (not an afterthought), designers can build boards that are quieter, faster, more stable, and far more predictable. The curse of the high-speed board is not complexity; it is ignoring the physics that govern it. Master inductance, and you master both SI and PI.

Key Points

- Inductance shapes nearly every facet of signal integrity and power integrity.
- Moving a voltage between two components requires moving energy (not a signal).
- Mutual inductance is when two magnetic fields react and is the root cause of crosstalk, switching noise, and timing uncertainty.
- In the time domain, a decoupling capacitor acts as a local energy reservoir, supplying instantaneous current to the processor during fast load transients until the voltage regulator can respond.
- In the frequency domain, that same capacitor reduces the PDN's AC impedance around its self-resonant frequency, enabling more high-frequency current to reach the IC with minimal voltage noise.
- Every capacitor includes an equivalent series inductance (ESL) component, which causes its impedance to rise at high frequencies.
- The power/ground plane pair acts almost like a perfect capacitor, with negligible inductance (no leads) and very low equivalent series resistance (ESR).
- The loop area formed by the capacitor, its vias, and the planes determines mounting inductance.

Resources

Beyond Design by Barry Olney: "Micro-ohm Power Delivery Network for AI-driven GPUs," "The Fundamental Structure of Spectral Integrity," "Embedded Capacitance Material," "The Impact of PDN Impedance on EMI," "The Target Impedance Approach to PDN Design," "PDN Planning." **I-CONNECT007**



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



BY MARCY LARONT, I-CONNECT007

The Need for Speed in Mil/Aero Manufacturing

With James Hofer



MILITARY AND AEROSPACE manufacturers are being asked to move faster, whether developing new drone technologies, responding to supply constraints, or getting new designs from prototype to production. James Hofer of Accurate Circuit Engineering sees that pressure as changing more than just manufacturing schedules. He talks about the growing role of quick-turn PCB fabrication, new relationships between defense primes and startups, and why even long-established approaches to standards and manufacturing have to become more adaptable.

Marcy LaRont: James, I'd like to begin our conversation with your work in the Mil/Aero market, including some of the conferences you've attended in that space. What are you seeing right now?

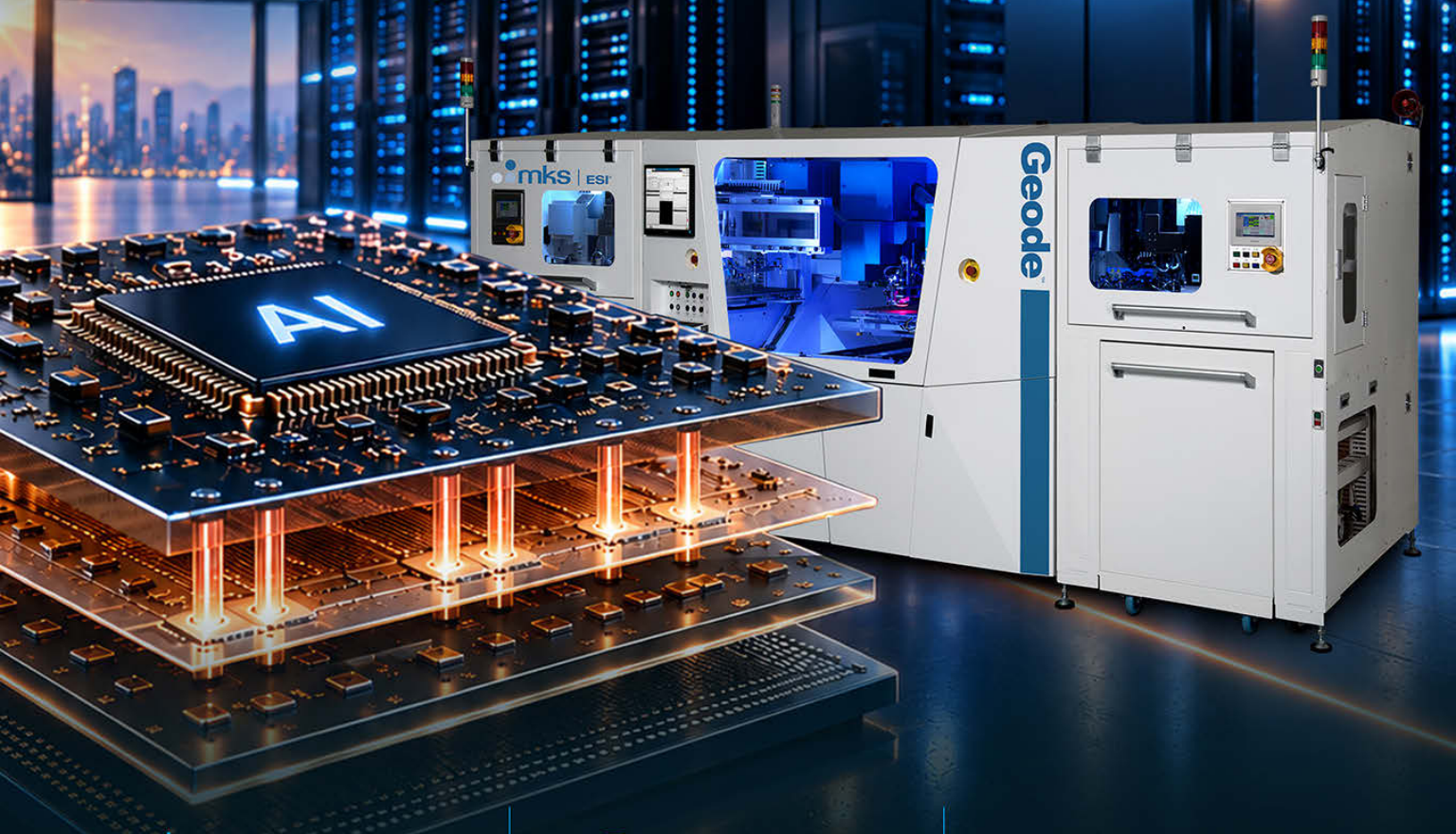
James Hofer: The industry is getting busier and, for many reasons, there is a need to increase the country's stockpiles of armament. We've learned that the defense industry needs to make a change, moving away from those multi-million-dollar missiles toward a more cost-effective consumable defense product.

mks
ESI

ADVANCED LASER VIA DRILLING

A NEW LINE OF POSSIBILITY.

For the innovators building what's next in AI.



PRECISION VIA GEOMETRIES.

CONTROLLED THERMAL IMPACT.

PRODUCTION-READY THROUGHPUT.

Discover what's possible with Geode™

Learn more at go.esi.com/geodeprecision

What is the primary driver for that?
It has been drones and their usage in military conflict. These multi-million-dollar missiles are basically giant drones locked onto a trajectory. Now, we have consumable drones that are much more adaptable than we previously thought, so we have to develop defenses against those drones.


James Hofer

We're learning that the aerospace industry needs to be able to pick something up from the battlefield, reverse engineer it, remodel it, and then repurpose it. That's exactly where ACE comes in because we are a quick-turn facility. There are plenty of manufacturers that can build tens of thousands of good boards in 10 to 20 weeks, but sometimes the military needs manufacturers that can build boards in a few days. We're not the only ones we are one of the fastest, and we definitely help fill that gap.

You also have to do it well, correct? A new PCB facility still has to fine-tune its process at the level that ACE is doing. Can one just pick up and do that?
No, you can't. You have to have what's called a process prove out, and there's just no way around that.

Right now, there are some other things going on as well. One of our resin suppliers in Saudi Arabia was damaged at the start of the Iran conflict, which has strained supplies of resins needed for epoxies. There's also an E-glass shortage, as AI data centers are trying to get boards as fast as they can and consuming more than their fair share of the raw materials used in many other types of PCBs. This is driving lead times for high-performance laminates to be above and beyond 10 weeks.

Tariffs are another issue; they are extremely high. Many copper-clad laminates are above 35% tariffs.

How are you managing those challenges?
It's affecting everyone, so in some ways, the field is level. But because our production needs change every day, we have developed special relationships with our suppliers, so we are able to stay on top of it. We've been staying busy and, as I said, we work quickly, which gives us a benefit in the market compared to some of our competitors.

I understand you've attended some conferences recently. How have those helped you understand the market better and increase business?
We actively participate in symposiums, conventions, and conferences throughout the year. It allows us to meet new customers and understand all the ways the industry is changing and where the priorities lie. At the Space Missile Defense Conference recently the most prominent topic was drone defenses.

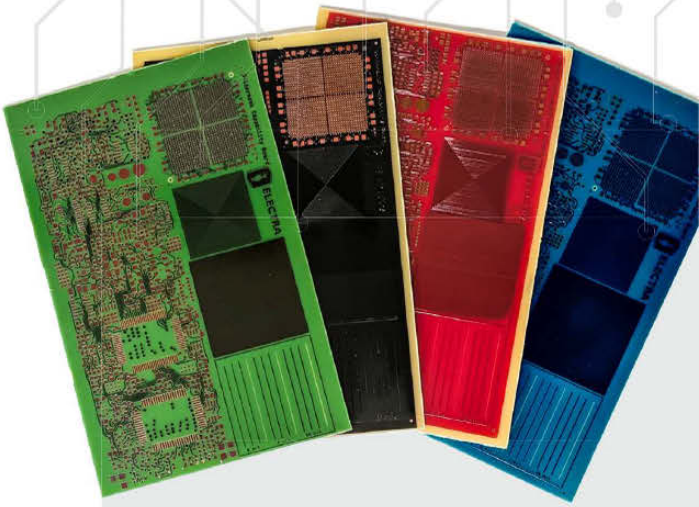
This also allows us to meet some of the defense startups that, once they get past proof of concept, often end up being backed by the large primes. If I see that, I know that's a company I want to work with because they're nimbler and can move faster than a traditional prime, which is what we need. We also like that they are like us in the ability to change, adapt, and respond quickly. The larger defense contractors tend to move a lot slower.

Looking at the bigger picture, what do you see as the state of the industry in aerospace and defense?
I see a complete renovation of how things are done. I see more diversification. I really think the primes want to partner with everybody. But I feel that once you bring these startups into the umbrella of the big companies, they become bogged down and slower. In the past, we tried to have everyone conform and make everything consistent, but we're learning that change is the new norm, and adaptability is key to success.

James, we've been talking about how quickly technology and manufacturing requirements are changing. What does that mean for standards and quality systems that traditionally emphasize consistency?
I think that might change our standards as well—AS9100 and IPC standards. We have always defaulted to: "Be consistent. Do exactly this. Develop and build this process and follow it." Instead, I think the future looks a bit more like LEGO® blocks. You can stack them this way, but you also have to be prepared to take them down quickly and stack them another way. We have long relied on our aerospace industry being very consistent. We're now learning that approach won't keep up. With AI coming into the

ELECTRA

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
- Black
- Blue
- Red

We Also Offer Flexible Soldermask:

- Green
- Amber

Key Features/Highlights

High Performance

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

Robust Formulation

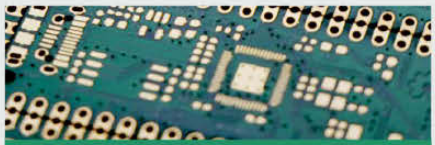
- Ink stable under a range of temperatures and environmental conditions
- No change in ink viscosity or particle size during use or shelf life of product
- Ink robustness critical for ensuring consistent printing with minimal problems
- Approved for use on leading inkjet printheads:
  - Konica Minolta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

Info@electrapolymers.com | electrapolymers.com

picture, change will be an everyday occurrence, and standards will need to be valuable within this reality.

That creates an interesting challenge for standards. Manufacturers may need to change much more quickly than the standards development process traditionally allows.

Absolutely. I believe it will become a roadblock at first. There will have to be exceptions, and then the question becomes whether the standards can keep up quickly enough to become more like living, breathing specifications rather than ironclad requirements developed by committees that can take months to work through a specification.

If the standards can't become nimbler, they could become less relevant, or potentially create a roadblock to getting products built. Manufacturers may end up relying more heavily on exceptions and sign-offs simply because the technology and processes have moved ahead of the existing requirements.

That means manufacturers and others working directly with these technologies will need to be involved. Those exceptions and new manufacturing realities will have to be brought forward quickly so they can be discussed and, when appropriate, incorporated into the standards.

Yes, that's exactly where I was going with this. The question is whether the standards process can keep pace with how quickly everything else is changing.

That seems especially important in industries like aerospace and defense, where standards and consistency exist for very good reasons. You don't want to lose the reliability and discipline those systems provide. At the same time, manufacturers are being asked to develop and build new technology faster than ever.

Right, we've always been very focused on consistency in aerospace, but nothing stays the same anymore. The only thing we can know will be consistent is change. I see that continuing over the next few years. It will no longer be, "This is what we do." It will be, "That's what we did last year. Now we do this." Maybe it becomes, "That's what we did last week. Now we do it this way." That's the difference I think we're going to have to learn to work with.

The minute we stop changing and growing and pushing that envelope and doing something that's not been done before is the minute we start to fall behind... we need to continue to create new technology every day."

To wrap up our conversation, given the limited capacity in U.S. manufacturing, do you see a situation where the government will require everyone to work together to create the volume that's needed?

I think it's very real, and if called upon, we would step up. We love what we do, and we would be very proud to do that for the right reasons. There will always be a need to keep making quick-turns for new technology. We're small compared to many other shops, but we're nimble, and we get things done more quickly than others can. We're used to being hands-on, so when someone gives us something that's never been done before, we understand and we work to do it. We do that type of work all the time.

The minute we stop changing and growing and pushing that envelope and doing something that's not been done before is the minute we start to fall behind. I firmly believe that we need to continue to create new technology every day.

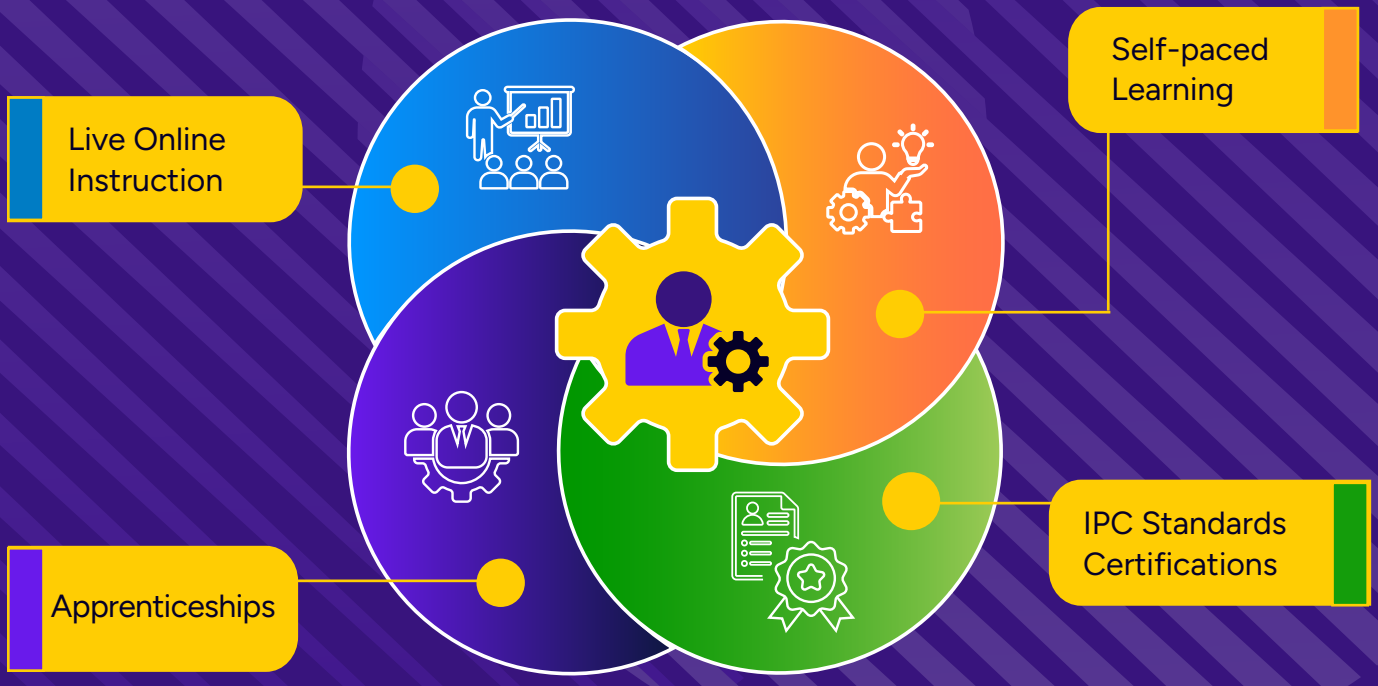
I hope that prototype PCBs are finally getting their day in the sun, because that is where some of the coolest work is done, and it's critically important in the overall ecosystem. James, thank you for catching up on these topics with me.

Yes, it's been great, thank you. **I-CONNECT007**

Electronics U
by Global Electronics Association

YOUR ELECTRONICS EDUCATION, CONNECTED

Electronics U and Global Electronics Association bring together self-paced learning, live online instruction, apprenticeships, and IPC Standards Certifications to create a connected education ecosystem for the electronics industry.



EXPLORE THE LEARNING ECOSYSTEM

www.electronicsu.org/connected-ecosystem-electronics-education



The Top 5 Scariest Things About AI in PCB Fabrication

BY STEVE WILLIAMS, THE RIGHT APPROACH CONSULTING

In last month's column, I wrote about how the "founding fathers of quality" would feel about AI. With the Halloween season upon us, I am looking at some cautionary points to consider when implementing AI in our factories.

A Genuinely Useful Technology With Genuinely Sharp Edges

AI is already inside the PCB fabrication process, whether a shop has formally adopted it or not. Automated optical inspection (AOI) systems have used machine learning for years. CAM and DFM checks are increasingly AI-assisted, and quoting engines predict yield before a panel ever sees copper. None of that is inherently frightening; most of it is a genuine improvement over manual methods that were slower and just as fallible in their own ways.

But the same properties that make AI powerful in this industry—speed, pattern-matching at scale, and the ability to make decisions faster than a human can

GREENSOURCE
FABRICATION

Built for What's Next in Advanced PCB Manufacturing



Reliability

Highly reliable stacked and staggered microvias (50 to 250 µm), backed by industry-leading blind via formation.

Sustainability

Environmentally sustainable Zero Liquid Discharge (ZLD) wastewater treatment system.

Capability

Sub-35 µm features with advanced mSAP and SAP trace formation.

100% USA Made | ISO 9001 | AS9100 | ITAR Registered

greensource.us

info@greensourcefab.com

Scan to learn more

review them—are exactly what make it dangerous when it goes wrong. When we are fabricating boards that fly in aircraft, power pacemakers, and guide munitions, "wrong" isn't an abstract risk. Here are the five developments that should keep quality engineers and program managers up at night.

