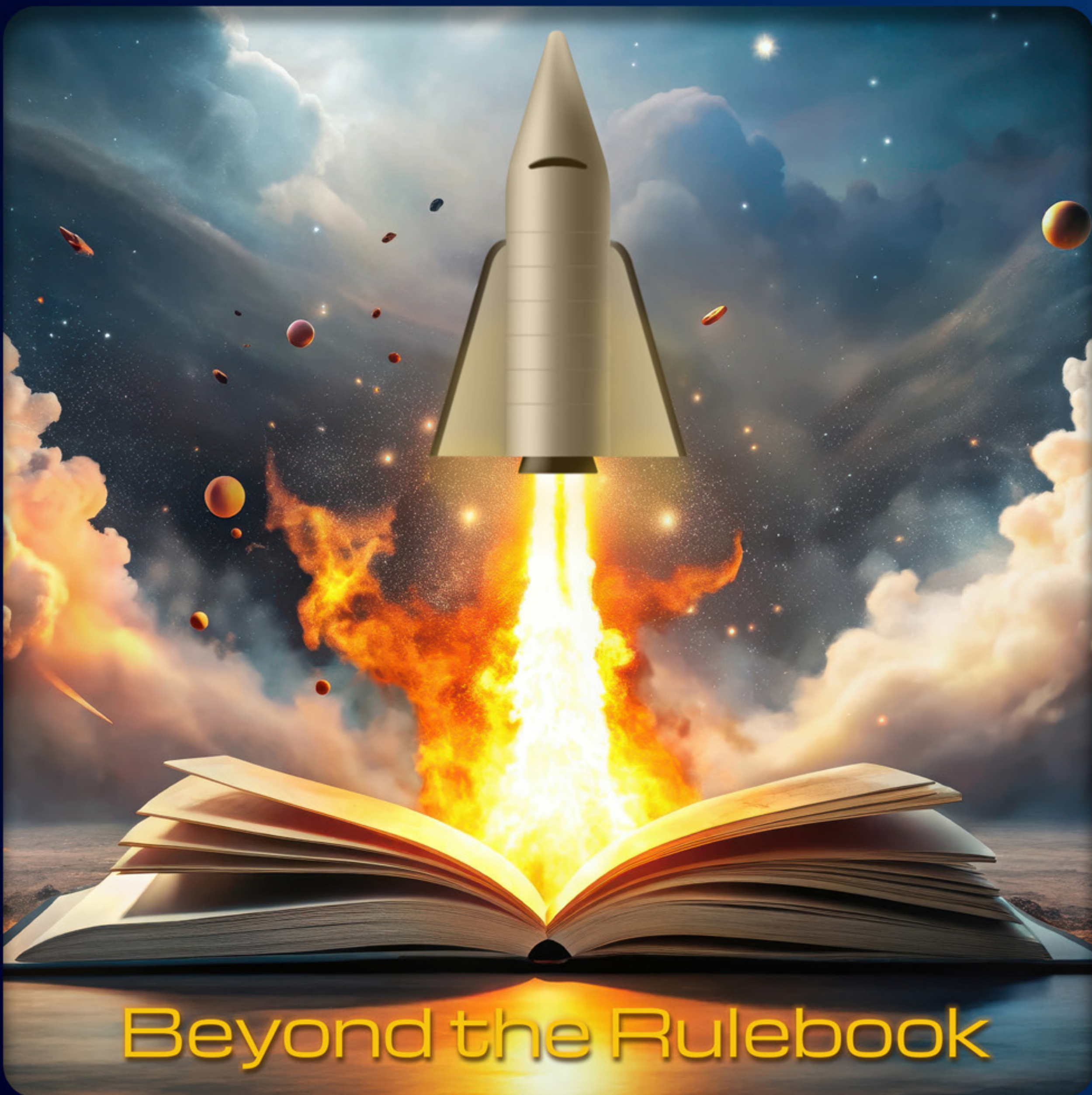
THE MAGAZINE

I-Connect007

INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY



PCB007
DESIGN007

APRIL 2026

Beyond the Rulebook

What happens when the rulebook is no longer useful, or worse, it was never written in the first place? Today's designers and fabricators are increasingly asked to design and build what has no precedent, no proven path, and no tidy checklist to follow. This is where "design for invention" begins.

THE MAGAZINE I-Connect007
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

PCB007

Features

Articles

Columns

LISTEN NOW!

I-Connect007 THE MAGAZINE
INTERCONNECTING THE GLOBAL ELECTRONICS INDUSTRY

Operating Without a Rulebook

BY MARCY LARONT, I-CONNECT007

What happens when the rulebook is no longer useful, or worse, has not yet been written? With electronics innovation happening at warp speed, we're increasingly asked to design and build what has no precedent, proven path, or tidy checklist to follow. "Design for invention" begins at the edge of known capability, where traditional DFM gives way to something far less certain, and far more exciting. It's not about breaking rules for the sake of it, it's about recognizing when the rules no longer apply, and having the insight, collaboration, and courage to move forward anyway.

In this month's *I-Connect007 Magazine*, we asked PCB designers, fabricators, and suppliers to discuss what it means to operate without a rulebook. Not surprisingly, there's a contrast in perspective, particularly between seasoned designers and experienced fabricators, but the common thread is that, to progress, you must push boundaries along an unclear path. This mindset has always been an undercurrent in our work.

From the design perspective, our contributors tackle this challenge head-on. Kelly Dack shares insights from a memorable conversation with an inventor-entrepreneur that helped shift his perspective. Kristin Moyer explores applications that demand entirely new approaches, which may include the use of new data formats altogether (model-based data structures instead of traditional layer-based) and the growing expectation that designers understand specialty materials. Kristin notes that design may no longer follow a rulebook at all, but rather a multidisciplinary decision tree guiding successful outcomes.

"It's not about breaking rules for the sake of it, it's about recognizing when the rules no longer apply, and having the insight, collaboration, and courage to move forward anyway."

John Watson emphasizes that "designing for X" is no longer a solo endeavor, highlighting the essential role of codesign. Stephen Chavez challenges readers to reconsider whether the very rules that once defined success may now be holding them back. In a special APEX EXPO interview, Vern Solberg points to the inevitable shift toward HDI, UHDI, and substrate design, something he is sure today's PCB designers will be taking on. Rounding out the section, Matt Stevenson examines advanced manufacturability through the lens of surface finishes, and the increased importance of flatness in surface finishes used in UHDI products. Anaya Vardya explores flex in advanced packaging, focusing on the need for standardization and current workforce challenges. Martyn Gaudion demonstrates a case study in crosshatch and how a little detective work made a big difference.

When it comes to operating without a rulebook in fabrication, materials take center stage. We feature a materials roundtable discussion with experts from Ventec, Four Peaks Innovation, and Nittobo, covering everything from ongoing glass shortages to the emergence of glassless materials for high-end applications. Innovation in materials is also front and center on Jiva's recyclable PCB substrate, a development that challenges long-held assumptions about what PCBs can be. At the same time, Mike Carano offers a more measured view, emphasizing that innovation doesn't eliminate the need for fundamental understanding. If anything, it demands a deeper grasp of core principles and closer collaboration between design and fabrication teams from the outset. Don Ball echoes this sentiment while pointing out a critical limitation: Standards simply cannot evolve quickly enough to keep pace with emerging technologies. His solution is a dedicated process development lab where new ideas can be tested without disrupting production.

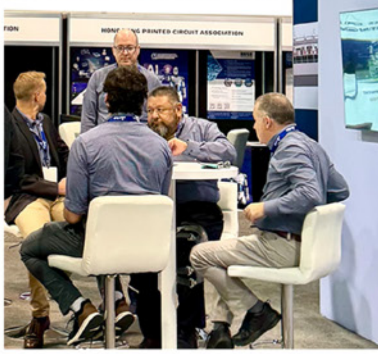
Elsewhere in this issue, Richard Nichols examines the realities of zero liquid discharge, while Steve Williams explores the implications of the DoD canceling MIL-PRF-31032. PCBAA raises concerns about the fragility of U.S. PCB manufacturing capacity amid the unprecedented drawdown of munitions since the onset of the Iran war. Manfred Huschka questions whether the China Plus One strategy is still a real phenomenon. Schmoll takes a deep dive into laser drilling technologies, and we celebrate Hall of Fame recipient Karen McConnell in Dan Feinberg's ongoing series.

It's another full issue, and one that proves you don't need a rulebook to move forward, only the willingness to rethink what's possible. **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.

