

THE MAGAZINE I-Connect007

INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY



Plus: Designing for
Display Electronics

PCB007
DESIGN007

JULY 2026

Protection That Performs

THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...

UV Curable Conformal Coatings

Addressing Future Manufacturing Speed and Need



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

I-007
BOOKS



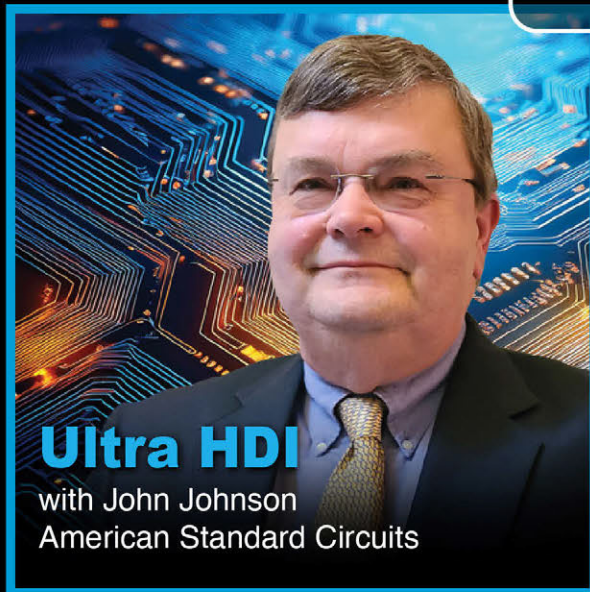
Look Inside

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

—Stanton Rak, PhD, SF Rak Company

I-007eBOOKS

Now Playing



PODCAST SERIES
WITH BONUS COMPANION GUIDES

Business Diversification and Display Electronics

Business diversification and designing for display electronics both demand strategic decisions that balance technical expertise with long-term success. This issue explores practical paths for PCB fabricators, unique design challenges with LED displays, AI's growing influence, and the technologies shaping tomorrow's electronics industry.

From the Editor

Marcy's Musings

8 From Business Diversification to Designing Display Electronics

by Marcy LaRont

PCB007

Features

The Chemical Connection

22 When History Repeats Itself on Diversification

by Don Ball

26 Beyond the Board: Diversification as Strategy

with Ray Cottrell

Happy's Tech Talk

50 Stop! Do You Have a Digital Strategy for the Future?

by Happy Holden

The Right Approach

70 The Pros and Cons of PCB Vertical Integration

by Steve Williams

Articles

12 Summit Interconnect Celebrates 10 Years

with Shane Whiteside and Gerry Partida

34 DFM Mistakes Designers Make When Moving to ELIC

by Anaya Vardya

Don't Buy AI: Learn AI

54 Your AI Doesn't Know Your Process

by Sean Patterson



18 Diversification: Where Do You Grow From Here?

by Dan Beaulieu



48 EIPC Summer Conference

by Marcy LaRont

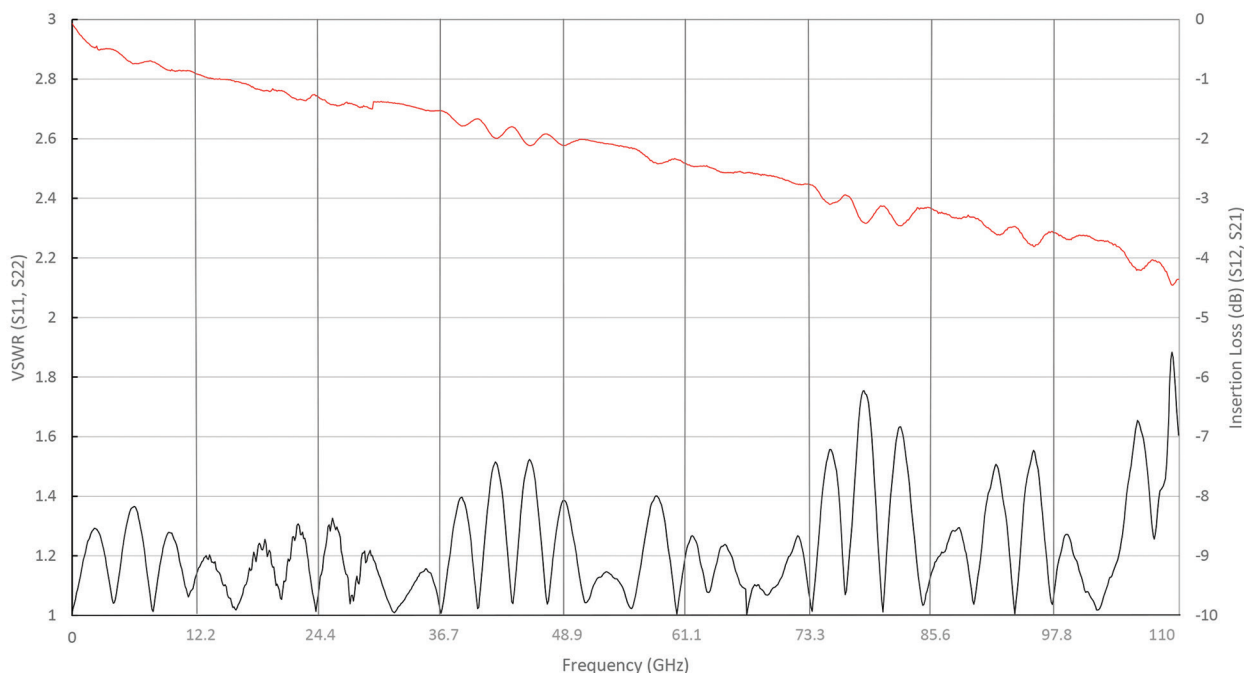
In Every Issue

- 76 MilAero007 Highlights
- 113 Career Opportunities
- 120 Educational Resources
- 121 Advertiser Index

THE MAGAZINE
I-Connect007
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

An Industry First!

Stripline to 110GHz.



Test Data: VSWR/ Insertion Loss



OPERATOR:	DD	DATE:	03/25/24	MODEL #:	24359-011SF
FILE NAME:	DataFile#5.s2p	PART #:	81W70350		
DESCRIPTION:	1.0mm J 2H VL STRPL, DEDD-001, BBRT	LOT #:	160475&158981-000		

■ S11 ■ S12 ■ S21 ■ S22

We are very pleased with our boards from Accurate Circuit Engineering. The data speaks for itself.

— Southwest Microwave

QUICK TURN RF, MICROWAVE, ANTENNA EXPERTS



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

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

PCB007 Columns

Trouble in Your Tank

44 At 40 GHz, Everything Matters, Part 2

by Michael Carano

American Made Advocacy

66 PCBAA's Fifth Annual Meeting: Growing Membership and Valuable Relationships

by Shane Whiteside

Powering the Future

104 Performance Is Won in Assembly

by Brian Buyea

DESIGN007

Features

Target Condition

82 Flex and Rigid-flex Solutions in A&D Display Electronics

by Kelly Dack

92 Understanding Design for Display Electronics

by Kristin Moyer

Flexible Thinking

98 'Let There Be Light'—Emitting Diodes!

by Joe Fjelstad

Elementary, Mr. Watson

108 The Display Revolution, Now With Fewer Tiny Keys

by John Watson

Articles

88 IPC Standards Released for Q2 2026

96 Smart Textile Integration With Flex PCBs

by Anaya Vardya

Columns

Connect the Dots

60 A Total Value Manufacturing Solution

by Matt Stevenson

JULY 2026

78 Three Lessons Every Designer Should Know About LED Displays

by Stephen V. Chavez

Shorts

97 From Prototype to Production: New Guideline Helps Industry Design, Integrate and Scale Reliable E-Textile Systems

by Global Electronics Association

111 Below the Surface Ceramic vs PCB: When 'Good Enough' Becomes System Failure

by Chandra Gupta



WORLD-CLASS SUPPORT

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

CALL 949.587.3200 FOR MORE INFORMATION

Welcome to Our Newest Suppliers



ELITE MATERIAL CO., LTD

For All Your 370HR & 185HR Needs and more

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

RESIN SYSTEMS FOR PCB LAMINATES

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



ARLON
ELECTRONIC MATERIALS

P85HP - Tomorrow's Polyimide Today

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

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

POLYIMIDES SYSTEMS FOR PCBS

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

PREPREGS FOR RIGID-FLEX & CAVITY PCBS

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



From Business Diversification to Designing Display Electronics

BY MARCY LARONT, I-CONNECT007

I love this time of year. Aside from summer, I appreciate the natural pause from the busyness of trade shows, conferences, airport delays, and back-to-back Teams meetings. Colleagues become a bit harder to reach as family vacations are prioritized, and we take focus off work for a little while. But no matter the season, the accelerated pace of technological advancement doesn't actually slow down. In this issue, we use this pause to look inward and forward.

In PCB fabrication, we examine the need for business diversification, especially for bare board fabricators in the U.S., where there are fewer than 150 shops, and technology expansion demands significant CapEx investment. The right decision to diversify or double down on what you already have always comes down to a convergence of issues.

Ray Cottrell of Flexible Circuit Technologies (FCT) speaks with authority on this topic. He joined FCT when president and founder Troy Koopman

STRONG PARTNERSHIPS GROW LONG BEFORE THEY'RE NEEDED.

Material shortages.
Longer lead times.
Supply chain uncertainty.

These challenges are affecting much of the PCB industry. Preparation makes the difference.

At FSQuality, decades of strategic supplier relationships and global sourcing strength help ensure reliable access to critical materials — keeping your production moving when others slow down.

When the market gets tough, the right partner stands out.



GLOBAL STRENGTH
Local Support



ON-TIME DELIVERY
When It Matters



TRUSTED QUALITY
Every Step



YOUR SUCCESS
Is Our Priority



chose to expand the assembly operations, and he provides foundational tenets for what successful diversification requires. Dan Beaulieu, who's been working with PCB companies for decades, shares insight into what diversification might look like and the important steps on how to begin.

Our columnists also take on this subject. Steve Williams harkens back to his master's degree dissertation on vertical integration and business diversification for U.S. PCB fabrication. Happy Holden approaches competing in this complex global environment from a slightly different perspective, reinforcing the critical need for fabricators not to delay automation. Don Ball points out the very cyclical nature of the advice to diversify. "The reasons for diversification or de-diversification during economic uncertainty are not unsound, but not for everyone," he writes.

In PCB design, we explore how display electronics, specifically LED displays, change a designer's approach. Kelly Dack states, "Displays and illuminated human-machine interfaces have become essential components of modern electronic systems. From aircraft cockpits and mission-control displays to medical equipment, industrial controls, and consumer devices, we depend on these visual interfaces to interact with technology." Kristin Moyer defines the different types of display technology and breaks down some of the design "gotchas" that one inevitably faces. John Watson writes, "The display becomes the center of the interaction, and the PCB is no longer hidden inside the product; it becomes the foundation of what the user sees and feels." Stephen Chavez examines the uniqueness of designing for LED display electronics, observing that it is one of the only times you are "engineering for perception." He rightly points out that the responsibility of meeting this challenge lies equally with

both design and manufacturing. Joe Fjelstad looks at the past and future of LED technology and offers a solution that might help designers and manufacturers achieve greater success.

Columnist Brian Buyea of Remtec talks about the importance of design for assembly, often an afterthought in the process. Anaya Vardya writes about the integration of smart textiles with flexible PCBs, while Michael Carano examines surface-finish choices for high-frequency 5G and 6G applications and extols the virtues of ISIG. We also include the second part of our discussion on every layer interconnect (ELIC).

In other news, Summit Interconnect celebrates 10 years, and my interview with Shane Whiteside and Gerry Partida takes us through the milestones that got them to where they are today. PCBAA just celebrated its five-year anniversary, and Chairman Shane Whiteside looks back at its impressive growth and milestones over that time. The EIPC Summer Conference took place last month, and we provide links to some of the coverage. Sean Patterson continues his "how-to" guide for the PCB fabricator to start implementing AI into his operation, and the Global Electronics Association released the IPC Q2 2026 standards update.

So, here's to the summer slowdown. Grab that icy cold glass of lemonade, put your feet up, and read on. **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](#).

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

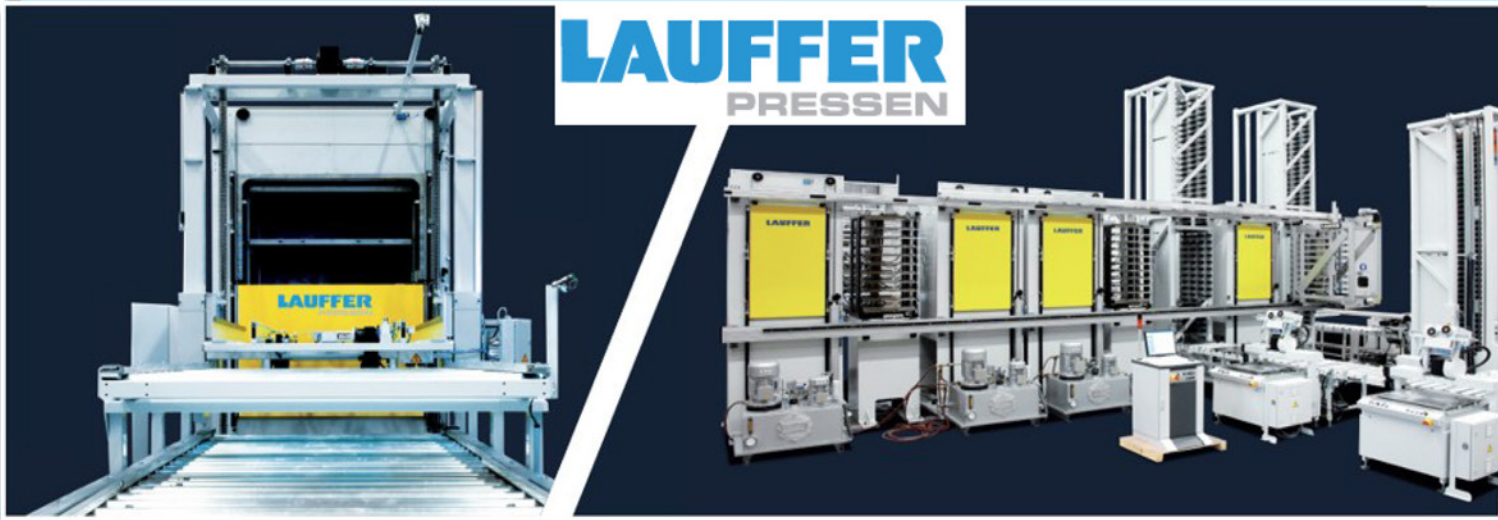
FOLLOW US



I-Connect007
The Global Electronics Resource

ADVANCED LAMINATION PRESS SYSTEMS

RMV SERIES | VACUUM LAMINATING TECHNOLOGY



RMV SERIES

Vacuum Laminating
Press Systems

Fast. Flexible. Efficient.

Modular Line Concept

Configure today and expand later as throughput or process needs change.

Automation Ready

Manual, semi-automated and fully automated press system layouts available.

Vacuum Lamination Control

Designed for precise pressure, temperature and vacuum process control.

Traceability + MES

Recipe management, process logs, OPC UA / SQL interface capability.

LAUFFER Lamination Software Control



Built for advanced lamination application

PCBs CCL Substrates Technical Laminates Busbars Smartcards

Summit Interconnect CELEBRATES 10 YEARS



Ten years ago, Shane Whiteside had a vision for the dwindling North American PCB industry: to develop a company at scale that would put North American PCB manufacturing back on the map. The result was the combination of six PCB organizations and complementary businesses to create Summit Interconnect, now the second-largest PCB manufacturer in North America.

In this interview, Summit CEO Shane Whiteside and Vice President of Technology Gerry Partida reflect on the achievement of Summit's 10th anniversary, the pride they take in what they've built,

and the enthusiasm they still bring to the business as they celebrate this milestone.

***Marcy LaRont:** Shane and Gerry, congratulations on 10 years. It's a great accomplishment. What was the impetus for starting Summit?*

Shane Whiteside: The U.S. market over the preceding 15 years had undergone a period of rapid decline due to geographic migration to lower-cost regions. It was very fragmented. We thought there was a case for future growth and that was part of our original strategic thesis.

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.

Who helped you get the company started?

Whiteside: I worked with HCI Equity Partners (now known as Oridian Capital Partners), a private equity investor group based in Washington, D.C., that I had worked with many years before. I knew they were interested in the North American PCB market, so together we conducted extensive research, developed a strategic thesis, and began outreach to select companies.

We have been doing this for a long time, and from the beginning, we understood the importance of scaling our investments to acquire the people, organization, and capital equipment to be at the leading edge of technology.

The cash flow requirements to be the kind of manufacturing partner that your customers require keep getting more expensive every year. For example, when Gerry and I first started in manufacturing, a film-based platform with UV lights cost about \$40,000. Now, an imaging platform costs 12-15x more because it's direct imaging. At one time, mechanical drills were the only choice; now a laser drill costs 5x more.

That's an important point. After securing the investment, how did you get started?

Whiteside: We identified two outstanding companies with excellent reputations that could form the foundation of Summit. We purchased KCA Electronics in April 2016, and Marcel Electronics (MEI) soon after. MEI was known for producing highly complex rigid boards, HDI constructions, and for using advanced materials. KCA specialized in advanced rigid-flex manufacturing. Both had very interesting individual customer lists. Importantly for us, both companies provided a broad end-to-end product mix.

We started consolidating, not only the back-end systems of payroll, accounting, etc., but also our sales forces to give this new company one face to our customers. We put a lot of emphasis on cross-selling between the two customer bases and intro-



ducing customers to the expanded capabilities we could now offer. That approach generated organic growth almost immediately. At that time, Summit was just a small fraction of the size we are now.

Leading up to 2016, investor interest in PCB manufacturing was relegated to a much smaller number than what we are seeing today in North America. Did you find that to be true?

Whiteside: That's correct, Marcy. Before we started Summit, North America's PCB industry had been in a 15-year decline. But we were convinced there were durability and growth prospects in supporting U.S. defense and in commercial prototyping, and this was part of our initial strategy. We felt those two areas would provide long-term growth opportunities in North America, and that remains where our company is focused.

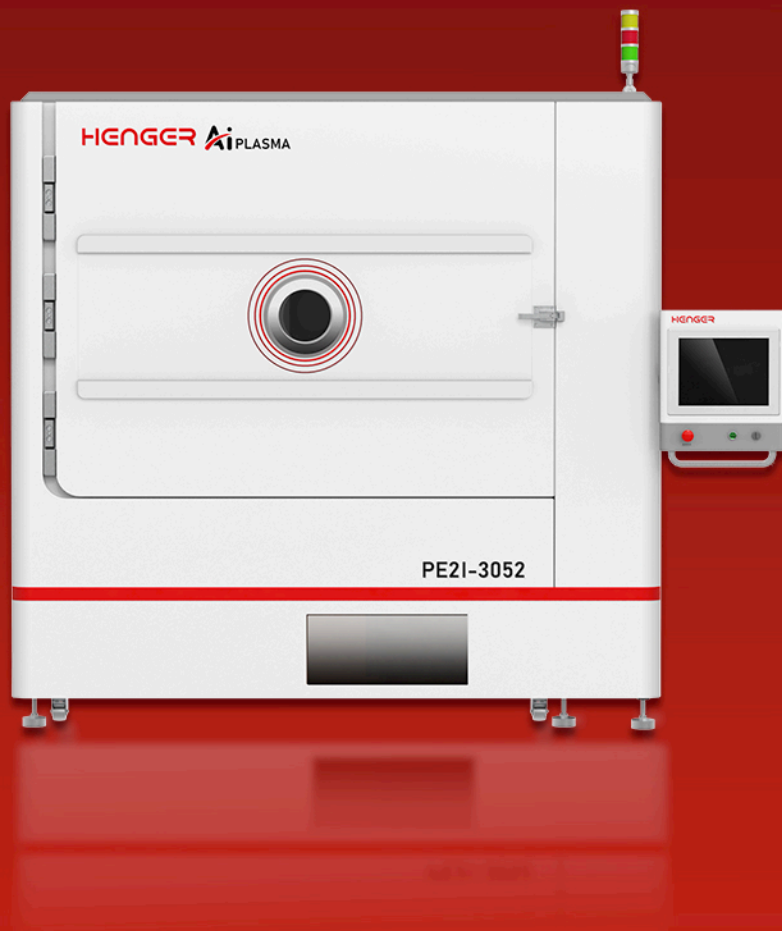
You created a sound strategic approach at the outset and believed you would create a substantial presence in this market. Was there any trepidation given the market at that time, and have you wavered from that strategy?

Whiteside: We saw some initial challenges. The North American market didn't really start growing until the end of 2017, and it's been consistently growing since. But we believed in the industry's long-term growth prospects and in our strategy, which was to provide a platform with end-to-end product capabilities. With our background, experience, and insight, we took on the challenge to grow in the market, and it has worked out well. Investors want to see that demonstrated ability to grow organically, and we've also made some interesting acquisitions along the way.

Something I find interesting about Summit has been the ability to create a cohesive organization through mergers and acquisitions. That is not a small feat, and you've done it seven times. What is your strategy?

AI plasma

Built for next-gen AI hardware, Henger AI Plasma processes M8/M9 ultra-low loss materials with reliable, uniform desmear across 32:1 aspect ratio PTH structures.



Uniform desmear
across top, mid, and
bottom vias



Multi-point
electrode
temp control



Real-time open
circuit monitoring
& fault traceability



Dual cooling
system for RF &
vacuum stability

Material	Aspect Ratio	Avg. Desmear (μm)	TP Yield (%)
EM-892K (M8)	32:1	9.36 - 10.56	85 - 90
EM-896K3 (M9)	32:1	13.08 - 21.99	81 - 89

Whiteside: Defense, specifically, is a very specialized market with many additional requirements to service it. KCA and MEI both brought a very high degree of proficiency with a deep bench of experienced, talented staff. That gave us a solid foundation to service the defense market.

Combining the cultures of two companies, particularly with an M&A strategy, can be challenging, but it must be a top priority. At Summit, everyone knows that our most valuable assets are our customer relationships. It is a bedrock of our culture. We don't have any unique IP or our own proprietary technology. We stand out as a customer partner in our capabilities and willingness to execute. Every day, we strive to be our customers' best manufacturing partner.

I couldn't agree more that we succeed and fail through the success of our customers. Gerry, what are your thoughts on creating a unified company culture?

Gerry Partida: When you bring companies together, you don't want to diminish what they are and try to make everyone the same. It just doesn't work. We wanted to preserve each company's strengths and unique skill sets, then share them across all facilities. That was really the most challenging part. Acquiring facilities is the event, but the actual work is bringing everyone to work as one company. Each company brings something important. It's their people, process knowledge, and the customer trust. We didn't want to erase those incredible strengths. Rather, we wanted to connect them all.

Through Shane's vision and leadership, we've formed a cohesive team. It has made it easy to share experiences and capabilities and be open-minded. As we have shared them from one facility to the other, we've created some best practices that we use across all of the sites.

Summit is dedicated to training and developing your young engineers which seems to be a core value for Summit. Why has this remained such a focus for you?



Gerry Partida

Whiteside: We have always recognized that developing talent was essential to our growth as a company. The two great companies we started with had great teams, but to really grow and expand, we needed to develop talent within our company beyond what was already there.

We understood that much of the talent across the industry was entering their later career years, and that challenge continues. One's knowledge, particularly in process and engineering, is developed throughout one's career. We saw an urgent need to develop this knowledge and talent more quickly in our employees who were earlier in their careers, so they will be able to take over that brain trust in our company when we lose some of our Gerry Partidas.

Gerry has been leading that effort right from the beginning. We were really excited when IPC launched its Emerging Engineers program a number of years ago, and we were very early adopters and champions of that program.

Partida: I started in this industry as an application engineer for CAM, but my true love was being a trainer. I always got a great deal of satisfaction from explaining things, showing people different skills, and sharing my knowledge. I believed it is extremely important, and I have done it throughout my career. I have met some of the best subject matter experts in this industry by happenstance or perhaps providence. I learned what was important and what was not, directly from them. They guided me to be great in my career.

I saw the Emerging Engineers Program as a way to create subject matter experts in three years, with people entering the industry. These newer engineers are being given an accelerated pathway to knowledge that has taken others three decades to acquire. They will be this industry's subject matter experts in 10 years, the ones who are deciding, engaging, and helping each other for the success of the entire industry. I give them guidance and an entry point into the program, but it's their own curiosity and desire to

learn and excel that create eventual excellence. I'm a witness and a champion for their success.

When you reflect on the past 10 years and your career in the PCB industry, what occurs to you now?

Partida: I've witnessed what has happened in our industry. When I first started, there were roughly 3,000 bare PCB manufacturers in North America. I have seen the slow slide as facilities have moved offshore to Asia, initially for price, but now for capabilities. That has changed how U.S. PCB manufacturing looks. Many remaining companies are using older equipment and are struggling to sustain their dwindling share of the market. It has created its own sort of dystopia where too many U.S. companies have become increasingly less significant and less capable.

I thank Shane for his bold vision to build Summit during a challenging time. Although the U.S. PCB market is still in decline, there are signs of change. I get to help prevent that further deterioration of knowledge, capability, expertise, and better ways of doing things.

On the engineering side, Summit has done some really amazing things, helping our customers build their products better, engaging with them early on for greater reliability in the field, and doing so in the most cost-effective manner. If we can build highly reliable boards, it's easier for us to build with higher yields. That speaks to a better bottom line for both of us. So, early engagement is a win-win, and this is how Summit has worked from the very beginning.

As you celebrate this milestone for Summit Interconnect, does anything come up as particularly pivotal or essential to where Summit is today?

Whiteside: All our acquisitions have been important, starting with the original two companies. We don't consider ourselves to be turnaround experts. We seek out well-run companies that add value to the Summit family. Of our more recent acquisitions, in 2018, we welcomed Streamline Circuits into the family, nearly doubling our size at the time. Streamline was complementary to what Summit offered in the type of technology it specialized in, its quick-turn culture, and the depth of knowledge in its staff.

In 2020, we acquired ITL Circuits, where we saw those same attributes, though their focus was in

much higher volume. They remain one of our higher-volume factories today. In 2021, we bought Eagle Electronics in Chicago, which enhanced our quick-turn offering with a different market and customer base than we had previously. In 2022, we acquired three facilities simultaneously: Advanced Assembly in Denver, Royal Circuit Solutions' PCB operation in Hollister, California, and South Coast Circuits in Southern California.

By staying true to our original strategic plan and vision, we have extended our capabilities, market reach, customer lists, and product profile. I'm really pleased that each of these facilities has grown since becoming part of Summit Interconnect, some by more than two times their size when acquired.

Partida: In the past 10 years, the growth has been there, but it's about more than that, especially coming into the U.S. PCB manufacturing industry when we did. This has also been about connecting knowledge, preserving capability and capacity, and building a stronger platform for our customers and for our national defense here in the United States.

I agree. What's next for Summit?

Whiteside: We like the path that we're on. We have seven manufacturing sites and a headquarters office in Irvine, California. We are further enhancing our value proposition, and we will continue to emphasize growth. We are really excited about growth in the North American market, and we think it's a great time to be in this industry.

Partida: It's been a fulfilling career, and it's more fun now than it's ever been. Yes, it's more challenging too, but in the electronics supply chain, making circuit boards lets us touch so much of what goes on around the world. Being a part of it always makes me excited.

Thank you both for the discussion, and congratulations again to Summit Interconnect on 10 successful years. I-CONNECT007

Sponsored links:

summitinterconnect.com/

summitinterconnect.com/about-us/

Diversification

Where Do You Grow From Here?



Our industry has long operated on a relatively straightforward formula: invest in equipment, build a solid production team, develop reliable manufacturing processes, and compete for business based on quality, delivery, and price. If a shop maintained strong relationships with customers and consistently delivered dependable work, there was every reason to believe the business would continue growing steadily for years.

In many ways, this model built the modern PCB

industry. But the environment surrounding electronics manufacturing has changed dramatically, and I am concerned that many U.S. domestic fabricators have not fully adjusted to these changes. Too many companies operate as though the industry still revolves entirely around producing bare boards, when the market has evolved into something far more complex, integrated, and demanding.

Today, simply being good at fabrication is no longer enough to guarantee long-term success.

NOW HIRING

TECHNICAL SALES MANAGER

SALARY

BASE \$105K – \$161K · OTE \$150K – \$200K +

#1

PCB Laminate Supplier
By Revenue Worldwide

#1

Rigid CCL Supplier
Worldwide

#1

AI Data Center PCB Material
Supplier Worldwide

WHAT YOU'LL DO

- Generate revenue growth across Defense & Aerospace OEM and PCB accounts in the US market
- Serve as technical liaison between customers and Arlon engineering
- Build long-term strategic partnerships in high-reliability and mission-critical programs

REQUIREMENTS

- **Academic Training**
BS or MS in Engineering or Science, or equivalent technical related degree
- **Work Experience**
5+ years in PCB / PCBA industry
- **Work Authorization**
Must be a U.S. Person per ITAR
- **Relevant Skills**
Good verbal and written communications

APPLY NOW

recruiting@arlonemd.com

This position is subject to ITAR/EAR regulations. Applicants must qualify as U.S. Persons.

Arlon EMD · Subsidiary of Elite Material Co., Ltd. · Equal Opportunity Employer · EOE/M/F/Vet/Disability

LinkedIn



In fact, if you are defined solely as a fabrication provider, you place yourself in an increasingly vulnerable position. It is extraordinarily difficult to succeed in a global market dominated by price pressures, rapid technological change, automation, and sophisticated customer expectations.

Domestic fabricators are competing against manufacturers the world over, many of which operate under dramatically different labor structures, government support systems, and cost advantages. With the prevalence of automated online quoting systems and digital procurement platforms, engineers and buyers are uploading designs, comparing suppliers, and receiving pricing from multiple sources faster than ever before. This lends to the perception that a bare PCB is more of a commodity than a specially manufactured custom part, and this reality has fundamentally changed the economics of PCB fabrication.