1. Black Box Defect Decisions on Boards That Can't Fail

AOI and automated X-ray inspection systems rely on deep learning models to classify defects like a solder void, a trace anomaly or a delamination signature, rather than the rule-based thresholds of a decade ago. The upside is real: These models catch subtle defects that rule-based systems miss entirely. The frightening part is that most of them can't fully explain why they flagged (or passed) a given board.

For a consumer product, an opaque pass/fail decision is a business risk. For a board destined for a flight control system or an implantable medical device, it's a traceability problem with legal and safety consequences. If a model passes a board that later fails in the field, "the algorithm said it was fine" is not an acceptable root-cause finding in an AS9100 or DO-178 audit trail. The industry is adopting inspection AI faster than it's building the explainability infrastructure to justify those decisions to a customer, an auditor, or an accident investigation board.

2. Your Stackup Is Training Somebody Else's Model

Every AI-powered DFM tool, thermal simulation platform, and quoting engine needs data to function, and much of that data arrives in the form of a customer's Gerber files, BOM, and layer stackup, which is among the most sensitive intellectual property a fabricator handles. When that data flows through a cloud-based AI service, the fabricator has to trust that it isn't being retained, aggregated, or used to improve a model that a competitor might eventually query.

This isn't a hypothetical. Design data uploaded to a poorly vetted AI tool can unintentionally contribute to a model's general knowledge in ways that are nearly impossible to audit or undo after the fact. For defense and aerospace work specifically, this collides directly with ITAR and DFARS data-handling requirements. A cloud AI tool with servers or training pipelines outside the continental U.S., or with unclear data retention policies, can turn a routine DFM check into an export-control incident. The scary part isn't malicious intent; it's how easy it is for an engineer under deadline pressure to paste a stackup into a convenient tool without asking where that data actually goes.

3. Institutional Knowledge Quietly Atrophying

A process engineer with 20 years on the floor knows things no dataset captures cleanly: how a particular laminate behaves in humid August weather in Wisconsin or Arizona, why plating line tank 3 runs differently than tank 1, or which customer's "standard" tolerance is actually tighter in practice than what's written down. AI models trained on historical process data can approximate a lot of this, but they approximate the documented, quantifiable version of that knowledge, not the tacit, contextual version that lives in an experienced engineer's head.

The frightening trajectory isn't AI replacing that engineer outright. It's a slower erosion: as AI-assisted tools handle more routine judgment calls, the next generation of engineers gets fewer reps making those calls themselves, and the tacit knowledge that used to get passed down through the model doesn't build up in the same way. A decade from now, when a genuinely novel process excursion happens, something that's outside the training data's distribution, the industry may have fewer people left who've built the pattern-recognition to catch it, precisely because AI had been catching the routine version of that problem for years.

4. A Much Bigger Attack Surface on Systems That Were Never Designed to Be Secure

AI-driven manufacturing execution systems, connected inspection equipment, and predictive maintenance platforms all depend on continuous data flow between shop-floor equipment and often cloud-based processing. Every one of those connections is a potential entry point, and PCB

fabrication equipment, much of it running on older, purpose-built control systems, was never designed with modern cybersecurity in mind.

For defense-adjacent fabricators already navigating CMMC 2.0 compliance, this is not a theoretical concern. An AI-connected pick-and-place machine or AOI system represents a new node on the network that has to be inventoried, secured, and monitored, often faster than the compliance program can keep pace. A compromised AI-integrated system doesn't just risk stealing data, it can be used to subtly alter process parameters or inspection thresholds, creating defects that pass inspection by design. The scariest version of this scenario isn't a smash-and-grab data breach; it's a quiet, sustained manipulation of quality data that nobody catches until a field failure forces the question.

5. Confidently Wrong, and Nobody Double-Checks

Generative AI tools are being used to draft root-cause analyses, suggest DFM corrections, and even propose process parameter adjustments. These tools are fluent and confident regardless of whether they're right, and that fluency is exactly the problem. A hallucinated root cause that sounds plausible and cites reasonable-seeming mechanisms can sail through a review process precisely because it reads like something a competent engineer would have written.

The danger compounds under deadline pressure, when a corrective action report needs to close out before a customer audit, and an AI-drafted explanation is the path of least resistance. Rubber-stamping an AI-generated root cause without independently verifying it against the actual physical evidence is how a plausible-sounding but wrong explanation becomes the official record, and how the real root cause keeps causing defects long after the paperwork says it was fixed.

The Common Thread

None of these five risks argues against using AI in PCB fabrication; the technology's benefits are too significant to walk away from. What they argue for is treating AI outputs the way any good quality system treats a new, unproven process: with verification, traceability, and a healthy default skepticism, rather than the assumption that speed and confidence are the same thing as correctness. **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.



BY ANAYA VARDYA, AMERICAN STANDARD CIRCUITS

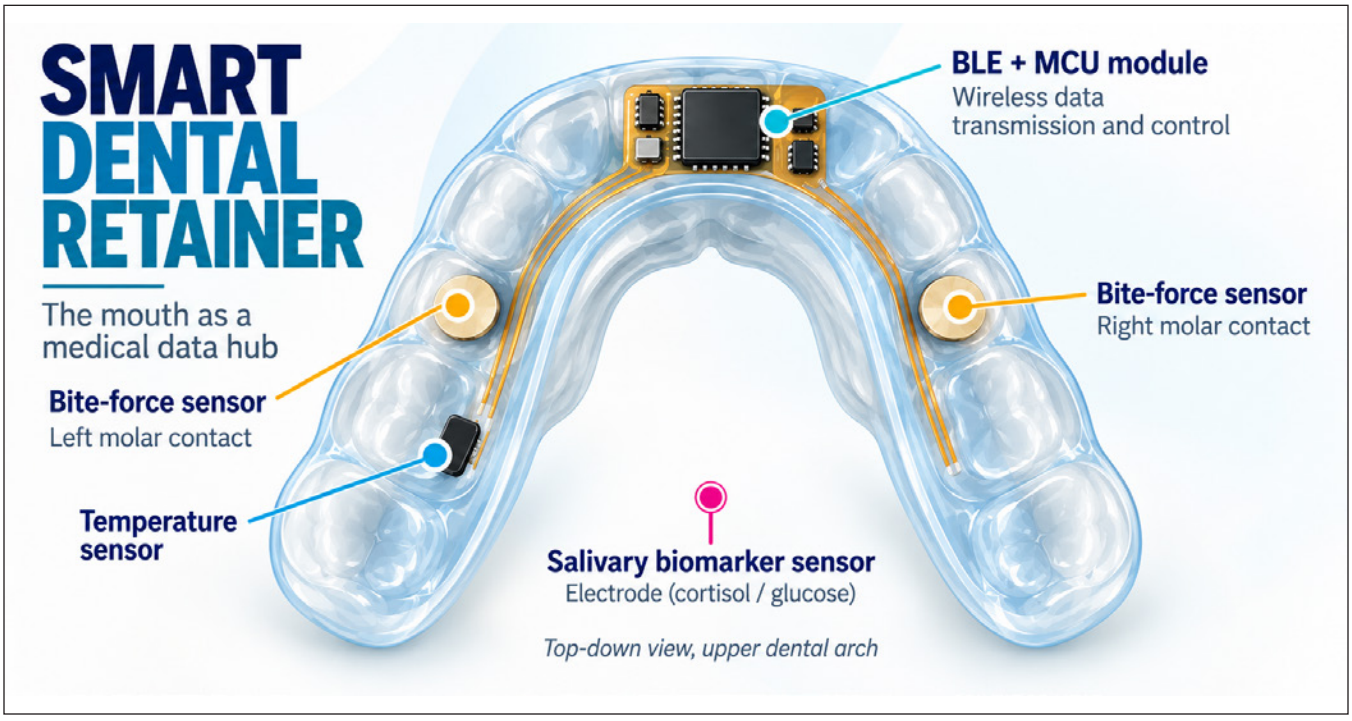
Smart Dental Retainers With Flex PCBs

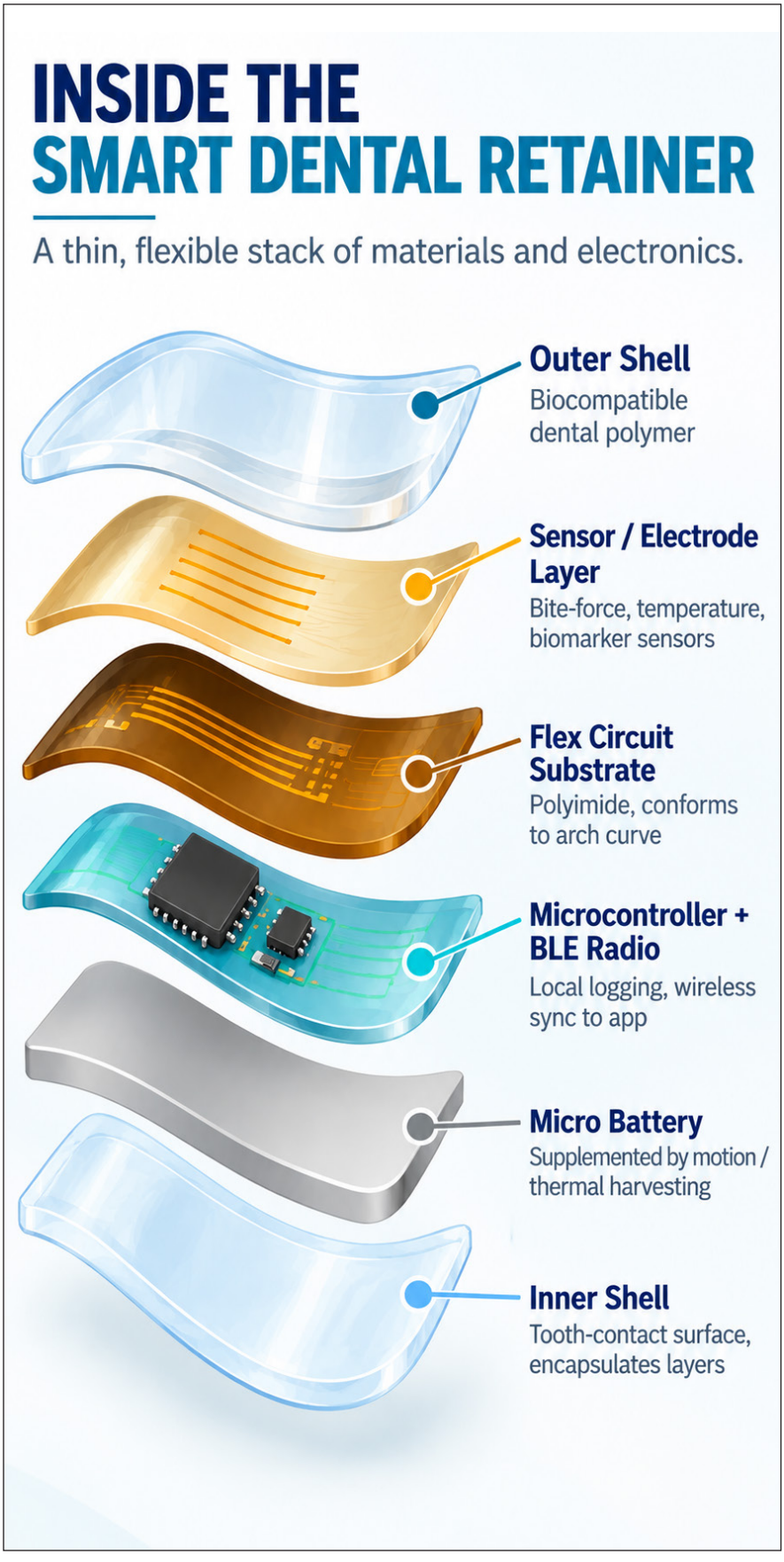
The retainer is one of the least glamorous devices in medicine: a thin plastic shell that patients are told to wear and mostly forget about. That obscurity is exactly what makes it interesting to device engineers right now. The mouth is a warm, wet, chemically rich environment sitting inches from the brain, in near-constant contact with saliva, teeth, and soft tissue. It is also one of the few places on the body a person willingly places a custom-fitted object for hours at a stretch, night after night, without complaint.

But embed a sensor package into that shape, and you have a monitoring platform that most other wearables can only approximate. A new generation of smart retainers is integrating flexible sensors, microcontrollers, and wireless radios into the same thermoformed or 3D-printed shells that orthodontists have used for decades, turning a passive compliance device into an active data-collection point.

What's Actually Inside

The engineering challenge starts with geometry. A retainer's shell curves in three dimensions and flexes as it's inserted and removed, which rules out





conventional rigid PCBs. The electronics package has to be built on flexible or rigid-flex circuitry thin enough to laminate into or between layers of dental-grade polymer without changing the fit the orthodontist prescribed.

Within that thin footprint, a functional smart retainer typically integrates several sensing elements at once. Bite-force sensors, often piezoresistive or capacitive elements printed directly onto the flex layer, track occlusal pressure and can flag bruxism (nighttime teeth grinding) from the pattern and intensity of contact. Temperature sensors monitor intraoral conditions, which can shift with inflammation or infection. Some designs incorporate electrochemical sensors capable of detecting biomarkers in saliva. Cortisol, glucose, and inflammatory markers have all been demonstrated in research platforms, positioning the retainer as a passive diagnostic tool rather than just a mechanical one.

A low-power microcontroller processes the sensor data locally, and a Bluetooth Low Energy radio transmits it to a companion phone app while the device is worn. Power remains the tightest constraint in the whole system: batteries have to be small enough to fit the shell and safe enough to sit against soft tissue, which has pushed several programs toward hybrid approaches: a small rechargeable cell supplemented by energy harvested from jaw movement or thermal gradients, extending run time between charges that happen while the retainer sits in its case during the day.

Why It Matters Clinically

The most immediate application is the one every orthodontist already wants solved: compliance. Retainers only work if patients wear them, and self-reported wear time is notoriously unreliable. A sensor that logs actual insertion time gives the clinician objective data, turning a guessing game into a chart they can review at a follow-up visit.

Beyond compliance, the sensing package opens two other clinical paths. The first is sleep and bruxism monitoring, because the retainer sits in the mouth overnight, exactly when bruxism and many sleep-disordered-breathing symptoms occur, it can capture bite-force and jaw-position data that a clinic-based sleep study only samples for a night or two. The second is longitudinal biomarker tracking. Saliva already carries diagnostic information about hydration, hormonal cycles, and early inflammation: a device that sits in contact with it for hours every day, rather than requiring a single lab draw, can build a trend line instead of a snapshot.

The Engineering Constraints That Keep This Hard

None of this is simple to build. Every material in the assembly—the flex substrate, the encapsulant, the electrode surfaces—has to meet biocompatibility standards for prolonged intraoral contact, which is a materially stricter bar than most wearable electronics face. Encapsulation has to survive not just moisture but the mechanical flexing of insertion and removal, thousands of cycles over a typical retainer's service life, without a seal failure that lets saliva reach the electronics.

Wireless performance is compromised by the attenuating effect of tissue and jaw structure, so antenna design has to work within a shape dictated by dentistry rather than radio-frequency convenience. And because these are Class II medical devices in most jurisdictions, once they carry a diagnostic claim, the regulatory pathway shapes the sensor selection as much as the underlying physics does.

Where This Goes Next

The trajectory is familiar from other corners of wearable medicine: a device built for one narrow clinical purpose accumulates sensing capability until it becomes a general-purpose monitoring platform. A retainer built to track wear time is a short step from one that tracks grinding, and a shorter step still from one that continuously samples saliva chemistry.

For flex circuit fabricators and dental device manufacturers alike, the near-term opportunity isn't a single breakthrough sensor; it's the disciplined miniaturization and packaging work needed to fit real diagnostic capability into a shape the mouth has been wearing, unremarked upon, for generations.



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*.



宝丰堂 boffotto

PLASMA TECHNOLOGY
FOR A MORE CONNECTED WORLD

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 m³/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, los 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

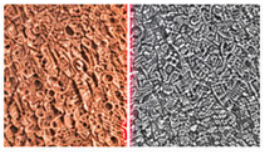
- Vertical electrode design
- Desmear, cleaning and activation
- For PCB, FPC and HDI applications
- Stable process control
- Flexible for multiple materials
- Compact footprint



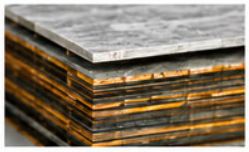
PROVEN PROCESS RESULTS



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
Flexibile for Your Applications
Scalable for Your Production Needs

all4-PCB® NORTH AMERICA

Northrop Grumman to Accelerate Extended-Range C-UAS Missile Tech for U.S. Army ▶

To address the evolving challenges of detecting and defeating unmanned aerial systems at extended ranges, Northrop Grumman will develop next-generation air-breathing propulsion for the U.S. Army, significantly enhancing the range and air defense capabilities of the Army's eXtended Range Counter-Unmanned Aircraft System (XRC).

Virtus Solis Signs 20-Year Space Solar PPA with Brae Systems for 250 MW Clean Power ▶

Virtus Solis Technologies, Inc., a pioneer in the delivery of space-based solar for terrestrial baseload power, and Brae Systems, announced the closing of a 20-year Power Purchase Agreement (PPA) in which Virtus Solis will supply continuous, space-based solar power to Brae's advanced computing infrastructure. The deal represents the first solar PPA in space and Virtus Solar's first in a series of commercial offtake agreements over the next several months.

PCB Technologies Announces $38M Engagement with U.S. Aerospace and Defense Manufacturer ▶

PCB Technologies continues to expand its operations in the U.S. aerospace and defense market, announcing a new agreement valued at approximately $38 million with a leading American defense and aerospace manufacturer.

Q/C Technologies Announces Research Collaboration with Sandia National Laboratories ▶

Q/C Technologies, Inc., a developer of next-generation optical computing solutions, announced a collaboration with the Center for Integrated Nanotechnologies (CINT) at Sandia National Laboratories (Sandia) to advance its proprietary optical processing unit (OPU) initiative for artificial intelligence inference. The Company is pioneering next-generation optical computing solutions designed to harness the properties of light to perform complex computations naturally via interference.

U.S. Navy Selects Boeing for F/A-XX Program ▶

The U.S. Navy announced Boeing has been awarded a multi-billion-dollar contract to produce its F/A-XX aircraft. The fighter will revolutionize advancements in open mission systems architecture, advanced mission systems and operational reach and capacity. F/A-XX is intended to replace the service's fleet of F/A-18 Super Hornets.

General Dynamics Lands $184M Navy Electronic Warfare Contract ▶

General Dynamics Mission Systems has received a $184.25 million U.S. Navy contract for full-rate production of the Next Generation Electronic Attack Unit (NGEAU), which will support the EA-18G Growler electronic warfare aircraft. The system uses an open mission systems processor architecture designed to provide advanced electronic attack capabilities while allowing the Navy to incorporate technology upgrades over the life of the platform.

