THE MAGAZINE
I-Connect007
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

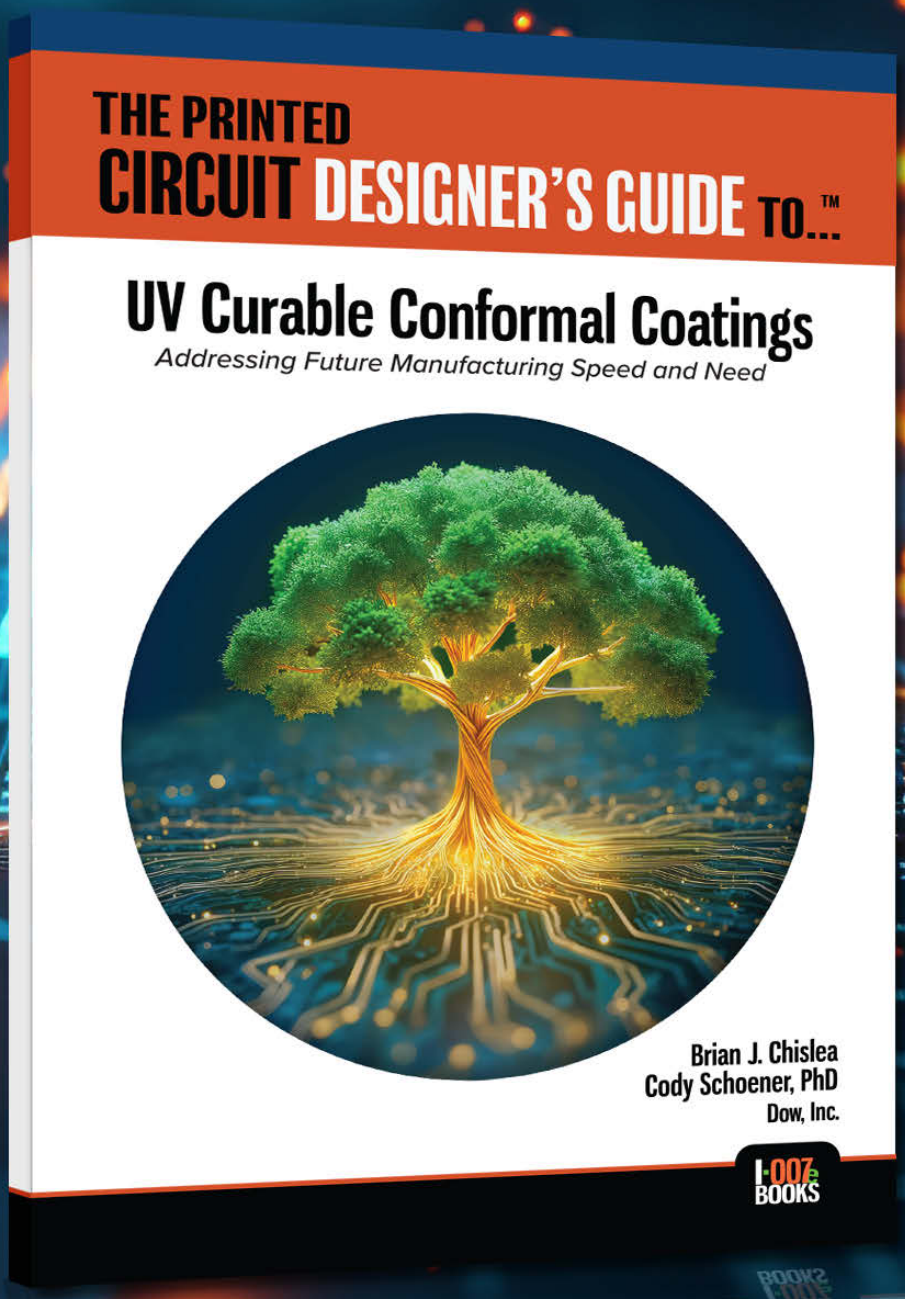
But Will it Fly?



PCB007
DESIGN007

NOVEMBER 2026

Protection That Performs



Look Inside

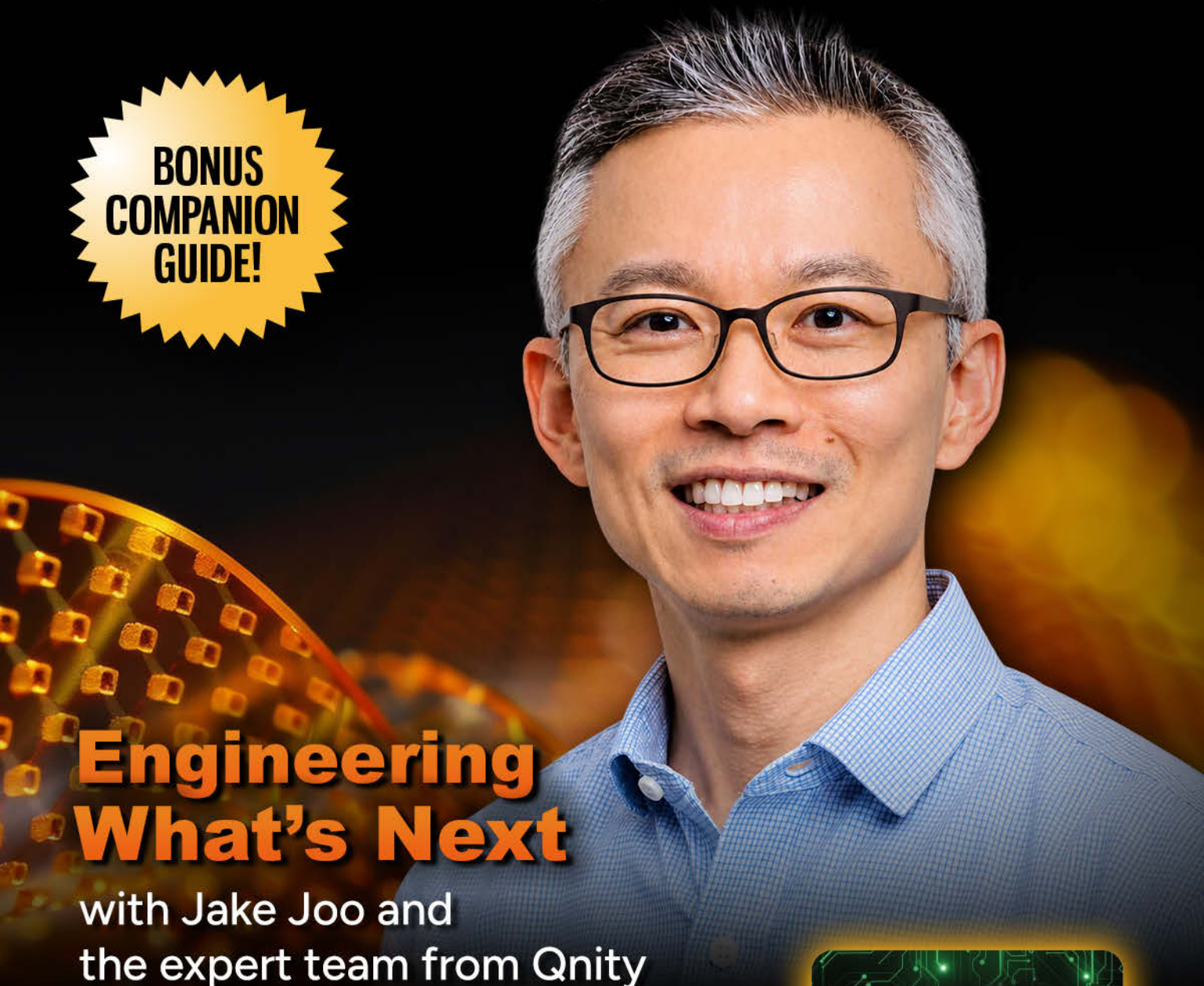
"This book contributes useful clarity to an area of growing importance in high-reliability electronics manufacturing."

—Stanton Rak, PhD, SF Rak Company

I-007eBOOKS

PODCAST | SERIES 9

BONUS COMPANION GUIDE!



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!



The Art of Getting It Right

Even the best-laid plans can produce a turkey. This issue explores what makes PCB designs perform, from polymer-glass interposers and power losses to ceramic substrates and flex circuits. We also examine fabrication processes, materials, and workforce development. Plus, four industry veterans share their most memorable career "turkeys." What's yours?

DESIGN007

Features

28 **My Biggest Design Turkey: The BGA That Cost Me Everything**
by Stephen Chavez

Target Condition

38 **My PCI Edge Card Connector Design 'Turkey'**
by Kelly Dack

Articles

70 **What Standard Power Supply Design Reviews Miss Before PCB Layout Sign-off**
by Kirsch Mackey

86 **Qualified Flexibility: More Choices We Can Trust**
by Jan Pedersen

Columns

The Pulse

56 **New People, New Book**
by Martyn Gaudion

Built for What's Next

66 **The Hidden Cost of 'Good Enough' PCBs**
by Michael Gleason

Beyond Design

100 **The Next Evolution of Power Delivery for AI Data Center PCBs**
by Barry Olney

Connect the Dots

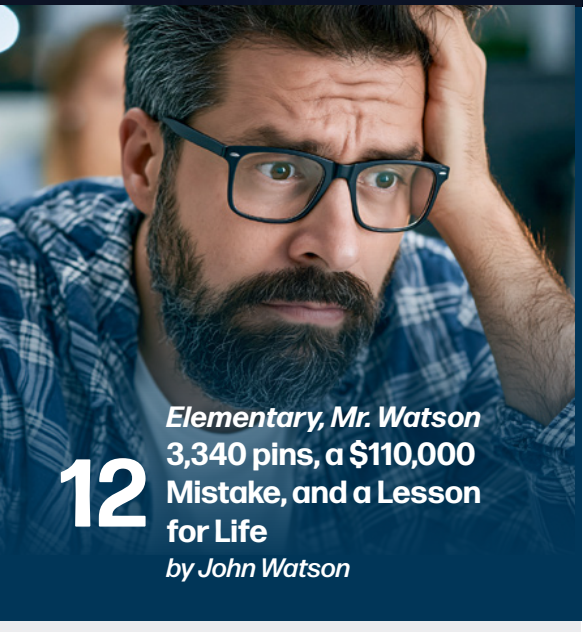
108 **From the Playbook to the Production Floor**
by Kelly Atay

Designers Notebook

112 **A Strategy for Polymer-Glass Interposer Design**
by Vern Solberg

From the Editor

8 ***Marcy's Musings***
A Turkey to Remember
by Marcy LaRont



12 ***Elementary, Mr. Watson***
3,340 pins, a $110,000 Mistake, and a Lesson for Life
by John Watson

In Every Issue

84 MilAero007 Highlights
121 Career Opportunities
128 Educational Resources
129 Advertiser Index

THE MAGAZINE I-Connect007
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

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)**

FEATURE

The Chemical Connection
96 A Career 'Turkey' Story for November
by Don Ball



Shorts

91 *Below the Surface*
From Substrate to System: Why Integration Is the Real RF Breakthrough
by Chandra Gupta

Flex007

Flex Connections
24 I Think I Need a Flex Circuit: Now What?
by Chris Clark

82 Plant Biometric Sensors: Flex PCBs Listening to the Language of Crops
by Anaya Vardya

THE MAGAZINE I-Connect007
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

PCB007

Articles and Interviews

16 **Why Advanced Chemistry and Surface Finishing Increasingly Determine Next-generation PCB Manufacturing**
by Harald Ahnert

32 **Schmoll America Sees Unprecedented Equipment Demand**
with Kurt Palmer and Evan Howard

46 **Through-hole Interconnection Using the Partition Method: Structural Design, Manufacturing, and Applications**
by Huang Mingan, Xiang Jinpeng, Huang Xiao, and Ye Feihua

76 **The Evolution of Horizontal Etching**
by Richard Nichols

Columns

Powering the Future
42 **The Real Ceramic Circuits**
by Brian Buyea

Trouble in Your Tank
60 **The Chemistry of Sustainability**
by Michael Carano

The Right Approach
94 **How AI Is Changing Quality Metrology**
by Steve Williams

American Made Advocacy
80 **A New Congress Needs to Know Why PCBs Are Essential to Security**
by Shane Whiteside

Don't Buy AI, Learn AI
106 **You Can't Buy a Baseline**
by Sean Patterson

Happy's Tech Talk #51
116 **DIY Spectroscopy (Colorimeter) Control**
by Happy Holden



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 Turkey to Remember

BY MARCY LARONT, I-CONNECT007

It's November, and that means it's turkey time for many of us. While the origins of Thanksgiving have become the subject of debate, the turkey's place at the center of the table remains largely undisputed. Personally, I find turkey delicious. Yet somewhere along the way, this beloved bird became slang for a loser or a dud, presumably thanks to its reputation for being, well, not terribly bright.

At one company where I worked, we opened every all-company meeting with "oops" stories. These were opportunities to share something that had gone wrong and what we learned or changed as a result. My boss understood that acknowledging mistakes, rather than punishing them, created a culture of honesty, transparency, and continuous improvement. It also made for some memorable team-building moments.



So, this month, we had a little fun with that second definition by asking some of our most esteemed and knowledgeable contributors to share a "turkey" story from their careers. It's a mistake or misstep that ultimately made them better at what they do. With recruiting, developing, and retaining skilled workers remaining a persistent industry challenge, the experiential wisdom shared by Don Ball, John Watson, Stephen Chavez, and Kelly Dack is both timely and inspirational. Sometimes our best lessons come from our biggest turkeys. (See the sidebar where I share one of my own biggest career "turkeys.")

Of course, there is plenty more to explore in this issue.

On the PCB design side, Vern Solberg guides PCB designers toward the sophisticated interconnect technologies more commonly associated with semiconductor fabrication by exploring a strategy for polymer-glass interposer design. Martyn Gaudion previews a new book from Polar, while Barry Olney tackles micro-ohm power considerations for AI data center PCBs, where even the smallest electrical losses can have significant consequences.

Brian Buyea of Remtec challenges conventional thinking about PCB substrates, making the case for ceramic alternatives to traditional organic materials. His discussion encourages designers to reconsider what a PCB can be and where different materials may offer meaningful advantages.

Kelly Atay of ASC Sunstone digs into PCB quoting, a process often viewed only through the lens of price, and explains its sometimes-overlooked connection to delivering quality boards on time. In a special contribution, Kirsch Mackey provides a practical PCB layout signoff checklist to help designers avoid costly problems during power supply design reviews. Michael Gleason explains that a board's design needs to be more than just "good enough." Designers need to consider today's demanding requirements for the application they are designing for.

Our flexible circuit coverage offers two particularly interesting perspectives. Chris Clark examines what happens when designers ignore IPC-2223, *Sectional Design Standard for Flexible/Rigid Flexible*

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

Marcy's Own Turkey Story

Some of you may be asking, "Marcy, what was one of your 'turkeys?'" Of the myriad errors I have made throughout my professional life, one in particular stands out.

I was managing a busy inside sales/customer service group at a quick-turn, prototype shop for a large PCB company in Northern California. My department was often understaffed, which left me to check capacity, quote jobs, and enter many orders into the system myself.

I don't know when I figured it out, but I distinctly remember walking into the production control manager's office and realizing that I had misentered a $25,000 order that was almost finished on the factory floor. Because of the error, we would have to scrap those boards, eat the cost, start over, and still ship to the customer in 48 hours.

You can probably imagine the feeling: My stomach turned, and I went a little out of my body, wondering how this might affect my employment status. I called in the GM and told him what happened. Luckily, he was pretty nice about it. I kept the job, and we remade the boards.

What did I learn? It made me slow down, be more careful, and manage my future mistakes proactively once I became aware of them. It also informed my approach toward managing my employees and their inevitable errors—something I believe to be an important part of management to this day.

Printed Boards. Unfortunately, it happens more often than you might think, and the consequences can be significant. Meanwhile, Anaya Vardya takes us into the fascinating world of plant sensors, where tiny flexible circuits may play a surprisingly important role in addressing one of humanity's greatest challenges: feeding a growing global population. It is a compelling reminder that the technologies we work with every day can contribute to solutions far beyond the electronics industry itself.

On the manufacturing side, FSQuality engineers discuss through-hole interconnect technology, particularly the challenges of volume production, in the first of a two-part series. Happy Holden focuses on DIY spectroscopy, and Steve Williams examines the essential role metrology has always played in manufacturing and how AI is elevating its capabilities, improving reliability, and reducing risk in ways previously out of reach.

Michael Carano takes a thoughtful look at sustainability, arguing that environmental responsibility and good business practices need not be at odds. He challenges us to think more strategically about how we approach these priorities for the benefit of all stakeholders.

Jan Pedersen addresses flexibility in the laminate material supply chain, why having options matters, and the risks facing OEMs, designers, and fabricators who fail to plan accordingly. Harald Ahnet of MKS' Atotech discusses surface finishes for advanced manufacturing, and don't miss my interview with Schmoll America to discover why the company is experiencing unprecedented growth.

Learn more about implementing AI on the factory floor from Sean Patterson, and how to get your voice heard in Washington from PCBAA Chair Shane Whiteside.

There is certainly no shortage of important stories to tell, and we feel fortunate to discuss the extraordinary range of technologies, challenges, and opportunities facing our industry. In December, *I-Connect007 Magazine* will take a closer look at workforce development and share additional interviews and insights from October's SEMICON West.

In the meantime, enjoy your Thanksgiving and remember: A turkey on the table is something to celebrate. A turkey in your career just might be, too.

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.

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



3,340 pins, a $110,000 Mistake, and

Lessons for Life

BY JOHN WATSON, PALOMAR COLLEGE

At the start of every PCB design course, I tell my students something that usually catches them off guard: “I'm no smarter than anyone sitting in this classroom. The only difference between you and me is that I have more experience.”

After more than four decades in the electronics industry, I've certainly collected plenty of experience. I've worked with talented engineers and designers, watched technology evolve, and been part of some amazing projects. But some of my greatest lessons didn't come from the projects that went right. They came from the ones that went terribly wrong.

That leads to the second thing I tell my students: “One of the reasons I'm qualified to teach at all

EMC

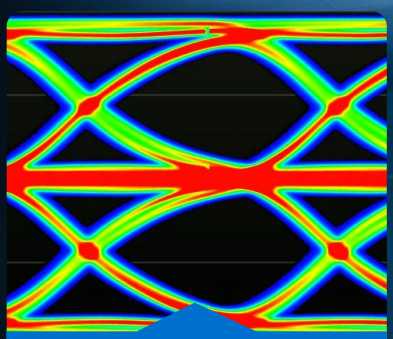
STRIVE FOR **EXCELLENCE**

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

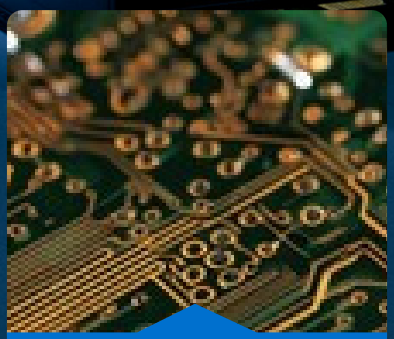
Introducing EM-896K2 and EM-896K3

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

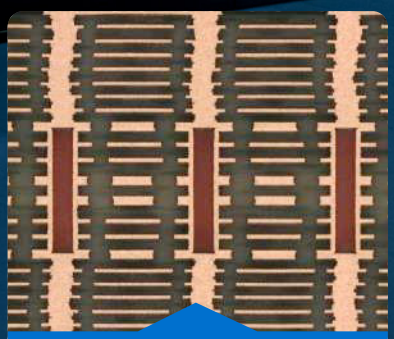
- Excellent Dk/Df performance for high-speed signal integrity (< 1.0 dB/inch @ 53GHz)
- Low X/Y-axis CTE for improved dimensional stability includes CoWoP applications
- Proven multi-lamination reliability (≥12x laminations)



Excellent Dk/Df performance



Ultra-low X/Y-axis CTE



Proven multi-lamination reliability

INSULECTRO

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



Yet despite these safeguards, mistakes still slip into our designs."

is simply because I've already made most of the mistakes you're about to make." That usually gets a laugh, but unfortunately, it's close to the truth. Some mistakes were small, easily corrected, and quickly forgotten. Others made my stomach drop, my blood pressure rise, and my career flash before my eyes. You know those moments when you stare at a design and think, "Well, this is going to be an interesting meeting."

Anyone who's spent enough time in engineering knows what I mean. PCB design gives us plenty of opportunities to get things wrong. A misplaced decimal point, an incorrect footprint, a reversed connector, or one innocent assumption can turn months of work into an expensive paperweight. To prevent these disasters, we rely on ERC, DRC, simulations, design reviews, and countless verification procedures.

Yet despite these safeguards, mistakes still slip into our designs. After all, engineering is a human endeavor, and even the most experienced among us can overlook a detail or make an incorrect assumption. No matter how sophisticated our tools become, nothing replaces careful judgment, attention to detail, and a willingness to question our own decisions.

I remind my students that some of life's most valuable lessons aren't found in textbooks or lectures. They're learned the hard way, usually when something goes wrong, and we wish we could turn back the clock 24 hours. But there's a catch: A mistake only becomes a lesson if we're willing to take it to heart. Otherwise, we're just getting better at making the same mistakes. There's a big difference between having 40 years of experience and having one year of experience repeated 40 times. Experience isn't simply measured in years, but by what we've learned and how those lessons have changed us.

I certainly have an entire library of stories to prove it. One involved a complex $30,000 AMD Versal Premium XCVP1402 FPGA chip with 3,340 pins; it was a $110,000 mistake, and a lesson I'll never forget. The design required 18-layer, high-density routing, multiple power and ground planes with controlled impedance, high-speed signals, differential-pair length matching, and thousands of connections that needed precise pin assignments.

Managing the pin assignments and ensuring every signal reached its intended destination was no small task. The schematic was completed, the PCB laid out, and the design released for fabrication and assembly. Everything appeared to be progressing as planned. Unfortunately, appearances can be deceiving.

We didn't discover the FPGA was wired incorrectly until the boards were fabricated and assembled. By then, the components were soldered in place, the money was spent, and correcting the problem had become considerably more expensive. What would have been a relatively simple design correction became a $110,000 mistake. Apparently, our FPGA wasn't the only thing with crossed wires. If you've ever had to explain to management how a design made it through fabrication and assembly before someone discovered a problem, you know that's a meeting nobody volunteers for.

Lesson Learned #1

When things go wrong, you need to own the mistake. It's tempting to blame someone or start making excuses. When you're staring at $110,000 worth of assembled boards that don't work as intended, that temptation can be overwhelming. But assigning blame doesn't fix the connections. The important questions are, "How did we get here, and what are we going to do about it?"

Accepting responsibility is easier to discuss than to practice. Nobody enjoys admitting they made a costly mistake, especially with their professional reputation on the line. My first reaction was a sinking feeling, followed by the hope that we'd overlooked

a simple explanation. But worrying about how bad things looked wasn't helping. The boards were sitting there, the problem was real, and someone needed to figure out what came next. I had to put my pride aside and look for a solution.

That's when the distinction between responsibility and problem-solving becomes important. Owning the mistake is the first step, but taking action is what makes the difference. Instead of dwelling on the $110,000 already spent, I needed to focus on what we could save. Could we correct the FPGA connections through reprogramming? Could we rework the boards? Would we have to start over? We could examine later how the mistake occurred and what needed to change in our design process. For now, our priority was finding a way forward. Feeling terrible about a mistake shows you care, but it won't get a circuit board working again.

Lesson Learned #2

Don't let the mistake's size grow bigger than your ability to solve it. It's easy to become so focused on the consequences that you lose sight of the problem itself. A $110,000 mistake certainly gets your attention, but staring at that number doesn't bring you closer to a solution. Neither does imagining how you'll explain it at the next management meeting. One of the most important things an engineer can do is separate emotion from engineering. The mistake has happened, and worrying won't change that. But we can influence what happens next.

After the initial shock, I reminded myself of something I've told countless students: Engineering is fundamentally about solving problems. Rather than viewing this as one enormous $110,000 disaster, we needed to break it into manageable questions. What exactly was wired incorrectly? Which connections were affected? What could be corrected, and what couldn't?

Most importantly, what options remained before declaring the boards a total loss? By addressing one question at a time, we could replace uncertainty with facts and frustration with a plan. Surely, somewhere among those 3,340 pins, there had to be a solution. The challenge was finding it before someone suggested dumping the entire project.

Lesson Learned #3

Once a failure has occurred, we can't change the mistake, but we can control our response. We can let it define the outcome or use what we learned to challenge our assumptions and find a way forward. Sometimes the hardest part is moving beyond the embarrassment and returning to what engineers do best: solving problems. In the words of engineer and author Henry Petroski. “Failure is central to engineering.”

As it turned out, the solution wasn't where we initially expected to find it. That's a story for my next column, where I'll share how we tackled a 3,340-pin nightmare, what it took to get those boards working, and how a $110,000 mistake became one of the most valuable engineering lessons of my career. **I-CONNECT007**



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

I-Connect007 Educational Resource Center

Podcasts | White Papers | Roundtables | Books | Webinars | Subscribe

BY HARALD AHNERT, MKS

Why Advanced Chemistry and Surface Finishing Increasingly Determine Next-generation PCB Manufacturing

The next phase of PCB manufacturing is defined not just by design capability, but also by the ability to reproduce advanced structures in large quantities with a stable yield and reliable performance. As high layer count multilayer and HDI PCBs transition to finer features, thinner dielectrics, and tighter assembly margins, achieving manufacturing consistency becomes paramount. In these circumstances, surface condition, seed layer and copper deposit quality, as well as final finish stability directly affect production yield and downstream performance (Figure 1).

As AI and HPC architectures increase interconnect density, bandwidth, and power delivery demands, PCB manufacturers must supply more advanced high layer count multilayers, HDI, substrate-like PCBs, and package substrates that meet tighter consistency requirements. In that environment, manufacturing becomes a determining factor in whether next-generation interconnect structures can be produced with acceptable yield, reliability, and assembly performance.



Figure 1: Illustration of an advanced HDI PCB build-up structure highlighting key steps for optimized interconnection between layers.



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.



Figure 2: Copper features and surface characteristics become increasingly critical as PCB designs move toward finer geometries, tighter tolerances, and more demanding performance requirements.

"As AI, high-performance computing, and advanced packaging continue to reshape the electronics industry, the conversation is no longer simply about creating smaller features or denser interconnects," says Dave Henry, Executive Vice President and GM of Materials Solutions Division, MKS. "The real challenge is manufacturing these increasingly sophisticated structures reliably, consistently, and at scale. What differentiates industry leaders today is not only design innovation, but the ability to control every critical interface throughout the manufacturing process.

"Surface preparation, metallization, copper deposition, and finishing all contribute to the final performance window of the system," he continues. "In many cases, the reliability and efficiency of tomorrow's AI infrastructure are being determined by manufacturing steps that remain largely invisible to the end user. As the boundaries between semiconductors, substrates, packaging, and PCBs continue to blur, success will increasingly depend on an integrated approach that optimizes materials, chemistry, equipment, and process control as a complete manufacturing ecosystem."



Dave Henry

Miniaturization Has Shifted the Focus to Process Interfaces

As interconnect density increases, even minor weaknesses can become systemic. For example, a marginal surface preparation step can lead to uneven metallization, unstable copper deposition, and finish variability. Ultimately, this can result in assembly fallout or risk to field reliability (Figure 2).

One practical lesson is that many modern defects are "handoff defects" between steps rather than isolated failures within a single tank or tool. This is why the discussion of wet-chemistry process chains spanning multilayer bonding, desmear/PTH, plating, surface treatment, and selective finishing is increasing. The focus is less on breadth for its own sake and more on controlling the interfaces where variability accumulates, particularly in PCB production contexts, where microvia integrity, pad condition, and final assembly yield are all linked to the stability of upstream processes.

Surface Preparation: Adhesion Without Paying the Signal Integrity Penalty

As PCB, HDI, and package-substrate designs evolve for higher-speed and high-frequency applications, surface preparation is no longer only about maximizing adhesion. The focus is increasingly on preserving reliable bonding while reducing conductor loss, line-geometry distortion, and process variability. Requirements such as low signal loss, controlled copper roughness, compatibility with low loss dielectric materials, and minimal or zero line-width reduction are therefore becoming more important in the assessment of inner-layer bonding and adhesion promotion.

MKS' Atotech addresses these requirements with its bonding and adhesion-promotion portfolio, including the BondFilm® HF series, BondFilm® EX, and NovaBond® PX-S2. These technologies are designed to balance

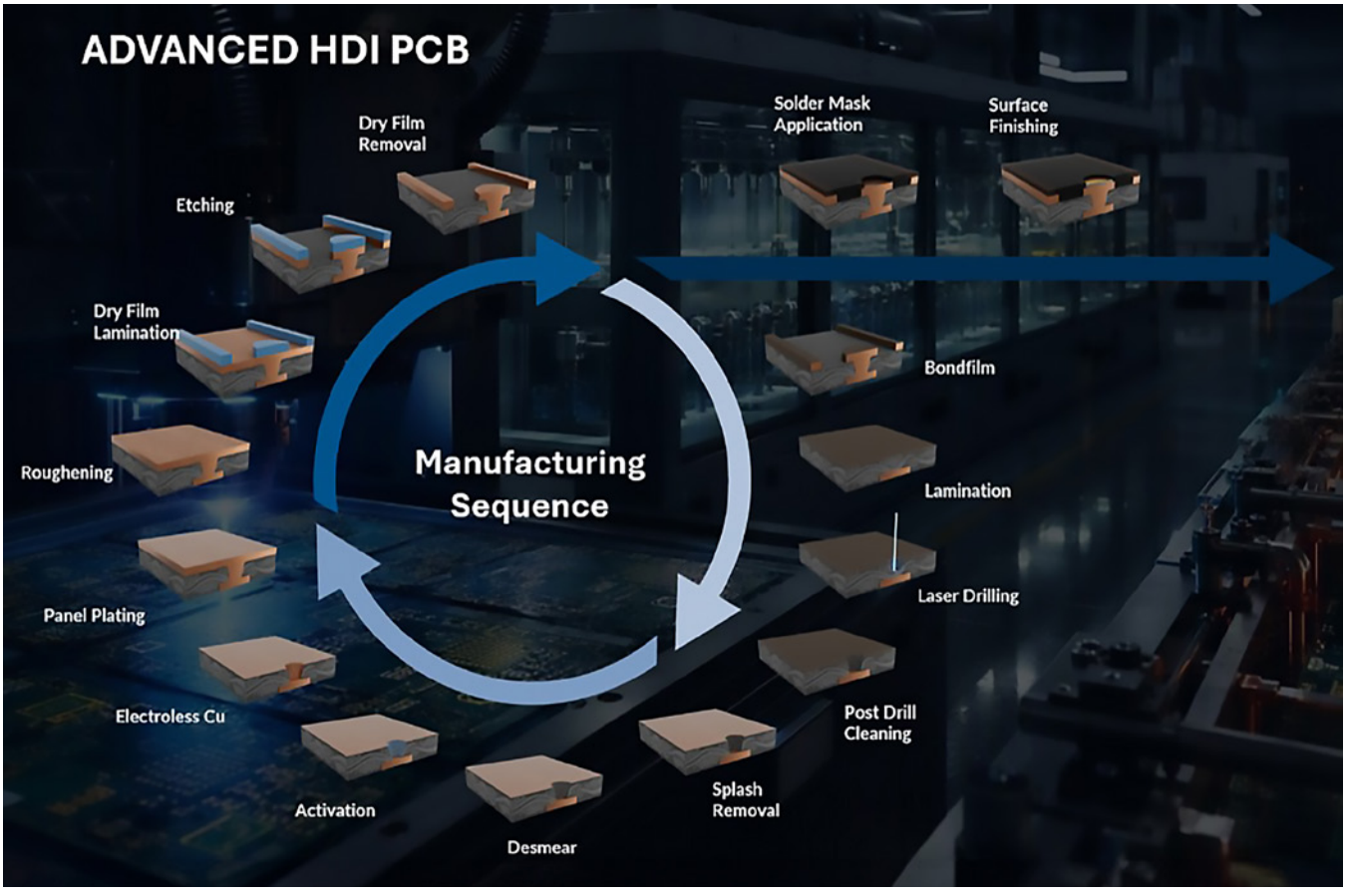


Figure 3: Advanced HDI PCB production involves multiple interconnected steps, from surface preparation and lamination through drilling, desmear, metallization, plating, and final surface finishing.

adhesion strength, reduced surface roughness, and compatibility with low loss dielectric materials for high-frequency applications.

The practical implication is that the copper–dielectric interface is becoming a critical design and process factor in advanced PCB, HDI, and package-substrate architectures. Surface treatment should therefore be evaluated not only as a safeguard for mechanical reliability, but also as a contributor to signal integrity, dimensional control, and stable long-term performance.

Desmear and Metallization: Foundation for Reliability

Since the original "through-hole" was developed, the electroless copper layer has created the basis on which subsequent processes are applied, making it critical to reliability, especially in new HDI and substrate-like architectures where vias are smaller and dielectric systems are more diverse (Figure 3).

In advanced manufacturing, metallization is more than just "coverage." It must deliver consistent Cu-Cu-Cu interconnect and uniform initiation across mixed materials and glass types because defects here tend to be costly later on.

Familiar performance themes emerge:

- Throwing power and coverage in challenging features, including blind microvias
- Compatibility with advanced dielectrics used in demanding applications
- Deposition quality that supports new reliability testing standards

For advanced substrate applications, new electroless copper developments are to provide blister-free deposition on smooth build-up dielectrics and high throwing power into small, high aspect ratio blind microvias.

Relevant MKS' Atotech examples in this area include the Printoganth® MV TP3 family and the innovative tool solution G-Plate, which supports ultra-thin electroless copper and particle control in SAP build-up technology; Printoganth® P2 for strong throwing power into multilayer core blind microvias and defect-free interface layers for the advanced AI PCBs; and Cupraganth® MV Activator for Pd-free activation in advanced applications.

Electrolytic Copper Deposition: Fill Quality, Distribution, and the HVM Reality

In the context of electrolytic Cu plating, the most relevant question is not whether a process can plate a feature once but whether it can consistently produce uniform results in the features that matter in production. This includes aspects such as conformally coated copper on challenging designs, including high hole density and isolated areas, and reliable via filling, including for high aspect ratio blind microvias and increasingly through-holes.

To meet these high-volume manufacturing requirements, MKS' Atotech combines production-proven Inpulse® chemistries with Uniplate® equipment to provide highly controlled continuous processing, reduce process variation, and support stable yield. Dedicated processes include Inpulse® 2HFU2 for microvia reinforcement and Inpulse® 2HT for high-aspect-ratio flash plating.

These processes promote improved crystal growth across the base copper, electroless copper, and electroplated copper interfaces, helping to reduce the risk of crack initiation. For horizontal blind-microvia filling, Inpulse® 2HF9 supports panel-plating applications, while InPro® THF2 is designed for vertical pattern plating. For vertical pulse plating, the Cuprapulse® series provides high throwing power and uniform copper thickness distribution, supporting increasingly complex PCB and package-substrate designs.

Final Finishing: The Last Microns That Decide Assembly Yield

The final finish is where fabrication meets assembly, and where small variations can result in significant costs. The finish must protect copper from oxidation during storage while remaining solderable and bondable throughout the assembly process and ensure robustness of the bonds during the lifetime of the finished electronic device.

Mainstream finishes used for PCBs include ENIG, ENEPIG/EPAG, immersion tin, and OSP. In high-end manufacturing, emphasis is placed not only on selecting the "right" finish but also on productivity, robustness, and process control.

Industry standards and performance requirements for immersion-tin applications have become more stringent over the years. To meet the growing demands for lower ionic contamination, improved solderability, and greater compatibility with solder masks, the world-leading Stannatech® 2000 process has been further developed. Stannatech® 2100 is the latest standard solution for PCB applications, while Stannatech® IC is designed to deliver enhanced performance for IC-substrate applications.

Advanced PCBs, including optical module PCBs and IC substrates, rely on high-frequency dielectrics and are moving to ever-finer features. This requires processes that are developed and tailored to be compatible with newer ranges of dielectric materials and that ensure isolation between lines and pads by reducing excessive plating and reducing coating thicknesses.

ENIG, ENEPIG, and EPAG developments utilizing upgraded pre- and post-treatment processes combined with the Aurotech® NIC electroless nickel, PD-Core® electroless palladium, and Aurotech® G-Bond 3 mixed reaction gold processes ensure high-quality results matching the latest and upcoming industry requirements while ensuring high productivity through short cycle times, long bath lifetimes, and lower overall process cost through reduced precious metal consumption.