One may argue total value proposition, but in a global sourcing economy, price inevitably becomes the dominant factor in the equation. For U.S. domestic manufacturers, unless you are strongly aligned with government military and aerospace contracts, the battle is difficult to win. Margins tighten, relationships become transactional, and inevitably, loyalty weakens. Every quote is turning into a race to the bottom.

After watching businesses survive, thrive, and sometimes fail over my many years in the industry, I see diversification as a must for PCB fabricators. More than a simple opportunity for growth, it is becoming a requirement for survival.

Companies best positioned for future success keep PCBs at the core of their business and identity, while building additional layers of value around them. They are transforming themselves from commodity manufacturers into strategic partners, and this distinction matters enormously.

Customers want more than “just” a board supplier. They prefer working with companies that can simplify their work and lives. They want partners that can help solve problems, reduce risk, improve manufacturability, accelerate development cycles, and coordinate multiple aspects of production simultaneously, rather than merely fulfilling a single isolated manufacturing function.

The good news is that this shift creates tremendous opportunity for companies willing to expand their capabilities thoughtfully and strategically. Here are some practical ways your company can look to diversify.

PCB Design and Engineering Support

The model of throwing a PCB design “over the wall” to your PCB fabricator is rapidly disappearing. Much of today’s electronics is simply too complex for purely transactional relationships to remain effective.

One of the most natural areas for expansion is engineering support and design collaboration. Be a company that provides guidance earlier in the development process. Customers need help with stackup optimization, impedance control, thermal considerations, manufacturability analysis, material selection, reliability concerns, and advanced packaging challenges. Be a partner that understands how to improve the overall product design before production even begins. Perhaps one of the best examples of this trend is the emerging practice of co-design. (See Kristin Moyer’s interview in the June issue to learn more about co-design.)

When a company becomes involved at the engineering level, you’re now part of the customer’s development process. This engenders trust and loyalty, resulting in significantly more stable long-term business relationships.

Sourcing and Supply Chain

For years, many domestic manufacturers resisted incorporating sourcing capabilities into their business models, viewing offshore manufacturing as competition rather than an opportunity. But rather than organizing their thinking around industry boundaries, your customers are trying to solve problems efficiently and keep costs down.

They may be developing a complex electronics program that requires simultaneous domestic quick-turn prototyping, offshore production scaling, component sourcing coordination, logistics management, assembly integration, and engineering support. If you can help manage all those moving parts, you are vastly more valuable than if you offer only one isolated service.

Assembly

Historically, fabrication and assembly often operated as separate worlds, but customers increasingly view the process as interconnected. When you can coordinate fabrication, assembly, testing, and logistics under one roof or through a single accountable relationship, your customers gain efficiency, clearer communication, and reduced management complexity. You become easier to work with and harder to replace.

Resilience

We all understand that the PCB industry is cyclical. Demand rises sharply and then contracts. Certain sectors surge while others weaken. Economic slowdowns impact orders rapidly, and supply chain disruptions create sudden instability. When you rely entirely on a single revenue stream, you often feel those downturns more strongly than your more diversified colleagues.

Multiple revenue streams help stabilize operations during uncertain periods. Again, this is where offering engineering or consulting services provides stability.

As manufacturing itself becomes automated, pure fabrication is even harder to differentiate over time. If you add automation, AI systems, and advanced packaging technologies, how will you stack up against competitors with similar access to these capabilities? The future belongs to companies capable of solving broader customer problems rather than simply producing components.

Domain Expertise

Everyone is talking about thermal management. Consider becoming a consultant on reliability topics, which is valuable to customers in the aerospace, defense, automotive, medical, and industrial markets. Become the experts in signal integrity and materials science that customers seek.

Visibility and Authority in the AI Era

The way customers discover suppliers has changed. Engineers are using AI platforms to research manufacturing challenges, compare technologies, explore reliability concerns, and evaluate suppliers. AI is the new authority, and companies that educate effectively will increasingly become the ones that AI

systems recommend most frequently.

In the old days, it was enough to rely on marketing strategies such as capabilities lists, certifications, equipment investments, and promotional messaging. Now, education leadership is a powerful differentiator. Spend the time and effort to publish meaningful technical content, engineering guidance, troubleshooting articles, reliability insights, manufacturability discussions, and supply chain expertise in third-party entities that already have the industry's trust and attention. That visibility influences how your future customers will discover their next supplier.

And the Winner Is...

The opportunity for growth in the PCB industry is not disappearing. In fact, demand for electronics will likely grow for decades. But the structure of that opportunity has changed. Some business leaders resist diversification because they fear losing focus. That concern is understandable, and careless diversification can certainly create problems. However, strategic diversification is entirely different.

The strongest companies will be those that become most useful to their customers across multiple dimensions by creating stronger relationships, greater loyalty, and more sustainable profitability.

The most effective diversification strategies build naturally on your company's current strengths while solving additional problems for your customers. These expansions are extensions of your core business, serving to create a "stickier" customer relationship. Customers tend to remain loyal to companies that consistently make their jobs easier.

Ultimately, the future will not belong solely to fabricators but to problem solvers. This is not because fabrication is less important, but because fabrication alone is no longer enough. **I-CONNECT007**



Dan Beaulieu is president of D.B. Management Group and an I-Connect007 columnist.

When History Repeats Itself on Diversification

BY DON BALL, CHEMCUT

Diversification, as a way to spread out risks,

maintain steady cash flows, and sustain stable earnings in unpredictable business environments, is a concept that has long been used by all businesses, not just PCB fabrication.

Is it really a sound business approach? With the benefit of 20/20 hindsight afforded by over 50 years of experience in the PC fab business, I have formed some definite opinions and observations on this subject, which I will now share with you (with some caveats).

First, I am an outside observer of the situation. I have never been part of the management or economic teams involved in decisions about whether to diversify; I'm just an observer of the results. Second, the opinions expressed here are entirely my own and do not necessarily reflect those of the management. So, with that in mind, let us continue.

One thing I have observed over the years is that the trend to diversify is cyclical. At some point, the advice from the economists is "diversify, diversify, diversify," for the reasons stated above. Then things quiet down for a while, followed a few years later by another economic downturn. Economists now say you should focus on strengthening your core business and avoid wasting time and resources on unrelated businesses.

In other words, the advice from the economic gurus is to de-diversify (undiversify?) and get on with the business you know. A few years later, as the business environment once again becomes more unpredictable (when does it not?), the advice is

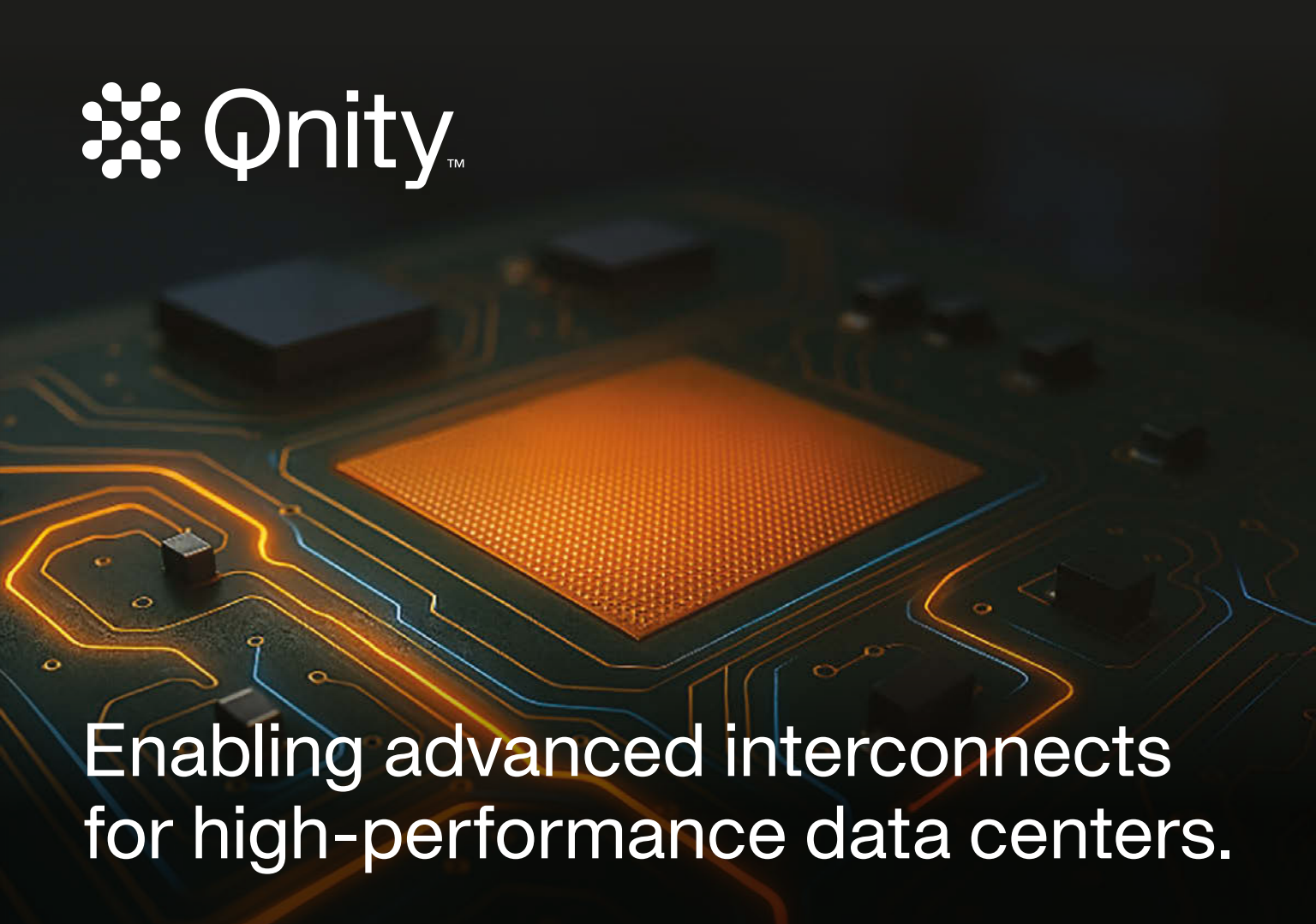
to once again diversify. I have experienced two of these cycles (maybe three) and have come to suspect that the advice to diversify or not is generational and not based on true economic analysis.

Generation 1 of MBA economic gurus needed to give advice during economic uncertainty and advised diversification. They grow old and retire, and when the next generation of MBAs runs into economically troubled times, they advise de-diversification because they must offer some advice, and everyone has already diversified.

Generation 2 grows old, retires, and, at the next economic crisis, the next generation of gurus advises diversification again.

In other words, the advice to diversify may be because that's where they are in the cycle, rather than a new, exciting economic analysis. The reasons for diversification or de-diversification during economic uncertainty are not unsound, but not for everyone. Consider whether your urge to diversify is your own decision or merely following a fad promoted by some economic gurus who may not have their heads in the same world where you exist.





Enabling advanced interconnects
for high-performance data centers.

Qnity powering AI at scale.

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

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

Scan to learn
more about
Kapton®



Scan to learn
more about
Pyralux®



The following will demonstrate the somewhat cyclical nature of the diversification advice.

As I have mentioned in previous columns, in the mid-1960s (even before my time, if you can believe it), management decided we needed to diversify because the world only needed so many etchers, and we needed to prepare for the inevitable downturn. The decision was toward household heating. Much time and effort were spent designing an efficient fuel oil injector for home furnaces that would significantly cut fuel oil costs. Unfortunately, it was discovered that to compete with companies specializing in this business, we would have to go all in and operate the household heating division as a separate business rather than an auxiliary subsidiary of the main business operating in a corner of the factory. We couldn't afford to do that, so the idea was allowed to lapse.

The urge to diversify struck again in the mid-1970s, when management decided it would be a good idea to add a screen printer to our product line, and they purchased a small screen-printing company. Technically, this was not a true diversification since screen printers were a major contributor to the imaging process needed to produce circuit boards at the time, but building and selling screen printers was a lot different than selling etchers. Today, the PCB industry has evolved, and the technical demands on screen printers have become harder to meet. Once again, it was found that to compete with companies that specialize in screen printers, we would have to commit to comprehensive R&D and design effort to stay relevant. We couldn't afford to do that either, so the screen-printing subsidiary was sold to a competitor who gleefully shut it down.

In the same timeframe, we were being acquired as part of someone else's diversification plan. In the early '80s, we were acquired by a German company that supplied electroplating chemistries and vertical plating lines. They, in turn, were a subsidiary of a giant German pharmaceutical corporation. The idea was to use us to introduce their plating chemistry and equipment in the U.S. and to use our equipment as a lure for their wet processing chemistry: dry film strippers, developers, etc.

Things did not go all that smoothly at first because, for some reason, American PCB manufacturers did

not break down the doors to get the German plating chemistry and equipment. Also, the first attempt to package wet processing equipment and chemistry was less than successful. This was a stripper-and-stripper-chemistry deal that gave the shop a free stripper in exchange for a contract to purchase the stripping chemistry. Unfortunately, when they did the initial audit, the sales guys didn't realize the old stripper leaked like a sieve and, as soon as the non-leaking new stripper was installed, the chemical usage dropped radically, and the return on investment ballooned to 10+ years rather than the approximately one year initially calculated.

In the early 1990s, the economy again took a downturn, and the economic gurus at the giant German pharmaceutical corporation thought, "We're making good money selling pills, why are we messing around in the electronics industry?" We were sold to a French oil company, which merged with a larger Belgian oil company, which asked the same question, "Why are we messing around with electronics when we're making money hand over fist in the oil business?" So, in the early 2000s, we found ourselves undiversified, sold to a group of a dozen employees and a local investment firm, and soon began operating as an independent LLC.

Economic unrest is again rearing its head due to COVID and politics, and calls for diversification are on the rise. You may have gathered by now that I am somewhat skeptical of advice given by the economic gurus. You would be right.

If you feel the urge to diversify, be sure it is your decision and not a response to a fad espoused by economic gurus who have no idea about what the PC fab business is all about. Think very carefully about whether you want to get into something totally unrelated to your core business. You may find you will have to put more resources into it than you initially thought. **I-CONNECT007**



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

MicroCraft's Product Range Built for What's Next



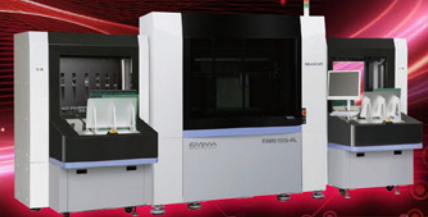
Moving Probe Testers

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



» **E8M6151G-AL**
8-Probe Loader & Unloader

» **E8S6151G**
8-Probe High Accuracy



CraftPix

On-Demand Inkjet Printers

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



» **CPA8165ST**
Stacker Type Loader & Unloader

» **C8K7265T**
8-Head Inkjet Printer



Sarra

Automated TDR & VNA Testers

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



» **S4V54**
4-Probe VNA Tester

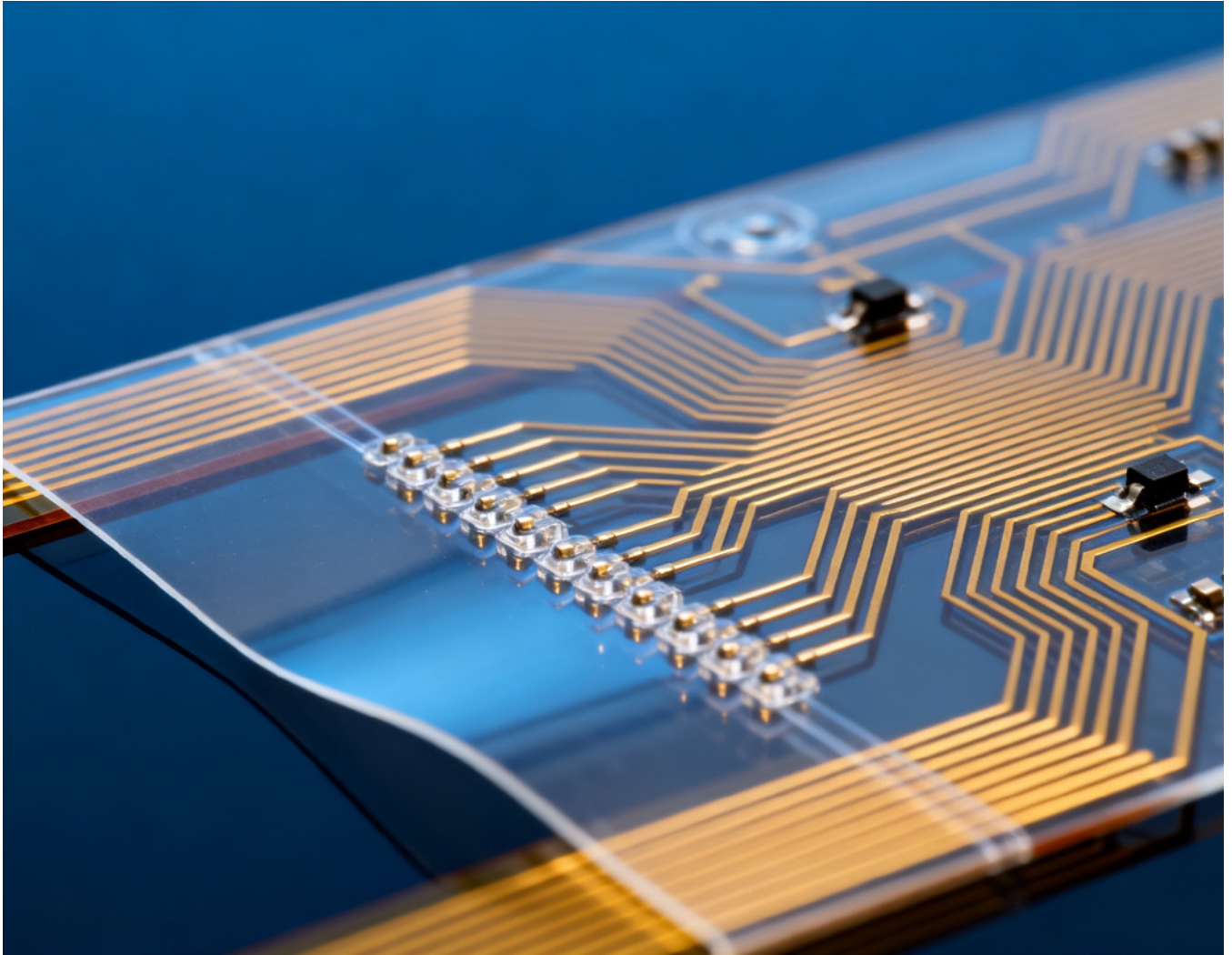
» **H2P45**
2-Probe Variable Pitch TDR Tester



MicroCraft

www.usamicrocraft.com sales@usamicrocraft.com

Beyond the Board Diversification as Strategy



Are the days of simply building boards and shipping them out the door coming to an end? For PCB fabricators and electronics manufacturers alike, diversification is moving from a growth strategy to one of survival. Supply chain disruptions, geopolitical uncertainty, tariff fluctuations, and accelerating technology cycles are forcing companies to rethink how they create value for customers.

At Flexible Circuit Technologies (FCT), that has meant expanding beyond world-class flex circuit manufacturing into assembly, vertical integration,

and a growing global footprint. Ray Cottrell, executive vice president of business development, discusses why diversification matters, how FCT is evolving from a supplier to a strategic partner, and what manufacturers should ask themselves as they prepare for the next wave of industry change.

Marcy LaRont: Ray, when we spoke two years ago, you had just joined FCT and were focusing on an important expansion area for the company. First, take a moment to reintroduce yourself.



WORLD-CLASS TOOLS

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

CALL 949.587.3200 FOR MORE INFORMATION

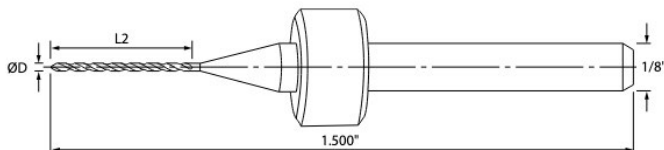


PRINTED CIRCUIT BOARD CUTTING TOOL SOLUTIONS

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

Drills, End Mills, Routers, Specialty Tools

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



4 Facet Point Geometry

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

Ray Cottrell: I'm the executive vice president of business development at FCT and inTFlex, our flex and EMS divisions, respectively. Since Troy Koopman started the business 26 years ago, our core focus has been on value-added flex design combined with world-class Asian manufacturing.



provides incremental value for our clients throughout a product's life cycle.

Ray, this month, we are talking about diversification. A couple of years ago, FCT was looking to build up its assembly operations and grow inTFlex. How did Troy come to those decisions?

It is appropriate to first provide some historical context. FCT has seen a lot of change over the course of almost three decades in business, and the customer has always been at the center of our strategic decisions.

Our customer base is diverse, with EMS and OEMs. We will continue to provide only flex solutions to our EMS partners, unless they request FPCAs. But early on, we had major medical and consumer partners needing advanced flexible circuits and value-added manufacturing services. We needed to invest in new facilities and services to support them, which led to the expansion of our flexible circuit capabilities, including assembly and full turnkey manufacturing.

Over the past two years, we have seen new disruptions arising from market needs, geopolitical risks, and tariff uncertainty. FCT has been adapting to that while planning initiatives to better position us to support our clients now and in the future. We continually ask ourselves, "How can we bring additional value to our clients around our core engineering strength?" To that end, we have increased our printed electronics division, expanded our flex offering to include catheters, and offer a low-cost manufacturing strategy with world-class facilities in China and Thailand.

We have also added injection molding with sterilization for medical products and expanded our EMS offerings in China and Thailand that complement our strong engineering presence in the U.S. Our design for excellence (DFX) approach

Can you talk more about DFX? With the acceleration of technology, we need more collaboration built into the whole process. There's co-design, new CAD tools, etc. What's your value-add on the design side?

Ideally, we engage with clients as early as possible on their innovative flex and manufacturing needs. If we can impact

their complex flex designs on the front end, for example, a medical client with catheters (long circuit with tight trace and space), we can help bring their products to market faster. It really helps to work hand in hand with an OEM's design and R&D teams when they are developing a new design. We have some wickedly talented flex engineers who have spent 35 years doing nothing but flex design work, and their expertise is widely sought.

Two examples are Mark Finstad and Chris Clark, whom many of your readers may know. They sit on several IPC committees to support, guide, and help write the standards for flexible circuit design and manufacturing. That level of expertise with our expanded global manufacturing teams allows us to design for improvement as product moves through the R&D cycle, and into design for test and design for manufacturability.

That is how we partner with the OEMs in all the markets we serve: medical, industrial, robotics, wearables, and consumer.

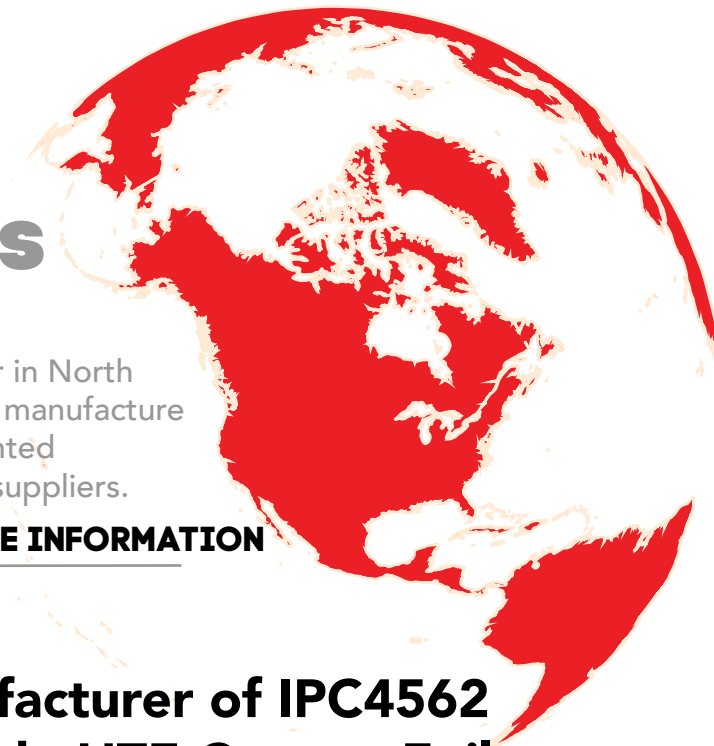
How has this informed your development in the manufacturing part of the ecosystem?

Of course, our design involvement often considers forward-looking production pricing, lead times, and options for manufacturing. We have invested in new advanced electronics manufacturing capabilities at our flagship inTFlex facility in China. This was a natural evolution by offering high-quality EMS in-house, including FPCAs, PCBAs, plastics, and sub-assemblies through full box builds. In 2026, we added Thailand to support our value-added EMS offerings and further diversify our footprint in response to client needs.

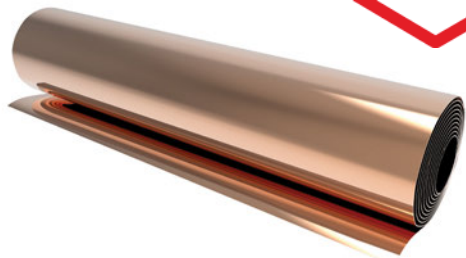
WORLD-CLASS SUPPLIERS

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

CALL 949.587.3200 FOR MORE INFORMATION

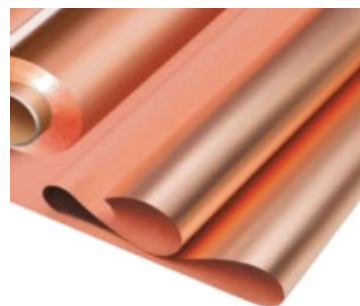


Manufacturer of IPC4562 B Grade HTE Copper Foils



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

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



Conformal Release Film

Pacothane provides innovative lamination

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



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

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

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



Precision is our specialty!

We are looking ahead for further geographic diversification in Vietnam, including full flex PCB fabrication in late 2027 and additional EMS too. So, there is a lot of investment going on.

How many employees does FCT have, and what does each region look like from a staffing and logistics viewpoint?

We have around 1,100 employees in China and Thailand, and 35 in other areas around the globe. We added commercial teams in Europe, Latin America, and the U.S. over the past two years. In the U.S., we have program managers, customer service talent, CAD teammates, and our world-class flex design engineering team. It is a unique offering in a space that has long existed but is changing rapidly.

So, in terms of business diversification, you are expanding services, both in design and assembly, and you've expanded your global footprint to de-risk and to mitigate geopolitical challenges, to be a better supply partner to your customers. That is a lot of capital investment. How important is access to capital, not only to survive, but to grow?

With all the changes in technologies that are driving the industry, you must be a company that is financially stable with working capital to weather the next storm, as well as invest in the future. We have seen so many competitors over the past few years who haven't been able to make it, have been acquired, or are looking for new partnerships just to stay viable. Thankfully, with Troy's backing and our financial success over the last 26 years, we're



The FCT/inTFlex Huizhou, China facility.

WORLD-CLASS SUPPLIERS

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

CALL 949.587.3200 FOR MORE INFORMATION



REVOLUTIONARY LAMINATING & BONDING TECHNOLOGY



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

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

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

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

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

230N - Next-generation inductive bonding machine



CAC

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

well-capitalized while staying focused on delivering value to our clients. There is a lot of innovation happening across the flex ecosystem.



We are focused on the value of having advanced electronics contract manufacturing in-house for our clients.”

Early supplier involvement can reduce revision cycles and costs, but in rigid PCB manufacturing, there are often several layers between the fabricator and the OEM. Flex seems to encourage closer collaboration. Is that level of engagement with customers and OEMs fairly seamless for FCT?

Yes, that is our business model. Early engagement depends on the company, their resources, and where they are in their design cycle, but we are active in design support as early as possible. When we become engaged with a customer, it really spans the entire product development and product lifecycle. We get clients that come to us very early on, and those that come to us only once they have a challenge. There are also customers that reach out after their initial product has been fully developed, seeking cost reductions as they look to scale.

FCT has made significant investments to expand beyond flex fabrication and into assembly and other services. What drove that decision, and how does it fit into the company's long-term strategy?

We have made it 26 years. To continue thriving, we knew we needed to invest in the future by building new facilities and capabilities. Over the past four years, we have invested about \$60 million in the latest flex and EMS equipment and in new facilities to offset some of the risks we discussed. However, our expansion is more about looking ahead and figuring out how we can deliver additional value and become more vertically integrated by offering plastics, sterilization, enhanced flex capabilities, and full EMS services.

We are focused on the value of having advanced electronics contract manufacturing in-house for our clients. This includes die bond, wire bond, and flip chip for the latest PCBA and FPCA designs. With our facilities in China, Thailand, and soon, Vietnam, we bring flex solutions and products to market and offer our clients a vertically integrated solution with risk mitigation in mind.

For all of us, tariffs remain a moving target, and we regularly have conversations with our clients about the options we provide. This includes warehousing, stateside logistics, and distribution offerings. We feel that being vertically integrated is a necessity.

Ray, end-to-end, other than being well-capitalized, what advice do you have for PCB fabricators looking to diversify? What questions should they ask, and what do they need to be successful in this attempt?

We feel diversification is fundamental to survival in today's market, whether that is expanding into a new geography or adding new services. For example, if you are only in Vietnam, looking at another country to support your manufacturing strategy and mitigate risk is probably warranted. What other value can you bring to your customers? What is your access to capital? The bottom line is to find ways to offer real value to the market and your clients. Otherwise, your customer will look for another supplier that does more than just the bare PCB or just flex. You want to simplify their supply chain and be a better supplier for them in the long term.

It has been good talking with you, Ray. FCT is a good example of strategic diversification to be a true solutions provider for your clients.

It has been a fun two years, and we're really looking forward to the future. Thank you, Marcy. **I-CONNECT007**



ADVANCED ELECTRONIC PACKAGING CONFERENCE

Component-to-System-Level Integration

april 3-8, 2027

anaheim convention center · ca
meetings · courses · april 3-8
conference · april 5-8
exhibition · april 6-8

The Call for Participation is now open for the Advanced Electronic Packaging Conference (AEPC) 2027 – *Component-to-System-Level Integration*.