Aitech Introduces New C165 Rugged Single Board Computer to Help Defense Programs Build and Modernize Fielded VME Systems ▶

Aitech, a global leader in AI-powered rugged embedded computing and connectivity solutions for defense and space applications, today announced the C165 Rugged 6U VME Single Board Computer (SBC), helping defense and aerospace organizations build or modernize mission-critical systems while preserving proven VME infrastructure already deployed across air, land and naval platforms.

Leonardo DRS Awarded $96 Million in Contracts to Provide Critical MK 41 Vertical Launching System Electronics Hardware ▶

Leonardo DRS, Inc. announced that it has been awarded contracts totaling $96 million to provide critical electronics hardware for the MK 41 Vertical Launching System (VLS). The MK 41 VLS is the U.S. Navy's premier shipborne missile launching system and is also fielded by 14 allied navies worldwide. The awards reinforce Leonardo DRS's role in supporting naval combat system readiness with trusted, mission-critical electronics.


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
ITAR

While Congress Debates, America's PCB Competitors Invest Billions

BY DAVID SCHILD, PCBAA

The story of offshoring is all too familiar in the electronics industry. For decades, PCB manufacturing shifted overseas, taking advantage of foreign subsidies, making it difficult for U.S. companies to compete. America once boasted more than 2,000 PCB shops. Today, that number has shrunk to less than 140. This is not simply the marketplace at work. As I have said repeatedly to our elected officials in Washington, American companies are not losing to their foreign competitors; they are losing to foreign governments.



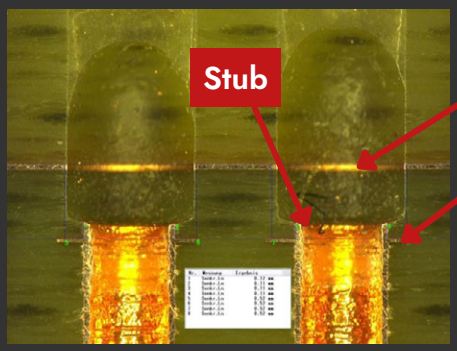
This did not happen overnight. Asian competitors, especially China, have directed their industrial policy toward market dominance. American business tends to focus on the next quarterly earnings call while our competition focuses on the next decade.

Because of an industrial policy that includes production subsidies, state-backed tax incentives, and special economic zones, China now controls more than 60% of the PCB market. China's designs to control supply chains don't stop at its border. In just the past five years, China has invested almost $2 billion in Thailand and another $310 million in Vietnam.

Over the same period the U.S. investment in PCB production was $118 million. Almost all of this was through the Defense Production Act (DPA). Understanding the critical nature of PCBs,

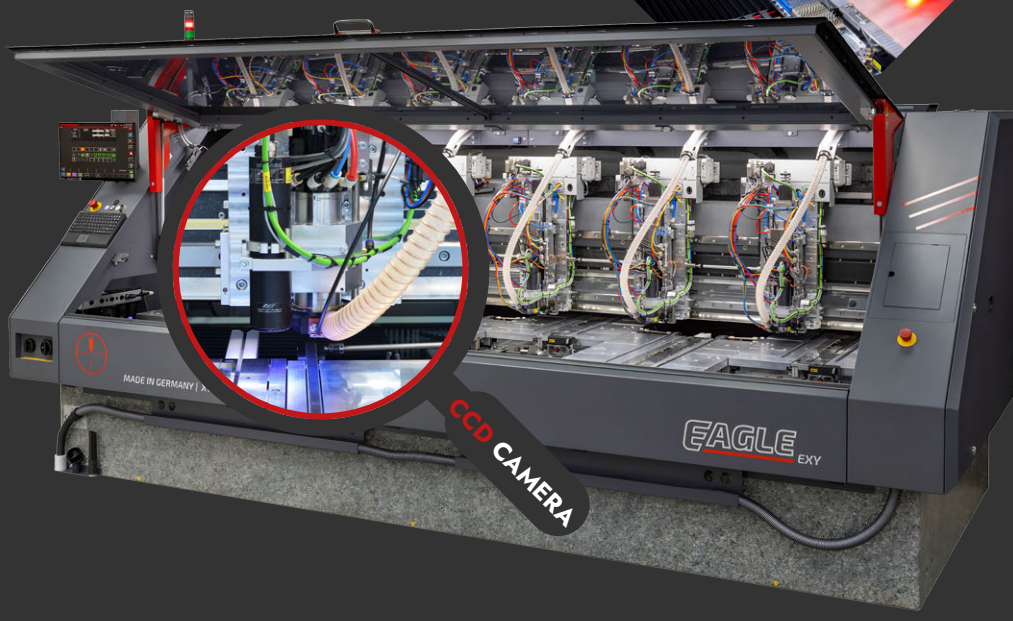
EAGLE EXY

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

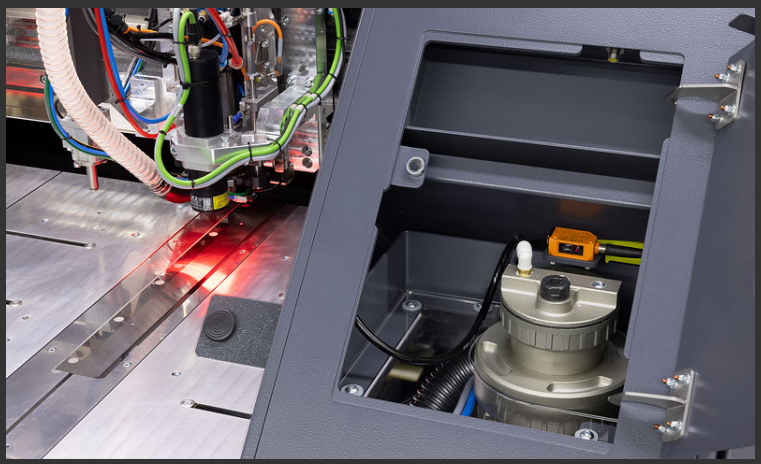




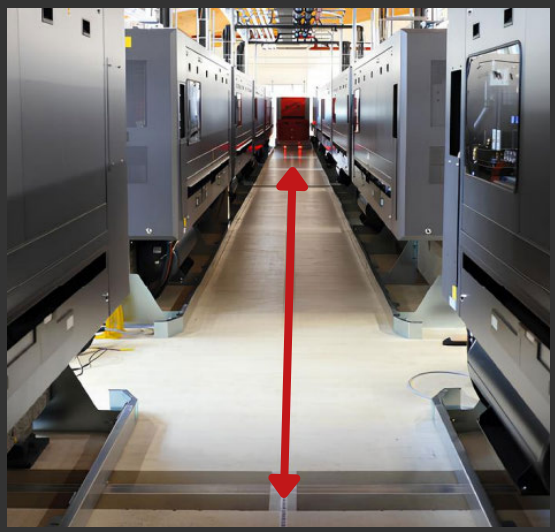


Depth Drilling: precise and reliable drilling to defined inner layers, enabling high-quality blind vias.

AUTOMATED COLLECT CLEANING (ACC)



AUTO LINE SHUTTLE (ALS) AUTOMATION



Congressional appropriators and Pentagon officials invested in three new facilities:

- Calumet Electronics opened a new facility in Michigan using DPA funds.
- GreenSource received DPA funding to upgrade its production facilities.
- TTM Technologies opened a new UHDI PCB manufacturing facility in Syracuse, New York, made possible by TTM investment and tax assistance from New York State and a grant from the Department of Defense for an extensive plant expansion.

These DPA investments prove that we can compete, but their scale is inadequate relative to the capital being deployed across Asia. If advanced PCBs are foundational to U.S. economic and national security, can an investment strategy of tens or hundreds of millions compete with an Asian PCB expansion cycle measured in billions?

We know that public action moves private capital. Policymakers should look to the CHIPS Act, which reinvigorated the American semiconductor industry by attracting more than $460 billion in private investment. The same scenario applies to the American PCB industry. American PCB manufacturers are ready to scale up to reduce the dependency on Asia, but they need supportive industrial policy, legislation, and government investment to reshore and restore capacity.

This is why PCBAA is on the ground in Washington, educating policymakers, legislators, and officials at the White House and Department of Commerce. A growing team gets better results, and we all have a role to play. From critical materials to fabrication and from assembly to OEM, every company in the PCB ecosystem benefits from smart industrial policy.

We are only a few weeks away from electing the 120th Congress. PCBAA looks forward to working with our elected officials to confront the challenges we face and turn shared ambition into lasting progress.



David Schild is executive director of PCBAA. To read past columns from PCBAA, click here.





The materials behind what's next.

As electronic designs become more demanding, engineers need materials that deliver proven reliability and performance.

For decades, Kapton® polyimide films and Pyralux® flexible circuit materials have helped enable innovation across the world's most advanced applications. Backed by a legacy of materials science leadership, Qnity continues to help shape what's next.

Scan to learn more about Kapton®

Scan to learn more about Pyralux®

© 2026 Qnity Electronics, Inc. All rights reserved. Qnity™, the Qnity Node Logo and all trademarks and service marks designated with ™, SM or ® are owned by affiliates of Qnity Electronics, Inc. unless otherwise noted.



Is a Printed Silver Ink Circuit Right for Your Application?

BY TERRILL SCHMIDT, FLEXIBLE CIRCUIT TECHNOLOGIES

When is the right time to design a flexible circuit with printed ink, usually silver, vs. a flexible etched copper circuit, a die-cut copper circuit, or a die-cut aluminum circuit?

First, a flexible circuit design must work in the application as designed and be cost-effective so the end product can be sold or used. What drives a designer to choose between an etched copper circuit, printed ink, flat flexible cable, die-cut copper, or die-cut aluminum? Multiple factors influence the design.

Unfortunately, limited knowledge or experience can prevent designers from considering alternatives because they may not know the design rules, limitations, or capability limits of each option. As a result, they often stay with what is known and proven, missing opportunities to reduce cost or provide another option to quote.

At this stage, several factors should be considered when determining how the flexible circuit should be designed:

Functionality: What electrical connections are required? Does the circuit need to support switching, sensing, lighting, components, or a full schematic? Are there high-speed, shielding, impedance, or resistance requirements?

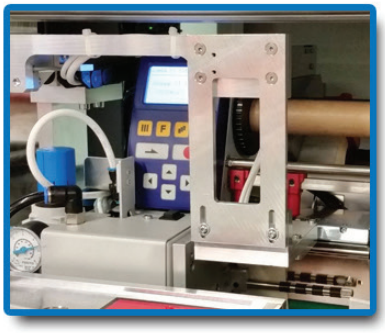
Component requirements: Does the design require LEDs, sensors, switches, or SMT components? How will they be attached? Are soldering or conductive epoxy required?

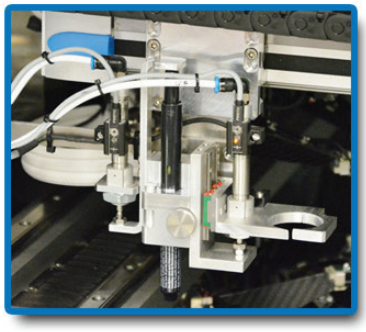
Temperature limits: What temperatures must

Unrivaled Test Speed with Full Automation



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

atg MYCRONIC

the circuit withstand during operation and during product assembly?

Manufacturability: What are the estimated line widths and spacing? Is this a fine-line application, an extremely long circuit measuring several meters, or a standard design?

Production volume by area: Is the product expected to run in hundreds, tens of thousands, or millions of units per year? Flexible circuits are usually measured in finished square area (square meters). What is the finished square area of the required production? Depending on the circuit type, cost may be driven by square meters, linear meters, or conductive ink usage.

Printed inks, including printed silver, are an option often missed. Printed inks have been used in circuits for decades, typically in membrane switches or graphic overlay applications. With material costs increasing, especially gold and silver reaching record levels, printed ink should be considered as an option. Printed inks have supported components for decades; millions of desk phones with LEDs were made with printed silver LED circuits. Most home appliances from the past 30 years with a touch panel or graphic overlay, such as microwaves, use printed ink circuits. These technologies have been around for a long time and can be cost-effective.

Over time, printed ink technology has also advanced, leading to several advantages:

Lower cost, reliable substrates. Lower-cost substrates can be reliable when the application does not require high-temperature performance or flame-retardant materials. In those cases, printed ink circuits may use substrate materials that meet the application's need without adding unnecessary material cost.

Volume processing. Lower-cost substrate materials can improve processing efficiency in volume manufacturing, supporting lower-cost production when the design is compatible with printed ink construction.

Improved inks. Improved inks have expanded the areas where printed ink circuits can be used. For example, human-biocompatible inks significantly grew the demand in medical applications such as disposable wearables for monitoring, diagnostics, and stimulations, as well as biosensors and assays. Clear conductive inks have enabled capacitive touch and other touch-sensing system designs in automotive and appliance applications.

Improved processing. Advanced screened dielectrics can now support more complex designs, including multiple layers. Combined with lower-cost substrates, volume processing, and improved inks, these process advances create additional cost-effective options for advanced designs using printed ink circuits.

As with each of the circuit types, every circuit technology has trade-offs.

Printed inks can offer several advantages, including lower cost, faster manufacturing, less tooling requirements, and improved environmental processing because they do not require harsh etching or plating chemistries. Potential limitations include lower maximum operating temperatures, larger minimum line widths and spacing, and limited SMT compatibility.

Production volumes also affect circuit selection and design. Many circuit types have ideal quantities where material usage and cost are optimized. This is typically evaluated by area rather than by the number of parts. With an etched circuit, cost is usually based on square meters. With a die-cut copper or aluminum circuit, cost is often based on linear meters of circuit from a roll. For printed ink, cost is a combination of the base substrate and the amount of ink used, usually measured in kilograms. As a result, ideal printed ink volumes differ from etched copper volumes, creating different price-point advantages at different production levels.

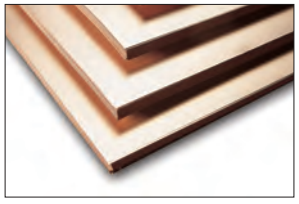
The key to designing a cost-effective flexible circuit is to meet the functional requirements at the lowest practical cost. To do that, designers should evaluate all viable constructions, including printed ink circuits, before selecting a path. When the best option is not clear, consult a flexible circuit design expert to help compare materials, processes, design rules, and cost drivers.

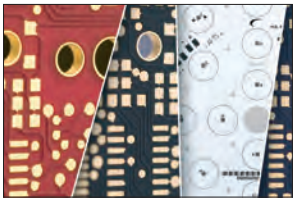


Terrill Schmidt is director of technology at Flexible Circuit Technologies. To read past columns from Flexible Circuit Technologies, click here.

Panasonic

Multi-layer & Flex Laminates





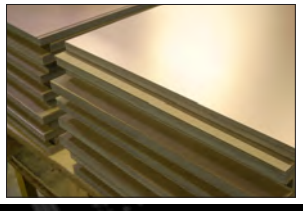
peters

Printed Circuit Coatings

CIRCUIT FOIL

Electro-deposited Copper Foils





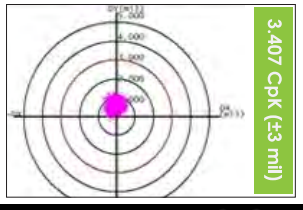
KB KINGBOARD

Rigid & Multi-layer CCL

PERFECT POINT PRECISION CARBIDE TOOLS

Drills & Routers





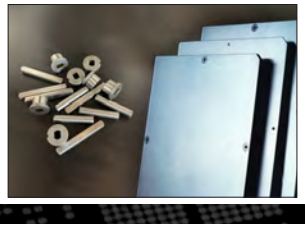
APEX ALUMINUM

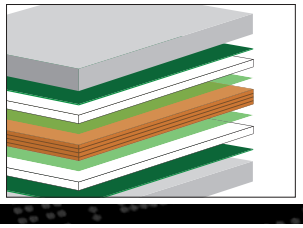
COOLSTRIKE LUBRICATED BACKUP MATERIAL

Drills Entry & Backup

IRON MOUNTAIN

PCB Press Plates & Tooling





EuroAssist PRODUCTS FOR PCB LAMINATION

PCB Lamination Assist Products

Hakuto

Dry Film Resist Lamination





AE ADVANCED ENGINEERING

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



Why Engineers Are Switching to *Ceramic Substrates*

BY BRIAN BUYEA, REMTEC

Walk into any design review, whether it's for an EV inverter, a radar system, or a high-frequency amplifier, and you'll hear the same concern repeated in different ways: "How are we managing the heat?"

That question, more than anything else, is driving one of the most important material shifts happening in electronics today. Engineers are moving deliberately and increasingly toward ceramic substrates. Not because they're trendy, but because they solve problems that traditional materials simply can't solve anymore.

What Is a Ceramic Substrate?

Let's break it down in plain language. At its core, a ceramic substrate is exactly what it sounds like: a base material made from ceramic—most commonly alumina (Al_2O_3) or aluminum nitride (AlN)—used to

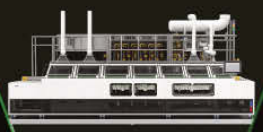
GREENSOURCE ENGINEERING

COMPLETE SOLUTIONS FOR

PCB & IC SUBSTRATE

ADVANCED TECHNOLOGY · END-TO-END SOLUTIONS

PRE-CLEAN
Remove contaminants and particles for



DEVELOPER
Remove photoresist to create the desired pattern



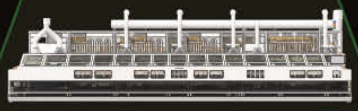
ETCH
Etch away superfluous copper for circuitisation



STRIPPING
Remove remaining resist and residues



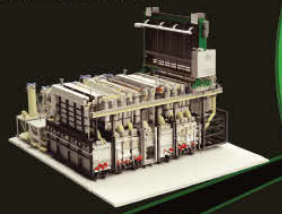
ADVANCED ADHESION PROMOTION
Optimized adhesion for superior buildup



SOLDERMASK DEVELOPER
Develop soldermask pattern for protection and insulation



VERTICAL ELECTROPLATING
Panel and pattern copper plating for PCB & IC substrate



CLOSED-LOOP ZERO LIQUID DISCHARGE (ZLD)
Process water recycling system

ARK™ AQUA REGEN KINETICS BY GREENSOURCE ENGINEERING

TURNKEY FACTORY PROVIDER



•ENGINEERED •INTEGRATED •DELIVERED
ONE PARTNER-ONE SYSTEM
COMPLETE SOLUTION

BUILT FOR PERFORMANCE. ENGINEERED FOR THE FUTURE.

- PCB
- HDI
- IC SUBSTRATE
- SPECIALIZED EQUIPMENT DESIGNED ON DEMAND
- AUTOMATION
- ENVIRONMENTAL SOLUTIONS

support and connect electronic components.

Think of it as the foundation of your circuit. Instead of fiberglass and epoxy (like traditional PCBs), you're building on a material that behaves more like a heat spreader, a structural backbone, and an electrical platform all at once.

Alumina is the workhorse: cost-effective, stable, and widely used. Aluminum nitride steps it up with dramatically higher thermal conductivity, making it ideal for high-power applications.