APEX EXPO 2026 by Global Electronics Association

FOR MORE SHOW FLOOR PHOTOS
VISIT: REALTIMEWITH.COM




isola
emx
APEX EXPO 2026
by Global Electronics Association


ELECTRA
electrapolymers.com


MicroCraft
CraftPix


MATRIX
CIRCUIT BOARD MATERIALS
KB


AGC


Mach3
SAA
DYNACHEM


CHEMCUT

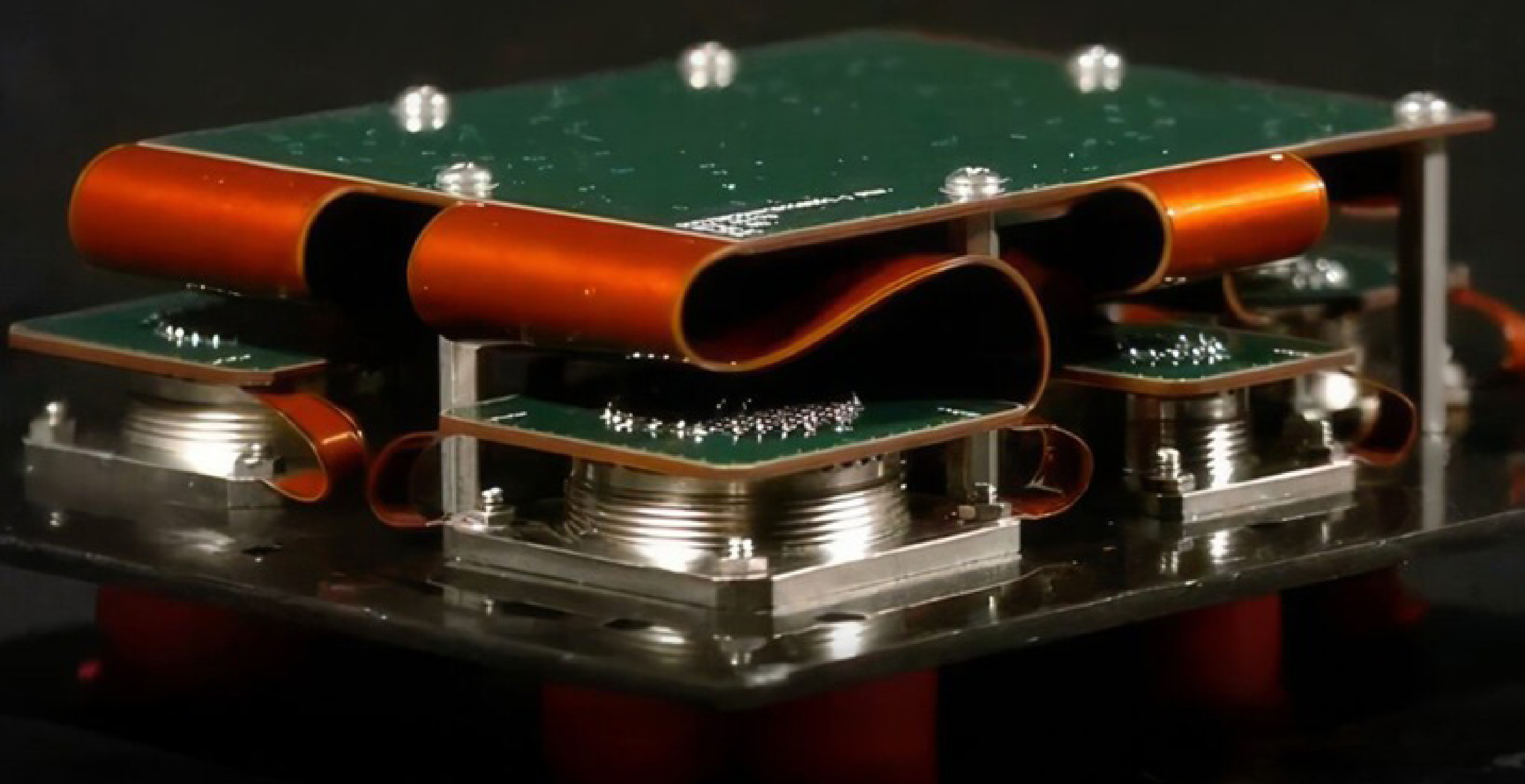

IPS
INTEGRATED PROCESS SYSTEMS
INSULECTRO
TECHNOLOGY VILLAGE


Eternal
X-VISION


all fle


INSULECTRO POWER CHATS
13.5-MIN
ORMET
InduBond

When DESIGN Outpaces MANUFACTURING



FEATURE COLUMN BY KELLY DACK, CIT CID+

Most PCB designers already understand why design for manufacturing (DFM) matters. If a product can't be built reliably, repeatedly, and at a price anyone will actually pay, it's dead on arrival. You may have the most elegant schematic and layout ever drawn in a CAD tool, but if the PCB suppliers can't make it, or the EMS sources refuse to quote it, you may well be considered more of a Nutty Professor or well-meaning inventor than a PCB designer.

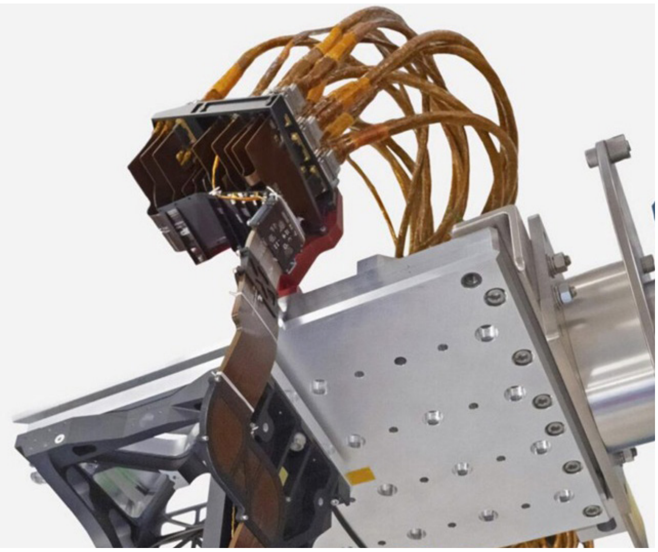
While designing for manufacturing makes sense, what about designing for invention? Maybe we should be talking about manufacturing for design (MFD), the idea that manufacturing sometimes has to evolve to keep up with the crazy ideas designers dream up. That tension of DFM vs. MFD isn't academic. It's exactly where innovation either takes off or crashes and burns as a PCB design project that devours the budget.

The Conversation That Sparked the Question

At IPC APEX EXPO 2025, I spoke with Keytronic Executive VP Chad Orebaugh about the moment

when an ambitious design collides head-first with the laws of manufacturing physics. DFM has been the rulebook for decades, and the premise is simple: if something can't be manufactured easily, it probably shouldn't be designed that way. Fair enough.

But at one point, our conversation took a hard left turn when Chad said something that seemed obvious once you heard it: We didn't land on the moon, and we certainly won't get to Mars, by designing within the limits of the machines, materials, and processing we already had. At some point, manufacturing had to catch up with the idea.



Figures 1 and 2: Mars Perseverance Rover utilizing extended length flex assemblies. (Source: Pioneer Circuits)

That's where MFD happens. Invention has an annoying habit of demanding materials that don't exist, requiring processes nobody has figured out yet, and asking machines to do things they've never done before. Innovation often leaps into advancement: Designers specify something just beyond current capability, and, if they want the business, manufacturers figure out how to make it happen. Sometimes that works, and sometimes the manufacturer laughs and hangs up.

Stop Saying 'You Can't Do That'

For decades, the PCB industry has been very good at telling designers what we can't do. DFM is essentially a long list of guardrails meant to keep designers from driving their layouts straight off a cliff: Too small. Too thin. Too weird. Too expensive, made of "unobtainium."

Those guardrails are useful, but MFD allows us to ask a different question: What if manufacturing's job wasn't just to enforce limits, but to remove them? The goal becomes providing designers with what you might call extreme design capability, meaning the shape, material stack, and architecture of an electronic system aren't dictated entirely by the constraints of traditional fabrication and assembly processes.

This isn't as far-fetched as it sounds. Consider the early prototype of the Apple-1 computer, hand-wired by Steve Wozniak in his apartment 50 years ago. At that stage, scalability was the least of anyone's concerns. The goal was simply to make the idea work. Only later did manufacturing figure out how to produce it. In some ways, modern prototyping tools are bringing us back to that mindset.

The Desktop Invention Lab

Today's designers have tools that would have looked like science fiction 50 years ago, including advanced CAD tools, powerful simulation engines, affordable additive manufacturing, and, increasingly, 3D-printed electronics.

Designers are experimenting with curvilinear circuit geometries, embedded components, flexible substrates, and structures that look less like traditional PCBs and more like something that escaped

Figure 3: 3D printer/printed part with circuitry.

from a mechanical engineering lab. The result is a new era of desktop-driven invention.

This is great news for designers, and slightly terrifying for manufacturers. Supporting those ideas may require entirely new processes, materials, and equipment. Manufacturers are investing heavily in Industry 4.0 technologies, driven by automation, digital process monitoring, and highly adaptable production lines. Fabrication techniques are evolving as well. Processes such as modified semi-additive processing (mSAP) and materials sputtering allow far finer conductor geometries than traditional subtractive etching ever could. Suddenly, trace widths that once sounded ridiculous are becoming routine, which is exactly how innovation sneaks forward.

The Material Problem

If manufacturing is the engine of electronics, materials are the fuel, but sometimes the fuel simply isn't available yet. Forward-thinking manufacturers are beginning to treat materials like strategic assets. Some are stocking advanced laminates—ultra-low-loss PTFE and ceramic-filled systems from Arlon, EMC, Isola, Qnity and Rogers—long before designers start asking for them.

That strategy isn't cheap. But when the next wave of high-frequency designs arrives, those suppliers aim to be ready.

Regulations are also reshaping the materials landscape. Environmental rules such as the Restriction of Hazardous Substances (RoHS) Directive and Waste Electrical and Electronic Equipment Directive are pushing manufacturers to explore recyclable and biodegradable substrates.

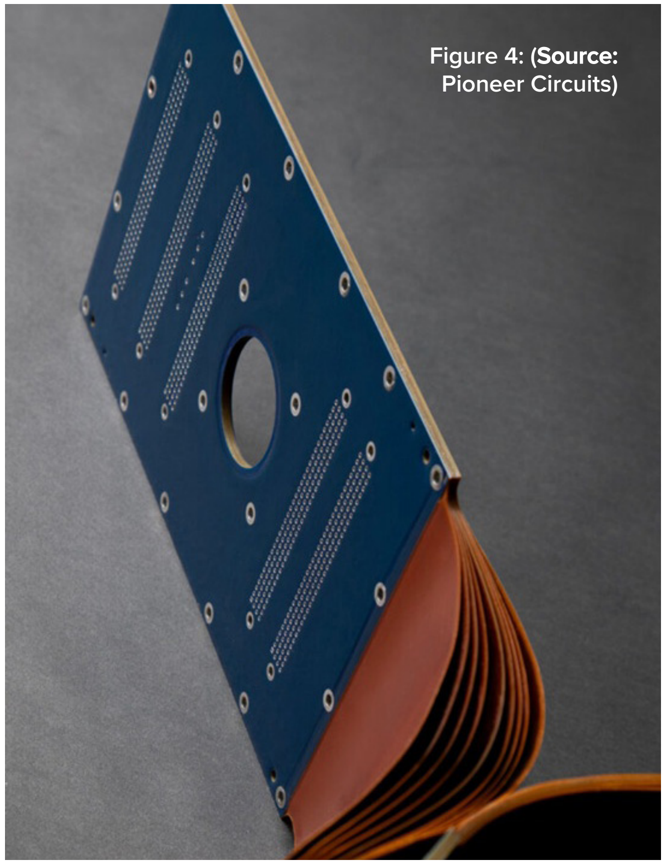


Figure 4: **(Source: Pioneer Circuits)**

At the same time, AI and EV applications are generating enormous thermal loads. Fabricators are responding with metal-core boards and new thermal management structures designed to dissipate heat far more effectively. Materials, in other words, are quietly becoming one of the biggest battlegrounds in electronics manufacturing.

Modeling the Impossible

Design and manufacturing are also converging in simulation. Many engineers prove their designs mathematically before hardware even exists. Electrical, thermal, and mechanical simulations can predict performance long before a prototype hits a fabrication line.

If manufacturers get access to those models early enough, they can create a powerful digital twin, essentially a virtual copy of both the product and the process required to build it. Manufacturing engineers can run simulated production scenarios, identify bottlenecks, and test process limits before anyone spends money on laminate and copper.

In other words, we can fail faster and far cheaper. Occasionally, we discover that the "impossible" design isn't impossible after all.

When DFM Isn't Enough

DFM still matters. Many PCB projects still go sideways because of poor documentation, unrealistic tolerances, or material selections that have no connection to the real supply chain. Standards developed by organizations like the Global Electronics Association exist for a reason. Good design still means choosing tolerances that manufacturing can hit, materials that actually exist, and processes that won't cause your fabricator to quietly blacklist you.

But there's another idea that needs more discussion: Sometimes the infrastructure needed to build the idea simply doesn't exist. When that happens, innovation stalls. Manufacturers might admire the idea, but still decline the job. If you've ever watched "Shark Tank," you've heard the phrase: "Great concept! But for that reason, I'm out."

A Lesson From the Space Race

History provides a useful reminder. The technological surge driven by NASA during the space race accelerated countless technologies we now take for granted: integrated circuits, satellite communications, GPS navigation, advanced materials, miniaturized sensors, and entire disciplines like systems engineering.

But getting those technologies into everyday products wasn't easy. Many early innovations were classified, expensive, or dependent on specialized manufacturing techniques that didn't exist outside government labs. Eventually, those ideas filtered into the commercial world, but it took time, translation, and a lot of stubborn engineers.