Share your latest research, innovations, and best practices with the global electronics community by submitting a proposal for a:

- Technical Presentation
- Technical Poster
- Professional Development Course

This is your opportunity to showcase your expertise, build your professional reputation, and help advance the technologies shaping the future of electronics packaging and manufacturing.

IMPORTANT DATES

Technical Presentations

Abstract Submission Deadline: September 18, 2026

Professional Development Courses

Abstract Submission Deadline: September 18, 2026

Technical Posters

Abstract Submission Deadline: December 4, 2026

**JOIN THE INDUSTRY'S LEADING EXPERTS AND HELP ADVANCE THE TECHNOLOGIES THAT
CONNECT COMPONENTS, PACKAGES, BOARDS, AND SYSTEMS.**

SUBMIT YOUR ABSTRACT TODAY →

Learn more and submit at apexexpo.org/event-info/call-participation

DFM Mistakes Designers Make When Moving to ELIC

Editor's note: This is the second part of an article series on every layer interconnect. [Read part one here.](#)

The transition from HDI to every layer interconnect (ELIC) is where the DFM gap between design intent and fabrication reality is most visible. Several common misconceptions appear consistently in the data packages we review at ASC.

The most common ELIC design mistake mirrors what we generally see in first-time UHDI builds: using process minimums everywhere because they appear on a capability chart. In ELIC, this is especially consequential. If only one region truly requires

aggressive geometry, say, a fine-pitch BGA escape, the rest of the design should preserve margin. Relaxing non-critical areas makes the overall build more manufacturable without sacrificing the dense region that drives the stackup.

Production capability charts reflect what is achievable under controlled conditions. Production introduces material variation, panel movement, plating distribution effects, and statistical process spread. ELIC compresses those windows. Small adjust-

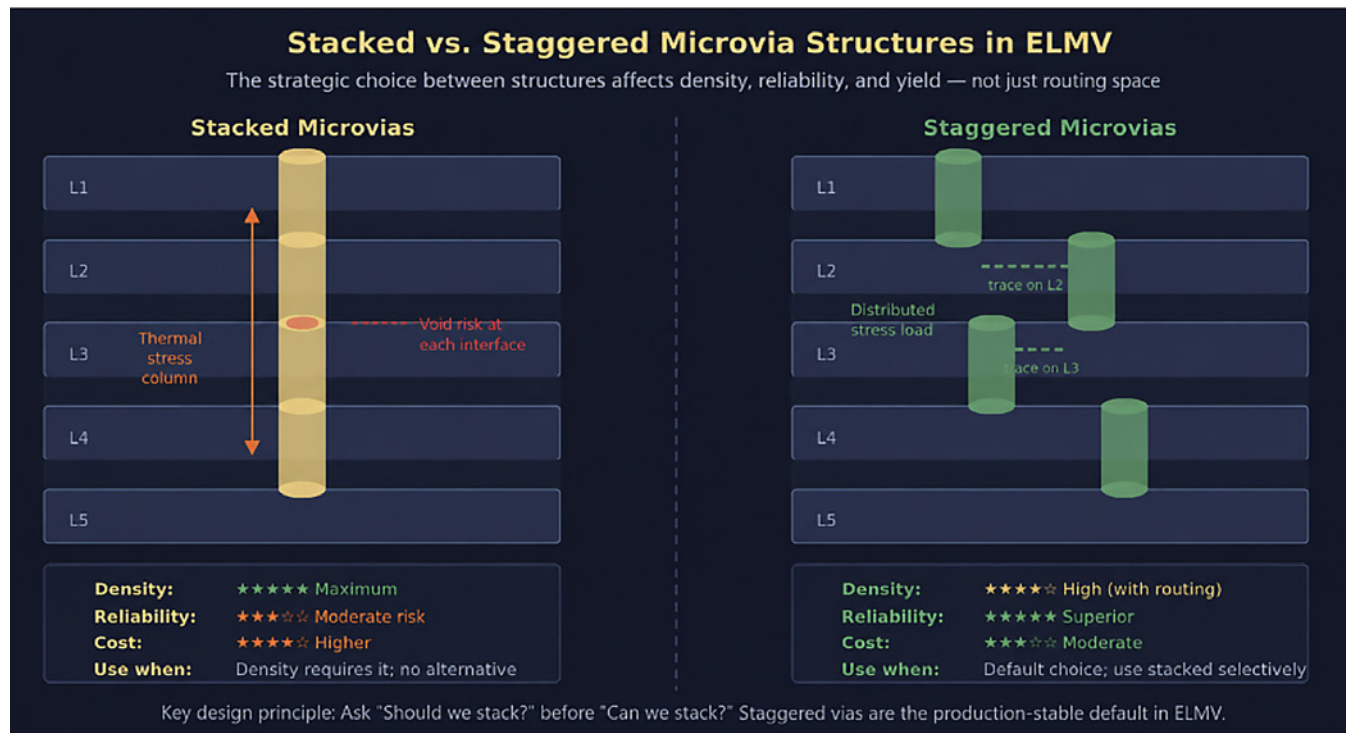


Figure 1: Stacked microvias maximize routing density but introduce a thermal stress column and cumulative void risk at each interface. Staggered vias distribute mechanical load and are the production-stable default. The strategic question is “Should we stack?” not “Can we stack?”

Providing Solutions that Advance the Manufacturing of Electronic Products



VIRTUAL REALITY ROOM

- Experience the process of innovation
- Transforming ideas into solutions
- How technology and sustainability can grow together



GEMINI DIRECT IMAGING SERIES

- High performance dual table
- Fast printing all resist and mask types
- Best in-call soldermask



SMARTLAM MODEL 5200

- Cut sheet laminator for full range of PCBs
- Stainless-class 1000 clean room construction
- Excellent temperature uniformity



ANTARES 700X

- World leader in automation solutions
- Pneumatic and electric spindle options
- Auto load/unload option (700XA)



TECHNICA, U.S.A.

technica.com



ments, such as widening a via capture pad by a few microns or relaxing a mask dam in a non-critical area, can shift a build from marginal to stable.

Via Stacking Rules That Do Not Transfer From HDI

Designers who have mastered HDI sometimes carry stacking rules into ELIC without recognizing that aspect ratio and fill requirements change with each additional level. A microvia correctly sized for a two-level stack may not achieve equivalent fill quality in a four-level stack. The target pad must be larger to account for positional uncertainty across multiple build cycles.

The 1:1 aspect ratio guideline is the foundational anchor: a 50 µm via requires an approximately 50 µm dielectric. This relationship affects impedance control, copper thickness, lamination cycles, and long-term reliability simultaneously.

Designers often miss the guidance that if a design terminates multiple ELIC via structures on external layers, the UHDI circuitry should be moved to internal layers. External layers carry the greatest cumulative registration uncertainty from the full build sequence, and placing multiple via terminations there compounds both the registration risk and the solder mask complexity.

In Table 1, staggered vias are the production-stable default in ELIC. Stacked vias should be used selectively, only where density requirements leave no alternative.

Treating Solder Mask as a Finishing Detail

In ELIC, the solder mask directly affects pad definition, isolation, assembly yield, and manufacturability. When pad openings approach 60 µm and mask dams shrink toward 50 µm, alignment tolerances do not scale proportionally. A layout that looks acceptable in CAD can become difficult to build if the mask strategy assumes more positional precision than the process can comfortably deliver.

Designers should evaluate the solder mask strategy before the layout is finalized. Key questions:

- Are solder mask-defined (SMD) pads appropriate for this pitch?
- Are narrow dams realistic?

Mixing SMD and non-solder mask-defined (NSMD) pads within a single fine-pitch BGA introduces variability that creates assembly challenges downstream.

Ignoring Thermal Implications of Eliminating Through-holes

Designers sometimes choose ELIC to maximize signal routing density without fully modeling the thermal implications of removing through-holes from the heat dissipation path. Through-hole copper barrels are effective thermal conductors from components to internal planes. When replaced by an all-microvia architecture, the thermal resistance

Table 1: Understanding this gap is foundational to ELIC design. Minimum capability is a process boundary.

DESIGN CAPABILITY	KEY BENEFIT	TRADE-OFF
Stacked microvias	Maximum routing density	Higher cost and complexity; cumulative stress column; void risk at each interface
Staggered microvias	Superior mechanical reliability; distributed stress load	Slightly more routing space required for lateral trace connection



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
- Blue
- Black
- Red



We Also Offer Flexible Soldermask:

- Green
- Amber



Key Features/Highlights

High Performance

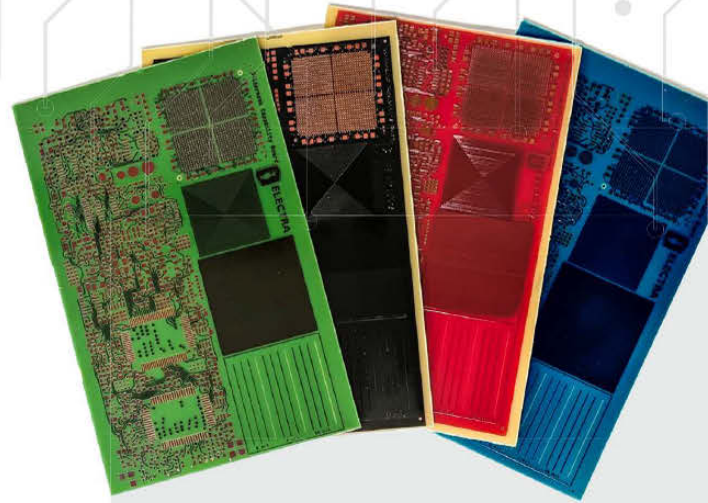
- EMJ110 designed to meet all key performance criteria
 - IPC SM840, UL-94V-0, no SVHCs, thermal cycling and heat storage

Robust Formulation

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

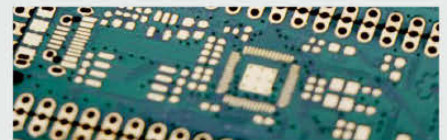
Reliability

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



Why Choose ElectraJet® EMJ110?

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



Join the Future of Soldermask Technology

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

Contact us today to learn more!

of the vertical interconnect structure increases. For power-intensive applications, this requires explicit thermal simulation and, in some cases, the strategic retention of thermal vias even within an ELIC architecture.

Design to a Real Process Window

This principle governs every other decision in ELIC. The right design target is not the best-case geometry on a capability chart. It is the actual process window the fabricator can repeatedly hold across production panels, material lots, and environmental variation. Preserving an extra increment of capture pad size, mask dam width, or layer-to-layer spacing where possible often has an outsized impact on yield and repeatability, and far greater than its apparent cost in routing space.

Early engagement with the fabricator answers the questions that matter most before the layout is locked: Is this mask strategy realistic? Is this capture

pad too aggressive? Where is registration most likely to drive yield loss? Which areas would benefit from a little more margin? That collaboration, before routing is complete, is where the most impactful DFM improvements happen.

Reviewing for Movement, Not Just for Clearance

Traditional design review asks: Do these features clear each other? ELIC review should ask: Do these features still work if they move slightly? That shift surfaces the right issues, where copper is barely protected, mask margin is smallest, vias are landing with the least tolerance, and structures depend on near-perfect alignment. It is a simple habit change with an outsized impact on first-build yield.

How Do Yield, Cost, and Cycle Time Scale?

Yield in ELIC is driven primarily by the cumulative defect opportunity at each sequential build cycle.

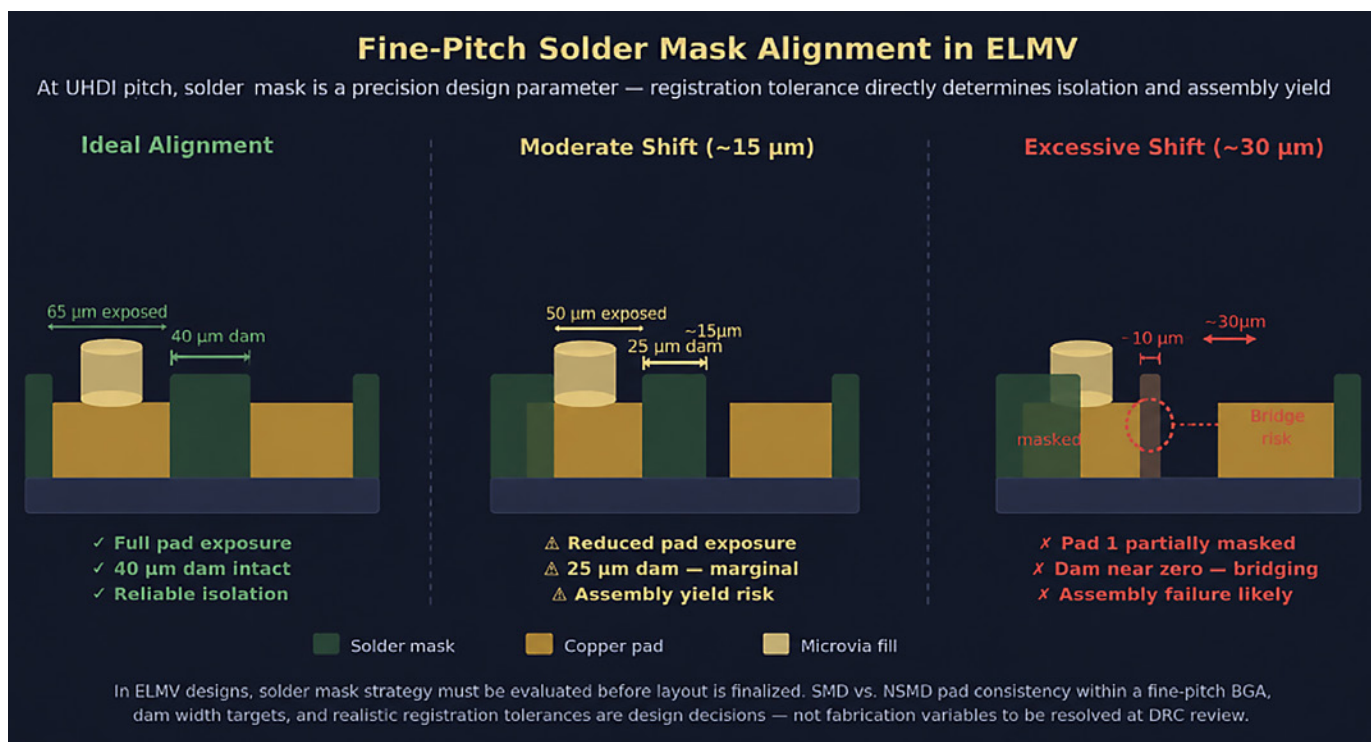


Figure 2: At UHDI pitch, a 30 µm mask shift can reduce dam width to near zero and partially mask an adjacent pad. Solder mask strategy, including SMD/NSMD pad definition, dam width targets, and alignment tolerances, is a design decision, not a fabrication variable.

In a conventional multilayer board, yield is relatively linear. In ELIC, yield is multiplicative across cycles. Based on our production experience at ASC, if process capability at each build cycle delivers 99% yield per cycle, a six-cycle ELIC build produces a cumulative yield starting point of approximately 94% before final electrical test, AOI, and assembly. A fabricator running 97% yield per cycle on the same six-cycle build arrives at a starting yield below 84%. That gap compounds with every layer and is the single largest driver of cost variability between ELIC suppliers.

This is why process maturity at each individual build step matters more in ELIC than in any prior technology category. Evaluating ELIC suppliers on production yield data for comparable builds—not just stated capability limits—is the right approach.

In ELIC, design variables are tightly interconnected in ways that have no parallel in standard HDI.

In Table 2, when a fabricator says “it depends” in response to an ELIC design question, this is the reason. No single variable moves independently. This is what makes ELIC closer to semiconductor packaging than traditional PCB manufacturing.

When ELIC Is Viable vs. Impractical for Production

The tipping point is best understood through three lenses: technical necessity, volume economics, and supplier readiness.

On the technical side, ELIC becomes the right choice when the design cannot be executed in a 2-4-2 or 3-2-3 HDI architecture. If a designer can achieve their density requirements with three or fewer sequential build-up cycles, they should do it. The yield and cost model will be better, and the supply chain is broader. Four or more build-up cycles is generally where ELIC becomes the appropriate architecture.

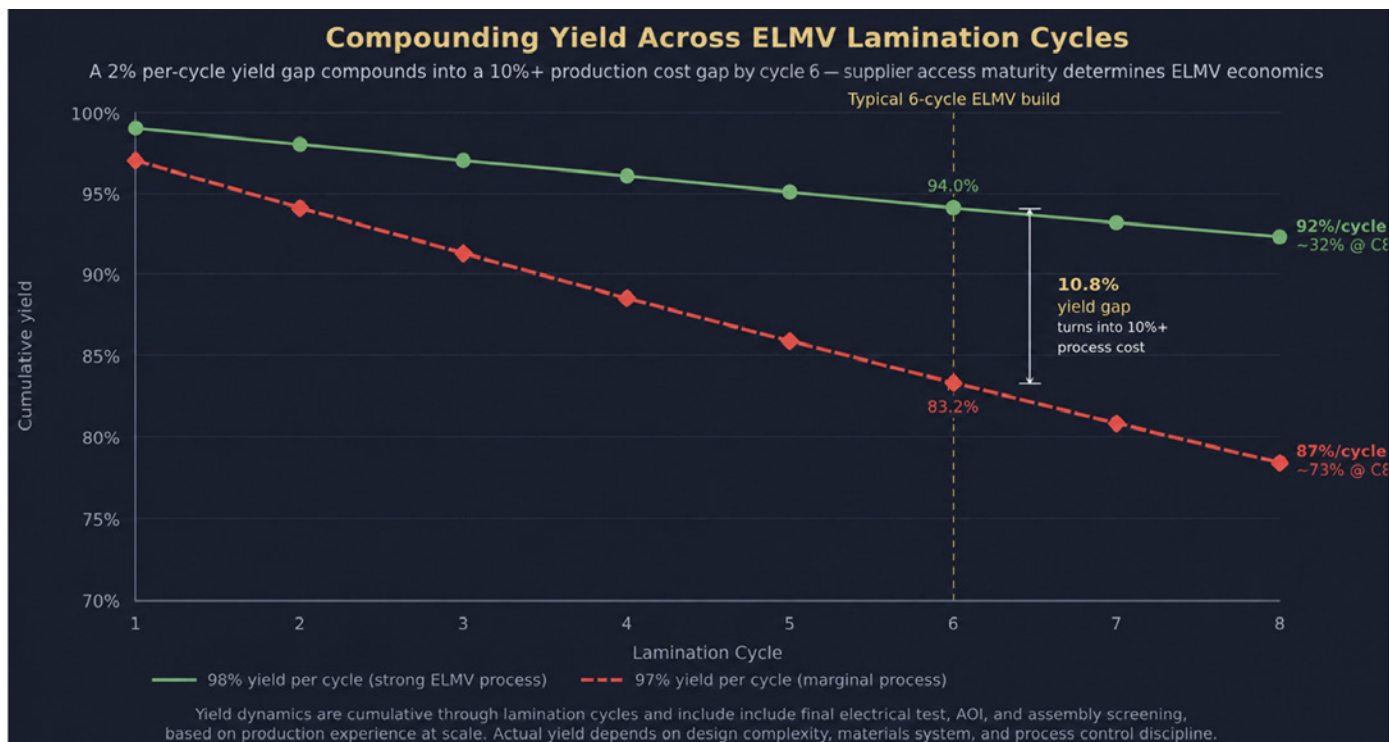


Figure 3: A 2% per-cycle process capability gap compounds into a 10.8% yield difference by cycle 6. This is the single largest driver of cost variability between ELIC suppliers, and the reason supplier process maturity matters more than stated capability limits.

ELIC Design and Fabrication: Frequently Asked Questions

Why choose ELIC over traditional HDI or stacked microvia HDI?

ELIC eliminates mechanical through-holes as the primary vertical interconnect, making every layer available for routing. This enables higher density than stacked or staggered HDI, supports sub-100-micron pitch BGA escapes, and reduces layer count, but at the cost of significantly more sequential lamination cycles and tighter process controls.

How dense can an ELIC board feasibly be in production?

A well-executed ELIC stackup can support line and space geometries down to 25 μm with HDI processes, and sub-25 μm with semi-additive processes such as SAP. Designs that would require 20+ layers in conventional multilayer can often be executed in 12–14 layers. Practical density limits are set by fabricator process capability and yield economics, not by theoretical geometry.

What is the biggest single challenge in ELIC fabrication?

Registration across the full sequential stack is the most systemic challenge. Because ELIC is built from the center outward—one layer added to the top and one to the bottom with each cycle—the outermost layers carry the greatest accumulated positional uncertainty. A shift of $\pm 10\text{--}15\text{ }\mu\text{m}$ that is inconsequential in standard HDI can materially reduce microvia landing margin in a six-cycle ELIC build. Via fill quality at each level is equally critical: incomplete fill (a known failure mode documented in IPC-WP-023) creates voids that standard AOI may not detect and that can cause field failures after assembly.

What are the most common design mistakes when moving to ELIC?

The most common mistakes are: 1) designing at minimum process capability everywhere rather than preserving margin in non-critical areas; 2) applying HDI via stacking rules without adjusting for the aspect ratio and fill requirements of deeper stacks; 3) treating solder mask as a finishing detail rather than a precision design parameter; 4) terminating multiple via structures on external layers without moving UHDI circuitry internal; and 5) not modeling the thermal implications of eliminating through-holes from the vertical heat dissipation path.

Where is the tipping point after which ELIC becomes viable or impractical for production?

ELIC becomes the appropriate architecture when designs require four or more sequential build-up cycles and cannot be routed in a 2-4-2 or 3-2-3 HDI configuration. The yield model is multiplicative: at 99% yield per cycle, a six-cycle build starts at ~94% cumulative yield before test and assembly. At 97% per cycle, that drops below 84%. This compounding effect means supplier process maturity, not theoretical capability, determines production viability.

When should the fabricator be involved in an ELIC design?

As early as possible, ideally before the stackup is finalized. By the time routing is complete, the structural decisions that drive yield and reliability are already locked in. Early fabricator engagement prevents the most costly DFM gaps: stackup over-aggression, via aspect ratio mismatches, solder mask strategy errors, external-layer via termination issues, and thermal management assumptions that do not survive the removal of through-hole copper.

Table 2: The impact of different design variables on ELIC.

DESIGN VARIABLE	DIRECTLY AFFECTS
Via aspect ratio	Dielectric thickness ↔ copper fill quality ↔ reliability
Registration tolerance	Stack depth ↔ capture pad size ↔ cumulative yield
Solder mask definition	Copper spacing ↔ pad isolation ↔ assembly yield
Lamination cycle count	Material thermal exposure ↔ CTE behavior ↔ Z-axis stability
Stacked vs. staggered via	Thermal cycling durability ↔ void risk ↔ fabrication cost

On the volume side, the economics improve with scale but not as dramatically as in standard technology. The fixed costs of process development, material qualification, and inspection tooling are higher and must be amortized across production quantities. For low-volume, high-complexity programs, which describe much of the aerospace and defense work we support at ASC, the per-unit cost is high but justified by the performance and form factor advantages ELIC enables. For high-volume commercial applications, the yield model must be very well characterized before production transition, because the cost of a field failure driven by a latent via-stack defect is orders of magnitude greater than additional upfront process development.

Supplier Readiness: The Underappreciated Variable

ELIC cannot be attempted without prior investment in equipment, materials, process chemistry, and the engineering talent to manage their interactions. At ASC, our path into ELIC capability was built on years of HDI production experience, our UHDI platform with SAP, and targeted investments in laser drill, registration, and electroplating systems specifically sized for ELIC tolerances.

Organizations evaluating ELIC suppliers should ask questions: What is your production yield data on comparable builds? What are your cross-section documentation protocols at each build interval? How deep is your process engineering support during design development? Those conversations will tell you more about a fabricator's actual ELIC capability than any capability statement document.

Closing Perspective

ELIC is a powerful technology. For the right design, in the right hands, it delivers density and performance capabilities that are not achievable through any other PCB fabrication approach short of IC substrate technology. But it is not a casual upgrade from HDI. In ELIC, the small things are no longer small.

The designers and programs that get the most out of ELIC are those who engage early, share design intent before routing is locked, design within production-ready ranges rather than minimum-capability limits, and choose fabrication partners with demonstrated process maturity. At ASC, we have built that capability deliberately and incrementally, and we look forward to the continued evolution of this technology as the semiconductor packaging landscape makes ELIC not just compelling, but necessary. **I-CONNECT007**

Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit*

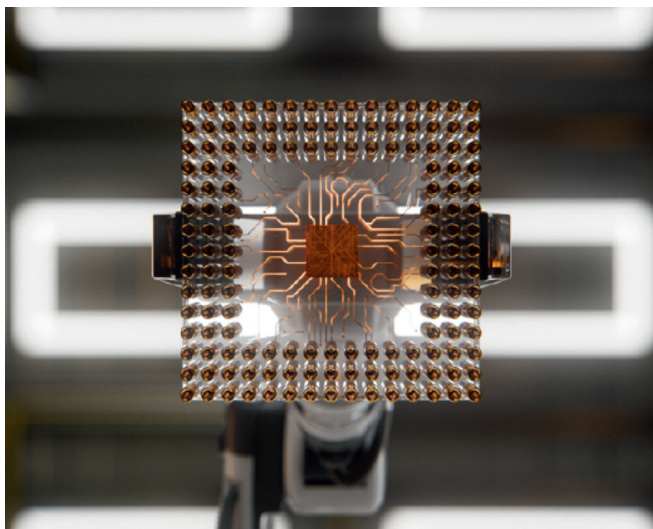


Designer's Guide to... Fundamentals of RF/Microwave PCBs and Flex and Rigid-Flex Fundamentals. He is the author of *Thermal Management: A Fabricator's Perspective*, *The Printed Circuit Designer's Guide to DFM Essentials*, and *The Companion Guide to Flex and Rigid-Flex Fundamentals*.

Visit [our library](#) to download these and other free, educational titles.

Wet metallization for glass substrates – From market trend to manufacturable flow

Glass is transitioning from pilot programs to commercial manufacturing because it offers dimensional stability, low warpage, smooth surfaces, and a coefficient of thermal expansion (CTE) similar to silicon's. These properties make glass highly attractive for heterogeneous integration, large-panel processing, and increasingly demanding AI/HPC package architectures. However, these properties also make metallization more challenging because glass is chemically inert and mechanically brittle.



Trends and manufacturing context

Glass is becoming an increasingly relevant option for advanced packaging applications that require low loss, tight overlay, and through-substrate interconnects. For PCB and substrate manufacturers, the key question is no longer whether glass will play a role, but how plating on glass can be translated into a stable, manufacturable panel-level process that meets both technical and economic requirements.

The solution – A three-step process

MKS' Atotech has developed a three-step wet metallization process for plating on glass. This process is designed to address three main challenges of the material: reliable adhesion, conformal seed coverage, and void-free TGV filling. The strength of this process lies in its sequential nature, with each step preparing the foundation for the next and creating a coupled process flow that supports downstream fine-line manufacturing.

1. Vitrocoat® GI – Adhesion promotion.

A nanometer-thin metal oxide layer forms a chemical bond to the glass which is then able to support wet metallization. This improves adhesion to inert glass surfaces, eliminating the need for a sputtered adhesion layer.

2. Cupratech® GI M – Electroless copper seed.

An autocatalytic copper seed layer is deposited onto the pre-existing vitrocoat layer. This provides uniform deposition across the glass surface, with unrivalled coverage into high-aspect-ratio TGVs, which can't be achieved with traditional PVD or "sputtered" methods.

3. InPro® Pulse TGV – TGV filling.

Reverse pulse plating guides copper growth toward the via center. This enables void-free filling with low overburden. It addresses one of the main challenges in TGV metallization and helps ensure electrical, thermal, mechanical, and surface planarity for downstream processes.

Outlook

Manufacturers that establish glass metallization capabilities during the current qualification phase will be better positioned for the next production increase. With panel sizes up to 515 × 510 mm already demonstrated, the focus is shifting towards industrialization, glass handling discipline, and process integration rather than basic feasibility.

Conclusion

As the use of glass substrates becomes more widespread, wet-chemical metallization provides a practical solution to the material's three main challenges: adhesion, conformal coverage, and void-free TGV filling. The above-described process creates a manufacturable path for plating on glass by combining adhesion promotion, electroless copper seeding, and reverse-pulse TGV fill in one coupled sequence. This sequence supports the fine-line downstream requirements of next-generation package substrates.

Roger Massey

Senior Manager Technical Marketing
Manager and Global Product Manager PTH/
STT at MKS' Atotech
roger.massey@mks.com
+44 7881 26 56 92



Metallizing glass for the next generation of substrates

A wet metallization process approach from MKS' Atotech for scalable adhesion, seeding, and TGV filling

AI and HPC are pushing substrate technology beyond the limits of traditional organics. Glass is emerging as the next platform, enabling finer redistribution layers, tighter registration, and higher interconnect density. MKS' Atotech wet metallization process solution is built to solve the key challenges of plating on glass, such as adhesion, conformal seed layer deposition, and reliable TGV filling, helping turn that promise into scalable manufacturing. The process features VitroCoat® GI, Cupratech® GI M, and In-Pro® Pulse TGV and supports high aspect ratios, copper-only seed removal, and panel-level productivity.