The Crossover With Advanced Packaging: When Copper Becomes a Device Enabler

As PCB and substrate technologies advance toward finer lines, smaller vias, higher layer counts, and greater interconnect density, the importance of wet processing to manufacturability and long-term reliability increases. Metallization and surface preparation directly impact conductor definition, microvia integrity, adhesion, and the stability of complex

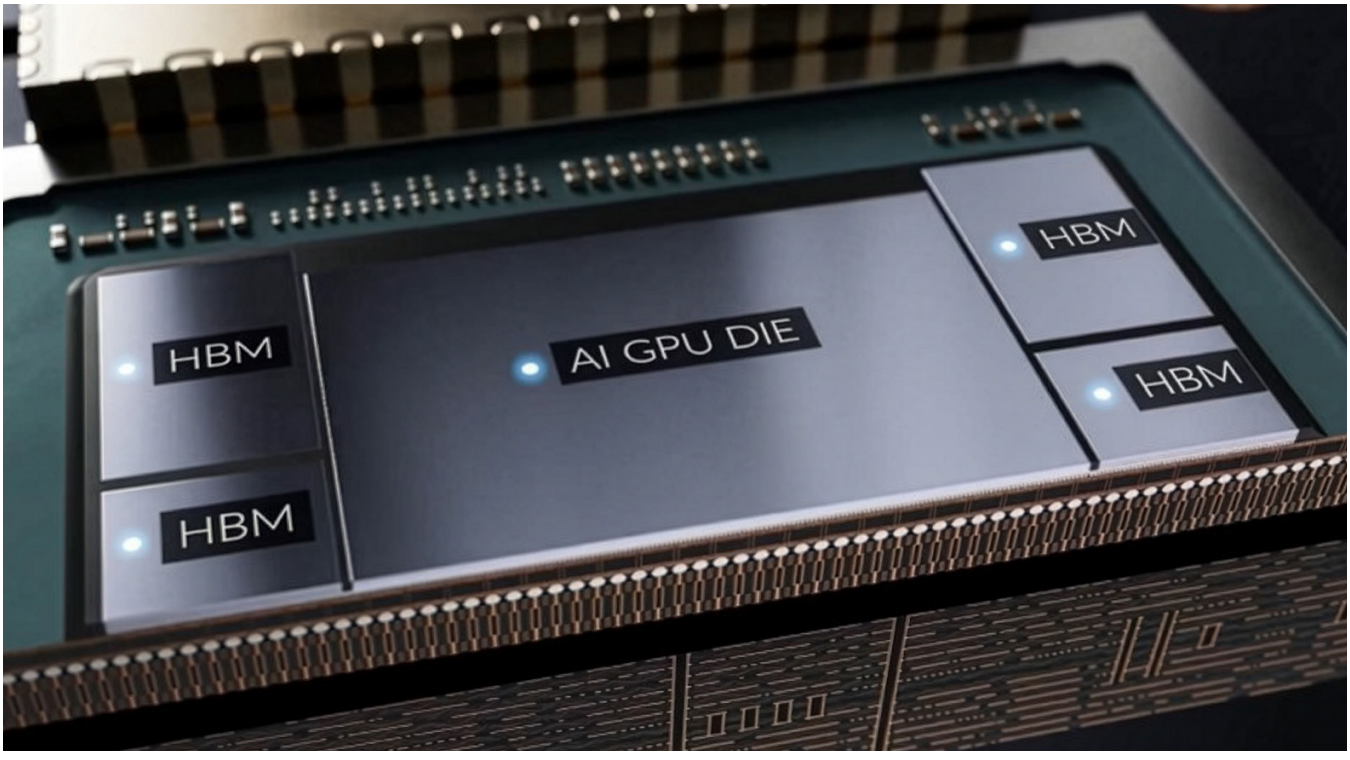


Figure 4: AI processors and high-bandwidth memory illustrate the growing interconnect complexity driving tighter requirements for copper deposition, surface preparation, and process consistency in advanced packaging.

interconnect structures. For PCB manufacturers, the influence of advanced packaging is more apparent here, as requirements once primarily associated with package substrates are increasingly relevant to high-end PCB and substrate-like PCB production.

This shift is reinforced by the convergence of process expectations in advanced substrates, substrate-like PCBs, and advanced packaging. Wet processes must now meet tighter requirements for deposit uniformity, impurity control, surface morphology, and interface consistency (Figure 4).

This explains why advanced chemistry and surface finishing have become more important in the manufacturing of next-generation electronics. Wet-process performance is being judged by interface readiness, which is the ability to provide the necessary cleanliness, roughness profile, metallurgical stability, and compatibility for the next process step. In many cases, the decisive factor is not the bulk deposit itself but rather the control and reproducibility of the top surface region. Future developments toward lower-roughness interfaces and ultra-thin copper will further increase the importance of this requirement.

At the same time, wet processing must be considered part of the entire manufacturing process. Its effectiveness hinges on close collaboration with laser drilling, desmear, activation, surface conditioning, and in-line process control. One of the clearest technology trends is the shift toward integrated process chains involving lasers, chemicals, and wet process equipment because interconnect capability is increasingly determined by how well these elements are matched in production rather than by the nominal performance of a single step in isolation. As interconnect architectures become more demanding, process capability is defined by the coordinated optimization of chemistry, equipment, and control strategy.

"One of the most important shifts we are seeing across the electronics industry is that performance can no longer be viewed as the result of a single breakthrough technology," Henry says. "Whether the application is AI infrastructure, high-performance computing, advanced packaging, or next-generation PCBs, success increasingly depends on how well

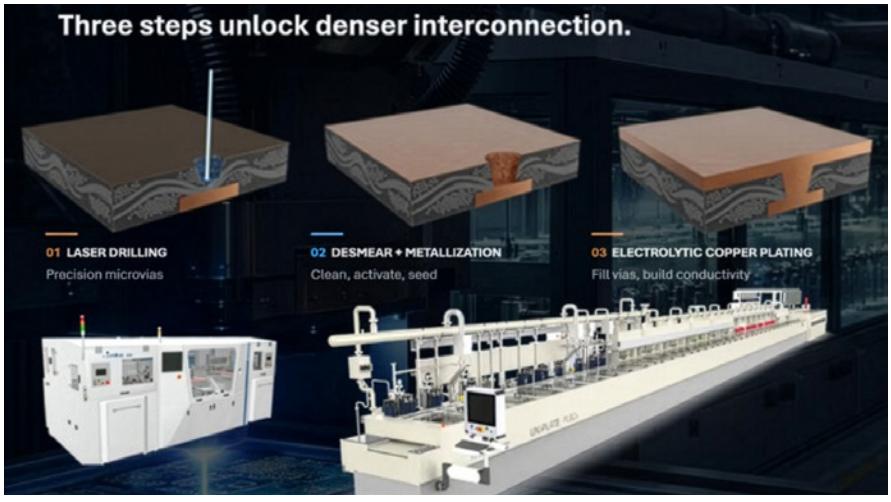


Figure 5: Laser drilling, desmear, and metallization, and electrolytic copper plating form a coordinated process chain for producing reliable, high-density interconnections.

multiple manufacturing processes work together.

"Surface preparation, metallization, copper deposition, finishing, laser processing, equipment performance, and process control all contribute to the final outcome," he continues. "As structures become smaller and tolerance windows narrower, even minor variations can have a measurable impact on yield, reliability, and long-term performance. The industry is moving toward a systems approach, where manufacturing capability is defined not by the strength of an individual process step, but by the ability to control and optimize the entire production flow as an integrated solution."

Why 'Systems Thinking' Is No Longer Optional

Process chemistry does not operate independently. In modern, high-volume production lines, outcomes are shaped by the combined behavior of chemistry, equipment, monitoring, and day-to-day operating procedures. This is why the industry is moving toward more integrated approaches. In practice, many yield losses originate from interactions such as contamination control, dosing stability, particle management, agitation uniformity, and drift detection, rather than from a single chemical step (Figure 5).

This manufacturing reality also explains why integrated systems approaches, combining chemistry, equipment, software, and services, are becoming more common. MKS' Atotech portfolio exemplifies this systems view with the following products: Horizon for surface treatment and finishing; Uniplate and Polygon platforms for horizontal desmear and metallization/plating; G-Plate for vertical curtain flow desmear and metallization for advanced substrates; vPlate for advanced vertical copper plating; and DFS (Digital Factory Suite) for condition monitoring, traceability, predictive maintenance, and energy optimization.

For PCB manufacturers, the key is repeatability. Miniaturization will continue, but maintaining stable process behavior across factories, materials, and product mixes while meeting the yield, throughput, and reliability requirements of advanced multilayer, HDI, substrate-like PCB, and package-substrate manufacturing will be key to competitive advantage.

In advanced electronics, performance limitations often originate much earlier in the process flow than where they finally become visible. Signal integrity issues, microvia failures, assembly defects, and reliability escapes can often be traced to variations in surface condition or copper quality introduced upstream. As AI and HPC systems become more complex, precisely controlling these foundational steps will be essential for manufacturing success.

I-CONNECT007

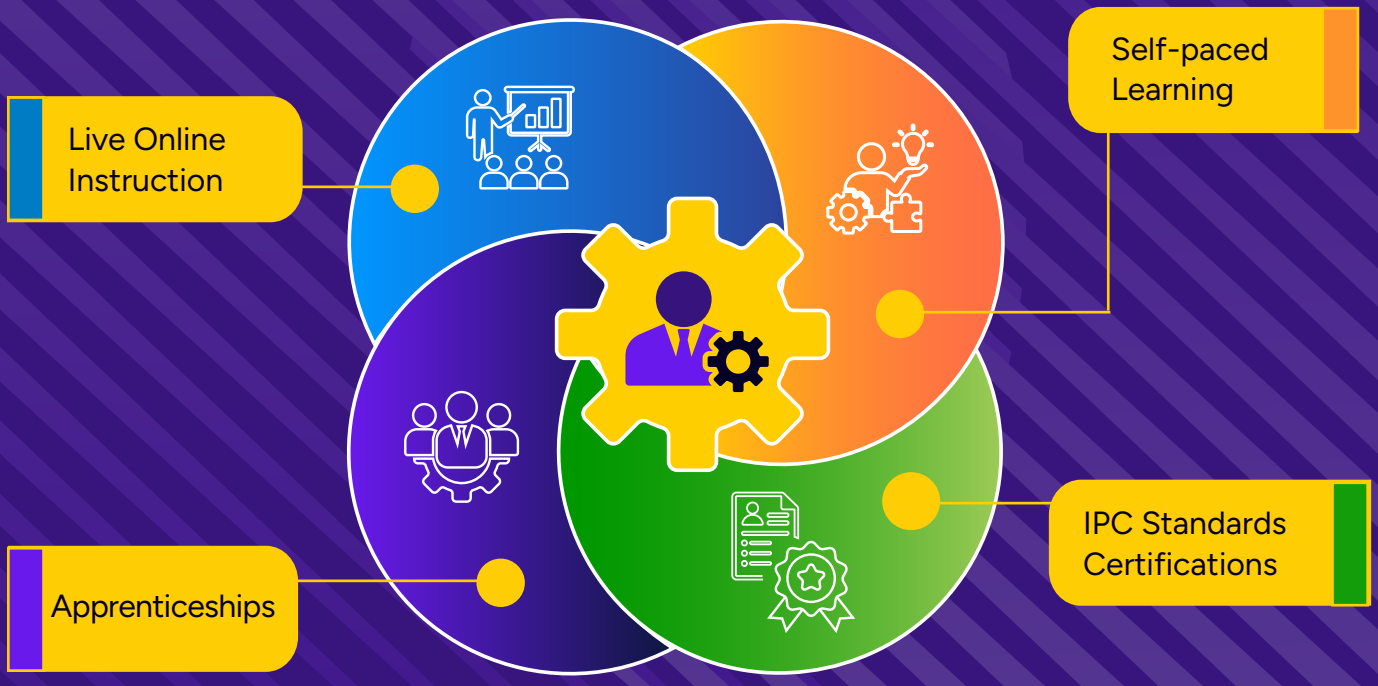


Harald Ahnert is SVP and GM Chemistry, Materials Solutions Division, MKS. He oversees the strategy, innovation, and business growth across the company's advanced chemistry portfolio. Having almost 30 years of experience in PCB, package substrate and semiconductor process technologies, he works closely with customers and the whole supply chain to address the industry's most complex technology challenges.

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

I Think I Need a FLEX CIRCUIT: Now What?



BY CHRIS CLARK, FLEXIBLE CIRCUIT TECHNOLOGIES

Over the past 35 years, I have been pulled into designs at all stages of their development. All too often, by the time a customer reaches out to us for a quote, they are pretty far down the design path. If they have done a good job, things generally work out, and we can either quote their design as-is or offer minor DFM feedback. There are, however, many instances where they have painted themselves into a corner.

The design, for example, may only be producible at a very low yield. At times the guidelines outlined in *IPC–2223, Sectional Design Standard for Flexible/Rigid Flexible Printed Boards*, have been completely ignored. Commonly, the available physical space is insufficient to meet the complex balance of mechanical and electrical needs imposed on the circuit.

Much of this could have been avoided if the customer had engaged with a circuit supplier early on, such as at the concept stage. Leverage the experts in the field, and they can coach you toward the best possible design. This will likely save you from multiple iterations.

YOUR PROCESS.
YOUR SPACE.
YOUR WET LINE.

OCCLEPPO®
WET PROCESS TECHNOLOGY

Built Around You.

PROCESS CONTROL

AUTOMATION

FACTORY LAYOUT

MES / ERP



Not a standard wet process line.
A manufacturing platform engineered around your factory.

CUSTOMIZED
Designed around your process.

FLEXIBLE
Adapted to your layout.

AUTOMATED
Industry 4.0 ready.

CONTROLLED
Every parameter monitored.

pluritec® GROUP
ENGINEERED FOR EXCELLENCE

Quick-turn facilities will generally build exactly the part you have designed. I have had many instances where customers said they built a round of prototypes with a quick-turn shop, and the parts were not working as they had planned. Ultimately, they did not save themselves any time, and it was no fault of the quick-turn house, which built exactly what it was provided with.

Here is an outline of the basic design steps that can help you avoid getting painted into that proverbial corner.

Define the Physical Shape of the Circuit

Start with a 3D model that defines the areas that need to be connected. If this is not an option, you can have a 3D-printed sample of your enclosure created. From there, you can try hand-cutting paper or Mylar to arrive at the desired outline of your circuit. This will be a great starting point. In some cases, paper dolls actually work better (I'm old-school).

Determine what termination options you plan to utilize. Factors to consider will be ease of installation, how rugged the connection will be (shock and vibration requirements), whether the option supports any current-carrying needs, and availability and cost. As for choices, anything that can be used in a rigid PCB can generally be incorporated into a flex design. This may include:

- Through-hole connectors
- SMT connectors
- ZIF connectors
- Crimp contacts
- Solder pads
- Wires
- Anisotropic adhesive that makes contact to pads on the flex

Determine your component needs and create a bill of materials. This will be required to determine whether you need a flex circuit (maybe with stiffeners) or a rigid-flex design.

A rule of thumb: If you need surface mount components on the top and bottom surfaces in the same region, you likely will need a rigid-flex design. This will also be required to achieve pad layouts and keep-out areas on the external layer(s) that need to be void of traces.

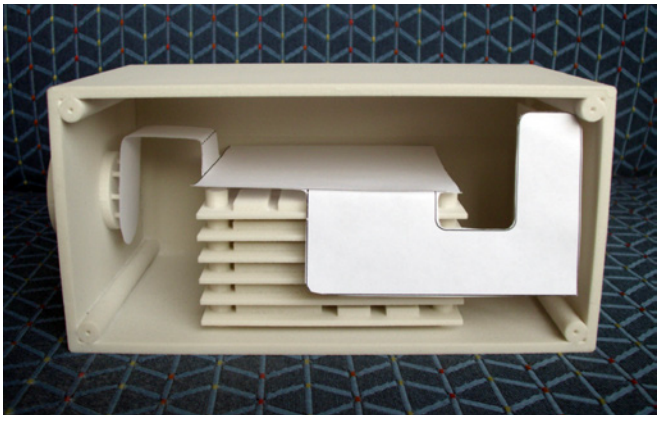


Figure 1: 3D printed model with a "paper doll" mockup to check fit.

Using the 3D CAD model, or your mockup, define the areas where bends need to occur. Keep in mind whether this is a static (bend-to-install) or a dynamic flexing application. This will be needed as you start to think about material selections, as well as where to place any required components.

Using your 3D model and all this additional information, flatten the circuit and create a paper doll to evaluate the fit within your device. If needed or desired, work with your supplier to get a circuit mock-up created using similar polyimide and copper thicknesses. Without the traces being etched and possibly plated, this will not be an exact representation but is a great starting point.

Create Your Schematic

This will be required for step 3. Depending on where nets need to terminate, you may need crossovers (vias) and the need for additional layers of circuitry.

Determine the Layers

How many layers will you need to achieve the desired number of traces, given the remaining available space (including terminations, vias, and components)?

Start with a very producible design; target 100-micron traces and spaces as a good starting point. These can be reduced, but it will come at a cost. Yields will be lower, and the smaller your features are, the more you may reduce your supply base. Not all suppliers can or will etch features below 75 microns.

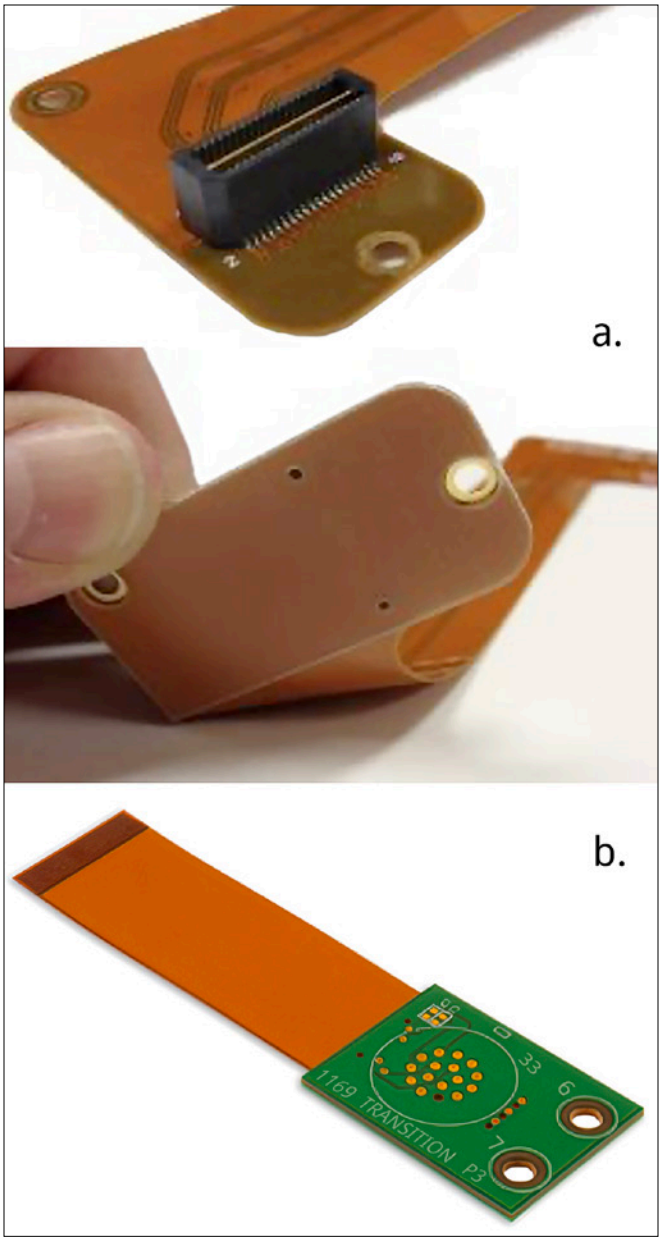


Figure 2: a) Stiffened flex b) Rigid-flex.

Start the Layout

Once you have these items defined, you are ready to start laying out the actual circuitry. This can be done most effectively with layout software, but it can also be done manually using various types of CAD software. If you are not comfortable with this step, most suppliers can offer this internally or through a third party.

Define the Stackup

Dynamic applications should be as thin as possible and utilize RA copper, but within reason. Some of the very thin polyimides (12-micron or thinner), copper below half an ounce, and adhesives under 25 microns do not lend themselves well to dynamic applications. A good starting point is one half-ounce copper, 1-mil polyimide dielectrics, and 1-mil adhesives. Avoid flexible solder masks in bend areas.

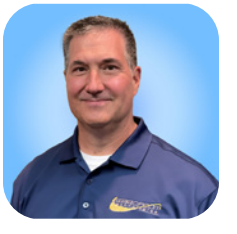
You will need to consider any electrical requirements, including how much current you need to carry on certain traces. This will determine the cross-sectional area of the conductors required to carry your current without overheating the trace. Impedance requirements usually come with a need for greater dielectric separation between signal and reference planes.

Often, your electrical requirements will push you in the opposite direction from where you would like to be to meet your mechanical needs (bend requirements). This is often a reiterative process to arrive at a finished design. At times, you will need to go back a step or two as you uncover issues.

Create Your Complete Procurement Package

- Use Gerber data.
- Prints including both assembly and the bare circuit level, as well as those required for custom components, should be generic where possible and detailed where needed.
- Do not over-dimension and apply overly critical tolerances. All too often customers apply tolerances similar to those found on a machined part. This is not realistic.
- Include a bill of materials.

The final step is to request a quote for your project. Obviously, there is more to each of these steps, but the important thing is to work with your supplier as much as possible along the way. Good suppliers should be more than willing to work with you on your designs. Everyone wins if the end product is as simple and producible as possible. **I-CONNECT007**



Chris Clark is a senior principal applications engineer at Flexible Circuit Technologies. To read past columns from Flexible Circuit Technologies, click here.

BY STEPHEN V. CHAVEZ, SIEMENS EDA AND PCEA



My Biggest Design Turkey: The BGA That Cost Me Everything

One engineering mistake can stay with you. Not because it's the only one you ever made, but because one particular mistake changes how you approach every design that follows. Mine happened roughly 20 years ago, when I was a lead PCB designer by day and running a PCB layout side business after hours. This mistake cost me a business opportunity, a promising startup, and a chance at what we all believed could be life-changing success. More importantly, it taught me a lesson about engineering accountability that I have never forgotten.

The Opportunity of a Lifetime

A small startup had approached me with an opportunity to help bring a highly complex network switch to market. Rather than take payment for my PCB layout services, I chose equity in the company. I became a shareholder and partner, joining a small group of people who believed we could build something special. Like many startups, we had scarce funding and limited resources, and we all contributed sweat equity while maintaining our day jobs.

Then came the opportunity we had been waiting for: a contract to develop a sophisticated network switch for data center applications. The schedule was brutally aggressive, with substantial technical challenges, and we had to deliver quickly.

I was responsible for the PCB layout. The design occupied a tight 3-by-9-inch envelope and included approximately 1,200 components, more than 2,000 connections, a 14-layer stackup, high-speed constraints, HDI technology, differential-pair routing, length tuning, and gigabit links connecting eight daughtercards and two network switches through a backplane processor. There was very little room for error.

To accelerate development, the team decided to outsource library creation to a third-party group known to one of my partners. I had reservations because I didn't know this team or its quality processes. I was outvoted, however, so I just had to trust my partners' judgment. That decision would

Flexible Circuit Technologies | inTFlex

The EMS Group of Flexible Circuit Technologies

FLEX SOLUTIONS
ENGINEERED FOR INNOVATION

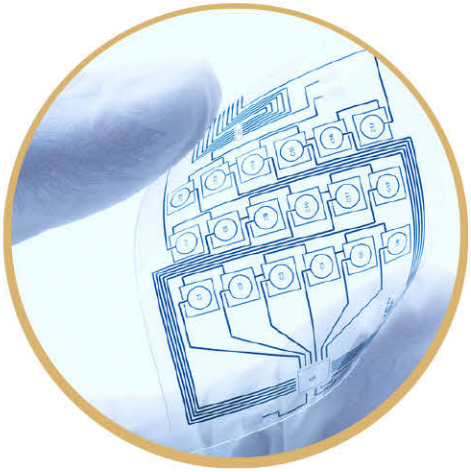
Industry-leading flex design services, premier in-house manufacturing of flexible circuits and EMS services



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

become part of a much larger failure, but this decision was not what ultimately sank us.

The Turkey Hiding in Plain Sight

During schematic development, the electrical engineer submitted a request for a library part representing the backplane processor, a BGA device that served as the brains of the entire assembly. The external library team created the part, and its quality checker supposedly verified it. The component entered the library, and the design moved forward.

I completed the PCB layout under tremendous schedule pressure. We conducted our electrical, mechanical, and layout reviews, and nothing obvious surfaced that would prevent us from releasing the design to fabrication. I performed a DFM review focused on fabrication, but I made a consequential decision: I skipped the assembly DFM analysis.

Why? Because we were nearly out of time and money. The delivery window was about to pass us by. I had designed plenty of complex boards, trusted my own abilities, and believed we had done enough to move forward. I was very confident in our workmanship. I also assumed the outsourced library team had properly completed its work.

Looking back, that was the critical mistake. I allowed confidence, schedule pressure, and assumptions about someone else's work to replace an independent verification step.

The manufacturing files went out. A few weeks later, the assembled circuit cards arrived. The photos looked fantastic, and excitement was running high. We believed we were close to landing the contract that could secure the company's future. Then the phone rang.

The Call I Will Never Forget

I was at my day job when my electrical engineering partner called. I expected good news from the initial testing I knew was underway that morning. Instead, I heard panic. During power-up, the backplane processor was getting dangerously hot, and the circuit card was not functioning as expected. He tried another assembly, then another. All five exhibited the same problem.

I found a private conference room and began working through the possibilities with him over the phone. I did not have my personal PCB layout workstation available, so I relied on him to review the design files while I opened and reviewed the processor manufacturer's datasheet.

> **"The very analysis I had skipped because we supposedly did not have enough time could have identified the problem before fabrication and assembly."**

First, I asked him to verify the assembly orientation. He checked, and the component was assembled correctly. That was when something clicked.

I asked him to open the actual Gerber data and locate pin one in the PCB land pattern. Then I asked him to compare that location with pin one on the physical BGA package while I checked the manufacturer's datasheet. As he described what he saw, and as I reviewed the BGA land pattern drawing in the datasheet, my stomach dropped: The land pattern had been created using the pin arrangement shown in the datasheet, but that drawing was explicitly labeled "VIEWED FROM BOTTOM SIDE." The library land pattern had been created incorrectly, effectively reversing the pin mapping.

I went silent for several seconds. My partner called out, "Steph? Steph? Are you there?" My response was not particularly professional, but it was certainly genuine. I realized we were not dealing with a fabrication defect or an assembly problem. We had a fundamental component library error that propagated through the entire design.

The processor was the heart of the system. Correcting the problem would require a complete board respin, including rerouting, reassessing the power distribution network, and revisiting component placement. With the schedule already exhausted and funding nearly depleted, there was no realistic path to recovery. The contract opportunity was gone.

The Discovery That Made It Worse

As I reflected on what could have been, I ran an

assembly DFM analysis using Valor on the design over a holiday break. Within seconds (yes, seconds), the tool flagged the land-pattern problem.

That result was devastating. The very analysis I had skipped because we supposedly did not have enough time could have identified the problem before fabrication and assembly. Instead, we had five nonfunctional assemblies and no time left to correct the design and meet the customer's deadline. Had we run it, we would have caught the error before fabrication. That would have given us a fighting chance to recover; it was a small chance, but a chance nonetheless.

The original error occurred during library creation, and the library team's quality check failed to catch it. But the mistake did not belong exclusively to the contractor. The incorrect land pattern passed through our design process, and I released the board without completing an assembly-focused verification that might have caught it.

I still cannot say whether the library team created the land pattern from scratch, downloaded an existing part, or performed an inadequate review. I never learned the full story. What I do know is that our process allowed one unverified detail to become a catastrophic system-level failure.

The startup eventually folded. My partners and I returned to our respective careers, and fortunately, we remain good friends. Every so often, we still joke about the retirement we might be enjoying if that project had succeeded—and the yacht we might have been sailing.

I keep one of those circuit card assemblies on my shelf today. It is a physical reminder of what could have been and of what I learned.

Why Every Designer Needs a Few Turkeys

Experience does not make you immune to mistakes. Sometimes, experience makes you more vulnerable because repeated success can create confidence that is not supported by sufficient verification. I knew better than to skip an analysis, yet I convinced myself that my experience, confidence, and our circumstances justified doing so. That is the turkey I own.

The lesson is not simply to double-check BGA pin assignments, although that is certainly part of it. It is to recognize that PCB design quality depends on a chain of verified decisions, from schematic capture and component-library creation through placement, routing, simulation, fabrication, assembly, and system validation. A failure at any link can compromise everything that follows.

Today, as PCB designs become denser, faster, and more complex, we cannot rely on visual inspection or individual experience alone. We need disciplined library verification, independent design reviews, appropriate electrical simulation, and both fabrication and assembly DFM analysis. Automated analysis and simulation tools are especially valuable because they can identify problems that human reviewers overlook. But those tools must be integrated into the workflow, not treated as optional extras when the schedule gets tight.

Most importantly, schedule pressure does not eliminate engineering risk. It merely changes how much risk we are willing to accept. When a critical verification step is skipped, the time saved may be insignificant compared with the cost of a failed build, a missed customer commitment, or a lost business.

I would never wish that failed experience on another designer. But I would encourage every engineer, especially those early in their careers, to be honest about their mistakes, take ownership, and learn from them. A design failure can be an expensive teacher, but only if we are willing to examine our own decisions rather than blame everyone else.

My biggest design turkey was not just a reversed BGA land pattern. It was believing that we could afford to skip verification because we were running out of time. Attention to detail is not a luxury, and verification and simulation are not overhead. They are essential parts of engineering.

If that circuit card on my shelf can remind even one designer to run the analysis, challenge an assumption, or take the extra time to verify a critical library part, then some good has come from our failure.

I-CONNECT007



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

BY MARCY LARONT, I-CONNECT007



Schmoll America Sees Unprecedented Equipment Demand

After a period of uncertainty surrounding tariffs and market conditions, demand for PCB manufacturing equipment has taken a significant turn for the better. For Schmoll America, the changes have brought unexpected growth and new challenges as manufacturers invest in both capacity and increasingly sophisticated capabilities. Kurt Palmer, president of Schmoll America, and Vice President Evan Howard discuss what they're seeing in the North American market, the forces behind the rising demand, and how the company is preparing to support its customers through the next several years.

Marcy LaRont: Kurt, when Schmoll America (SAM) established its North American operations in April 2024, the market looked quite different. What has changed over the past two years, particularly in terms of equipment demand and the factors driving it?

Kurt Palmer: When we opened our Chicago-area office in April 2024, we had 15 people supporting about 350 machines. By the end of 2025, the outlook was uncertain due to tariffs and general market instability. Then, in February, demand started to climb, and it hasn't slowed down since.

Two forces are driving it. First, AI: The surge in



PCBAA.ORG

PCBAA
Printed Circuit Board Association of America

AI-related work has much of Asia's PCB capacity allocated, and North American fabricators are investing to pick up the slack. Second, military and aerospace demand here at home keeps growing. Together, they've created machine demand unlike anything we've seen before, so much so that we're now forecasting all the way through the end of 2028. We expect our North American install base to reach 1,000 machines by then, and we're investing heavily to support it.


Kurt Palmer

Today we have 33 people on the team, and we're growing to 38 by mid-2027. That's more than double where we started, in just over two years. In my 35 years in this industry, I've never seen anything quite like it.

What kind of lead times can customers expect in the U.S. and how is Schmoll responding to that?

Palmer: Of course, everyone wants their machines as fast as they can get them, and lead times have stretched across the board. That's a challenge, but Schmoll Maschinen was already preparing for it. Well before the boom, the company started building a large, state-of-the-art facility near its headquarters in Rödermark, Germany, with the plan of consolidating four existing facilities into one. Today's demand has changed that plan. Schmoll will finish the new facility and keep the others running, because it needs the capacity.

Honestly, we were a little lucky. We didn't predict this volume, but because we were already building for growth, we're ready for it. From a lead-time perspective, I think that puts us a step ahead of most equipment manufacturers.

When will the new facility be finished?

Palmer: About one-third of it is complete, and that section alone has the capacity to build about 60 machines at one time. The rest of the facility will be finished by the end of 2027.

Kurt, what kind of support are U.S. customers asking for? What do they need, and are you seeing anything more prevalent in this market than in the past?

Palmer: We've always been laser-focused on customer feedback, and right now customers are telling us they need two things: capacity and capability. They need both, and they need them fast, because their own customers are demanding faster turns. So we're building machines as quickly as we can while continuing to add technology, from controlled-depth drilling to integrating multiple laser sources into a single machine. One trend really stands out: Our North American customers are buying some of the most complex machines that Schmoll manufactures.

Evan, let's talk about the service aspect of Schmoll America. How are you able to support this growing need in the North American market?

Evan Howard: As Kurt said, by the end of 2028, it's realistic that we'll have a North American install base of over 1,000 machines. We have grown from about 350 machines when SAM started in 2024 to about 650 today. Looking to grow beyond 1,000 North American installations over the next two years is significant.

That being said, our top priority is to support the machines already in the field. If we help the current customers, it will naturally create opportunities to add



more machines. Providing the highest level of technical support is a pillar of Schmoll's business, which continues to prove out in our growth numbers.

Kurt and I manage this at a higher level by supporting our technicians, keeping their workloads manageable, and keeping them healthy in ways we can control. We monitor the machine-to-technician ratio to keep that at a reasonable level of around 20 to 30 machines per technician.



Evan Howard

language with Schmoll.

It is a great collection of people, with some getting ready for retirement and some who are just kicking off their careers; all of them are good at different things. It's created a nice, well-rounded department with a lot of diversity. Though SAM is focused on North America, Schmoll is a global business. Our service team now represents more than 15 different countries. It's a really well-rounded group.

How is Schmoll America responding to the challenges of finding and securing a qualified workforce?

Howard: I started my career with Schmoll building machines in Germany. When I moved back to America in 2018, I was one of three service engineers supporting the North American market. There were a lot fewer machines then, so the ratio wasn't as crazy as it sounds. But North America started very thin.

When we began to staff up, we worked with many local universities using a program called Handshake, where we were able to post our jobs through their career centers. We didn't hire engineers all from one school or area. I think that is one of the reasons our service department has become really successful.

We've hired some people with not an ounce of PCB industry knowledge; they're just fresh out of school with an engineering degree and a curious mind. We put them in front of a machine, and they just fall in love with what this industry provides. We also have many experienced technicians who have been in the industry for decades. They've seen it all, inside and out, and now they're just learning a new

What do you feel is most important in how you support customers?

Howard: It starts with the culture of Schmoll Germany. It's not just about constantly fixing machines anymore. We see our visits as opportunities to identify areas where we can improve your process, support your training to make operators better, or assist your process engineers in learning more.

A company can hire somebody with a doctorate in engineering, and they will still likely come in knowing nothing about printed circuit boards. It's an industry that you may never see in an engineering course. So, we do a lot of work to bridge that knowledge gap, especially as people retire out of the industry and customers bring in new people. The training we do is key for our customers.

With such high expectations for growth in the next two years, where do you go from here?

Palmer: We're hiring for five new positions right now. We've created a new Technology and Quality department, which includes applications engineering. We're adding a training manager to develop and track





Julian Leon Degen



As an employee hired in 2025, Julian Leon Degen, marketing manager of the Schmoll "Group," shared his positive experience coming into the company and being trained firsthand.

"In 2024, I joined Schmoll Asia Pacific as an intern based in Taiwan and moved in 2025 back to Germany as a production intern to bridge the gap between my studies in international business and the technical knowledge I needed. Since then, I have worked in Asia and Germany and connected with colleagues from Schmoll America. Wherever I went, I felt part of the same family. We might have worn different badges on our T-shirts, but we all belonged.

"I remember standing on the production floor in Germany alongside colleagues from Asia and America and really feeling that sense of community. That is something Schmoll does exceptionally well.

"At Schmoll America, you may sometimes work independently—on your own little island—but you are empowered to take responsibility, solve problems, and show what you can do. And you always have support behind you.

"I saw that firsthand in Asia. When our technical engineers needed help at a customer's site, they called colleagues at the application center in Germany. Despite the time difference, they could reach the right people and get answers that helped them solve problems faster.

"As a new employee, that means so much. Wherever you are, whatever time it is, you know someone has your back. Even when you are standing by yourself in front of a customer, you never feel alone. You have a whole team behind you."

training requirements for our staff and help each person grow their skills, because we want everyone here to see a real career path with Schmoll.

We're adding a project manager. That person doesn't need to be a technician; the role is about coordinating our technicians and keeping Schmoll Germany and our service management team in sync. The goal is to grow our team ahead of the install base, so service never becomes the bottleneck for our customers.

What makes Schmoll America a great place for you both to work? What culture have you created here?

Howard: Schmoll Germany is managed like a family business. Stephan and Thomas Kunz have their kids working in the business. They're down on the factory floor designing and building these machines, and driving the R&D. You can't invest more in your company than your children.

Kurt and I foster that same type of culture here. It can be difficult when you're servicing a big area like North America. Our technicians work in high-pressure situations when they're in production facilities trying to bring machines up, and we don't want them to feel like they're on an island.

We run team-building events and put participants in situations where they will succeed. We make them feel supported, that they're not doing it alone. We want them to feel, "Although I might be the guy in front of the machine by myself, I've got 30 other service engineers in my pocket waiting to answer any questions I run into. They're just a phone call away." This is a company that values and takes care of its people.

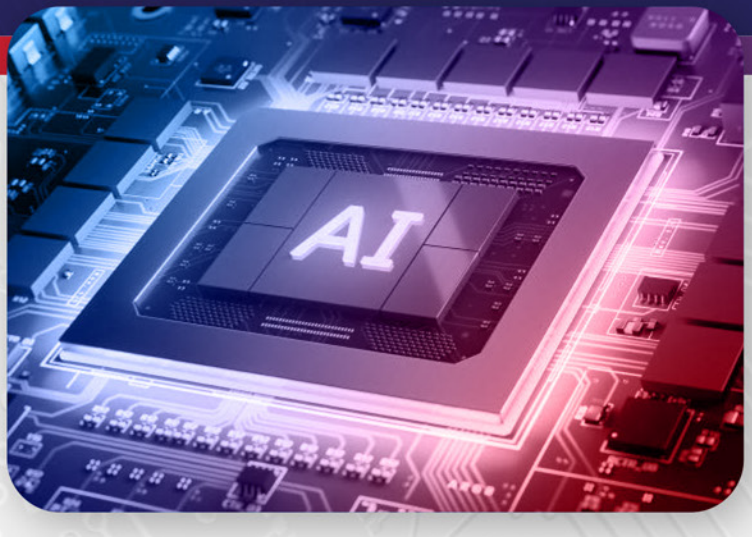
Palmer: I agree. The culture comes from the top, from Thomas and Stephan Kunz. Their customer focus is contagious, and it makes the work fun. As we're talking, it's 8:30 p.m. in Germany. If you wanted to test me and you said, "Let's add Stephan to this call," I could dial him right now, and I guarantee he'd pick up. The same goes for Thomas. That attitude carries throughout our entire organization.

I-CONNECT007

Kurt and Evan, it is always a pleasure. Congratulations on a successful two years and all the growth ahead.