The Ecosystem Problem

Innovation doesn't happen in isolation; it happens in ecosystems. Government programs fund risky research. Entrepreneurs identify real-world opportunities. Manufacturers build the infrastructure needed to turn ideas into products. When those three forces line up, industries are born. When they don't, the idea goes back in the drawer.

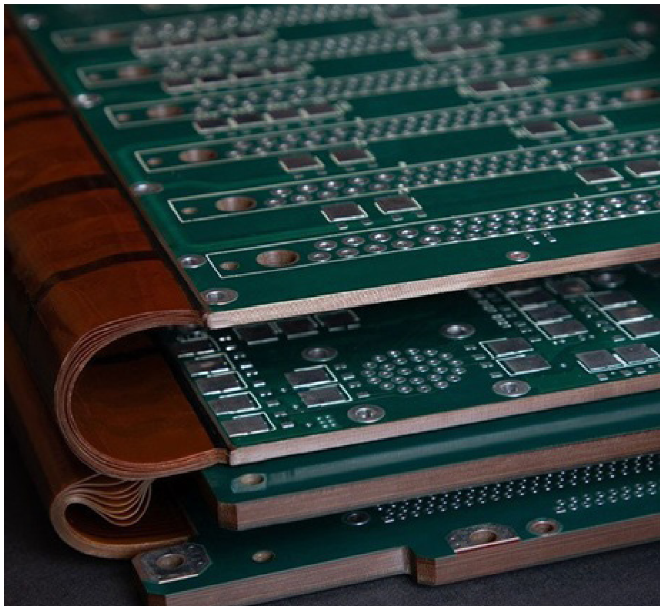


Figure 5: GPS III satellite rigid-flex assemblies. (Source: Pioneer Circuits)

Where Does That Leave Us?

The future doesn't belong exclusively to DFM or MFD; we need both. DFM keeps designers from doing ridiculous things that manufacturing can't support. MFD pushes manufacturing to evolve so that tomorrow's ridiculous ideas eventually become the norm.

Bad design ignores reality. Good design understands the ecosystem it lives in and pushes it just far enough to make progress. Bad manufacturing falls behind design trends. Good manufacturing anticipates where those trends are headed.

Innovation rarely happens by staying safely inside today's limits. It happens when someone designs something that shouldn't work, and manufacturing steps in to prove them wrong.

Kelly Dack, CIT CID+, specializes in DFx-driven PCB design and applications engineering at Pioneer Circuits, Inc., a global leader in high-reliability flex and rigid-flex printed wiring boards for defense, aviation, near-earth orbit and deep-space exploration systems.

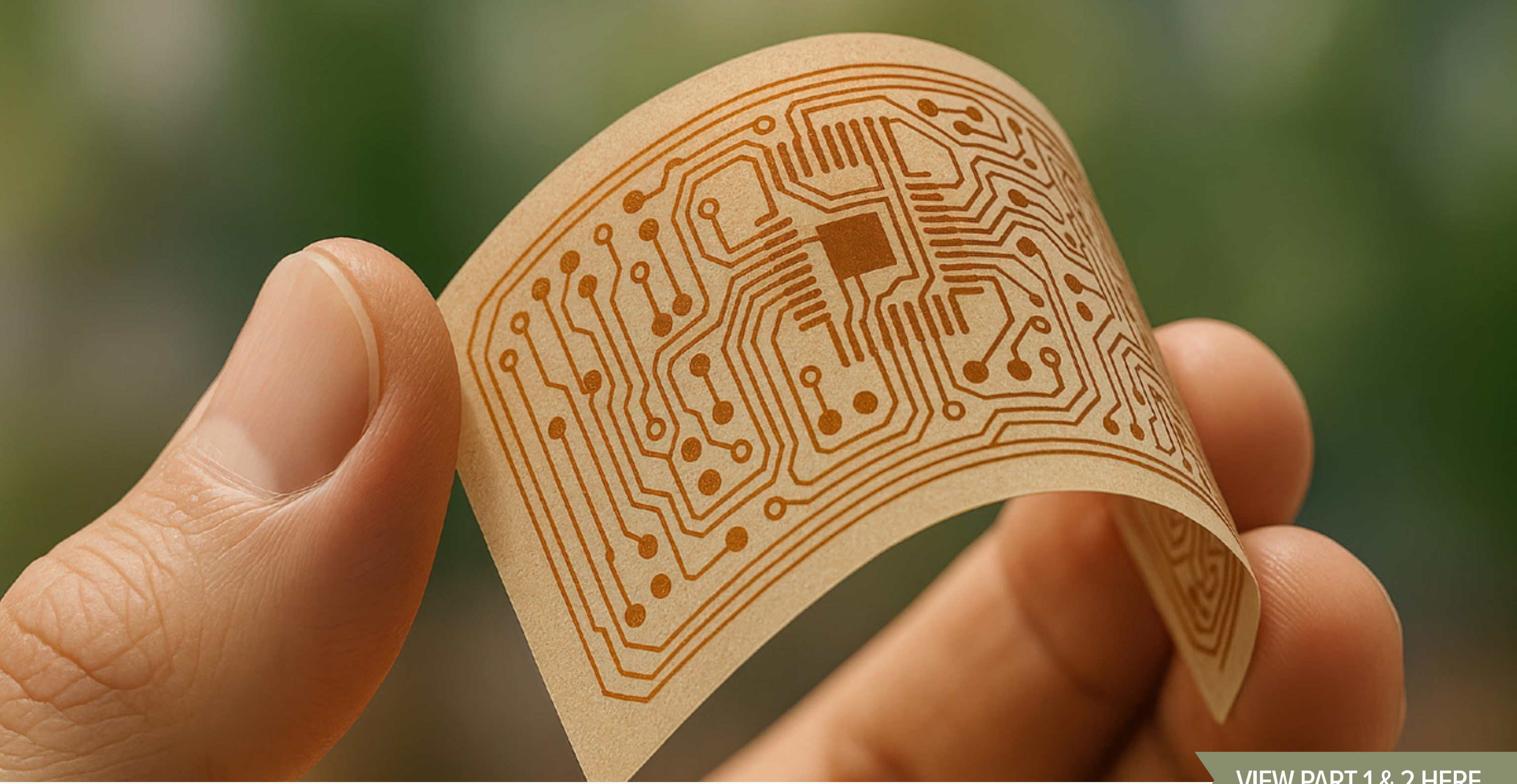
To read past columns, click here.

ARTICLE

BY ANAYA VARDYA, AMERICAN STANDARD CIRCUITS

Standardization, Workforce, and the Road Ahead for FLEX–PACKAGING INTEGRATION

PART 3



VIEW PART 1 & 2 HERE

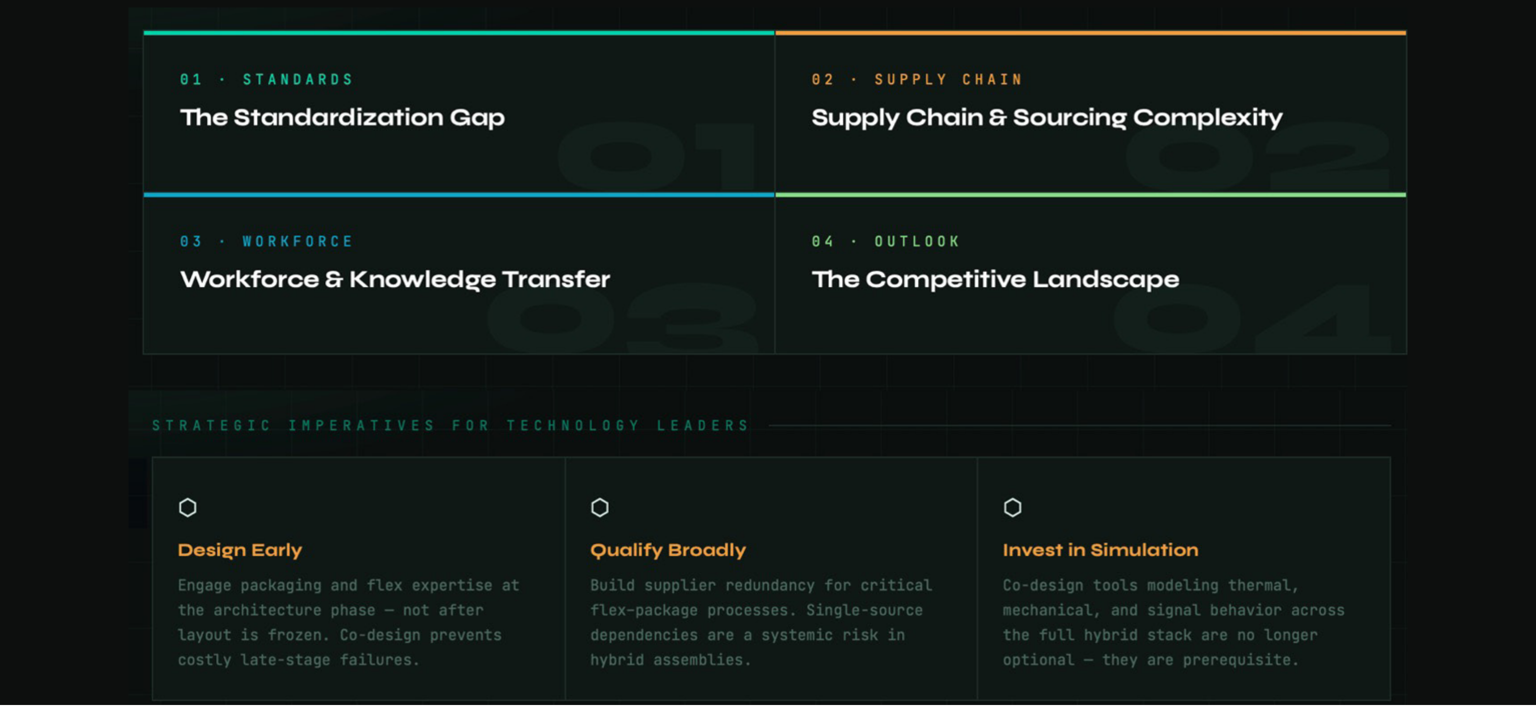
Parts 1 and 2 of this series established the technical foundation and application landscape for the convergence of flexible PCBs and advanced semiconductor packaging. Part 3 addresses what comes next: the standards frameworks, talent pipelines, and strategic imperatives that will determine whether the industry can scale this convergence reliably and competitively.

The Standardization Gap

Despite rapid technological progress, standards governing the flex–packaging interface remain fragmented. IPC, JEDEC, and IECEx each govern portions of the design and qualification space, but no unified framework addresses hybrid flex-package assemblies end to end. This creates costly inconsistencies across supply chains and slows customer qualification cycles. Industry working groups are beginning to bridge these gaps, but meaningful harmonization will require active participation from OEMs, EMS providers, and substrate manufacturers alike.

Supply Chain and Sourcing Considerations

Flex–packaging integration introduces unique sourcing complexity. Key considerations include:



- Dual-qualified suppliers for both flex substrates and advanced package assembly
- Traceability requirements that span the chip-to-flex boundary
- Geopolitical exposure across specialized material and tooling sources

Workforce and Knowledge Transfer

The talent gap is one of the most underappreciated obstacles to scaling flex-packaging integration. Traditional PCB engineers and semiconductor packaging specialists have historically operated in separate disciplines with distinct vocabularies, tools, and process assumptions. Bridging this divide requires targeted cross-training programs, updated university curricula, and industry-led certification pathways. Organizations that invest in hybrid expertise now will hold a significant competitive advantage as demand accelerates.