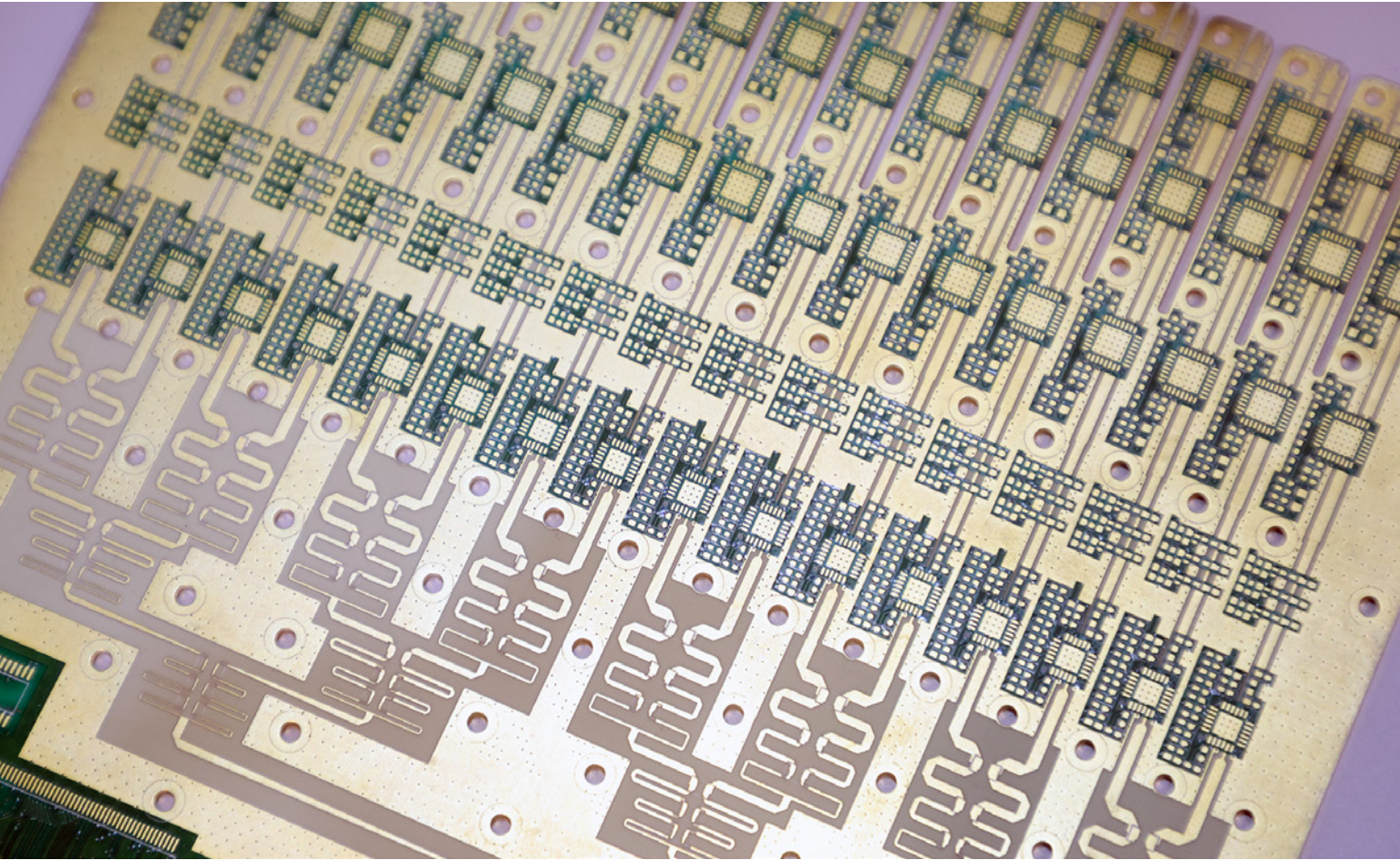
Contact us to learn more about moving glass metallization toward production scale.

info-atotech@mks.com



www.atotech.com

At 40 GHz, *Everything Matters*



BY MICHAEL CARANO, GLOBAL ELECTRONICS ASSOCIATION

In Part 1 of this article, we examined alternative solderable finishes for 5G/6G applications and reviewed literature on their performance at high frequencies.

We found that certain solderable finishes are more susceptible to signal loss as frequency increases. These attenuation losses become increasingly significant at frequencies approaching and exceeding 40 GHz. Because of their high conductivity, silver and

copper generally outperform finishes such as nickel and palladium. Taken together, these findings underscore the importance of finish selection in high-frequency applications.

Part 1 described, however, that modified nickel (higher phosphorous content, thinner deposit, smoother topography) mitigates some of the signal loss when compared to conventional electroless nickel deposits. But this may not be a sufficient

Get the **CapStone™** Advantage



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

CapStone™

ACCURACY

MAXIMUM AVERAGE VELOCITY

MAXIMUM DRILLING VELOCITY

POWER CONTROL

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

>10 mm/s

10,000 mm/s

Precision™ Pulse real-time

Discover the CapStone advantage – www.ESI.com



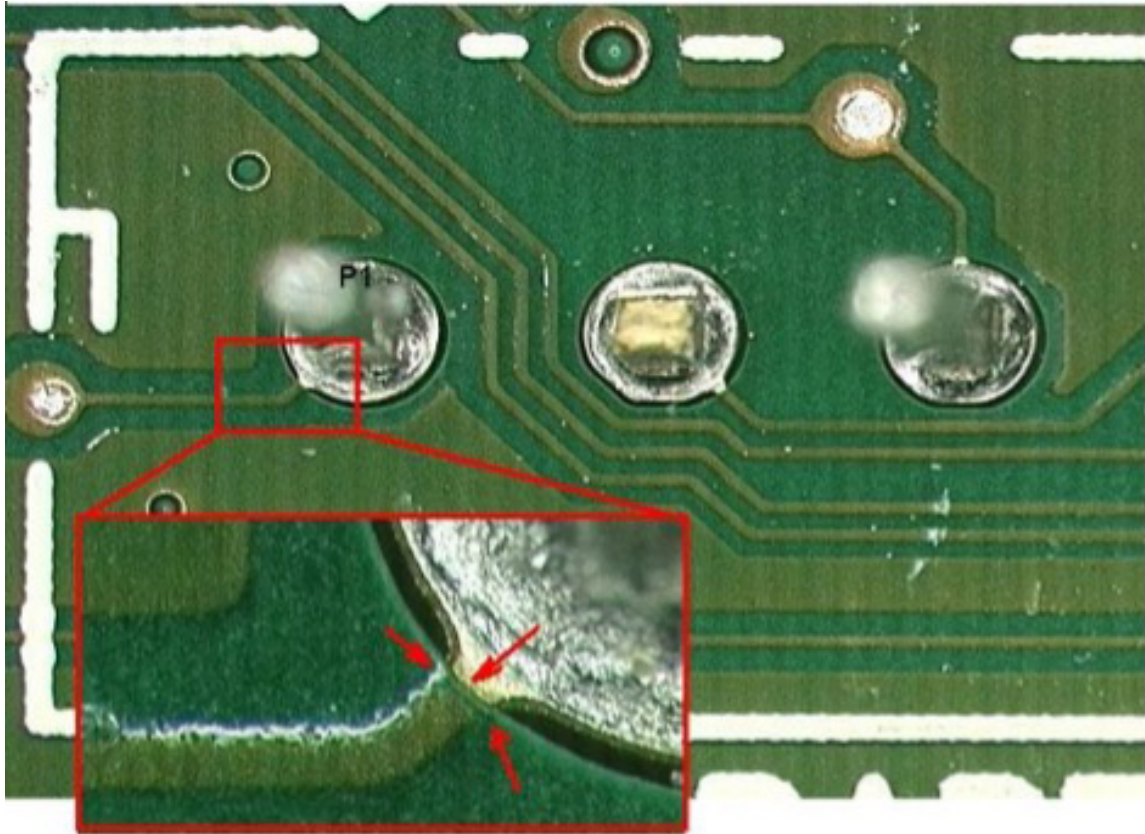


Figure 1: Solder mask interfacial attack (SMIA).

solution as designers are faced with 1.6T/3.2T 6G. We are seeing frequencies above 77 GHz and beyond. So, what is the answer?

Immersion Silver-Immersion Gold (ISIG)

Immersion silver has been used commercially as a solderable finish for more than 25 years. Its high conductivity makes it especially well-suited for high-frequency applications, where signal integrity becomes increasingly important.

Alkaline Immersion Silver

Most immersion silver processes are acid-based. The process described here uses an alkaline formulation as an alternative approach. This chemistry was developed to support high-frequency performance while addressing some of the limitations associated with conventional acid-based systems.

Development of ISIG

To further enhance the performance of alkaline immersion silver, a specialized immersion gold process was developed for deposition over the

silver layer. This combined finish, immersion silver-immersion gold (ISIG), was designed to improve signal integrity and reduce attenuation loss in 5G applications.

The initial development effort focused on immersion silver alone, since silver is already recognized for its ability to improve high-frequency performance because of its superior conductivity. However, concerns about silver tarnishing through environmental interaction led to the addition of a thin immersion gold overcoat, creating a more robust dual-finish system.

The alkaline formulation of immersion silver has mitigated one of the most annoying issues associated with acidic immersion silver processes: solder mask interfacial attack. This is often known as trench etch (Figure 1). The interfacial attack is often blamed for the genesis of creep corrosion, which is often related to exposure of the circuits to harsh environments.

Nitrate-based formulations are overly aggressive on copper. This leads to voiding along the



Figure 2: Voids also known as champagne bubbles.

surface of the copper during the galvanic reaction. As copper is displaced, the silver deposits on the copper surface, forming a silver “roof” over the voided area. Upon assembly, the silver dissolves, leaving microvoids as shown in Figure 2. This issue was named “champagne bubbles.”

The alkaline-based process will not attack the copper as aggressively as any nitrate-based process. In addition, the silver ions in the alkaline silver process are chelated. This allows for more controlled deposition of silver. This results in a denser, less porous silver deposit.

While gold is slightly less conductive than silver, the benefits of an immersion gold deposit would outweigh any additional signal loss due to the presence of gold over silver. In addition, gold provides

additional protection for the silver and enhances wire-bonding performance. With a thin coating of immersion gold, the underlying silver is protected from tarnishing.

In general, the ISIG process presents a more controllable and efficient final finish for 5G/6G. The result of a nitrate-free immersion silver process is a deposit with insertion loss identical to copper, and now providing protection very close to electroless nickel. This makes nitrate-free immersion silver a viable process for 5G and millimeter-wave applications well into the future. **I-CONNECT007**



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

jobCONNECT007

**INDUSTRY-FOCUSED
HIRING
STARTS HERE.**

The Right Talent.
Easy to Find.

START HIRING

EIPC Summer Conference 2026

In June, Marcy LaRont attended the EIPC Summer Conference in Lithuania and enjoyed an impressive lineup of business and technical presentations, along with a field trip to tour TLT's impressive new PCB factory. Under the new leadership of industry veteran Rico Schlueter, the EIPC conference was full of energy, emanating from both presenters and attendees. If you missed I-Connect007's EIPC Summer Conference coverage, click on the articles below to read more.



AI, Robots, and Historic Vilnius, Lithuania



BY MARCY LARONT, I-CONNECT007

Historic downtown Vilnius, Lithuania, hosted the sold-out EIPC Summer Conference, June 9–10, where 175 participants gathered at the AC Hotel, including representatives from the Global Electronics Association, FED, and four other industry associations. This year also included representation from 14 European PCB companies, and one—TTM— from the U.S., something that hasn't happened for quite some time.

Among those notable companies gracing the roster were Eurocircuits from Hungary and Germany, Schweitzer and Wurth from Germany, Thales from the Netherlands, Aspocomp from Finland, ACB from Belgium, and TTM, based in the United States. In all, there were 14 presentations and several panel sessions.

[To continue reading, click here.](#)



Automation as a Survival Strategy

INTERVIEW WITH AURELIJUS BELECKIS, ELINTA ROBOTICS



Aurelijus Beleckis

Aurelijus Beleckis, head of robotics at Lithuania-based Elinta Robotics, challenged manufacturers to rethink how they evaluate automation investments. Elinta has been growing its robotics business for 10 years and is the leading Lithuanian company in this market. In this interview, Aurelijus explains the value of automation, even in regions with high labor costs, and why delaying automation may be a far greater risk than the investment itself.

[To continue reading, click here.](#)

Where Is Direct Imaging Headed?

INTERVIEW WITH DENNIS PUSCH, SCHMOLL MASCHINEN



Dennis Pusch

Dennis Pusch, sales and business development manager for direct imaging at Schmoll Maschinen, examined the latest developments in UV light source technologies and the real-world trade-offs manufacturers must consider. If you're evaluating an imaging system upgrade or simply want to understand where the technology is headed, this interview is well worth your time.

[To continue reading, click here.](#)

Nomenil Aims to Make Additive Manufacturing Production-Ready

INTERVIEW WITH BAS LE GRAND AND DR. LUCA GAUTERO, NOMENIL



Dr. Luca Gautero

Bas Le Grand

Can additive manufacturing finally move beyond niche applications and into mainstream PCB production? Nomenil founders Bas Le Grand and Dr. Luca Gautero believe the answer lies not in the hardware, but in the software. Drawing on years of inkjet experience, they're developing tools designed to make additive manufacturing more predictable, scalable, and accessible for manufacturers.

[To continue reading, click here.](#)

STOP!

Do You Have a Digital Strategy for the Future?

BY HAPPY HOLDEN, I-CONNECT007

I am sure you have been wondering how you will survive this new age of AI, tariffs, China, and the ever-increasing cost of materials and PCB equipment. You cannot just sit back and watch it roll over you. If you wish for your company to survive, you need a “digitalization strategy,” and that includes a way to afford it.

After working in the industry for 50 years, I believe profits will not magically expand to a level that can be maintained for much longer. You need a strategy that will enable you to compete in this industry and expand your capabilities to attract new customers. Some of the challenges you face:

- Scarcity of suitable new employees with skills
- Maximum first-pass yields on ever-increasing complex multilayers
- Difficulty getting technical support for complex problems
- Providing short lead times and DFM knowledge to customers
- Increasing material, labor, utilities, and operating costs
- Fending off the growing presence of large Asian PCB fab competition

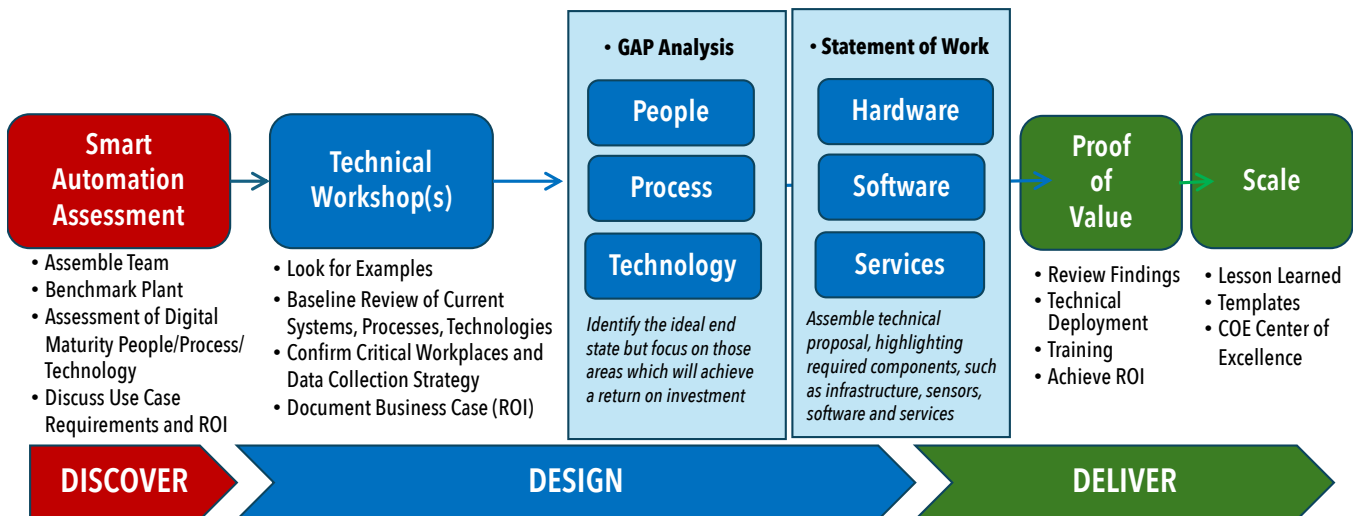


Figure 1: Automation Strategy: Think big, start small, prove value, scale fast.

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

Driven by integrated circuits and software, electronic devices are increasingly complex in density, size, and layer count.

How do you compete in this arena? You automate. But automation is complicated and expensive, so how do you handle this challenge?

The Nature of the Problem

One way is to find out how others are meeting this challenge. I read an interesting article¹ by Jim Zahora, CEO of The Abilities Connection (TAC), a Springfield, Ohio-based manufacturer and long-time supplier of cargo nets to the U.S. Air Force. His company needed to be more cost-competitive to secure future orders from the Air Force. But he faced these obstacles:

- TAC had diversified its workforce to include people with disabilities (to gain a larger population of workers), and training was increasing costs and slowing the implementation of newer technologies
- Attempts to reshore TAC's materials and equipment suppliers from Asia were not happening, and tariffs were not inducing the Asians to move manufacturing here

TAC tried a different tack: Invest in automation, diversification, and vertical integration. Today, 100% of TAC's product is manufactured in the U.S.: lead times are shorter, throughput has increased, and the workforce is performing more complex tasks with machine help.

The Working Solution

Your approach to automation doesn't always need to mean buying expensive equipment; you can work with the resources you already have.

You have experts on PCBs and their materials, processes, and tooling. In maintenance, you have experts on your equipment and how to keep it running. So, the first task is to identify individuals who want to learn about automation planning and can implement automation solutions, and then start planning the implementation of digitization and automation. If you want it done, you have to do it yourself.

“
Your approach to automation doesn't always need to mean buying expensive equipment; you can work with the resources you already have.”

For the rest of the year, my columns will discuss various techniques, devices, solutions, and equipment that you can fabricate (even locally) to solve your automation and other needs.

The Initial Steps

If you have decided that your company needs this strategy to survive, the first task is to create a team dedicated to accomplishing these goals. Though I believe recruiting a new college engineering graduate is an important part of developing your team, training a new graduate could cost up to \$20,000. You already have employees who have been trained and established, especially in CAM-tooling and maintenance. These are ideal candidates who can be upskilled for your task. They have the hands-on skills, combined with grit and curiosity, giving them an advantage over an engineer with credentials.

Once the team is established, look for partners, resources, and further education opportunities. Resources are machine shops and mechanical fabricators that can build the devices that we will talk about in the next few columns. Partner with community colleges or technical high schools that

train youth for our industries. This allows you to build a pipeline for talent. It may require some of your time volunteering in an after-school club, but it could evolve into a self-reinforcing partnership. Education should start with the basics of automation planning and sensor technology.

The Capstone Project for Interns

An alternative is to recruit engineering students for the summer, as many are seeking experience and a way to earn money during the school break. With interns, you can assign a simple project they already understand and have them execute it. One idea is a good manufacturing practices (GMP) project in which they document everything from the user requirements specification (URS) to the functional requirements specification (FRS) to the system design specification (SDS). They can handle PLC and SCADA code implementation, as well as commissioning and acceptance testing.

Along the way, the activities might increase the likelihood that, after graduation, they will consider returning to work for you. These activities include building a schedule, estimating durations, and comparing and reporting on estimates vs. reality. With this approach, the intern has received training and a healthy dose of context for future projects in about two weeks, having touched on documentation, coding, testing, and project management.

There are a lot of sharp, motivated people out there who can do this work, but I admit that finding them is hard.

Apprenticeship Over Time

Apprenticeships are also a popular activity, and one that can create the “automation engineer” you have been longing for. It also provides soft skills for approaching difficult issues, learning new processes, and understanding when it is appropriate to ask for help under the real constraints, timelines, and consequences inherent in automating machinery.

Conclusion

The Automation Strategy (of Digitization) will follow the process seen in Figure 1. After training the team, a thorough automation assessment will be performed to discover opportunities, followed by

the design and delivery phases.

This follows what Jim Zahora found useful. He concluded:

“For manufacturing leaders looking at something similar, let me offer a few final thoughts: Resources exist. We leaned on our Manufacturing Extension Partnership, which is part of NIST, for research and evaluation. That was especially helpful because we do not carry engineers on staff who specialize in every piece of automation equipment. Industry associations helped, too. You do not have to take the leap without a guide.

“Change does not happen overnight, and big changes in the defense supply chain sometimes do not move at commercial speed. But the payoff on the other side—a multi-year long-term contract, a more resilient supply chain, and production capacity we can flex with demand—is why we kept at it.

“One thing our operations team often says enough that it is worth repeating: ‘If you are still doing things the way you did them 20 years ago, you are in trouble and do not know it. Someone else will automate. Someone else will cut the lead time. Someone else will match the quality. Sooner than you think, it will be too late to catch up.’ I'd rather take the leap.”

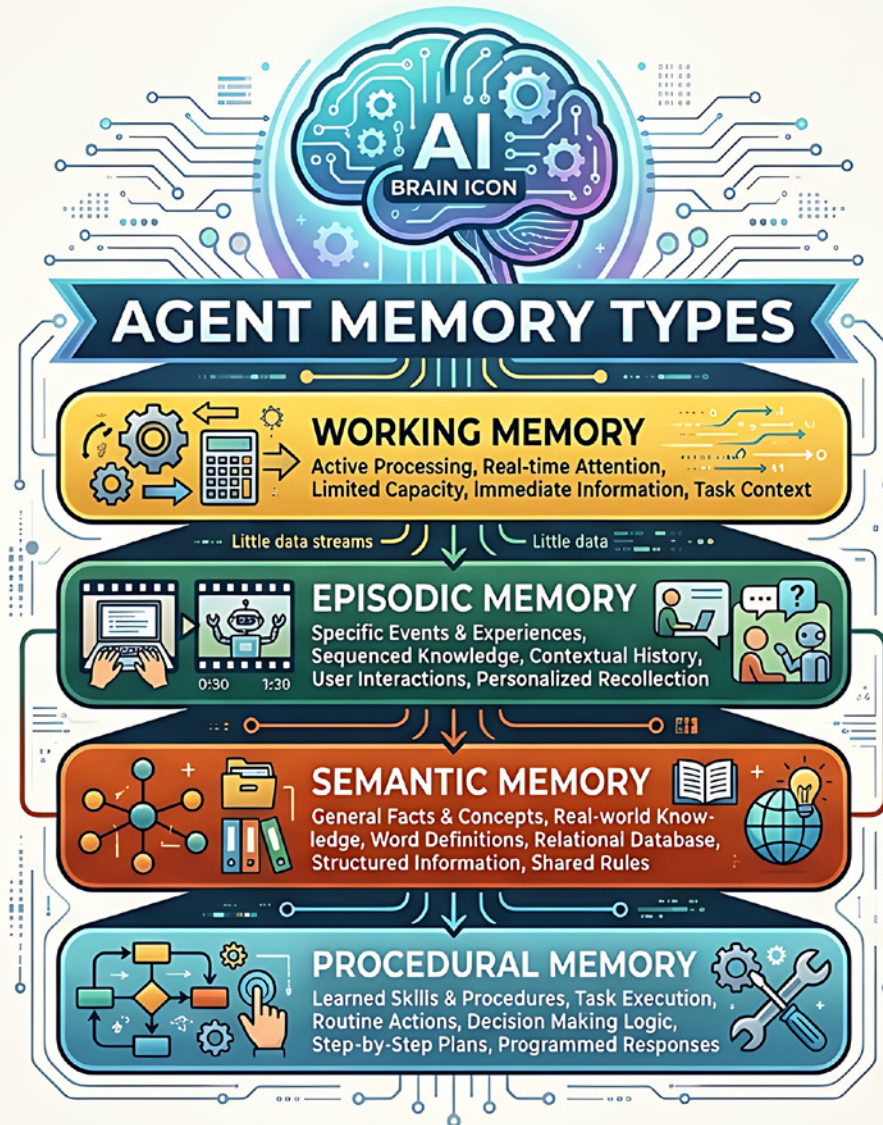
References

1. “We Need Automation to Reshore Our Supply Chains,” by Jim Zahora, Industry Week, June 1, 2026.



Happy Holden has worked in printed circuit technology since 1970 with Hewlett-Packard, NanYa Westwood, Merix, Foxconn, and Gentex. He is currently a contributing technical editor with

I-Connect007, and the author of Automation and Advanced Procedures in PCB Fabrication, and 24 Essential Skills for Engineers. To read past columns, [click here](#).



Your AI Doesn't Know Your Process

(Here's How to Teach It)

Last month, I made the case that you should not buy AI in the hope that it will fix the business. You need to learn it yourself first.

If your AI is failing, it's not because it is lazy or broken. It is failing because it does not know your plant. It does not know your quoting rules, how material gets released to the floor, what your customer expects in an 8D1, or which spec outranks

another. It certainly does not know where your team hides the real answer once the formal procedure and reality have drifted apart.

AI is like a smart new engineer on day one, bright and fast and generally informed, but with no idea of your customers, process names, capability edges, handoffs, approval rules, or evidence chain. If you want useful output, you have to teach it those things.

The Recognition Moment

You have probably seen this before. You ask AI to draft a customer response and it sounds polished, but overpromises. You ask for a material shortage note, and it misses the one fact the floor needs. You ask for an 8D, and it hands back a tidy report that sounds finished before the evidence is in.

You gave it a task but never taught it your process. In a PCB shop, that matters more than people think. Our work is context-heavy, document-heavy, and exception-heavy. A generic answer can sound good while being wrong for your plant.

What Context Actually Means

When people hear "add more context," they assume it means making the prompt longer. That is not the point. Context is the information that teaches AI how to work inside your world, and it helps to treat it as a small library you reuse rather than a one-time paragraph.

Here are the building blocks:

1. **Plant profile:** What kind of PCB shop are you: high-mix, quick-turn, aerospace-heavy, prototype-focused? What does the work generally look like?
2. **Workflow summary:** What process are we talking about? Who owns it? Where are the handoffs? What happens when things go wrong?
3. **Source hierarchy:** Which documents win if there is a conflict? Customer requirement, internal traveler, ERP record, or work instruction? Tell the AI what to trust first.
4. **Output format:** Do you want an email, a risk summary, a checklist, an escalation note, or a draft 8D section?
5. **Ask-before-answer instructions:** Tell the AI to ask clarifying questions one at a time before drafting. This is where it becomes a thought partner, not a search engine.
6. **Guardrails:** Tell it what it is not allowed to do: invent capability data, make disposition

decisions, claim evidence that is not in the record, or treat standards from memory as final authority.

Why One Giant Chat Starts to Get Sloppy

After a few good interactions, most people hit a snag: one long chat collects baggage. Your new request rides on top of old instructions, yesterday's problem, and a different tone, and the work drifts as stale history comes along. A long thread is like a conference table buried in notes from every meeting this week. Some still matter, some are outdated, and some belong to a different problem entirely.

Stay in one chat while you are working on the same problem, and the history helps. Start a clean one when the task changes, the department changes, or the AI keeps dragging in irrelevant assumptions. When in doubt, make a new chat window.

This is why a separate chat for each recurring job beats one catch-all thread. Give each job its own clean window. The sales chat receives the customer request and response rules; the material-issue chat receives the shortage facts and escalation format; and the 8D chat receives the complaint facts and evidence list. Start all three from the same base context—plant profile, terminology, source hierarchy, tone, capability boundaries—then add only what that job needs.

Use CRIT so the AI Learns Before It Writes

I introduced the CRIT framework in Article 1, and this is where it starts to earn its keep.

- **C = Context:** Teach the plant, workflow, constraints, and source rules
- **R = Role:** Tell the AI what perspective to take
- **I = Interview:** Make it ask questions before it answers
- **T = Task:** Define the exact output you need

The Interview step is the one most people skip, and it is what keeps AI from acting more confident than it should. If the tool does not know enough, make it ask.

Sales Process

Sales is a great place to build this habit, because the work is repeatable, text-heavy, and full of hidden context. A weak prompt like "Help me respond to this RFQ" produces something polished that may promise things your plant does not actually do. A better prompt teaches first.

CRIT Prompt: Sales RFQ Clarification

Context: We are a PCB shop focused on high-mix, quick-turn rigid multilayer work for industrial and defense customers. Our standard quoting process requires us to flag unusual stackups, tight tolerance items, nonstandard materials, and delivery expectations before committing. Customer requirements override our internal templates. If information is missing, do not assume.

Role: Act as an experienced PCB sales support specialist who understands plant capability boundaries and internal handoffs.

Interview: Ask me clarifying questions one at a time before drafting anything. Ask first about missing technical data, delivery expectations, special quality requirements, and anything that should be escalated to engineering or quality.

Task: After the interview, create: 1) a customer-facing clarification email, 2) an internal capability-risk summary, and 3) a short handoff note to engineering. Clearly separate known facts, open questions, and risks.

This works because it teaches the plant type, escalation behavior, and document priority, and it forces clarification before drafting. In sales, AI should help with the consistency and speed of your communication, not become quote authority.

Material Issue Moving to the Manufacturing Floor

Material issues are where individuals feel the value of using AI, because it sharpens communication when things get messy. A shortage or hold hits, and the floor, operations, and quality all need the same picture fast. The traveler says one thing while inventory says another. AI can help, once you teach it what the floor needs and what it can and can't decide, all while keeping a human in the loop.

CRIT Prompt: Material Hold/Shortage Communication

Context: We have a material issue affecting release to the manufacturing floor in a PCB plant. The floor, operations, and quality all need the same picture fast. That picture covers what happened, what is affected, what is contained, what is still unknown, and who decides next. AI is not allowed to decide release, substitution, or disposition, and unknowns must stay labeled as unknowns.

Role: Act as an operations support coordinator who writes clear, consistent internal plant communications during material issues, keeping every audience working from the same set of facts.

Interview: Ask me one question at a time about the material ID and lot traceability, the impacted jobs or travelers, the current hold and containment status, what is still unknown or unverified, and who owns the next decision.

Task: After the interview, draft three aligned draft communications built from one set of facts: 1) a short floor note, 2) an escalation note for operations and quality, and 3) a traceability question checklist of the open items the team still needs to confirm. Flag clearly what requires a human decision and who owns it.

With this structure, AI pulls scattered information into one clear picture of what happened, what is affected, what is contained, what is still unknown, and who decides next, so everyone reads from the same page instead of three different ones. What it must not do is decide whether material can run. That call belongs to your documented owners, approval rules, and evidence. AI organizes the communication. It does not make the disposition, you do.