MacDermid Alpha ELECTRONICS SOLUTIONS

Maximize Advanced Multilayer PCB Performance with PC 710

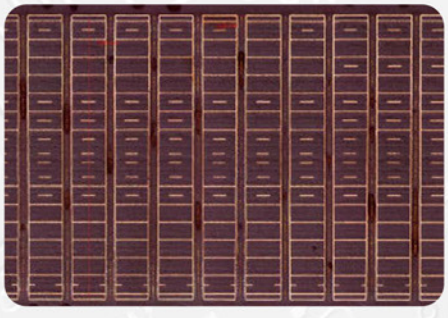


DISCOVER THE PERFORMANCE ADVANTAGES OF PC 710

As high-performance computing (HPC) and next-generation electronics drive thicker and more complex PCB architectures, PC 710 periodic pulse reverse plating technology delivers enhanced performance for demanding multilayer designs.

PC 710 - PERIODIC PULSE REVERSE PLATING TECHNOLOGY

- Superior throwing power for high-aspect-ratio through holes
- Exceptional thickness uniformity for complex board designs
- High performance without compromising deposit quality
- Long bath life and wide operating window
- Reliable, repeatable copper deposition



Reliable, consistent plating for ultra-high-aspect ratio through-holes

Reliable, Efficient Copper Plating for Advanced PCB Manufacturing

- **>8mm** — Exceptional Thickness Uniformity
- ✓ — Reliable Performance
- **High AR** — Superior Through-Hole Performance
- **HPC** — Optimized for AI Performance

macdermidalpha.com

© 2026 MacDermid, Inc. and its group of companies. All rights reserved. "(R)" and "TM" are registered trademarks or trademarks of MacDermid, Inc. and its group of companies in the United States and/or other countries.

My PCI Edge Card Connector Design 'Turkey'



BY KELLY DACK, CID, CID+, PIONEER CIRCUITS

In the 1990s, computer backplanes were undergoing a massive design evolution that challenged every layout rule in our playbooks. Dedicated peripheral PCB cards were plugging into desktop CPU backplanes to provide the necessary interfaces for printers, plotters, audio devices, and those screeching dial-up modems connecting us to the brave new world of the internet. With so many reasons for users to connect, motherboards were starving for real estate, utterly challenged to provide enough card slots to accommodate the exploding variety of add-on hardware.

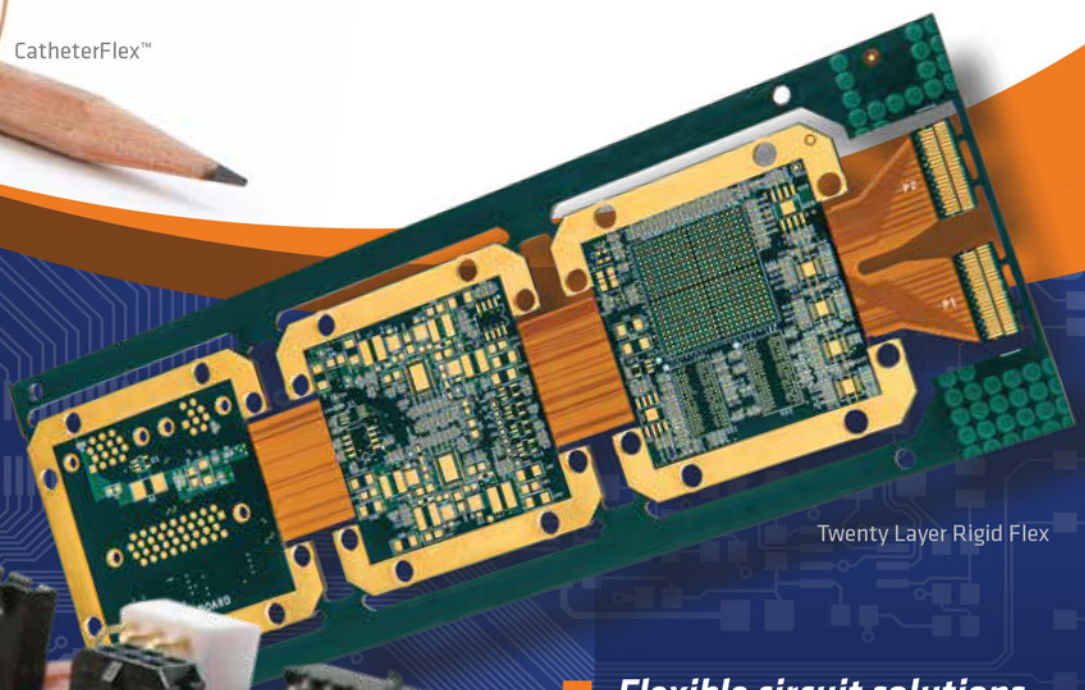
To make matters worse for those of us routing the boards, the industry was smack in the middle of a major form-factor transition. We were moving away from the legacy ISA edge card connector standard, with its comparatively "beefy" 0.100"-spaced contact "fingers," and shrinking to a newer, tighter PCI standard that cut the spacing in half to 0.050" pitch. It was an interconnect standard that feels grossly huge by today's sub-millimeter standards, but at the time, it helped a lot to gain more real estate.

That was right when I signed on for a PCB designer gig at an innovative telecom startup down in San Diego. Our mission was to help the industry survive

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

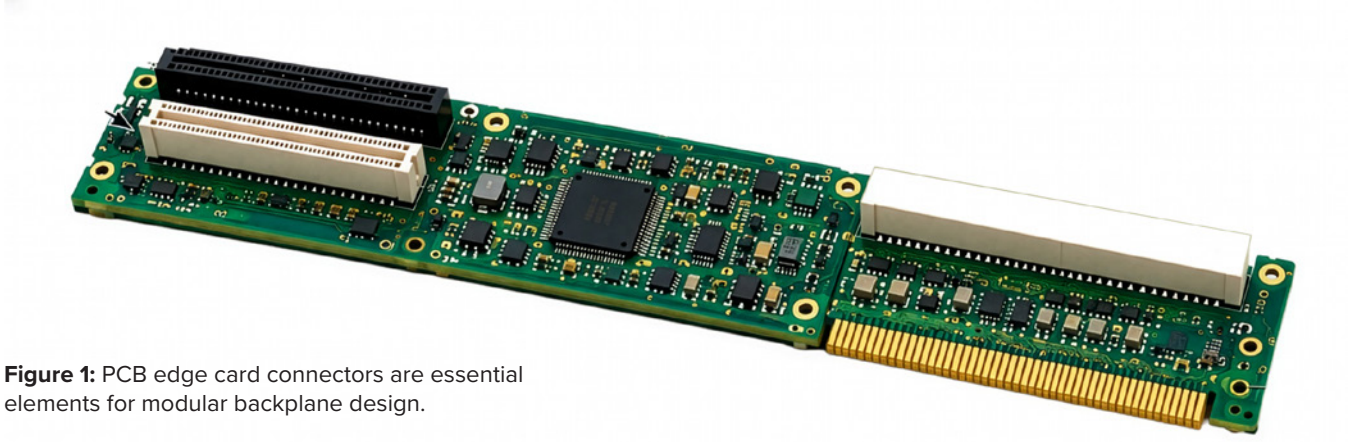


Figure 1: PCB edge card connectors are essential elements for modular backplane design.

this messy ISA-to-PCI transition by rethinking backplane architecture entirely. We set out to design a line of modular, snap-together backplanes. The goal was to give users the ultimate design agency: the ability to snap together modular card slots as needed by their system, configuring a custom backplane with the perfect ratio of standard edge connectors on demand.

But as every designer must learn (and believe me, I'm no exception), what looks simple on a conceptual schematic or block diagram can quickly devolve into a manufacturing and signal integrity nightmare the second you start laying down copper. Snapping physical PCBs together meant routing high-speed bus lines right across those mechanical seams. While the older, sluggish 8.33 MHz ISA bus was incredibly forgiving, I would learn the new 33 MHz PCI bus was a whole different animal. It exposed my lack of understanding regarding my role in solving for controlled impedance and consideration for PCB signal integrity.

Suddenly, those modular snap-together interfaces threatened to act like tiny antennas, inviting catastrophic signal reflections, impedance mismatches, clock skew, and severe crosstalk if our trace geometries weren't flawlessly executed.

I had heard about designing for controlled impedance but found it easier to pass the buck onto the PCB fabricator to solve it. I would soon find out what I needed to learn during this phase of my PCB design career. It culminated in an expensive, teachable moment when I learned that some PCB designs are steeped in electrical design concepts that are affected by basic mechanical design constraints.

The Turkey's Footprint

To create those new PCI edge card fingers, I first had to build a CAD library part from scratch. Today, designers are spoiled for choice with endless sources of pre-verified, accurate CAD footprint libraries. Back in the '90s, they were rare, pricey, and often forced us to roll our own, so to speak. But, as I was about to learn one fateful afternoon, nothing is more expensive than the layout error you inflict upon your own design when you don't know what you don't know.

In CAD terminology, an edge card connector is treated as a component footprint. But there's a massive catch: the male end of an edge card connector isn't a part you physically install. It is manufactured directly into the rigid edge of the PCB. The copper contacts are part of the board itself, completely devoid of a plastic connector body. According to the standard, these contacts are numbered alphanumerically per side: 1A, 2A, 3A on the primary side, with matching contacts on the opposite side identified as 1B, 2B, 3B. Simple enough on paper.

However, I learned a few brutal lessons after creating what I thought was "the perfect edge card footprint." No matter how flawless a symmetrical footprint shape looks on your screen, if the CAD tool allows it to be placed, rotated, or flipped without constraints, it becomes a ticking time bomb for a catastrophic physical connectivity error.

In my case, I had created only the raw artwork using a series of "padstacks" to form the copper fingers of the edge card pattern. Because there was no physical component body to install, I omitted a silkscreen component outline legend and skipped any graphic indication of pin numbering, Side A, or Side B. Fortunately, I was blessed with a decent amount of mechanical aptitude, so I did have the

foresight to include a mechanical alignment mark so the pattern could at least be positioned perfectly along the PCB's routing outline.

Hard Lessons Learned

Well, as any seasoned designer can imagine, what can go wrong usually does in PCB layout. I ran all my automated electrical checking routines, and they proudly flagged a flawless netlist with perfect connectivity to the top and bottom contacts. I confidently shipped the CAM data off to the prototype shop along with a basic stackup detail. Then, as was my naive custom at the time, I simply asked them to adjust the dielectric layers on their end to achieve my specified impedance targets.

Lesson #1: Give Your Board Fabricator a Fighting Chance of Hitting Your Impedance

Originally, I had chosen my trace widths based purely on ideal print-and-etch DFM constraints. I foolishly left the other two critical variables of impedance control—distance to the reference plane and the material's dielectric constant—entirely up to the fabricator. I learned just how irresponsible that practice was when the CAM engineer called me back. He patiently explained that to hit my impedance targets with traces that wide, the finished PCB would need to be 0.100" thick. That was a massive problem, considering the edge card standard strictly mandated a standard overall PCB thickness of 0.062".

That was the exact moment I realized I couldn't just pass the buck. I needed to start utilizing at least a basic impedance calculator to choose line widths that actually complemented the required laminate material thicknesses. Back then, field solver options were incredibly sparse on the internet. Today, there is absolutely no excuse. Most, if not all, modern PCB design software suites have advanced stackup solvers built right into the design environment. It is every PCB designer's job to get the impedance math close to a solution so the board fabricator can simply dial it in from there.

I re-routed the board with thinner trace widths and finally brought the overall thickness down to 0.062". I shelled out the extra expedite fees to make up for the lost time. When the fresh prototypes arrived, I stood right there over the shoulders of our soldering techs, watching them assemble the connectors and components onto the board. Everything looked great. On to the bring-up phase.

Next, the engineer I had designed the board for began probing some of the assembled board's contacts. After some time, he looked at me, and in a patient, empathetic voice that I will never forget, said, "Sheesh, Kelly, I think you've got the A and B sides of this connector flipped!"

He was right. I did.

Lesson #2: Build Electronic Component Footprints, Not Turkey Footprints

The error didn't happen because I accidentally rotated the pattern or reversed the pin order. It happened because the A-side padstacks were placed directly on top of the B-side padstacks in the CAD library part, making manual visual verification incredibly difficult without proper silkscreen legend or etched alpha side indicators on the outer layers. The pattern was flipped, causing top side routes to connect to the bottom side and vice versa.

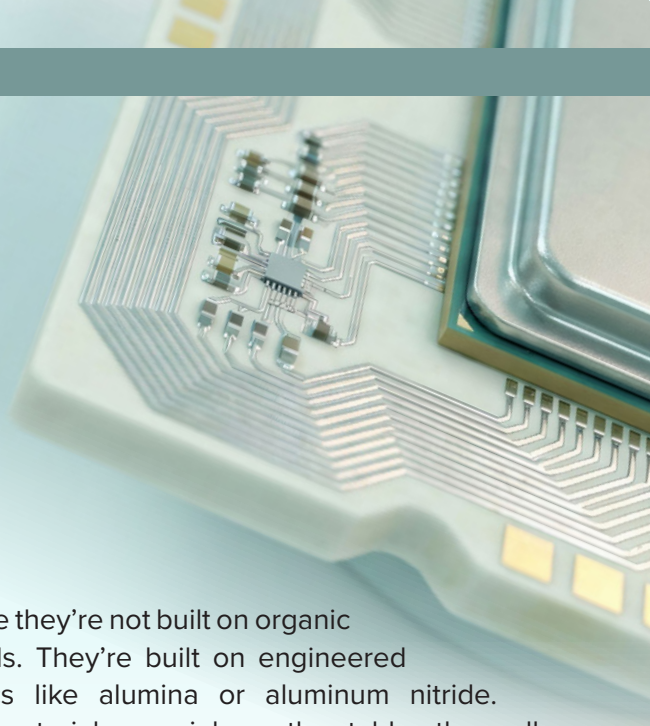
Fortunately, the engineering group I worked with had seen this kind of rookie mistake before, but they weren't going to let me off the hook without turning the event into a permanent teachable moment. They handed me one of the bare prototype boards, a Dremel tool, a soldering station, a spool of blue rework wire, and a female edge card connector.

I spent the next two days of my life hunched over that bench, tediously hacking and jumping traces to create a hideous "Frankenstein" connector-converter just so we could test the electrical functionality of the rest of the PCB assembly. It was a true design turkey born of my own making, but it was an opportunity to learn a couple of lessons that would completely transform how I approached impedance control specification and CAD library discipline for the next 30 years. Gobble, gobble. **I-CONNECT007**



Kelly Dack, CIT CID+, specializes in DFx-driven PCB design and applications engineering at Pioneer Circuits, Inc. To read past columns, click here.

The Real Ceramic Circuits



BY BRIAN BUYEA, REMTEC

There's a shift happening in our industry. It's not as loud as the move to electrification or as headline-grabbing as AI. But make no mistake, it's just as important. It's the shift away from traditional PCBs toward true ceramic-based circuitry.

If you're still thinking of ceramics as just another material option inside a PCB stackup, you're already behind, because what we're really talking about isn't a better board, but a completely different way to build electronics.

It Starts With a Misunderstanding

For decades, the industry has tried to improve performance by modifying the same basic platform of laminate-based PCBs: Add ceramic fillers, improve resins, reinforce substrates, and push the limits.

That's worked to a point, but at some stage, incremental improvement hits a wall. You can only make a material designed for general-purpose use perform so well in extreme environments until confusion begins. People see "ceramic-filled laminates" and think they're getting the benefits of ceramics, but they're not. They're getting a compromise.

True Ceramic Circuits Are Not PCBs

A ceramic circuit is not a PCB with ceramic in it. It is an entirely different platform mechanically, thermally, and electrically.

Traditional PCBs are built on organic substrates, typically glass-reinforced epoxy. These materials are versatile and cost-effective, but they come with inherent limitations. They expand, degrade under heat, and struggle with high power densities. They introduce signal losses at higher frequencies.

Ceramic circuits don't have those problems because they're not built on organic materials. They're built on engineered ceramics like alumina or aluminum nitride. These materials are inherently stable, thermally conductive, and well-suited for high-performance environments. This isn't an upgrade. It's a leap.

Heat Changes Everything

If you want to understand why ceramics matter, start with heat, the silent killer of electronics. It degrades materials, weakens interconnects, and shortens product lifespans. As power densities increase, especially in EVs, power modules, RF amplifiers, and defense systems, the problem only gets worse.

Traditional PCBs manage heat. Ceramics eliminate it as a bottleneck. Aluminum nitride, for example, has thermal conductivity that dwarfs anything you'll find in a laminate system. That means heat moves away from critical components faster, more efficiently, and more predictably.

When heat is controlled, everything improves: Reliability goes up, performance stabilizes, and lifecycles extend. That's not a small advantage. It's a design enabler.

Stability Is the Real Story

But heat is only part of it. The real story is stability. Ceramic materials don't behave like laminates. They don't absorb moisture, warp, or expand unpredictably under thermal stress. Their CTE is much closer to that of semiconductor materials.

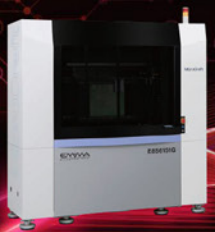
That means fewer mechanical mismatches, failures, and surprises. If you've ever dealt with field failures in high-reliability applications, you know that those "fewer surprises" are everything.

MicroCraft's Product Range
Built for What's Next

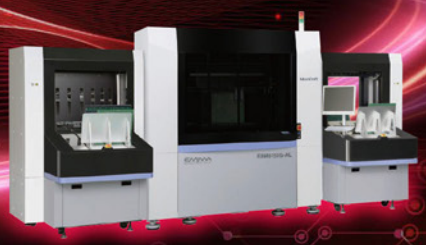
EMMA
Moving Probe Tester

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

Conductivity Without Compromise

One of the biggest misconceptions about ceramics is that they somehow limit electrical performance. That might have been true decades ago, but it's not true anymore.

Technologies like plated copper thick film have changed the game. They allow ceramic substrates to carry high current, support fine features, and maintain robust interconnections, all while preserving the inherent advantages of the ceramic base.

What you get is a platform that combines the best of both worlds: the electrical performance of copper with the thermal and mechanical stability of ceramics. That's a breakthrough.

Integration Reduces Risk

Another key difference between ceramic circuits and traditional PCBs lies in how they are integrated into the final product. With laminate-based systems, you're often dealing with multiple steps, multiple suppliers, and multiple opportunities for failure. Substrate fabrication, assembly, and packaging are frequently disconnected.

Ceramic solutions allow for a more integrated approach. Substrate and assembly can be designed, manufactured, and optimized together as a single system. That reduces interfaces, and every interface you eliminate is one less place for something to go wrong. In high-power, high-reliability applications, that matters more than most people realize.

Designed for the Real World

Where do these differences matter?

- Electric vehicles, where power modules must handle extreme thermal cycling without failure
- Defense systems, where reliability is a mandate
- Industrial drives, where uptime directly impacts profitability
- RF and microwave systems, where signal integrity and thermal stability are critical

These are not environments where "good enough" works, but where failure is expensive or unacceptable, and that's exactly where ceramic circuits shine.

A Shift in Thinking

The biggest challenge isn't the technology; it's the mindset. Engineers have been trained to think in terms of PCBs: It's the default, it's familiar, and it's what they know.

But the next generation of electronics is pushing beyond what traditional materials can handle: higher power, higher frequencies, smaller form factors, and greater reliability demands.

At some point, you have to stop asking, "How do we make a PCB work?" and start asking, "What's the right platform for this application?" More often, the answer is ceramics.

When Should You Use Ceramic?

You should use ceramic when performance, reliability, and thermal management are critical to success. If your design is pushing limits—thermal, electrical, or mechanical—ceramic is a necessity. If your application can't afford failure, ceramic isn't an option; it's the solution.

Powering the Future

At the end of the day, this isn't about replacing PCBs. They still have their place, and they always will. This is about recognizing that not all electronics are created equal, and not all platforms should be treated the same.

Ceramic circuitry represents a distinct class of solution designed for environments where performance isn't negotiable, reliability isn't optional, and failure isn't acceptable. That's where the future is being built.

Companies that understand this shift are the ones willing to rethink their assumptions and embrace new platforms. They are the ones that will lead. The rest will keep trying to push yesterday's technology just a little bit further.

In the end, the question isn't whether ceramic circuits are different, but whether your application can afford to ignore that difference. **I-CONNECT007**



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

Through-hole Interconnection Using the Partition Method:

Structural Design, Manufacturing, and Applications

BY HUANG MINGAN, XIANG JINPENG, AND HUANG XIAO, FSQUALITY, AND YE FEIHUA, ZHAOQING UNIVERSITY

Editor's Note: *This technical paper is presented in two parts. Part 1 introduces the partition method for through-hole interconnection, explaining its structural design, manufacturing process, and applications in thick-copper PCBs, high-density BGA fan-out, and outer-layer routing. Part 2, appearing in the next issue, examines reliability testing, compares the technology with conventional manufacturing methods, and explores its potential applications.*

Modern electronic products, driven by the rapid development of 5G communications, AI, automotive electronics, and other emerging technologies, demand both high-speed signal transmission and high-current-carrying capability. Servers, communication base stations, and similar equipment require high density PCB interconnections to ensure signal transmission efficiency, while power amplifiers, high-voltage modules, and other power electronic devices rely on thick-copper structures to support large current loads and efficient heat dissipation.

Under these demanding requirements, conventional PCB manufacturing technologies are facing several critical limitations. HDI technologies such as blind vias, buried vias, and sequential build-up, require multiple lamination and drilling cycles, resulting in complex manufacturing processes and degraded signal integrity. Thick-copper PCBs (typically ≥4 ounces, where 1 ounce ≈ 35 µm) suffer from severe lateral etching and are prone to lamination voids. Meanwhile, the pin pitch of high-density BGA packages has been reduced to as small as 0.30 mm, leaving extremely limited routing space for fan-out design. Furthermore, conventional through-hole structures occupy valuable routing and component placement area on the outer layers of the PCB.

To overcome these challenges, the development of through-hole any-layer interconnection technology is aimed at achieving four primary objectives:

1. To overcome the limitation of conventional plated through-holes that carry only a single electrical signal by enabling multiple electrically independent signal paths within a single through-hole and allowing flexible interconnection between arbitrary PCB layers.
2. To provide a manufacturing solution that simultaneously supports thick-copper applications and high density interconnection, satisfying both high-current carrying capability and fine-line routing requirements.
3. To optimize fan-out design for high density BGA packages by expanding routing flexibility while improving signal integrity.
4. To release valuable routing space on the outer layers through partition technology, thereby supporting the continued miniaturization of electronic products.

The proposed process is founded on the concept of "achieving multiple performance objectives through a single manufacturing process." It represents an integrated, simplified, and highly reliable approach to the next generation of PCB manufacturing technology.

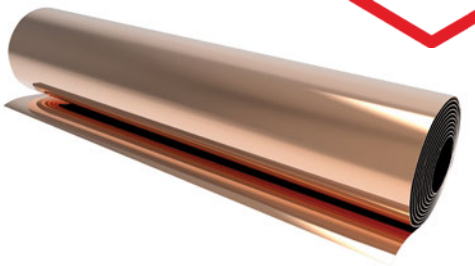
INSULECTRO

WORLD-CLASS SUPPLIERS

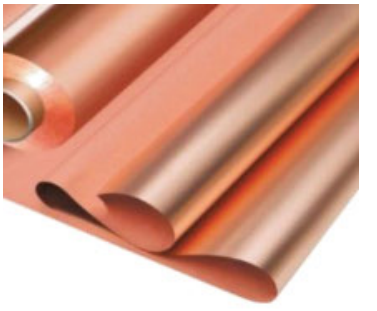
Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, salutes our premier suppliers.

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

Structural Design and Manufacturing Method

Core Structural Design

The proposed technology is based on the concept of enabling multiple independent electrical connections within a single plated through-hole. The key structural innovation lies in partitioning the conductive region inside the plated hole by introducing recessed isolation rings.

Recessed isolation rings are formed at both ends of the interlayer dielectric isolation zone through selective etching of the inner-layer isolation pads. The etching depth is controlled within 0.10–0.25 mm, while the isolation ring height ranges from 5–50 µm with an aspect ratio greater than 5, ensuring reliable electrical isolation.

Within the same plated through-hole, the copper wall between adjacent isolation zones remains electrically connected to the corresponding inner-layer circuitry and outer layers. By selectively arranging these isolation structures, electrical interconnection can be established between any desired PCB layers while maintaining complete electrical isolation between different signal networks.

The structural concept is illustrated in Figure 1, where the recessed isolation rings, plated copper wall, and interlayer dielectric isolation zones together create multiple independent signal transmission paths within a single through hole.

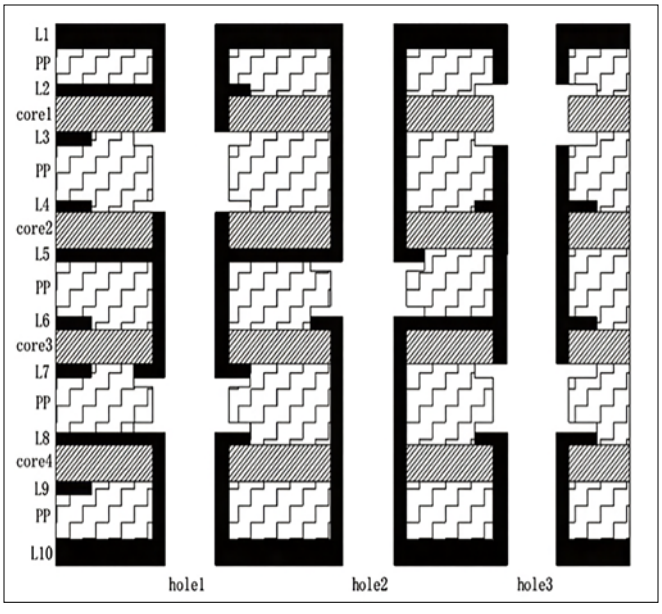


Figure 1: Structure of through-hole any-layer interconnection.

Manufacturing Process

The proposed through-hole any-layer interconnection technology requires only one lamination cycle, one drilling operation, and one electroplating cycle. The manufacturing process consists of the following key steps.

1. Inner-Layer Fabrication and Lamination

After completion of conventional inner-layer fabrication and automated optical inspection (AOI), an alkali-soluble ultraviolet (UV)-curable ink is selectively deposited onto the pad surfaces of the inner-layer core using inkjet printing technology.

Following UV curing, the printed ink forms the isolation pads. Standard inner-layer oxidation treatment and multilayer lamination are then carried out according to the PCB stackup design.

2. Drilling and Recessed Isolation Ring Formation

Through-holes are mechanically drilled at the designated locations to expose the embedded isolation pads.

The laminated PCB is subsequently immersed in a 10 wt.% sodium hydroxide (NaOH) solution maintained at 60°C for 30–120 minutes. During this process, the alkali-soluble isolation material is chemically dissolved, leaving recessed isolation rings with a tapered profile inside the plated through-holes.

3. Electroless Copper Deposition and Electroplating

A thin conductive copper layer is first deposited by electroless copper plating, followed by DC flash plating at a current density of 0.8–1.5 ASD for 3–6 minutes to establish an initial copper coating on the hole walls.

Reverse pulse electroplating is then employed using the following parameters:

- Forward current density: 3 ASD
- Forward pulse duration: 10 ms
- Reverse current density: 9 ASD
- Reverse pulse duration: 10 ms

This process selectively removes copper from the isolation regions while maintaining continuous copper deposition in the intended conductive areas,

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.

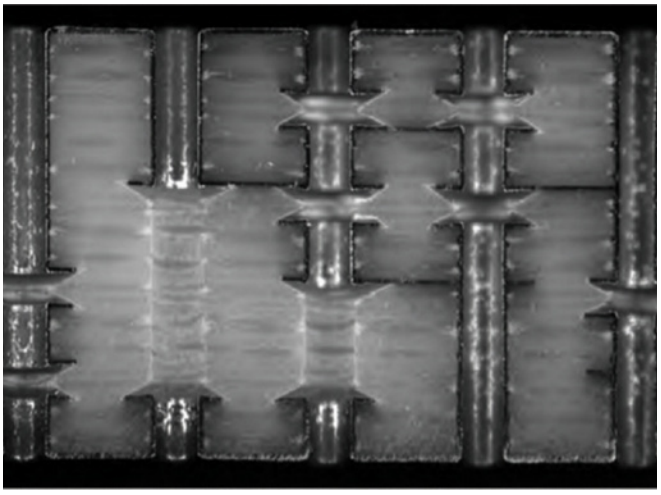


Figure 2: Cross-section of through-hole any-layer interconnection.

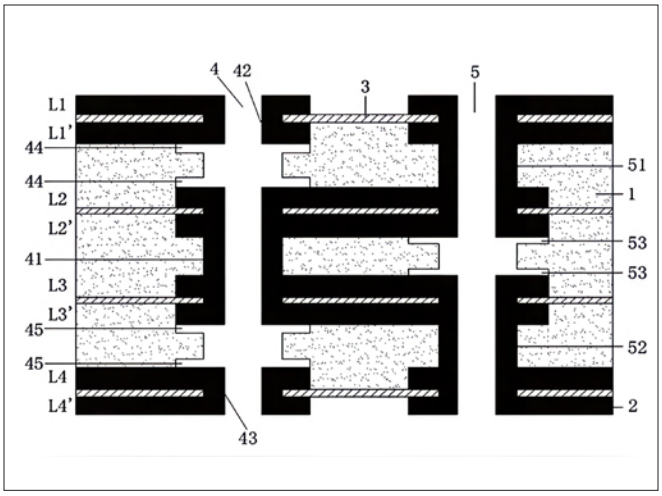


Figure 3: Thick-copper PCB structure.

thereby producing precisely partitioned conductive paths within a single through-hole.

Finally, conventional DC electroplating or a seconwdary pulse-plating process is performed to increase the final copper thickness, ensuring both electrical conductivity and long-term structural reliability.

Process Development and Optimization

The manufacturing process was optimized through several rounds of experimental validation.

During the initial trial, the isolation zones partially overlapped with the inner-layer pads, resulting in incomplete electrical partitioning and preventing independent signal transmission.

The second trial successfully achieved electrical partitioning; however, excessive copper extension occurred around the recessed isolation regions, leading to localized plating defects.

By optimizing both the dimensions of the isolation pads and the electroplating parameters during the third development cycle, stable transmission of multiple independent signal networks through a single plated through-hole was achieved, thereby realizing the intended through-hole any-layer interconnection concept.

Representative cross-sectional micrographs of the optimized structure are shown in Figure 2.

Core Application Solutions Based on Through-hole Any-layer Interconnection

Building upon the through-hole any-layer interconnection technology described above, several application-specific solutions have been developed to address three major challenges in modern PCB manufacturing: thick-copper adaptation, high density BGA fan-out optimization, and efficient utilization of outer-layer routing space. Together, these solutions establish a comprehensive technology platform that combines a fundamental interconnection process with application-oriented optimization.

Thick-copper PCB Solution

Thick-copper PCBs are widely used in high-current applications such as automotive power electronics, industrial control systems, and power conversion equipment. However, conventional manufacturing processes often suffer from poor etching accuracy, excessive undercut, and lamination voids when copper thickness reaches 4 ounces or greater.

The proposed through-hole any-layer interconnection technology addresses these issues through an innovative “copper layer segmentation + through-hole interconnection” architecture.

1. Structural Design

Instead of employing a single ultra-thick copper layer, the proposed design divides a copper layer of 4 ounces or more into two overlapping copper layers of 2 ounces or greater, which are electrically connected through plated through-holes.

From an electrical perspective, the current path is equivalent to two conductors connected in parallel,

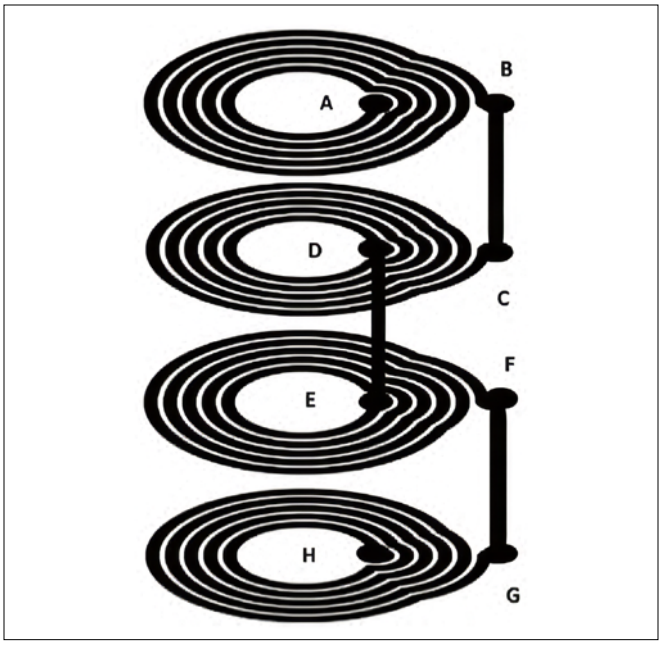


Figure 4: Coil PCB routing design.

thereby maintaining the required current-carrying capability while significantly reducing manufacturing difficulty.

The PCB structure incorporates at least four thick-copper layers separated by dielectric materials, allowing flexible interconnection between arbitrary layers through the proposed through-hole architecture.

2. Coil PCB Routing Design

For coil-board applications, the four thick-copper layers are connected sequentially in clockwise or counterclockwise directions to maintain a consistent magnetic flux orientation throughout the entire conductor path. This routing strategy significantly reduces occupied board area while preserving electromagnetic performance. Representative routing layouts and cross-sectional structures are illustrated in Figure 4 and Figure 5, respectively.

High-density BGA Fan-Out Optimization

As BGA packages move toward finer pitches, routing density has become one of the primary limitations in PCB design. To address the routing challenges associated with 0.30 mm pitch BGA packages, the proposed technology introduces a direct through-hole interconnection with ordered inner-layer fan-out strategy.

Unlike conventional sequential build-up technology, this approach eliminates the space occupied by blind-via capture pads while reducing signal discontinuities introduced by multiple interconnection transitions.

The design methodology includes the following features.

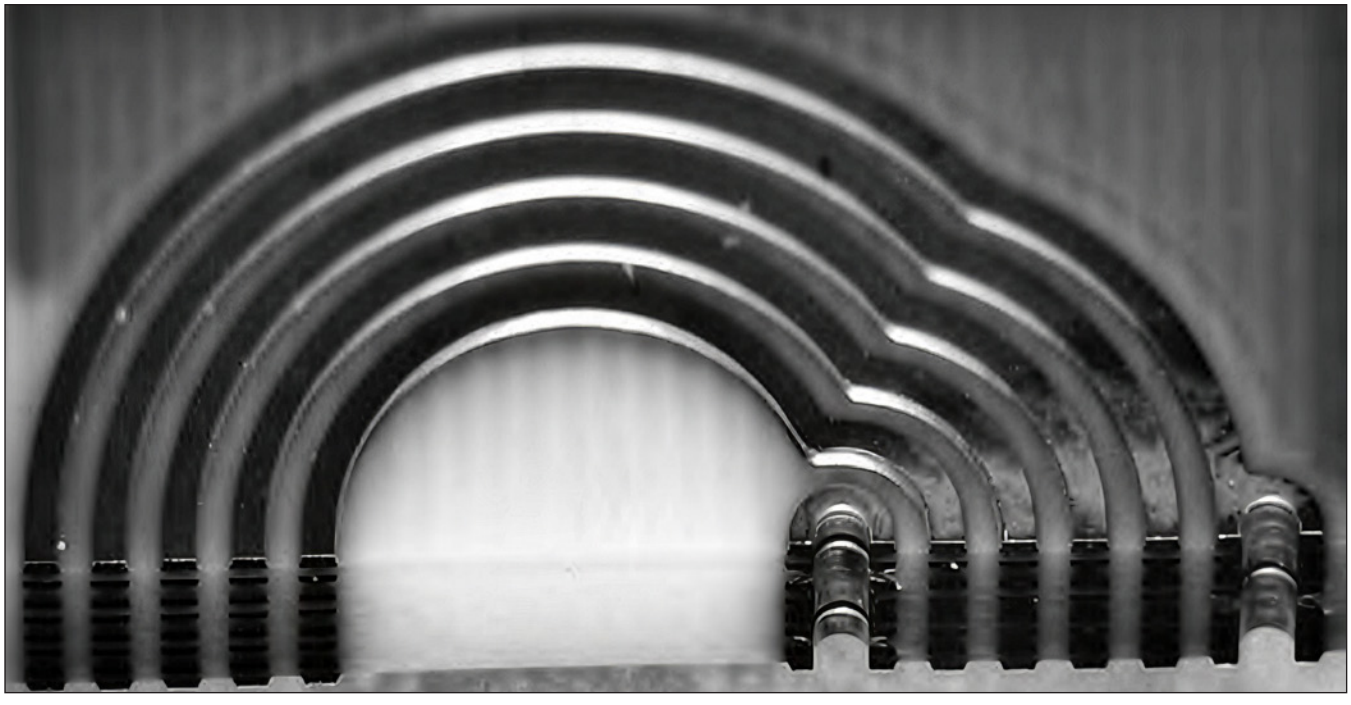


Figure 5: Cross-section of coil PCB.

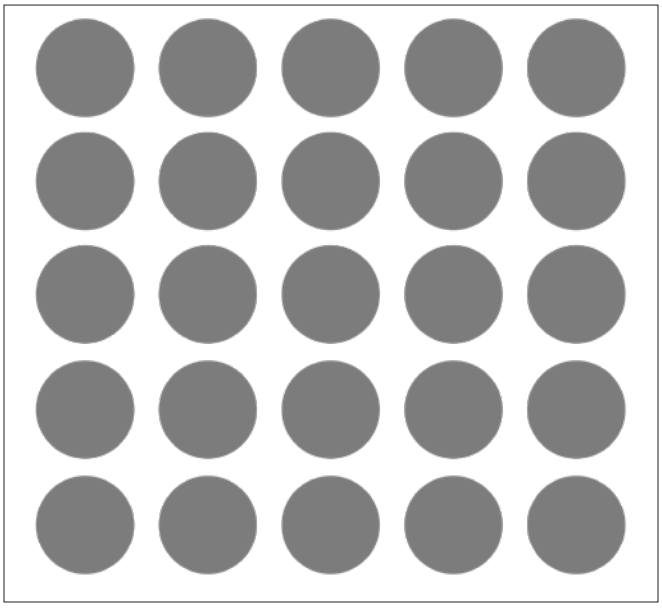


Figure 6: Top-layer BGA pad design.