Therefore, a simple definition is that a ceramic substrate is a high-performance circuit base material designed to handle heat, power, and environmental stress far beyond what standard PCB materials can tolerate.

How Ceramics Differ from Traditional PCB Materials

Most engineers cut their teeth on FR-4. It's familiar, versatile, and affordable, but it was never designed for the demands we're placing on electronics today.

FR-4 is a composite, woven fiberglass reinforced with epoxy resin. It's electrically reliable and mechanically sound for a wide range of applications. But when thvings get hot, it starts to show its limits.

Ceramics are fundamentally different because they don't soften under heat, rely on resin systems that degrade over time, or trap heat. In fact, they move it.

Where FR-4 insulates, ceramics conduct. Where FR-4 manages, ceramics perform. So, this isn't about replacing PCBs across the board. It's about recognizing that different problems require different materials, and in high-performance environments, ceramics are often the better answer.

Thermal Conductivity Explained in Real Terms

When you remove the jargon, you understand that thermal conductivity is simply how well a material moves heat away from where it's generated. FR-4 has a thermal conductivity of roughly 0.3 W/mK. Alumina sits around 20–30 W/mK, and aluminum nitride can exceed 170 W/mK.

That's not a small difference. That's a completely different category. In real-world terms, this means:

- Heat doesn't build up around components
- Devices run cooler and more efficiently
- Thermal management systems can be simpler and smaller
- Reliability improves dramatically over time

If you've ever had a design fail not because of electrical issues but because something overheated, you already understand why this matters.

Why Heat—Not Just Signal—Is Driving Decisions

For years, material selection in electronics was driven by signal integrity, cost, and manufacturability. Those still matter, but they're no longer the limiting factors. It's heat.

Power densities are rising, devices are shrinking, and performance expectations are climbing. In many applications, thermal failure is the first failure: You can design a perfect circuit electrically, but if you can't get the heat out, it won't matter.

Engineers are rethinking materials at the substrate level because solving thermal challenges after the fact—with heat sinks, fans, or complex cooling systems—is often less effective and more expensive than addressing it at the source.

Ceramics don't just help manage heat. They eliminate it as a bottleneck.

Reliability in the Real World

Now let's talk about what happens outside the lab. Electronics today don't live in controlled environments. They're in vehicles, aircraft, satellites, industrial systems—places where conditions are anything but stable.

Ceramic substrates excel here because they handle high temperatures without degradation and resist thermal cycling—repeated heating and cooling—without cracking or delaminating. They maintain mechanical integrity under vibration and shock.

You can clearly see the difference. FR-4 can perform well under normal conditions, but push it into extremes, and its limitations show up fast. Ceramics are built for those extremes.

Where Ceramics Outperform

There are specific applications where ceramic substrates aren't just better; they're becoming essential.

Electric vehicles are a prime example. Power



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

Every day we get questions like those. And we love it. We have more than 600 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/faq-about-printed-circuit-boards/#ask

Reliable answers. Reliable PCBs.

NCAB GROUP

modules in EVs generate significant heat, and efficiency is everything. Ceramic substrates, especially aluminum nitride, help manage that heat while maintaining electrical performance.

In aerospace and defense, reliability isn't optional. Systems must perform in extreme temperatures, under constant vibration, and over long lifecycles. Ceramics provide the stability that those environments demand.

In RF and microwave applications, signal integrity and thermal management go hand in hand. Ceramic materials support high-frequency performance while dissipating heat effectively.

Satellite communications, radar systems, and industrial drives are no longer edge cases. They're growth markets, and they're driving material innovation.

Cost vs. Value: A Different Conversation

Let's address the obvious question. Yes, ceramic substrates are more expensive than traditional PCBs, but that's the wrong comparison. The real question is: What does failure cost?

If a lower-cost material leads to overheating, reduced lifespan, or field failures, the initial savings disappear quickly. Add in redesigns, downtime, warranty issues, and lost reputation, and suddenly the "cheaper" option becomes the most expensive one you could have chosen.

Ceramics shift the conversation from cost to value. They reduce risk, improve performance, and extend product life. In high-reliability applications, that's not a luxury. It's a requirement.

When Not to Use Ceramics

Now, ceramics are not the answer for everything. If your application is low power, cost-sensitive, and operating in a controlled environment, FR-4 or other traditional materials are often the right choice. Consumer electronics, simple control systems, and many standard applications also don't require the performance that ceramics provide. Forcing ceramics into those designs doesn't make them better; it just makes them more expensive.

So, good engineering is about choosing the right material for the job. Not the most advanced, but the most appropriate.

The Bottom Line

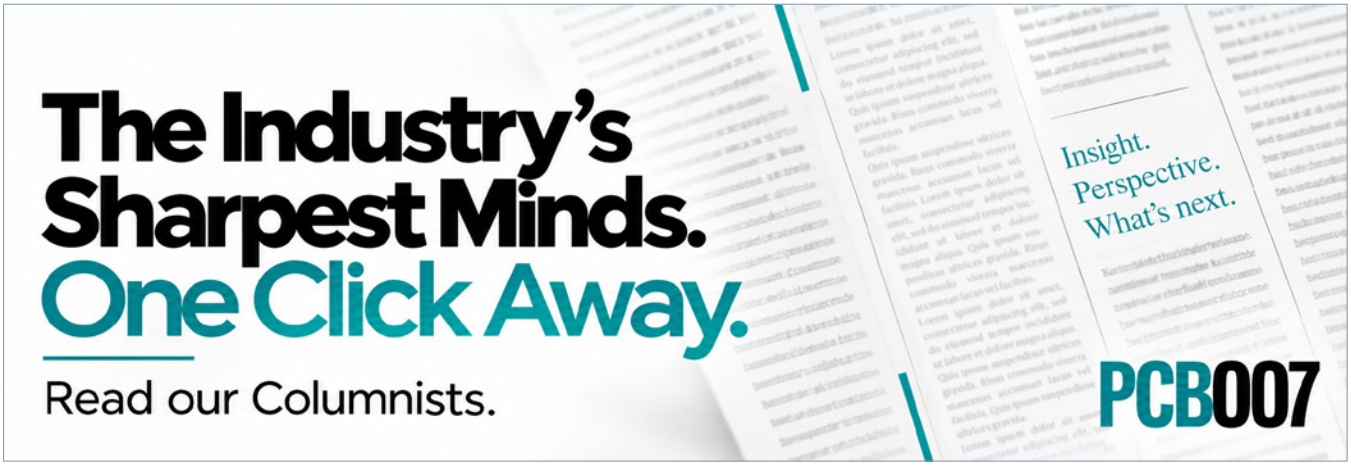
Ceramic substrates are not new, but their role in modern electronics is changing rapidly. As power increases and reliability expectations rise, environments become more demanding, pushing engineers to rethink the foundations of their designs.

Ceramics can solve heat at the source, perform under pressure, literally and figuratively, and they offer a level of reliability that traditional materials struggle to match in high-performance applications.

So, when should you use a ceramic substrate? Use it when heat is a problem, when reliability is critical, and when failure is not an option. Because in today's world, it's not just about making electronics work. It's about making them last.



Brian Buyea is president of Remtec Inc. To read past columns, click here.





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

Every day we get questions like those. And we love it. We have more than 600 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/faq-about-printed-circuit-boards/#ask

Reliable answers. Reliable PCBs.

NCAB GROUP



Design With the Quote in Mind

BY KELLY ATAY, ASC SUNSTONE

The quoting process has always been an important part of high-quality PCB prototyping and manufacturing. With supply chain disruptions and intensifying demand for domestically produced boards, that is even more true today. As we've previously discussed in recent articles, as well as in our book *The Printed Circuit Designer's Guide to Designing for Reality*, board designs are becoming more complex. So is manufacturing them.

Everything from inventory management to shop floor staffing has gotten harder in recent years, all while the pace of business accelerates. Project development teams need to move fast, and even though designs are more complex, they need their PCB quote requests turned around and their boards manufactured tout de suite.

Designing with the quote in mind means preparing for the quote process while you are still working on the design. Be thinking about the components you will include in your request and whether the complete quote request package contains everything your manufacturer needs to respond within the allotted time.

Complex PCB designs, such as those for UHDI boards, usually require careful analysis by the manufacturer to determine whether specialized manufacturing processes or materials are needed. Juxtaposing elements like a higher layer count and tighter tolerances against an aggressive production deadline takes time to work through.

Give Your Manufacturer Everything They Need

With complex designs in particular, the quote process should be collaborative and focus on value drivers, not on filling information gaps. It is common for key elements to be missing from quote requests, most notably in the BOM. The more complete the quote request, the more collaboration between designer and fabricator focuses on goals such as accelerating production timelines, substituting hard-to-find materials, improving yield, or reducing costs.

Quote requests should include:

- **Gerber files:** Are they complete? Do they include all necessary specifications for copper layers, solder mask, nomenclature, and drilling?
- **Fab drawings:** Do they include all required dimensions and tolerances?

- **BOM:** It is of the utmost importance and should include all material descriptions, reference designators, and required quantities.
- **Fabrication specifications:** Be extra thorough in defining your specifications, including layer count, board thickness, material, copper weight, surface finish, and color.
- **Quote timing and quantity:** How many boards do you need, and how fast do you need them? The answer can have a big impact on which manufacturing partner to choose and how much the boards will cost.
- **Assembly requirements:** Make sure you include a complete list so your boards are prepared for the next production step.
- **Design name:** Give your design a unique identifier instead of default names like PCB1.

A complete quote request is the price of admission these days, but it isn't a guarantee of success. Even if the BOM and fab drawings are complete, it doesn't mean the manufacturer can produce your board. In many instances, the designer and fabricator will have worked together before, making the designer familiar with the fabricator's manufacturing capabilities and material availability. But if you are soliciting a quote from an unfamiliar PCB manufacturer, it makes sense to research ahead of time to ensure their capabilities match your needs.

Know the Quote Process

If you're an engineer at a large organization that is always developing new electronic products, the PCB quote process is no mystery. But our industry thrives because of innovators who are often less experienced with PCB design and quote solicitation. Designers who don't have a lot of experience working with fabricators benefit from understanding how a quote request is processed.

In general, these are the steps fabricators take when responding to a quote request:

- **Review the quote.** Do the Gerber files, BOM, and fab drawings contain everything needed to respond? Can a response be generated as fast as the requestor asks? Are there materials on hand and manufacturing resources available to produce the boards in the requested time window? Do the specs, such as layers, layer thickness, materials, and surface finish, match design requirements?
- **Address design ambiguities.** Generate questions for the requestor.
- **Suggest design changes to improve DFM** if an opportunity exists to increase yield or reduce costs.
- **Look ahead to assembly.** This is done by confirming that the components called out are readily available for assembly now and in the future. If they are not, suggest alternatives.
- **Calculate total cost of manufacturing.** Include shipping.
- **Present the quote to the requester.** Negotiate terms as needed to finalize it and generate a production order.

Not every quote turns into an order, so unless you are working with a trusted manufacturer, don't put all your eggs in one basket. Quick-turn or online quoted orders for simple designs are straightforward for fabricators, so in most cases, soliciting two or three will ensure quick receipt of a response that will meet requirements. Custom, more complex designs will need more back-and-forth with the manufacturer. It makes sense for designers to consult with several manufacturers in order to find one that fits their needs.

Requesting quotes from multiple fabricators will not only give you options but also provide peace of mind regarding completion timing and board costs. Having relationships with multiple fabricators makes sense because, when you are designing with the quote in mind, you focus on successful outcomes rather than tailoring your needs to a single manufacturer's capabilities.



Kelly Atay is the sales and marketing manager at ASC Sunstone. To read past columns from ASC Sunstone, click here.

BY TULIP GU, ECIO & MARCY LARONT, I-CONNECT007

FSQuality in Thailand:

From Automotive and Industrial Controls to AI and Optical Modules



Left to right: Mu Chao, FSQuality sales director; Edy Yu, ECIO; Tianming Liu, CEO, FSQuality; Marcy LaRont, I-Connect007, and Tulip Gu, ECIO, at the Amata City Rayong Industrial Estate in Rayong Province, Thailand. The factory is approaching two years of operations.

One year after their first visit, the ECIO team returned to FSQuality Co., Ltd.'s (a subsidiary of Sihui Fuji Electronics Technology Co., Ltd.) Thailand factory to see what had changed over an eventful 12 months. This time, the team was joined by Marcy LaRont, executive director of I-Connect007, following the Thailand Electronics Circuit Asia event in Bangkok.

Having met with FSQuality Chairman Liu Tianming at the show the week before, we were eager to see firsthand how the company has evolved its customer strategy, expanded manufacturing to support AI and optical module applications, and added another new building to further increase its high-end manufacturing capabilities and strengthen Thailand's growing PCB value chain.

When we visited a year ago, the facility had just become operational. Notably, this visit marked Mr. Liu's declaration that, after nearly two years of investment, construction, and process optimization, the factory has become profitable. The current expansion will add the inner-layer processing and fine-line capabilities needed to produce high-end multilayer PCBs and optical-module products.

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

'Who Are Our Customers?'

Yet, the most significant change at FSQuality is not the addition of another factory building or a few more pieces of equipment. Rather, it is the company's answer to a fundamental question: "Who, exactly, are our customers?" That answer is now reshaping the Thailand factory's product strategy and equipment planning, while also prompting the company to rethink the role and value of overseas manufacturing.

In 2023, FSQuality moved quickly to launch its Thailand project after securing a site. Liu recalled that, with time of the essence, the company opted for a single-story factory design and completed construction in about six months, followed by another three to four months for interior fit-out and equipment installation.

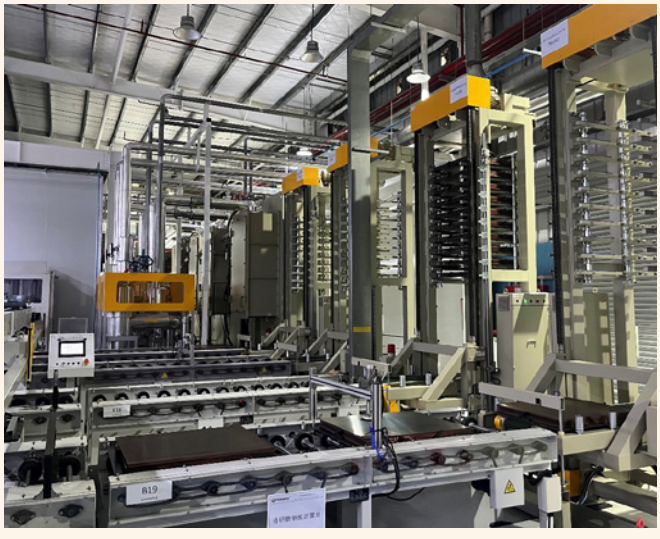
> **"This is more than a simple geographic shift in where orders are produced; it represents a fundamental redefinition of the factory's role and value proposition."**

At the outset of the project, the company envisioned leveraging Rayong Province's automotive industry cluster and the broader Southeast Asian market, starting with automotive electronics and industrial control products before gradually expanding its reach into global markets. However, actual operations have shown that international trade policies and customers' supply chain strategies have evolved, and European and Japanese customers have not shifted their orders to Thailand on the scale initially anticipated.

Meanwhile, PCB manufacturing in China offers clear advantages in terms of cost, lead times, and the completeness of its supporting supply chain ecosystem. For customers without specific regional manufacturing requirements, continuing to source from China is often the more economical option.

"When we first arrived, we were full of ambition," Liu said. "The future was uncertain, but it was also full of promise." Since then, FSQuality has gone through a journey from optimism to uncertainty, and ultimately to a strategic reset. "It was like riding a roller coaster."

Liu acknowledged that some industry peers had a smoother path with their Thailand projects, particularly those that had secured communications-sector customers in advance or whose customers had explicitly designated overseas manufacturing





A look inside the FSQuality Thailand plant.

capacity. FSQuality, by contrast, gradually recalibrated its customer and product positioning through actual operations, directing the Thailand plant's incremental capacity more clearly toward demand from the U.S. market in AI, communications, and optical modules.

This is more than a simple geographic shift in where orders are produced; it represents a fundamental redefinition of the factory's role and value proposition. A "second growth curve" in overseas manufacturing does not emerge automatically once a factory is completed. It must be continuously tested and recalibrated against real customers, real costs, and real orders.

The Answers Lie in High Value-Added Products

The signals prompting FSQuality to adjust its direction came from costs and financial performance.

Manufacturing costs in Thailand are generally 15–20% higher than in China, although the gap varies by product type. A significant proportion of key materials still comes from China.

Although a growing number of suppliers are establishing a presence in Thailand, their arrival does not create a mature local supply chain overnight. Some currently function primarily as distribution hubs and local service providers. Scale, yield, raw material sourcing, and service capabilities all take time to develop. When products lack sufficient added value, it is difficult for a Thailand-based factory to compete directly with factories in China on price, no matter how aggressively it works to reduce internal costs.

Thailand's higher overall cost base, therefore, acts as a natural filter. Products that can remain viable for long-term manufacturing in the country must either be subject to clear overseas supply chain requirements or offer sufficient technical complexity, quality requirements, and delivery value to justify the additional costs.

'All in AI, All in America'

Liu summed up the company's new direction in one phrase: "All in AI, all in America."

This does not mean that FSQuality is moving away from its automotive electronics and industrial control businesses. Rather, its Thailand factory needs is building a product portfolio that is distinct from the more conventional manufacturing capacity of its China operations. In Mr. Liu's view, the factory will increasingly focus on products that "offer healthy margins, involve greater technical sophistication and manufacturing complexity, and require stringent quality standards," rather than competing for orders primarily on price.

Seen from this perspective, the shift toward AI and optical modules is not simply about chasing fast-growing sectors. It is the result of a combination of factors: the plant's cost structure, the geographic distribution of its customers, and the value proposition of its products.





The mSAP Roadmap Takes Shape

FSQuality's Thailand plant has already established a solid manufacturing foundation. The Phase 1 facility has a single-story, linear layout, where production flow is designed to minimize material movement and turnaround time from panel cutting through final shipment, while robotic arms, barcode scanning systems, and MES integration help reduce errors associated with manual setup. Critical production areas are maintained to high cleanroom standards, providing the infrastructure needed for high-precision manufacturing.

In what Liu describes as Phase 1.5 of their expansion, the focus is not simply on adding floor space and overall capacity, but more importantly on strengthening the plant's inner-layer processing capabilities for high multilayer PCBs and developing finer-line processes tailored to optical module applications, such as supporting 1.6T optical modules and future generations of even higher-speed products.

Toward this end, they will soon introduce an mSAP production line to enable finer-line circuitry HDI boards, optical module PCBs, and high-speed communications PCBs.

Under its phased roadmap, FSQuality plans to progressively advance from its current, relatively mature line width/spacing capabilities toward 35/35 µm, before using mSAP to target 25/25 µm and laying the groundwork for even finer-line capabilities in the next stage. In terms of its product roadmap, Liu describes this strategy as staying "half a step ahead," initially focusing on 1.6T products, where clear market demand has already emerged, while also closely tracking the subsequent evolution toward 3.2T.