Strategic Imperatives for Technology Leaders

For engineering and business leaders, the convergence of flex and advanced packaging demands a proactive strategy across several dimensions:

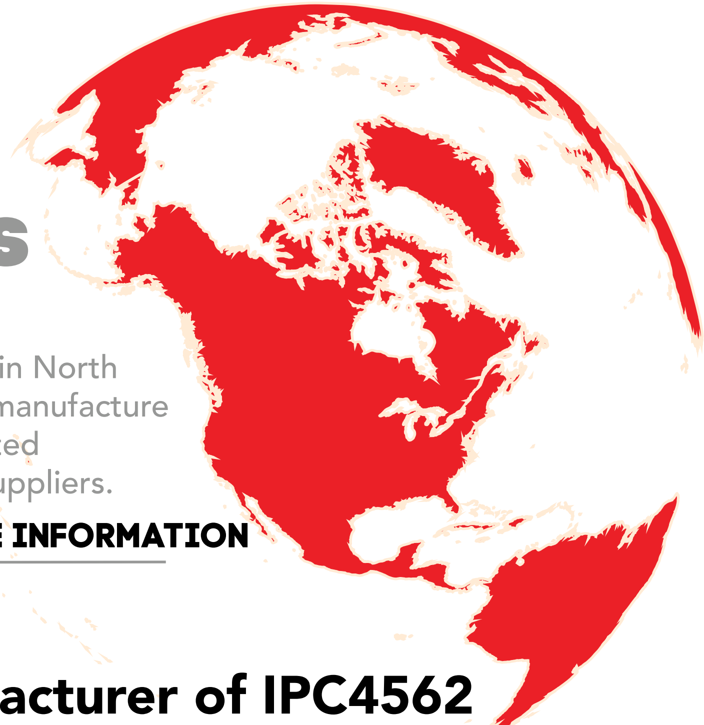
- **Design early:** Engage packaging and flex expertise at the architecture phase, not after layout is frozen
- **Qualify broadly:** Build supplier redundancy for critical flex-package processes
- **Invest in simulation:** Co-design tools that model thermal, mechanical, and signal behavior across the full hybrid stack are no longer optional

Conclusion

The convergence of flexible PCBs and advanced packaging is not a distant prospect. It is unfolding across automotive, medical, consumer, and defense platforms today. What remains unfinished is the infrastructure around it: the standards, the trained workforce, and the strategic frameworks that will allow this convergence to scale. The organizations that close those gaps first will define the next decade of electronics innovation. **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*.

INSULECTRO



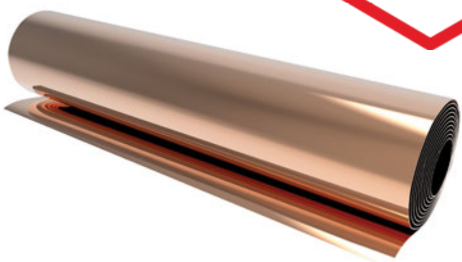
WORLD-CLASS SUPPLIERS

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

CALL 949.587.3200 FOR MORE INFORMATION

CCP

Manufacturer of IPC4562 B Grade HTE Copper Foils



CCP is a leader in the manufacture of high-quality electrodeposited copper for the printed circuit board industry. This privately held and Taiwanese owned has been supplying foil since 1987. They produce a full range of grade 3 HTE foils in both matte side and shiny side (RTF) treatments for high performance markets covering HSD and RF with foil profile options as smooth as <1 micron.

PACOTHANE TECHNOLOGIES

Conformal Release Film

MADE IN USA

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

FOCUStech™

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

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

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

Precision is our specialty!

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

We invited the following group of companies to ‘Take the Mic!’ at APEX EXPO 2026 to talk about why their work truly matters. These short, editorial-style interviews were held in the participants’ booths and are designed to highlight voices we believe are shaping what’s next for the electronics industry.

AGC’s Advanced PCB Material Solutions

AGC Multi Materials America’s line includes advanced PCB materials for critical industries like aerospace, defense, and medical. This interview with George Pinzon and Jim Pavelek highlights AGC’s commitment to North American sourcing and solutions to today’s challenges, along with specialized automotive materials such as fastRise, a low-loss prepreg for high-frequency applications such as 77 GHz radar.

Notion Systems Inkjet Solder Mask

Marcy LaRont interviews Celia Wenzler to explore Notion Systems’ innovative inkjet solder mask technology at APEX EXPO 2026. This green, clean, and precise solution offers rapid color changes and high-precision inkjet systems for various production needs. Discover how their expandable, platform-based machines enhance throughput and efficiency in PCB manufacturing.

Integrated Thin-film Resistive Foil Technology

John Andresakis of Ohmega Ticer shares his advanced solution for next-generation PCBs, enabling miniaturization, enhanced electrical performance, and stable resistance. This interview explores how rising demand for high-frequency phased-array antennas in commercial space and military sectors is driving the need for superior RF performance and simplified beam forming.

Photo Chemical Systems: 50 Years Strong

Photo Chemical Systems is celebrating 50 years in the bare board PCB market. In this interview, David Graves and Jason Averette discuss their expansion into assembly, how strong relationships and a customer-centric approach help navigate supply chain challenges, and the innovative solutions and growth strategies, including AI integration and new market ventures, driving their continued success.

Global Expansion Insights

Join Dan Beaulieu and Jimmy Fang, VP of global development at Sunshine Global Circuits (SGC), in this booth interview. SGC has extensive global operations, including its new expansion in Malaysia. Fang outlines his company’s commitment to high-tech PCB manufacturing and shares insights into the company’s capabilities, market focus, and future growth strategies.

Highlighting Karen McConnell

Editor's note: Dan Feinberg continues his series on the Hall of Fame, spotlighting the achievements of past Hall of Fame members of the Global Electronics Association.

Many members who have contributed significantly to the Global Electronics Association (formerly IPC) and our industry have been awarded the Raymond E. Pritchard Hall of Fame (HOF) Award. Though many early HOF members have passed away and are not known to today's Association membership, their contributions still resonate. This special series on Hall of Fame members provides a reminder of who was honored and why.

This Hall of Fame Spotlight Features Karen McConnell

In 2021, Karen McConnell was awarded the Raymond E. Pritchard Hall of Fame award in recognition of her contributions to the Association and the electronics industry. As a senior staff member and CAD/CAM engineer at Northrop Grumman Enterprise Services, her primary responsibility was to develop a common, shared EDM (Electronic Document Management) library to support the electrical and PCB design tool initiatives across Northrop Grumman Mission Systems.

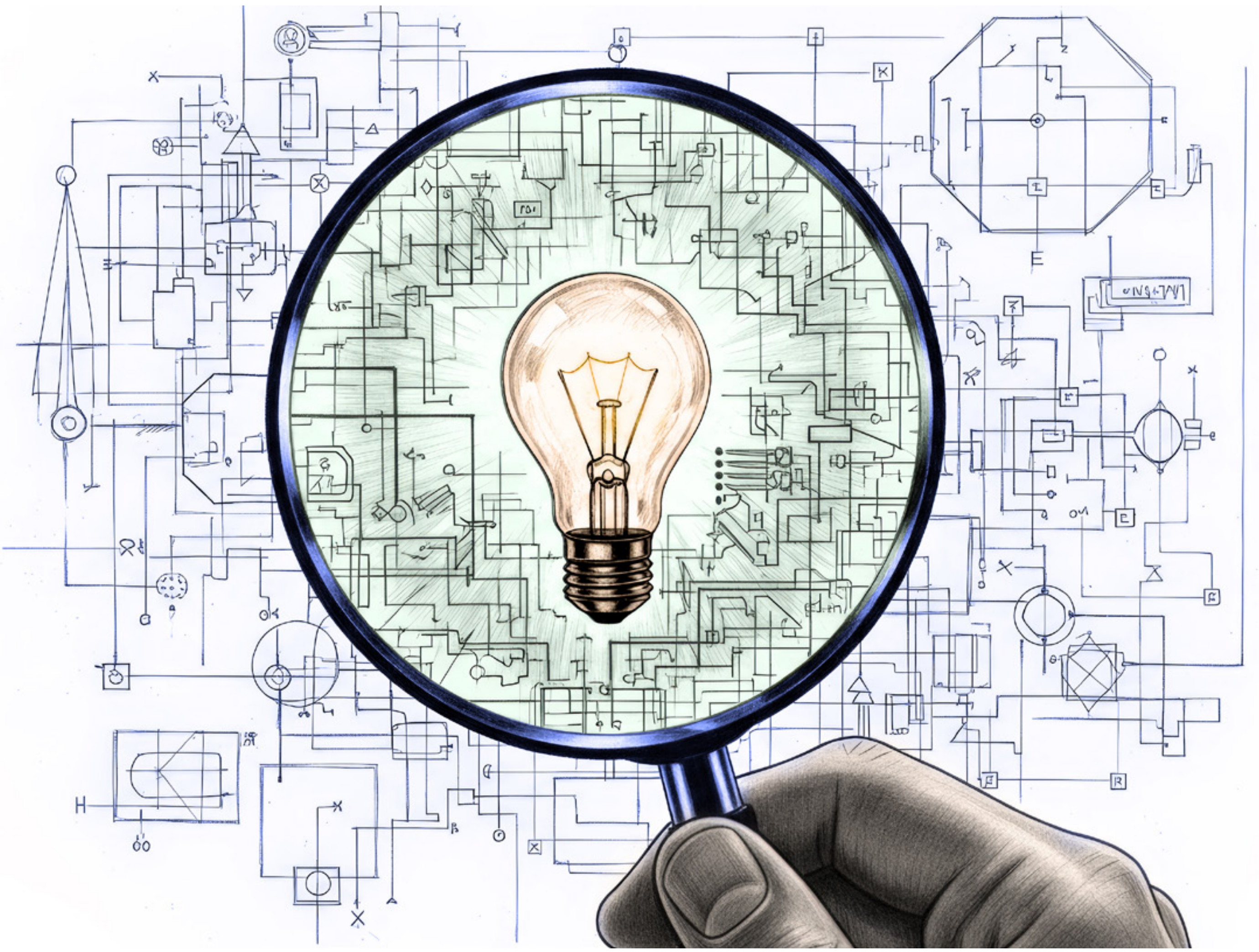
An active Global Electronics Association member and volunteer for over 30 years, Karen first got involved while working for Lockheed Martin, when one of her bosses presented at a meeting. She told Patty Goldman in a 2021 interview upon hearing of her HOF honor that she became engaged during "the GenCAM/ODB++ data wars of the 1990s," and that "IPC decided to host the peace talks." She recounted working closely with Dieter Bergman on the 2-10 and 2-16 committees. Lockheed Martin was very involved in standards creation and review. Karen recalled that it was a high priority, and feels fortunate that her current employer, Northrup Grumman, has remained supportive of her involvement.