Figure 7: Cross-section of BGA fan-out routing.

1. Top-layer Pad Design

The top-layer BGA pads are designed with a diameter of 0.25 mm, while pad spacing is reduced to 50 µm.

No fan-out traces are routed on the top layer. Instead, signals are transferred directly into the designated inner routing layer through plated through-holes.

2. Inner-layer Fan-Out Design

The inner-layer fan-out uses 150 µm-diameter vias while maintaining a minimum copper-to-copper spacing of 150 µm. This configuration provides sufficient routing space for 50 µm trace widths between adjacent vias.

The principal design parameters are summarized as follows:

- Via pitch: 300 µm
- Trace-to-via clearance: 50 µm
- Pad-to-pad spacing: 50 µm

Capture pads are provided only on the top layer and the designated fan-out layer. Additional vias are progressively fanned outward, reserving routing channels for signals originating from the inner rows of the BGA package.

Compared with conventional HDI structures, this design provides shorter signal paths, improved routing efficiency, and enhanced signal integrity.

Outer-layer Partitioning and Routing Space Optimization

Based on the through-hole any-layer interconnection process, two outer-layer partitioning techniques have been developed to eliminate unnecessary through-hole occupation on the PCB surface and recover valuable routing area.

1. Solder Mask Filling Method

After completion of through-hole electroplating, the vias are vacuum-filled with resin and planarized.

During subsequent outer-layer circuit fabrication and solder mask processing, the exposed copper around the partitioned via is naturally removed through the normal etching process and replaced with solder mask material.

As a result, the outer-layer portion of the via becomes electrically isolated while routing traces can be placed adjacent to the partitioned region.

2. Plated-Over-Filled-Via (POFV) Method

Following resin filling and planarization, windows are selectively opened in the outer copper layer above the partitioned vias to interrupt the electrical connection between the outer layer and the internal conductive structure. The vias are subsequently filled again, planarized, and processed through

INSULECTRO

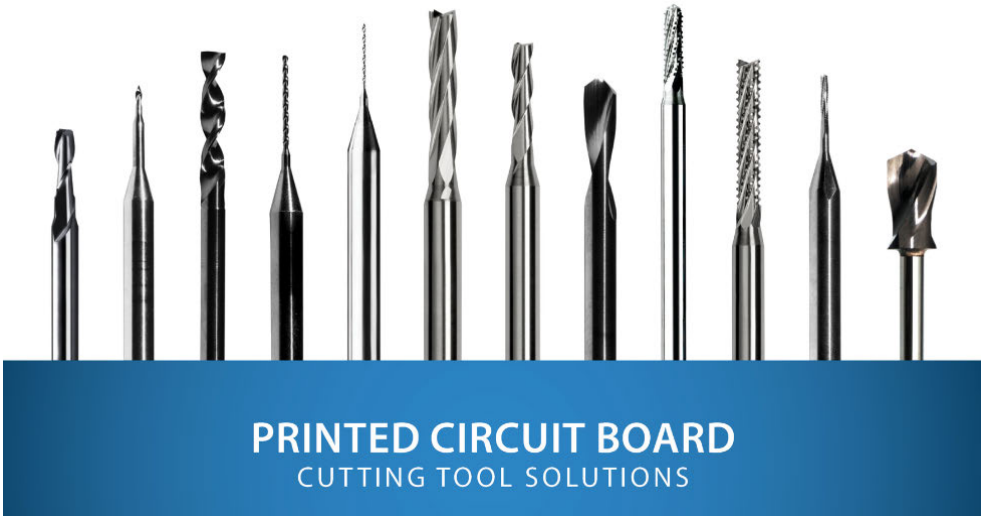


WORLD-CLASS TOOLS

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

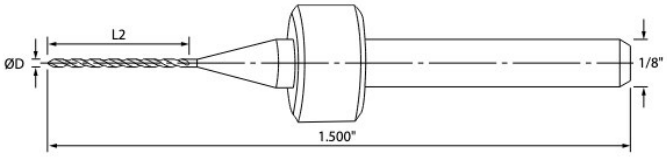
KYOCERA



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.

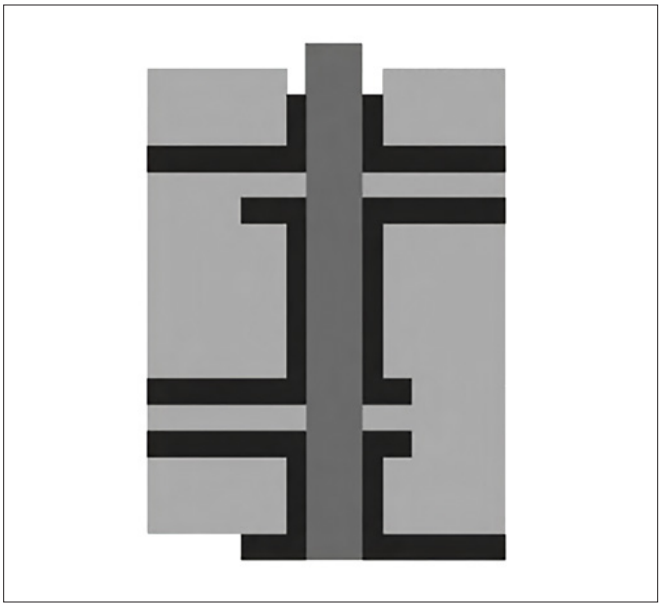


Figure 8: Outer-layer partition by solder mask filling.

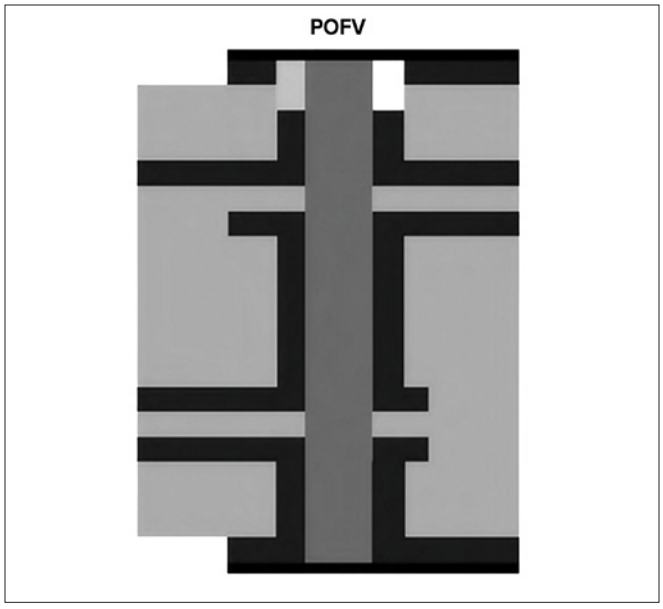


Figure 9: Outer-layer partition using POFV technology.

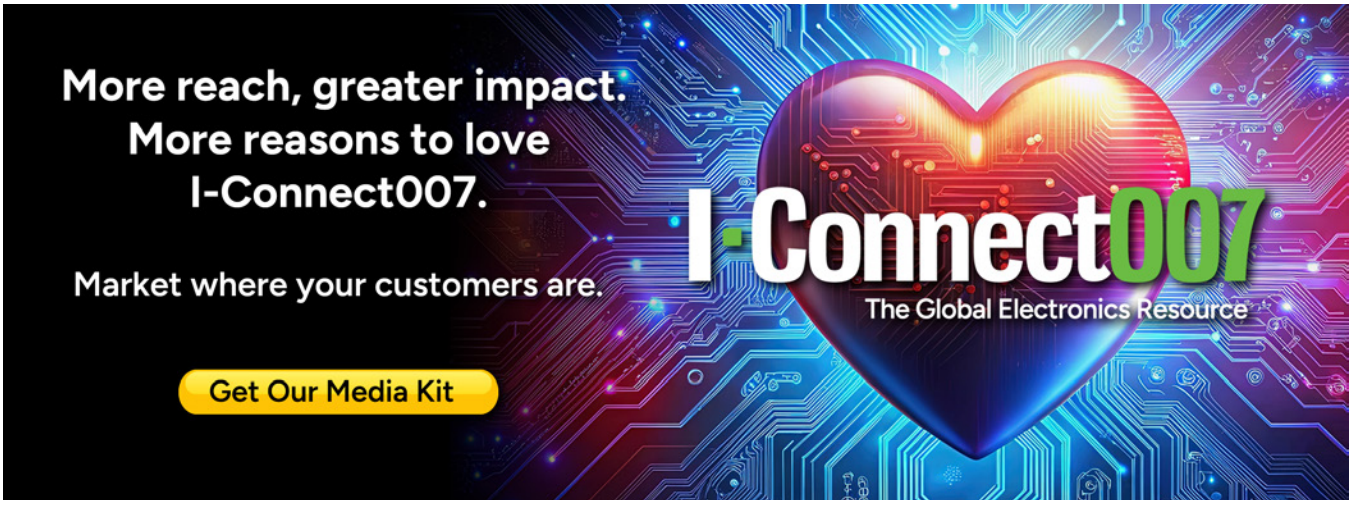
electroless copper deposition followed by electroplating to form a plated-over-filled-via (POFV) structure.

This approach not only provides complete electrical isolation but also enables routing directly over the filled vias, maximizing routing density on the outer layers.

The through-hole any-layer interconnection process offers an alternative approach to several challenges in PCB manufacturing. By creating electrically independent connections within a single plated through-hole, the method provides greater flexibility in interlayer routing while reducing the need for multiple lamination, drilling, and plating cycles. Its applications in thick-copper structures, high density BGA fan-out, and outer-layer partitioning demonstrate how a common manufacturing process can address different design requirements.

However, manufacturing efficiency and routing flexibility are only part of the equation. The reliability of these structures is equally important, particularly for applications involving thermal stress, high current, and demanding operating conditions.

Part 2 *examines the results of reliability testing, compares the proposed method with conventional build-up technology, and considers the challenges and opportunities associated with broader industry adoption.*

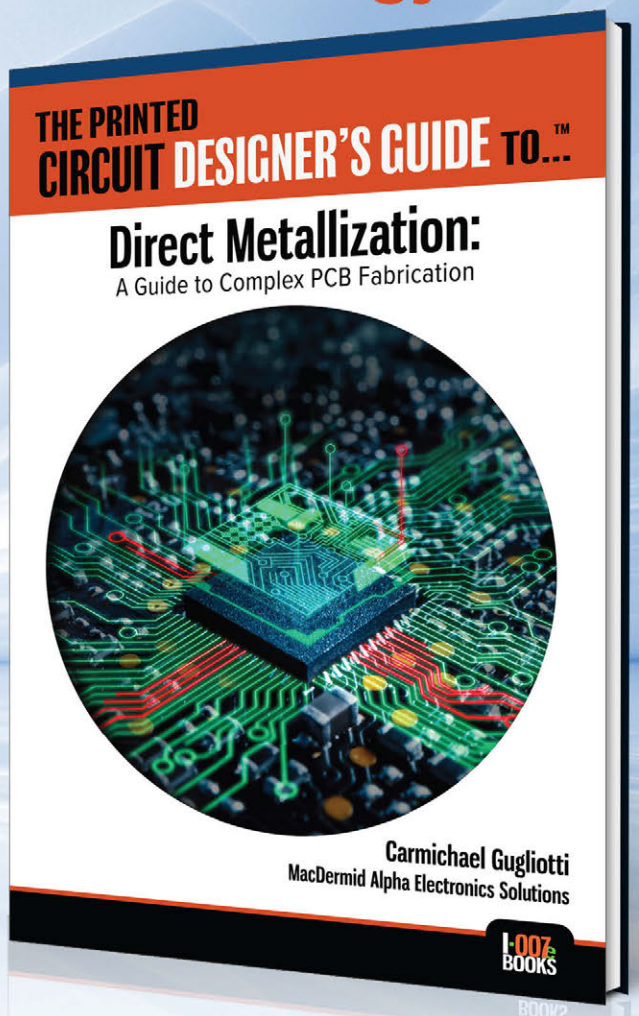


DIRECT METALLIZATION

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

New People, New Book

BY MARTYN GAUDION, POLAR INSTRUMENTS LTD

It's a slightly different "Pulse" this month as I write a prelude to Polar's upcoming book with I-Connect007, an introduction to impedance and insertion loss control. Neil Chamberlain, Polar's signal integrity product manager, frames it as a prequel to Polar's other two books, *The Printed Circuit Designer's Guide to… Secrets of High Speed PCBs* and *More Secrets of High Speed PCBs*.

The idea for an entry-level book emerged from my regular check of the most-visited pages on the Polar website. I noticed that a booklet called "An Introduction to Controlled Impedance PCBs" regularly appears among the top 10 most-downloaded files. Originally written more than a quarter of a century ago, the core information is still relevant, but much has changed in the industry during that time, so I have rewritten and added much more content relevant to today's new entrants to the industry.

The new book covers many topics familiar to "The Pulse" readers, but it also aims to provide a very readable, digestible overview for new entrants. When I approach a new and unfamiliar topic, the reading material often dives too deep too quickly without framing the outline for "why;" it often doesn't give enough supporting information to allow me to dig in deeper. Without that, I find it difficult to approach topics.

Maybe that's me, but in parallel, I find applied math more understandable than pure math. That's because there is a reason for it, and it's there to do a job, not just academic searching. Both have their place, but this book aims to give new entrants to the industry enough to ask the right questions.

The topic of questions reminds me of when I first met Polar's late founder, Doug Campbell. He kept a notice on his office wall: "Dumb questions are preferable to dumb mistakes." These wise words are for those who are reluctant to ask for clarification. It's a rule inside Polar that has kept us grounded in our first 50 years.

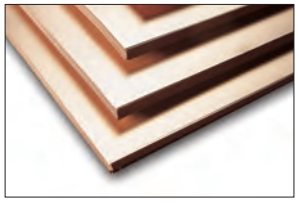
What Else You Can Expect

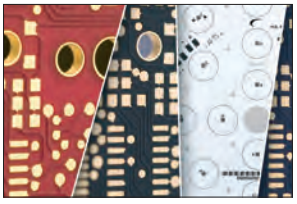


Figure 1: Reference coaxial air lines are the standard that PCB transmission lines are measured against.

Panasonic

Multi-layer & Flex Laminates





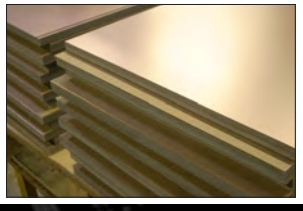
peters

Printed Circuit Coatings

CIRCUIT FOIL

Electro-deposited Copper Foils





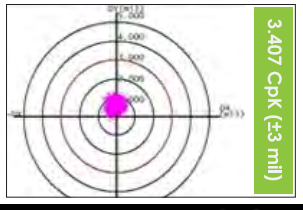
KB KINGBOARD

Rigid & Multi-layer CCL

PERFECT POINT PRECISION CARBIDE TOOLS

Drills & Routers





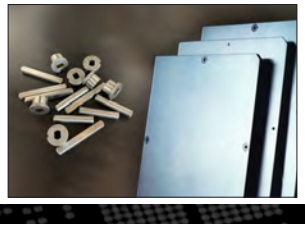
APEX ALUMINUM

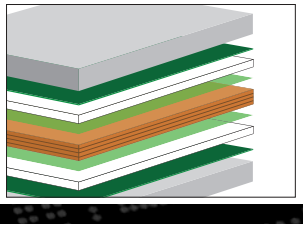
COOLSTRIKE LUBRICATED BACKUP MATERIAL

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

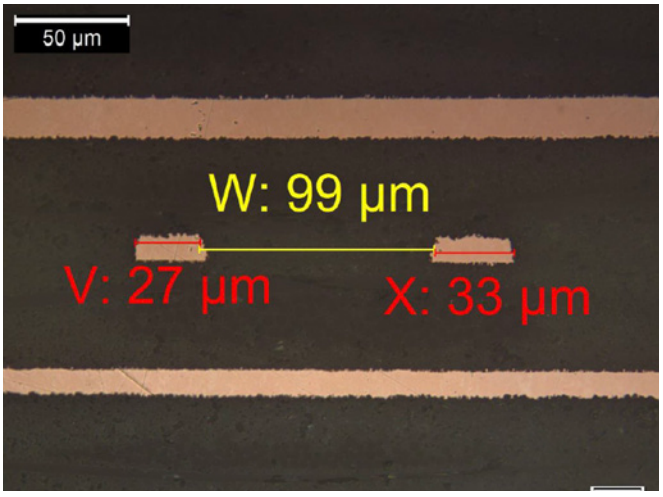


Figure 2: UHDI transmission lines take high speed into new dimensions.

The book will also cover why transmission lines are needed in PCBs at high speeds and where you are likely to encounter them, as well as some of the consequences of omitting to consider interconnects as transmission lines.

Then we discuss how fabricators control the characteristic impedance of on-board transmission lines through careful material selection and control of line widths and trace separation. For higher speed designs, we introduce the concept of insertion loss and explain its interplay with characteristic impedance. More importantly, we list the physical drivers that control insertion loss and contrast them with those that govern characteristic impedance.

I find it fascinating that in the early days of high-speed electronics, coaxial cables were used to route the high-speed signals point-to-point on a PCB. The cost and scale reduction by using transmission lines built into the structure of the PCB itself has been one of many key factors in the advancement of PCB capability over the half-century that Polar has been part of in the industry.

The Introduction of UHDI

We published our first introduction to controlled impedance when 10- or 15-mil traces were commonplace. The 10- to 25-micron traces of UHDI were unheard of, but they are with us now, and Chapter 14 explains what to expect when measuring and modelling at these line dimensions. At the time, those line widths would have been in silicon, not the PCB fabrication arena.

People who are new to the industry are faced with an enormous variety, from simple single- and double-sided PCBs, which are still workhorses in many designs, through to the most complex and demanding high layer count rigid and flex-rigid PCBs. I trust this new introduction sheds some light on one of the many specialist areas that are required for PCB production.

It has been interesting and refreshing to update introductory information for a new generation of PCB enthusiasts. I hope it encourages deeper learning and a hunger to empathise with all the technology specialists who are called on to mix disciplines and produce modern high layer count high-speed printed circuits. Please keep an eye out for publication dates. **Click here** to get on an early waiting list. **I-CONNECT007**



Martyn Gaudion is managing director of Polar Instruments Ltd. To read past columns, **click here**. Martyn is the author of *The Printed Circuit Designer's Guide to... Secrets of High-Speed PCBs,* Part 1 *and* Part 2*,* and The Printed Circuit Designer's Guide to... More Secrets of High-speed PCBs.



Your Idea. Our Expertise. Complete Solutions.



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

Our PCB production capabilities include:

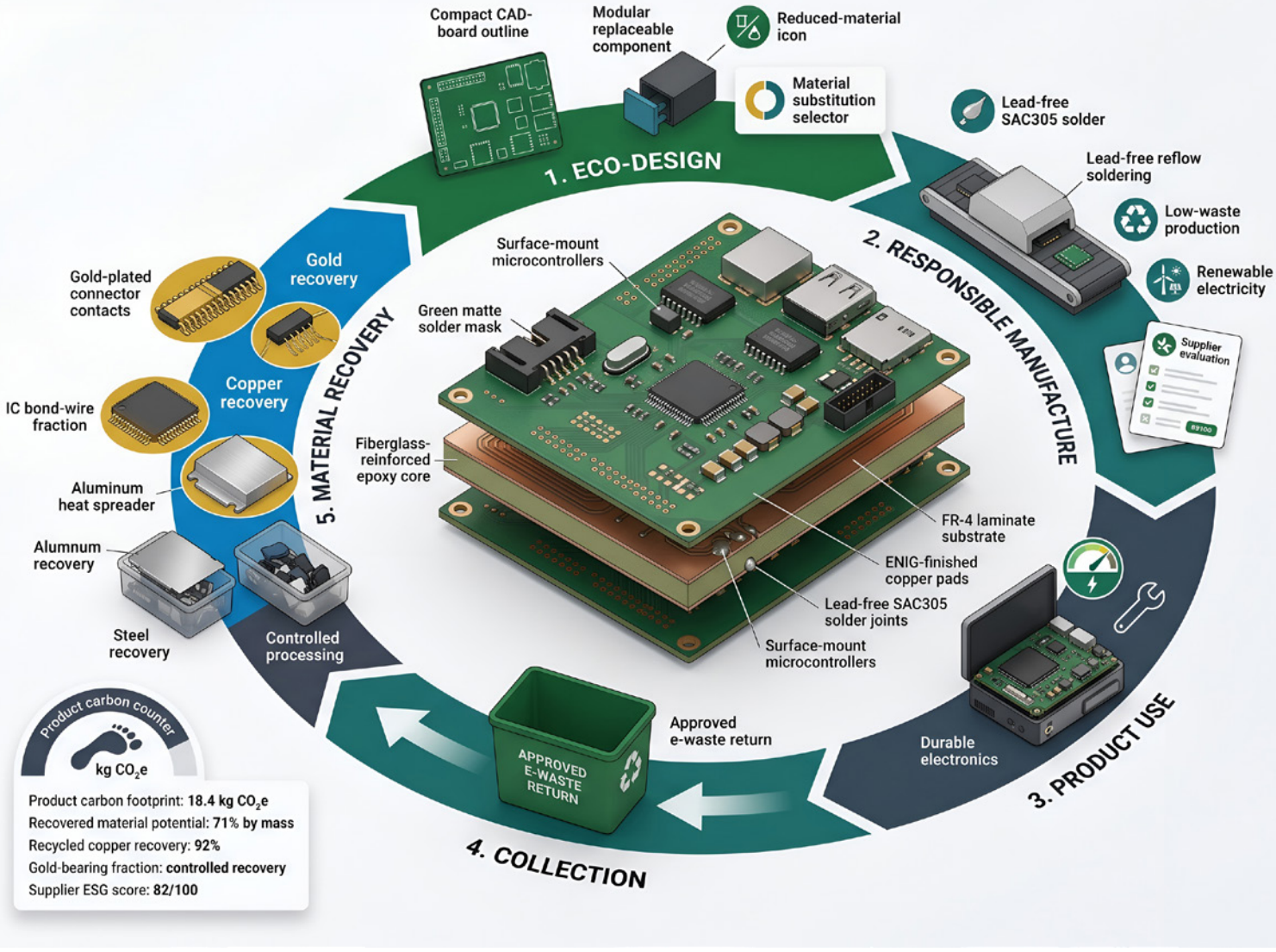
- State-of-the-art 45,000 sq. ft. manufacturing facility
- Precision drilling and plating
- A variety of final finishes and lamination options
- Advanced technologies, including RF and HDI
- Testing and quality assurance

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

OUR CAPABILITIES ▶

UNITED ELECTRONICS CORPORATION

sales@unitedel.com **847-671-6034** **www.unitedel.com**



The Chemistry of Sustainability

BY MICHAEL CARANO, GLOBAL ELECTRONICS ASSOCIATION

Sustainability is more than an environmental goal. It is a responsibility shared by individuals, businesses, and governments to protect economic prosperity, natural resources, and human health and safety. History offers ample evidence of the consequences of neglecting that responsibility: Efforts to promote growth and employment have sometimes damaged the environment and put people and wildlife at risk.

In his book *Collapse*, author Jared Diamond describes how societies can undermine their own futures by damaging the natural systems on which they depend.[1] His discussion reinforces a central principle of sustainability: Long-term prosperity

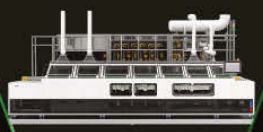
GREENSOURCE ENGINEERING

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



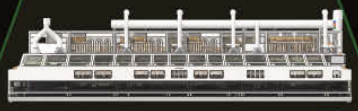
ETCH
Etch away superfluous copper for circuitisation



STRIPPING
Remove remaining resist and residues



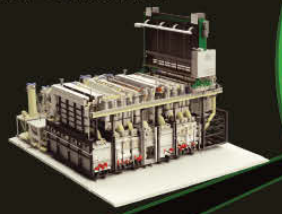
ADVANCED ADHESION PROMOTION
Optimized adhesion for superior buildup



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

depends on how responsibly we use natural resources. The question is not whether we should pursue progress, but how we can do so without compromising the systems that sustain us.

Balancing these priorities is not simple. Businesses must remain competitive, economies must grow, and communities need jobs and opportunity. At the same time, water and air pollution, greenhouse gas emissions, and resource consumption demand attention. These are not separate concerns; they are interconnected challenges that require thoughtful decisions.

The goal is to advance technology while protecting people and the environment. That requires looking beyond immediate results, considering the long-term consequences of our actions, and recognizing that even well-intentioned policies or process changes can create unintended problems. Sustainability is a commitment to continual improvement, not a one-time solution.

The Conundrum: What Can the PCB Industry Do?

For the PCB industry, sustainability begins with a practical reality: Manufacturing consumes water, energy, and chemicals. Our responsibility is to manage those resources in ways that support both a healthy business and a healthy environment. Waiting for these challenges to resolve themselves is not an option.

Chemical regulation is one part of that challenge. Federal policies and approaches to chemical risk evaluation continue to evolve, with substances such as formaldehyde, phthalates, tetrabromobisphenol A (TBBPA), and N-methylpyrrolidone (NMP) drawing scrutiny. Formaldehyde, used in electroless copper plating formulations, is particularly relevant to electronics manufacturing.

The industry can engage constructively with the U.S. EPA by focusing on two priorities:

1. Develop stronger data and documentation on how chemicals are used, what risks they present, and how those risks are controlled.

2. Work with suppliers to evaluate environmentally preferable alternatives and develop safer processes that also reduce regulatory risk.

Regardless of the regulatory timetable, manufacturers should understand which chemicals and materials they use, why they use them, and how they manage them. That knowledge is the foundation of responsible chemical management and informed process improvement.

Why wait for a restriction before taking action? The industry can make progress now by:

- Improving supply chain transparency and traceability for chemical and material content

- Using supply chain data to guide supplier selection and product design

- Documenting the safe use of essential chemicals and materials throughout the product life cycle

- Evaluating safer chemistries and processes through trade-off analyses and life-cycle assessments to determine whether alternatives offer an overall improvement

The same innovation that makes electronics more capable and reliable should also help reduce waste, lower emissions, and improve material safety. The transition to lead-free technology offers an important example of how environmental goals and technical performance can be addressed together.

Going Lead-free (Pb-free)

Some manufacturers remain hesitant to adopt lead-free PCB assembly because they question whether it can deliver the reliability of tin-lead solder. Research by the High Density Packaging Users Group (HDPUG) challenges the assumption that reliable solder joints require lead. Two phases of its Harsh Use Environment Alloy Evaluation project demonstrated strong performance in accelerated thermal cycling (ATC) and vibration testing. A separate project compared performance under thermal cycling and thermal shock.

Many third-generation, high-reliability Pb-free solder alloys demonstrated better thermal fatigue

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

performance than SAC 305. These findings support evaluating Pb-free alternatives on their demonstrated performance rather than dismissing them based on longstanding assumptions.

Design for Environment (DFE)

Reliability, however, is only part of the decision. In partnership with the EPA and electronics industry participants, the University of Tennessee's Center for Clean Products conducted a life-cycle assessment (LCA) of tin-lead and lead-free solders used in electronics.[2] The project evaluated tin-lead solder alongside three lead-free alternatives:

- 99.3% tin and 0.7% copper
- 95.5% tin, 4.0% silver, and 0.5% copper
- 92.3% tin, 3.4% silver, 1.0% copper, and 3.3% bismuth

The study gave solder manufacturers, users, and suppliers a basis for incorporating environmental considerations into material selection. Rather than examining only one manufacturing step, an LCA considers potential impacts across a product's life-cycle. For this project, those stages included:

1. Raw material extraction or acquisition and material processing
2. Solder manufacture
3. Solder application
4. End-of-life disposition

This broader perspective helps manufacturers evaluate solders that meet application requirements while potentially reducing impacts on public health and the environment. The assessment also identified areas for further research and provided information relevant to electronics purchasing and end-of-life management. The important lesson is to evaluate the full lifecycle, rather than assume that a material substitution improves every environmental outcome.

Making Modern Electronics Possible and Sustainable

Manufacturers have a responsibility to protect the resources that make their businesses, and modern life, possible. In PCB fabrication, that means paying close attention to water use, wastewater treatment, hazardous chemicals, and potential toxic emissions. Some essential manufacturing materials present health and environmental risks. We must understand those risks, control them effectively, and pursue opportunities to reduce them without compromising quality or performance.

Sustainability goes beyond regulatory compliance. It requires us to examine processes, question established practices, and evaluate alternatives using sound data. Even incremental reductions in resource consumption and waste can contribute to meaningful progress. The goal is not to abandon the materials and processes that make modern electronics possible. It is to use them more responsibly and keep improving how we do it. **I-CONNECT007**

References

1. *Collapse: How Complex Societies Choose to Fail or Survive, by Jared Diamond*, Penguin, 2005.
2. "Solders in Electronics: A Life-Cycle Assessment," by Jack R. Geibig and Maria Leet Socolof, U.S. EPA.

Resources

- "What is Sustainability? Definition, Three Pillars and Career Paths," by Matthew Mason, Feb. 7, 2026.
- Design for Experiment, ScienceDirect.



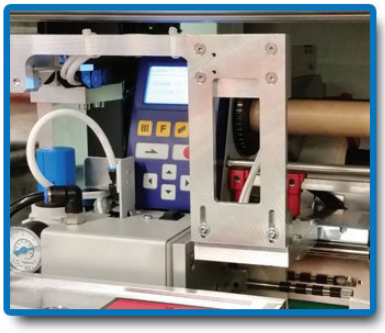
Michael Carano brings over 40 years of electronics industry experience with special expertise in manufacturing, performance chemicals, metals, semiconductors, medical devices, and advanced packaging. To read past columns, click here.

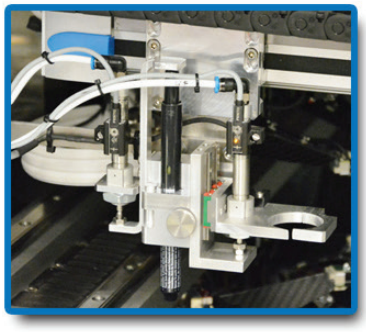
I-007e Get smarter here.

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

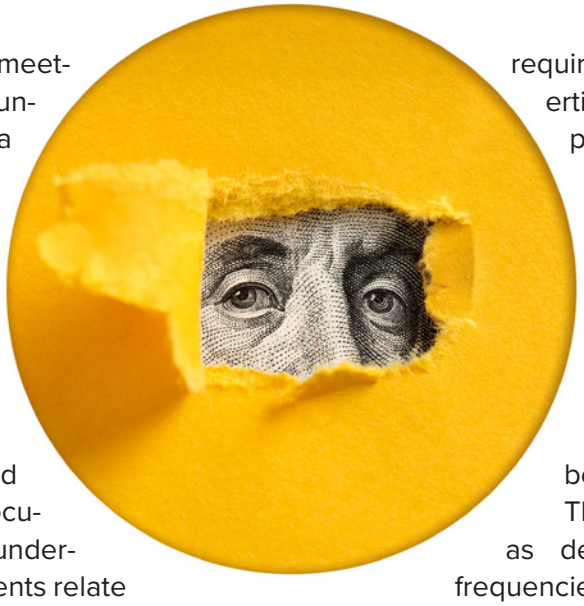
Get more info

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 HIDDEN COST OF 'GOOD ENOUGH' PCBS

BY MICHAEL GLEASON, GREENSOURCE FABRICATION



In PCB manufacturing, meeting specifications is fundamental to delivering a reliable product. Drawings, material requirements, and defined tolerances provide the framework that allows designers and fabricators to work toward a common result. But as electronic systems become more complex, there is value in looking beyond whether a board meets its documented requirements and understanding how those requirements relate to the performance of the finished product.

"Good enough" is often used to describe a board that satisfies its specifications at an acceptable cost. In many applications, that is an entirely appropriate standard. A prototype intended to demonstrate a concept will have different requirements from a board designed for long-term operation in a demanding environment. The opportunity lies in defining what "good enough" means for each application, identifying where additional precision adds meaningful value, and understanding how manufacturing decisions influence performance over the life of the product.

Understanding the Application Behind the Design

A fabrication drawing communicates what needs to be manufactured, but it doesn't always provide the full context behind the design decisions. A particular tolerance may be important because of assembly requirements, while material properties can influence electrical performance or thermal behavior in ways that aren't immediately apparent from the drawing. Understanding the intended application helps fabricators recognize which characteristics require closer attention and where manufacturing alternatives may be worth considering.

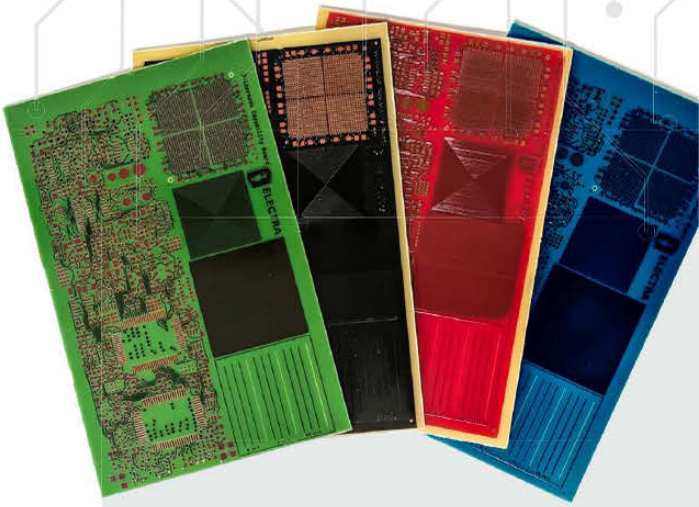
This is particularly relevant as designers work with higher frequencies, increasing power densities, and more complex board architectures. The relationship between materials, geometry, and manufacturing processes becomes increasingly important as performance requirements evolve. At the same time, tighter tolerances and advanced materials aren't necessarily beneficial in every application. Applying them without a clear technical reason can introduce unnecessary manufacturing complexity and expense. The objective is to establish requirements that reflect the actual needs of the product and support consistent, repeatable results.

These considerations also depend on the accuracy and completeness of the information provided to the fabricator.

In practice, we regularly encounter fabrication drawings that reference specifications that haven't been provided, contain hole counts that don't match the design data, or lack clarity around via fill and cap plating requirements. These details are important because they establish the requirements 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

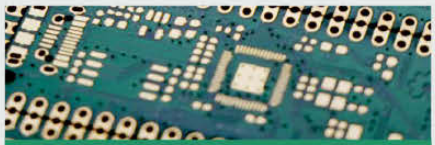
- Ink stable under a range of temperatures and environmental conditions
- No change in ink viscosity or particle size during use or shelf life of product
- Ink robustness critical for ensuring consistent printing with minimal problems
- Approved for use on leading inkjet printheads:
  - Konica Minolta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

fabricator needs to follow and can influence how a board is manufactured. Ensuring that this information is complete and clearly communicated from the beginning allows both designers and fabricators to work from the same understanding of what the application requires.

Why Manufacturing Consistency Matters

Every manufacturing process involves some degree of variation, and PCB fabrication is no exception. The important consideration is how much variation a design can accommodate while maintaining its intended performance. In applications with generous operating margins, small differences in materials or feature geometry may have little practical impact. In more demanding applications, those same differences can influence electrical characteristics, thermal performance, or long-term reliability.

As board designs become denser and operating requirements become more sophisticated, understanding these relationships earlier in the development process can help engineering teams make better-informed decisions. A board may meet its nominal specifications while still exhibiting variation within the permitted range. Whether that variation matters depends on the application, which is why process consistency and the ability to maintain critical characteristics across production builds deserve attention alongside basic specification compliance.

Manufacturing consistency also has implications beyond the board itself. When performance varies unexpectedly, engineering teams may need to investigate several possible causes before identifying the contributing factors. Additional testing, component evaluations, or design adjustments can all be reasonable parts of the development process, but they require time and resources. A better understanding of manufacturing variables can help teams focus their investigations and establish more effective controls where they are needed.

Copper plating provides a useful example of how manufacturing consistency can influence subsequent fabrication processes and the electrical performance of the finished board.

Maintaining consistent copper plating thickness across a production panel and from lot to lot is an important consideration, particularly for impedance-controlled designs. Variations in plating thickness can affect subsequent etching processes, potentially requiring requalification of etch parameters or adjustment of compensation factors. These changes can influence the finished geometry of traces and spaces, which in turn can affect electrical performance. Statistical process control (SPC) helps manufacturers monitor plating processes and maintain consistency within established specifications. The design itself also plays a role, as variations in copper density across a plated layer can complicate achieving uniform plating thickness.

Looking Beyond the Initial Board Price

PCB pricing provides a clear basis for comparing manufacturing options, but the purchase price represents only one part of the overall cost of bringing a product into production. Engineering time, validation requirements, production yield, and long-term reliability can all influence the economics of a design. These considerations become especially relevant when products require extensive qualification or are intended to operate in environments where consistent performance is essential.

This doesn't mean the most expensive manufacturing option is necessarily the best choice. Additional precision or more demanding specifications should provide a measurable benefit to the application. A tighter tolerance that doesn't improve performance may simply increase manufacturing costs, while an appropriately controlled characteristic can reduce the likelihood of additional testing or revisions later in development. Understanding that distinction allows engineering teams to direct resources toward the requirements that have the greatest influence on the finished product.

There is also value in considering these decisions before a design reaches production. When designers and fabricators discuss the intended application early, they can evaluate manufacturing requirements in the context of performance expectations rather than treating them as isolated specifications. That exchange can identify opportunities to simplify certain features, refine material selections, or adjust tolerances without compromising the product's objectives.

One example we see regularly involves the

relationship between via structures, copper plating requirements, and fine-feature circuit routing.

Designs which combine fine-feature routing on plated layers with multiple via terminations can present particular manufacturing considerations. These structures will require multiple plating cycles to meet the individual via requirements, increasing the overall copper thickness on the layer. As copper thickness increases, achieving fine-feature resolution becomes more challenging. Adjustments to circuit routing or changes to the stackup and via structures can help address these limitations. Making those decisions early in the design process is considerably easier than revisiting them after layout is complete and the board is ready for fabrication.