Though well informed and prepared by the success of their Chinese plants, the new Thailand factory is being built to serve high-end global customers, which means its equipment and process capabilities must be aligned with customer requirements from the start. For this reason, FSQuality has favored newer-generation equipment since the early stages of the plant's development, rather than relocating older or phased-out capacity from China to Thailand. But equipment is only the starting point.

Consultant Qin, a Taiwan-based expert who participated in the onsite discussions, noted that the first priority for an mSAP project is not choosing an equipment brand, but clearly defining the product: First, determine what types of PCBs the company intends to manufacture and what specifications it aims to achieve in the initial stage, then it define the required equipment capabilities. In turn, work backward to determine the configuration of the entire production line and the plant layout. "Product type, target specifications for each stage, equipment capabilities, and plant layout—the sequence matters and must not be reversed."

This sequence may seem straightforward, but it



Aerial view of the plant and the new building under construction.

isola

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.

isola-group.com

Under its phased roadmap, FSQuality plans to progressively advance from its current, relatively mature line width/spacing capabilities toward 35/35 µm, before using mSAP to target 25/25 µm and laying the groundwork for even finer-line capabilities in the next stage

ultimately determines whether a new production line becomes an integrated manufacturing system aligned with customer requirements, or merely a collection of expensive equipment that does not work together to its full potential.

The First Qualified Board Is Only the Beginning of Mass Production

For a fine-line process such as mSAP, producing the first qualified sample is a long way from achieving stable, high-volume, and profitable production. FSQuality is well aware of this and ready for the challenges, some of which Liu outlined specifically.

First, as circuit features become increasingly fine, physical contact between the panel surface and rollers during conventional horizontal developing may have a greater impact on delicate circuit patterns. This makes it necessary to reduce such risks through improved conveying systems and non-contact handling designs.

Second, the requirements for process uniformity in exposure, developing, etching, and plating become significantly more stringent. Equipment precision, chemical process control, and shop-floor process stability must therefore be improved in parallel.

An even greater challenge comes from sequential build-up processing. Unlike conventional multilayer PCBs, where multiple inner layers are laminated together in a single stack, some HDI and mSAP products require repeated cycles of layer build-up, lamination, laser drilling, and via filling and plating. Each additional process cycle introduces new risks involving flatness, interlayer adhesion, via copper integrity, and circuit integrity. As these risks accumulate across multiple layers, final production yield cannot be determined by any single piece of equipment alone.

This also explains why, even when funding, equipment, and materials are all in place, mass-production performance can still vary significantly from one manufacturer to another.

FSQuality hopes to engage with customers at an earlier stage by taking on prototype orders for its R&D projects, allowing six months or more to refine the process and build technical know-how, rather than rushing to secure large-volume orders as soon as the production line becomes operational. With a new process, if delivery pressure begins to override R&D objectives too early, the team may become preoccupied with getting prototypes out the door instead of systematically identifying and resolving every hidden problem in the process.

More dangerous than a temporary lack of POs for an overseas factory is having orders, certifications, and production capacity already in place while its yield performance and team capabilities have yet to mature at the same pace.

Building Cohesion in a Multinational Team: Talent Matters

When discussing the key to making the new production line successful, Liu's first answer was not equipment or materials, but talent.

He noted that the company had previously sent more than 20 Thai employees to its factory in China for hands-on training. However, most of those employees eventually left the company. The experience led FSQuality to realize that a one-off cross-border training program would not automatically translate into sustainable local capabilities that could remain embedded in the overseas factory.

FSQuality, therefore, began to place greater

emphasis on sending Chinese engineers directly to the Thai plant. Their first task was to get the products successfully into production and then train Thai and Myanmar employees through a "show them how, then let them do it themselves" approach. He believes that local employees have certain strengths when it comes to following clearly defined standards.



Chairman Liu speaks with tech journalist Tulip Gu.

But success depends on the company first establishing those standards and ensuring that knowledge and experience are continuously accumulated and embedded in day-to-day operations, illustrating a fundamental challenge of overseas expansion: Equipment can be installed and brought online relatively quickly, but organizational capabilities cannot be transferred at the same pace. True localization is not simply a matter of achieving a certain percentage of local employees. Rather, it is achieved when the factory can maintain consistent quality, deliver on schedule, and drive continuous improvement on its own.

In the near term, FSQuality will continue to rely primarily on its China-based team for strategic decision-making, R&D introduction, and the resolution of complex technical issues, as well as its mature supply chain.

'Coming to Thailand Means Playing in the World Cup'

FSQuality has already invested approximately RMB 700 million in its Thailand factory and plans to invest another nearly RMB 300 million to introduce mSAP capabilities and AI-related products, in addition to further expanding into the AI and optical module markets.

Beyond the scale of investment, what deserves greater attention is the changing role of the Thailand plant. In the past, Southeast Asia was often seen as a destination for relocating relatively mature, labor-intensive manufacturing capacity from China. But amid the latest wave of PCB capacity expansion, some new factories in Thailand are being equipped from the outset with more advanced machinery to serve international customers with specific requirements for regional supply chains. Rather than simply replicating China's existing manufacturing capacity, these factories are rebuilding high-end manufacturing capabilities around a new customer base.

Drawing on his many visits to local factories, FSQuality consultant, Mr. Qin offered a vivid analogy: "Coming to Thailand means playing in the World Cup, not competing in a regional tournament."

For FSQuality, the Thailand factory's recent move into profitability represents an interim payoff from the adjustments made over the past two years, but it is by no means the end of the journey. The mSAP production line has yet to be built and ramped up; 1.6T and subsequent products must still go through sampling, qualification, and mass production; and local R&D and engineering capabilities will take time to develop.

From establishing a regional foothold to targeting U.S. demand for AI-related products, from conventional multilayer PCBs to advanced mSAP processing, and from reliance on China-based teams to building local capabilities, what makes FSQuality's Thailand project worth watching, therefore, is not whether it has already produced a successful formula that others can replicate, but how the company is gradually transforming a completed overseas factory into a manufacturing operation capable of meeting the demands of high-end global customers. **I-CONNECT007**

BY TULIP GU, ECIO & MARCY LARONT, I-CONNECT007

Sundell Technology

Laying the Groundwork for a 'Certain' Future



Sundell Technology is in Rojana Industrial Park, Ayutthaya Province, about an hour's drive north of Bangkok. The entrance to the industrial park rises over a gentle slope. This design was introduced in the aftermath of Thailand's devastating 2011 floods, with an approximately three-meter-high floodwall standing beyond the slope.

In early September, Edy Yu and Tulip Gu of ECIO, and I-Connect007's Marcy LaRont visited Sundell Technology's Thailand facility at the Rojana Industrial Park in Ayutthaya Province. Two years into operation, the factory has devoted considerable investment and effort to things that are largely "invisible."

Registered in February 2023 and commencing production in 2024, the company has invested nearly RMB 1.2 billion in the facility, which covers approximately 267,000 square meters and was initially planned for a monthly production capacity of 120,000 square meters. Among Chinese-funded PCB manufacturers that have established factories in Thailand, Sundell Technology ranks near the top on almost every measure.

Sundell Technology's growing list of investments includes most key materials, chemicals, and auxiliary materials that are now sourced locally, supported by a three-tier supply system comprising direct local supply, vendor-managed inventory (VMI) consignment, and strategic inventory reserves; automation exceeds 95% in core processes; and every PCB can be traced throughout the entire production

INSPECTA & D6 MULTI X

INTELLIGENT CLOSED-LOOP REGISTRATION SOLUTION

1. X-RAY DATA

X-ray imaging identifies internal reference targets and captures real-time registration data.



2. AI CORRECTION

AI analyzes the data, compensates for distortion, and optimizes the true position of every hole.

AI
CORRECTION ENGINE

- DISTORTION COMPENSATION
- TARGET OPTIMIZATION
- INTELLIGENT LEARNING

3. PRECISION DRILLING

Optimized data is executed by D6 Multi X for maximum drilling accuracy and consistency.



HIGHER ACCURACY

Consistent inner-to-outer layer registration at the highest level.

LESS SCRAP

Automatic error correction reduces defects and rework.

MORE YIELD

Higher first-pass success improves throughput and profit.

AI-DRIVEN ACCURACY. LESS SCRAP. MORE YIELD.

INTELLIGENT TECHNOLOGY. SUPERIOR RESULTS.

pluritec® GROUP

pluritec.it



Sundell's solder mask and legend line.

process at the individual-piece level. None of these initiatives directly generates dollars. What they produce instead is something else: certainty, which is a highly valuable but invisible product that PCB manufacturers must create after when expanding into Thailand.

Sundell Technology is still ramping up capacity quarter by quarter. While outsiders are focused on when the factory will reach full capacity, the sequence of Sundell Technology's moves is worth noting: first, create certainty, then wait for orders to materialize. During our visit, Sundell Technology President He Zixiu, General Manager Song Bo, and Quality Director Peng Min were interviewed about how a high-end PCB factory creates "certainty, "and when it can convert that certainty to orders.

Creating Certainty Through the Details

Why did Sundell choose Ayutthaya for building in Thailand, rather than Prachin Buri, where Chinese-funded companies are more heavily concentrated? Different than the answer expected, their response was simple and focused: people and water.

"Ayutthaya is close to Bangkok, has a large resident population, and sees substantial population flows; this was our first consideration," said Song Bo, general manager of Sundell Technology, adding that the second consideration was less obvious: Thailand is a major agricultural country, and during the dry season, priority must be given to domestic and agricultural water needs. Industrial water withdrawal is not permitted upstream but is allowed downstream. "Prachin Buri is upstream, while we are downstream, close to the river mouth. The stability of water resources was a hard constraint factored into our site-selection decision."

To ensure stable water resources, Sundell Technology designed its wastewater treatment system to meet standards more stringent than current discharge requirements, while also reserving capacity for future expansion. The system is designed to accommodate potential tightening of discharge standards over the next 10 years and beyond, thereby minimizing the impact of future local environmental policies on continuous factory operations and ensuring reliable delivery.

The devastating floods of 2011, which inundated much of Thailand's industrial belt, remain a shared memory for all companies with factories in Thailand. The flood protection measures are twofold. Rojana Industrial Park built a raised entrance and an approximately three-meter-high floodwall around its perimeter. Sundell Technology also raised its factory floor 1.1 meters above the park's road level, constructed the exterior walls entirely of concrete from ground level up to a height of 1.2 meters and below the windowsills, and installed removable flood barriers at the factory entrances.

"If floodwaters ever do reach us, we can simply put the barriers in place, and that gives us two layers of protection," Song Bo said. "But since the factory was completed, we haven't had to use them even once. We hope we never will.

When it came to their people, Sundell Technology displayed the same level of detail thinking and planning, arranging six-month visas for personnel stationed onsite by its suppliers. "We can't have people working here while constantly being checked by immigration authorities," Song Bo said. "They need a sense of stability."



Sundell's well equipped final inspection stations.

The Cost of Time: Certification Cycles for a Broad Product Strategy

Sundell Technology's product portfolio covers nearly all the mainstream markets that a PCB factory can serve: automotive electronics, laptops, AIPCs, servers, communications and switches, and energy applications. Its Phase I capacity of 120,000 square meters per month for general-purpose high-end PCBs requires a sufficiently diversified customer base to fully utilize it.

But a broad product strategy carries a cost: Each new product line requires customer qualification, and each new customer category can bring another round of factory audits. As capacity continues to ramp, customer qualifications, yields, and utilization must keep pace, while growing order volumes gradually absorb the fixed costs of the initial investment. Their real challenge is not simply getting the equipment running, but synchronizing customer qualification, stable delivery, and capacity growth while developing multiple product lines in parallel.

Thailand is not a low-cost manufacturing base. This truth was acknowledged by all the manufacturers we visited. Customers shift orders to Thailand primarily to meet compliance requirements for overseas. Orders are not simply "taken," they are "screened." Some are filtered out by cost, others by customer qualification. Ultimately, what remains are those that must be produced in Thailand. Another price of pursuing a broad product strategy is that every product line must go through this screening process separately.

Investments and Partnerships Matter

In February 2025, Japan's Meiko Electronics invested approximately $20 million in Sundell Technology's parent company, JIARUIAN PTE. LTD., acquiring a 14.9% stake and appointing a director to Sundell's board. For Sundell Technology, the significance of this strategic investment lies not only in the capital itself but also in the customer resources and technical exchanges it brings.

"Meiko does not get deeply involved in our internal management," Song Bo said. "First, it provides capital; second, it brings customer resources; and third, it facilitates mutual technical exchange. We visit its factories to learn, and there are regular exchanges and visits between the two sides."

However, none of these resources—capital, customers, or technical exchange—produces an immediate return. For a factory still expanding capacity each quarter, the greatest benefit of a hands-off industrial shareholder may not be the orders it brings, but the time it allows to complete certifications carefully and improve yields steadily. Capital markets and financial reporting cycles may be less patient. Managing this tension between operating needs and financial expectations is a central challenge for Sundell Technology.

Laying the Groundwork for 2028

In contrast to its measured pace of construction, Sundell Technology has laid out an ambitious and tightly packed roadmap. Using 18-layer PCBs as the benchmark, the company plans to ramp up production in stages, in parallel with more advanced HDI process capabilities. On the technology front, development will proceed step by step along a clear path, from sample delivery and small-batch validation to full-scale mass production. Key iteration milestones have already been mapped out for both

high layer count and HDI product lines, closely synchronized with the pace of capacity expansion.

Behind this roadmap is a set of hard technical metrics, which Song Bo rattled off in quick succession:

- Minimum line width/spacing of 35/35 µm, surpassing the 40/40 µm mass-production limit of conventional dry-film tenting processes
- Minimum laser blind-via diameter of 75 µm
- Compatibility across the full product range with M6–M9 ultra-low-loss laminates
- Stable back-drill stub control at 2–6 mil; and impedance tolerance of ±7%

To meet leading customers' requirements for 40–78 layer N+M and N+N+N hybrid-stack designs, the factory is adding integrated hot-and-cold lamination presses and high-precision CCD vision-guided back-drilling machines in batches. Its HDI lines are also being fully equipped with Mitsubishi laser drilling systems and horizontal electroplating equipment for via filling.

On the measurement and analysis side of the equation, Peng Min described Sundell's central analytical laboratory. It is equipped with a VNA, ion-milling SEM, EDS, and 3D X-ray systems, enabling electroplated crystal structures to be examined at 5,000–10,000X magnification for nanoscale defect localization. The laboratory also offers a full suite of reliability testing capabilities, including TCT, CAF, IST, and HATS. They plan to complete ISO/IEC 17025 accreditation this year.

The manufacturing system follows the same philosophy of "laying the groundwork first." Automation rates exceed 95% across core processes, enabling full-process traceability down to the individual PCB level. APS (advanced planning and scheduling), WMS (warehouse management system), QMS (quality management system), EAM (enterprise asset management), and BPM systems form a closed-loop, data-driven architecture, while AGVs (autonomous guided vehicles) handle material transport throughout the workshop.

In the fourth quarter of 2023, the industrial park received a Gold Award in the U.S. MUSE Design Awards for factory design; in January 2025, it earned an EcoVadis Silver Medal, placing it among the top 15% of companies assessed globally.

What these figures have in common is staying



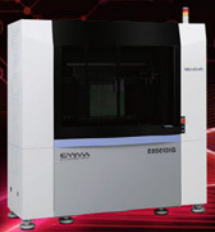
Brown oxide surface treatment line.

MicroCraft's Product Range
Built for What's Next

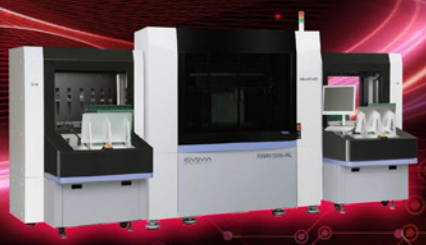
EMMA
Moving Probe Tester

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,
Sarra adds advanced impedance
and signal integrity testing
to your toolkit



» H2P45
2-Probe Variable Pitch TDR Tester



» S4V54
4-Probe VNA Tester

MicroCraft

www.usamicrocraft.com sales@usamicrocraft.com

ahead of the curve: The capacity roadmap extends through 2027, the technology roadmap through 2028, and key certifications are scheduled for the fourth quarter. Step by step, Sundell Technology is turning each element of its plans into certainty for their customers.

The Supply Chain

According to Song Bo, since trial production began in Q3 2024, Sundell Technology has spent nearly two years mapping out Thailand's local supply ecosystem. Today, most of its key materials, including copper-clad laminates and prepregs, along with process chemicals and auxiliary materials, can already be sourced locally.

Shengyi Technology Co., Ltd (SYTECH), Taiwan Union Technology Corporation (TUC), and Kingboard Laminates Holdings Ltd. have all established mass-production operations in Thailand, while Nanya New Material Technology Co., Ltd. and Zhejiang Huazheng New Materials Co., Ltd. plan to bring production online in Thailand in 2027.

Building on this foundation, Sundell Technology's three-tier supply system means suppliers without local warehousing rely on cross-border logistics, with lead times of 12–15 business days by sea, 5–7 business days by land, and 3–5 business days by air. Safety stocks are dynamically replenished, with shelf-life controls in place to manage materials approaching expiration.

People and Talent

On the talent side, Song Bo said, "An independently operated overseas PCB factory cannot simply replicate the workforce model used elsewhere. It has to adapt to local regulations, education systems, and culture."

Frontline staffing follows a dual-track, compliance-based model: the factory primarily recruits graduates from Thai vocational schools and experienced local workers, while supplementing its workforce with Myanmar labor recruited in strict compliance with Thai labor regulations, ensuring uninterrupted 24-hour production.

For engineering and technical talent, Sundell Technology has signed long-term industry-university partnership agreements with two Thai universities, creating targeted recruitment pipelines for science and engineering graduates into core functions, including engineering, quality, the central laboratory, and equipment operations.

Training is structured across three levels. Fresh graduates enter an 18-month "100-Person Growth Program," receiving one-on-one mentoring from senior staff and rotating through different roles. Frontline technicians undergo hands-on training backed by standardized skills certification and clearly defined promotion pathways. At the management level, bilingual training has become routine, with managers learning Thai and expatriate engineers receiving English-language training.

"Adaptation" is the operative word at Sundell Technology. If a single supply channel is not enough, build a three-tier system; if one recruitment channel is insufficient, open a second. The objective is to make the factory self-sufficient in local talent as early as possible, rather than continuing to rely on personnel transferred from China.

Focus: Betting on Computing Power

Once that certainty has been built, the next question is where to put it to work. He Zixiu's answer is computing power. Phase II will add dedicated capacity for computing-related products, with standalone production lines prioritizing orders for computing hardware. Over the medium to long term, the goal is "to become Southeast Asia's technologically leading supplier of PCBs for computing applications."