Quality 8D for Customer Corrective Action

This is where healthy skepticism belongs. AI can help write an 8D, but AI cannot replace evidence. Its real value is structure, clarity, and questioning: separating fact from assumption, drafting containment language, flagging where the story jumps ahead of the evidence, and turning rough notes into a cleaner customer-facing draft.

CRIT Prompt: 8D Structure and Missing-Evidence Review

Context: We are preparing a customer corrective action using the 8D method for a PCB quality issue. I will provide the complaint summary, known facts, containment actions, investigation notes, and evidence still missing. Do not present hypotheses as facts. Do not claim root cause unless supported by evidence I provide. AI is supporting structure and questioning only.

Role: Act as an experienced quality engineer from Boeing, helping organize a disciplined 8D draft.

Interview: Ask me one question at a time about the defect description, affected product, containment taken, investigation status, and proof supporting each conclusion.

Task: After the interview, produce 1) a draft D1-D8 structure, 2) a list of statements that are factual vs. assumed, and 3) a list of missing-evidence questions the team should answer before sending the report to the customer.

It is not something to ship straight to a customer, but it is a strong draft to bring into a quality review meeting.

The Habit That Makes This Stick

Before you ask AI for help, stop for 60 seconds and answer four questions.

1. What does this tool need to know about my plant?
2. What does it need to know about this workflow?
3. What is it allowed to do, and what is it not allowed to do?
4. What should it ask me before it starts writing?

That is your pre-prompt checklist, and two weeks of it will change your output quality. The next level is to stop rewriting the same context from scratch and save your best blocks: one plant profile, one sales profile, one material-issue template, one 8D support template, one standard guardrail block.

Why This Matters Beyond Your Own Desk

Context compounds. It makes AI more useful at the individual level, more useful at the team level, more consistent, and, at the company level, an asset. The real advantage starts when your people can teach AI your plant, so sales drafts sound like your company, issue summaries follow your logic, and new employees ramp faster on context that is already reusable.

Context control is not a minor prompting trick. It is the beginning of organizational memory.

For Monday Morning

Pick one lane: sales RFQ clarification, material issue communication to the floor, or 8D draft support. Then do this:

1. Open a fresh chat.
2. Write a 150-300-word context block for that one workflow.
3. Add one guardrail block that says what AI may not do.
4. Use CRIT and require one-question-at-a-time clarification.
5. Save the prompt if the output is useful.
6. Reuse it this week and improve it once.

HOMEWORK

Quick win (5 minutes): Give AI context about your plant. Build a short teaching block for one lane (sales, material issue, or 8D) that names your role, what the plant makes, the audience, which documents matter, and what the AI should not assume. Then ask the same question twice, once without the block and once with it. The side-by-side shows why context is the difference between a generic answer and a useful draft.

Practice (15 minutes, pick the one for your lane): Run it with a CRIT context block and tighten the output until it reflects your real process language and handoffs.

Stretch (30+ minutes): Build your three-chat context library. Create one shared plant profile, start three clean chats from it (sales, material issues, 8D), and add only the lane-specific instructions, documents, audience, and output to each. Compare the results against doing all three in one long thread.

Teach the AI your process, make it ask questions, and verify the output like a professional. If Article 1 was about opening the gym door, Article 2 is about learning the first movement correctly. **I-CONNECT007**

References

1. The Eight Disciplines (8D) methodology is a structured, team-based approach to problem-solving that aims to identify the root causes of issues and implement effective corrective actions, Six Sigma.



Sean Patterson is the founder of CrossGen AI. He previously served as COO and CTO at Summit Interconnect, held senior multisite roles at TTM Technologies, and was CRO of Nano Dimension, a former Amazon executive and U.S. Navy Submarine Veteran.

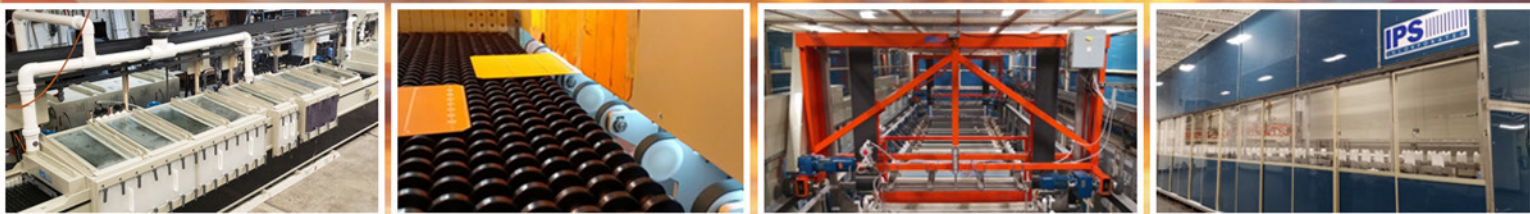
**More reach, greater impact.
More reasons to love
I-Connect007.**

Market where your customers are.

[Get Our Media Kit](#)

I-Connect007
The Global Electronics Resource

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



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



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

A Total Value Manufacturing Solution



BY MATT STEVENSON, ASC SUNSTONE CIRCUITS

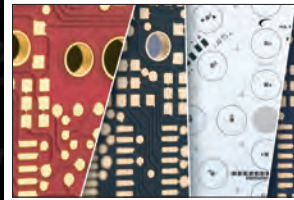
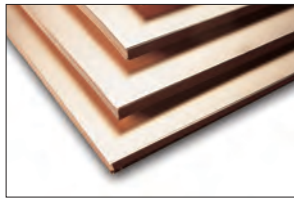
In my last column, I discussed total value management, an approach to the quality-versus-cost balancing act that integrates components of total quality management (TQM) and Lean manufacturing. Like those other methodologies, total value management prizes customer focus, continuous

improvement, and employee empowerment.

Total value manufacturing applies all these quality and improvement focused concepts to the PCB fabricator. A total value manufacturing solution partner is collaborative, innovative, nimble, and resourceful. They can help you formulate the best

Panasonic

Multi-layer & Flex
Laminates

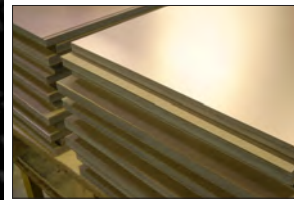


peters

Printed Circuit
Coatings



Electro-deposited
Copper Foils



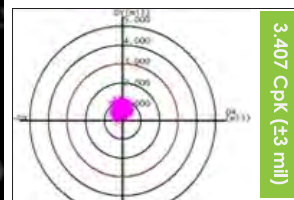
KB

KINGBOARD

Rigid &
Multi-layer CCL



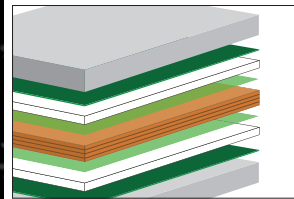
Drills & Routers



Drills Entry & Backup



PCB Press Plates
& Tooling



PCB Lamination
Assist Products



Dry Film Resist
Lamination



PCB Automation
Solutions

and more....



MATRIX
CIRCUIT BOARD MATERIALS

Supplier to the North American
Circuit Board Industry since 1977

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

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

path forward for your PCB prototyping and manufacture, be it domestic or offshore.

For customers facing increased decision-making complexity and evolving financial risk and reward models, it helps to focus on traditional value propositions:

- More effective, easier communication with the fabricator
- Reliable, dependable delivery
- Higher yield, better quality
- Smooth, faster production
- Reduced risk

Value at Each Stage of Production

Manufacturing value starts before the first blank board begins its journey through the production facility and continues well after the final products are shipped out. The first opportunity for manufacturers to deliver value is at the initial point of contact, when many customers are addressing the global question: Where to manufacture?

Providers of both domestic and overseas manufacturing solutions are well-positioned to objectively consult with customers to solve the

domestic vs. offshore equation, which is more complicated than simply calculating per-unit pricing. When you factor in yield, reliability, and durability, domestic PCB manufacturing is often the most cost-effective choice, even for higher-volume projects. You must identify variables and run all the numbers to find out.

With larger production volumes, even modest per-unit savings often associated with offshore production can appear attractive on the surface. However, those savings paint only half the picture. Our experience tells us that less easily quantifiable cost drivers pose greater obstacles to offshoring than just volatile import taxes and port delays. It is the value equation that tells the story.

Is the risk associated with a globally sprawling supply chain offset by the value of slightly cheaper boards? Can you cost-effectively warehouse a sufficient supply to be resilient? Can delays and cost variances from shipping, tariffs, and transport fees be absorbed? How much will lesser yield or issues with quality impact the cost savings?

These are the real, indirect costs associated with offshoring that may minimize or even eliminate potential savings. It might still be the right decision to go overseas, but a value-based partner can work with you to clarify which path offers the most value.

Best Practices for Total Value Manufacturing

Total value delivered to the customer through support, on-time delivery, quality boards, and high yield is how manufacturers can meet customer needs for PCB production in all circumstances. Its foundation is quality.

Schedule Adherence

It does not matter how good the boards are if they are not available for production of the electronic device they were built for. If a manufacturer ships an average of 1,000 orders per day, the difference between 95% on time delivery and 97% annualizes to over 5,000 additional late shipments per year. Every late delivery has a cost. Because it affects each customer's bottom line, manufacturers that are focused on total value work every day to improve their on-time delivery rate.

The key question facing every designer: Can the manufacturer build the board I designed?"

The road to American manufacturing is paved with PCBs.

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

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

JOIN US TODAY

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



PCBAA.ORG



Design Adherence

The key question facing every designer: Can the manufacturer build the board I designed?

Designs with manufacturability issues cause production delays and can drive up costs. Design adherence helps the design team concentrate on the behavior of the design during prototyping and debugging, because the PCB manufacturer built what the design team specified.

The standard or widely accepted RMA rate for PCB manufacturers is roughly 2%. Fabricators that strive to provide total value manufacturing maintain a 1% RMA rate.

Responsiveness

PCB manufacturing is a collaborative process. A good partner will help the design team keep revision cycles to a minimum during prototyping and move the project along as fast as possible. Responsiveness is key to achieving this goal. Design teams are often working on tight project deadlines. They need prompt, reliable access to their fabricator to get clarity.

A value-focused manufacturer will provide ready, constant access to experts who speak the language of design and product, and they will do so with customer service phone call wait time measured in seconds, not minutes.

Yield

Production yield levels are a key leading indicator for ongoing service levels during the product's life-cycle. Sound production processes, coupled with deep manufacturing expertise, a focus on quality, and a commitment to continuous improvement, produce higher yields.

Partnering With a Total Solution Provider

When a PCB manufacturing partner has deep-rooted cultural values of quality and collaboration with customers, the result is accurate, efficient, and speedy order fulfillment, no matter the project, factory, or scale. Customers benefit from reduced decision-making complexity in pre-production, greater transparency across each phase of manufacturing, and higher-quality boards. **I-CONNECT007**



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

Educational Resource Center

Podcasts 

Books 

Roundtables 

White Papers 

Webinars 

Subscribe 

I-Connect007

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's Fifth Annual Meeting

Growing Membership and Valuable Relationships

BY SHANE WHITESIDE, PCBAA

The Printed Circuit Board Association of America recently celebrated its fifth anniversary. We began with five member companies that banded together to influence legislation favorable to our industry. Today, we have 90 members across the entire ecosystem: OEMs, fabricators, assemblers, materials providers, testing companies, and applied AI startups. Our member-

ship ranges from companies with long histories to ventures just embarking on their PCB journeys.

Our growing association has raised the profile of the PCB industry where our members live and work, as well as in Washington, D.C. Members have hosted legislators in their home districts, and PCBAA representatives have frequent direct engagements on Capitol Hill. Our annual meeting last month included





ventec
INTERNATIONAL GROUP

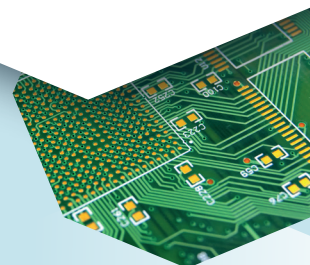
As material demand rises, **optionality** becomes essential.

Demand for advanced PCB materials is growing, making supply security and material flexibility more important than ever.

In today's constrained market, securing reliable supply is a constant challenge. That's why material equivalence matters. Access to qualified alternatives helps maintain production continuity, reduce risk, and strengthen supply chain resilience.

With a broad portfolio of qualified material alternatives, Ventec provides greater sourcing flexibility, helping customers secure supply and maintain more stable lead times.

Contact Ventec today to discuss your material requirements.



Ventec Equivalent Materials for Supply Chain Continuity

Wherever **technology** takes you, Ventec delivers

FR4 High Tg, HF		Low Loss					
	Halogen-Free VT-447V(I)	tec-speed 2.0 VT-464	tec-speed 3.0 VT-464L	tec-speed 4.0 H-PK VT-464LT	tec-speed 6.0 VT-462S	tec-speed 6.0 / 6.1 H-PK VT-770 / VT-770(LK)	tec-speed 8.1 / 8.2 H-PK VT-772(LK2)
	For Automotive HDI Applications	Middle Loss	Middle Loss	Low Loss	Very Low Loss	Ultra Low Loss	Extreme Low Loss Super Ultra Low Loss
Dk Df	Dk: 4.40 Df: 0.012	Dk: 3.8 Df: 0.0095	Dk: 3.7 Df: 0.008	Dk: 3.9 Df: 0.0068	Dk: 3.5 Df: 0.005	Dk: 3.3 ~ 3.4 Df: 0.0024	Dk: 2.90 Df: 0.0013
SYL	Autolad 3G	S7045G S7045GH S7040G	S7038	S7439 S7439C S7439G S7439HW	Synamic 6 Synamic 6X Synamic 6G Synamic 6N	Synamic 8 Synamic 8G Synamic 8GN	Synamic 8NE Synamic 9GN Synamic 9N2
ITEQ	IT-858G	IT-170GT IT-170GRA1	IT-180A	IT-958G IT-170GRA2	IT-968 IT-968G IT-968GSE	IT-988GSE	IT-988G
TUC	TU-865	TU-865 TU-862HF TU-872LK TU-862S	TU-863 TU-872LK TU-872	TU-872SLK TU-863+	TU-883 TU-883SP	TU-933 TU-933E TU-933+	TU-943R
NY	NPG-180BH	NY3170M NY3170HF	NY3170M NY3170HF	NY6200 NY3170M2 NY6180L	NY6300 NY6300S	NY-P1 NY-P2 NY-P3	NY-P4
Panasonic	-	-	R-1755V R-5725	R-5725 R-5725S	R-5775 R-5775N	R-5785 R-5785(N/GE/GN)	R-5795(N) R-5795(E) R-5795(U)
EMC	EM-370(Z) EM-370(Z)-15	EM-370(D) EM-828G	EM-888	EM-526 EM-888S EM-888K	EM-891 EM-528 EM-528K	EM-890 EM-890K	EM-892K
Doosan	-	-	DS-7409	DS-7409D(X)	DS-7409DV	DS-7409DVN	DS-8502SQN
Isola	IS 550H	I-Speed FR-408HR	I-Speed IS 580G	I-Speed I-Tera MT40	iTera MT40	Astra MT-77 Tachyon 100G	TerraGreen-400G2

a day on Capitol Hill, giving executives a chance to meet with members of Congress and staff to tell their personal stories and to ask for support for our two PCB bills: HR3597 and S4569.

This face-to-face contact is important. Many legislators and staff still don't understand where PCBs fit in the technology stack. Our Hill meetings raised awareness of the need for trusted and secure PCBs to support national security and critical infrastructure. These meetings also generated additional sponsors for our bills.

“
Our annual meeting grows every year because relationships matter, and PCBAA brings people together to make a real impact in Washington.”

We also heard from other branches of the federal government. We were encouraged by remarks from the PCB Executive Agent Office outlining what is being done to improve requirements and processes, and how they can work with us to achieve the most productive outcomes. We will engage more fully in the second half of the year as Defense budget decisions are made.

A senior Department of Commerce official gave remarks that clearly showed his depth of knowledge and understanding of both the issues facing our industry and what is being done to help us reshore and restore manufacturing in the U.S. We're proud of the strong relationship we have developed with the Department of Commerce, and we are excited about the current and emerging financial programs available for companies needing help to scale up

operations, buy expensive specialty equipment, and invest in other capital expenses.

On the customer side, we hosted an OEM supply chain panel comprised of representatives from SpaceX, Lockheed Martin, and Raytheon. The biggest companies in the world depend on trusted and secure PCBs and were unanimous that communication is important up and down the supply chain. Hearing this from our major customers reinforced the need to reshore and restore domestic PCB capacity.

Compelling storytelling has been a hallmark of PCBAA's first five years. We were proud to share a recent 15-minute CNBC story,¹ which was orchestrated by our team and provided a clear explanation of our industry and the challenges we face. PCBAA and the Alliance for American Manufacturing jointly produced a 20-minute documentary film, “The New Frontier,”² which was also shown. After so many years of being invisible to the mainstream media, it's gratifying to have this level of national exposure.

Our annual meeting grows every year because relationships matter, and PCBAA brings people together to make a real impact in Washington. If you are ready to be part of our “Educate, Advocate, and Legislate” mission and join us in Washington next year, please contact me today. **I-CONNECT007**

References

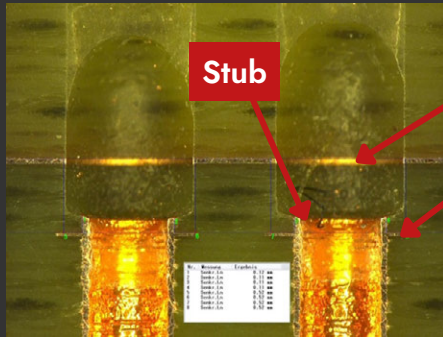
1. “Hidden beneath AI chips, Chinese-made circuit boards raise national security concerns in U.S.,” by Katie Tarasov, CNBC, June 3, 2026.
2. “The New Frontier: Inside the Comeback Factories Securing America's Tech Future,” a documentary produced by the Alliance for American Manufacturing and Printed Circuit Board Association of America, youtube.com.



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

EAGLE EXY

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

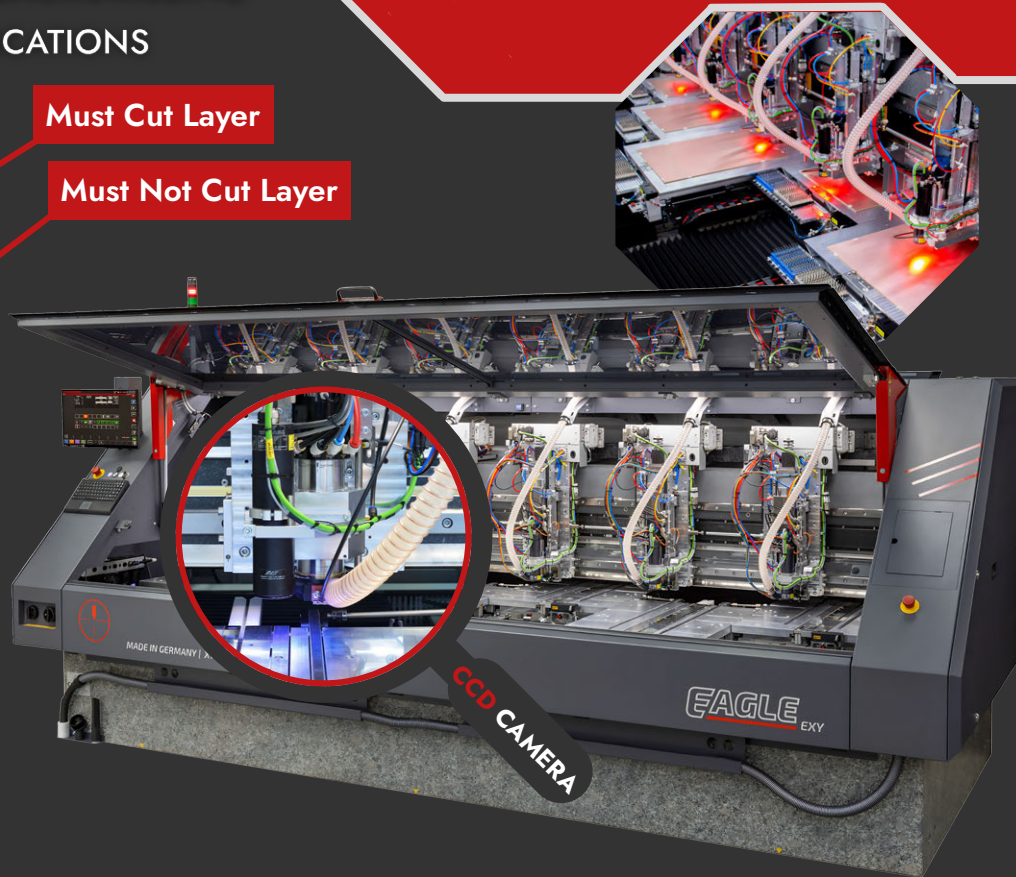


Back Drilling: stop of depth drilled hole before must-not cut layer within stub length tolerance

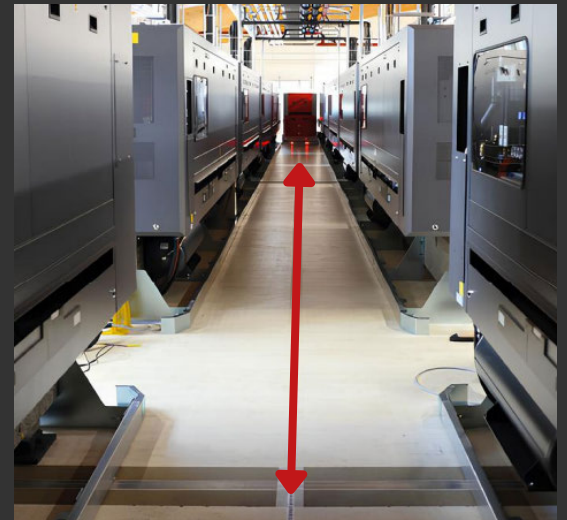


Depth Drilling: precise and reliable drilling to defined inner layers, enabling high-quality blind vias.

AUTOMATED COLLECT CLEANING (ACC)



**AUTO LINE SHUTTLE (ALS)
AUTOMATION**



THE PROS AND CONS OF PCB Vertical Integration

BY STEVE WILLIAMS, THE RIGHT APPROACH CONSULTING

Vertical integration in PCB fabrication is a genuinely contested strategic question with real tradeoffs on both sides. The core tension: Fabricators integrate up into design and down into assembly to gain margin and stickiness, but risk diluting their manufacturing focus and alienating the EMS and design-house partners they depend on.

A Master's Thesis to Reality

I earned my MBA 25 years ago while working at Plexus, the \$4 billion global contract manufacturer, and I decided to do my thesis on "Should Plexus Acquire a PCB Manufacturer as a Captive Fabricator?" Managing a global PCB supply base of around \$150 million in spend (which would more than double in the following decade), I had been toying with this idea of vertical integration for quite a while.

Now, why would I risk investigating something that could potentially eliminate the need for my position? I was early in my 15-year tenure with Plexus, and if the conclusion was yes, my next step after presenting the results to the leadership team was to leverage my 22 years of leadership in PCB manufacturing to convince them that I should run the new division.

But after eight months of research, quantitative analysis, case studies, and endless "what ifs," the data-based conclusion was a resounding "no"; it did not make financial or strategic sense to vertically integrate with a board shop. However, there was a bit of poetic irony that the Plexus business model had always combined electronic assembly with design capabilities, so upward integration made sense while downward integration did not.

Back in the heyday of PCB fabrication, when the U.S. was the global leader and had over 2,000 board shops, almost every major OEM had its own captive PCB manufacturing operations. When OEMs

began shifting production overseas to lower-cost countries, they shuttered their captive shops and purchased on the open market. My thesis research had proved this was the right decision.

Twenty-five years later, I stand behind my conclusions for EMS/PCB integration. With the ongoing quantum changes in technology, equipment, and processes, no single shop, or even group of shops, can handle all the technology needs of today's market. I still believe it is most cost-effective to source PCBs in a competitive market and utilize a stable of shops that are experts in a select portfolio of technology. While those are my personal biases, let's explore some unbiased, objective, specific pros and cons of vertical integration.

Pros

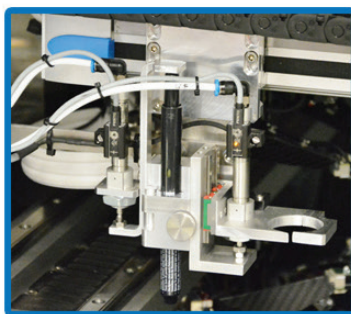
For the PCB fabricator:

- **Margin capture.** Design services and assembly both carry higher margins than bare-board fabrication, which is a commodity business (to the customer) under constant price pressure from Asian competitors.
- **Design-for-manufacturability lock-in.** When the same company designs and builds, DFM feedback is immediate and iterative rather than arriving late via back-and-forth with an external EDA firm. Yield goes up, and respins go down.
- **Customer stickiness.** Once a customer's IP, libraries, and process knowledge live inside the fab's ecosystem, switching costs are high. The relationship becomes hard to unwind.
- **Data continuity.** Fabricators that also assemble have complete visibility from Gerber

Unrivalled Test Speed with Full Automation



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



Highlights:

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

Watch video

Get more info

atg Luther & Maelzer GmbH

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



files to finished board to test results, enabling closed-loop process optimization that siloed companies can't replicate.

- **Faster cycle times.** No handoffs between design house, fab, and assembler mean prototype-to-production timelines compress significantly. This is critical in aerospace, medical, and defense markets where speed matters more than unit cost.
- **Single-source accountability.** Customers with high-reliability requirements (medical, mili-

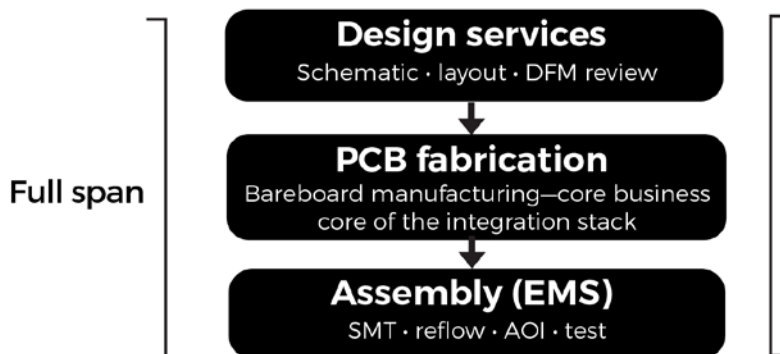
tary, automotive) often prefer “one throat to choke.” A vertically integrated fabricator can offer that.

For the customer:

- Simpler vendor management.
- Tighter feedback loops between design intent and manufactured reality.
- Potentially lower total cost (fewer margin layers in the supply chain).

VERTICAL INTEGRATION IN PCB FABRICATION

Should fabricators absorb design and assembly?



✓ PROS



✗ CONS





pluritec® GROUP

Advanced Technology for Printed Circuits Boards



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



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



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

X-CUT

FROM LAMINATION TO
DRILLING IN ONE STEP



D.E.S.

"SMART" ETCHING CONTROL
AND ULTRATHIN 1 MIL
MATERIAL HANDLING

SPRAY COATING OVEN

COAT AND TACK CURE READY FOR
IMAGING IN 20 MINUTES



Cons

For the PCB fabricator:

- **Channel conflict.** The moment a fabricator announces design services, every EMS company and design house that sends its boards starts evaluating alternatives. The existing partner network, which may represent the majority of revenue, sees the fabricator as a competitor.
- **Cultural mismatch.** PCB operations are process-driven, high-discipline, and continuous-improvement cultures. Design engineering is creative, iterative, and tolerates ambiguity. Running both under one roof is organizationally difficult.
- **Capital and talent requirements.** Adding real design capability means hiring senior PCB layout engineers, signal integrity specialists, and application engineers, expensive talent in a thin-margin business. Assembly adds SMT lines, reflow ovens, AOI, and X-ray equipment.
- **Focus dilution.** The fabricators that dominate on quality do so because they are obsessively focused on the highly complex manufacturing processes. Adding adjacent businesses introduces distraction at the operational level.
- **Technology mismatch risk.** A fabricator optimized for rigid multilayer production that integrates assembly will be tempted to bias its design recommendations toward what it already makes well, potentially steering customers away from rigid-flex or HDI solutions that would serve them better but that the fabricator is less equipped to build.

For customers:

- **IP concentration risk:** All design files, schematics, and trade secrets sit with a single vendor.
- **Reduced competitive tension on pricing;** no easy way to rebid to an alternative fabricator without re-engaging a design and EMS provider.

- **Potential for conflict of interest in design recommendations** (the fabricator recommends what it can build most profitably, not necessarily the optimal solution).

Where Does It Work Well?

Vertical integration succeeds when it targets a specific, well-defined niche: military/defense fabricators moving into ITAR-compliant assembly, medical device fabs offering turnkey prototype services, or flex PCB specialists adding design services specifically for flex and rigid-flex, where the design-process interaction is tight enough that integration genuinely adds value rather than just capturing margin.

It struggles when attempted as a broad strategy against entrenched EMS players that have an assembly scale no fabricator can match.

The historical pattern suggests that partial integration, offering DFM consulting and design review without fully displacing the customer's design house, preserves partner relationships while still capturing some of the margin and stickiness benefits. Full vertical integration is a strategic bet that changes who your customer is, which not every fabricator has the organizational capacity to execute.

Conclusion

So, the answer to whether it makes sense for PCB fabricators to vertically integrate is an unconditional, definitive "it depends." It depends on the company, the market sector, the competition, and customers involved, and whether they would see vertical integration as a competitive advantage or a threat. Perhaps a hybrid and/or partnership approach provides the optimal balance of the pros and cons.