Building Better Decisions Into the Process

The growing sophistication of electronic systems is encouraging a more integrated approach to design and manufacturing. Fabricators have valuable insight into how materials behave during processing, which features require particular attention, and how manufacturing methods influence repeatability. Bringing that knowledge into the design process gives engineers a clearer understanding of the options available and the trade-offs associated with different approaches.

This becomes particularly important as manufacturing capabilities continue to evolve. Advances in fabrication processes can expand the range of designs that can be produced reliably, allowing engineers to evaluate approaches that were previously difficult or impractical to manufacture. Understanding those capabilities, along with their limitations, helps ensure that design requirements remain aligned with what can be consistently achieved in production.

Our focus on advanced PCB manufacturing includes capabilities such as fine-feature formation using mSAP and SAP processes, along with stacked and staggered microvia structures. These technologies support complex board architectures, in which the relationship between design requirements and manufacturing execution warrants careful consideration. By working with customers to understand their applications and evaluate fabrication requirements, we aim to help establish practical manufacturing approaches that support the intended performance of the finished product.

With advanced HDI designs, early engagement between the fabricator and the customer is particularly important because many PCB attributes are interrelated. Providing accurate design rules gives engineers an essential starting point for layout, but understanding how those requirements interact during fabrication is equally important. A design may satisfy individual rules while creating manufacturing conflicts when those requirements are considered together. Working through these relationships early allows designers and fabricators to identify potential limitations, evaluate alternatives, and establish a design that can be manufactured consistently.

Defining Value Through Performance

The definition of a successful PCB has always depended on its intended application. What is changing is the level of understanding required to translate increasingly sophisticated product requirements into reliable manufacturing outcomes. As fabrication technologies advance, designers have new opportunities to evaluate materials, refine board architectures, and establish requirements that reflect the actual demands of their products.

The most effective decisions come from knowing where precision matters, understanding what manufacturing processes can consistently deliver, and recognizing when additional complexity provides a meaningful benefit. A board that meets its specifications is an essential starting point, but the broader measure of value lies in how well those specifications support the performance and reliability of the finished product. As the industry continues to develop more capable manufacturing processes, that connection between design intent and manufacturing execution will become an increasingly important part of building what's next. **I-CONNECT007**



Michael Gleason is director of product development and strategic growth at GreenSource Fabrication. To read previous columns, click here.

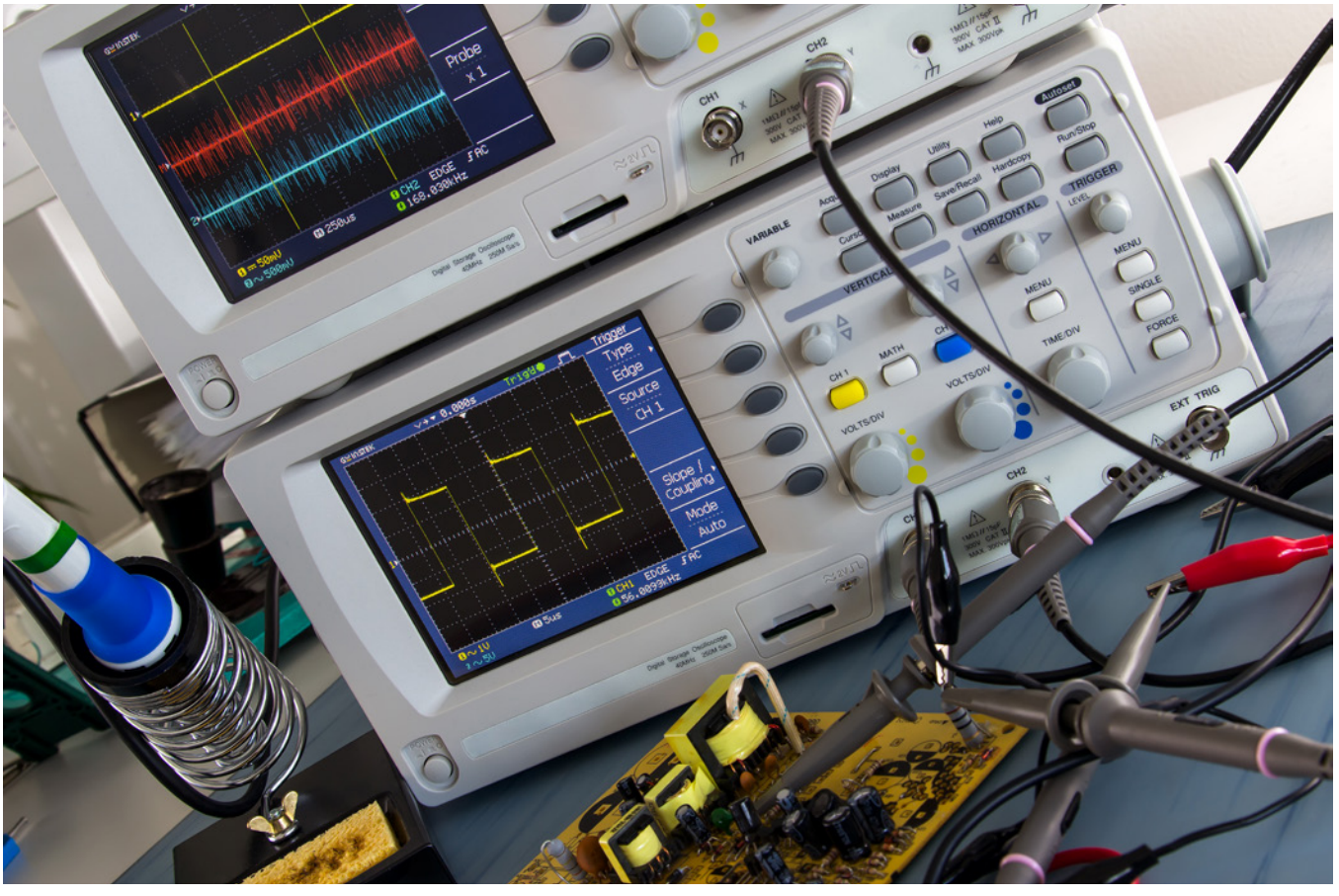
BY KIRSCH MACKEY, HASOFU

What Standard Power Supply Design Reviews Miss Before PCB Layout Sign-off

Early in my career, I learned that getting a power supply design to work and getting it ready for production are two very different things. On one project, our team used a hardware reference design, application notes, circuit analysis, and simulations on a 6-layer high speed PCB. The board worked, but we still went through four respins over six months and exceeded our budget by 300% because we hadn't accounted for critical requirements such as UL safety and emissions testing.

The experience was stressful and taught me that a functional design is only the beginning. A design that passes in the lab is a proof of concept, while a design that is truly ready must be reliable, compliant, manufacturable, and ready for customers.

That experience showed me the value of a comprehensive power supply design checklist. In industry, successful designs require much more than converter calculations and simulations. They also require consideration of standards, regulatory




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



Figure 1: Higher frequency return path (illustrated) current avoiding voids on a flyback converter PCB designed in Allegro X PCB layout.

requirements, testing, and other details that are often buried in meetings and email threads. A repeatable design and review process helps teams build power supplies that are more predictable, reliable, and production-ready.

In this article, I'll share the checklist I use for power supply design reviews and explain how it can help both new and experienced engineers avoid common pitfalls and create designs that are ready for the real world.

The Problem With the Typical Power Supply Design Review

Most engineering workflows don't enforce enough requirements for a working power supply the first go-round. This experience exposed me to just how incomplete the power electronics design engineering process is.

Every hardware test and regulatory requirement exposed what we hadn't considered or had skipped over in the earlier part of the process. So, whether we addressed the problems sooner or later, they were always addressed. We just needed to decide how much more time and money we wanted to spend finding something out that could have been prevented in the first place.

I figured, "The reference design was done by engineers and works for them, so it'll work for me." This experience is common enough because not everything that's published is correct for my problem or application.

Top Five Oversights on Most Power Supply Designs

Power supply designs can fail in the field due to a few overlooked details. For each oversight listed below, we'll cover:

- Why it's missed in a normal power supply design review
- What happens when you overlook it
- What the checklist tells you to do so you avoid those problems altogether

Let's start with the oversights that cause the most problems, as foundational problems wreak the most havoc. Then we'll look at how to prevent them.

Problem 1: Return Path Discontinuity

Why it's missing in review: Engineers tend to check return paths simply by visual inspection, if even reviewed at all. Unless the return path is embedded in your constraints, it is likely to get overlooked or have gaps.

What happens when you overlook it: Return path discontinuities allow EM waves and fields to leak. The leakage of fields causes EMI noise issues for other devices, fails emissions testing, generates unwanted noise, and more. Even misoperation of components depends on their sensitivity level to said noise.

How to avoid this problem: Create an unbroken return plane that connects to ground. Especially put that return plane underneath the hot loops. The hot

loops produce most of the EMI issues, so you need something to protect against those.

Always fix the return path before PCB layout sign-off. It affects everything else.

Problem 2: Hot Loop Inductance

Why it's missing in review: Engineers often estimate the hot loop inductance through visual inspection (aka eyeballing) rather than extracting the parasitic values. But without real inductance measurements, the designer might underestimate the trace parasitics that cause severe switching noise and voltage spikes across power MOSFETs.

What happens when you overlook it: When you don't do a thorough check, or they're overlooked, you get higher inductance, therefore higher voltage spikes across the switch (about 10–20 nH per inch of copper trace, multiplied by a fast current rise time).

How to avoid this problem: Minimize loop area by placing components close together. Then run a simulation for accurate numbers to see how much inductance is still present. The exact values on inductance will give you the truth and where to update your placed components, like decoupling and bypass capacitors.

Problem 3: Stability at Only One Operating Point

Why it's missing in the review: Engineers who simulate tend to focus on one key scenario, one circuit state, or one operating condition. But that doesn't mean the circuit is stable in other operating conditions.

What happens when you overlook it: When you only simulate at one operating point, you don't know if a load step change will kick the circuit out of stability. That instability can happen when you're outside normal operating conditions even without a step change.

How to avoid this problem: To avoid this, simulate at multiple operating points: What is the phase margin at minimum voltage, nominal voltage, and maximum input voltage? For each of those conditions, what's the maximum and minimum load? In

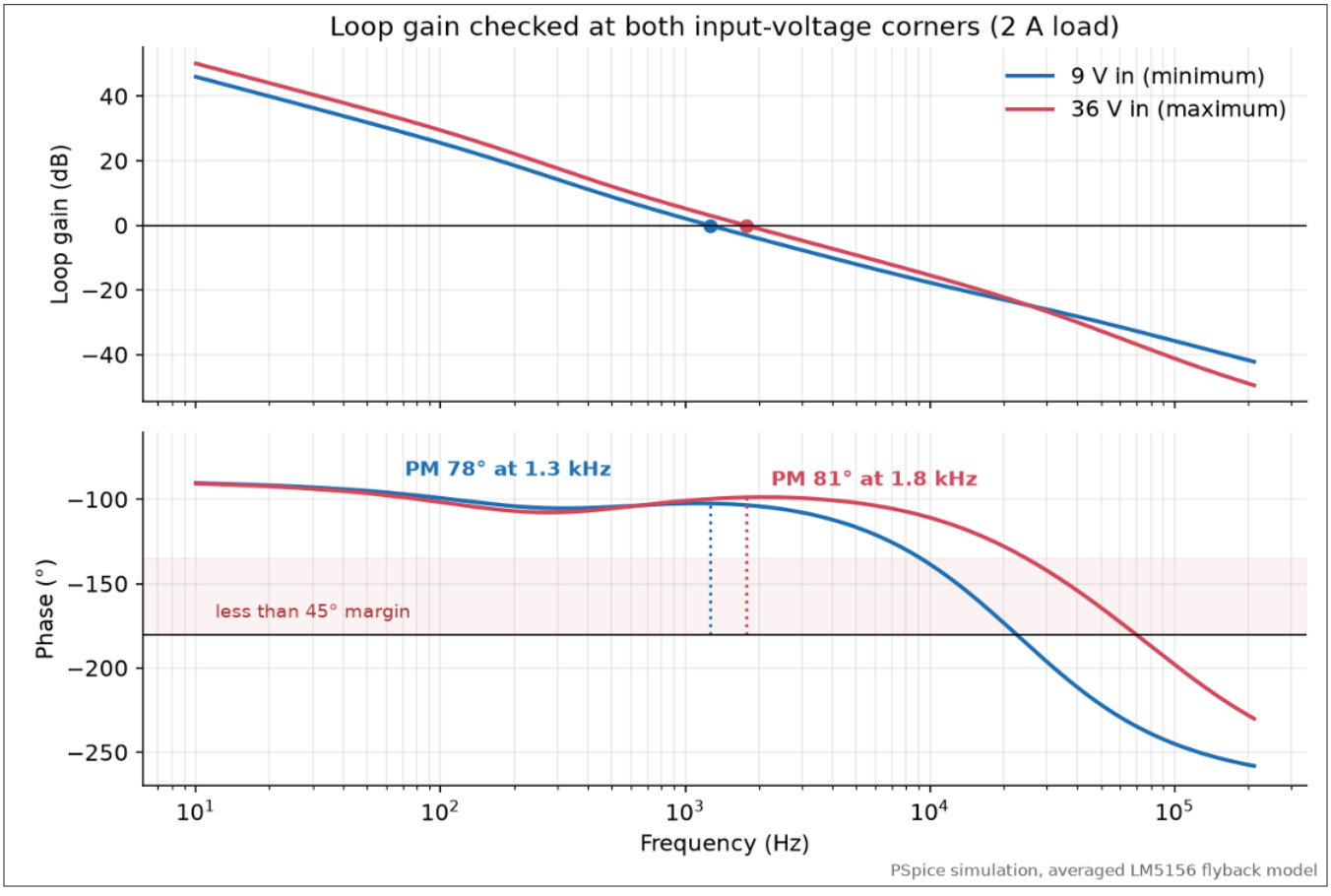


Figure 2: PSpice Bode plot of a flyback at minimum and maximum input.

running those various simulations, you want your phase margin to be at least 45° (ideally 60°).

Problem 4: Worst Case Voltage Stress

Why it's missing in review: It's time-consuming and complex to run analyses on every component that is overly stressed in the design, because you have to take key components (if not all) into account to get good estimates.

What happens when you overlook it: If you don't check, your boards are at risk of becoming overstressed and more likely to fail (without derating) under worst-case conditions. If parts fail early and customers experience boards failing earlier than expected, you're at risk for product recalls, losses in sales, and customer trust. We can't have losses in sales. Cash flow keeps things going.

How to avoid this problem: If possible, use software that runs this kind of check automatically or scripts that can calculate stresses for you. Whether you need a combination of AI, datasheets, or automation to run calculations for you, you'll want a report explaining which devices are at risk for voltage degradation. Or run a quick check using a PCB layout tool that lets you analyze electrical overstress right within your design environment.

Problem 5: Not Enforcing Your Outcome Requirements

Why it's missing in review: Capturing requirements is cumbersome because they're often in multiple places like written notes, emails, and presentation slides. So, requirements often fall through the cracks. Also, there isn't a consistent method for ensuring a design meets certain requirements.

What happens when you overlook it: The problem shows up too late in the design process. The cost of retroactively changing or fixing the design to meet a requirement is very high in engineering man-hours. Another problem is that if you don't meet the requirements but your design works, you have to redo the board anyway to meet them, so you are only making the work more difficult by missing them in the first place and not enforcing them.

How to avoid this problem: Have your requirements in one location and implement them in the Constraint Manager. For example, to improve visibility, leave comments in your design environment that let you tag other designers so they can contribute to the design. You can also enforce allowed noise, crosstalk rules, and electrical clearance and creepage, depending on which electrical features and requirements matter most to your design.

Then once you've selected all electrical analyses you want to check for, enforce the rules with values and constraint sets, like trace width rules for current carrying capacity, clearance and creepage, and even DFX rules directly in the Constraint Manager. The software itself implements the requirements and constraints, then creates markers to let you know when those rules have been violated. This is proper rule enforcement.

Conclusion

For months, we struggled to pass EMC testing or UL certification. The extra travel, testing, and respins put us about $100,000 over budget and forced us to bring in a UL expert to get our regulatory process right.

Then a seasoned power electronics design engineer reviewed our schematic and PCB, applied first principles and his experience, and got us to finally pass EMC testing with a $6 solution. No respin.

Experiences like that are exactly why resources like the power supply design guides are useful. They walk through the steps for designing a robust power supply using a flyback converter reference design that takes you from schematic to PCB layout while highlighting the kinds of issues that often lead to costly delays, test failures, and respins.

Now that you know some of the biggest bottlenecks and contributors of power supply design, you can design your board with more accuracy, speed, and confidence. **I-CONNECT007**



Kirsch Mackey is an electrical engineer, educator, and content creator with more than 15 years of experience in electronics design, embedded systems, and power engineering. He is also the founder of HaSofu, where he shares practical insights and educational content for the engineering community.

cadence

One Platform.
Every Stage of PCB Design.

Allegro X Design Platform

HANDOFFS BREAK

Schematic and layout drift apart.

KEEP DESIGN INTENT CONNECTED

Synchronize intent and constraints from schematic to layout.

ISSUES SHOW UP LATE

SI, PI, and thermal problems surface after layout.

FIND ISSUES EARLIER

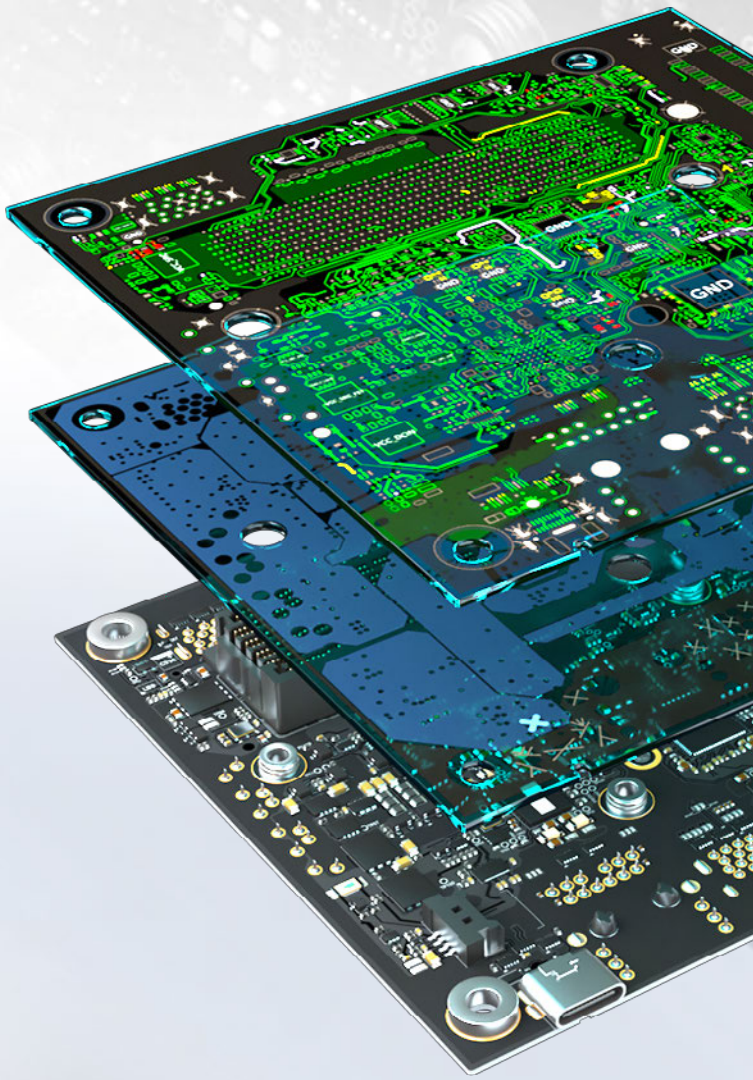
Analyze SI, PI, and thermal during layout, while changes are still easy.

TOOLS DON'T TALK

Disconnected tools create conflicts and delays.

WORK IN ONE ENVIRONMENT

Design concurrently in one connected environment.



Fewer late-stage surprises.
More first-pass success.

WATCH THE ALLEGRO X DEMO

BY RICHARD NICHOLS, GREENSOURCE ENGINEERING

The Evolution of Horizontal Etching

As we've pushed the boundaries of performance in AI server infrastructure and robust high-voltage power distribution systems, the integrity of the copper interconnects has become the defining factor for success. Among the various chemical processes involved in PCB manufacturing, copper etching remains the most critical juncture for ensuring signal integrity and electrical reliability.

Specifically, achieving superior etch uniformity in horizontal cupric chloride systems is no longer just a quality target; it is an absolute necessity for modern, high density, and high power applications.

The Challenge of Modern Circuitry: AI Servers and High Voltage

The current generation of AI server panels operates at unprecedented speeds and power densities. These boards are marvels of engineering, packed with high speed signal lines that require exact impedance control. Any variation in the copper line width or profile due to non-uniform etching can cause signal reflections, crosstalk, and timing jitter, all of which compromise the performance of the AI processing units they support.

Simultaneously, the transition toward electrified systems and renewable energy infrastructure has placed a renewed focus on high-voltage circuits. Unlike high speed digital signals, these circuits deal with significant current-carrying requirements that necessitate thick copper foils. Etching these thicker foils presents a unique set of physics-based challenges, particularly when using horizontal conveyorized systems.

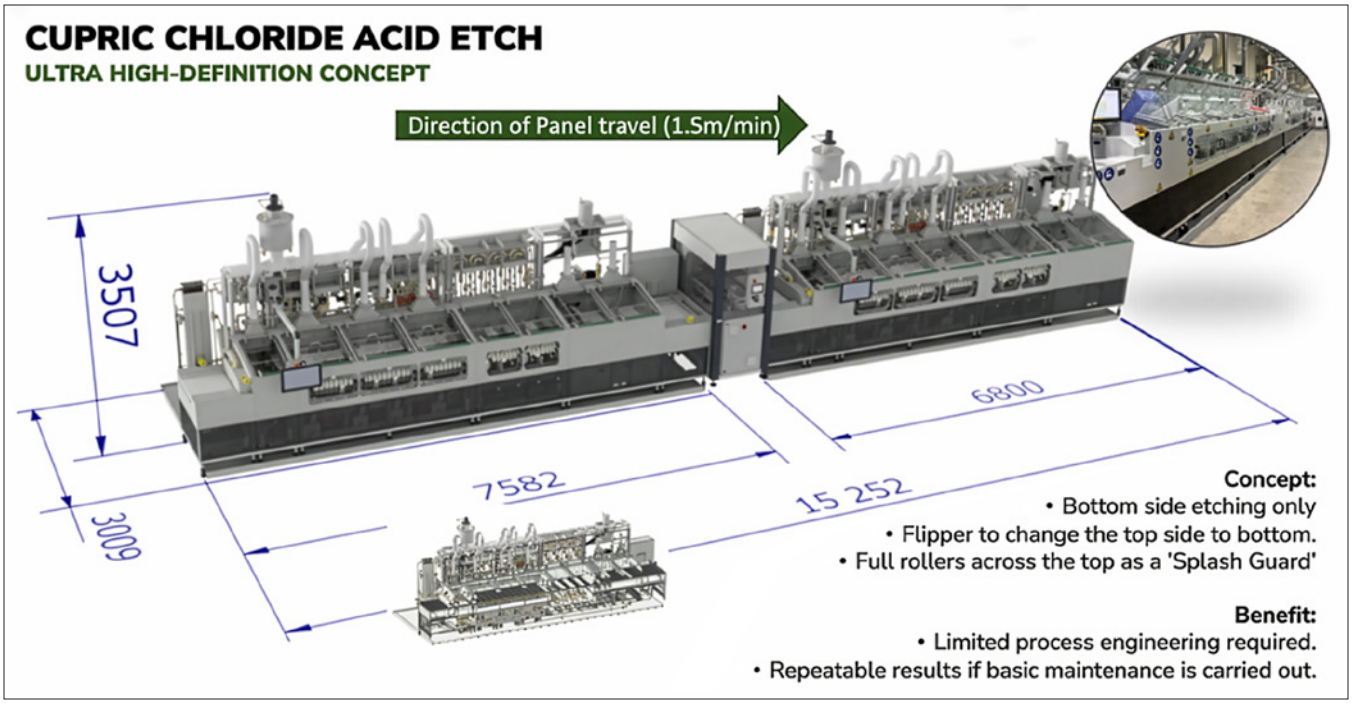


Figure 1: Equipment overview.

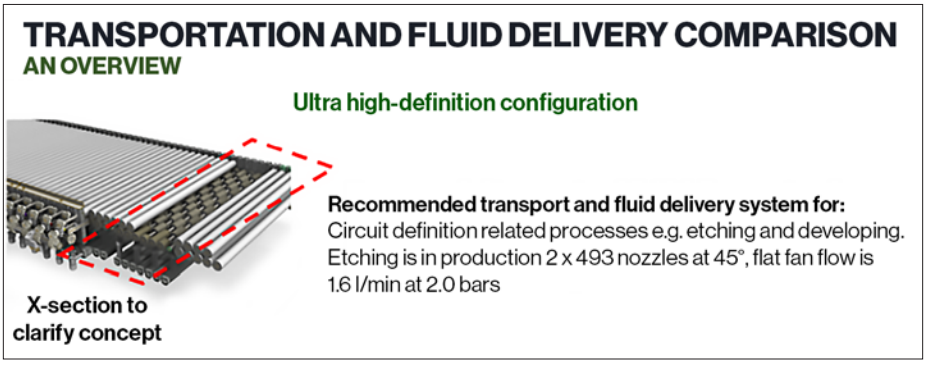


Figure 2: Transportation modifications.

The Problem of Horizontal Etching: Puddling and Differential Etch

When processing panels in a horizontal cupric chloride etcher, the etchant is sprayed onto the board's surfaces. While theoretically balanced, gravity introduces a pervasive challenge: top-side puddling. As the cupric chloride solution interacts with the copper, it creates reaction byproducts. On the bottom of the board, gravity helps these spent etchants drain away, allowing fresh chemistry to continuously access the copper surface.

On the top side, however, the solution tends to collect, creating a layer of stagnant liquid called the "puddle," which inhibits the flow of fresh etchant and alters the local concentration of the chemistry, leading to poor etch uniformity. This can have a significant impact on differential etching, or "etch-down." Effectively, it means the bottom of the board etches faster and more aggressively than the top. For fine-pitch high speed traces, this results in an uneven trapezoidal profile, where trace widths vary uncontrollably. For thick copper high-voltage traces, this unevenness can lead to undercutting, reduced current capacity, and potential reliability issues due to excessive etching on one side relative to the other.

Solving the Etch-Down Dilemma

Historically, manufacturers have attempted to mitigate this by adjusting spray pressures, nozzle angles, or conveyor speeds, often at the cost of throughput or footprint efficiency. However, these are workarounds rather than solutions and are heavily labor-intensive in process engineering. As etch chemistries are usually generic rather than proprietary, the task of balancing up the etch uniformity on the top and bottom cannot be passed along to the supplier. To truly master the etching of thick copper for high-voltage applications, the industry requires a system that treats the top and bottom surfaces as equals, irrespective of the physical constraints imposed by gravity.

This is where GreenSource Engineering is recognizing that the footprint of modern manufacturing facilities is at a premium, and the engineering team has developed a novel approach to horizontal etching. By optimizing fluid dynamics and nozzle geometry specifically for the cupric chloride process, we have engineered a solution that actively prevents top-side puddling without requiring an oversized equipment footprint.

A New Standard for Reliability

This approach focuses on "active management" of the etchant stream. Using high-velocity, controlled-flow delivery systems, the solution effectively shears the etchant from the panel surface. A flip between etchers prevents puddling and allows the etch chemistry to operate with uniform aggression across the entire copper surface, resulting in consistent trace profiles on both the top and bottom of the board.

For the manufacturer, this translates into several key advantages:

- **Enhanced impedance control:** Uniform trace profiles ensure impedance remains consistent across the panel, which is critical for high-speed AI signal integrity.
- **Superior reliability for high voltage:** By eliminating differential etching, manufacturers can ensure that thick copper circuits maintain their structural integrity and design width, providing reliable power delivery for high-voltage systems.
- **Improved yields:** With fewer variables to manage and a more stable process, scrap rates are reduced, and the cost of ownership decreases.

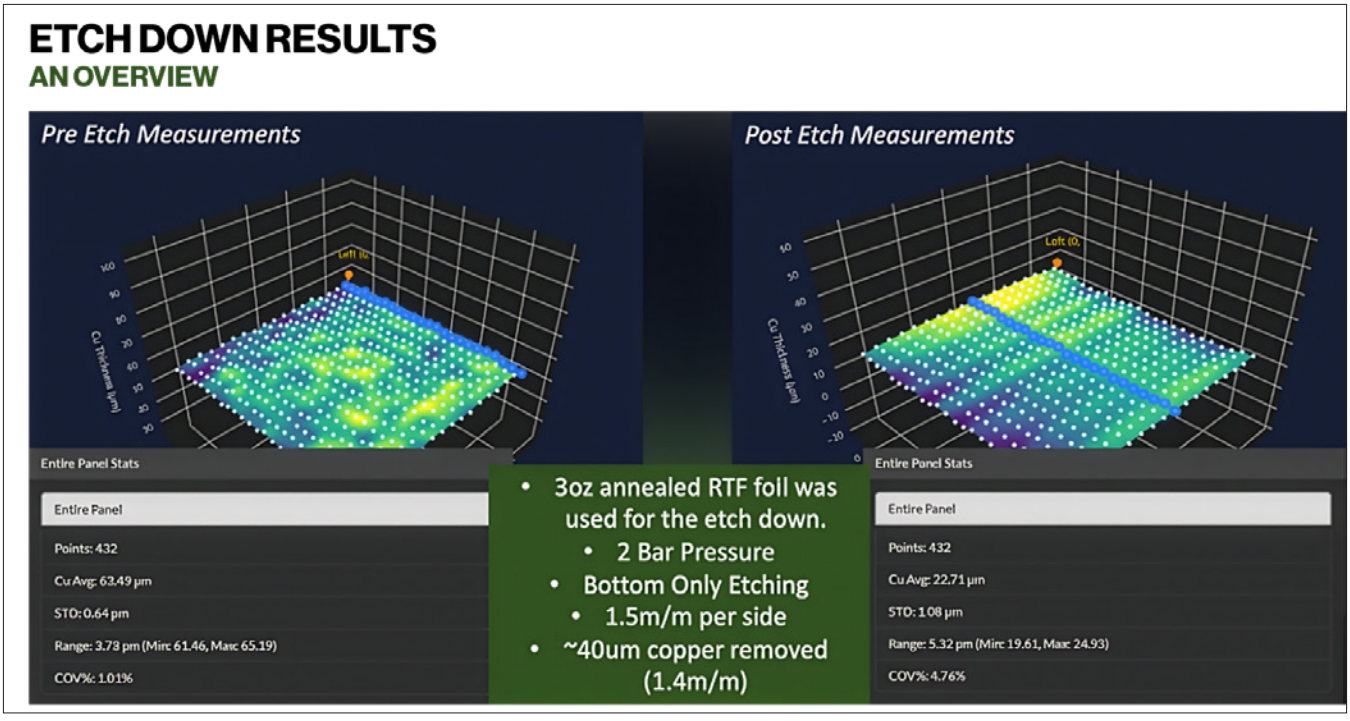


Figure 3: Demonstrating that even after etching 40 µm of copper, the COV% is low.

Efficiency: Achieving these results without increasing equipment size allows facilities to upgrade their capabilities without reconfiguring their entire production line.

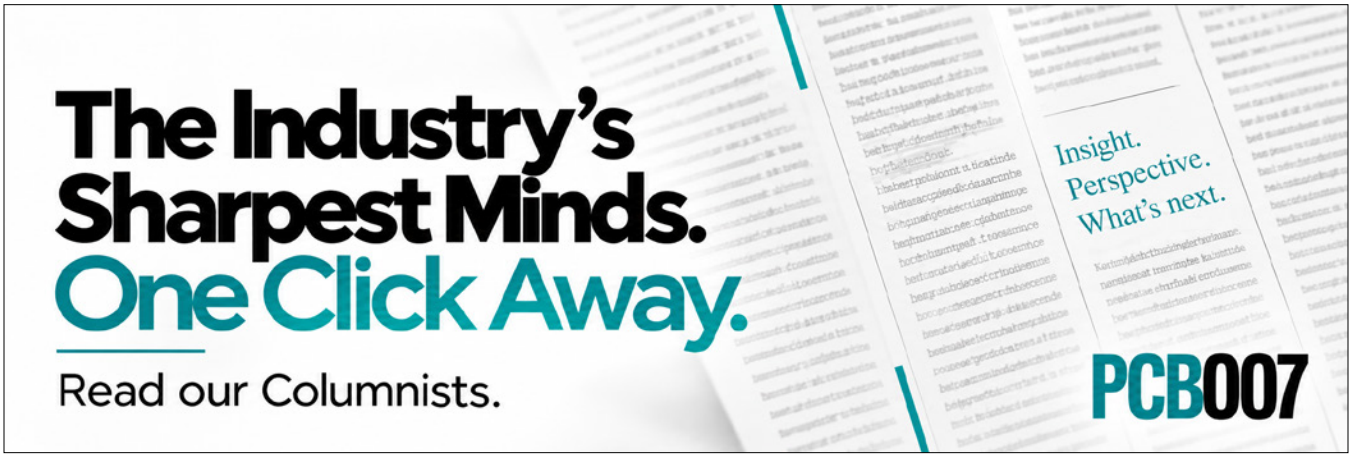
Conclusion

As the electronics industry pivots toward the massive data processing power of AI servers and the high-power needs of future energy grids, the manufacturing processes that support these technologies must evolve. Copper etching is the unsung hero of this evolution.

By addressing the fundamental challenges of horizontal cupric chloride etching and the top-side puddle, we can help build tomorrow's boards with the precision and reliability demanded by future technologies. The transition to high-uniformity, high-throughput etching is not just an equipment upgrade; it is a vital step toward sustaining the pace of technological innovation. **I-CONNECT007**



Richard Nichols is technical marketing director at GreenSource Engineering.





ventec
INTERNATIONAL GROUP

Special Applications & NF/LF Prepreg

Bringing Ventec quality, performance and value to specialist circuits.

Bring Ventec quality, performance and value to circuits with special requirements. Like those needing mechanical flexibility - whether formed once to allow installation during product assembly or flexing dynamically with moving parts such as a printer head or optical drive, our flex and flex-rigid formulas withstand reflow temperatures and maintain their structural integrity to prevent fatigue or corrosion.

Contact your local sales representative today.

venteclaminates.com

Wherever technology takes you, Ventec delivers

Tell the New Congress Why PCBs Are Essential to Our Security

BY SHANE WHITESIDE, PCBAA

The 2026 midterm elections are over, and many of the names and faces in Congress will change. Those new members of Congress will all need to be educated about the critical importance of the PCB industry.

Newly elected and returning members are both charged with representing your interests. There is a tremendous need to educate them on the critical role PCBs play in the microelectronics ecosystem, because even among veteran members, few know the history of our industry and how little U.S. capacity remains to manufacture PCBs today.

There is a lot of talk about supporting AI data centers, defense systems, and other critical infrastructure, but little realization that PCBs made in other countries are used in every one of those systems. America is dependent on PCBs made in countries that are either adversarial in unfair trade practices or gathering nefarious military intelligence, or both.

The overseas boom in PCB production is not an accident. The uneven playing field evolved through smart industrial policies, incentives, and public and private investments. Billions have flowed into China, Vietnam, Thailand, Malaysia, South Korea, and Japan while investment has mostly flowed out of the U.S.

Our elected officials have been discussing what to do while our global competitors have been deploying capital. The new Congress must understand, and should hear firsthand from companies in the PCB industry, that the American PCB industry needs the same type of incentives that semiconductors received under the CHIPS Act, which demonstrated that private capital follows public action. Government support for the PCB industry is a signal to Wall Street that reshoring and restoring industrial capacity remain national priorities.

There are already two PCB bills in Congress: HR 3597 and SB 4649. The provisions in those bills would give the PCB industry the boost it needs to compete in the global marketplace and to provide trusted and secure PCBs for America.

In the meantime, American PCB manufacturers, materials providers, and assemblers are competing with countries with favorable industrial policies and government investment. Without those ingredients, the U.S. will remain a small player in this growing and critical market.

There is something everyone in the industry can do: Contact your member of Congress and explain the local economic impact of the PCB ecosystem and the need to support our bills.

We invented the PCB here and, at one time, produced 30% of the world's supply. That share has now eroded to less than 4%. American companies deserve the benefit of the same smart industrial policies and bold action that our competitors have been pushing for decades.

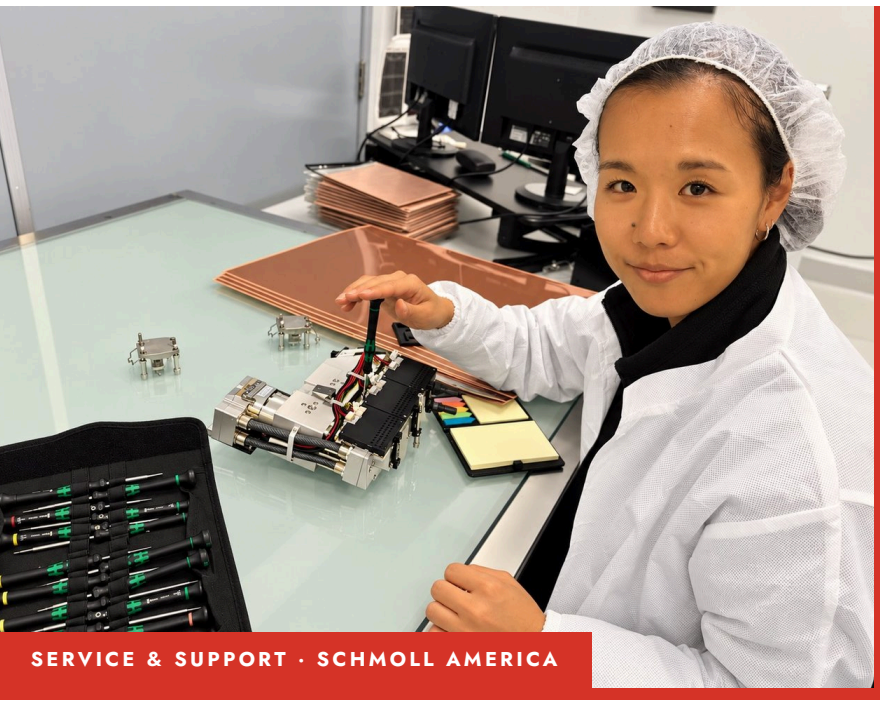
Join PCBAA's ongoing campaign and tell your members of Congress to act in support of American manufacturing. Don't sit on the sidelines: Use this link to make your voice heard. **I-CONNECT007**



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

SCHMOLL AMERICA · SERVICE & SUPPORT

Exceptional Machines. Outstanding Service.