The industrial logic behind this choice is straightforward. AI computing is one of the few downstream markets where qualification and delivery reliability can outweigh cost considerations—precisely the kind of market that can offset Thailand's cost disadvantage. While automotive PCB orders may remain in China due to cost considerations, computing-related orders are increasingly shifting to Thailand due to supplier qualification requirements.

Of course, no matter how much certainty an individual manufacturer builds into their operation, market uncertainty and geopolitics are not their control. The best and only thing to do is to excel at what you can control, and Sundell does that. They are ready for a bright future. **I-CONNECT007**

IPC®
by Global Electronics Association



THE STANDARDS YOU NEED.
BUILT AROUND THE WAY YOU DESIGN.

IPC Design Suites bring together complementary standards by design function, helping you quickly find the guidance relevant to your work and apply it throughout the design process.

DESIGN CORE SUITE

Build a strong foundation with essential guidance for rigid, flex, rigid-flex and high-frequency PCB design, from board structure and layout to performance requirements.

DFM ENABLERS SUITE

Design with manufacturing in mind using guidance for DFX, land patterns, impedance and current-carrying capacity to support a smoother path from design to production.

MATERIALS SUITE

Make more informed material and finish selections with standards addressing performance, reliability, thermal considerations and manufacturability.

PACKAGING APPLICATIONS & EMBEDDED GUIDANCE SUITE

Navigate the design and manufacturing considerations of advanced packaging and component technologies with application-specific guidance.

ONE INDUSTRY.
ONE COMMON LANGUAGE FOR ELECTRONICS DESIGN.

Whether you're establishing design requirements, preparing for manufacturing, selecting materials or working with advanced packaging, IPC Design Suites put relevant standards together in one place.

Find the suite that fits your work.

BY TULIP GU, ECIO & MARCY LARONT, I-CONNECT007

Delton

Turns Its Thailand Location Into a Rules-driven Smart Factory



In Prachin Buri, Delton's new Thailand factory is progressing rapidly through a mass-production ramp-up that would typically take considerably longer. Featured here is Zen Zhijun (second from left), assistant to the CEO.

On Sept. 1, 2026, ECIO's Edy Yu and Tulip Gu, along with I-Connect007's Marcy LaRont, toured Delton's Thailand facility in Prachin Buri. The factory officially began production in June 2025 and, impressively, was profitable by December. Phase I capacity utilization has already exceeded 90%, and further expansion is moving quickly—a remarkably fast ramp-up for an overseas PCB operation.

Amid the wave of PCB investment in Thailand, not every company is starting from the same place. Some companies build their factories and install equipment first, then work to demonstrate to customers that they can deliver. Others, by contrast, are driven overseas by customer demand. This was the case for Delton.

According to Zeng Zhijun, assistant to the chair of Delton Technology in Guangzhou and general manager of Delton Technology in Thailand, strong incoming orders, particularly from U.S. customers, have accelerated the factory's original three-phase expansion plan, with Phases II and III now progressing more concurrently. However, it takes more than a steady order backlog to guarantee success in such a tremendous endeavor.

EMC

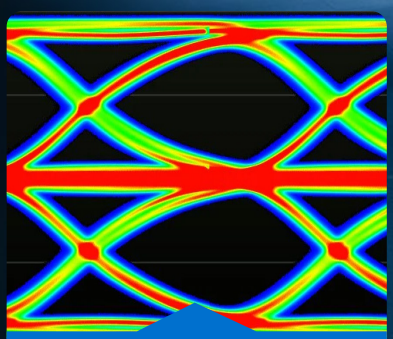
STRIVE FOR **EXCELLENCE**

www.emctw.com | EMC.INFS@mail.emctw.com

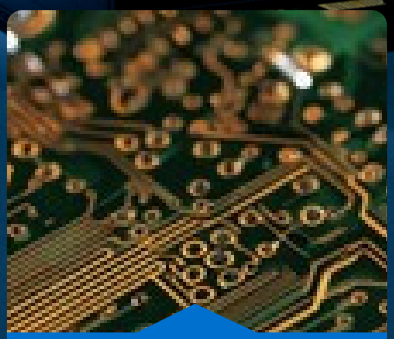
Introducing EM-896K2 and EM-896K3

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

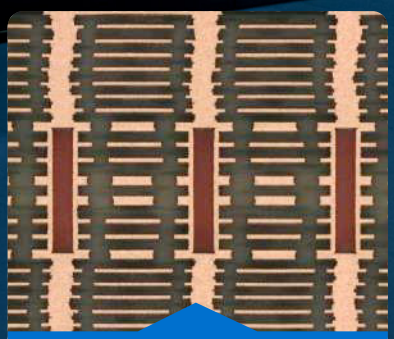
- Excellent Dk/Df performance for high-speed signal integrity (< 1.0 dB/inch @ 53GHz)
- Low X/Y-axis CTE for improved dimensional stability includes CoWoP applications
- Proven multi-lamination reliability (≥12x laminations)



Excellent Dk/Df performance



Ultra-low X/Y-axis CTE



Proven multi-lamination reliability

INSULECTRO

North American Master Distributor
(949) 587.3326 • insulectro.com

But from our perspective, the more interesting part of Delton's story addresses a fundamental challenge facing Chinese PCB manufacturers expanding overseas: How do you transfer advanced manufacturing capabilities outside China's mature supply chain and engineering ecosystem?

Delton's executive management team quickly learned that customers did not simply want a conventional PCB factory located in Thailand, but an overseas manufacturing base capable of producing advanced products for servers, switches, accelerator cards, backplanes, and, eventually, advanced PCB products for GPU applications. This meant that Delton could not follow the traditional approach of "moving simpler products overseas first and upgrading gradually over time," Zeng Zhihun said.

Delton's answer was not simply to send more engineers and skilled workers overseas or replicate its domestic operations. Instead, the company broke down its manufacturing know-how into explicit rules, embedded them in digital systems, and used them to automate factory management.

Evaluating Costs

Delton management said that the overall manufacturing costs of PCB factories in Thailand are approximately 20%–25% higher than those in China, a statistic repeated by multiple sources during our visits. The cost premium comes from several factors, including Chinese expatriate personnel, material transportation, procurement, utilities, and the relative immaturity of the local supply ecosystem, which requires some materials and chemicals to be sourced through Delton's China-based supply chain.

Because of this, it was unrealistic to expect profitability from competing in low-end PCB markets, so building operations in Thailand had to support advanced products. Therefore, Delton Thailand was quickly positioned as a high-end manufacturing operation. Additionally, in Delton's view, only by focusing on high-end products and building an advanced overseas manufacturing base can the company establish a firm foothold in today's rapidly changing and highly competitive market.

Delton's product portfolio is centered on server motherboards used in general-purpose servers, AI racks, and switch trays. The Thailand factory can support PCIe 6.0 platforms, while also possessing the process capabilities required for PCIe 7.0 and 800G switch products. It is now further expanding its technical capabilities in advanced HDI, and next-generation switches and AI servers.

The Heart of Smart Manufacturing

As you walk into Delton's factory, the high level of automation is immediately apparent. Interconnected production lines, robotic arms, automated guided vehicles (AGVs), automated loading and unloading systems, and automated logistics systems have significantly reduced the need for manual handling and intervention between manufacturing processes.

Transferring its mature manufacturing capabilities to Thailand was not a copy/paste from Guangzhou. Though equipment and materials do not fundamentally change simply because they are used in a different location, the Thai factory has its own characteristics in terms of equipment configuration, automation and digital architecture, logistics, and process flows, requiring existing technologies and processes to be adapted and recalibrated accordingly.

What truly sets the bar for this factory, however, is not its impressive hardware. Delton has built an integrated system architecture connecting production planning, manufacturing, quality, equipment, and logistics.

APS (Advanced Planning and Scheduling) handles production scheduling, MES (Manufacturing Execution System) manages manufacturing execution, QMS (Quality Management System) governs quality rules, EAM (Enterprise Asset Management) manages equipment assets, WMS (Warehouse Management System) and MCS (Material Control System) coordinate warehousing and material flows, while EAP (Equipment Automation Program) connects production equipment and collects process parameters.

General-purpose modules can be supported by third-party software, while the core systems designed specifically for PCB process control are developed and refined in-house.

From an operational perspective, the factory has two logical layers: a rule layer that defines how production should run, and an execution layer that applies those rules to equipment and processes on the shop floor. In a conventional factory, the Control Plan (CP) and Inspection Plan (IP) often exist primarily as documents,

with execution dependent on operator experience and onsite management. Delton instead digitizes those requirements, turning CP and IP into rules that are systematically enforced throughout production.

The value of this system lies not in the volume of data it generates, but in its quality. Poor data simply creates more noise, while reliable data depends on consistent connectivity across the factory. In this sense, a smart factory is more than a collection of automated equipment; it is a manufacturing system in which machines operate according to embedded process knowledge and rules.

That distinction becomes particularly important for server and AI PCBs. Manufacturing complex products with high layer counts, large board sizes, high aspect ratios, back drilling, and advanced HDI cannot be reduced to a fixed set of equipment parameters. Success depends on accumulated engineering knowledge of the relationships among materials, equipment, processes, and production anomalies, which is predicated upon good data.

Talent Is Still Key

Compared with equipment parameters, the skills and experience of engineers and frontline manufacturing personnel are the most difficult to replicate. The ability of onsite personnel to understand critical control points, identify abnormalities, and make the right judgments can only be developed gradually through continuous training and hands-on production experience.

To address this, Delton's Guangzhou R&D center brings together resources from across the group to develop key technologies and tackle specific technical challenges. R&D and process engineers in Guangzhou and Thailand then communicate directly on a one-to-one basis, transferring process routes, critical control points, and validation experience to the factory in Thailand while keeping the chain of information exchange and technology transfer as short as possible.

In addition, the engineering team in Thailand develops its own specific process flows and parameter windows based on customer product requirements and local equipment and process configurations. At the same time, the Guangzhou and Thai facilities share the same process database, quality-control standards, and core processes, ensuring that the same product meets identical requirements for impedance, back-drill stub length, via copper thickness, reliability, and batch-to-batch consistency. Zeng Zhijun said, "The two sites may differ in their equipment and process configurations, but there is no compromise on quality standards."

Meanwhile, the factory in Thailand is gradually building up its own independent capabilities in R&D, testing, new product introduction, and basic failure analysis, enabling it to respond quickly to overseas customers. Projects that require substantial resources or involve major new technologies or complex failure mechanisms, however, will continue to be developed centrally in Guangzhou, with the resulting mature technologies and solutions subsequently transferred to Thailand.

When manufacturing knowledge can flow continuously between Guangzhou and Thailand and be executed consistently through digital systems, individual expertise is transformed into an organizational capability that can be replicated, validated, and continuously improved.

People make mistakes, so reducing reliance on human intervention has been a central objective in building the factory's management system. But that does not mean the factory no longer needs people. Rather, it means that the role they play is changing. Operators are no longer the primary agents whose manual actions directly determine product quality; instead, their roles are increasingly focused on equipment maintenance, recovery from production anomalies, and executing system instructions. Engineers, meanwhile, are moving away from creating and compiling data toward analyzing it and solving problems more quickly.

This model is particularly important in Thailand. There is no shortage of local talent with experience in conventional double-sided PCBs, four-layer boards, or automotive PCBs, but technical and managerial professionals with expertise in high-end server PCBs remain scarce.

Delton is working with universities to develop targeted training programs that build a local talent pipeline. In the short term, digitalization lowers the experience threshold for operational roles. Though Thai nationals already account for more than 70% of frontline employees—meeting the Board of Investment target number for lower-level skilled

workers—their representation in technical and management positions remains in the single digits. Over the longer term, however, the factory must still develop local engineers who can understand the logic behind manufacturing processes, handle complex production anomalies, and drive continuous improvement. Localization, therefore, remains one of the factory's most difficult challenges.

Delton follows this comprehensive approach:

- Basic operational know-how is digitized to create smart systems and equipment
- Anomaly-handling capabilities remain with the on-site engineering and manufacturing teams
- New process development and critical technical capabilities are accumulated and retained at the group level

Customers Trust Systems, Not People

High-end customers do not lower their audit standards simply because a factory is located in Thailand. On the contrary, some customers are imposing even more stringent requirements, especially regarding digital data and tracking. The days of manually recording data entries into a spreadsheet are over. It is not uncommon for a customer to require that data be generated directly within the system, be shareable, and be available for review at any time. Under this new audit logic, digitalization is no longer merely an internal tool for improving factory efficiency. It has become part of how customers assess their suppliers.

Customers are not dismissing the value of people; rather, they are unwilling to stake the consistency of quality in high-volume, highly complex products on whether individual employees are sufficiently attentive or skilled.

They want a system that is traceable, cannot be bypassed, and can be verified in real time. At Delton, every PCB is assigned a unique ID, enabling traceability of its processes, equipment, parameters, and inspection data both upstream and downstream. Only then is product quality truly verifiable.

Smart Factory Next Steps

Delton acknowledges that its Smart factory journey is a continuing process. Data collection and report generation are well-established, but engineers must still manually access various reports and sift through large volumes of information to identify problems. In the next phase of Delton's Smart factory, AI will do some of that heavy lifting, transforming this process to analyze data, detect anomalies, and proactively alert relevant personnel to problems.

Achieving this will require machine-learning technologies to build predictive models for parameter recommendations and early warnings of potential problems. Moving the project forward and identifying meaningful anomalies effectively is no easy task, and it places even greater demands on the knowledge and judgment of engineering and technical personnel. In the AI era, therefore, people will not be replaced. On the contrary, more will be expected of them.

Delton, meanwhile, has taken a measured approach to digital twins. Zeng Zhijun acknowledged that although the company already has the necessary data foundation and is conducting simulations in selected areas, it has yet to systematically deploy a factory-wide digital twin.

This restraint is worth noting. For a PCB factory, truly valuable digitalization does not necessarily mean a more sophisticated 3D visualization of the shop floor on a central control-room display. What matters more is whether the system can detect anomalies earlier, predict outcomes, and deliver actionable recommendations to the people responsible for making decisions.

Delton Thailand: Exporting Success

On the surface, the factors behind Delton Thailand's success are not mysterious: strong customer demand, high-end product positioning, substantial investment in automation, and technical support from its sister facility in Guangzhou. But only when these elements are connected through an integrated system do they translate into achieving profitability within just six months.

Delton has created, not merely a PCB production line, but a model for manufacturing success. Achieving monthly profitability within six months of commencing production in Thailand is a striking result. But beyond that, it has demonstrated what is possible. **I-CONNECT007**

INSULECTRO

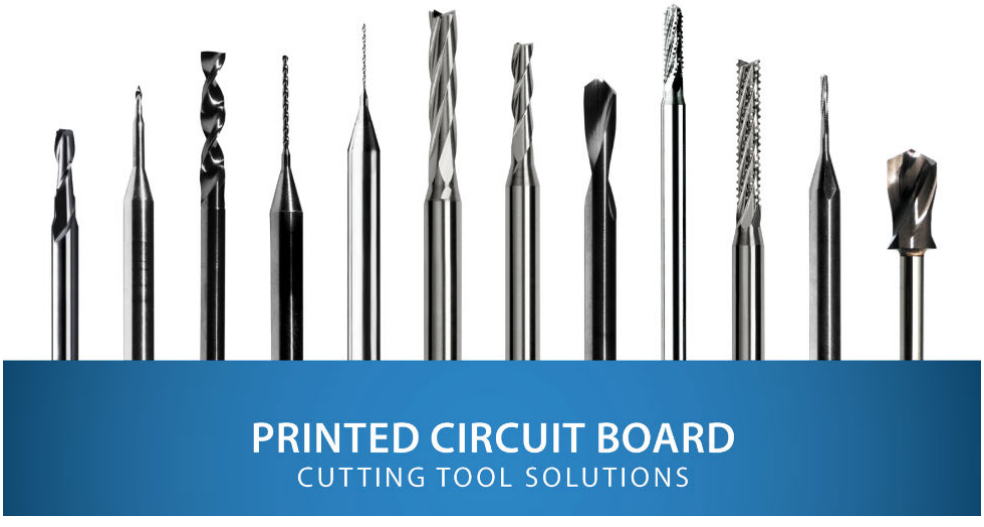


WORLD-CLASS TOOLS

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

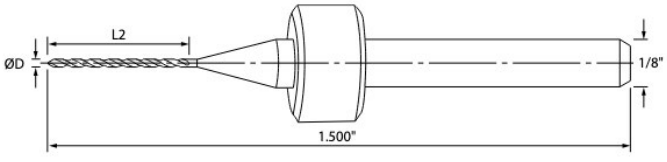
KYOCERA



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.



A Few Things That **Still Scare Me**

BY DON BALL, CHEMCUT

The PCB fabrication industry is constantly changing and throwing new challenges our way. Initially, these changes and evolutions can be frightening, but we've always managed to adapt, incorporate these them into our processes, and continue on. But what do some of us find frightening in the industry environment now and in the near future?

Emerging technologies can be frightening, especially since they represent the unknown. It usually means changing how we do things and finding new (and uncomfortable) ways of operating. That could mean spending more on new equipment and processes, which can also be frightening.

This can be especially so for smaller shops that don't have the extra budget funds for new equipment. Some emerging technologies of the past seemed to threaten the importance of PCBs or even attempt to eliminate our industry. Yet somehow there has always been the need for interconnects, and the industry has survived and even thrived.

Yes, some companies have failed. But others have embraced new technologies and grown. Such is the nature of the business. However, while there are things about new technologies to be afraid of, they also present new opportunities, and so far, this industry has successfully integrated them into its businesses. But to put things into perspective, let's take a quick look at some of the previous "emerging" technologies that raised concerns.

New Technologies Can Be Scary

The 1950s (before my time): What? You want to put thin sheets of copper on fiberglass and form circuits by removing unwanted copper by spraying it with hot acid? Don't you know that's what wires are for?

The 1960s (still before my time): What? You want to put copper sheets on both sides of the panel, then drill holes through the panel and plate copper inside the holes on fiberglass and epoxy to provide an electrical interconnect between the top and bottom of the panel, so you can use both sides for circuits? Are you crazy?

The 1970s (now we're talking): What? You want to stack these double-sided panels on top of each other to form multilayers, then line them up and laminate them together so we can drill and plate hundreds of smaller holes to electrically connect all the layers, using even less space. Are you out of your mind?

The 1980s: What? Now you not only want to use even thinner fiberglass substrates (0.002" thick) for your multilayers so you can stack even more layers in the same thickness, but you also want to use flexible substrates so you can bend them to fit into even tighter spaces? Oh, and you also want to make the circuit lines even smaller, like less than 0.005". C'mon, man, get real.

The 1990s: What? You want to put the entire circuit board on a tiny chip? Are you trying to put us out of business?

It's now 2026, and thus far, the industry has navigated the shoals of new technologies. I don't see this changing anytime soon. While new technologies certainly cause anxiety, they should be seen as opportunities rather than something to be feared.