Among her list of Association achievements, Karen received the President's Award in 2013. At one time, she was the chair of nine committees, including the Technical Activities Executive Committee, the Technical Activities Executive Global Committee, and the IPC Design Community Leadership team. She has been a staunch advocate for new engineers, serving as one of the first mentors for the Association's Emerging Engineer program.

When she was awarded her Hall of Fame trophy in 2021, John Mitchell said, "Karen's contributions to IPC are extraordinary, and her passion for the industry and commitment to the growth of younger engineers moved them, as well as IPC (the Global Electronics Association) and the overall electronics industry, forward". About her work with the Association, she has said, "Everybody I have met at IPC cares about the industry, cares that things are done correctly, and cares that it costs their company less." **I-CONNECT007**

CAUGHT IN THE CROSSHATCH

A Cautionary Tale of Detective Work



BY MARTYN GAUDION, POLAR INSTRUMENTS LTD

A chance meeting at a family wedding the other week led to a conversation about numbers, an introduction to a book entitled *Humble Pi,* and how numeric misinterpretation can lead to all kinds of unexpected outcomes, some just costly, others tragic. It's a good and amusing read, and as a result of this conversation with someone I had previously never met, I feel somewhat (at least temporarily) enlightened. One of the takeaways of the book is that humans are born to think logarithmically, and linear math has to be formally educated into our brains. That got me curious for more.

Back in the World of Work

The Monday morning after that chance meeting, an email landed in my inbox from a colleague in our Singapore office: "One of our OEM customers is getting very different results when compared with his supplier, despite them both using identical Polar software. What could cause that?" As is often the case, customer confidentiality meant that


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

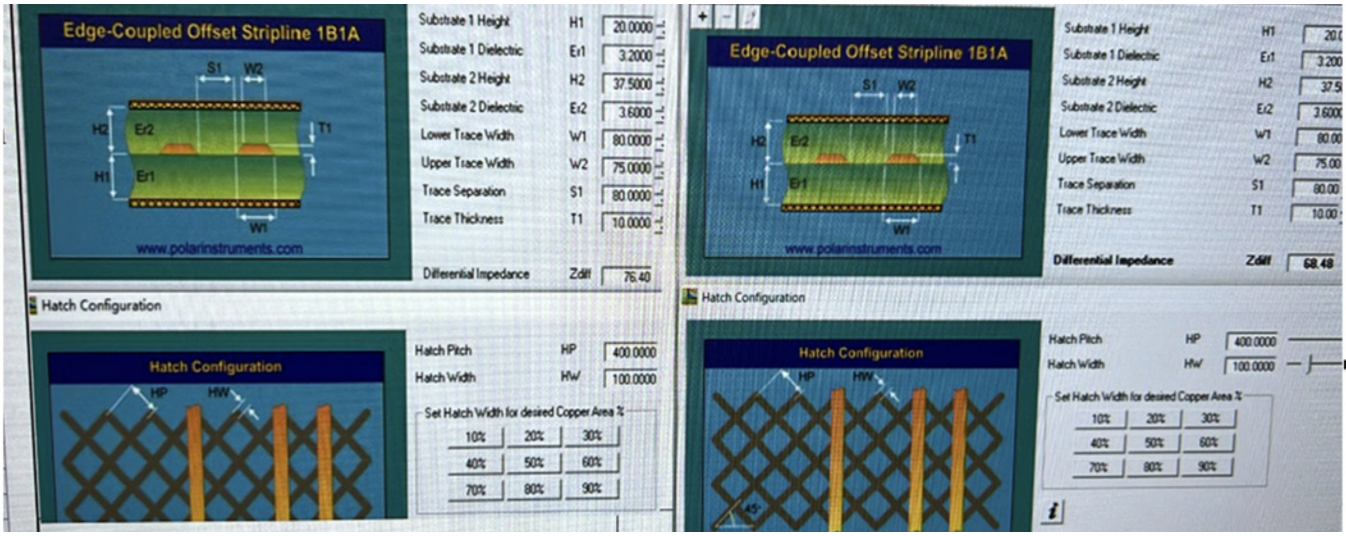


Figure 1: Two software instances side by side in an Excel sheet (from a mobile phone picture).

limited information was available, and all I had to work with was a somewhat redacted mobile phone picture of the two software instances side by side in an Excel sheet (Figure 1).

These scenarios usually lead to a degree of detective work to isolate the root cause of the discrepancy. So, I set about manually entering the data and checking on my own version of the field solver. The scenario involved modelling and using crosshatched (mesh) ground planes. I thought it would be good to revisit the rationale for using mesh planes and the dimensional (and other) considerations when deploying them.

Crosshatch Planes Revisited

From a signal integrity purist's point of view, crosshatched planes are non-ideal: A full copper plane provides a consistent return path. However, PCB engineering often involves a compromise between the mechanical and electrical worlds, especially with flex circuits. Meshing/crosshatching is extremely useful on flex as it keeps the flex, well, flexible. It's no use having perfect signal integrity if the flex becomes brittle and fails owing to forcing solid copper plains in a stripline to act as an "I" beam, making the flex "inflexible." Here, the use of crosshatch is vital if the flex circuit is to survive its design lifetime of flexing. This is clearly more important in flex, such as with hinged and moving applications, rather than flex-to-fit, which may only be flexed a few times during assembly and repair.

Crosshatch is also useful on thinner stackups, flexible or not, to achieve a given impedance whilst keeping the line width manufacturable. A meshed return plane will require a wider trace for a given impedance. This can be helpful when a fabricator is bouncing on the lower limits of process capability.

Considerations

It's important to keep the mesh as small as possible. You can make large apertures in the mesh, but this will lead to EMI problems and, in extreme cases, cause impedance to vary along the line. Think of fiber weave effects on steroids. But if kept small in relation to the wavelength, the crosshatch is a viable and practical way to keep things flexible.

Single-ended and differential?: Interestingly, crosshatch has a greater effect on single-ended transmission lines than differential because in differential pairs (depending on the coupling percentage), much of the return current flows in the complementary side of the pair, so the closer the coupling, the less the effect of the hatch.

Alignment: Often, you will see that guidance is to align over the hatch at 45 degrees and over the center of the crosses. However, this is not always possible. Often, the hatch is specified to the fabricator and added at the CAM stage rather than in CAD. It's best to know that much of the crosshatching in flex is added at the fabrication stage—that might be changing, and if you think otherwise, let me know. At Polar, we get requests to add crosshatch to flex impedance coupons in the CGen

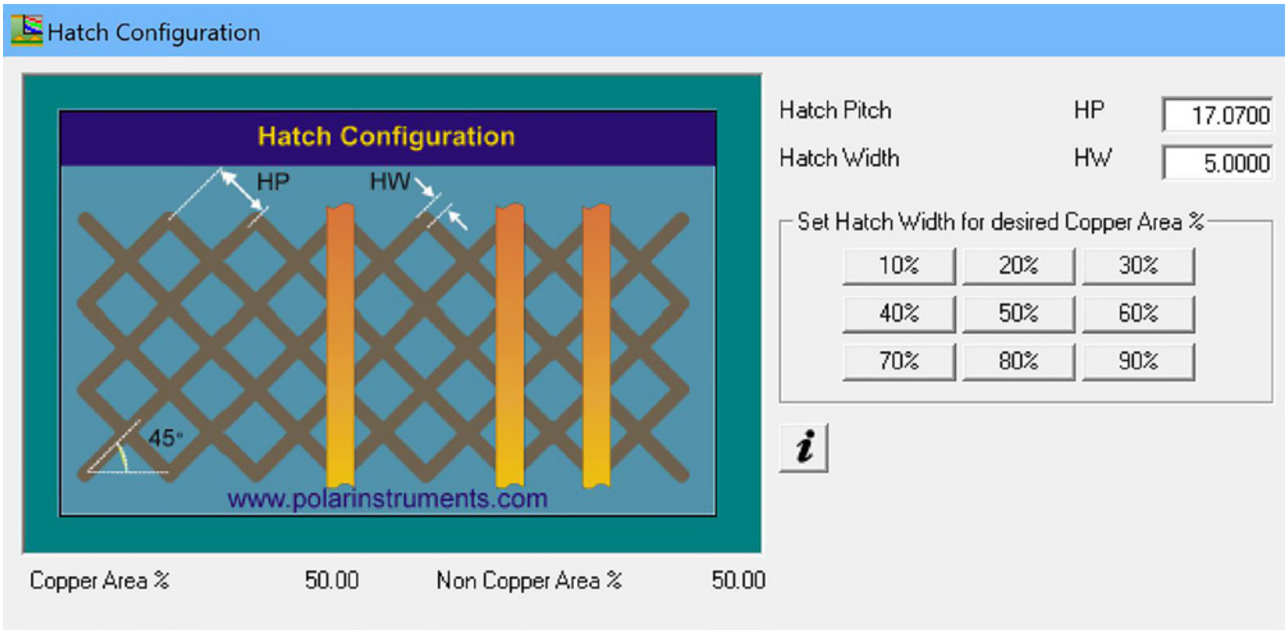


Figure 2: Figure alignment is the ideal situation, but not always possible.

coupon generator, but for that reason, we haven't done it, as it might mean the hatch on the coupon hatch is created differently from the hatch under the circuit itself.

How Did the Variation Arise?

I started by noting that a customer reported different impedance results from the same modelled data than from their supplier, so I took a long, hard look to see whether units were mixed or there had been some form of input error. But as hard as I tried, I could not replicate the discrepancy. My numbers only aligned with one of the predictions sent to me. Then, as with all detective work, a minor point struck me that I had missed on first look: One of the screenshots had a "+" and a "–" in the top left of the structure image (Figure 3).

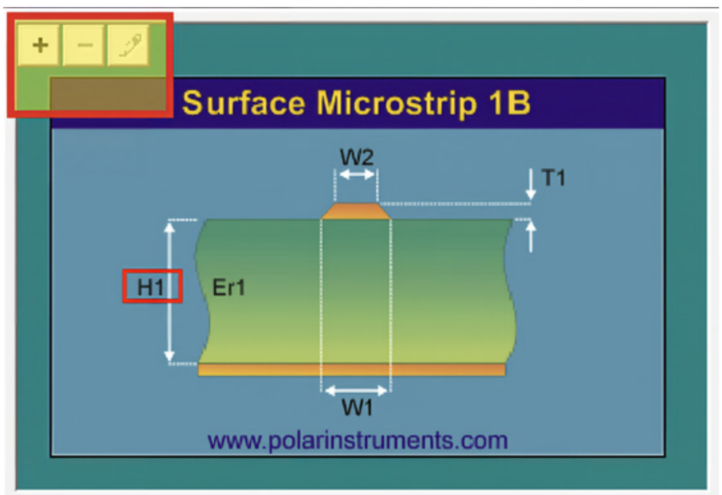


Figure 3: Detective work led me to find a plus and minus sign in the top left.

The other image from the OEM's supplier did not contain that control. A look in the archives showed me that this change in our software dated one version at a release (at least 11 years older than the latest, maybe 15), which was in the early days of crosshatch modeling for Polar. Our tools are regularly updated and fine-tuned, and so I find myself now in a further search to see what has changed and what might have improved since those early days. A lesson, perhaps, in making sure that software is regularly updated, especially when working on critical designs.