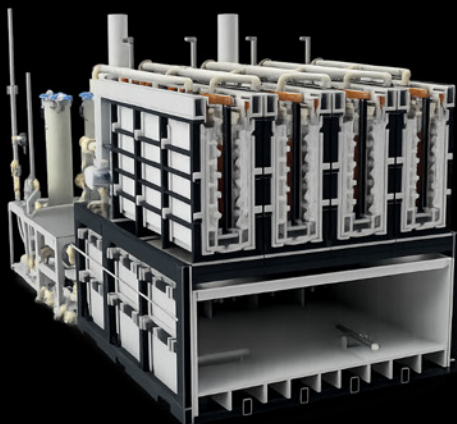
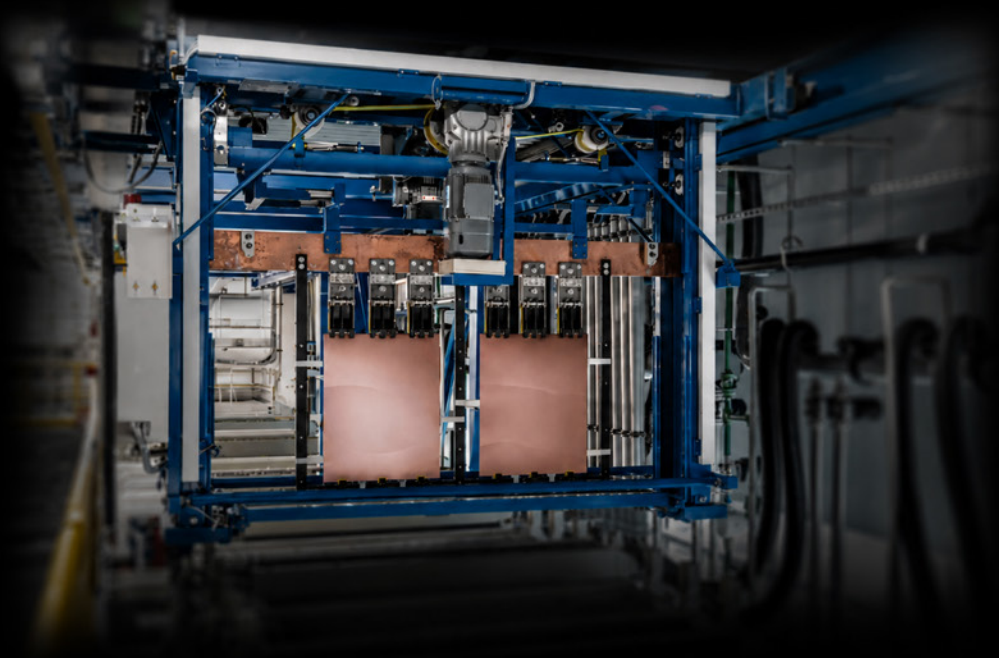
Steve Williams is president of The Right Approach Consulting. He is also an independent certified coach, trainer, and speaker with the John Maxwell team. To read past columns, [click here](#).

RESOLUTION™ V

Electrolytic Plating Innovation



GREENSOURCE
ENGINEERING



High CD application capability:

- Up to 6 ASD for DC and complex RPP
- Every single cell can be controlled independently

MMO coated DSA Inert anodes with CuO replenishment:

- Anode to Cathode (panel) distance = 70mm

Separation of plating cell and sump tank propagates:

- Excellent solution exchange - no temp hot spots
- Dosing distanced from active area - no conc hot spots



GREENSOURCE
ENGINEERING

Airbus Awarded Aeolus-2 Wind Sensing Satellite Contract ►

Airbus Defence and Space has signed the initial contract for the design and build of the Aeolus-2 wind sensing satellite from the European Space Agency (ESA) at a ceremony at ESA's UK headquarters ECSAT at Harwell. Aeolus-2 will follow the first Aeolus wind sensing satellite built by Airbus and launched in 2018.

Mission Space Selected by NASA to Advance Lunar Environmental Monitoring Technology ►

Mission Space has been selected by NASA under the Announcement of Collaboration Opportunity (ACO) to advance LEEMR, a compact lunar dust and surface-charging monitoring instrument developed for future lunar surface operations, in collaboration with NASA's Goddard Space Flight Center in Greenbelt, Maryland. The project supports Mission Space's expansion from orbital space-weather intelligence into lunar surface environmental monitoring.

L3Harris to Provide Communication Systems to Support German Forces ►

L3Harris Technologies has received two Foreign Military Sales orders for Falcon® systems to support Germany's Digitalization –Land Based Operations (D-LBO) and special operations forces requirements. "The battle-tested Falcon systems will enhance German forces' secure, interoperable communications with European Union and NATO allies," said Chris Aebli, president, Mission Critical Communications, Communications & Spectrum Dominance, L3Harris.

Lockheed Martin Awarded \$3 Billion to Produce GMLRS Rockets Supporting U.S. Army Readiness ►

Lockheed Martin has been awarded the FY26 contract for GMLRS rocket production, supporting the U.S. Army's continued modernization of long-range precision fires and its efforts to expand and sustain munitions readiness. This \$3 billion award enables continued production and delivery of standard and extended-range GMLRS rounds.

Boeing Selected by U.S. Space Force to Extend Secure Military Communications Worldwide ►

The U.S. Space Force has selected Boeing to advance the next phase of a program that will help extend and improve secure military communications for users around the world. The contract award for Mobile User Objective System Service Life Extension (MUOS SLE), valued at up to \$2 billion, includes two narrowband communications satellites that prioritize reliable voice and essential data, helping users stay connected. The first satellite is scheduled to be delivered in 2031.

Lockheed Martin, Insta Advance Europe's First MLRS CFCS Sustainment Centre ►

Lockheed Martin and Finnish technology company Insta are continuing to establish Europe's first sustainment centre for the M270A2 and HIMARS® MLRS CFCS at Insta's facility in Tampere, Finland. This will be Europe's first MLRS fire control system maintenance and sustainment centre. The M270A2, manufactured by Lockheed Martin, is a tracked heavy rocket launcher system.

NASA Selects Rocket Lab to Launch Sun, Earth Sciences Missions ►

Rocket Lab Corporation, a global leader in launch services and space systems, announced it has been selected by NASA to provide three Electron launches for two separate NASA missions—PoSIR and TSIS-2—from early next year. Rocket Lab's extensive flight heritage across more than 90 launches, Electron's proven deployment accuracy, and the company's ability to meet the tight turnaround schedules for both missions were key driving factors behind Rocket Lab being selected as the launch provider.

As SpaceX Goes Public, the Market Hunts for the Next Investable Launch Company ►

For two decades, the most important company in the modern space economy was one that public investors could not buy. That changed in 2026, when Space Exploration Technologies Corp. (SpaceX) moved to list on the Nasdaq under the proposed ticker SPCX, in what has been reported as one of the largest initial public offerings in history.

Technology, and People, You Can Trust.

Specialties:

- RF/ Microwave
- UHDI
- Rigid-flex
- High Reliability
- Aerospace/Medical

...and Specialists:

- DFM / DFX Support
- Collaboration
- Project Guidance



asc-i.com



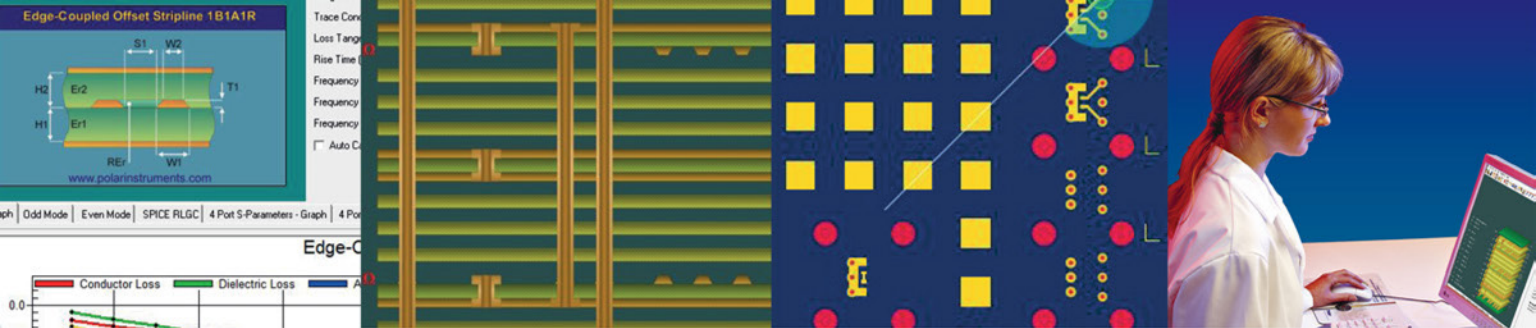
Three Lessons Every Designer Should Know About *LED Displays*

A display is one of the few electronic subsystems where PCB design decisions become visible to the end user. A few milliohms of resistance, a slight impedance discontinuity, or a few degrees of temperature difference may never matter elsewhere in the product. In an LED display, however, those same design decisions can become a visible defect. Behold the power of the human eye.

Over the years, I've reviewed hundreds of PCB layouts spanning everything from industrial controls and aerospace electronics to medical devices and high-speed computing systems. Every design pres-

ents its own unique challenges, but display electronics have always stood apart. They demand a different way of thinking because success is measured by what the customer sees, not solely by electrical performance.

I've often said that experienced PCB designers struggle with LED displays, not because they lack technical knowledge, but because they apply the same design mindset that has served them well elsewhere. Display electronics require a subtle but important shift in perspective. You're no longer designing only for electrical performance; you are



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

PCB Signal integrity tools for design & fabrication

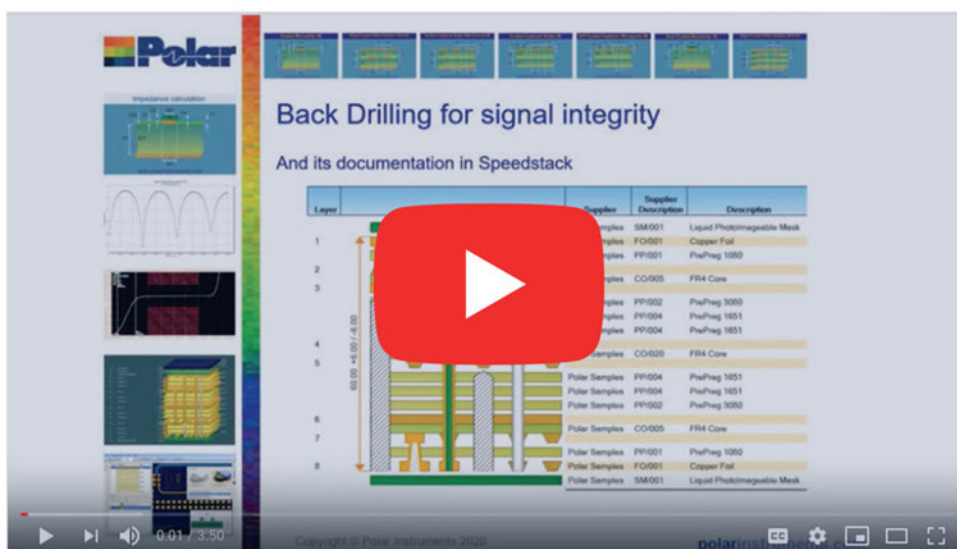
- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of SI topics

SUBSCRIBE



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

DOWNLOAD BOOK



polarinstruments.com

designing for human perception.

Here are three lessons I believe all new and experienced PCB designers should keep in mind when designing an LED display for the first time.

1 Every Electrical Decision Becomes a Visual Decision

One of the first things I look at during a display board design review isn't the LED array itself. I look at the power distribution network. I know that's surprising, because most engineers naturally focus on signal routing, LED drivers, or interface circuitry. Those are certainly important, but I've learned that many display problems begin with how power is delivered across the board.

Unlike many electronic systems, where voltage variations remain invisible to the user, LED displays convert electrical inconsistencies directly into optical inconsistencies. A slight voltage drop across a copper plane may seem insignificant during simulation, yet it can create subtle brightness differences that the human eye detects immediately. The irony is that the design may satisfy every electrical requirement.

I've seen layouts that passed design reviews, simulations, and bench testing without concern, only to reveal slight brightness gradients once the display was powered on. Nothing was technically "wrong," and the display functioned exactly as designed. Unfortunately, the customer was evaluating visual quality rather than electrical measurements.

The human eye is remarkably sensitive to inconsistency. We instinctively notice when one section of a display appears slightly brighter, a color subtly shifts, or one region responds differently than another. Those differences may represent only a tiny electrical variation, but perception doesn't measure millivolts; it measures uniformity. That's why power integrity becomes far more than simply supplying adequate current.

Current balancing, low-impedance power distribution, copper geometry, connector placement, decoupling strategies, and return path continuity all influence what the customer ultimately sees. Every milliohm matters because every milliohm has the potential to become visible. When reviewing display layouts, I always encourage designers to stop asking, "Will this circuit work?" and instead ask,

"Will the customer notice the difference?" Those are two very different questions.

2 Design for Thermal Uniformity, Not Just Thermal Performance

If power distribution determines how a display performs today, thermal management often determines how it performs tomorrow. Most of us understand that excessive temperature reduces reliability. That's hardly new. What many first-time display designers don't fully appreciate is that thermal variation often becomes visible even before components begin to fail.

LEDs are current-driven devices where optical characteristics change with junction temperature. As temperatures increase, you'll see brightness decrease, color characteristics shift, efficiency decline, and long-term degradation accelerate. When neighboring LEDs operate at different temperatures, the display loses the uniform appearance customers expect.

I've participated in design reviews where thermal analysis concluded everything was comfortably within specification, yet the display still exhibited noticeable visual inconsistency because thermal performance and thermal uniformity are not the same thing.

A display cares about temperature consistency across the entire viewing surface, making PCB design an integral part of the thermal solution rather than simply a platform for mounting components. In these designs, copper planes become heat spreaders, thermal vias become thermal highways, and component placement influences airflow. Board stackups also affect heat distribution. Material selection, copper weight, substrate construction, and even connector locations all contribute to maintaining thermal balance across the display.

Outdoor displays make this challenge even more demanding. In a lab, you don't have to account for solar loading, elevated ambient temperatures, or continuous high-brightness operation. Designs that appear robust under controlled conditions may behave very differently after years of field operation.

One lesson I reinforce repeatedly: Don't design thermal management to keep LEDs alive. Design thermal management to keep every LED looking the same. Those are not always the same objective.

3 You're Designing a Manufactured Optical System

Perhaps the biggest misconception I encounter is the belief that designing an LED display is primarily a challenge in PCB layout, when it's actually a manufacturing challenge. Successful display products happen through a combination of electrical engineering, mechanical design, thermal management, manufacturing processes, assembly quality, calibration, and long-term reliability. The PCB simply serves as the foundation that brings all those disciplines together.

I've reviewed technically impressive layouts that were unnecessarily difficult to manufacture, inspect, calibrate, or repair. In production, those designs often proved to be expensive lessons. Displays have very little tolerance for inconsistency.

Things that influence the finished product include slight assembly variations, connector alignment, solder joint quality, component placement accuracy, LED binning, and mechanical tolerances. Individually, each variation may appear insignificant. Collectively, they determine whether customers perceive a display as premium or problematic.

That's why DFM isn't something you verify at the end of the project. It becomes part of the architecture from the very beginning. I encourage designers to ask questions that extend well beyond PCB layout and determine the long-term success of the product:

- How will this board be assembled and inspected?
- How will defective modules be replaced?
- How will technicians access connectors during service?
- What happens after thousands of thermal cycles?
- Can manufacturing consistently build this design tens of thousands of times?

One observation I've made over the years is that the best designers for LED displays spend as much time thinking about manufacturing as they do routing. It shows engineering maturity.

Engineering for Perception

Display technology is advancing toward higher pixel densities, smaller pitches, brighter outputs, and thinner form factors. As such, the engineering challenges will only become more demanding. Designers will have to deal with higher-speed interfaces, greater power densities, tighter mechanical integration, and complex manufacturing processes. These challenges raise the bar for everyone involved in product development.

Fortunately, the fundamentals haven't changed. Successful display designs still begin by understanding that electrical, thermal, mechanical, and manufacturing decisions cannot be made independently. Every discipline influences the final image the customer sees.

Conclusion

Over the years, I've come to believe that LED display design is really an exercise in engineering for perception, and display designers must also design for visual performance. The oscilloscope, thermal camera, and simulation tools remain essential, but none of them represents the final judge. The human eye does.

That reality changes how we distribute power and manage heat, as well as how we think about manufacturability, consistency, and quality. Most importantly, it changes how we define success.

The next time someone tells me they're designing an LED display, I probably won't ask about the processor, the interface, or even the LEDs. I'll ask them how they intend to design for the human eye. Because after more than three decades of reviewing PCB designs, I've learned that the most successful display boards aren't necessarily electrically perfect. They're the ones where engineering disappears, leaving the customer to notice only the image, never the technology behind it.



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

Flex and Rigid-flex Solutions in A&D Display Electronics

BY KELLY DACK, CIT CID+



Displays and illuminated human-machine interfaces have become essential components of modern electronic systems. From aircraft cockpits and mission-control displays to medical equipment, industrial controls, and consumer devices, we depend on these visual interfaces to interact with technology.

As display systems become thinner, lighter, brighter, and more integrated, we need flexible and rigid-flex circuit technologies as critical enablers. I recently spoke with Pioneer Circuits' Applications Engineer Dan Hansler, and Sales and Marketing Director Ruben Lugo about how decades of aerospace and defense (A&D) experience are influencing today's lighting and display designs.

Kelly Dack: Dan and Ruben, why are flexible and rigid-flex circuits so important in high-tech lighting and display applications?

Dan Hansler: Modern displays are much more

than screens. Behind every display is a network of supporting electronics that may include LED back-lighting, display drivers, touch sensors, processors, communications interfaces, and power management circuitry. In many applications, these elements cannot occupy the same physical space.

Flexible and rigid-flex circuits provide an efficient way to connect these sub-systems while reducing weight, minimizing assembly complexity, and eliminating many of the connectors and cable assemblies that can become reliability concerns over time.

This is especially important in A&D systems, where displays are often integrated into highly constrained mechanical environments such as cockpit instrument panels, heads-up displays, mission computers, control consoles, and operator interfaces. The same technologies are now being applied to automotive displays, medical equipment, industrial controls, and next-generation consumer electronics.

Precision PCB Advancing with Precision, Built on Expertise

Electronics Manufacturing
experts focused on scalability,
efficiency, and speed.

QUICK-TURN PCB AND ASSEMBLY

26+ Years of Excellence in PCBs

- ✓ **PCB Design Support**
- ✓ **PCB Manufacturing**
- ✓ **PCB Assembly**



Get a quick quote by email today at
sales@precision4pcb.com

Strategic Solutions

- Avoid Communication Gaps
- Prevent Engineering Delays
- Single Point of Contact - Sales Through Delivery
- Streamlined Process - FAB to Assembly
- Consistent Quality Results
- Advanced Technology PCBs
- Quick-Turn Prototyping
- High Mix-Low Volume Production
- Fine Pitch SMT - Thru-hole Technology
- BGA/LGA 3D X-Ray/ AOI
- Turnkey and Consignment
- Box Build Services
- Full Functional Testing

Get Started Now

888-228-9440

www.precision4pcb.com



Precision Technologies, Inc. dba: Precision PCB | Made in USA

ITAR | SAM registered | UL approved | ISO 9001:2015 Registered | IPC-A-610 / J-STD Certified



Dan Hansler



Ruben Lugo

Ruben Lugo: Lighting systems benefit as well. Whether we're talking about LED backlighting, illuminated controls, status indicators, or advanced display assemblies, rigid-flex circuits allow designers to package electronics in ways that would be difficult or impossible using conventional rigid boards.

By eliminating unnecessary interconnects and utilizing 3D packaging space, designers can create systems that are lighter, more compact, and often more reliable. That's a major reason why flex and rigid-flex technologies have become such valuable tools for modern display and lighting applications.

A&D platforms have relied on sophisticated displays and illuminated controls for decades. How has that experience influenced today's flex and rigid-flex design practices?

Hansler: Like so many other technologies, display technology is changing as well, with many of the industry's most demanding reliability requirements originating in A&D applications. For decades, Pioneer Circuits has manufactured flexible and rigid-flex circuits supporting mission-control interfaces, radar systems, navigation equipment, secure

communications, electronic warfare platforms, and other mission-critical systems.

In these environments, display failures can impact mission effectiveness and operator performance. As a result, designers have learned that long-term reliability depends on much more than electrical functionality. Material selection, bend-radius management, conductor geometry, strain relief, thermal performance, and manufacturing consistency all play critical roles.

I've seen that many of those same considerations are now important in commercial display applications as products become more compact and mechanically complex.

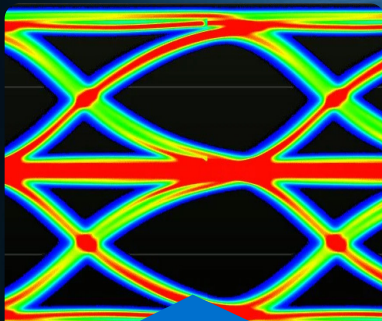
Lugo: A&D programs have effectively served as real-world laboratories for flex-circuit reliability. These systems operate in environments characterized by vibration, shock, thermal cycling, altitude extremes, and long service lives. Those conditions quickly expose weaknesses in both design and manufacturing.

The lessons learned from those applications have shaped many of today's best practices. As

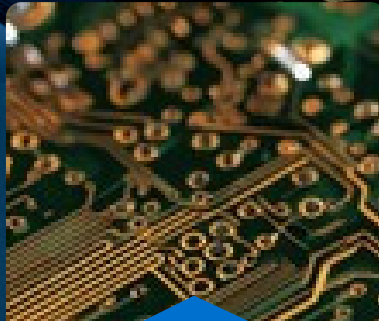
Introducing EM-896K2 and EM-896K3

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

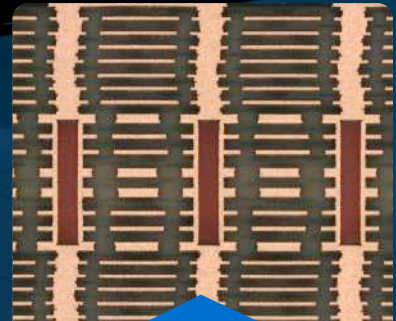
- Excellent Dk/Df performance for high-speed signal integrity (< 1.0 dB/inch @ 53GHz)
- Low X/Y-axis CTE for improved dimensional stability includes CoWoP applications
- Proven multi-lamination reliability (≥ 12 x laminations)



Excellent Dk/Df
performance



Ultra-low
X/Y-axis CTE



Proven multi-
lamination reliability

Dan mentioned, the design of a proper bend radius is critical. Also, dynamic-flex considerations, rolled-annealed copper selection, strain-relief techniques, and advanced inspection methods should not be overlooked.

What's interesting is that modern display technologies are experiencing similar challenges. Foldable consumer devices, automotive digital dashboards, wearable electronics, and advanced medical displays all require reliable interconnections within increasingly compact mechanical structures. The reliability knowledge developed for A&D applications is directly applicable to these emerging products.



If there's a single point to sum up this discussion, it's that reliable display systems don't happen by accident."

Yes, agreed. What final advice would you offer engineers developing lighting and display systems that incorporate flexible and rigid-flex circuits?

Hansler: The most important message I can emphasize is that reliability starts early. The flex and rigid-flex performance attributes of power, signal integrity, mechanical, and potential thermal management of heat are largely determined during the phases of development, material selection, mechanical packaging, and interconnect planning, not during fabrication alone.

Early collaboration between electrical designers, mechanical engineers, and manufacturing specialists can prevent many issues before they become costly production or field failures.

Lugo: Whether the application is an aircraft display, an industrial control panel, a medical imaging system, or a consumer device, the objective remains the same: delivering reliable visual information to the user.

The most successful designs recognize that displays, lighting systems, mechanical packaging, and interconnect technology are all part of a single system. When electrical, mechanical, materials, and manufacturing considerations are addressed together, engineers can maximize both performance and long-term reliability.

That's a lesson A&D programs have taught for decades, and it's one that remains highly relevant to modern display and lighting designers today.

Thank you, Dan and Ruben, for sharing your insights.

Conclusion

If there's a single point to sum up this discussion, it's that reliable display systems don't happen by accident. Whether the application is a fighter aircraft, a medical imaging platform, or the dashboard in your next car, successful flex and rigid-flex designs result from thoughtful collaboration between design, materials, manufacturing, and mechanical engineering teams.

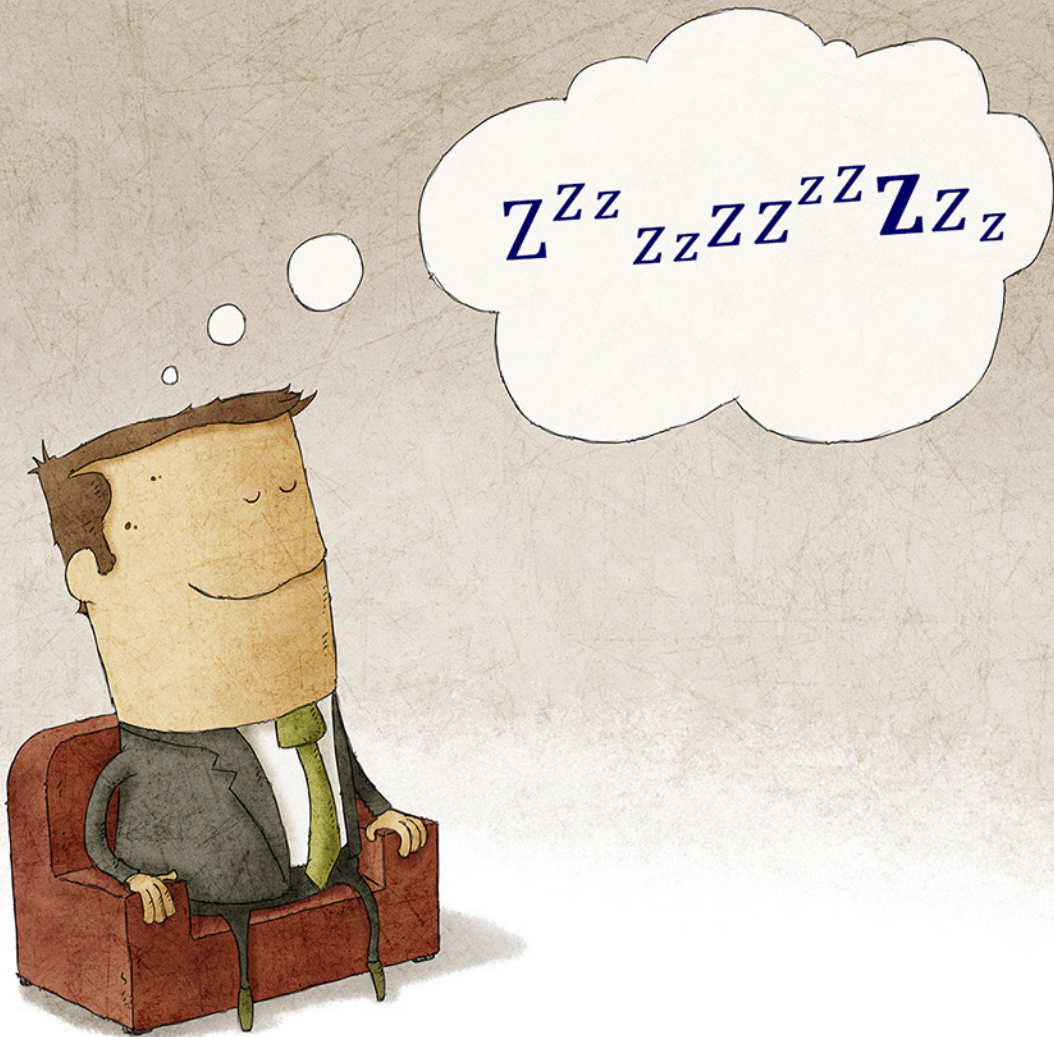
The A&D industries have spent decades teaching us that interconnect reliability matters just as much as the electronics it connects. As displays become thinner, brighter, lighter, and more complex, those lessons remain highly relevant.

From a Target Condition perspective, the goal is simple: Create interconnect solutions that perform long after everyone has forgotten who designed them. After all, the best flex circuit is the one nobody ever has to think about again.



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



www.icd.com.au

IPC Standards Released for Q2 2026

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

These are the latest releases for Q2 2026.

IPC-1401B, Environmental, Social Governance (ESG) Management System Standard

IPC-1401B specifies the requirements and best practice guidelines for an effective environmental, social, and governance (ESG) management system to help an enterprise integrate ESG as a customer requirement into products and value chain activities, as well as to identify and manage ESG risks and opportunities through cooperation with customers and suppliers, so as to enhance the competitive advantages of the enterprise and its supply chain.

IPC-2223F, Sectional Design Standard for Flexible/Rigid-Flexible Printed Boards

IPC-2223F establishes the specific requirements for the design of flexible and rigid-flexible printed board applications and their forms of component mounting and interconnecting structures. The flexible materials used in the structures are comprised of insulating films, reinforced and/or non-reinforced, dielectric in combination with metallic materials. These interconnecting boards may contain single, double, multilayer, or multiple conductive layers and can be wholly flex, or a combination of flex and rigid.

IPC-4202C, Specification for Flexible Base Dielectrics for use in Flexible Printed Boards

The IPC-4202C standard establishes the classification system, qualification, and quality conformance requirements for flexible metal-clad dielectric materials to be used for the fabrication of flexible printed boards. The purpose of IPC-4202C is to classify and characterize base materials for use in flex and rigid-flex boards for electronic applications. It is to be used for procurement and quality assurance activities.