Fears About AI

As I mentioned in last month's column, I am leery of the impending AI revolution. Not so much about the implementation of AI itself—that will happen whether I like it or not. But I'm leery of the rush to use it before it's proven to reduce the workforce.

We've all read about tests with autonomous AI running various businesses that have failed, usually spectacularly. I'm reminded that the learning algorithms built into them are developed by human coders who have a reputation for not living in the real world like the rest of us and for not being especially socially adept.

My feeling is that no AI should be allowed to run any process or business without human supervision. AI has no human empathy. Those of you who have tried to call a help line with an AI operator know what I mean.

Here's a story that I think illustrates my point. Recently, I had a problem with my card from a national bank, and I had to spend at least 15 minutes finding the right wording to reach a real person on the line who could help with a problem not on the AI menu. I call these AI reps Artificial Idiots. They are especially annoying because you can't yell at them to relieve your frustrations as you can with a natural human idiot. After all that, I ended up canceling my national bank card and going with a local bank. At least I can talk to a real person when I have a problem.

I'm all right with AI, but not AI without human supervision and oversight. I think those who try to use autonomous AI to replace skilled and entitled workers for lower pay and less skilled workers are in for trouble.

Skilled Workforce Fears

Okay, enough about my AI concerns. I am also concerned about having a skilled workforce, not so much for our company but for the industry in general. My company is fortunate to have a low employee turnover rate, so when people retire or otherwise leave, it's not too much effort to promote from within to fill those positions. When we need to expand the workforce, we can hire entry-level candidates with confidence that they will stay long enough to become productive members of the team.

One exception is more staff for our process lab. It is difficult to find candidates willing to work with the chemistries and processes involved in our testing and development work for our equipment (and to work with the two crazy people already there). I'm sure this is not a surprise, as many of you are facing the same problem and, as a result, are increasingly turning to your suppliers for process support. Which is why we are looking for more process people.

What concerns me is that, while we have gained a lot of process experience over the years and are more than willing to share it with our customers and provide guidance and help where we can, we are an equipment manufacturer, not a process engineering service company. Being willing to help but not having the resources to help everyone who needs it is a great concern.

These are just a few of the things I find "scary" this Halloween season. I see that I am running out of column space (again), so I'll leave it here.



Don Ball is a process engineer at Chemcut. To read past columns or contact Ball, click here.



Are PCB Fabricators Ready for the Next Technology Curve?

BY MICHAEL CARANO, GLOBAL ELECTRONICS ASSOCIATION

The electronics industry's demand for smaller, faster, and more powerful devices is driving major advances in PCB manufacturing. One of the central priorities is to increase circuit density while maintaining performance and reliability.

Manufacturers produce finer lines and spaces so more components can fit on smaller, more sophisticated boards. They are also refining SMT features to support tighter pitches without compromising performance.

As components shrink, smaller-diameter microvias and through-holes are becoming essential for connecting multiple PCB layers. Stacked microvias support dense signal routing and chip-to-chip communication, while 6+2+6 build-up constructions are becoming standard.

Meeting Future Market Demands

By advancing fabrication technologies and continuously improving manufacturing processes, PCB manufacturers must position themselves to meet the electronics market's growing requirements for miniaturization, density, and performance.

In the electronics manufacturing industry, the imperative to uphold signal integrity and ensure overall design reliability cannot be overstated.

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

Designers are recognizing these critical challenges by addressing several key aspects to achieve optimal outcomes. Foremost among these considerations is the vital need for efficient heat dissipation, a paramount factor in averting overheating issues that could lead to debilitating component failures.

As a result, the integration of innovative heat dissipation techniques into PCB designs has become a pivotal endeavor, enriching thermal management capabilities and safeguarding the seamless functioning of electronic systems. The design of a PCB entails a rigorous focus on long-term reliability, and necessitates an intricate analysis of diverse operating conditions and stress factors. Engineers ensure the device's longevity and enduring performance, bolstering end-user trust and confidence.

Furthermore, the pursuit of higher data rates and unparalleled performance necessitates the adoption of cutting-edge materials with low loss and ultra-low-loss properties. Such innovative materials decisively minimize signal loss, thereby preserving signal integrity and leading to superior electronic device performance. We are moving from 800G to 1.6T and, as some predict, 3.2T in the next decade.

Material science has yielded remarkable advancements, manifesting in the development of high-performance and ultra low loss materials tailored to meet the exacting requirements of cutting-edge electronic devices. By enhancing the mechanical properties of PCBs, these materials instill exceptional robustness and resilience, thereby mitigating vulnerabilities to external stresses and enhancing the overall reliability of electronic systems.

Manufacturing Process Innovation

To support denser, more complex designs, PCB manufacturers are moving from traditional subtractive processing to more efficient additive techniques. These methods improve precision, reduce material waste, and, when combined with build-up films, enable higher-density multilayer boards with greater feature integration. Advanced surface finishes are also replacing conventional solderable options to improve performance and reliability.

Business and Workforce Challenges

These advances also create significant financial and operational pressures. Key challenges include:

- Funding advanced manufacturing technologies and regular equipment upgrades
- Maintaining a stable supply chain to protect production schedules and control costs
- Recruiting and developing skilled workers who can operate increasingly sophisticated processes

Without sustained investment in equipment, supply resilience, and workforce development, companies may struggle to adopt new manufacturing techniques.

Compliance and Environmental Priorities

The electronics manufacturing industry must address compliance and environmental concerns. Current priorities include:

- Evaluating safer alternatives to PFAS because of environmental and health concerns
- Complying with the Restriction of Hazardous Substances directive to reduce hazardous materials in electronic products
- Reducing water use and adopting more resource-efficient manufacturing practices
- Managing the additional production costs associated with the transition to lead-free materials

Addressing these priorities will be essential to balancing continued industry growth with environmental responsibility. Investing in equipment and processes to reclaim and reuse metals such as copper that have been etched from the board will reduce costs and help conserve vital resources.

The Path Forward: Jumping the Technology Curve

The electronics industry urgently needs more efficient process flows and sustained technological transformation. Advanced packaging is now the

MADE IN THE 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!

IPS INTEGRATED PROCESS SYSTEMS INCORPORATED

U.S. Coast to Coast • Parts and Service Fast

ips-vcm.com · sales@ips-vcm.com · 435-586-1188

PCB fabricators should think like a semiconductor company: razor-sharp focus and relentless pursuit of higher yields, even as technology becomes more complex."

norm, while growth is shifting from consumer electronics such as smartphones and tablets toward millimeter-wave technologies, 5G/6G, high-performance computing, CoWoS (chip-on-wafer-on-substrate), ADAS, and AI.

Supporting these advances requires substantial upgrades in materials, processing equipment, laser drilling, in-circuit testing, registration, and lamination. Progress demands more than a change in mindset; it requires strategic investment in technology, capacity, quality, and workforce development. Companies must prepare engineers and manufacturing personnel to manage critical aspects of workflow, quality, delivery times, and yield. These improvements carry significant costs, but they are essential to moving up the technology curve.

PCB fabricators should think like a semiconductor company: razor-sharp focus and relentless pursuit of higher yields, even as technology becomes more complex. Moving into sub-50-micron lines and spaces, and 75-micron-or-less blind-via diameters, is the new norm. To be even modestly successful, mastering laser via formation, laser direct imaging, precise registration, improved lamination presses, thin materials handling, precision metallization, electroplating, and etching is essential.

Look outside the box and think about how you can minimize handling (to improve workflow as well as yields). A good example is the adoption of modular plating equipment that performs like a semiconductor wafer plating tool. Horizontal processing, including precision etching and metallization, is a game-changing process investment. As HDI and UHDI add more layers, why not invest in alternative processes to make the vias conductive? Direct metallization is an alternative that not only reduces impact on the environment but also offers a precise, highly productive process compared to conventional electroless copper.

Investments Needed

Jumping the technology requirements curve today and moving forward requires significant investment. You can no longer bootstrap your way to prosperity. It is not only about microvias and substrates. There will always be a need for higher aspect ratio plated through-hole circuit boards. High layer counts and aspect ratios reaching and exceeding 30:1 are here. Just look at the server board requirements for AI. Mechanical drilling and plating of these high aspect ratio boards will require the latest in drill-bit design and CNC drilling equipment. Layer counts are exceeding 32 layers.

If that is not enough, thermal management becomes more difficult as these AI devices generate significant heat. Tight control of stub length and back-drill depth tolerance is important for high-speed designs. This matters for 800G/1.6T switch PCBs. Of course, blind and buried vias eliminate the unwanted stub. However, HDI with many sequential laminations is not always the best solution, especially for large, high layer count networking boards.

First, however, fabricators must master key technologies and processing if they wish to be a credible supplier of this growing technology. This includes:

- Tooling and materials selection
- Small hole drilling and micro-drilling
- High aspect ratio via formation
- Back-drilling expertise
- Laser via formation

Table 1: At a minimum, these investments must be made

Investment	Cost (approximate) in 000**
Combo UV/CO2 laser via formation equipment	$300–$800
Laser Direct Imaging	$600–$800
Copper Via Fill plating Cell	$150–$200
High Speed Mechanical Drills/Routing	$500–$800 (with back drill capability)
Lamination Presses (state of the art)	$750–$1,200***
Etching/developer upgrades	$350–$500
Registration systems (optical/pinless bonding)	$400–$600 (pinless optical)
Direct Metallization (horizontal)	$350–$450
Flying Probe Tester	$200–$400 (high end tester)
Via fill technology (paste fill)/planarizer	$150–$250
Electroplating Line new/upgrades	$750–$1,000
Vacuum Laminator	$100–$200 (higher with fully automated line)
Plasma Systems	$400–$500
Engineering/IT upgrades	$300–$400
Online plating solution analyzers	$125–$150
In-house reliability testing/QC	$400–$500

**Cost approximate and could vary depending on style of each piece of equipment

*** State-of-the-art lamination presses are required for leading-edge PCB fabrication. The press is not simply bonding the layers together; it serves to control resin flow, dielectric thickness, copper balance, layer registration, Z-axis CTE, warpage, and final drill registration.

- Desmear, metallization, and hole plugging
- Fine-line and tight-registration, image transfer, and etching
- Blind via plating (super fill copper)
- Quality control and qualifications (test vehicles, reliability verification)

The investments noted in Table 1 should not be considered trivial. They are the cost of entry for competing in a lucrative, less crowded market. The industry is pushing technological boundaries and creating unprecedented opportunities. Demand from AI, high-performance computing, electric vehicles, aerospace, defense, and autonomous systems is accelerating the adoption of heterogeneous integration, chiplet architectures, advanced substrates, and HDI technologies.

These developments increase the importance of coordinating design, materials, assembly, testing, reliability, and manufacturing capabilities. Investment in technologies, materials, tools, and processes will enable fabricators to improve yields, increase productivity, and enter the realm of advanced packaging and high-performance circuit board fabrication.



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.

BY SEAN PATTERSON, CROSSGEN AI

Turn Your AI Into Action: Interview Your Plater



Every shop has knowledge that lives in only one person's head, and then you find out just how much when that person takes a vacation or leaves the company. The line starts acting up, the yield slips, and everyone realizes that they should have captured that person's knowledge.

The cost is not limited to the parts that must be reworked or scrapped. Production slows down as people try to replicate the decisions the experienced operator made without much thought. Supervisors start calling around for answers, engineering gets pulled into the line, and the new operator becomes afraid to touch anything without asking permission. One missing person can quickly turn into five people losing time.

It's not like you didn't already know this. So, to prevent this situation, you try handing the veteran plater a work-instruction template to fill out. A few weeks later, the template comes back half-completed, or it contains the same basic steps already posted beside the line: check the temperature, verify the chemistry, start the cycle, record the readings. All are correct, but none of them capture the experiential knowledge base that makes that plater so valuable. Yet, this has been the standard half-hearted and wholly ineffective attempt to do just that.

So, why doesn't the template approach work? The problem is that most templates are built to document a process after the important decisions have already been made. They are good at holding limits,



PCBAA.ORG

PCBAA
Printed Circuit Board Association of America

steps, and sign-offs. They are not good at uncovering why an operator checks one thing before another or why two readings that appear acceptable can still point to a problem.

The missing information is in the small decisions of what he looks at first, which reading he doesn't quite trust, the sound that makes him turn around, and the condition that tells him to wait five minutes before making an adjustment. After 20 years, those decisions don't feel like decisions anymore; they're simply part of running the line. If you ask how he knows a bath is starting to drift, he may say, "You can just tell." That information has to be drawn out before it can be organized. AI can help.

Start With a Conversation

Instead of trying to pull that knowledge out of your plater with a form, use AI to start a conversation. You just need one experienced operator, one interviewer, and a small block of time. The interview can happen in one sitting or over several shorter conversations on different days. With the operator's knowledge, record each conversation so nobody has to take notes. Explain that you're trying to capture how the process is actually run, so the next operator can learn faster and make fewer mistakes. Most experienced people are happy to teach what they know. They just don't want to spend an afternoon staring at empty boxes on a computer screen.

Start With One Process

Choose one process for the first interview instead of trying to document the whole department. This gives the operator something real to walk through. Ask him to picture the beginning of a normal shift and take you through what happens from the moment he approaches the line. If possible, have the current work instruction, control plan, and a recent job traveler nearby. These can remind him of steps without forcing the conversation to follow the paperwork.

Note: The interviewer does not need to be the company's top plating expert. In fact, there's an advantage to using someone who is still fairly new since new people ask questions that the rest of us have stopped asking:

- "What do you look at first?"
- "What does the line look like when it's running well?"
- "What do you notice before a problem shows up in the data?"
- "What mistakes do new operators make?"
- "What do you do that isn't written in the current instruction?"

It also helps to ask the operator to compare situations, which gives him a way to explain details that may not come to mind when he is only asked to describe a normal day. What is different on a Monday morning after the line has been idle all weekend? What changes after a heavy production day?

If his/her answer to a question is, "You just know," ask about the last time that "knowing" occurred. What caught his attention? Where was he standing? What did he check next? What would have made him stop the line? That's where the useful details begin to surface. "The pump didn't sound right" might mean a particular rattle that only happens when flow starts dropping. Or a reading may still be inside the process window, but an experienced operator notices that it's moving faster than it should. Those are the details a standard work instruction usually misses.

Don't worry if the conversation jumps around. Experienced operators may remember one problem

Use This Exact AI Prompt

Use only the information in this transcript. Do not add steps from outside knowledge.

Organize the material into five sections: what the operator checks, what normal operation looks like, warning signs, the action he takes, and why he takes it. Identify every place where the operator says "usually," "it depends," or "you just know," and turn those statements into follow-up questions. If information is missing, mark it as "not stated."

INTERVIEW TRANSCRIPT:
[Paste the interview transcript here.]

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

PCBAA
Printed Circuit Board Association of America

while describing another. Ask what they saw, what they ruled out, and why they chose one response instead of the other options.

AI can help the interviewer come up with follow-up questions, especially when the conversation stalls. AI can organize the discussion afterward, but it cannot recover a detail that nobody took the time to ask about.

Turn the Interview Into Something Useful

After each conversation, transcribe the recording. This can be done easily with many available tools, including standard AI chat tools. Sending recordings "outside" for transcription is unnecessary and should never be done to safeguard any customer information that may have come up during the process of documenting the process. That matters if the discussion includes customer information, controlled processes, or export-controlled work. Use company-approved equipment, involve IT or compliance when necessary and know where the recording and transcript will be stored.

Do not assume the transcript is accurate. Plating terms, chemical names, equipment brands, and shop-floor noise can trip up even a good transcription tool. Keep the original recording so questionable sections can be checked. Before asking AI to organize anything, have somebody familiar with the department make a quick pass through the transcript. Correct the names of chemistries, tanks, test methods, and equipment so the same item is not treated as two different things. Mark places where the audio is unclear rather than guessing at what the operator said. A clean transcript gives you a much better starting point. Next, use your company-approved AI system to organize the transcript. The instruction should be narrow and specific.

Organize and Separate 'Observations' and 'Actions'

You can also ask the system to separate observations from actions. "The solution is darker near the return" is an observation. "Check the filter and flow" is an action. If you want the system to do this, add this sentence to the prompt: "Separate observations from actions, and organize them into two clearly labeled sections." Keeping the two separate makes it easier for the operator and the engineer to see whether the response actually follows from the condition being described. It also shows where the interview captured a warning sign but never established what should happen next.

The result is a working draft. AI may misunderstand a statement, leave out an important detail, or write something that sounds perfectly reasonable but was never directly said. The draft still needs to go back to the person who knows the process.

Review the draft with the operator in person.

- Read it one section at a time and ask whether he/she would be comfortable training somebody from it.
- When he/she changes a word such as "always" to "usually," stop and ask about the exception. When he/she changes the order of two checks, ask why the order matters.

These corrections are not small edits; they are often the judgment you were trying to capture in the first place.

Review Is Critical

After the operator is comfortable with the draft, engineering and quality should review it. The goal is to capture experience, not turn every habit into an approved practice. The reviewers should confirm that the limits align with the controlled process, that the recommended actions are safe, and that an old workaround hasn't quietly become the new standard.

No document will replace the experience and judgment of a veteran plater. A new operator still needs to see the line run, hear the equipment, handle parts, and watch conditions change. But with this documentation to assist, instead of standing there without knowing what to look for, the new operator arrives with better questions. The veteran spends less time explaining the obvious and more time teaching judgment.

Once the Document Has Been Reviewed, Use It

Do not wait until the experienced plater is out of the building to find out whether the instructions make sense. Have a different operator follow the

"procedure" and sequence as captured in the new document. Have that individual point to the conditions being described, and explain what action he would take. Any place where he hesitates is something to go back and ask the original operator to get further clarification and guidance.

The finished product does not have to become one oversized procedure. Some information may belong in a formal work instruction, while some may fit better in a troubleshooting guide, a setup checklist, or a short training document with photos. Put each piece of knowledge where an operator is most likely to need it.