SERVICE & SUPPORT · SCHMOLL AMERICA



ENGINEERING · RÖDERMARK, GERMANY

At Schmoll America, we know your success depends on more than great machines – it requires tailored, top-notch service.

With the expertise of Schmoll Maschinen GmbH behind us, we deliver:

- **Minimal downtime**
  Quick response times and fast on-site intervention
- **Fast access to spare parts**
  Original Schmoll parts when you need them
- **Expert guidance when you need it**
  Process know-how, remote and on site

We're dedicated to providing the support you deserve, because the best machines should always come with the best service.

From quicker response times to customized solutions, Schmoll America ensures your operations run smoothly and efficiently – backed by the engineering team in Rödermark, Germany.

Let's elevate your production to the next level.

SERVICE HOTLINE

Support@SchmollAmerica.com

+1 888-SAM-0321 or 1-888-726-0321

schmollamerica.com

SCAN TO CONTACT US

schmoll america

... one step ahead.

BY ANAYA VARDYA, AMERICAN STANDARD CIRCUITS



PLANT BIOMETRIC SENSORS:

Flex PCBs Listening to the Language of Crops

Plants speak. It's not in words, but in electrical signals, pressure gradients, gas concentrations, and biochemical cascades that are legible, if you have the instruments precise enough to listen. For most of agricultural history, those instruments did not exist. Farmers read crop health through visible symptoms: yellowed leaves, stunted growth, wilting stems. By the time any of those signs appeared, the problem was already well advanced. The opportunity to intervene early had passed.

Flexible PCBs are changing that fundamental dynamic. Thin, conformable, and engineered to live on or inside living plant tissue, flex PCB-based biometric sensors are giving researchers and growers a real-time window into the physiological state of crops at the cellular level, before stress becomes visible, before yield losses become inevitable.

What Plant Biometrics Actually Measures

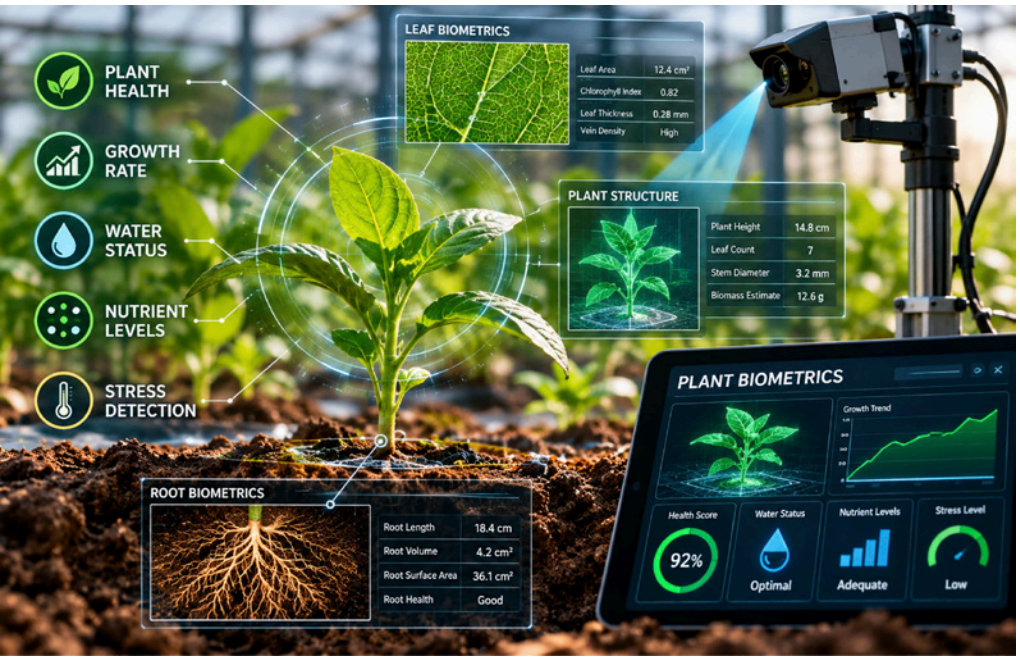
The term "plant biometrics" encompasses a surprisingly broad range of measurable signals. Stomatal conductance, the degree to which the microscopic pores on leaf surfaces are open or closed, is one of the most information-rich indicators in plant physiology. Stomata regulate gas exchange and water loss, and their behavior reflects the plant's integrated response to water availability, light, temperature, and CO_2 concentration. A sensor mounted conformally on a leaf surface can track conductance in real time, providing a continuous readout of the plant's water status hours before wilting becomes visible.

Beyond stomata, flex PCB sensors can monitor stem diameter fluctuations driven by daily cycles of water uptake and loss, a phenomenon known as stem psychrometry. They can measure leaf temperature differentials that indicate evapotranspiration rates, track surface pH changes associated with disease presence, and detect volatile organic compound (VOC) signatures emitted by plants as chemical distress signals. Some platforms integrate electrochemical electrodes that measure ion concentrations directly in plant sap, providing a near real-time readout of nutritional status.

Why Flex PCBs Are the Enabling Technology

Rigid electronics have no place on a growing plant. Leaves flex in the wind and stems expand as they mature. Fruit swells over days and weeks. Any sensor platform that cannot conform to these dynamic fluctuations will either damage the tissue it contacts or lose contact and fail. Flex PCBs built on polyimide substrates, with serpentine or fractal conductor traces that accommodate deformation without fracturing, solve this problem elegantly.

The most advanced implementations go further, using ultra-thin substrates measuring 25 to 50 microns that are effectively transparent to the plant's normal mechanical behavior. Researchers at several agricultural universities have demonstrated sensor patches that can be adhered to leaf undersides for weeks without affecting photosynthesis rates,



Plant biometrics in a smart greenhouse.

stomatal function, or overall plant growth. The sensor becomes, in effect, invisible to the plant; present but not disruptive.

Wireless communication adds the final critical capability. Integrated near-field communication (NFC) or Bluetooth Low Energy (BLE) transceivers embedded within the flex circuit allow sensor data to be read without wired connections, enabling deployment across an entire field rather than in isolated laboratory conditions. Energy harvesting circuits that capture ambient light or even the electrochemical potential of the plant itself extend sensor operational life without battery replacement.

From Individual Plants to Field-scale Intelligence

The power of plant biometric sensors multiplies when deployed at scale. A single sensor tells you about one plant at one moment. A network of sensors distributed across a field tells you about spatial variability in soil moisture, microclimate differences between rows, the early spread pattern of a fungal infection, or the differential stress response of crop varieties to the same environmental conditions. That spatial and temporal density of data is simply not achievable through conventional scouting or satellite imagery.

When integrated with machine learning platforms, data streams from flex PCB sensor networks begin to support predictive, rather than reactive, crop management. Models trained on historical biometric signatures can flag early drought stress days before visible wilting, identify nutritional deficiencies based on ion flux patterns, and even distinguish between water stress and pathogen-induced stress. These conditions look similar on the surface but require entirely different interventions.

The Broader Implication for Agriculture

Precision agriculture has long promised the ability to apply exactly the right input—water, fertilizer, and pesticide—at exactly the right time and place. That promise has been partially fulfilled through satellite imagery, soil sensors, and weather modeling. Plant biometric flex sensors represent the missing layer: direct, real-time physiological data from the crop itself.

Conclusion

With global food demand projected to increase significantly by 2050, against a backdrop of potential water scarcity and climate volatility, the efficiency gains enabled by crop-level biometric monitoring are not merely agronomic; they are strategic. Flex PCBs, once confined to consumer electronics and medical devices, are finding their most consequential application in the fields that feed the world. The language of crops has always been there. We are finally learning to hear it. **I-CONNECT007**



Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs* and *Flex and Rigid-Flex Fundamentals*. He is the author of *Thermal Management: A Fabricator's Perspective, The Printed Circuit Designer's Guide to DFM Essentials*, and *The Companion Guide to Flex and Rigid-Flex Fundamentals*.

Little Streams Make Big Rivers: Creating an Industry Voice in Brussels ▶

There is a French saying I am fond of, and I used it to open my presentation in Paris: "Les petits ruisseaux font les grandes rivières," translated as "Little streams make big rivers." It is, to my mind, the whole of European government relations in just a few words. No single company, however large, and no single association, however active, changes the direction of European policy on its own.

Airbus Awarded Contract for Two Additional Galileo Satellites ▶

The European Space Agency (ESA), acting on behalf of the EU Agency for the Space Programme (EUSPA), under a delegation of the European Commission, has awarded Airbus a contract to build two additional second-generation Galileo satellites.

indie Selected for Space-Qualified Satellite Program ▶

indie Semiconductor, an automotive solutions innovator, today announced that superluminescent light-emitting diodes (SLEDs) from its photonics portfolio have completed final space qualification testing and have been selected for use in the Copernicus Hyperspectral Imaging Mission for the Environment (CHIME).

NASA Awards LeWiz Phase III Avionics Networking Contract ▶

LeWiz Communications, Inc. has been awarded a NASA Small Business Innovation Research (SBIR) Phase III contract to develop an advanced avionics networking system designed to improve communications and interoperability among spacecraft electronic systems.

Micron Invests in New York Critical Minerals Processing With Indium Corporation ▶

Micron Technology, Inc. announced an investment in critical minerals processing and high-purity materials manufacturing that is expected to support the expansion of Indium Corporation's capabilities in Central New York, restarting domestic processing of critical minerals essential to advanced semiconductor manufacturing.

Kratos Wins $33.6M NASA Contract for Nuclear-Powered Spacecraft System ▶

Kratos Defense & Security Solutions, Inc., a leading technology company in the defense, national security, and space domains, announced a $33.6 million contract award to design, build, and deliver the power conversion system for NASA's Space Reactor-1 Freedom. Kratos will be executing this work as a subcontractor to Sierra Lobo, Inc., under their NASA SpaceDOC III contract.

ESI Supplies Solar Array Components for Missile Defense Programs ▶

ESI Motion Inc. is proud to announce that it is supplying the Solar Array Drive Electronics (SADE) to Lockheed Martin for its work on the Space Development Agency's Proliferated Warfighter Space Architecture (PWSA) programs. Under the agreement, ESI Motion's SADE technology will support SDA as the PWSA is developed.

CIEL Defence, ASELSAN Sign MoU to Advance Canadian Defence Industrial Cooperation ▶

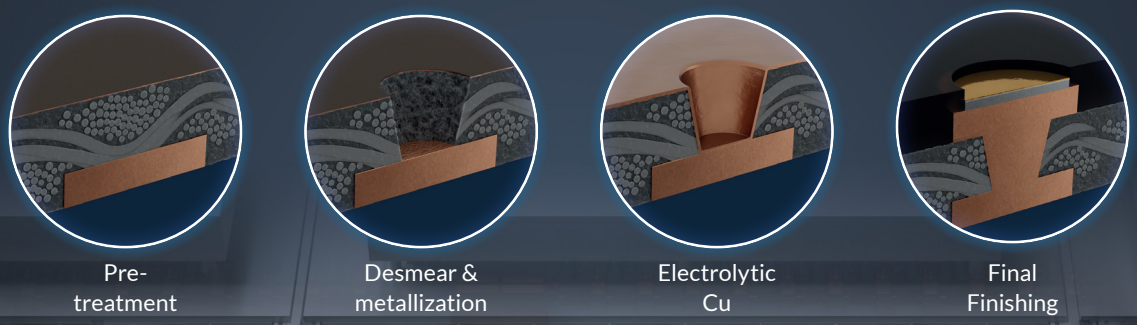
CIEL Defence Inc., the defence-focused investment arm of Canadian private investment firm CIEL Capital Inc., has signed a memorandum of understanding (MoU) with ASELSAN to explore opportunities to develop majority Canadian-owned defence industrial capabilities. The agreement was signed on Oct. 23, 2026.

Iris Technology Secures Strategic Investment from OceanSound Partners ▶

Iris Technology Corporation, a provider of radiation-hardened, space-qualified TRL 9 electronics for mission-critical national security space payloads, as well as rugged power management solutions for tactical military radio applications, announced the completion of a strategic investment from OceanSound Partners, a private equity firm that invests in technology and technology-enabled services companies serving aerospace, defense, government, and highly regulated end markets.

mks
Atotech

Why wet-process control is becoming a strategic lever



From adhesion to final finish – MKS' Atotech helps turn critical chemistry steps into yield, reliability, and manufacturing advantage

AI and HPC are reducing process margins in PCB, substrate, and advanced packaging manufacturing. As feature sizes shrink and materials diversify, achieving high yields increasingly depends on controlled surface conditions, reliable metallization, stable copper deposition, and robust final finishes. The integrated wet metallization process solutions from MKS' Atotech support these requirements. Solutions span from NovaBond® PX-S2 for low-roughness adhesion, over Printoganth® MV TP3 series for ultra-thin electroless copper seeding, and InPro® SAP7, for uniform blind microvia filling at high current densities, to Aurotech® G-Bond 3/ Plus, for ENIG performance in high-end HDI production. For manufacturers pursuing finer features with repeatable HVM results, wet-process control is no longer secondary; it is strategic.

To find out more, scan the QR code to the right.

info-atotech@mks.com

www.atotech.com

BY JAN PEDERSEN

Qualified Flexibility

More Choices We Can Trust



The laminate specified on your PCB drawing is unavailable. Another material can be delivered sooner, and your supplier asks whether you will accept it. What would you need to know before saying yes? We should have the evidence needed to answer before the question becomes urgent.

In my previous article, I discussed why advanced and conventional PCBs do not exist in separate supply chains. Even an ordinary six-layer board can be affected by competition for materials and production capacity elsewhere in the industry. Understanding that pressure is important.

At a seminar on 10 September, I highlighted three requirements for that ordinary PCB: qualified laminate and solder mask, and a qualified factory. Those requirements belong together. Having more material names to choose from is useful only when we know which combinations can deliver the performance and reliability our product needs.

"Qualified flexibility" is the freedom to choose among alternatives that have been evaluated and approved for the intended use. The work behind that confidence includes evaluation, testing, manufacturing experience, and clear documentation.

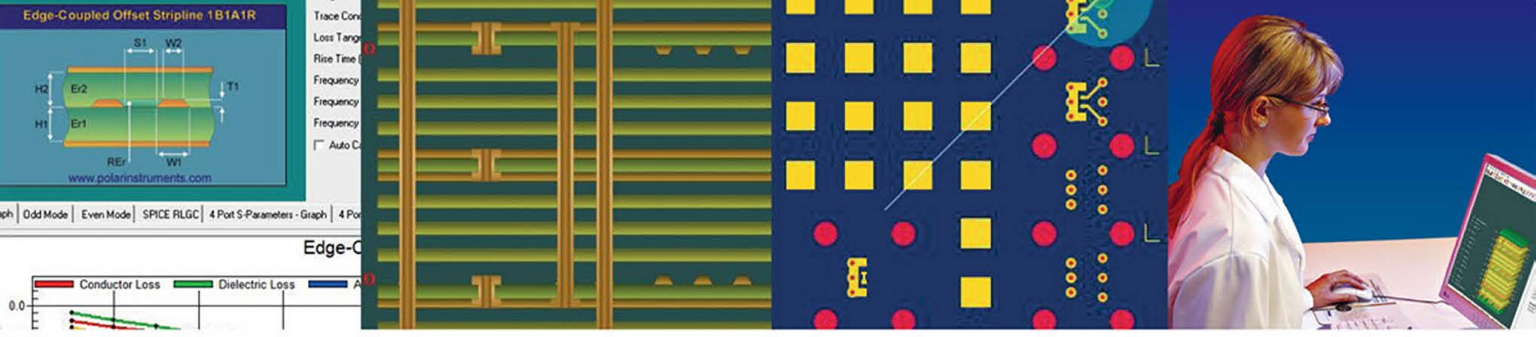
Qualification Creates the Freedom to Choose

Qualifying additional materials take effort, but the benefit is that engineering and purchasing can act on evidence when supply conditions change because we have already investigated the alternatives and agreed where and how they can be used.

Having both flexibility and qualification are essential for PCB fabricators. Flexibility without qualification equals risk. Qualification without flexibility equals vulnerability. To be successful, you need both. If only one suitable solution exists, we need to recognise that dependency and plan for it. If several solutions are possible, qualification gives us the confidence to use them.

Start With a Performance Window

Before comparing materials, we need to understand the application. What temperatures will the board experience during assembly and service? What



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

PCB Signal integrity tools for design & fabrication

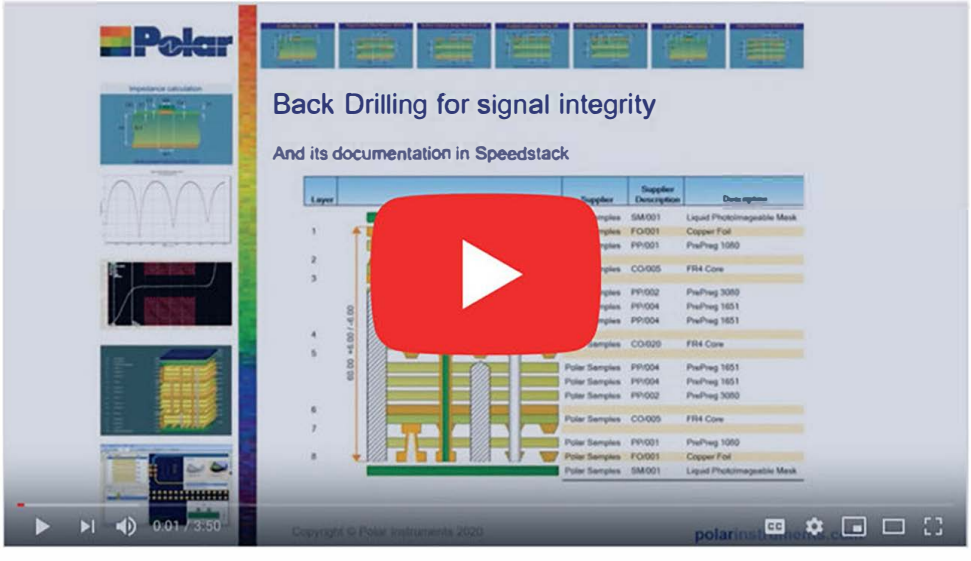
- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics

SUBSCRIBE



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

DOWNLOAD BOOK



polarinstruments.com

electrical performance is required? What operating environment and product lifetime must the board support? These and other examples in the application may bring other requirements that must also be considered. The answers will define the performance window: the parameters, acceptable limits, and operating conditions that determine whether a material is suitable.

For one product, thermal expansion and the ability to withstand assembly stresses may be the main concerns.[1] For another, dielectric properties and signal loss may be critical.[2] Moisture behaviour, insulation performance, and resistance to conductive anodic filament (CAF) formation may also need attention.[3] The selection should reflect the application. A Tg value cannot describe all of this. Glass transition temperature is one property; it does not by itself establish long-term thermal endurance or the reliability of a finished board.

We should also avoid copying every datasheet parameter into every specification. We need to identify which parameters and limits matter for the product's performance and reliability, and why they are required. To compare materials properly, we also need to understand the test methods and conditions behind the values. Two dielectric-constant values measured at different frequencies or using different methods can give a misleading comparison.[2]

IPC-4101 provides a recognised framework and important starting point for specifying laminate and prepreg, including the requirements in its specification sheets.[4] We still need to establish whether a candidate meets the requirements of our particular application and construction. Two materials can meet the same IPC-4101 slash sheet and still have different properties. Both may meet the specified limits, but that does not mean they are interchangeable in every PCB.

My suggestion is to define the performance window first, then identify and qualify materials that meet the requirements of that performance window. If a particular brand is essential, document the reason. If alternatives are acceptable, document the conditions.

Qualified by Whom, and for What?

In my presentation, I discussed several routes to qualification: IPC Validation Services, NCAB, and third-party laboratories.[5] Each can help establish confidence in a material. To understand a material qualification, we need to know who carried out the evaluation, who accepted the evidence, and what the approval covers.

Different organisations may be responsible for testing, reviewing the results and granting approval. A laboratory can produce the test evidence on which a customer or PCB supplier bases an approval. A qualification programme may combine testing with manufacturing-site audits. A supplier may qualify both the material and the factory processes in which it will be used.

Before approving the alternative for production, we need to check that the combined qualification evidence covers the requirements of the board we need to manufacture.

What IPC Material Qualification Tells Us

IPC Validation Services provides a formal route through the IPC-4101 Qualified Products List. The public listing identifies particular materials and specification sheets from suppliers including Arlon, Elite Material, Isola, TUC, and Ventec, with reports and dates. Unfortunately, the IPC-4101 QPL currently covers only a limited selection of laminates.[4] In my view, it cannot on its own provide the choices we need for the range of PCB applications we work with. If we restrict ourselves to listed materials, we may exclude suitable alternatives that have been qualified through other recognised routes.

This makes qualification by PCB suppliers, manufacturers, customers, and competent third-party laboratories an important part of qualified flexibility. The evidence must still match the application and the intended production process.

IPC's qualification approach also addresses manufacturing sites and processes. Its published explanation describes onsite audits for the locations manufacturing qualified products.[5] For a base-material listing, this means the facility manufacturing the laminate. It does not automatically establish that a downstream PCB factory has qualified the material for our board.

The reports deserve attention too. For example, the linked Arlon 33N report is explicitly labelled "Self-Tested Data."[6] It describes the test evidence submitted. IPC is still the organisation

“Before approving the alternative for production, we need to check that the combined qualification evidence covers the requirements of the board we need to manufacture.”

granting the QPL listing. This is why it is useful to distinguish who tests from who qualifies.

A QPL entry provides valuable evidence about the material and the requirements covered by the qualification. But we still need to establish how that material will perform in the proposed PCB construction and fabrication process. It is important to note that a material that is absent from the QPL may still be suitable. It may have been tested and qualified through another route.

Combining Material and PCB Factory Qualification

For me, the most complete practical basis for qualified flexibility is to qualify the material together with the PCB production facility and the processes that will use it. This brings the evidence closer to the product we are actually buying.

I did much of this qualification work during my time with NCAB, covering a broad range of laminates and solder masks, conducted alongside PCB factory qualification. Its public capability information describes a documented material qualification procedure and additional material approval and qualification by its Preferred Supplier List factories.[7,8]

The connection matters. We need confidence in the material’s properties and in the factory’s ability to process it consistently under the approved conditions.

Other PCB suppliers, manufacturers and customer qualification programs can establish this same connection, using internal expertise, external laboratories, or a combination. The principle can be applied wherever the necessary evidence, responsibilities and production controls are in place.

For laminate, you want to know whether the supporting evidence covers the proposed cores and prepregs, as well as the construction and processing conditions. What has the factory already manufactured with this material? Which reliability requirements has the testing shown it can meet? Are there conditions or limitations attached to the approval?

The answers determine whether a material is a usable alternative for this board. A familiar trade name, a matching slash sheet, or successful use elsewhere can help the evaluation, but we still need to check suitability at the factory that will produce the board.

The same applies to solder mask. The exact product must be considered alongside its application, imaging, development, and curing conditions, where applicable. Compatibility with the surface finish and assembly process also matters.

Taiyo’s PSR-4000 BW01DI technical data sheet illustrates the evidence available from a material supplier: processing guidance and reported adhesion, chemical resistance, insulation and thermal testing against IPC-SM-840E requirements.[9] Factory qualification checks whether the factory’s process can consistently achieve the required solder mask performance on the board.

This combination is what‘s most valuable: an identified material, a capable production facility, a controlled process, and evidence supporting their use together.

An Approval Has Boundaries and Needs an Owner

A factory’s approval should identify the technologies and manufacturing limits it can support. A material’s approval should establish where and under which conditions it may be used. Both approvals

“This combination is what‘s most valuable: an identified material, a capable production facility, a controlled process, and evidence supporting their use together.”

must cover the actual PCB construction.

Supplier qualification does not automatically authorise a material change in the customer’s product. Any required customer approval must also be secured before the alternative is released for production.

Qualification also needs continuing attention. Changes in materials, equipment, processing conditions, or product requirements can affect the basis of an earlier decision. The responsible organisation must decide whether the existing evidence still applies or whether further evaluation is needed.

In my presentation, I also discussed capability approval, change control, audits and ongoing process monitoring.[8] They illustrate how an approval can be maintained throughout production. Other organisations may structure the work differently, but they need to preserve the same connection between the approved conditions and what happens in the factory.

For the customer, the useful question becomes who has approved this combination of material, construction, and production facility, and what evidence supports that decision? With those answers, we can decide whether the alternative is ready to use.

Make the Approval Usable Before the Shortage

Consider a simple example: a six-layer industrial PCB. Material A is the established choice. Material B appears to satisfy the relevant requirements and is available sooner.

If B has only been compared on paper, important work may still be needed. The team may still need to check the available cores and prepregs, confirm the stackup, review the factory’s experience, obtain test evidence, and secure product approval.

Now suppose that work was completed earlier. The alternative construction is documented, the supporting evidence is accepted, and the customer has approved its use under defined conditions. The supplier can then use the approved alternative, following the agreed requirements for notification and traceability. That is where the earlier qualification work pays off.

For each alternative, the record should make clear what is approved, where it may be manufactured, what evidence supports it, and which changes require further review. Engineering, quality, purchasing, and the supplier must have the same understanding.

The phrase “or equivalent” leaves too much open unless equivalence and the approval process have been defined. A long list of approved materials also offers little flexibility if nobody knows which ones can be used for the product.

To track the materials used, I recommend assigning a unique PCB article number to each approved material choice or to each defined combination of materials. The specification linked to each article number should identify the exact laminate, prepreg, and solder mask, as applicable, together with the approved construction and production facilities. Production and delivery records must link that article number to the actual materials, material batches, and factory used. This gives us the freedom to use qualified alternatives while keeping a clear record of how each batch of boards was manufactured.

We also need to ask whether the alternatives reduce our dependence on the material or capacity causing the delay. Two PCB factories waiting for the same constrained material may offer little relief. Even different laminate brands can share upstream constraints. Qualification establishes technical acceptability; supply-chain understanding tells us how much resilience the alternative adds.

From Trusted Alternatives to Available Products

Qualified flexibility gives purchasing more room to act because the technical work and approval decisions have been made in advance. It can reduce the delay between identifying a supply problem and implementing an acceptable response. It cannot guarantee that capacity or material will always be available.

The value lies in being prepared. We know our options, understand their limits, and have the evidence to support their use.

There is also an earlier opportunity to improve those options. Design decisions determine whether the product can make practical use of the materials and factories we have qualified. That is the subject of my next article: Design for Availability. Qualified flexibility establishes which alternatives we can trust. Design for availability helps ensure our designs can use them. When the supplier calls to say the specified laminate is unavailable, are we ready to offer an informed answer? **I-CONNECT007**

References

1. "Lead Free Assembly: A Practical Tool for Laminate Materials Selection," by Erik J. Bergum and David Humby, Isola, 2005.
2. "Digging Deeper into Dielectric Constant for PCB Materials," Rogers Corporation, 2020.
3. "Material & Process Influences on Conductive Anodic Filamentation (CAF)," Isola.
4. "IPC Validation Services—Qualified Products List (QPL), IPC-4101," IPC, accessed Oct. 9, 2026.
5. "Validation Services—Frequently Asked Questions," Global Electronics Association.
6. "Test Report (Self-Tested Data): Arlon Product 33N, IPC-4101/40," Arlon, Feb. 17, 2023.
7. "Technical Capability," NCAB Group.
8. "Design for Availability: Building PCB Supply Chain Resilience Through Smarter Design," by Jan Pedersen, NCAB Norway, Sept. 10, 2026.
9. "PSR-4000 BW01DI Technical Data Sheet," Taiyo America, revised Sept. 22, 2017.



Jan Pedersen is an independent PCB technology and circularity consultant, and industry writer.

BELOW THE SURFACE

From Substrate to System: Why Integration Is the Real RF Breakthrough

BY CHANDRA GUPTA, REMTEC

In my last column, I described the ceramic substrate itself, and why material choice matters so deeply in RF, microwave, and millimeter-wave applications. Dielectric constant, loss tangent, thermal conductivity, stability across temperature, and frequency are the physical rules that govern whether a signal arrives intact or collapses into noise.

But focusing only on the substrate is a bit like judging a city solely by its geology. Bedrock matters, of course. Without it, nothing stands. But it's how a city emerges from the bedrock: roads connect, utilities are routed, neighborhoods interact, and systems layer on top of one another. RF performance works the same way.

Once a ceramic substrate is chosen, the real story is in how that material evolves into a package, and how packages evolve into modules. It's where integration becomes the defining variable and where many RF designs either succeed quietly or fail expensively.

The Substrate Is the Beginning, Not the Solution

A ceramic substrate on its own is inert. It doesn't amplify signals, filter noise, or dissipate heat…

Continue reading this column here.

Factory Data Is Everywhere. Operational Insight Is Not.

For Flex PCB manufacturers, stronger performance depends on turning fragmented equipment data into information people can understand and act on.

Flex PCB manufacturers have spent decades improving performance through faster equipment, tighter process control, higher yields, automation, and capital investment. These advances have created increasingly sophisticated production environments, but they have also introduced a new challenge.

Modern factories generate enormous volumes of data. Yet manufacturing teams do not always have a clear view of what that information means for overall performance. The next opportunity for improvement may not begin with another machine or major process change. It may begin with understanding what is already happening across the factory.

The Difference Between Data and Visibility

Production equipment continuously generates machine states, process parameters, alarms, utilization information, and subsystem data. The challenge is rarely a lack of information. The greater challenge is fragmentation.

Data may be distributed across individual machines, log files, spreadsheets, and isolated dashboards. Engineers can often find what they need, but locating, interpreting, and correlating it can take considerable time.

A machine can report what happened within that system. A broader factory view can help teams understand how patterns and conditions affect the entire operation.

The Small Losses That Add Up

Major equipment failures receive immediate attention. Smaller performance losses are less obvious.

A few minutes may be lost between jobs. A laser system may wait for an operator or production material. An abnormal condition may not receive immediate attention. Engineers may repeatedly search through logs to diagnose a familiar issue.

Individually, these events may appear insignificant. Collectively, they can consume meaningful production time. The business problem is not simply downtime. It is also the time required to recognize a condition, understand its cause, make a decision, and take corrective action.

When those steps are slow or disconnected, preventable losses can become accepted as normal operating behavior.

From Managing Downtime to Engineering Uptime

Uptime has traditionally been treated as a maintenance measurement: equipment is available or unavailable, running or stopped. That definition is useful, but incomplete.

Uptime also reflects how effectively equipment, people, information, and decisions work together. It is affected when operators cannot recognize abnormal conditions quickly, when engineers must assemble information from multiple sources, and when recurring events are treated independently instead of as a pattern.

The goal is not simply to create more dashboards. More data without context can create additional noise. The goal is actionable visibility: presenting the right information to the right people early enough to improve the outcome.

A More Purpose-Built Approach

General-purpose software platforms can offer extensive capabilities, but they may require significant discovery, customization, and application knowledge before delivering value in a specialized Flex PCB production environment.

A purpose-built approach starts with an understanding of the laser systems, drilling applications, operating conditions, and production decisions that shape performance. Software created by the people who design the laser systems and understand Flex PCB via drilling can provide important context for the data those systems generate.

As Flex PCB manufacturing becomes more connected, advantage will belong to factories that can turn data into understanding, action, and sustained improvement.

The factories that succeed will not simply measure uptime after production has been lost.

They will engineer it.

by 

Tyler Wilson
Product Marketing Manager, MKS Inc.

Turn factory data into operational insight. Discover DFS at go.esi.com/dfs

mks | ESI

mks
ESI

ENGINEER UPTIME.

Turn factory data into clearer insight, faster action, and stronger performance.



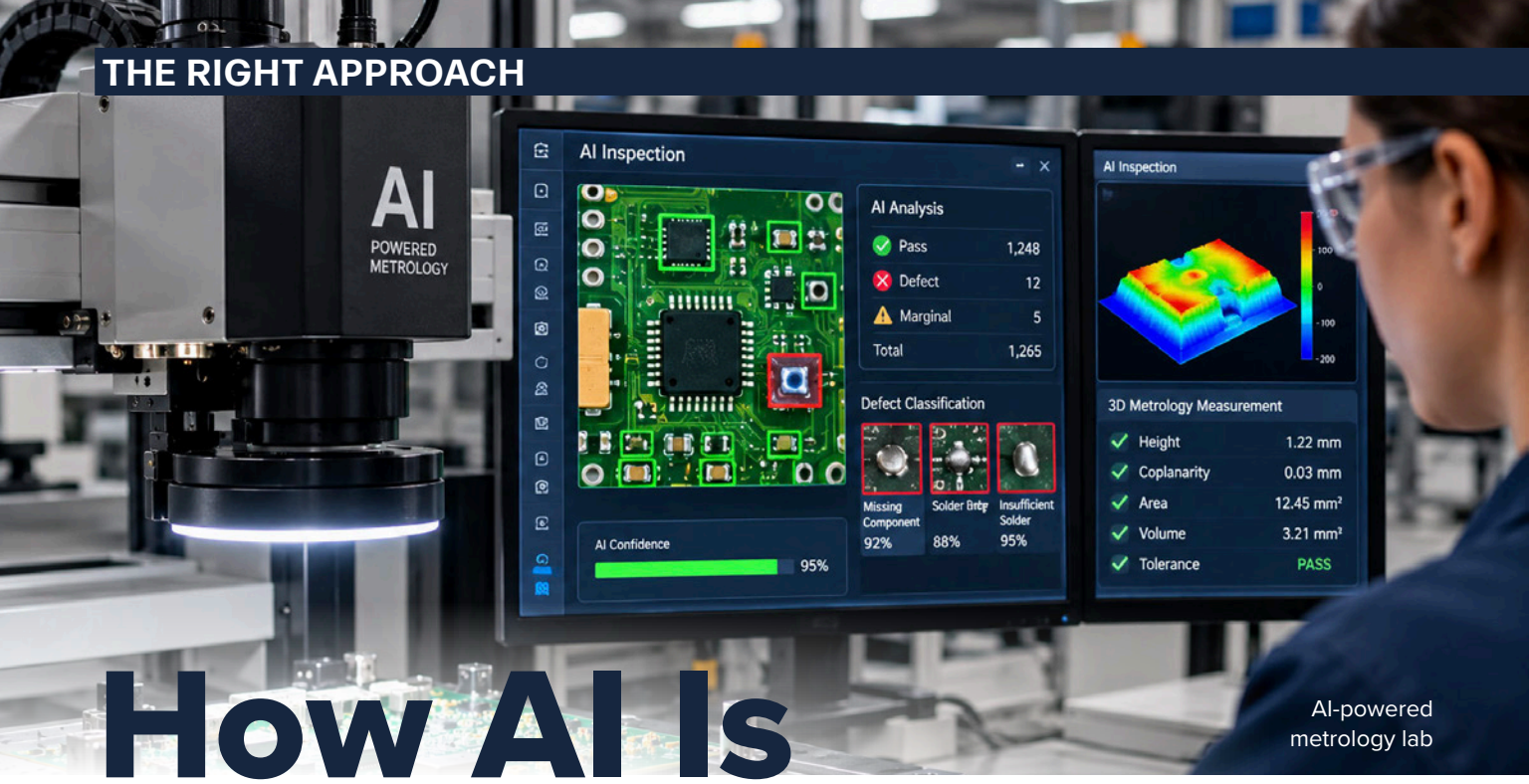
SEE CLEARLY.
Bring critical system and factory conditions into view.

RESPOND FASTER.
Recognize important conditions and act sooner.

IMPROVE PERFORMANCE.
Turn operational insight into better factory outcomes.

Purpose-built by the people who design the laser systems and understand Flex PCB via drilling.

Discover DFS. Visit go.esi.com/dfs



AI-powered metrology lab

How AI Is Changing Quality Metrology

BY STEVE WILLIAMS, THE RIGHT APPROACH CONSULTING

Metrology, the science of measurement, has always been the foundation of quality assurance. Before a part can be accepted or rejected, a process can be declared in control or out of it, or a tolerance can be called met or missed, something must be measured with sufficient accuracy and certainty to make that judgment defensible.

Augmenting Judgment, Not Replacing It

For most of manufacturing history, that "something" was a skilled metrologist with calibrated instruments, a stable environment, and years of accumulated judgment about the numbers.

AI is not replacing that judgment. It is, however, augmenting it in ways that are fundamentally changing what metrology can do, how fast it can do it, and how much of the manufacturing process it can reach. The transformation is unfolding across three intersecting dimensions: the automation of measurement itself, the interpretation of measurement data at scale, and the predictive application of that data to prevent nonconformance before it occurs.

From Sampling to 100% Inspection

Traditional quality metrology has always operated under a statistical compromise. Measuring every part in a production run is expensive, slow, and often physically destructive, so quality professionals sample. They measure a fraction of output, apply statistical process control, and infer the population from the sample. The inference is sound under stable process conditions, but it carries irreducible risk: Nonconforming parts exist in unmeasured gaps between sample points, and the gap between detection and correction allows defects to propagate.

AI-powered machine vision systems are collapsing that compromise. Computer vision models trained on hundreds of thousands of labeled inspection images can evaluate surface finish, dimensional conformance, solder joint quality, weld integrity, and assembly completeness at line speed; not on a sample of parts, but on every part and cycle.

In electronics manufacturing, AI vision systems now routinely perform automated optical inspection at throughput rates that no human inspector could match, flagging anomalies that fall below the threshold of human visual acuity. The effective sample size has gone from a statistical fraction to 100%, without a proportional increase in inspection cost or cycle time.