IPC-6921, Requirements and Acceptance for Organic IC Substrates

IPC-6921 establishes and defines the qualification, performance requirements, and acceptance requirements for organic IC substrates, including wire bonding and flip-chip IC substrate products. The purpose of IPC-6921 is to provide requirements for the qualification and performance of rigid organic substrates based on the following constructions and/or technologies. These requirements apply to the finished products unless otherwise specified:

- Double-sided organic IC substrates with or without plated through-holes (PTHs)
- Multilayer organic IC substrates with or without PTHs or buried/blind vias/microvias
- Passive embedded circuitry on organic IC substrates with capacitive planes or resistive planes

IPC-9711, Generic Requirements for Automated Inspection Process Control

IPC-9711 provides generic requirements for automated inspection systems to define, set up, estab-



inTFlex

INNOVATIVE PRECISION FOR YOUR INNOVATIVE PROJECTS

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



**Global Experts in
Flex Design**



**High Reliability &
Advanced
Manufacturing**



**State-of-the-Art
Facilities**



**Comprehensive
Services**



We provide precision engineering, quality production, and cost-effective solutions that will bring your products to market faster!

CONTACT US FOR A FREE FLEX DESIGN REVIEW

sales@flexiblecircuit.com

763.545.3333

www.flexiblecircuit.com

lish, and apply process controls. Requirements include inspection parameters, calibration, detectability, resolution, threshold limits, and program setups, measurement system analysis (MSA), maintenance, and verification protocols. The standard establishes generic requirements applicable to automated inspection systems and is intended to be used in conjunction with product/process-specific standards of the IPC-971X series.

IPC-9712, Requirements for Automated Inspection Process Control for IC Substrates

The IPC-9712 standard provides requirements for automated inspection systems—automated optical inspection (AOI), automated visual inspection (AVI), and metrology—to define, set up, establish, and apply process control for manufacturing integrated chip substrates, including general and specific process and equipment conditions. Requirements include those for operating and inspection parameters, vision systems, lighting conditions, calibration, detectability, resolution, threshold limits, process

windows, program setups, measurement system analysis (MSA), maintenance, and verification protocols, in addition to the generic requirements of IPC-9711.

IPC-9716A, Requirements for Automated Inspection Process Control for Printed Board Assembly Processes

IPC-9716A provides specific requirements for automated inspection for printed board assembly processes in addition to the generic requirements of IPC-9711. Requirements are based on the use of inspection systems such as solder paste inspection (SPI), automatic optical inspection (AOI), and automatic X-ray inspection (AXI).

The purpose of IPC-9716A is to set industry-defined requirements for inspection systems to reduce false calls, ensuring quality and reliability while improving throughput and shortening cycle times. This standard also supports electronics manufacturers to enable advanced manufacturing, real-time data analytics, and control capabilities.

I-CONNECT007

Insight, Delivered

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



SUBSCRIBE NOW →



I-Connect007
The Global Electronics Resource

The Answers Behind Better PCB Design

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

PCB technology continues to evolve, bringing new design challenges, tighter performance requirements, and increasing reliability demands.

Electronics U's live online PCB design courses are taught by industry experts and designed to help you build practical skills you can apply immediately, whether you're developing a strong foundation or advancing your expertise.

PCB Design I

Aug 4 – Sep 10

PCB Design for Military, Aerospace, and Extreme Environments

Aug 3 – Sep 9

PCB Design II

Aug 4 – Sep 24

PCB Advanced Design Concepts

Aug 4 – Sep 24

PCB Design for Radio Frequency Boards (RFB)

Aug 3 – Sep 9

Looking beyond design?

Understand one of the industry's most common reliability concerns.

Reliability of Electronics – Tin Whisker – All You Should Know

Aug 18 – Aug 20

FIND THE ANSWER TO YOUR NEXT PCB DESIGN CHALLENGE.

Can't make the August session?

Several of these courses will be offered again beginning in October.

Need training for your team?

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



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

Electronics U
by Global Electronics Association

Understanding Design for Display Electronics

For Kristin Moyer, designing display electronics is all about scale. As displays shrink in size and grow in resolution, every engineering decision becomes more demanding. In this interview, the Global Electronics Association design instructor explains the strengths and limitations of today's dominant display technologies—LCD and LED—as well as the emerging role of AR/VR displays. Kristin also explores the PCB design, manufacturability, and reliability challenges that accompany each.

Marcy LaRont: Kristin, display electronics have become part of nearly every product we use today. From a PCB designer's perspective, what makes designing for displays different from designing for other types of boards?

Kristin Moyer: From an engineer's point of view, what's in the stadium LED is no different than what I design for a cellphone display; it's really a matter of scale.

What are the categories of display electronics?

The industry is really centered around two dominant categories—LCD and LED—with a third emerging category in AR/VR, like the type found in smart glasses, where the technology etched into the transparent medium provides the augmented

reality. There is technically a fourth category, plasma displays, but that technology largely peaked two decades ago.

Before we dive into the design challenges, give us a quick overview of today's display technologies.

LCD and LED displays solve the same problem in different ways. LCDs use a backlight and liquid crystals to control the light passing through the display, while LEDs generate their own light. From a design standpoint, the biggest difference is manufacturability. As displays become smaller and demand high resolutions, LEDs can be fabricated at much finer pitches than LCDs, making them the preferred technology for many modern applications.

Is that why we have seen a shift toward LED technology?

Exactly. Higher pixel density means higher resolution. Think about early Mario video games, where you could see every square pixel. Today's microLED displays pack thousands of LEDs into the same space, producing photorealistic images using less power. For PCB designers, though, that increased density creates its own challenges in routing, placement accuracy, thermal management, and manufacturability.

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

What's the first design challenge that comes to mind when you're working with LED displays?

Thermal management is one of the biggest. Whether you're designing an LED light bulb, a consumer display, or a large outdoor video board, LEDs generate heat that must be dissipated to maintain performance and reliability. As power levels and display sizes increase, designers have to think carefully about substrate materials, power delivery, cooling, and overall system design.

That's why you'll often see aluminum-core PCBs or other thermal management techniques in high-power LED applications. The challenge isn't the LED itself. Instead, it's about getting the heat away from it efficiently.

The challenge simply scales up for large stadium displays: Many high-power LEDs need to be bright enough to be seen on a sunny day, but how do you dissipate the heat? How are we dealing with all the thermal and power control, even more than the LED technology itself? All these must be considered at the level of system and PCB design.

What other special design considerations are there when designing PCBs for LED boards?

One key design issue is parts placement. For example, when designing a large display board for scrolling marquee signage, where the entire board is visible, IPC standards allow a certain amount of placement tolerance. That's fine for standard circuitry, but in display circuitry, and LED specifically, pixels must be placed exactly, which makes the soldering process so challenging. Density is another big challenge in the design process.

That's certainly a common theme, isn't it?

Yes, and it's specifically problematic here. For displays smaller than a cellphone screen that require high-resolution graphics, each LED must be mounted directly on the board. But how do you fit that many LEDs together while still routing traces or vias to each one without forcing the LEDs farther apart? How do they get soldered?

This leads most companies to purchase commercially manufactured LED display modules. Rather than placing thousands of discrete LEDs, manufacturers fabricate the entire LED array as a single

display assembly using processes more closely aligned with semiconductor fabrication. We can go through the same process but fabricate the complete LED array into its own ready-to-use package. That's how we typically get LED-based cellphone displays. It is a single fabrication process that now resembles silicon or IC chip fabrication.

It becomes easier for the designer to simply buy an off-the-shelf display that connects to your board on your display driver. You can buy these displays with or without their own interposer or controller board. Overall, it has made the design process more streamlined and easier for PCB designers.

Given how pervasive displays are, how likely will today's PCB designer encounter a display design at some point in their career?

Very likely, especially in consumer goods, avionics, and aerospace products. You probably won't see this in power electronics for a motor or similar technology, but most board designers will encounter some form of display technology.

Thermal management and placement are challenges that designers often think about. One that's less obvious is controlling the light itself. What is light bleed, and why does it matter?

Light bleed occurs when light from one LED spills into an adjacent viewing area instead of staying confined to its intended indicator or display element. Imagine a simple 4x4 LED array. Each LED may represent a different function. When one turns on, you don't want that light leaking into the neighboring windows. Preventing that requires more than PCB layout. It also requires close coordination with the mechanical design.

Display electronics are among the applications where PCB designers have to work closely with the mechanical team. Good electrical design isn't enough. You also have to think about how the light is controlled once it leaves the board.

It sounds like you would need some shielding, but what does that look like on that scale and with so much light power being emitted?

A light pipe is an old-school technology that uses a molded rigid plastic component. When you have a

vertical LED and need the light to go out to the right, you attach the light pipe, and it essentially acts like a fiber-optic guide.

There are several ways to deal with this issue, from flexible fiber optics to shielding mechanisms in how you build your enclosure (with ribs that sit between your LEDs) to an enclosure cover very close to your LEDs.

Do you have a case example that might help illustrate this challenge?

I had a job with one LCD graphics display and separate LEDs for diagnostics and status. They sat at two heights, so the enclosure cover had to be at the height of the LCD display, but the diagnostic LEDs were so low that we were getting light bleed. We solved it by using light pipe technology.

Interesting. What about designing for very small displays?

We are seeing a need for an almost HDI-level routing capability. Traditional through-hole via technology is too large, which forces LEDs farther apart unless you're trying to create a densely packed group of LEDs without using an off-the-shelf, ready-to-use LED panel display.

For the purposes of this discussion, we'll assume that's not happening.

Many of today's LED displays are pre-etched, ready-to-go, single-panel, prefabricated. But when you're doing it yourself, you worry about how small a trace you can make, given the thickness of the copper layers to be etched. Designers need to understand the relationship between minimum trace width, via size, starting copper thickness, and manufacturability.

That relationship often leads to other problems. To have LED display technology on the same board where you're generating power, you need thick copper for power delivery and very thin copper for fine-line display routing.

How do you achieve both in a manufacturable design? A designer must make some conscious tradeoffs. If the only way to achieve the LED density we want for certain display capabilities is to use the high-density footprint, there is a higher risk of assembly-related issues because we have a much

smaller solder drop.

In a high-reliability environment, a designer and product engineer must consider how to mix high density and high reliability. Ultimately, these decisions affect manufacturability, reliability, and cost.

You mentioned AR and VR as an emerging category of display electronics. How is that changing the way designers think about PCB design?

AR and VR are interesting because they blur the line between PCB design and semiconductor manufacturing. Conductive features have become so small that many of the manufacturing techniques look more like silicon fabrication than traditional PCB fabrication. Most PCB designers won't design AR display glass itself, but they will see more additive manufacturing processes and finer geometries making their way into mainstream PCB design.

What does that mean for today's PCB designer?

There's now a crossover between silicon and board design disciplines, merging out of necessity. Designers have to move beyond thinking only about subtractive PCB processes. They need to understand when additive manufacturing, like SAP and mSAP, makes sense and how manufacturing capabilities influence design decisions.

Is this an exciting development for PCB designers, and does it require new designers to have more breadth and depth than in the past?

I think it does. Engineers and designers increasingly have to think outside the box to address challenges. Solving those engineering and manufacturing challenges is what creates value for customers. The designer who understands the available materials, manufacturing methods, and design techniques is no longer just laying out circuit boards; they're solving complex engineering problems. That broader knowledge makes them a more complete PCB designer or engineer and a far more valuable contributor to the product development process.

Kristin, you've really helped us better understand the challenges and implications of today's display electronics technology.

It's an evolving field, Marcy. Happy to talk about it.

I-CONNECT007

Smart Textile Integration

With Flex PCBs

The idea of clothing that monitors your health sounds like science fiction. It is not. Flex PCBs, integrated with conductive yarns and polymer substrates, are turning ordinary fabric into diagnostic platforms and turning your shirt into a medical device. The implications for healthcare, athletics, and workplace safety are profound.

Smart Textile Integration

The fundamental engineering challenge of wearable electronics has always been the interface between two incompatible worlds: the rigid, planar world of conventional electronics and the soft, stretchy, washable world of textiles. Early smart garments solved this by simply attaching a rigid sensor pod to clothing, which was functional but bulky and uncomfortable. Flex PCBs collapse that boundary.

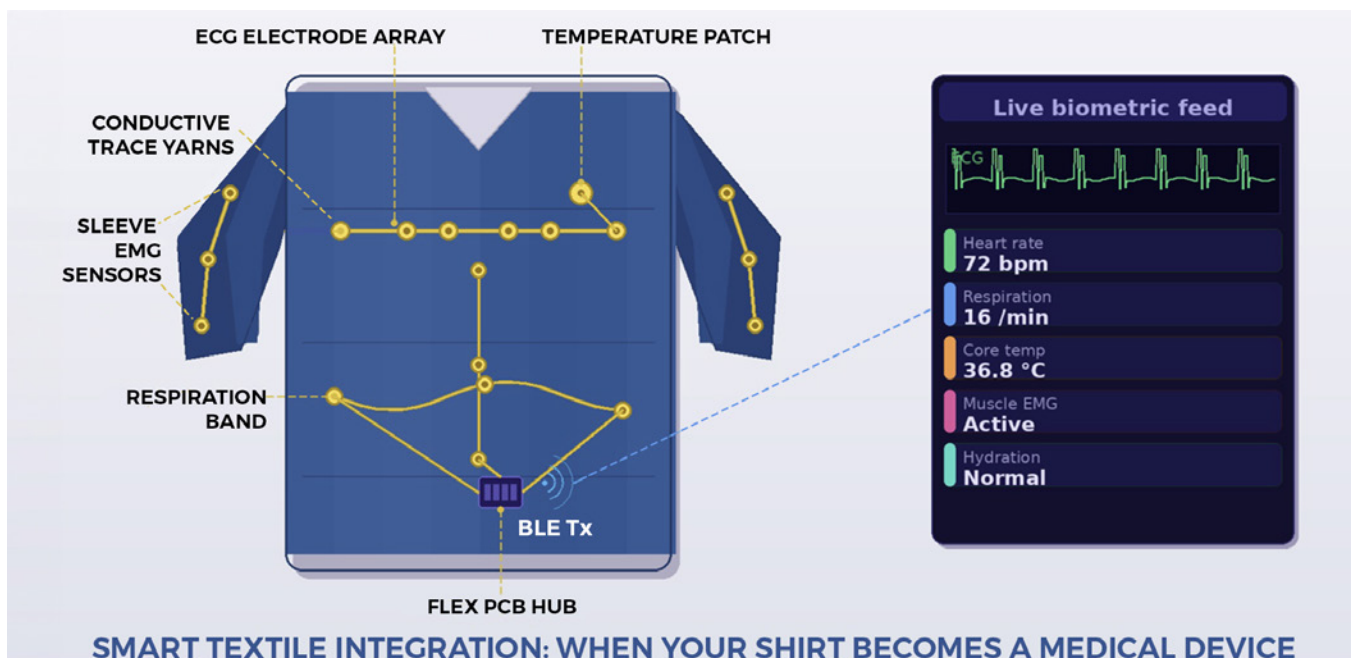
Modern smart textile integration uses flex circuits in two distinct modes. In the first, thin-film flex PCBs are laminated directly onto fabric panels; their polyimide substrates are thin and compliant enough to move with the garment during normal wear.

In the second, and more technically ambitious, approach, components are mounted on stretchable flex substrates using serpentine trace geometries that accommodate elongation without fracturing. A serpentine trace behaves like a miniature spring, straightening under tension and returning to its coiled shape when relaxed.

The biometric sensing applications are extensive. Electrocardiogram monitoring requires electrodes at specific positions on the torso. A flex array embedded in a compression shirt precisely positions those electrodes and maintains skin contact through movement, in ways no adhesive patch can match. Respiratory rate can be derived from subtle changes in chest impedance during breathing. Skin temperature gradients map blood flow. Electromyography patches on sleeves track muscle activation patterns during rehabilitation or athletic training.

Sports and Safety Applications

For athletes, the data density achievable with smart garments is transformative. A single smart jersey



can simultaneously capture cardiac rhythm, core temperature, hydration status through sweat electrolyte sensors, and the biomechanical strain patterns of specific muscle groups. Coaching staff receive this data in real time, enabling fatigue modeling and injury prediction that was simply not possible with wrist-worn devices alone.

In industrial and military contexts, smart textiles take on life-safety roles. Firefighting gear with embedded flex sensors can monitor core body temperature and alert a firefighter or their commander to dangerous heat stress before cognitive impairment sets in. Military uniforms with embedded biometric systems can relay a soldier's physiological status to field medics during combat operations.

Remaining Challenges

Washability remains the central engineering challenge. Consumer garments face hundreds of wash cycles over their lifetimes, each one a mechanical and chemical assault on delicate electronic components. Encapsulation strategies using flexible silicone overmolding and waterproof conformal coatings have extended wash cycle ratings dramatically, with some commercial products now rated to 50 or more machine wash cycles.

Summary

The convergence of flex PCB manufacturing with advances in conductive yarn, stretchable substrates, and low-power Bluetooth has brought smart textiles to the edge of mass-market viability. Within this decade, the shirt that knows your heart rate better than any watch may be as unremarkable as the smartphone in your pocket. **I-CONNECT007**

Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit*

Designer's Guide to... Fundamentals of RF/Microwave PCBs and Flex and Rigid-Flex Fundamentals. He is the author of *Thermal Management: A Fabricator's Perspective*, *The Printed Circuit Designer's Guide to DFM Essentials*, and *The Companion*

Guide to Flex and Rigid-Flex Fundamentals.

Visit I-007eBooks.com to download these and other free, educational titles.



From Prototype to Production

New Guideline Helps Industry Design, Integrate and Scale Reliable E-Textile Systems

The Global Electronics Association announces Fundamentals and Best Practices for E-Textile System Development, a new next-generation guideline that provides a structured framework for the design, testing and deployment of e-textiles systems.

As e-textiles increasingly move from prototype to production, development teams face unique challenges, requiring coordination among disciplines that often use different design approaches, terminology, performance expectations and test

methods. Developed by a global group of industry experts, this guideline establishes a common technical foundation for e-textiles; covering materials selection, integration methods, interconnection approaches, reliability, testing, cybersecurity, and sustainability. It serves as an educational resource for organizations seeking to better understand e-textiles technologies, bridging knowledge gaps between textiles and electronics industries.



[Continue reading here.](#)

‘Let There Be Light’— Emitting Diodes!

BY JOE FJELSTAD, VERDANT ELECTRONICS

Display technology has become ubiquitous in today’s electronic products, driven largely by smartphones. This touchscreen technology increasingly dominates our means of communication, both for voice and text. Our connection to the world outside our homes, cars, and offices is now enjoyed 24/7 via the internet, to which they are

invariably connected. In terms of time and rate of social change, such devices have gone from “nice to have” to “indispensable” in the seeming blink of the proverbial eye.

The key has been the mind-numbing advances in visual display technology, particularly the light-emitting diode (LED). The technology has roots

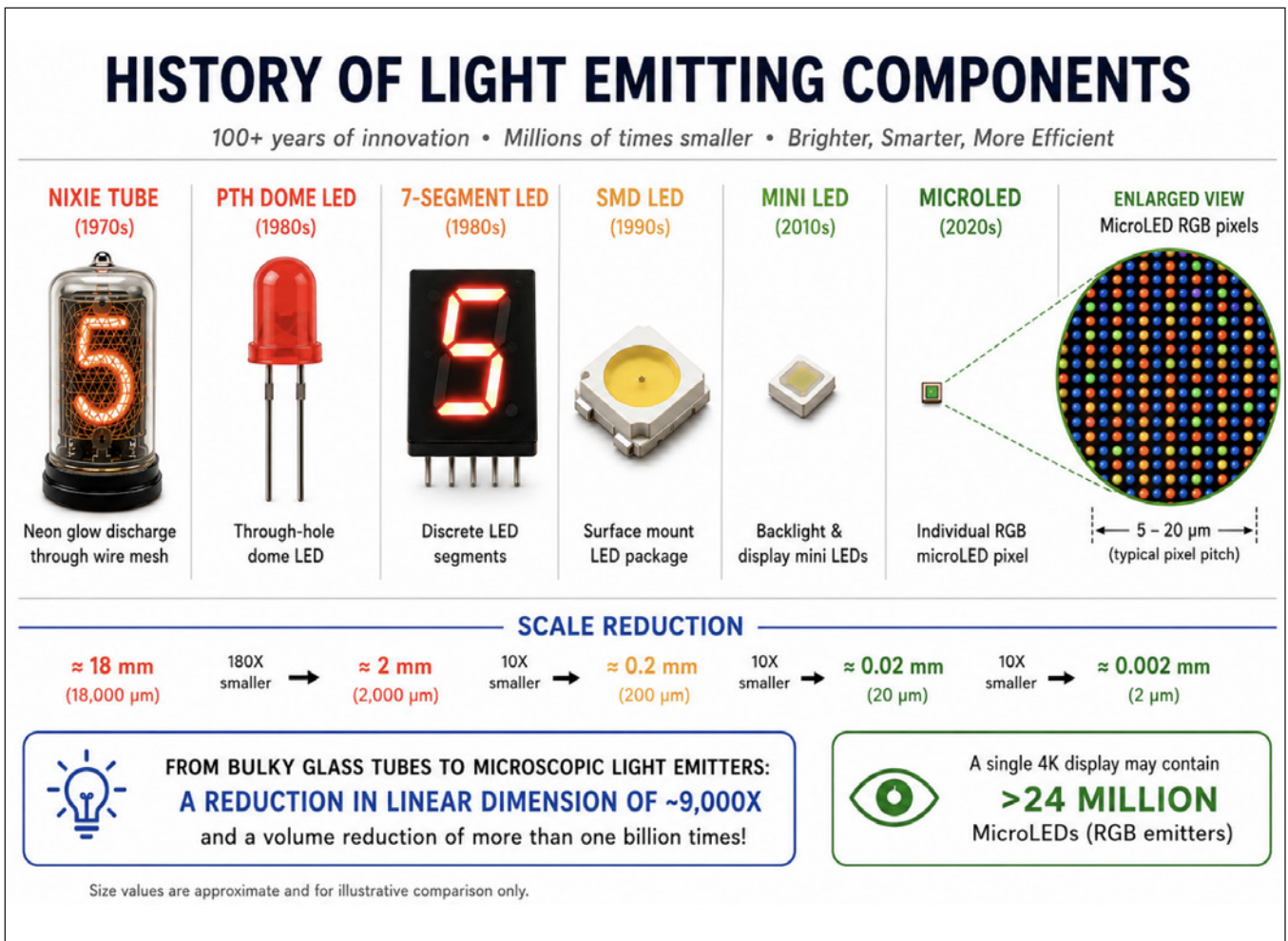
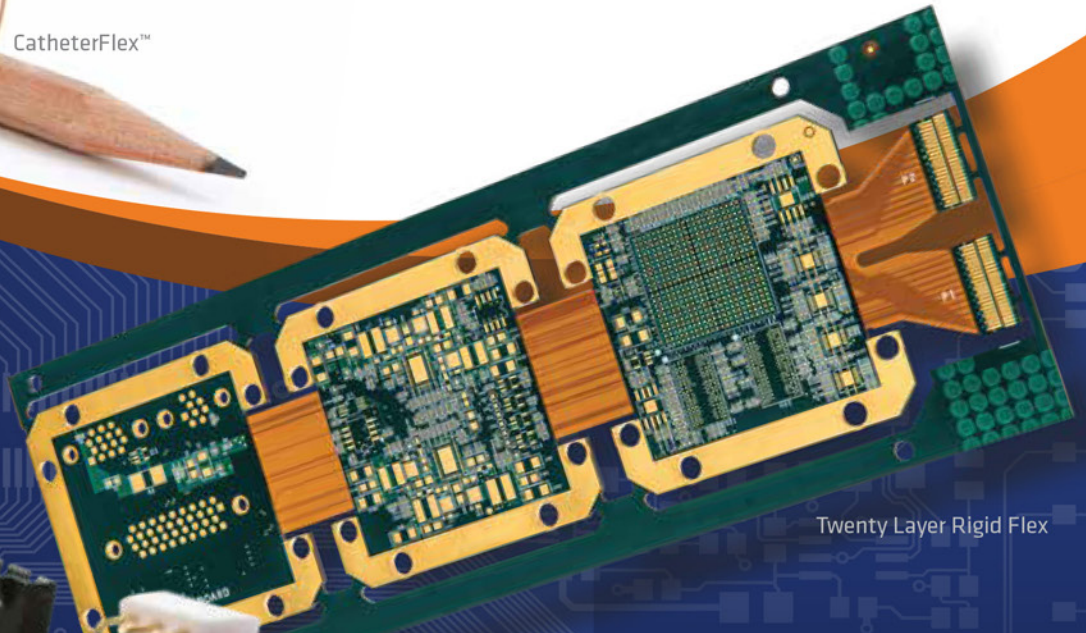


Figure 1: This AI-generated graphic based on the article’s content nicely illustrates the history of light-emitting components.

ALL YOUR FLEXIBLE SOLUTIONS IN ONE PLACE



CatheterFlex™



Twenty Layer Rigid Flex



And, Assembly Tool!

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

All Flex Solutions

1601 Cannon Lane
Northfield, MN 55057

(507) 663-7162
AllFlexInc.com



SCAN ME


all flex
SOLUTIONS

back to the discovery of electroluminescence by H. J. Round in 1907, but did not see much commercialization until the red-light LED was demonstrated by Nick Holonyak at GE in 1962. Holonyak is commonly referred to as the “Father of the LED.”



In 2000, a white light LED delivered roughly 20 lm/W; by 2010, it delivered 100 lm/W, and today the efficiency is sitting at an amazing 150–250+ lm/W.”

The devices were not all that bright, but they were much less power hungry than the neon orange light-emitting “Nixie tubes,” which were common in the 1970s. They set the stage for the development of formats and colors: orange, yellow, green, and blue ushered in low power, high lumens per watt (lm/W). In home lighting, these long-lasting white-light LEDs have replaced incandescent bulbs, where most of the energy consumed is lost as heat. Efficiency has also increased rapidly. In 2000, a white light LED delivered roughly 20 lm/W; by 2010, it delivered 100 lm/W, and today the efficiency is sitting at an amazing 150–250+ lm/W.

Display technology using LEDs has relatively quickly gone through several stages in package formats. Initially, it was the dome-shaped or tubular LED, familiar to many electronics hobbyists. These are still relatively large and are most often used for larger-format outdoor displays because the individual pixels cannot be easily distinguished at a distance; however, they are unsuitable for personal electronics.

Developers created newer formats, including the first surface mount versions. What is truly amazing

is that the internal construction of the original LED, a semiconductor die mounted on a lead frame with a bond wire and clear encapsulant, is nearly indistinguishable, conceptually, from modern LED packages.¹ The major difference is that today's microLEDs can be 1,000 to 10,000 times smaller than those early through-hole devices.

Mini and microLEDs are often used in televisions, monitors, and smartphones. The technology progression for light-emitting electronic components is shown in Figure 1. This brings us to the present and the significant challenges of assembling devices as small as a mote of dust. The limitations of conventional pick-and-place assembly, the assembly of microLED displays, are summarized in the technology comparison chart shown in Table 1.

MicroLED technology employs microscopic light-emitting diodes, typically fabricated from gallium nitride (GaN), that individually function as red, green, or blue pixels. Depending on display resolution and application, driven by the distance of the viewer's eye to the display, LED devices used in displays may range from approximately 3 to 50 μm in size, with many advanced display architectures targeting device dimensions in the 5–20 μm range. For perspective, a human hair is approximately 70–100 μm in diameter.

The manufacturing challenge is staggering. A 4K television contains more than 8 million pixels, and because each pixel generally requires separate red, green, and blue emitters, the display may require placement of over 24 million individual microLED devices. Future 8K displays may require nearly 100 million emitters.

Therein lies the rub. Conventional pick-and-place systems, which excel at placing packaged components measured in millimeters, become increasingly challenged when required to handle millions of fragile semiconductor devices measured in micrometers. At these dimensions, device handling, alignment accuracy, throughput, contamination control, and yield become major obstacles to commercialization. Even a small defect rate can translate into thousands of failed pixels, and even a single pixel failure in a field of millions can be picked up by the human eye.

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.

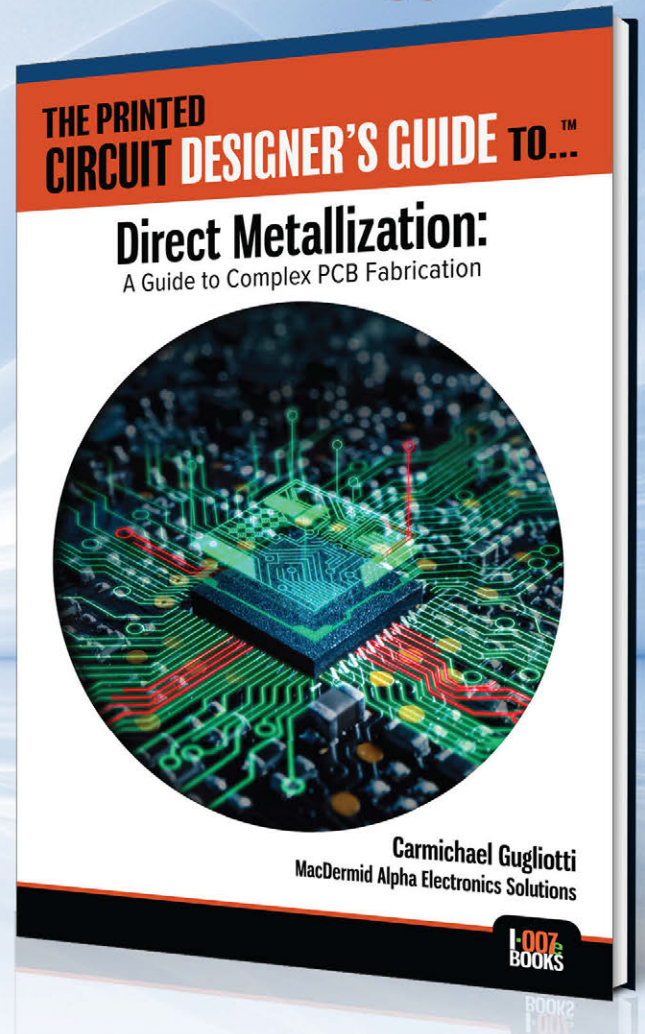
Featuring insights relevant to PCB designers, fabricators, OEMs, and process engineers, this book provides the technical foundation needed to understand, evaluate, and implement direct metallization technologies in next-generation manufacturing environments.