So, as with many investigations, the original supposition turned out to be wrong, but it did point to further analysis and triggered the search for the cause of the difference. **I-CONNECT007**

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

ARTICLE

BY NOLAN JOHNSON, I-CONNECT007

Volatile Metals Market Creates PCB Pricing Headache

Market volatility for precious metals is very real. Financial organizations have reported elevated volatility, with record highs and steep corrections; in 2025 alone, gold has increased by over 60%, silver over 120%, and copper over 35%. Each is a critical raw material used in electronics manufacturing, where pricing is already fraught for business owners and their customers due to tariff uncertainty and a critical supply chain that resides mostly in China. The volatility of precious metals markets adds yet another layer of complexity for manufacturers, pushing up raw material costs.

Though it may seem obvious, analysts concur that the volatility is driven in large part by global macroeconomic data, shifting U.S. policy, and inflation numbers. Traders are rotating between risk assets (stocks, etc.) and safe havens (metals and, to a lesser degree, bonds). When investors make a

INNOVATIVE PRECISION FOR YOUR INNOVATIVE PROJECTS

FLEXIBLE CIRCUIT Technologies | inTFlex

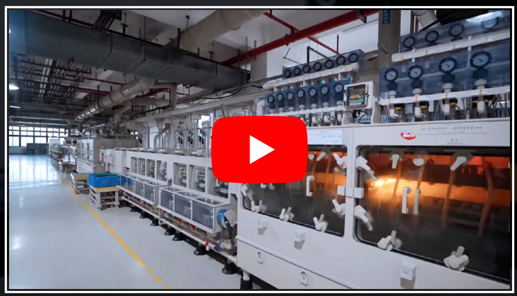
Industry-leading flex design services, premier in-house manufacturing of flexible circuits and EMS services!

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

Metal	Where it appears	Share of PCB fab cost
Copper	Foil, plating, vias, heavy copper layers	15–30%
Gold	ENIG, hard gold, edge connectors	3–10%
Silver	Immersion silver, specialty pastes	1–5%

Figure 1: Typical cost exposure by metal (fabrication only).

Metal	Recent Spot Range	60-day Move	Volatility
Silver	~US$26–34/oz	~±25–30%	Extremely high
Copper	~US$3.70–4.35/lb	~±15–18%	High
Gold	~US$2,200–2,450/oz	~±10–12%	Above average

Figure 2: Recent price behavior (rounded ranges).

run on metals and prices spike, other investors respond to those run-ups with a profit-taking sell-off, triggering a correction.

An industry-wide average puts silver at 3–8% of printed circuit board fabrication costs, gold at 5–15%, and copper, of course, leading the way at 15–25% of the total fabrication cost. An increase in copper prices of 20% over a year can trigger an increase in PCB fabrication costs of 3–5%, which may be the full profit margin for a company building production-level quantities of PCBs.

Looking at data from COMEX and the London Metal Exchange, the following table captures just how volatile prices have been in the first quarter of 2026. Gold's volatility is mostly driven by investors, while silver is experiencing swings characterized by analysts as "speculative-grade volatility." Copper may sit between the two, but copper's volatility tends to come from high industrial demand countered by supply risks and tight inventory.

So, how does this look when translated to a specific work order? By running some research tools, we estimated the impact of current metal costs on the cost per piece for three common board configurations: standard ENIG, heavy copper, and hard gold.

Scenario A: Standard 8-layer ENIG board

Price Changes

- Copper +15%
- Gold +10%

Net effect: ~+2.0–2.8% fab cost increase and increasingly quoted as a line-item surcharge, not absorbed

Scenario B: Heavy copper/power electronics board

Price Changes

- Copper +20%
- No gold (OSP finish)

Net effect: ~+3.5–5.0% fab cost increase and high risk of mid-quarter repricing

Scenario C: High-reliability connector board (hard gold)

Price Changes:

- Copper +10%
- Gold +25%

Net effect: ~+2.0–3.0% fab cost increase, driven by gold thickness specs

The PCB industry already runs on slim profit margins. In our ENIG example above, a net margin of 11% ±3 % would be typical. Absorbing 2–5% on costs will, for some jobs, represent an unsustainable drop in net margin.

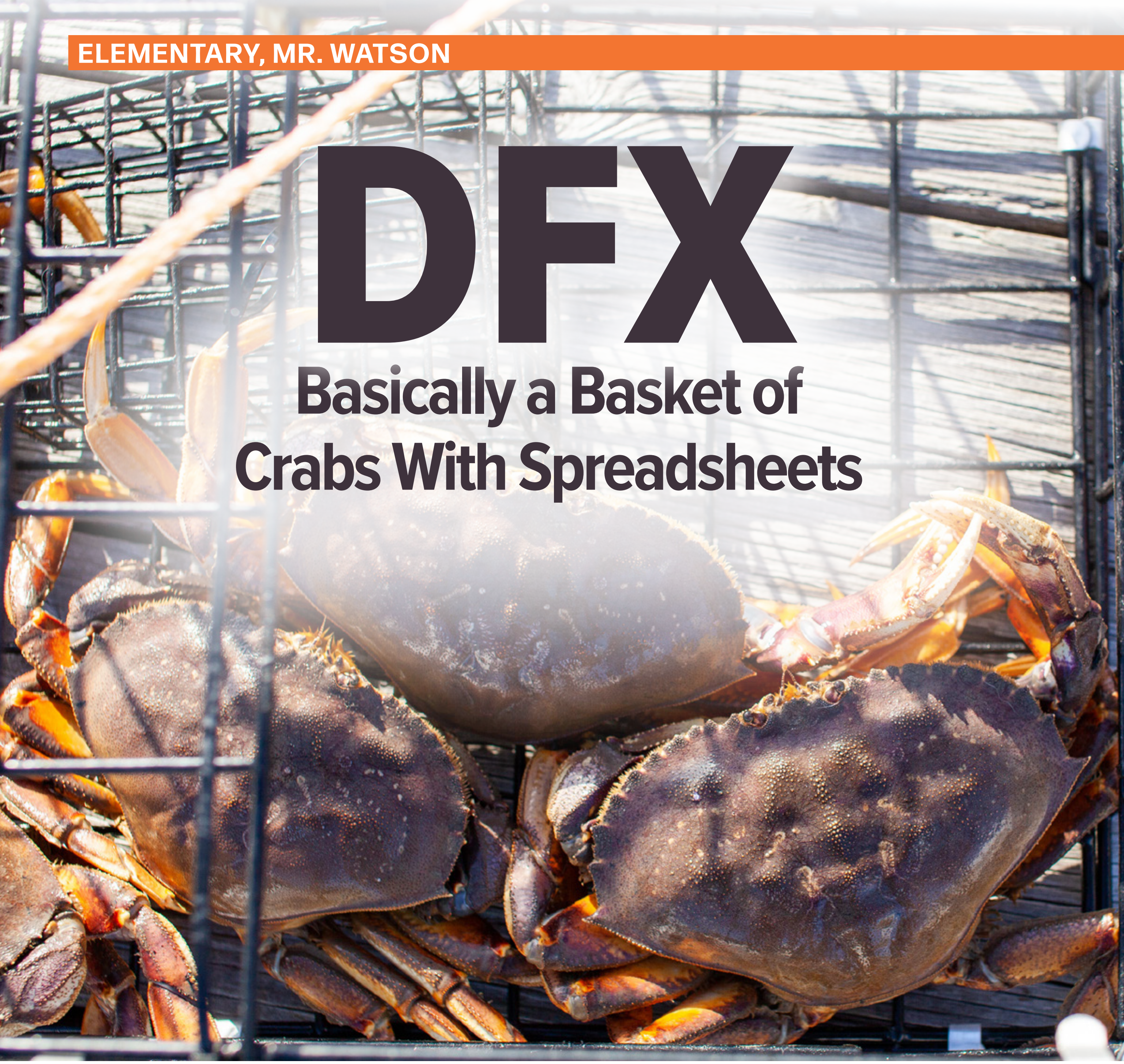
Understandably, PCB manufacturing can only absorb this for so long before changing their pricing tactics, such as adding copper surcharges tied perhaps to an exchange average like the London Metal Exchange, encouraging more affordable finishes when the board specifications allow, shortening the window for honoring quotes on metal prices, or even separating the base board quote from the metals quote.

What can we expect in the next 24 months? An article by Crux Investors takes a somewhat bearish stand on copper futures, citing Goldman Sachs when it makes the case that copper prices may drop. "In January 2026, Goldman Sachs published a revised outlook suggesting that copper could retrace toward $11,000 per tonne by December 2026, representing an 18% correction from recent highs. This is not framed as a bearish call on copper's long-term relevance, but rather as a normalization thesis following an extended period of sentiment-driven price appreciation. Margin sensitivity and balance-sheet strength are likely to reassert their importance as the macro tide recedes."

The mixed messages from analysts about copper futures mirror overall market volatility. What we know for sure is that copper demand is strong enough—and the inventory tight enough—that volatility may well continue through 2027, with wide oscillations, as market dynamics show no sign of easing. **I-CONNECT007**

Resources

1. "Why Is Silver Falling with Gold? Silver Price Crashes 3rd Hardest in 6 Years," Finance Magnates, Feb. 13, 2026.
2. "Gold Rises Over 2% as Soft Inflation Data Rekindles Fed Rate-Cut Hopes," Reuters, Feb. 12, 2026.
3. "What Happened to Gold, Silver, Platinum and Copper Today? Why Are Precious Metal Prices Crashing Sharply Today—Is This the End of the Bull Run for Metals?" Economic Times, February 11, 2026.
4. "Precious Metals Price Update: Gold, Silver, PGMs Face Another Bumpy Week." Reuters, Feb. 18, 2026.
5. "How Can 5 Strategies Maximize Printed Circuit Board Profitability?" Startup Financial Projection, Oct. 28, 2025.
6. "Copper's Price Ceiling Comes Into Focus as Speculative Capital May Unwind." Crux Investor, Jan. 21, 2026.



DFX

Basically a Basket of Crabs With Spreadsheets

BY JOHN WATSON, PALOMAR COLLEGE

One of the joys of living in San Diego is that I'm never far from the ocean. Aside from burying my toes in the sand at the beach, I particularly enjoy going down where the fishing boats come in. I once watched a fisherman standing beside several baskets of crabs. Most had lids tied on, but one was completely open. Curious, I asked why he wasn't worried about them escaping. It reminded me of a well-known story.

The Parable of the Crabs

In this story, as one crab begins climbing toward the top, pulling itself closer to freedom, another crab grabs it and pulls it back down. Then another attempt is made, and the same outcome occurs. The crabs aren't merely grabbing each other; their legs are hooked and intertwined, so that when one moves, several others move with it. None is truly climbing alone. No lid needed," says its fisherman.

"They're all connected. They won't let each other out."

The same kind of reaction happens whenever you bring up any topic related to "design for X." Mention design for manufacturing, design for innovation, design for test, or design for cost, or one of the hundreds of "design for X" principles to a room full of engineers, and you can almost feel the temperature change. Someone groans, others cross their arms, and before long, you get responses like, "Why are you making my job harder?"

"These principles are not stand-alone; they're interdependent by nature, like crabs in a basket. Each design force influences the others, whether or not teams account for it."