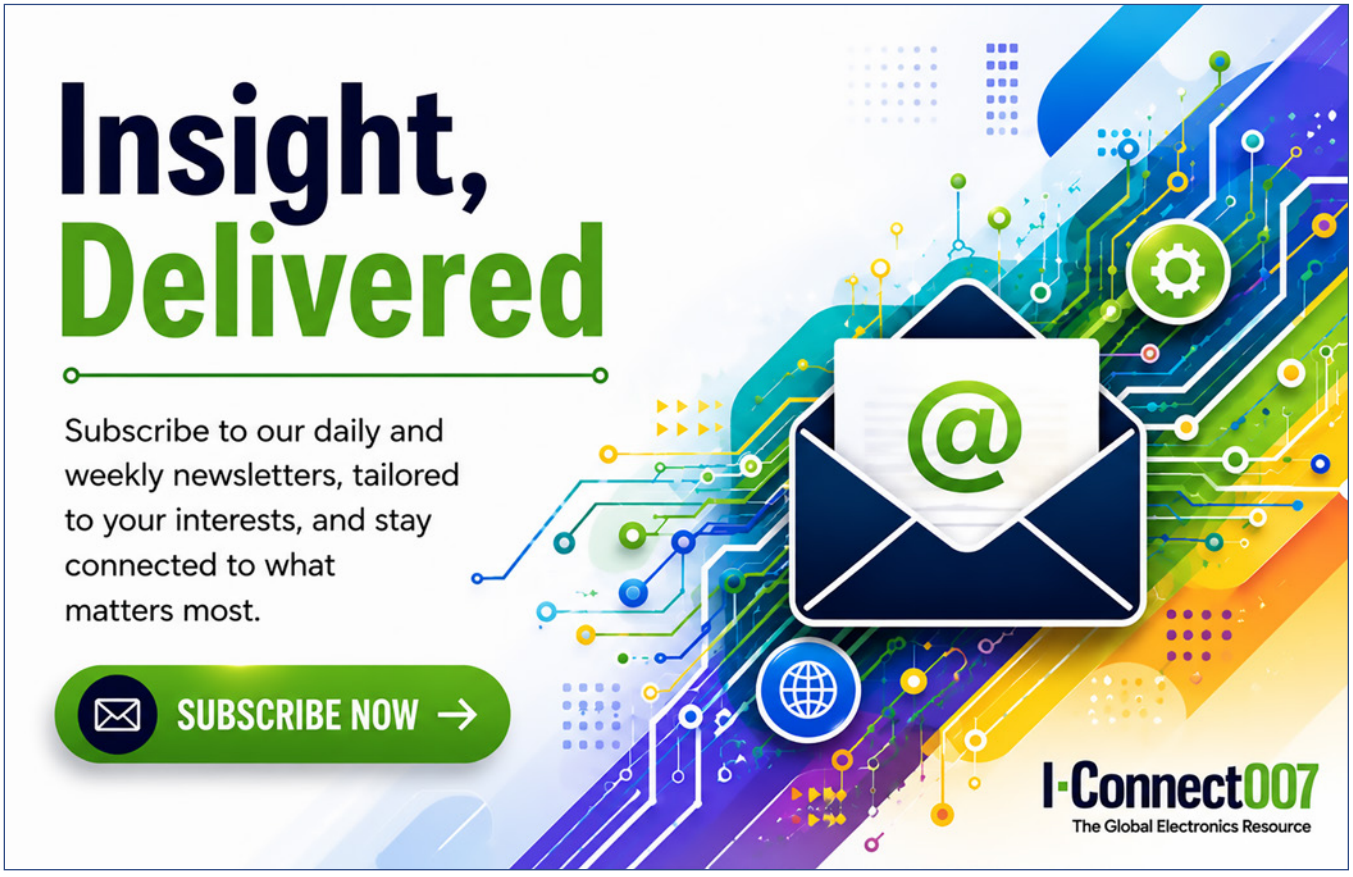
Ownership and Continuous Improvement

Finally, assign the document an owner and a reason for review. Processes change, equipment is repaired, chemistries are updated, and the operator's own thinking continues to develop. Bring the document back out after a significant problem, a process change, or a training session. Ask what the team learned and whether the document could have helped them reach the answer sooner.

You won't capture 20 years of plating experience in a single AI document, but if you can save future operators from learning every lesson the hard way and wasting countless hours of production time, you've made a worthwhile start. The important part is to begin before knowledge walks out the door. Pick one process, schedule the first conversation and listen closely. The first interview will not capture everything, but it will show the team what is hiding between the written steps once AI organizes the conversation. Once people see the value, the next conversation becomes easier.



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, was CRO of Nano Dimension, an Amazon executive, and is a U.S. Navy submarine veteran.



BY GLOBAL ELECTRONICS ASSOCIATION

IPC Standards Released for Q3 2026



Each quarter, the Global Electronics Association releases a list of standards that are new or have been updated. To view a complete list of newly published standards and standards revisions, translations, proposed standards for ballot, final drafts for industry review, working drafts, and project approvals, visit electronics.org/ipc-standards.

These are the latest releases for Q3 2026.

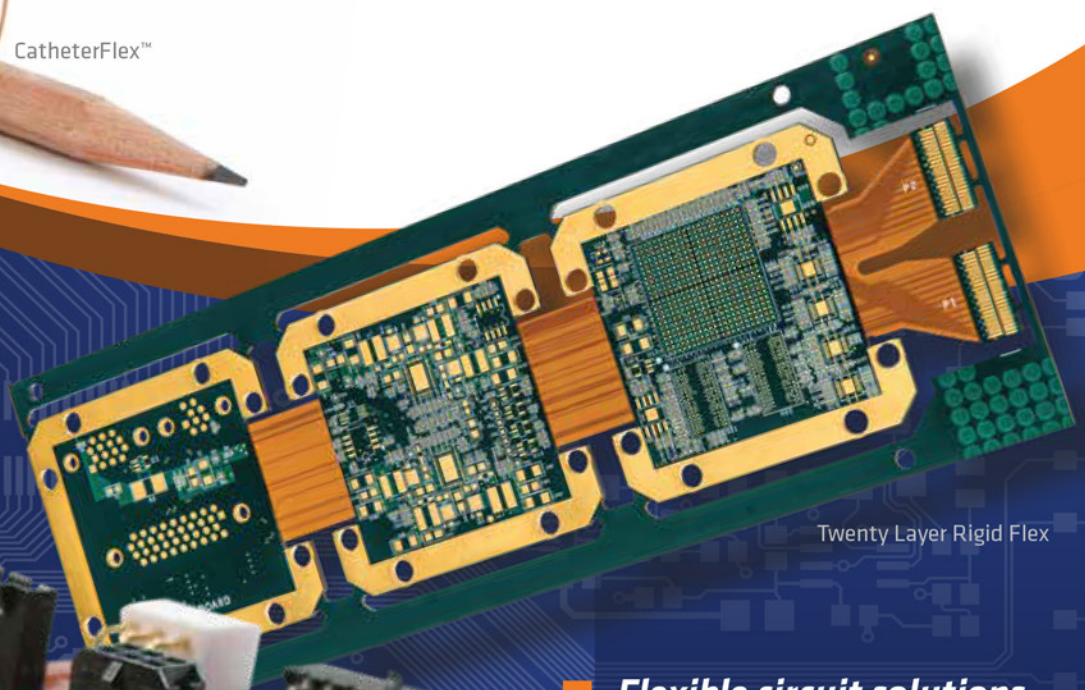
IPC-6013F, *Qualification and Performance Specification for Flexible/Rigid-Flexible Printed Boards*

IPC-6013F covers qualification and performance requirements for flexible printed boards designed to IPC-2221 and IPC-2223 and utilizing materials qualified to the IPC-420X series. The flexible printed board may be single-sided, double-sided, multilayer, or rigid-flex multilayer. All these construc-

ALL YOUR FLEXIBLE SOLUTIONS IN ONE PLACE



CatheterFlex™



Twenty Layer Rigid Flex



And, Assembly Too!

- *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

all flex
SOLUTIONS

tions may include stiffeners, PTHs, microvias, and blind/buried vias. Revision F incorporates new and updated requirements for rigid-to-flex transition zones, printed board edge plating, printed board cavities, dewetting, copper penetration as a function of wicking, etchback, and hole formation, and also minimum dielectric spacing.

IPC-2591 *V2.1, Connected Factory Exchange (CFX)*

IPC-2591 V2.1 standard establishes the requirements for the omnidirectional exchange of information between manufacturing processes and associated host systems for assembly manufacturing. This standard applies to communication between all executable processes in the manufacture of printed board assemblies, whether automated, semiautomated, or manual, and to related mechanical assembly and transactional processes. IPC-2591, Version 1.7, includes enhancements for closed-loop between AOI and SPI, and fields added to Heartbeat to provide repeated active fault and recipe information in each message. Visit www.ipc-cfx.org for software developer resources, training programs and information on the IPC-CFX-2591 Validation System and Qualified Products List (QPL).

IPC-Hermes-9852 V1.7, *The Global Standard for Machine-to-Machine Communication in SMT Assembly*

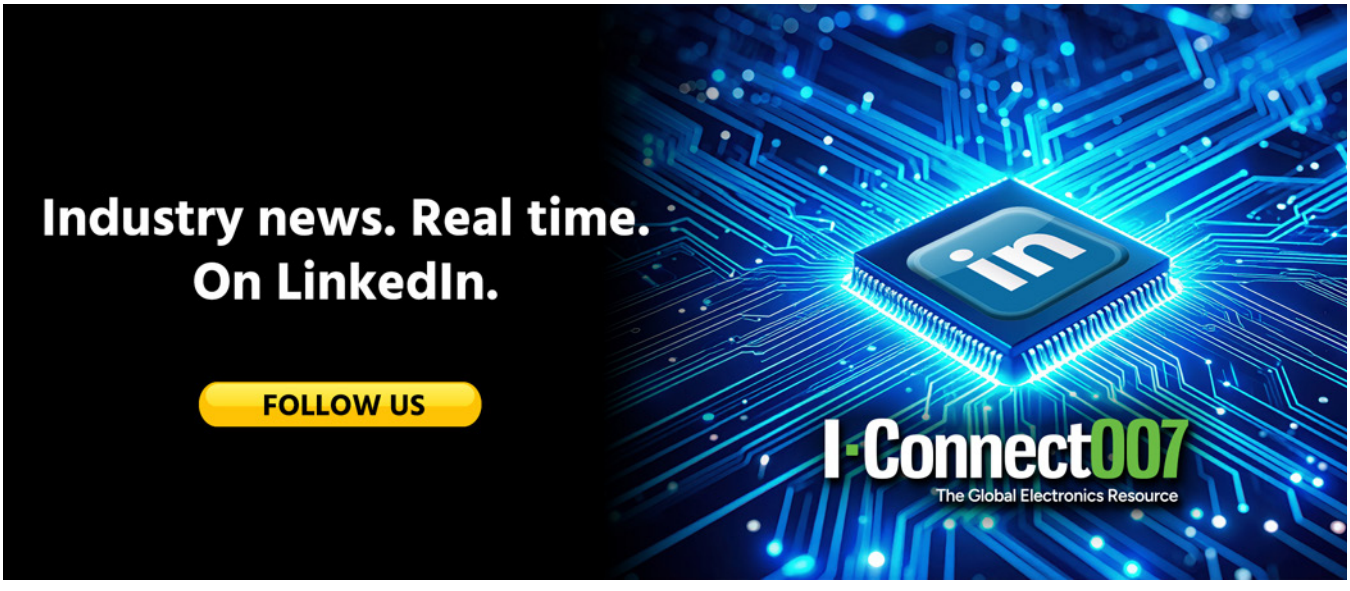
IPC-Hermes-9852 V1.7 standard is an open, vendor-neutral standard that replaces the legacy SMEMA interface to enable seamless board-handover communication between machines on an SMT assembly line. By transmitting rich product and process data alongside the physical handoff, it eliminates manual setup errors, reduces changeover times, and serves as a foundational building block for factory automation.

IPC J-STD-002F, *Solderability Tests for Component Leads, Terminations, Lugs, Terminals, and Conductors*

IPC J-STD-002F prescribes test methods, defect definitions, acceptance criteria, and illustrations for assessing the solderability of electronic component leads, terminations, solid wires, stranded wires, lugs, and tabs. The IPC-J-STD-002E standard also includes a test method for the resistance to dissolution/dewetting of metallization. IPC-J-STD-002F is intended for use by both supplier and user. The IPC-J-STD-002F standard was developed by ECIA, IPC and JEDEC.

IPC-4105, *Specification for Metal Base Copper Clad Laminates for Rigid Printed Boards*

IPC-4105 is a new document that covers the requirements for metal-base materials that are referred to as metal-based laminate listed in the specification sheets that are contained in the list of the main body. These are to be used primarily for rigid and multilayer printed boards for electronic interconnections. **I-CONNECT007**

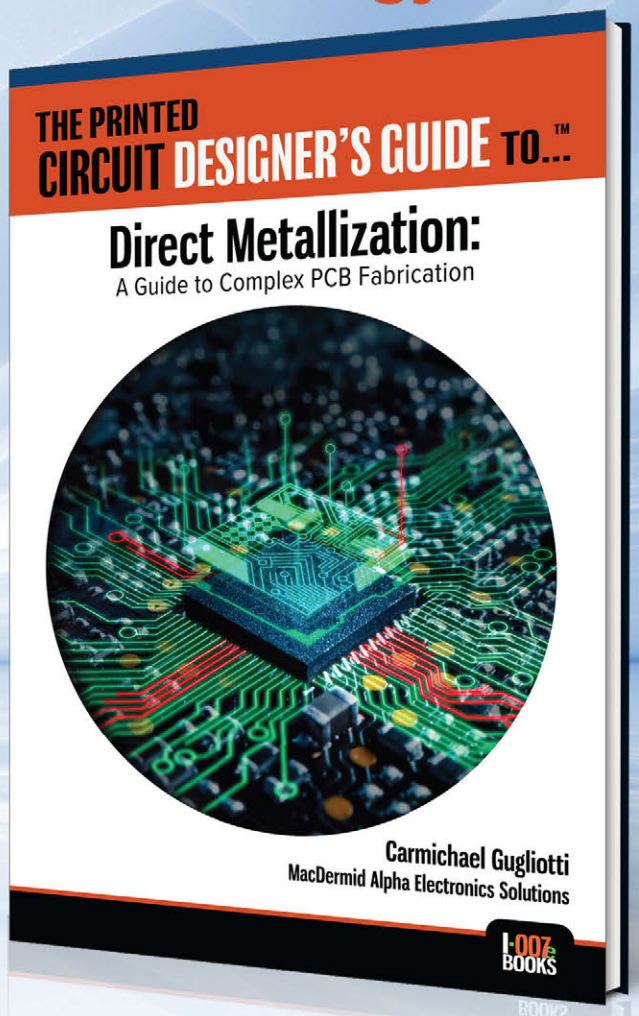


DIRECT METALLIZATION

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

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!

I-Connect007
The Global Electronics Resource

SMT007 PCB007 DESIGN007

CAREER OPPORTUNITIES

all4-PCB®
NORTH AMERICA

FIELD SERVICE ENGINEER

PCB & Advanced Manufacturing Equipment

all4-PCB is seeking an experienced Field Service Engineer to install, commission, maintain, and support advanced production equipment used in PCB, semiconductor substrate, and high-technology manufacturing.

Equipment includes **direct-imaging systems for solder mask and dry film, AOI/AFI and machine-vision inspection systems, vacuum plasma equipment**, and other automated PCB manufacturing equipment represented by all4-PCB.

The ideal candidate has hands-on experience with industrial automation or capital equipment and strong electrical, mechanical, optical, and software troubleshooting skills. Experience with cameras, lighting, optics, motion control, servo systems, PLC-controlled equipment, vacuum pumps, gas systems, RF power, UV/LED or laser imaging, industrial PCs, and networking is highly valued.

Responsibilities include equipment installation, calibration, preventive maintenance, troubleshooting, process setup, customer training, technical documentation, and coordination with overseas manufacturers.

This position primarily supports customers in the **Texas, California** and requires extensive North American travel, plus occasional travel to Europe and Asia for factory training.

Benefits: Health insurance • Life insurance • 401(k) Paid time off • Ongoing technical training

Apply Now!

HT
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!

CAREER OPPORTUNITIES

Pioneers SYSTEMS

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!

PCB RECRUITMENT LTD

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!

CAREER OPPORTUNITIES

ARLON ELECTRONIC MATERIALS

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!

NCAB GROUP
Integrated PCB Production

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!

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!

REMTEC®

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.

Email careers@eptac.com to apply.

Apply Now!

CAREER OPPORTUNITIES

P·K METAL

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!

TTC The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

Apply Now!

CAREER OPPORTUNITIES

ACE
ACCURATE CIRCUIT ENGINEERING

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!

INSULECTRO

Are You Our Next Superstar?!

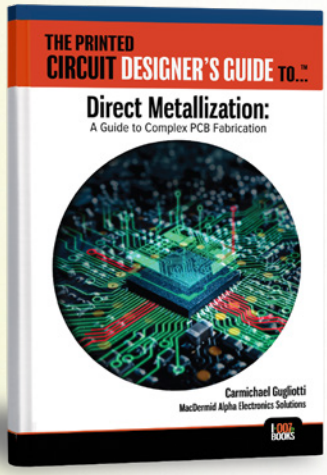
Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers to build better boards 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!

I-Connect007
The Global Electronics Resource

EDUCATIONAL RESOURCES



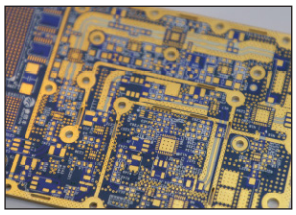
BOOKS

"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



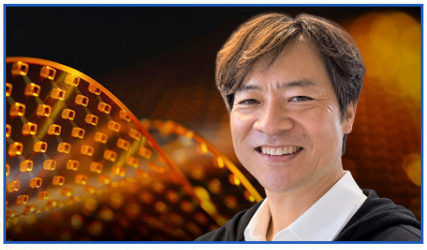
PCB Cost Drivers

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 Jin Hyun Hwang, Qnity

LISTEN NOW >



Voices of the Industry

with Kelly Davidson, NCAB

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
MICHAEL DEL PURGATORIO

Ad Design
TOBEY MARSICOVETERE

Director of Digital Operations
and Media Systems
BRYSON MATTIES

Cover Design
TOBEY MARSICOVETERE

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.

October 2026, Volume 1, Number 10
I-CONNECT007 MAGAZINE is published monthly,
by IPC Publishing Group, Inc., dba I-Connect007.

ADVERTISER INDEX

Accurate Circuit Engineering 7
All Flex Solutions 109
all4-PCB 53
ASC Sunstone 11
atg Mycronic 61
Candor Industries 97
Chemcut 73
Electra Polymers 43
Elite Materials 89
Flexible Circuit Technologies 25
Global Electronics Association 45, 87
GreenSource Engineering 65
GreenSource Fabrication 47
I-007eBooks 2, 111
Insulectro 5, 29, 31, 93
Integrated Process Systems 99
Isola 77
Matrix USA 63
Millennium Circuits 35
MKS' Atotech 17, 19
MKS' ESI 41
NCAB Group 67, 69
On the Line with 3
PCBAA 103, 105
Pluritec 81
Polar Instruments 23
Qnity 59
Schmoll America 57
Technica USA 9
US Circuit 55
USA Microcraft 85
Ventec International Group 21


Problems solved!
SIGNAL INTEGRITY
PROPER PLANE DESIGN
DECEMBER 2025
THE AUTOMATION ADVANTAGE
THE MAGAZINE
I-Connect007
October 2026
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY
Enabling the Interconnect Technologies That Transform Our World
MKS Inc.
Plus: Q&A with MKS Inc. CEO and President, John T.C. Lee
PCB007 DESIGN007
PCB MAGAZINE
Advanced Materials... Advanced Solutions
SMT007 MAGAZINE
OCTOBER 2026
SMT007 MAGAZINE