"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



Table 1: Conventional pick-and-place vs. mass transfer for microLED manufacturing

CONVENTIONAL PICK-AND-PLACE	MASS TRANSFER APPROACH
Pick one LED device	Pick thousands to millions of LEDs simultaneously
Move one device	Move entire arrays in a single operation
Place one device	Place complete pixel populations in parallel
Repeat millions of times	Repeat only a small number of transfer cycles
Serial manufacturing process	Highly parallel manufacturing process
Throughput limited by placement speed	Throughput scales with transfer area
Mechanical handling of every device	Reduced handling of individual devices
Greater risk of emitter damage	Lower mechanical stress on fragile microLEDs
Placement errors accumulate over millions of operations	Improved global placement accuracy and registration
Increased opportunities for particle contamination	Reduced contamination exposure
Yield losses from repeated handling and placement	Higher potential manufacturing yield
Higher capital and operating costs	Lower cost per transferred device
Practical for larger LEDs and lower volumes	Essential for ultra-small microLEDs and high-volume production

The simple reality is that microLED devices are extremely thin and fragile, and any transfer process must avoid mechanical damage to the delicate devices while maintaining placement accuracy, often measured in a few micrometers. Furthermore, the economics of display production require that these transfers occur not one device at a time, but in massively parallel fashion.

For what it's worth, I was invited to write about this topic for July's issue, but it was quite propitious as I have been friends since 2008 with and an advisor to Dr. Jayna Sheats, founder of Terecircuits (it has

changed names over the years). Dr. Sheats is a brilliant scientist with a PhD from Stanford and valuable experience as a researcher at HP Labs in Palo Alto.

The reason I say "propitious" is because Sheats and her team at Terecircuits have developed a unique and useful technology called Terefilm that seems almost custom-designed for microLED assembly, though its usefulness goes far beyond. The novel photopolymer platform is particularly interesting in that, rather than relying on mechanical pickup and placement of individual devices, Terefilm enables temporary bonding and highly selec-

tive release of semiconductor and/or other discrete devices (i.e., not just microLEDs) through photochemical activation and total gasification of the bonding material. Devices or groups of devices can be released from a donor substrate and transferred with high precision while minimizing mechanical handling and obviating cleaning.

In Table 1, the significance of the Terefilm approach to microLED assembly lies in its ability to replace serial assembly operations with highly parallel transfer processes as illustrated in the above chart. Note again that the technology is not limited to microLEDs alone but can be adapted to handling and assembling bleeding edge and prospective next generation discrete devices such as the 008004 (0.25mm × 0.125mm) and the experimental <0.2mm × 0.1mm.

For microLED manufacturing, the Terefilm approach thus offers several potential significant advantages:

- Simultaneous transfer of thousands or millions of devices
- Reduced mechanical stress on fragile emitters
- Improved placement throughput
- Lower contamination risk
- Enhanced yield potential
- Reduced manufacturing cost

While microLED displays represent one of the most visible applications for Terecircuits' technology, the same challenges increasingly appear today in advanced semiconductor packaging, chiplet integration, heterogeneous assembly, silicon photonics, RF modules, and embedded electronics. In each case, manufacturers are confronted with the task of accurately positioning often vast numbers of extremely small and delicate devices at economically viable production rates—a challenge that will not get easier with current methods.

As electronic systems continue to shrink while component counts increase, the industry's future may depend less on how quickly machines can pick up components and more on how effectively materials and processes can move them in parallel.

Technologies such as Terefilm may therefore represent not merely an improvement in assembly efficiency, but a fundamental shift in how next-generation electronic systems are manufactured.

A Closing Thought

Terefilm combines temporary bonding and debonding functionality with lithographic pattern ability. Through photochemical activation, individual devices or groups of devices can be selectively released from a donor substrate and transferred with high precision. Upon activation, the material decomposes cleanly into gaseous products without leaving residue, minimizing contamination and eliminating many cleaning requirements associated with conventional release methods. It seems to me, and to a growing number of others, that the technology has a definite “place at the electronics assembly table.” Check it out.

References

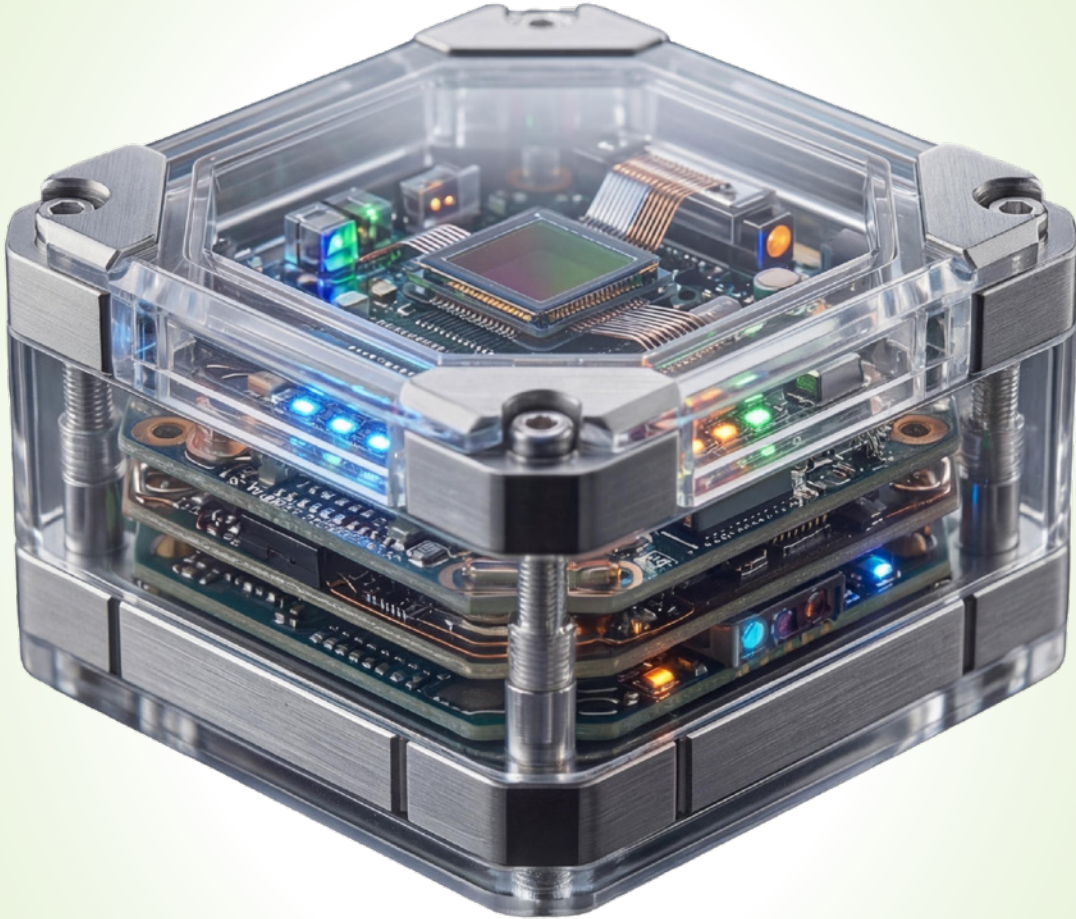
1. My former company, Tessera (thrice renamed and now called Aedia), applied for a patent for one of my disclosures, “Semiconductor Package Having Light Sensitive Chips in 1997. It concerned semiconductor packaging structures incorporating light-emitting semiconductor devices within a package architecture, and according to AI, the patent is cited in later optoelectronic and semiconductor packaging patents and appears repeatedly in patent citation records.



Joe Fjelstad Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or

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

Performance Is Won in Assembly



BY BRIAN BUYEA, REMTEC

There's a moment in every design cycle when the conversation shifts from the elements of a substrate—material selection, thermal conductivity, circuit density, layout, and performance on paper—to the how it performs in assembly. No matter how good your substrate is, if the assembly isn't right, none of it matters.

Engineers are coming to terms with the fact that performance is more than the circuit. It's now about

the system, and that's where assembly stops being an afterthought and becomes the whole game.

Assembly Isn't an Add-On, It's the Product

When people hear "assembly," they often think PCB assembly, pick-and-place, solder, and reflow. In ceramic electronics, and especially in high-power, high-reliability applications, assembly means something very different:

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

- Die attach that can survive extreme thermal cycling
- Wire bonding that maintains integrity under vibration and current load
- Encapsulation that protects without compromising thermal performance
- Packaging and interconnects that are hermetically sealed
- Housing integration that turns a component into a system



The future of electronics is being built at the intersection of materials, design, and assembly, where performance is won, reliability is proven, and systems either succeed or fail.”

These describe microelectronics assembly and packaging, where electrical, thermal, and mechanical performance all converge. If you get it wrong, failure shows up in the field, in the air, or in outer space.

Why Integration Matters More Than Ever

In the old model, one company made the board, another assembled it, then another company handled packaging, and everyone hoped it would all work together. Sometimes, it did, but more often, there were delays, redesigns, and cost overruns.

That happened because whenever you hand off a product, you introduce risk: misaligned tolerances, thermal mismatches, material incompatibilities, and communication gaps between teams.

What companies like Remtec have done is eliminate those handoffs. We’ve pulled substrate, metalization, and assembly into a single, integrated process built around our proprietary Plated Copper Thick Film (PCTF®) technology.

Now, the same team that designs the substrate understands how it will be assembled, and the same engineers who build the circuit are thinking about electrical performance, heat flow, stress, and reliability from the start. This process reduces failure points before they ever exist.

The Real Work Happens at the Die Level

System succeed or fail in the die attach, where the semiconductor meets the substrate. That’s where heat starts to move or doesn’t. In high-power applications—EVs, industrial drives, RF amplifiers—that interface is everything.

As I mentioned, a poor die attach creates thermal resistance, leading to heat buildup and failure. Advanced ceramic assemblies solve this by:

- Using materials with high thermal conductivity
- Optimizing attach methods for minimal resistance
- Designing for uniform heat spreading across the substrate

These circuits keep things cool and maintain performance under load, over time, in the real world.

Wire Bonds: Small Details, Big Consequences

Wire bonds don’t get much attention because they’re tiny and hidden. Although they seem simple, they are not. In high-frequency or high-current applications, wire bonds are critical to signal integrity and electrical performance. They also have to survive vibration, temperature swings, and mechanical stress.

Poor bonding leads to signal degradation, intermittent failures, and catastrophic opens under stress. In an integrated assembly environment, wire bonding is engineered to match to the substrate, the materials, and the application. It turns a fragile connection into a reliable one.

Encapsulation and Protection Without Compromise

Once the die is attached and the bonds are in place, the next challenge is protection. Encapsulation not only seals the device, it protects without introducing new problems: If it's too rigid, you create stress fractures. If it's too soft, you lose structural integrity, and if it's too insulating, you trap heat.

Advanced ceramic assemblies can balance mechanical protection, thermal conductivity, and environmental resistance because the environments these systems operate in aren't forgiving. Some examples where you would see this include under-the-hood automotive conditions, aerospace vibration and altitude extremes, and defense systems operating in harsh, unpredictable environments. You don't get a second chance in those applications.

Housing Integration: Turning Components Into Systems

Now, at some point, the assembly stops being a component and starts becoming a system. This is the role of housing integration. Instead of designing a circuit and then figuring out how to package it, integrated assembly approaches design with everything together: substrate, interconnects, thermal paths, and mechanical housing.

The result is a system that's more compact, efficient, and reliable. It's designed to work as a whole, not as a collection of parts.

Why does this matter for today's applications? If you're designing low-power consumer electronics, you can get away with a lot, but you can't when you're designing high-power, high-reliability systems.

I'm seeing integrated ceramic assembly showing up in places where performance matters, from electric vehicle power modules and industrial motor drives to RF/microwave systems, aerospace and defense electronics, and medical devices.

These applications demand high power density, efficient heat dissipation, and long-term reliability under stress. These demands are increasing.

The Cost Conversation

Yes, integrated ceramic assemblies cost more up front, and there's no point pretending otherwise.

But it's the wrong comparison. The questions should be:

- What do performance trade-offs cost?
- What does failure cost?
- What does redesign cost?
- What does downtime cost?

When you look at total system cost over the life of the product, integrated assembly often comes out ahead because it reduces failure rates, rework, field issues, and warranty exposure. In other words, it reduces risk.

From Vendor to Partner

Companies aren't just looking for someone to build their substrate. They want partners who can understand the application, design for performance, manufacturability, and reliability, integrate assembly from the start, and deliver a complete, working system. It's a different kind of conversation that requires a different kind of capability. Remtec combines ceramic substrates, metallization, and assembly into a single offering.

Where the Future Is Being Built

The future of electronics is being built at the intersection of materials, design, and assembly, where performance is won, reliability is proven, and systems either succeed or fail. The next time you're evaluating a design, don't stop at the substrate. Instead, ask who's building the system. That's what your customer is buying, and what will determine whether your product performs in the real world, where it matters most. **I-CONNECT007**



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

The Display Revolution

Now With Fewer Tiny Keys

BY JOHN WATSON, PALOMAR COLLEGE

From the earliest days of electronics, engineers have worked within some shared assumptions about how systems should be built and how people should interact with them. These assumptions often feel so natural that they are rarely questioned; they define what "normal" is.

Thomas Kuhn, the American philosopher of science best known for introducing the idea of "technical paradigm shifts," observed, "What someone sees depends upon what he looks at and also upon what his previous visual-conceptual experience has taught him to see." In other words, we do not readily notice the framework we are in while using it, nor how it relates to the broader picture of innovation.

After many years in the electronics industry (and I am being kind to myself with "many"), I have learned that change is often easier to recognize

in the rearview mirror. While it is happening, it may look like another new tool, process update, customer requirement, or a "minor improvement" that somehow changes everything. Only later do we realize how much the industry has shifted.

A good example is the way people have interacted with electronics. Specifically, for nearly 40 to 50 years, the same basic model shaped how computers interacted: input occurred in one place, and output appeared elsewhere. A keyboard, keypad, mouse, switch, and, yes, even punch cards or buttons told the system what to do. A screen, a light, a meter, or a printed result told the user what happened.

But over time, that model began running into practical limits. Physical controls worked well, but they were fixed in place. A button could only

We are dedicated
to excellence through

innovation, technology and most importantly, service.



COMMERCIAL • MILITARY • MEDICAL • BAREBOARD



U.S.CIRCUIT

www.uscircuit.com



do what it was designed to do. A keypad needed space. A label had to be printed. A control panel could quickly become crowded as products added more features. As electronic products became more capable, the physical interface grew more complex.

At the same time, users began expecting more from their devices. They wanted more information, clearer feedback, faster responses, and simpler operation. Products were becoming more capable, but that added capability created a new problem: How do you give the user access to more functions without turning every device into the cockpit of a 747?

A personal example was the BlackBerry I had around 2003. Like everyone else at the time, I thought it was the greatest thing ever made. It put email, messaging, contacts, and business communication right in my pocket. At the time, it was revolutionary because it was portable, practical, and powerful. For many of us, it changed how we communicated.

But looking back, the BlackBerry still followed the old paradigm: the screen displayed information, and the built-in keyboard handled input. That felt normal because it was the world we were used to. The limitation, however, was hard to miss. The keyboard took up valuable space, which kept the screen small. And those tiny keys posed a very personal user interface challenge: My fat fingers on a small keyboard are not a productivity feature. They are a spelling disaster with a battery.

That is how paradigm pressure begins. The old model still works, but its limits become harder to ignore. The BlackBerry keyboard was useful, even brilliant for its time, but it also showed the constraint of fixed hardware. The keys were always there, whether you needed them or not. They took up space whether you were typing, reading, browsing, or looking at a photo. The interface was physically locked into the product.

Then came 2007 and the release of the iPhone. Steve Jobs did not just introduce another phone; he helped reveal a new paradigm. Several technologies had converged at the right time: a better display, capacitive touch, faster mobile processing, improved software, and a user interface that felt natural.

The key shift was simple: The screen was no longer just an output device, but where interaction happened.

That changed the display's role. The keyboard no longer had to be permanently built into the hardware; it could appear only when needed. The same surface could become a keyboard, a menu, a camera viewer, a map, a browser, a game controller, or a control panel. The screen became flexible in a way that physical controls could never be.

Just a historical point: BlackBerry is a paradigm shift that did not survive. It was built around a world in which input and output were separate, with the physical keyboard as the core of the user experience. That model worked extremely well and made BlackBerry dominant.

But the industry shifted to large, touch-based, software-driven screens where the display became the main interface. The keyboard went from essential to optional. BlackBerry tried to adapt, but it was still anchored to its original design philosophy while the market had already moved on.

In simple terms, the foundation that made BlackBerry successful was the same thing that made it hard to transition into the new paradigm. Just like that, with a new paradigm, my fat fingers now had a fighting chance.

This shift did more than change how we use electronics. It changed how those electronics are designed. Once the display became the main point of interaction, the PCB behind it became more important. The display is no longer just a feature added to the product. In many cases, it is the part of the product the user sees, touches, and judges first.

The user sees the screen. The PCB designer sees what makes that screen work: power, signals, connectors, drivers, heat, grounding, and reliability. A display may look simple from the outside, but behind it is a complete electronic system.

Correct display design matters because the display now serves two roles at once. It presents information, but it is also often where the user interacts with the product. That makes it one of the most visible and important parts of the system. If the display works well, the product feels modern, clear, and reliable. If it does not, the product immediately feels flawed.

For PCB designers, this raises the stakes. Problems that might stay hidden in another part of the circuit become obvious when they affect the display. Flicker, dimming, uneven brightness, slow response, touch errors, or early failure immediately shape the user's opinion of the product. The user may not know whether the issue is power delivery, signal routing, grounding, thermal design, or component placement. They just know the product doesn't feel right.

In simple terms, the display is where the product meets the user. The PCB determines whether that meeting feels clear, reliable, and professional.

Kuhn's observation fits this shift well. For decades, electronics followed a familiar framework: input occurred in one place, and output appeared elsewhere. That model shaped product design, user

interaction, and engineering thinking. But the display is no longer just a window into the system. It has become the point of interaction, communication, and judgment.

This month, I have focused on the larger shift, the "why" behind the growing importance of displays. Next month, I'll move into the "how" by examining the design specifications and layout details required to ensure display electronics operate reliably in real products. **I-CONNECT007**



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

BELOW THE SURFACE

Ceramic vs. PCB

When 'Good Enough' Becomes System Failure

BY CHANDRA GUPTA, REMTEC

Let's start with the question engineers are increasingly asking, not in design reviews, but quietly, and often too late: When should I use ceramic instead of FR-4? It's a fair question. FR-4 has been the backbone of electronics for decades. It's affordable, widely available, and for many applications, it works just fine. Until it doesn't.

That's the uncomfortable truth: Good enough is often the most dangerous decision in high-performance electronics. Not because it fails immediately, but slowly and expensively over time.

The Real Limits of Traditional PCB Materials

FR-4 has long been the default substrate for PCBs, but at its core, it is a compromise material. It was never engineered for the demands of extreme environments, high power densities, or multi-GHz signal integrity. Instead, it was designed to be broadly usable, balancing cost, mechanical strength, electrical insulation, and manufacturability for mass-market electronics.

This "good enough" philosophy becomes a limitation as performance requirements push beyond its original design envelope.

- Thermal conductivity is low (typically ~0.3–0.4 W/m·K), - heat builds up, not out
- Dielectric properties vary with temperature and frequency
- Mechanical expansion is inconsistent under stress
- Moisture absorption changes performance over time

In low-demand systems, these limitations are manageable. In high-performance systems, they become failure mechanisms. Yet, many designs still default to FR-4 because it's familiar, easy to source, and inexpensive—on paper.

[Continue reading this column 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!

CAREER OPPORTUNITIES



FAE Wanted

San Francisco Bay Area, United States

PCB Recruitment Ltd - the **global PCB and electronics industries recruitment specialists** - are working in partnership with our client to source a world-class FAE in the Bay Area.

Outstanding Field Application Engineers in the San Francisco Bay Area – we have an exceptional opportunity that's well worth your attention.

Join one of the world's leading manufacturers of high-performance server PCBs at a pivotal stage of growth.

We're looking for someone with **specialist expertise** in high-speed signal PCBs and experience supporting AI infrastructure, GPUs, 800G–1.6T networking, or similar high-performance computing applications.

Salary: \$250,000–\$300,000 plus an excellent, industry leading benefits package.

For a confidential, no-obligation conversation, contact william@pcbrecruitment.co.uk.

This opportunity is not to be missed and could be an immense step in your career.

Apply Now!



TECHNICAL SALES MANAGER

Arlon Electronic Materials. Rancho Cucamonga, CA

Arlon Electronic Materials, a subsidiary of Elite Material Co., Ltd., is the #1 PCB laminate supplier by revenue worldwide, the #1 rigid CCL supplier, and the #1 supplier of AI data center PCB material. We are hiring a Technical Sales Manager to drive growth across the U.S. defense, aerospace, and high-reliability electronics market.

In this role you will generate revenue across Defense & Aerospace OEM and PCB fabricator accounts, serve as the technical liaison between customers and Arlon's engineering team, and build long-term strategic partnerships that support mission-critical programs. It is a hunter-farmer role for someone who can translate material science into customer value, with roughly 50% travel to customers and industry events.

We welcome experienced engineers and technical professionals from the PCB/PCBA industry ready to step into sales. Requirements: a BS or MS in Engineering, Science, or an equivalent technical field; 5+ years in the PCB/PCBA industry; and strong written and verbal communication. Must qualify as a U.S. Person per ITAR.

Compensation: base \$105K–\$161K, with OTE of \$150K–\$200K+ and uncapped bonus potential.

Apply today: recruiting@arlonemd.com

This position is subject to ITAR/EAR regulations. Arlon EMD is an Equal Opportunity Employer.

Apply Now!

CAREER OPPORTUNITIES



Manufacturing IT/ERP Specialist

Remtec is growing and has two exciting opportunities at our dynamic, team-oriented, and innovative tech-manufacturing company, based in our newly renovated headquarters in Canton, MA (just south of Boston).

In a manufacturing environment, the Manufacturing ERP Specialist is the backbone of daily operations and frontline IT support. This role supports and enhances the company's Enterprise Resource Planning (ERP) system while also providing general IT support to ensure reliable systems, accurate data flow, and strong user adoption across the organization.

The ERP Specialist partners closely with Production, Supply Chain, Purchasing, Finance, Quality, HR, and external service providers to maintain system integrity, streamline workflows, reduce errors, improve productivity, and support company growth. This position provides hands-on ERP and IT support to employees at all levels and serves as a key liaison with third-party technology vendors.

Preferred Skills: Epicor ERP, cloud-based ERP systems and SaaS environments, basic networking concepts and endpoint management. Experience in supporting manufacturing users and onboarding/offboarding processes in collaboration with HR is desired.

Join Remtec and contribute to a collaborative culture focused on quality, integrity, and innovation.
Learn more and apply here:

Apply Now!



eptac
TRAIN. WORK SMARTER. SUCCEED.

Instructor – Electronics Manufacturing

EPTAC is seeking experienced professionals from the electronics manufacturing industry to join us as Instructors. This role offers flexible part-time or full-time opportunities. Ideal candidates will have expertise in soldering, quality control, PCB design, or related fields

Responsibilities:

- Teach IPC Certification and electronic 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 2 locations. Our headquarters is in Salem, NH. We offer a modern, positive environment with benefit including health care, paid time off, retirement savings, and professional development.

Email careers@eptac.com to apply.

Apply Now!

CAREER OPPORTUNITIES



Independent Sales Representatives Wanted

P. Kay Metal / MS2 Technologies, headquartered in the United States, is seeking established **independent sales representative firms** to add a proven, patented product to their portfolio.

We manufacture and supply **MS2® Molten Solder Surfactant**, a patented solution designed for both **Tin/Lead and Lead-Free soldering processes**, delivering measurable performance and cost benefits to electronics manufacturers.

MS2® already has successful customer installations throughout the United States, as well as a growing international footprint with active customers in Mexico, El Salvador, Honduras, Ecuador, and the Dominican Republic. We ship globally and support customers worldwide.

This opportunity is ideal for rep firms currently calling on electronics manufacturing, PCB assembly, soldering, or process engineering accounts who are looking to expand their offering with a differentiated, high-value consumable.

Strong technical products, international growth potential, and factory support provided.

To learn more about our company and technology, visit **www.pkaymetal.com**.

Interested firms are encouraged to reach out for additional details to Gladys Flores at gladysflores@pkaymetal.com.

Apply Now!



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



Sr. Test Engineer (STE-MD)

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

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

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

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

Apply Now!



Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

Apply Now!

CAREER OPPORTUNITIES



Quality Assurance Specialist—Bare Board PCB Manufacturing

Accurate Circuit Engineering seeks an organized, technically proficient Quality Assurance Specialist dedicated to bare board PCB production. You will champion compliance with industry standards, lead internal audits, manage certifications, and drive continuous improvement based on product performance data and customer feedback.

Key Responsibilities:

- **Standards Compliance & Certification:** Enforce IPCA600, IPC6012 (CIS/CIT preferred), and ISO 9001 quality standards throughout fabrication
- **Internal Process Auditing:** Conduct scheduled and ad hoc audits of incoming materials, fabrication stages, testing protocols (etest, AOI), and documentation traceability
- **Employee Training & Development:** Create and deliver training programs for inspectors and production staff on IPC standards, quality procedures, and inspection tools to maintain certification
- **Failure Analysis & Corrective Actions:** Investigate nonconforming boards—including internal findings and customer returns/RMAs—analyze root causes, and lead corrective/preventive actions (8D/CAPA)
- **Procedure Optimization:** Analyze quality trends and RMA data to update processes, inspection checklists, and control plans

Qualifications:

- Associate degree or equivalent experience in electronics manufacturing
- 3+ years in bare board PCB QA, with IPCA600/ CIS and IPC6012 certification
- Strong auditing, training, documentation, and cross-functional collaboration skills
- Proficient in rootcause failure analysis

Join us to ensure rigorous compliance, elevate fabrication quality, and continuously improve manufacturing standards.

Contact brandon@ace-pcb.com and James@ace-pcb.com to apply.

Apply Now!



Are You Our Next Superstar?!

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

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

Apply Now!

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.

View all open roles and apply here:
<https://www.linkedin.com/company/arlon-emd/jobs/>

Apply Now!

I-Connect007
The Global Electronics Resource

THE ELECTRONICS INDUSTRY'S HIRING NETWORK

**Your Next Opportunity
Starts Here.**



START HIRING

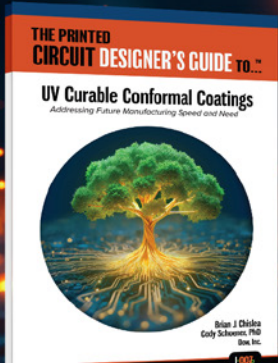
Unlock a World of Technical Knowledge INSTANTLY



START LEARNING NOW!

I-007eBOOKS

EDUCATIONAL RESOURCES



"A great read on materials and processes that help electronics succeed in harsher conditions."
—Jason Keeping, Celestica

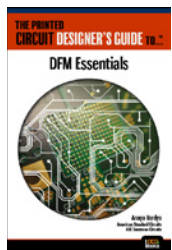
Look Inside



More Secrets of High-Speed PCBs

by Martyn Gaudion, Polar Instruments

This book is designed to help bridge the gap between the idealized environment of your CAD or CAM system, the specifications for PCB procurement, and the complex realities of manufacturing processes. [Read it now!](#)



DFM Essentials

by Anaya Vardya, American Standard Circuits, ASC Sunstone Circuits

One of the biggest challenges facing printed circuit board designers is not understanding the cost drivers in the PCB manufacturing process, particularly the manufacturing of advanced technology PCBs. The guidelines offered in this book are based on both ASC recommendations and IPC standards. [Download your copy today.](#)

PODCAST SERIES



I-Connect007

The Global Electronics Resource

EXECUTIVE DIRECTOR | MANAGING EDITOR

MARCY LARONT

marcy@iconnect007.com • (480) 280-5229

Associate Editor

MICHELLE TE

michelle@iconnect007.com

Technical Editor

PATTY GOLDMAN

Contributing Technical Editor

HAPPY HOLDEN

happy@iconnect007.com

Business Development Manager

BARB HOCKADAY

barb@iconnect007.com

Marketing Services Manager

TOBEY MARSICOVETERE

tobey@iconnect007.com

Digital Marketing Manager

BRITTANY MARTIN

brittany@iconnect007.com

Art Director | Production Manager

SHELLY STEIN

Magazine Layout

TYLER ZAPOLUCH

Ad Design

TOBEY MARSICOVETERE

Director of Digital Operations

and Media Systems

BRYSON MATTIES

Cover Design

SHELLY STEIN

Cover Image

ADOBE STOCK © VADYM

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.

July 2026, Volume 1, Number 7

I-CONNECT007 MAGAZINE is published monthly, by IPC Publishing Group, Inc., dba I-Connect007.

ADVERTISER INDEX

Accurate Circuit Engineering	5
all4-PCB	11
All Flex Solutions	99
Arlon	19
ASC Sunstone.....	77
atg Luther & Maelzer GmbH.....	71
Candor Industries	105
Chemcut	51
Electra Polymers	37
Elite Materials.....	85
Flexible Circuit Technologies.....	89
FS Quality	9
Global Electronics Association	33, 91
GreenSource Engineering	75
Henger	15
I-007eBooks.....	2, 101, 119
In-Circuit Design Pty Ltd	87
IPS	59
Insulectro	7, 27, 29, 31
Isola	13
Matrix.....	61
MicroCraft	25
MKS' Atotech.....	42, 43
MKS' ESI	45
On the Line with.....	3
PCBAA	63, 65
Pluritec	73
Polar Instruments	79
Precision PCB	83
Qnity	23
Schmoll America	69
Technica USA	35
United Electronics Corporation.....	93
US Circuit.....	109
Ventec International Group.....	67



Problems solved!