In fact, there are now so many design-for-X considerations that managing them individually just isn't practical. Modern products can involve dozens of DFX priorities, and trying to track each one separately leads to conflicting feedback, review overload, and slow decisions. That's why many companies organize them into DFX domains, grouping related constraints such as manufacturing, reliability, electrical, business, and compliance into coordinated clusters so trade-offs can be managed efficiently rather than one rule at a time. These principles are not stand-alone; they're interdependent by nature, like crabs in a basket. Each design force influences the others, whether or not teams account for it.

After researching DFX for many years, I have clearly seen that each one is tied to the rest. This affects manufacturability and cost, shifts reliability and testability, and requires responses to changes in materials and compliance. They're interconnected even more tightly than those crabs, where one motion affects the whole group. The lesson is the same: You're never designing against a single constraint, but inside a system where everything is linked.

When I was asked to compare only two DFX principles and "pull them out of the crab basket" to examine them separately, something became clear immediately: They didn't come out alone. Every time I tried to isolate one, others came with it. What was intended to be a simple comparison quickly turned into a discussion of systems. It was like trying to lift a single crab without disturbing the others; you immediately saw they were all connected.

Setting Up a Fighting Match

This sets the stage for a clash between two titans of industry: design for innovation (DFI), the bold challenger pushing the limits of what's possible, and design for manufacturing (DFM), the reigning champion grounded in what's actually achievable.

In this analogy, the challenger's corner represents design for innovation, a fresh face on the scene who's bold, fast, and boundary-pushing. DFI is a design philosophy focused on generating new ideas, achieving breakthrough performance, and creating differentiation. Its purpose is to explore what is technologically feasible before addressing production limits, cost optimization, or process constraints. It prioritizes creativity, new architectures, emerging materials, and unconventional approaches to discover solutions that don't yet exist in standard practice. It is state-of-the-art and lightning-fast.

This discipline operates at the front end of development, where the goal is vision rather than refinement. It thrives on "what if," "why not," and "has anyone tried this before?" Innovation accepts risk as part of progress and recognizes that many ideas will not succeed, but the ones that do can redefine an entire product category and industry for the future.

DFI is a powerful contender because it stretches boundaries, challenges assumptions, and constantly pushes designs beyond their comfort zone. It's exactly why stepping into the ring with the long-time champion becomes such a compelling matchup.

In the other corner is our reigning champion: design for manufacturing. They're battle-scarred and seasoned, disciplined, and grounded in reality. DFM is the design philosophy focused on ensuring a product can be built reliably, repeatedly, and cost-effectively at scale. While DFI seeks what's possible, DFM seeks what's practical. It evaluates whether materials are available, tolerances are achievable, processes are stable, yields are high, and production can run without surprises.

DFM has held the title for so long because, ultimately, every product must face it at some point in its lifecycle. A design can be brilliant on paper, but if it can't be fabricated, assembled, and tested consistently, it never leaves the lab. DFM is the longtime champion because it's the real-world proving ground where ideas must stand up to physics, process capability, and production economics.

Why Fighting It Out Matters

With our "contenders" in the ring, the bell rings, and the ultimate showdown begins. At first, it appears to be a classic fight. DFI throws the first punches: bold ideas, new technologies, unconventional approaches. It moves fast, takes risks, and pushes limits. Then DFM counters with precision: Can it be built and scaled? Will it yield? Can it be repeated thousands of times without failure? Every strike from DFI is met with a grounded response from DFM. The match seems to go back and forth, neither side able to eliminate the other.

As the rounds go on, something becomes very clear: Neither fighter can actually win alone. If DFI dominates, you get brilliant designs that can't be produced. If DFM dominates, you get safe products that no one is excited about. Instead of the fight ending with a knockout, it ends with both realizing they've been training for the same outcome all along.

The Result: Everybody Wins

What appeared to be a fight between DFI and DFM is more like the crab basket. While it appears the crabs are competing, each pulling the other down and preventing escape, they aren't. They are connected. Every movement from one affects the others. No crab moves independently, whether it is intended to or not.

DFI, DFM, and many other DFX principles behave similarly. Innovation seeks to climb, manufacturing seeks stability, and the motion of one immediately influences the other. They're not isolated forces taking turns; they're part of the same system, reacting in real time. What appears to be opposition is, in fact, interdependence.

So, the lesson from both the ring and the crab basket is the same: The goal is to understand how they move together. Because in real engineering, just like in that basket, nothing moves alone.

I-CONNECT007

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

BY STEPHEN V. CHAVEZ, SIEMENS EDA AND PCEA

Designing Without a Rulebook

WHEN ENGINEERING BECOMES INNOVATION

What if the very rules that made you successful as a PCB designer are the ones now holding you back? This reminds me of walking the floor and attending sessions at both PCB West 2025 and APEX EXPO 2026, where one common theme stood out: More designs with traditional PCB "best practices" simply don't apply. It's not because they're wrong, but because the problems we're solving have fundamentally changed.

In some cases, those best practices can actually limit performance. This is where PCB design moves beyond optimization and into something far more challenging: designing without a rulebook.

When 'Best Practices' Stop Being Best

For most of our careers, PCB design has relied on proven guidelines, rules that ensure manufacturability, signal integrity, and reliability. These practices are built on decades of experience because they work.

But what happens when you're working on something that hasn't been done before?

> "For teams, it keeps everyone aligned, even as the design changes. Without this structure, designing without a rulebook can quickly turn into chaos."

This could be designs for:

- Ultra-high-speed interconnects
- AI and HPC hardware pushing extreme bandwidths and power density
- Advanced packaging and heterogeneous integration
- Flexible and hybrid electronics

In these cases, the question shifts from "What's the right way to do this?" to "Why was it ever done that way, and does it still apply here?" That's a very different mindset and one that many teams are still learning how to navigate.

A Practical Approach: The Rulebook-free Design Framework

Designing without precedent doesn't mean guessing. It requires a structured way to deal with uncertainty. Here's a practical framework I've seen work that turns uncertainty into something manageable and repeatable.

1. **Challenge assumptions:** Don't blindly apply rules. Understand where they came from and whether they still apply.
2. **Define constraints:** Replace generic rules with clearly defined design intent using constraint-driven design.
3. **Connect the ecosystem:** Ensure electrical, mechanical, and manufacturing teams are aligned through a connected workflow.
4. **Explore rapidly:** Use simulation and early prototypes to test ideas quickly.
5. **Scale what you learn:** Capture insights so they can be reused and refined across the organization.

Back to Basics: First Principles Still Win

When the rulebook doesn't apply, you fall back on fundamentals. That means going deeper into electromagnetics for signal behavior, thermal analysis for heat management, mechanical constraints for real-world integration, and material properties that go beyond standard FR-4. At this level, PCB design becomes true system engineering.

Constraint-driven Design: Replacing Rules With Intent

If there's one thing that helps bring order to this kind of complexity, it's constraint-driven design. So, instead of relying on generalized rules, you define:

- Electrical intent (impedance, timing, coupling)
- Physical intent (spacing, geometry, stackup)
- Manufacturing intent (fabrication limits, yield)

The key is that constraints evolve as you learn more. For designers, this provides direction when things aren't clearly defined. For teams, it keeps everyone aligned, even as the design changes. Without this structure, designing without a rulebook can quickly turn into chaos.

Co-design Isn't Optional Anymore

One of the biggest shifts is how tightly coupled everything has become. Electrical, mechanical, and even software decisions are made simultaneously, not in sequence. A change in one area immediately impacts the others:

- A mechanical constraint can drive routing strategy
- Thermal limits can impact component placement
- Software requirements can influence timing and architecture


Visit us at DMEMS
Booth 153 - 22 to 23 April 2026
DMEMS2026
Del Mar Electronics & Manufacturing Show
Visit us at PCB East
Booth 605 - 29 April 2026
PCB EAST 2026
Conference & Exhibition
ventec
INTERNATIONAL GROUP
pro-bond &
thermal-bond
Glass Free Materials:
Next-Generation Performance.
Ventec's pro-bond series marks a significant advancement in dielectric material technology, offering high-performance, glass free laminates engineered for the most demanding electronic and thermal management applications.
By eliminating woven glass reinforcement, pro-bond eliminates fiber weave effects, delivering superior signal integrity, uniformity, and reliability. This innovation is essential for high-speed and high-frequency designs in aerospace, automotive, and next-generation communication systems.
By eliminating woven glass in thermal-bond, the thermal insulating effect of glass is removed from the dielectric, optimizing thermal performance and heat dissipation in MiB and embedded component multi-layer build up structures and component packaging applications.
Wherever technology takes you, Ventec delivers.
www.venteclaminates.com
JOIN VENTEC'S
GLASS FREE
REVOLUTION

This is where co-design becomes essential, not just collaboration, but true integration across domains.

Why the Digital Thread Matters More Than Ever

As complexity increases, disconnected workflows start to break down. The digital thread becomes critical because it connects design and simulation, design and manufacturing, and engineering decisions across teams.

That continuity enables something extremely valuable: a reliable digital twin. This isn't just a static model either. It evolves as you run simulations, build prototypes, and validate performance. Over time, it becomes a much more accurate representation of the product and a powerful tool for decision-making.

Simulation, Prototyping, and Learning Faster

In traditional design flows, success often means getting it right the first time. In innovative designs, success comes down to how quickly you can learn. Simulation helps you explore ideas before building. Prototyping helps you validate what simulation can't predict. The teams that move fastest today aren't the ones who avoid mistakes, but learn from them the fastest.

A Real-World Example

Take a next-generation AI accelerator board. You're dealing with extremely high data rates, tight power and thermal constraints, and complex packaging interactions.

Traditional spacing rules and stackup guidelines only get you so far. At some point, you have to define new constraints, rely heavily on simulation, and iterate through prototypes.

You're not optimizing a known design; you're figuring out what the design even needs to be.

The Organizational Challenge Most People Overlook

Here's something we don't talk about enough. Designing without a rulebook is hard both technically and organizationally. It challenges schedules, budgets, and expectations around predictability.

Many teams struggle not because they lack engineering talent, but because their processes are built for repeatability, not exploration. Innovation fails because the organization isn't set up to support how innovation actually works.

What This Means for Leadership

For engineering leaders, this shift is significant. If your organization is focused only on efficiency, you will run into problems. What matters more is how quickly your team can iterate, how well decisions are connected across domains, and how effectively you capture and reuse what you learn. This is where investments in constraint-driven design, digital thread infrastructure, and digital twin capabilities start to pay off; not just in one project, but across the entire organization.

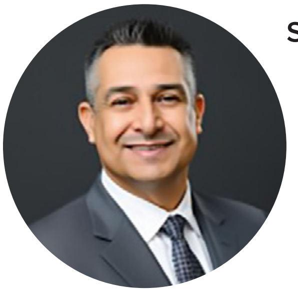
A New Mindset for Designers

For PCB designers, this shift changes their role in a meaningful way. You're no longer just implementing a design; you are helping define it. That requires becoming more comfortable with uncertainty, more collaborative across disciplines, and more grounded in first principles. Because at the end of the day, you don't optimize what's never been built, you discover it.

Final Thoughts

Designing without a rulebook isn't about throwing away everything we know. It's about knowing when those rules no longer apply, and having the tools, mindset, and processes to move forward anyway. It's challenging, no question. But it's also where the most meaningful innovation is happening in our industry today. For those ready to embrace it, it's an opportunity to not just build better boards but to help define what we can do next.