Dimensional Metrology and the CMM Revolution

Coordinate measuring machines have been the backbone of dimensional metrology for decades. A CMM probes a physical part at discrete points, compares the coordinates to a nominal CAD model, and reports deviations. The process is accurate, traceable, and slow; a complex aerospace part may require hours of CMM time, which makes 100% CMM inspection impractical for high-volume production.

AI is transforming CMM utilization in two ways. The first is measurement path optimization, where machine learning algorithms analyze the geometry of a part and the tolerance requirements of its critical features to generate optimal probe paths that maximize information per unit of machine time. Inspection cycles that previously took 2 hours are now completed in 40 minutes without sacrificing measurement confidence.

The second transformation is the application of AI to structured light and laser scanning systems, which capture point cloud data of an entire part surface in seconds. AI models interpret that dense geometric data, compensating for fixture variation, identifying form errors, and extracting GD&T-compliant measurements, at a speed and completeness that conventional software cannot approach.

Predictive Metrology: Measuring What Has Not Yet Gone Wrong

The most consequential shift AI brings to metrology is conceptual, not technological. Traditional metrology is retrospective; it measures what has been made and determines whether it conforms. Predictive metrology, enabled by AI analysis of continuous measurement data streams, asks a different question: Given what the process measurements are telling us right now, what will the next parts look like, and are we trending toward nonconformance before we get there?

This capability emerges from the fusion of in-process sensor data, spindle load, coolant temperature, vibration signatures, and tool wear, with part measurement history fed into machine learning models that have learned the relationship between process variables and dimensional outcomes for a specific machine and part family.

The model does not wait for a part to go out of tolerance. It detects the process signature that precedes that outcome and triggers a corrective intervention: a tool offset adjustment, a coolant flow change, or a maintenance alert, while parts are still conforming. In high-value manufacturing environments such as aerospace, medical device, and defense electronics, this shift from detection to prevention represents a fundamental change in the economics of quality.

Calibration, Uncertainty, and the Human Expert

AI does not eliminate the foundational requirements of metrology. Measurement traceability, the unbroken chain of calibration that links every instrument reading back to national and international measurement standards, remains as essential as ever. An AI vision system trained on mislabeled data or operated with an uncalibrated illumination source will produce confident incorrect results at machine speed, which is considerably more dangerous than a slow uncertain answer from a careful inspector. The introduction of AI into metrology workflows raises the stakes on calibration discipline, measurement uncertainty analysis, and the competence of the metrologists who design, validate, and maintain AI-augmented measurement systems.

Future State

What AI is genuinely changing is the ceiling of what metrology can accomplish within the constraints of production economics. The expert metrologist of the near future will spend less time at the eyepiece of a comparator and more time designing measurement strategies, validating AI model performance, interpreting the process intelligence generated by continuous measurement, and translating that intelligence into engineering action. The measurement itself—precise, fast, and ubiquitous—will increasingly be the work of algorithms. The judgment about what to do with what was found will remain irreducibly human. **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**.

A Career 'Turkey' Story for November

BY DON BALL, CHEMCUT



When it was first put to me that I share a "turkey" from my career and show how it ultimately led to a better understanding of the technology, customer relations, and how to learn from mistakes, I had some reservations. It's not that I haven't made many mistakes. I've always admitted them (most of the time, anyway), but have never had the desire to prolong the experience beyond the initial admission of guilt.

They say a smart person learns from their mistakes, and by that standard, I am very well educated indeed. However, it was rightly pointed out that mistakes provide learning experiences, and it is a noble thing to share those learning experiences so that others may benefit. With that in mind, here goes.

In the mid-1980s, we had an inquiry from a company that provided brass parts to many manufacturers. They had been producing their parts by stamping, but they saw their business was changing. Instead of long runs of standardized parts, they were getting orders for short runs of many different parts. Stamping dies was expensive, and they wondered if the parts could be etched.

We told them, of course, they could and offered to demonstrate, since this was new technology for them. They visited with some of their parts, which we duly etched for them. They were delighted, and I was asked to calculate how many etch chambers would be needed to handle their business volume. Well, this was an easy assignment since we had just etched the parts and knew exactly what the etch rates were. It was a quick calculation.

I reported that they needed three etch chambers. The quotes were prepared and accepted, and the equipment was purchased, built, delivered, and installed.

Things went well for the first several weeks. Then the customer asked why the etch rate was starting to fall off. We went through the regular troubleshooting steps (plugged nozzles, low free acid, etc.), but nothing helped. While we were trying to figure out what was wrong, the etch rate stabilized but was 30% slower than the new cupric etch bath they had started with, and they were no longer able to keep up with their production.

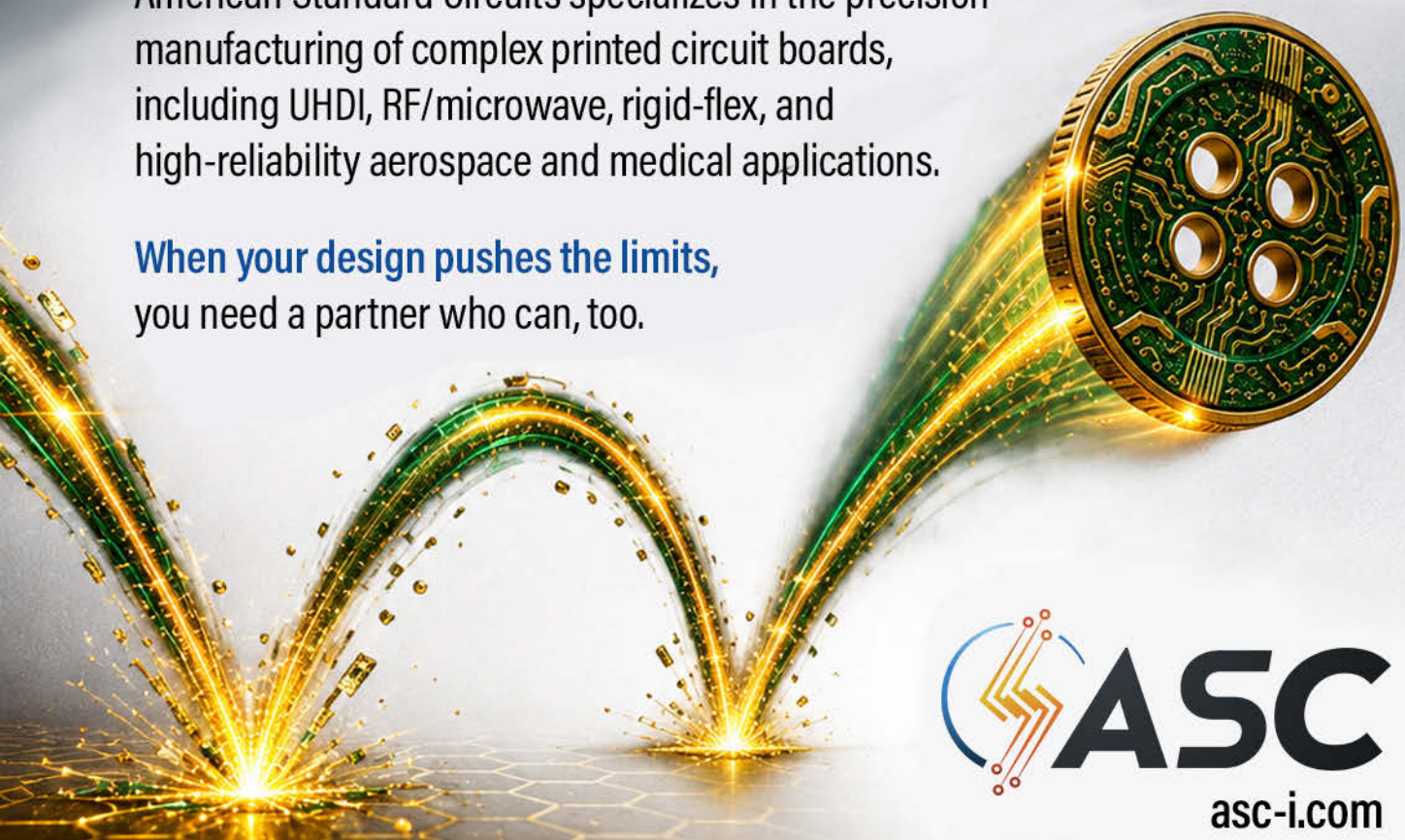
Stressed By TIGHT Tolerances?



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

As soon as I heard the 30% slowdown in etch rate, I had that sinking feeling: I knew what the problem was and that it was caused by a mistake I had made. I had forgotten to consider that brass was around 70% copper and 30% zinc. With a regenerated cupric chloride bath and etching nothing but brass, the metallic composition of the etch bath soon matched the composition of the alloy being etched.

Once the etch bath reached a 70% copper/30% zinc composition, the etch rate stabilized at 70% of that of a new bath. I had done the initial calculations based on our demonstration tests with an etch bath that had no zinc.

Recalculating with zinc accounted for showed they needed four etch chambers to keep up with production once the bath stabilized, not three. Oops.

Fortunately, the customer was very understanding about the whole thing, especially since we provided an additional etch chamber and installation at our expense, and we remained in their good graces for many years after. Also fortunate for me, my company culture did not require immediate dismissal for costly mistakes. They just wanted you to learn from those mistakes and not do it again.

So, it was forgiven, but don't think for a minute it was forgotten. For years, I was reminded of that mistake, usually with great glee from the person doing the reminding. Eventually it was forgotten, but likely only because I'm the only one left with the company that still remembers that incident; everyone else has retired. There are some benefits to getting older, after all, in that you can outlive some of your mistakes.

So, what did I take from this rather serious screw-up? First, we obviously sell equipment to etch metals other than copper, and the composition of those metals can have a major effect on the etch chemistry over time, especially if chemical regeneration is involved. Double-check your etch parameters before you calculate.

Second, the customer's reaction to the mistake showed that it is better to come clean when you make a mistake and not try to weasel out of the blame. The customer was, of course, upset but our technical sales manager got me on a conference call with the customer and readily admitted that it was our mistake and we would rectify it at our cost. The customer immediately calmed down, and we were able to set a timeline for delivery and installation of an extra etcher that was satisfactory for both companies without a lot of acrimony.

The whole thing turned into a running joke between their process engineer and me for several years thereafter.

Lastly, I learned that even I (gasp) could make a major mistake with all the best intentions in the world. I was pretty full of myself at that point in my career (I was in my mid-30s) and full of confidence that I had mastered the technology needed for my job. This was a wake-up call that knocked my confidence down a peg or two.

In all honesty, this was a good thing at that point in time. Did that experience keep me from making more big mistakes during my career? Nope. Fortunately, most of them have been caught internally before escaping into the wild. Just last week I calculated a system for a customer to do one panel a minute, although the specification clearly indicated that the desired production rate was two panels per minute. That one was caught by our sales engineer before the quote was made, so there was no problem, but it reminded me that I can still make serious errors despite all due diligence.

I have also found that admitting my mistakes has made me more tolerant of others' honest mistakes, especially from new hires and interns. I think making mistakes is part of the learning process, especially when they make the same mistakes I did when I was learning the business. A calm approach to explaining those mistakes and showing them how to fix them is much more productive than ranting and raving. Careless or lazy mistakes are one thing, but if everyone is afraid of making mistakes, innovation stagnates and newer, more efficient approaches to business are never tried. **I-CONNECT007**



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

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.

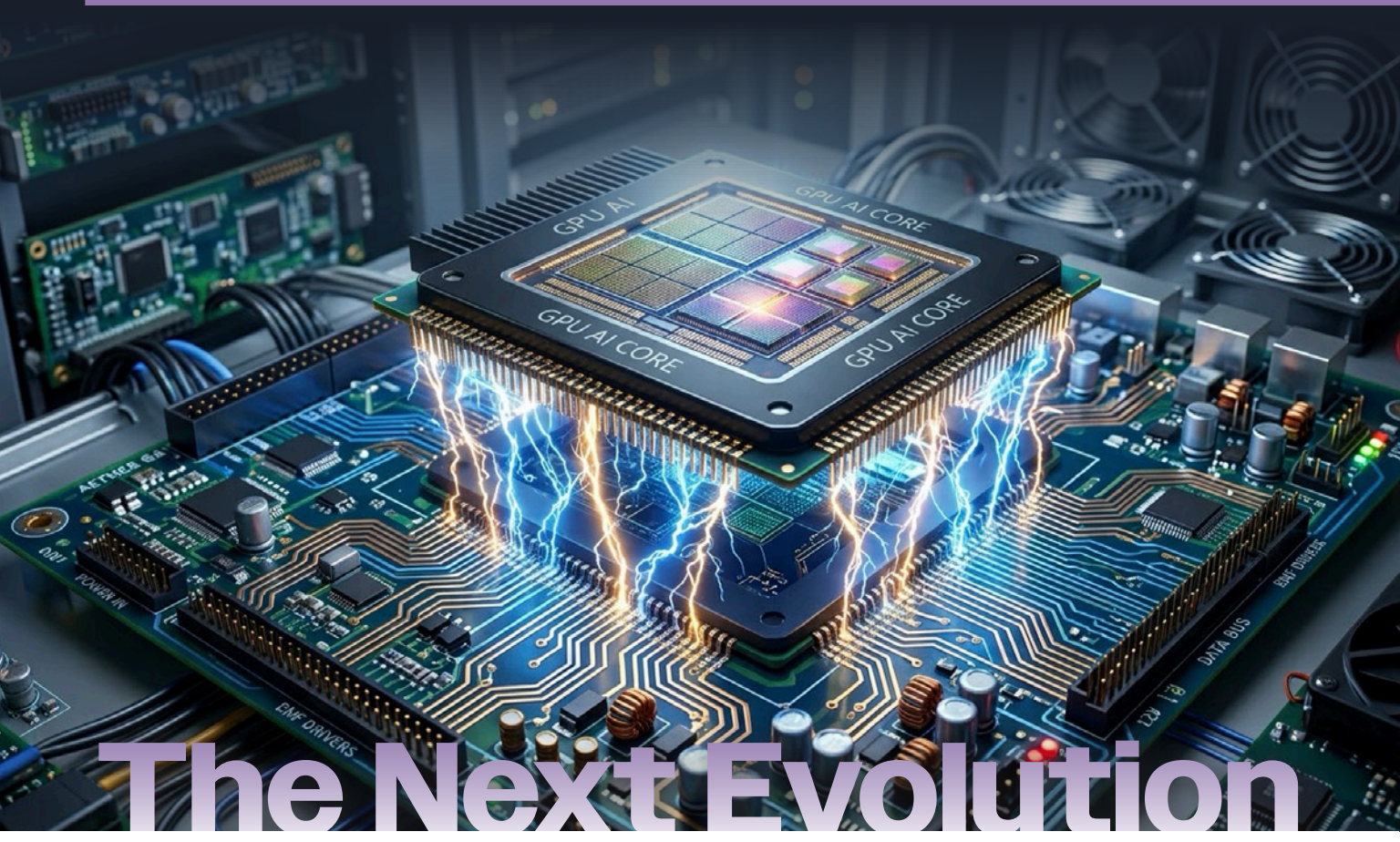
PACKAGING APPLICATIONS & EMBEDDED GUIDANCE SUITE

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.



The Next Evolution of Power Delivery for AI Data Center PCBs

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

AI-driven GPUs have pushed conventional PCB power delivery networks to their limits. As detailed in my previous column, achieving micro-ohm AC impedance requires extreme measures: multiplane stackups, heavy copper, dense via lattices, embedded capacitance, and meticulous power distribution network (PDN) optimization.

Modern accelerators consume 700–1,000W at only 0.85V, exposing the fundamental weakness of lateral PDNs. Even with these aggressive techniques, the lateral architecture used in devices such as NVIDIA's H100/H200 GPUs is now hitting a hard physical ceiling. In this month's column, I focus on the next evolutionary step in AI power delivery architecture.

Electromagnetic Fields Shape Power Delivery: How It Really Works

When a processor draws 1 kA, the intuitive picture is electrons surging through copper like water in a pipe. But at those magnitudes, that mental model breaks down. What is actually being delivered to the die is electromagnetic energy. The copper merely guides the boundary conditions; the real power flow occurs in the electric and magnetic fields surrounding the conductors. Power lives in the fields, not in the metal. The actual energy flow is described by the Poynting Vector ($S = E \times H$) that points through the dielectric between conductors, not inside the copper itself. The copper's thickness

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

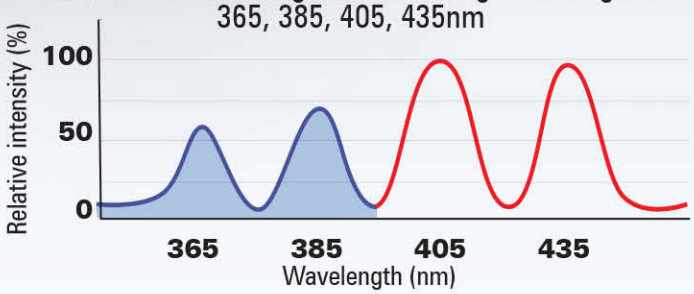
4 Smart Automation
Seamless integration with MES systems

5 Excellent Product Traceability
Multiple tracking methods to enhance QC

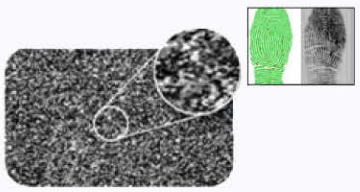
6 User-friendly Interface
Supports industry-standard data formats

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

only determines how efficiently those fields can form, propagate, and deliver energy to a processor.

High current is, in reality, an intense magnetic field. A 1 kA rail creates a very strong H-field encircling the conductor. The return current in the adjacent plane creates an opposing field, and together they form a tightly coupled transmission structure. The inductance we fight in the PDN is literally the energy stored in these magnetic fields.

Voltage, conversely, is the electric field established between conductors. The E-field exists between the power and ground planes. At sub-1V, the field strength is modest, but because the planes are closely coupled and the area is large, there is significant stored electric energy or planar capacitance.

The PDN behaves like an electromagnetic waveguide rather than a stack of copper sheets. Its role is to shape the EM fields so energy reaches the silicon efficiently and with minimal loss. At high switching speeds, it functions as a fully distributed EM structure, not a simple lumped network.

- Current steps launch field changes that propagate through the structure.
- Any discontinuity (splits, gaps, long detours) distorts the EM fields, causing reflections, resonances, and droop.
- Micro-ohm impedance ensures the EM fields reconfigure fast enough and smoothly enough to support those current transients.

Losses are field phenomena as well. I^2R loss results from fields driving current through finite-resistance conductors, converting energy into heat. Inductive droop arises from magnetic field energy resisting rapid change in current (L x di/dt).

So, when a processor pulls 1 kA at 0.85V, what you're really managing is a high-density electromagnetic energy pipeline: controlling the space in which the fields form, how they couple, how quickly they can change, and how little they dissipate on the way. The "micro-ohm PDN" is the physical manifestation of a well-controlled EM field environment around that device.

Lateral Power Delivery Limits

For decades, processors have relied on multiphase voltage regulators arranged closely around the package, feeding power laterally on planes through the PCB. This architecture is mature and well understood, but it is ultimately constrained by physics. As processor current climbs into the hundreds of amps, the PDN's resistive and inductive losses rise sharply. Even with heavy copper, optimized stackups, and dense via fields, lateral delivery forces power to travel through long, often narrow, winding paths before reaching the die.

Those extended routes accumulate parasitic resistance and inductance, eroding efficiency and slowing the PDN's transient response. In AI systems (where current demand can swing violently within

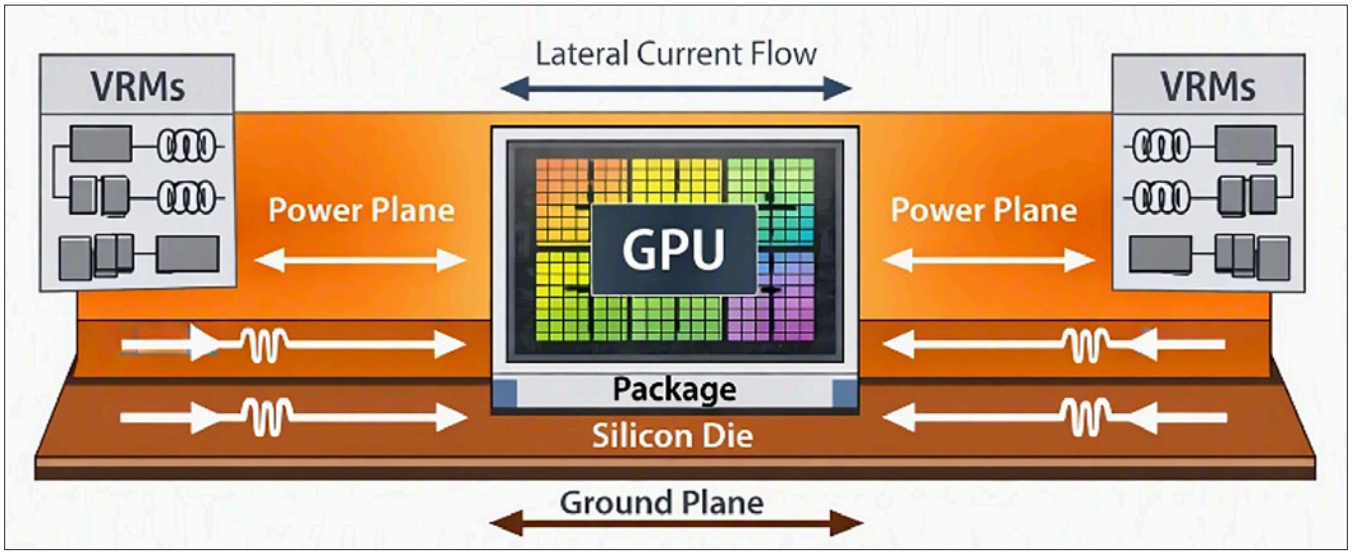


Figure 1: Lateral power delivery.

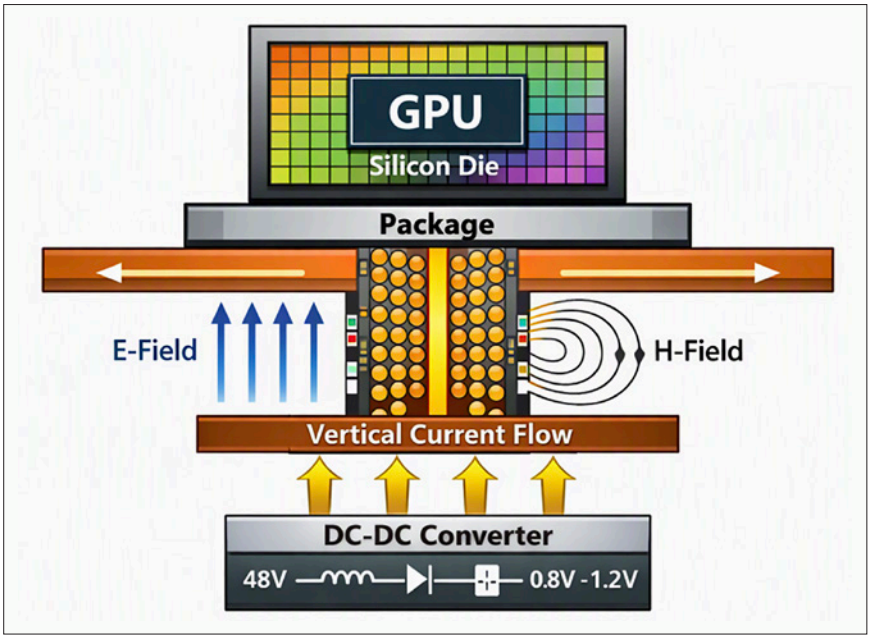


Figure 2: Power-on-Package employing vertical power delivery.

nanoseconds), these parasitics become a critical liability. Lateral PDNs (Figure 1) simply cannot react fast enough, nor can they sustain the escalating current densities demanded by modern accelerators.

Enter Vertical Power Delivery

Power-on-Package (PoP) overturns the traditional PDN model by collapsing the final stage of power delivery from a lateral, centimeter-scale copper network into a tightly coupled, millimeter-scale vertical structure beneath the processor. By relocating the DC-DC conversion stage to the backside of the PCB and feeding the SoC through dense via arrays, PoP removes the resistive and inductive penalties of lateral routing and enables the multi-kiloamp current levels required by next-generation AI accelerators. This vertical architecture transforms the PDN into a field-controlled energy conduit, reducing loop area and strengthening field coupling. Figure 2 illustrates how PoP collapses the PDN into a short vertical column, eliminating the lateral last inch.

At the core of PoP is vertical power delivery (VPD), which delivers energy directly through the substrate rather than across the PCB. Instead of forcing high current to meander through copper planes and via fields, VPD drives power straight into the die through large-diameter (12–20 mil) via arrays. This dramatically improves efficiency, power density, and transient response.

At the system level, VPD distributes 48V across the PCB, a far more manageable voltage for long-distance transport. The final DC-DC conversion is then performed directly beneath the processor package. By keeping the heavy current in a short, vertical column rather than sprawling lateral planes, VPD eliminates the resistive and inductive penalties that plague traditional PDNs.

The physics is simple: Raise the delivery voltage to 48V, and the current required at the board level drops proportionally. Lower current means lower I^2R loss, lower heating, lower copper mass, and dramatically improved PDN impedance.

Vertical Power Delivery Challenges

Thermal: PoP changes the heat paths, pushing heat up into the substrate rather than spreading in the planes. Therefore, there is a reliance on backside cooling technologies.

Mechanical: Driving hundreds or thousands of amps through large-diameter via arrays introduces mechanical stresses and reliability considerations. These vias are no longer sparse interconnects; they form a dense, load-bearing column that must survive extreme thermal, electrical, and mechanical conditions.

Embedded components: Embedded capacitors and inductors must be placed inside the substrate, creating active, integrated energy delivery.

Electromagnetic: Every di/dt event is an electromagnetic shockwave. When current jumps by hundreds of amps in a few nanoseconds, the magnetic field around the PDN must collapse and reform at the same speed.

Supply chain: The shift to PoP/VPD requires a coordinated supply chain capable of delivering advanced substrates, backside power modules, embedded passives, and high-voltage distribution infrastructure.

PCB Design Implications

VPD also imposes a profound shift in the PCB design, redefining stackup formulation, via architecture, current-path geometry, and overall reliability margins. Power distribution is forced into the Z-axis, with high current delivered through dense via farms beneath the SoC, making the PCB an active element of the PDN rather than a passive carrier. This elevates the importance of return-path integrity under the package and requires tightly coupled reference planes within the PoP footprint.

Designers must accommodate high-current vertical conduction, ultra low inductance power paths, dense escape routing, and uninterrupted reference plane continuity within a highly constrained footprint. The top layer is consumed by BGA breakout and memory routing, while VPD occupies inner layers with power via arrays, effectively eliminating two to three routing layers around the device.

Escape channels rely on microvias, skip-layer vias, and buried vias, and the high-speed interfaces must maintain strict reference plane continuity and avoid the power via zones. The primary current path becomes the via-in-pad array, with hundreds of filled and capped microvias operating in parallel. Each BGA power and ground ball may require multiple vias. The via farms are enclosed by densely stitched ground vias to maintain a stable return path. These ground vias form a continuous cylindrical return wall, compressing the magnetic loop.

Power delivery is fundamentally an electromagnetic problem, not a copper-routing exercise. This shift marks a decisive architectural evolution: from planar copper distribution to field-controlled vertical energy transport. As AI workloads surge toward multi-kilowatt and multi-kiloamp regimes, vertical power delivery becomes not just an innovation but a necessity. It's the foundation for the next generation of efficient, high-density computing systems. The roadmap is clear: The future of AI power delivery is vertical. **I-CONNECT007**

Resources

- Beyond Design: Micro-ohm Power Delivery Network for AI-driven GPUs by Barry Olney.
- "Infineon Announces Collaboration with NVIDIA on Power Delivery Chips for Future Server Racks."
- "Vertical power delivery reduces losses in AI processor designs" by Bill Pelletier.
- "Vertical power delivery structure for ultra-high current applications," tdcommons.org.
- Image and diagrams generated by AI



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**.



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

You Can't Buy a Baseline

BY SEAN PATTERSON, CROSSGEN AI

Nobody in a PCB shop changes a plating chemistry because a vendor says it "feels better." You run a sample, compare it with what you had, and look at the numbers. If they move in the right direction, you write the change into the process. If they don't, you change it back.

That discipline is so normal in our industry that most of us barely notice it anymore, yet it is missing from almost every AI effort I see.

AI usually enters a shop when someone tries a tool, gets an impressive answer, and tells everyone it's great. A few people start using it, some eventually stop, and six months later the question comes up in a meeting: Is this actually helping?

Nobody knows, because there was never a "before" to compare it with. The whole effort runs on feelings, which is the one thing we would never accept on the floor.

That is why I keep saying don't buy AI, learn AI. You can buy a license, but you cannot buy a baseline. That baseline has to come from your process, your people, and your numbers, and only you can collect it.

Treat Every AI Habit Like a Process Change

The good news is that fabricators already know how to do this. We run SPC, use change control, and understand the difference between a trend and a one-off, so the next step is to take the habits we already trust and point them at AI.

Start with one piece of recurring work, not something broad like "AI for quality," but one specific task. Maybe it is the RFQ review that takes up the first half of every Monday for your sales engineer, the shift handoff summary that costs the outgoing lead 20 minutes, or the first draft of a corrective action report.

Before anyone touches AI, write down what that work looks like today. Track how long it takes, how often something gets missed or sent back, and how many times a customer comes back with a question the RFQ review should have caught. You do not need a complicated system to do this.

Then change one thing by bringing AI into that task in the same way each time for the same number of weeks. When the trial is over, compare the results. Did the review get faster, did fewer questions come back, or did the handoff catch the things that used to slip through?

If the numbers move in the right direction, keep the change and write it down so the next person does it the same way. Stop if they don't. That is not a failure, but rather the experiment doing its job, and it keeps you from building a habit around something that does not work.

Measure What the Work Is For

Speed is one of the easiest numbers to measure, but it is also the one most likely to fool people. AI will almost always make a first draft faster, though



the real question is whether the finished work got better or whether someone downstream is now spending the saved time fixing it.

That is why you need to measure what the work is actually for. For an RFQ review, look at missed requirements and quote revisions. For a shift handoff, track how often the incoming shift has to call someone and ask what happened. For a corrective action report, measure how often the customer sends it back.

Keep the guardrails where they belong, too. AI can help draft, summarize, organize, and compare, but it does not set process parameters or make disposition calls. Measuring its performance does not change that. It simply makes it easier to prove where the line belongs.

Question the Old Way on Purpose

One surprising part is that once you start measuring AI this way, you begin to notice how many of your existing habits were never measured either: the report everyone fills out that nobody reads, the meeting that exists because it always has, and the form that still includes fields for a customer you lost years ago.

AI is a good excuse to question how work gets done, but we should question it the same way we question any other process, with data and one variable at a time. Changing things on a whim because a new tool showed up is how shops end up with a dozen half-adopted systems and a team that tunes out the next announcement.

The people closest to the work are usually the first to see where a change might help, so ask them. Then measure what they suggest the same way you would measure anything else.

Why This Matters More Than the Tool

Two shops can buy the same AI tool on the same day, yet a year later, only one of them may be able to tell you exactly where it helped and where it didn't. The other may have nothing more than a vague sense that it is "useful sometimes."

The first shop learned something, while the second one bought something. That gap compounds over time because every measured win becomes a standard, and every new standard frees up time for the next experiment.

If you want a place to start this week, start with one task, measure how it works today, and test one change with AI. Until you can compare the result with what came before, you have not improved the process; you have only bought a tool. **I-CONNECT007**



Sean Patterson is the founder of CrossGen AI.

From the Playbook to the Production Floor

BY KELLY ATAY, ASC SUNSTONE

A well-executed, collaborative quote process is vital to manufacturing quality PCBs. PCB designs are like football plays in that a lot can happen between the chalkboard (aka CAD tool) and the playing field (aka production floor). No matter how well the play is drawn up, you still have to snap the ball and play the game. Designers should be prepared. Even when designers and manufacturers check all the boxes and work closely together, things can still go wrong.

That's because events don't always go as planned between the quote phase and product shipment. In football, when the ball is in the hands of the quarterback and the defensive line is coming at them, planning is only one part of preparation. If the players aren't schooled in how to deal with specific intangibles, the play can devolve into chaos.

The shop floor doesn't get as messy as a blown football play, but if the team doesn't account for potential issues during the quote phase and take appropriate action, there can be unexpected problems during production:

- Production delays are more common today because of continued supply chain disruption. Raw materials shipments can get delayed, creating inventory shortages and slowing overall manufacturing. We recommend designers build some padding into their timelines, if possible, just in case.



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

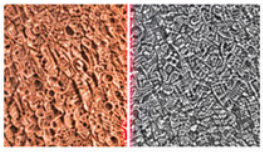
- Vertical electrode design
- Desmear, cleaning and activation
- For PCB, FPC and HDI applications
- Stable process control
- Flexible for multiple materials
- Compact footprint



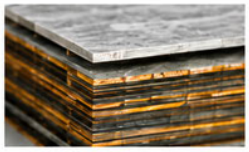
PROVEN PROCESS RESULTS



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

Trick plays require additional preparation

The nature of the quote itself can create intangibles for the team. Designers need to be aware if their design creates a need for specialized manufacturing processes. Does the quote require advanced testing or inspection? Will it create non-standard engineering costs? Is specialized tooling, setup, or a custom test fixture needed? Designers should be aware of how specialized requirements can impact costs, so they are not surprised by increases to the per-unit price or post-quote cost add-ons.

- Since it is often hard to pad timelines in a fast-paced environment, manufacturers can find themselves rushing to respond to quote requests. If a quote response is rushed or the fabricator is not prepared for quick-turns, mistakes become more likely. When teams hurry, tasks such as the DFM check can be approved too quickly, which can lead to mid-production discoveries that result in surcharges or increased scrap risks. Fabricators should not sacrifice thoroughness for speed.
- PCB manufacturers can only carry so much inventory, and these days it is more likely than not that the next supplier shipment will be more expensive than the last. Sourcing issues can also lead to cost overruns. When suppliers' costs go up, they pass them on to fabricators, sometimes with little warning. Quote responses are often based on the most recent component costs, but if prices change or long lead times necessitate an alternate supplier, this can affect the final assembly bill. To avoid this issue, it is important for manufacturers to keep their supplier's costs up to date and quote accordingly.

Best Practices for Dealing With Production Intangibles

Here are some ways designers and fabricators can work together during the quote phase to avoid problems that can derail production:

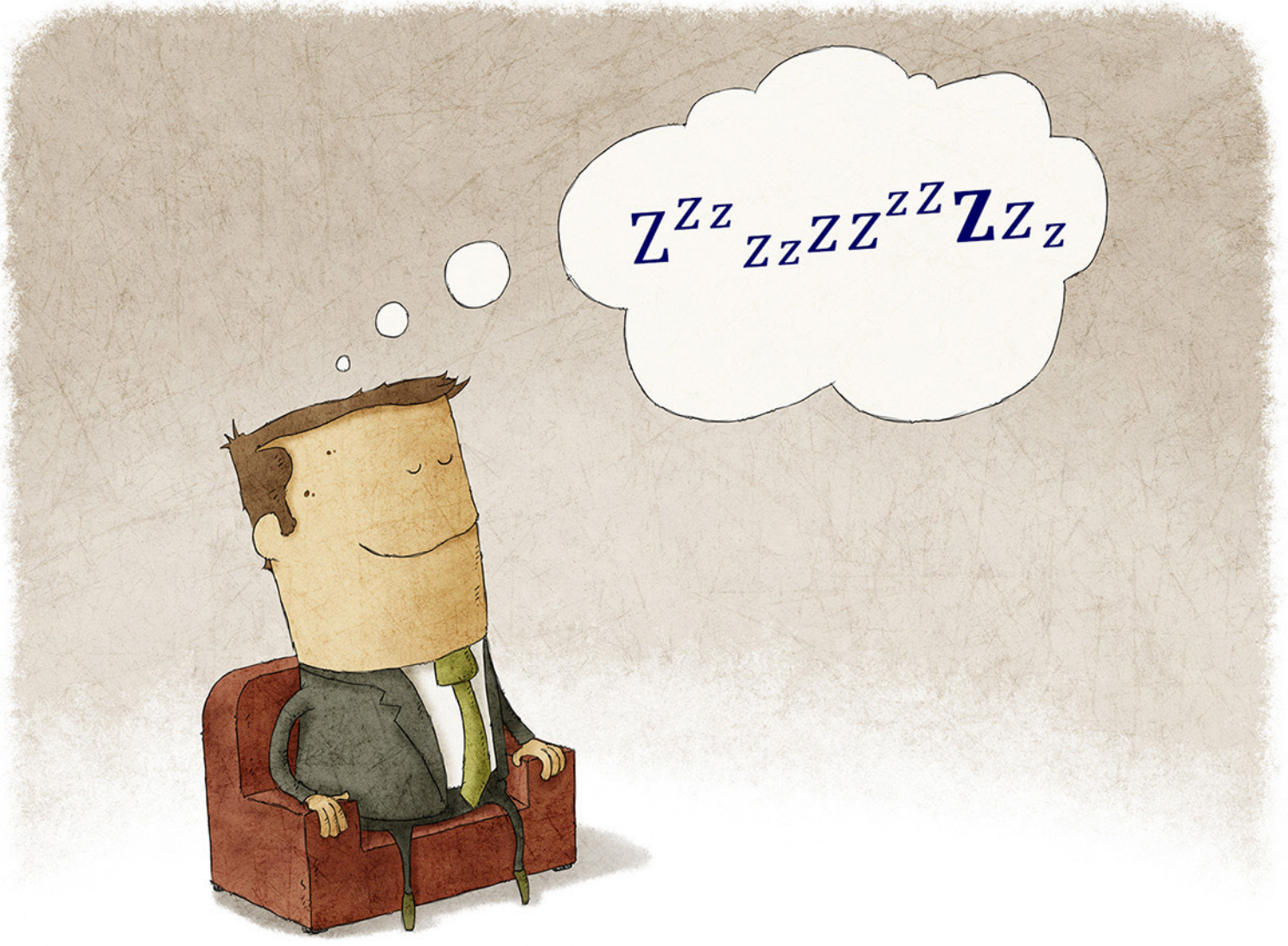
- Instead of trusting ChatGPT's research on general industry availability, designers should reach out to manufacturing partners to determine material availability and cost while still in the design phase.
- Design complexity for vias and controlled impedance can impact the time and effort needed to produce a board. Elements like non-conductive vias introduce complexity to quoting that makes it difficult to quickly turn an accurate quote. Designers need to be aware of how these components impact the quote process and plan accordingly.
- Complex designs require more time to quote and build. PCB designs with many layers, multiple blind via sets, and specialized materials will naturally take longer to quote. If a design takes more time to quote, it may also take more time to build. If a project is on a tight timeline, designers can work with the fab team to find ways to accelerate production and keep product development on track. Time spent in the prototyping phase listening to each other is particularly valuable.
- Context is important. Fabricators often do not know how the PCBs they manufacture will be used. This places additional burden on the designer to produce a near-perfect quote request and leaves the manufacturer unaware of how environmental factors could impact board performance. In many cases, if designers can inform manufacturers about how the board will be used, it can kick-start a conversation that will speed the prototyping phase, help optimize performance, and reduce production costs.

To ensure PCB manufacturing produces the desired quality and yield, much like the offense changing or altering the play at the line of scrimmage because of a defensive shift, the designer and fab team can work together to tweak the design or its specifications for any number of reasons, including materials availability, time constraints, or unearthed information about the finished board's application. **I-CONNECT007**



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

We DREAM Impedance!



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

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

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

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

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



iCD
In-Circuit Design Pty Ltd

www.icd.com.au

A Strategy for Polymer-Glass Interposer Design

BY VERN SOLBERG, CONSULTANT

Editor's note: This is the first of a two-part series. Part 2 will appear in the December 2026 issue of I-Connect007 Magazine.

The trend toward onshoring semiconductor manufacturing and the establishment of domestic packaging capability is opening a whole new window of opportunity for the board-level design engineering professional.

Although they may be proficient at solving complex interconnect challenges for board-level products, PCB designer engineers who apply their expertise to developing package-level interposer applications will take them to a whole new level. Interposer design software is essential for creating layouts that efficiently integrate multiple semiconductor components. The CAD tools we're familiar with were initially developed for printed circuit design, but their variations provide the more advanced circuit interconnect capability that is required for ultra high density interposer design applications.

Now, with the adoption of multi-core processor applications, traditional monolithic system-level products simply cannot keep up. The increasing complexities of the single monolithic die platform significantly affect process yields and seriously impact overall unit cost. Recently, chiplet advocate Synopsys posted, "the shift from monolithic to multi-die design is inevitable."

Interposer development for advanced multi-core processor applications and multiple semiconductor system-in-package (SiP) technologies presents many challenges. Recognizing the practical limits of the single-die system-level semiconductor product, manufacturers have rapidly adopted multi-die integration.

According to ICChipNet, the key issues influencing semiconductor package development are:

- **Miniaturization:** The trend toward smaller devices increases packaging complexity, making it difficult to maintain performance while reducing size.
- **Material compatibility:** Different materials used in packaging can lead to reliability issues, necessitating careful selection and testing.
- **Process integration**: Combining various semiconductor technologies, such as 3D ICs and chiplets, presents unique challenges in design and manufacturing.
- **Thermal management:** Effective heat dissipation is critical as devices become more powerful, requiring advanced materials and designs to prevent overheating.
- **Cost efficiency:** Balancing performance improvements with cost-effective manufacturing processes remains a significant hurdle.

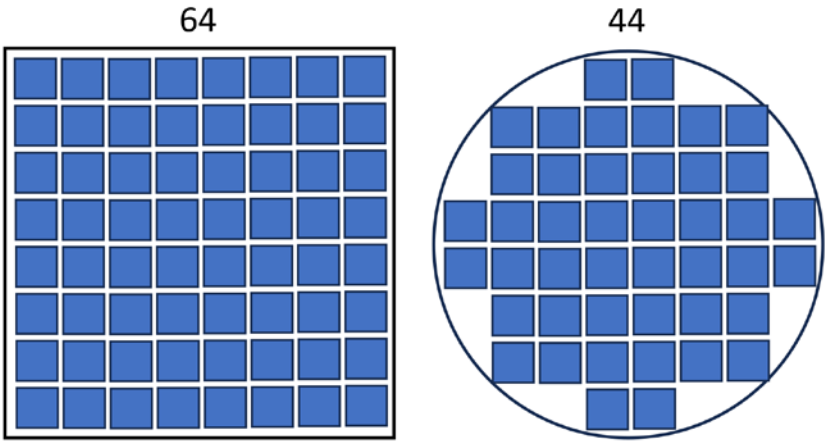


Figure 1: Comparing 1.5-inch square interposer layout, 310 mm panel to 300 mm wafer format.

This development is achieved by separating the complex core processors into less complex support functions: individually packaged chiplet die elements. Although multiple uncased die (hybrid) packaging and multichip module (MCM) assembly using packaged die have been used for decades to develop system-level products, semiconductor technology has evolved. IBM originated the concept of chiplet technology in the early 2000s by describing a number of chiplet applications.

The first practical implementation using chiplets occurred in 2013 when Xilinx (now an AMD company) collaborated with TSMC to use a chip-on-wafer-on-substrate (CoWoS) process, marking a pivotal moment in chiplet design and integration.

Initially, the processor die elements and the system support die were placed and interconnected on a silicon interposer. The supply sources and fabrication infrastructure for silicon wafer processing were well established and capable of responding quickly to the industry's immediate interposer needs. Technically, silicon is a terrific medium for multiple-die package interconnect, but the process for producing the silicon wafer is highly specialized and expensive, and wafers used for semiconductor processing are round.

Although configured for the existing fabrication process systems expressly developed for individual semiconductor die fabrication, they cannot be practically used in all interposer applications. For example, interposers are square or rectangular in the number of units that can be arranged within the circle.

Companies have found that using an alternative base material and a uniform panel format yields greater efficiency. In fact, advanced epoxy-glass composites have proven a practical alternative for low- to moderate-interconnect applications, and glass-core panels are currently considered the most viable substitute for silicon.

In fact, SEMI member companies have already established standard guidelines for panel-level interposer fabrication. The selected panel size is 310 mm x 310 mm, compatible with existing semiconductor processing systems, and suitable for alternative base materials (glass or epoxy-glass) processing systems. Figure 1 compares the wafer-level interposer layout to the layout efficiency using the rectangular panel format.

Polymer-Glass Interposer Momentum

As an alternative to silicon in wafer-level packaging applications, in 2010, Professor Rao Tummala at the Georgia Institute of Technology initiated a glass-based interposer development. Today, the panel-based industry consortium comprises about 50 global companies, including members from IDMs, OSATs, OEMs, and supply-chain manufacturers of materials and design and process tools.

Although glass is the base material for semiconductor packaging and is widely used by many prominent companies today, the materials and fabrication processes have required considerable time and resources to develop.

Through active collaboration with several materials and fabrication companies, Georgia Tech has established a proactive ecosystem to apply glass-based interposer technology to complex, multi-die chiplet system-in-package applications, as illustrated in Figure 2, as well as to high I/O-density single semiconductors.

The two primary suppliers of glass panels, Asahi Glass in Japan and Corning Glass in America,

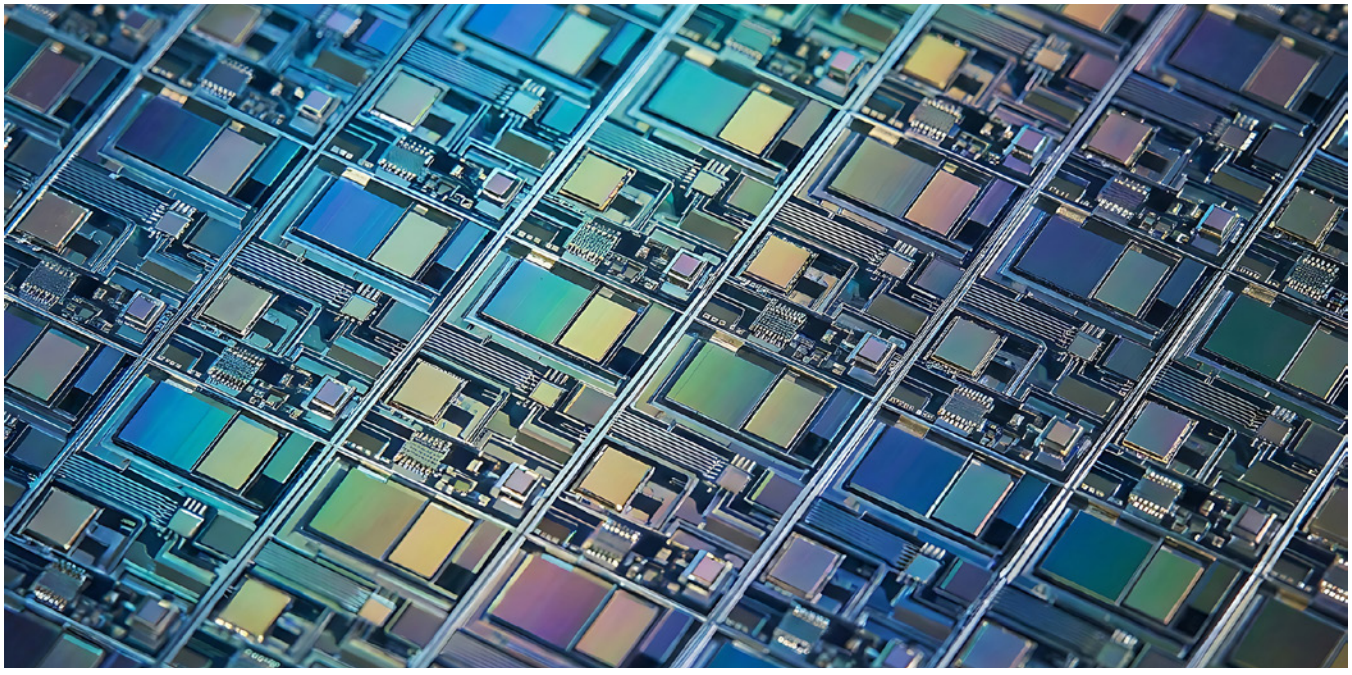


Figure 2: Chiplet configured panel-level, system package units.

contributed guidance during the SEMI standards development activity. Some are primarily concerned with the handling and processing of these thin glass panels, but experienced fabricators of silicon materials have these procedures in place. The proprietary formulations of glass from these suppliers, known as "gorilla glass," are very tough, delivered in sheets at thicknesses ranging from 100 µm to 400 µm (0.0394" to 0.1575").

Multilayer Polymer-Glass-based Interposer Fabrication Process and Concerns

The multilayer polymer glass interposer fabrication process includes various critical stages:

- **Material selection:** The process uses specific polymer glass materials with desirable optical and mechanical properties.
- **Layering technique:** Multiple layers are deposited to create the interposer structure, allowing for complex integration of components.
- **Patterning methods:** Techniques such as photolithography are employed to define the optical and electrical pathways within the interposer.
- **Via creation:** High-quality through- and blind vias are fabricated, with diameters as small as 20 µm.

Historical note: In six decades, the electronics industry moved from the basic integration of common working elements (digital logic gates, amplifiers, oscillators, timers, and counters), individually packaged for PCB interconnect, to a monolithic configuration that combines all the functional elements into a single silicon-based system-level product. The first commercial microprocessor was introduced by Intel Corporation in 1971.

In part 2, I will focus on polymer-glass interposer design strategy on multilayer copper/polymer material variations, fabrication process, and current design guidelines for interconnect routing. **I-CONNECT007**



Vern Solberg is an independent technical consultant, specializing in SMT and microelectronics design and manufacturing technology. He is the author of several books, including *Design Guidelines for Surface Mount and Microelectronic Technology*. To read past columns, click here.

schmoll america

... one step ahead.

www.schmollamerica.com

NEWS INSIDE SCHMOLL

The latest from Schmoll Maschinen – drilling, laser and automation technology for PCB and substrate production, engineered in Germany and supported in North America by Schmoll America.

DRILLING



NOVEMBER 3, 2026

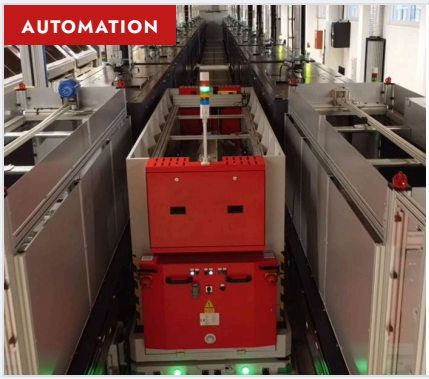
INDIVIDUAL TRACEABILITY PER STATION

Data Matrix reading on Schmoll drilling machines

Multi-station drilling machines now read a Data Matrix code at every station, so each station knows exactly which panel or stack it is processing. Drill program, tool usage, timestamps and each station's scaling coefficients are logged under the panel's code and handed to your MES – traceability for automotive, medical, aerospace and defense boards.

Scan for the full article
schmoll-maschinen.de/news

AUTOMATION



NOVEMBER 3, 2026

AUTOMATION: FROM MACHINE TO LINE

People in Precision – November edition

Schmoll builds the automation for its entire range in-house – drilling, routing, direct imaging, X-ray, punching and laser – so machines and handling share matched interfaces and software. Fewer manual transfers mean less handling, less risk to sensitive panels and longer unattended runs. With ASSxLHMT, whole production sections come from one partner.

Scan for the full article
schmoll-maschinen.de/news

LASER



OCTOBER 12, 2026

MICROSCOPE INSIDE THE LASER MACHINE

Quality pre-control directly on the machine

Selected laser drilling models now come with a built-in microscope. Operators inspect the first holes of a new batch right on the machine – no trip to a separate inspection station, no loss of alignment – and can run test structures to tune pulse energy, pulse count or focus before volume production. The result: shorter setup, less material loss and fewer test runs.

Scan for the full article
schmoll-maschinen.de/news

LET'S TALK ABOUT YOUR PRODUCTION.

Schmoll America · 10 North Martingale Road, Suite 400 · Schaumburg, IL 60173
Phone 1-888-726-0321 · support@schmollamerica.com

www.schmollamerica.com/contact

SCAN TO CONTACT US

DIY Spectroscopy (Colorimeter) Control

BY HAPPY HOLDEN, I-CONNECT007

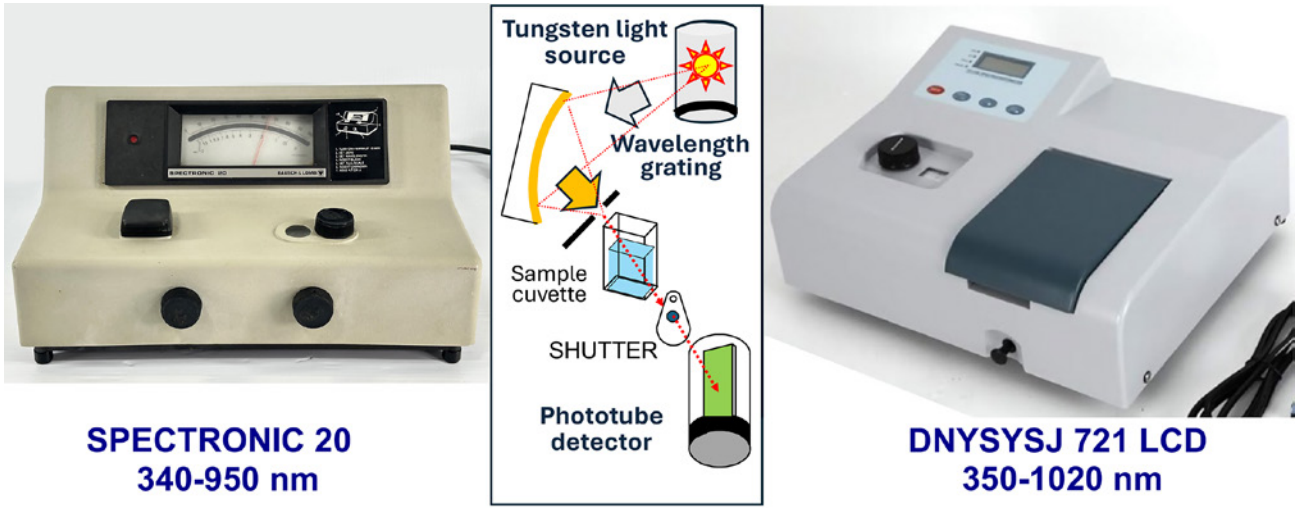


Figure 1: Stages of developing UV-Vis spectroscopy from 1960 to 2025.

I was introduced to my first sensor, the Spectronic 20, a tube-amplifier-based photo-spectrometer, in my ninth-grade chemistry course. You probably can't even find one of these anymore (it was 65 years ago, after all), because it has been replaced by a modern digital UV-Vis spectrophotometer that you can buy online for less than $200. Many schools are having students build their own, which I will cover later.

Spectrophotometers (Colorimeters)

Light waves (electromagnetic radiation) have been a focus of humanity for centuries, ranging from rainbows to reflections in puddles. I covered these in detail in my book, Automation and Advanced Procedures in PCB Fabrication.[1]

Other than my chemistry courses and keeping the chlorine level in the family pool at the proper level, I did not deal with spectroscopy until working in printed circuit manufacturing at Hewlett-Packard, where, in 1972, we had the opportunity to test HP's new generation of UV-Vis spectrophotometers (Figure 2).

The HP 8450A was a breakthrough because it used a holographic grating and a fixed array of photo-sensors instead of moving the grating to focus light on a single photodetector. This allowed the instrument to scan the entire UV-Vis spectrum in milliseconds rather than minutes.

The latest version of HP's diode-array spectrophotometers (DAS) is the Agilent 8453E, which you can only find on eBay. In Chapter 4 of my Automation book, you can see the extensive research we performed on using the new computer-controlled DAS to rapidly analyze many of the chemicals used in printed circuit processes. This allowed us to set up and use these principles online for real-time process control.



Figure 2: Various DAS UV-Vis spectrophotometers.

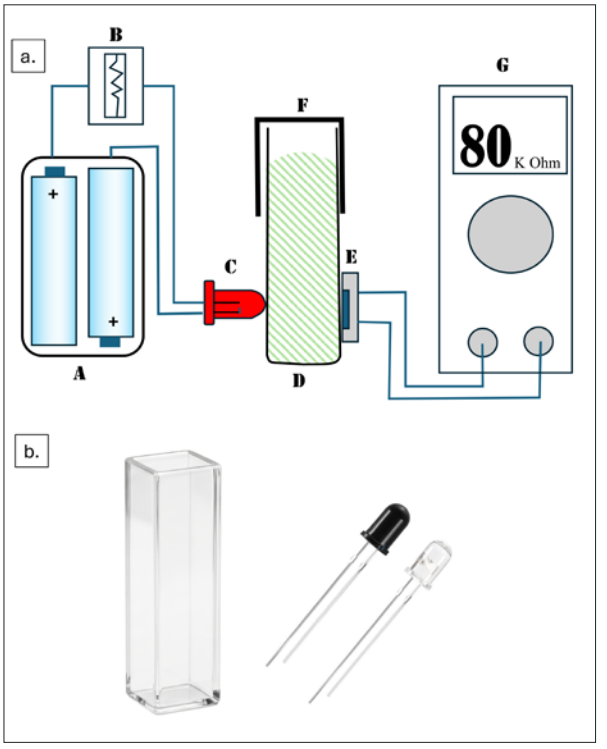


Figure 3: a) Simple schematic of a colorimeter: batteries (A), control resistor (B), LED light source (C), sample cuvette (D), photo sensor (E), sample cap (F), display device (G); b) The three principal elements.

Principle of Operation

The simplest method of analysis is shown in Figure 3a. An LED is used to emit a specific wavelength of light, which passes through a glass cuvette containing a solution, where the light is absorbed by the liquid and detected by a photoreceptor. Figure 3b shows these three elements.[2]

Sensor Design

A tungsten light is typically used as the Vis-IR light source, and a deuterium-tungsten lamp is used for UV-Vis. The wavelength separation is performed by a ruled grating or monochromator, using a photodetector, photoresistor, or cadmium sulfide sensor. The old Spectronic 20 used a photomultiplier tube, but current instruments use various solid-state detectors and sensors, such as the DAS. Three of these components are illustrated in Figures 4b and c.

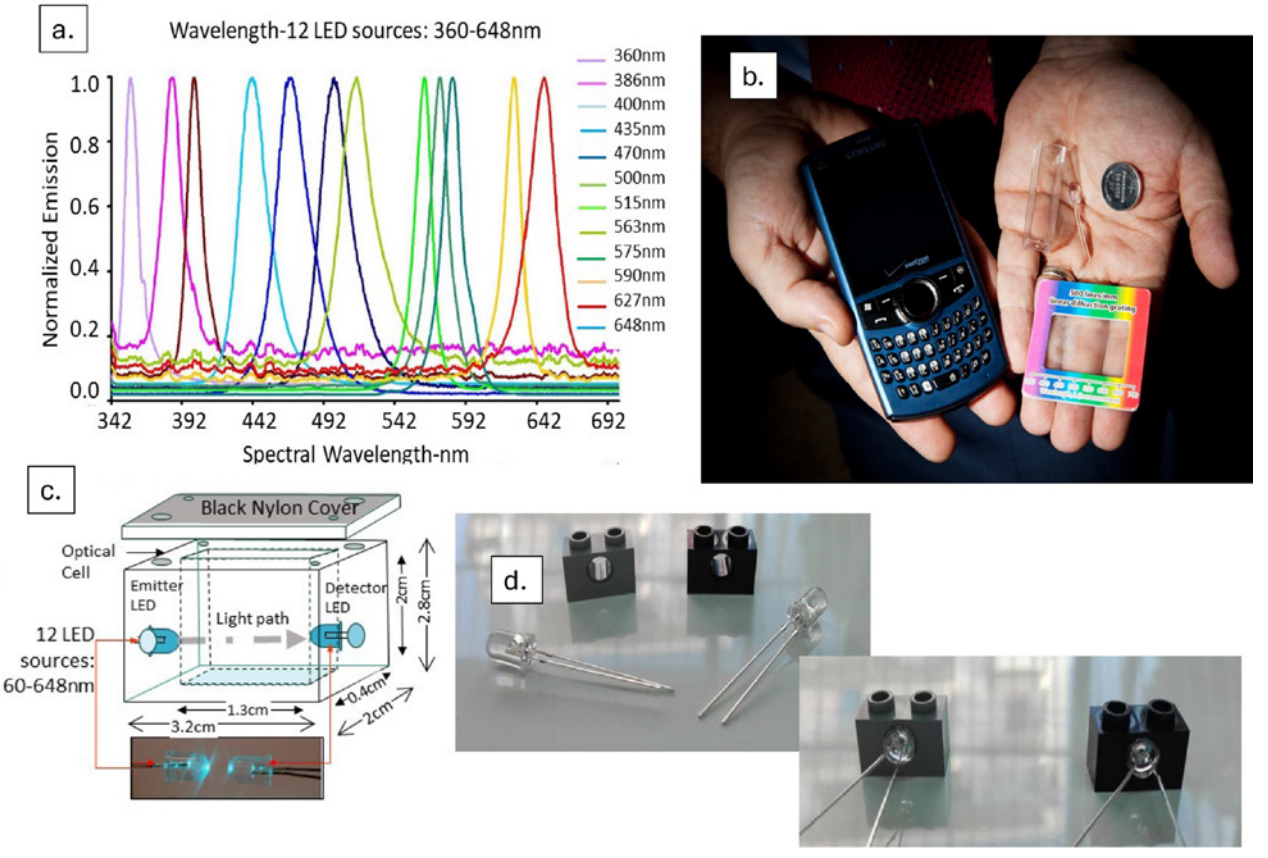


Figure 4: a) The various wavelengths of LEDs available today: UV and infrared LEDs, from 200 to 940 nm; b) LEGO® blocks are constructed for the case and housings; c) An old camera phone can be used as a sensor, and a monochromator slide can filter white light. Pieces of old CD can be used as gratings as well because of the density of pits in the polycarbonate plastic; d) The LED source and the LED photodetector are mounted in LEGO single brick pieces.

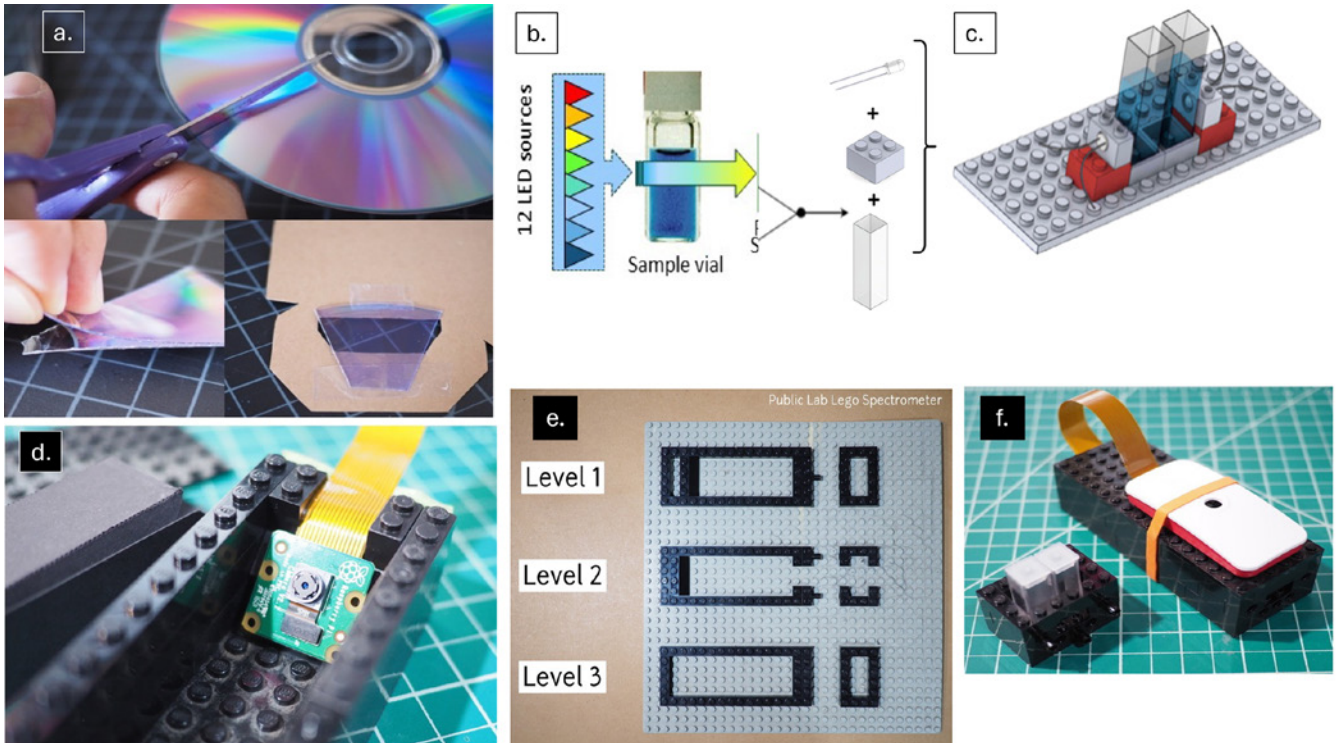


Figure 5: Basic DIY spectrophotometer for the (LEGO) type: a) Cutting a piece of old CD for the monochromator; b) One or more of the 12 different LEDs for light sources; c) LEGO parts without the sealed box; d) Photosensor purchased and placed in the LEGO box; e) Different levels of LEGO bricks for the unit; e) Finished LEGO unit using the Raspberry Pi and its sensor.[2,3]

Construction and Example

The colorimeter has several DIY publications due to its popularity in middle and high schools (Figure 5). The most popular is the LEGO spectrophotometer. Either the Raspberry Pi or the Arduino Uno can be used as a controller, and sensors can be from either the Pi or the Arduino. You could even substitute an old smartphone as long as it has a camera. See schematics, instructions, and programs here.[2]

Controllers

A controller is constructed by taking the sensor

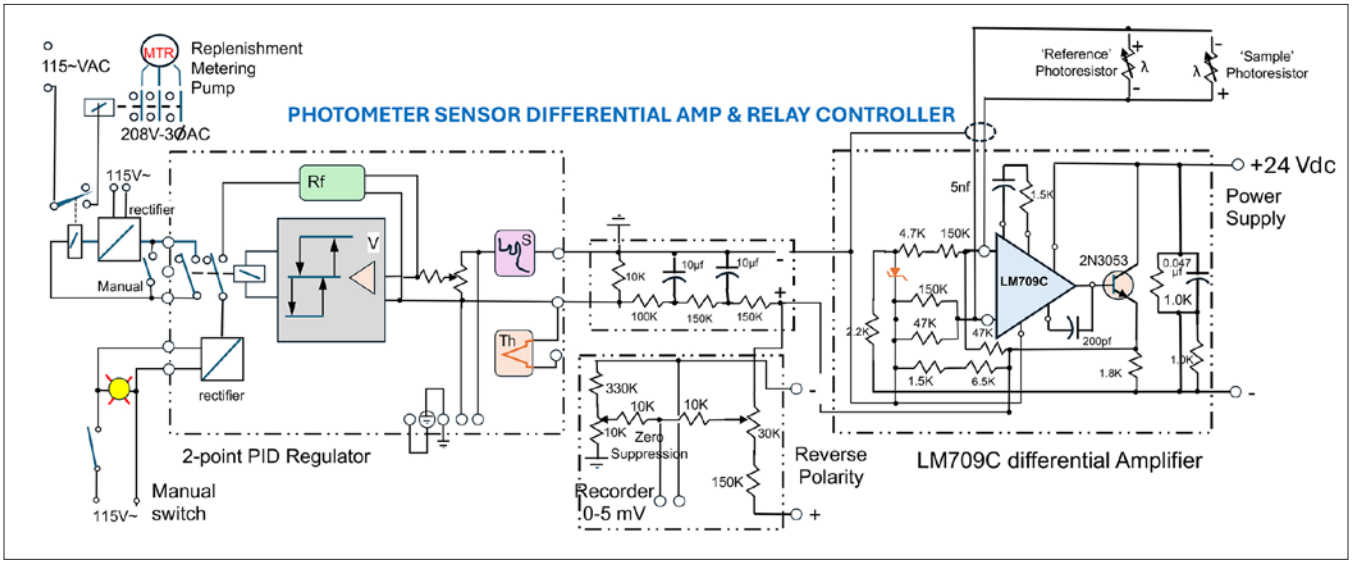


Figure 6: The schematic of building the colorimeter and controller from bare parts.

output or sensor electronics output and comparing it to the set point. The simplest controller is an on/off relay, and the most complex and expensive is a PID controller. There are controller programs available for all the popular maker microcomputers on the market, as well as simply writing your own. Figure 6 is one such example.

Programming

The Micro:bit from the UK's BBC is used in UK middle and high schools to teach computer programming to students. These are provided free of charge to UK middle school students, and nearly four million have been sent out. I purchased one for my grandson for $16. As such, there are extensive lesson plans, software, examples, and instructions available on the Internet to learn microcomputer languages such as LINUS, Python, and C.[4]

Replenishment Actions

Replenishment of chemicals is accomplished with various valves, whether proportional or on/off, or with pumps, the proportional replenishment pump being the most popular. You can fabricate systems with your own designs, also with a connection to a DAS or any monitoring software. Examples are shown in Figure 7.



Figure 7: Control actions can be a simple alarm or replenishment of chemicals or water with various valves, be they proportional or just on/off, or with pumps.

Summary

All these sensors, except for the specific gravity sensor, are electronic and require an electronic setpoint.

Because virtually nothing has been published about the use of UV-Vis spectroscopy in printed circuit analysis or control, HP's PCB organization did extensive research and published a number of technical papers for the American Chemical Society and their Pittsburgh Conferences.[1]

While doing our research, we had to rely on German and Russian analytical textbooks on colorimetric techniques, but we pursued this track because of the excellent control capability of an online colorimeter for advanced and UHDI processes. Because PCB wet processes are water-based, this sensor has applications everywhere and is a good place to start.

Because of the current popularity of aquaculture/hydroponics, home aquariums/swimming pools and hot tubs, and microbrewing of beer and wine, as well as environmental studies in school science classes, many suitable sensors and controllers can be purchased locally or online, all at reasonable prices. **I-CONNECT007**

References

1. *Automation and Advanced Procedures* in PCB Fabrication, Chapter 4, by Happy Holden, 2024.
2. Spectrometers and Spectroscopy Equipment, Ossila.
3. "Arduino Rig Does Spectrophotometry," by Zoe Skyforest, Hackaday, Aug. 24, 2020.
4. "Happy's Tech Talk #20: Teaching Coding to Kids—The UK's Micro:bit Tool," by Happy Holden, *PCB007 Magazine*, June 2023.
5. For additional instructions on sensor technology, visit "Teach Engineering-STEM Learning for K-12."



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

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

schmoll asia pacific

SCHMOLL ASIA PACIFIC — an international machinery company serving the electronics industry (approx. 500 employees in Germany, 300 across Asia, EUR 150m annual revenue) — is expanding in China.

Director of Product Management & Technology – Asia
Guangdong, China | Reporting to the Group CTO

Key Responsibilities:

- Develop and maintain the regional product strategy, technology roadmap, and product lifecycle management
- Monitor competitor technologies and PCB manufacturing trends; prepare market intelligence reports
- Translate customer insights into future product requirements through visits, workshops, and benchmarks
- Lead new product introductions, including launch planning, sales enablement, and pilot installations
- Support regional sales with technical expertise and strategic sales opportunities
- Act as the primary interface between sales, customers, and German R&D

Candidate Profile:

- Degree in Mechanical/Electrical Engineering, Mechatronics, Automation, or related discipline
- 10+ years in PCB equipment manufacturing, preferably drilling/routing; candidates from leading PCB manufacturers are also welcome
- Experience with Via Mechanics, Schmoll, Han's Laser, Tongtai, or similar is preferred
- German speaker
- Strong leadership, strategic thinking, and cross-functional communication; frequent travel within Asia

We Offer:

- A senior position shaping regional product strategy, with close cooperation with German teams
- Competitive package: 13-month salary, performance bonus, 30 days leave, relocation support, apartment, and company car

Apply now and join our international team.

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

Cogiscan

Solutions Consultant – Americas

We are seeking a Solutions Consultant to provide technical expertise and pre-sales support to customers across North and South America. Working with Account Executives, Account Managers, and partners, you will help electronics manufacturers evaluate how Cogiscan solutions can improve factory connectivity, traceability, material and process control, and operational performance.

You will lead customer discovery, define technical requirements and solution architectures, deliver product demonstrations and technical presentations, and develop Statements of Work, proposals, and project documentation. You will also collaborate with Product Management, Operations, and implementation teams to validate solutions and ensure a smooth sales-to-operations handoff.

What we're looking for:

- 3+ years in Solutions Consulting, Sales Engineering, Applications Engineering, Manufacturing Engineering, or a similar customer-facing technical role
- Experience with electronics manufacturing and MES, traceability, machine connectivity, industrial software, or factory analytics
- Strong communication, presentation, analytical, and problem-solving skills
- Ability to translate customer requirements into practical technical solutions and documentation
- Bachelor's degree in a related technical discipline strongly preferred

This is a primarily remote, customer-facing role requiring approximately 20–30% travel throughout the Americas.

Candidates must be United States citizens due to customer facility access requirements.

Apply Now!

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!

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!

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!

CAREER OPPORTUNITIES

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!

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!

CAREER OPPORTUNITIES

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!

PK M·E·T·A·L

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!

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?!

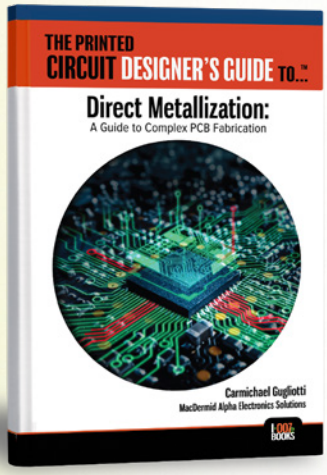
Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers to build better boards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



A Collaborative Quote Process

By American Standard Circuits

Learn how effective collaboration between PCB designers and fabricators improves quoting accuracy, reduces costs, and enhances manufacturability.

DOWNLOAD



Short on Glass? Build on Ceramic.

By MacDermid Alpha Electronics Solutions

Explore how MacDermid Alpha's Micromax ceramic circuit technology addresses woven glass shortages while improving thermal management, electrical performance, design flexibility, and manufacturing efficiency.

DOWNLOAD

PODCASTS



Engineering What's Next

with Siri Kore, Qnity

LISTEN NOW >



Voices of the Industry

with Kelly Davidson, NCAB

LISTEN NOW >



Meet the Author

with Carmichael Gugliotti, MacDermid Alpha Electronics Solutions

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
SHELLY STEIN

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.

November 2026, Volume 1, Number 11
I-CONNECT007 MAGAZINE is published monthly,
by IPC Publishing Group, Inc., dba I-Connect007.

ADVERTISER INDEX

Accurate Circuit Engineering 7
All Flex Solutions 39
all4-PCB 109
ASC Sunstone 97
atg Mycronic 65
Cadence 75
Candor Industries 11
Chemcut 105
Electra Polymers 67
Elite Materials 13
Flexible Circuit Technology 29
Global Electronics Association 23, 99
GreenSource Engineering 61
I-007eBooks 2, 55
Incircuit Design Pty 111
Insulectro 5, 47, 49, 53
IPS 9
Isola 63
MacDermid Alpha Electronics Solutions 37
Matrix USA 57
MKS' Atotech 85
MKS' ESI 92, 93
On the Line with... 3
PCBAA 33, 45
Pluritec 25
Polar Instruments 87
Qnity 17
Schmoll America 81, 115
Technica USA 101
United Electronics Corporation 59
US Circuit 71
USA Microcraft 43
Ventec International Group 79

Problems solved!