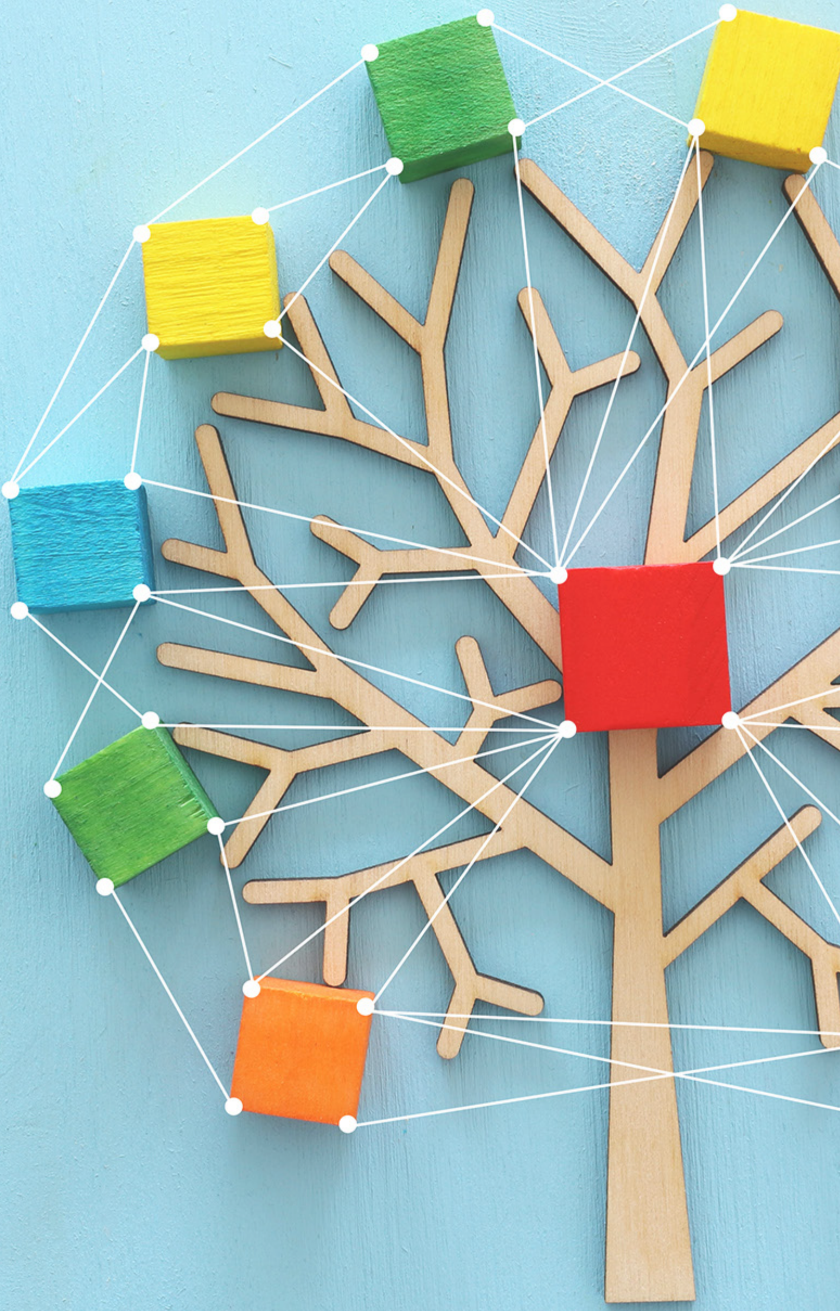
I-CONNECT007



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

BY KRISTIN MOYER, GLOBAL ELECTRONICS ASSOCIATION

Rethinking Stackup, Materials, and Tolerances *in Modern Designs*



The simple rectangular rigid PCB is becoming increasingly infrequent. This reality necessitates designing with concepts well outside traditional rigid PCB methodologies. For example, the designer of wearable electronics may need to implement conductive fibers integrated into the textile material. Heads-up displays, like those in VR/AR headsets and glasses, require transparent circuitry etched into the display glass. The process of designing without a rule book usually starts with something other than the traditional board design process.

In this process, the complete schematic is sent to the PCB layout tool. The first step is usually to define or import the mechanical board outline from the mechanical design engineer's or the customer's specification. Next, you define the board stackup and place all the parts into the design, followed by routing and validation. Finally, you have documentation and manufacturing file generation.

In new designs, there are often several steps that differ from this traditional approach. First, when considering modern designs, it may be necessary to select specialty materials with properties that may not have been accounted for in the designs' electrical requirements. You may need to select materials for thermal, chemical, or mechanical needs. In the case of 3D-printed electronics, there is no reinforcement material (glass) like that used in a traditional PCB design.

In this case, conduct a physics study of the materials first, focusing on the conductive

INSULECTRO

WORLD-CLASS TOOLS

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

CALL 949.587.3200 FOR MORE INFORMATION

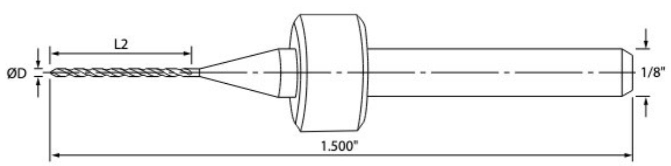
KYOCERA

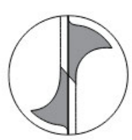
Kyocera designs and manufactures tight tolerance precision carbide cutting tools for PCB applications. Products include drills, routers, end mills, and specialized cutting tools ranging from: 0.05mm to 6.70mm diameters (0.0020" - 0.2638").

Drills, End Mills, Routers, Specialty Tools

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

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.





4 Facet Point Geometry

"So, as this one example shows, the rulebook for PCB design that has been so successful for design engineers for so long is no longer the only approach."

material, as printed conductive materials are typically not pure crystalline lattice metals like copper foil but instead are copper- or silver-impregnated inks or resins. This means that the conductivity of the material, while sufficient for electrical signal propagation, may not be as good as a pure conductor. This means that there may be a greater IR drop on a printed conductive path compared to a pure conductor path.

Additionally, a resin structure without the reinforcement materials in traditional PCBs will exhibit different CTE rates, and the CTE will be significant in all three axes simultaneously. This means that a different approach to thermal expansion must be considered for these designs. Typical Z-axis copper-balancing techniques do not necessarily work when the conductor material is placed in all three axes.

Manufacturing file generation is also different in these designs. Rather than traditional layer data formats like Gerber or ODB++, new designs for additive manufacturing will need 3D modeling file formats, such as STEP (Standard for the Exchange of Product model data, ISO 10303), to define the data to be 3D printed.

Tolerances and other fabrication limits must also be re-evaluated by the designer. When using additive processes for large-scale design features like these, the deposition and feature registration controls now have three simultaneous axes of tolerance rather than a 2+1 tolerance. In traditional designs, the feature registration along the X-Y axis on the same layer is controlled during the etching process. On the other hand, the layer-to-layer Z-axis registration is controlled during the lamination process.

In 3D printed electronics, you are concerned not only with the shape of the design but also with the 3D support structures that provide mechanical support during the printing process (and that are removed after printing). Removal of the structures usually leaves an artifact or imperfection on the surface of the board, which will require additional cleaning and shaping of these sites to ensure the surface profile requirements are met.

Parts placement is an additional challenge. In a traditional two-sided PCB, parts are placed on the top surface or bottom surface. Whatever orientation the PCB is mounted in will produce physical forces on the board in only two orientations, from the top or from the bottom. These forces include load and pressure applied to the surface of the board due to the mass of the parts, torque from cantilevered parts off the board, and thermal stresses from the thermal energy transferred to the PCB.

With parts mounted on only two flat surfaces, the magnitude and directionality of these forces are relatively straightforward to calculate. For a three-dimensional structure, with parts mounted at any angle and on any surface or orientation, the complexity and difficulty of these calculations and determinations increase by an exponential order of magnitude.

So, as this one example shows, the rulebook for PCB design that has been so successful for design engineers for so long is no longer the only approach. Instead, the next generation of designers will need a thorough understanding of all the scientific disciplines to grasp everything that will go into the design process. The flowchart of the PCB design process will need to be significantly expanded to account for all these new processes and effects.

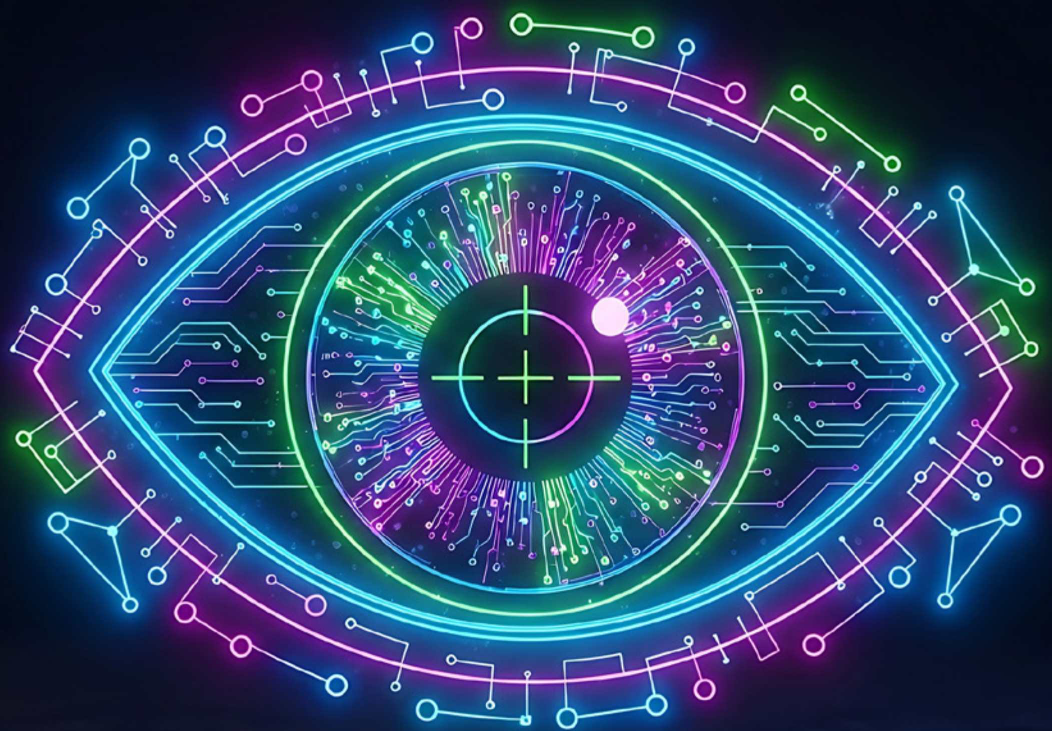
Design will no longer have a rulebook to follow; instead, it will use a multidisciplinary decision tree to help the designer ensure a successful design, the first and every time, regardless of the unique requirements of the design. **I-CONNECT007**

Kristin Moyer is an instructor for the Global Electronics Association.

FEATURE INTERVIEW

BY MARCY LARONT, I-CONNECT007

A Designer's Focus on HIGH DENSITY

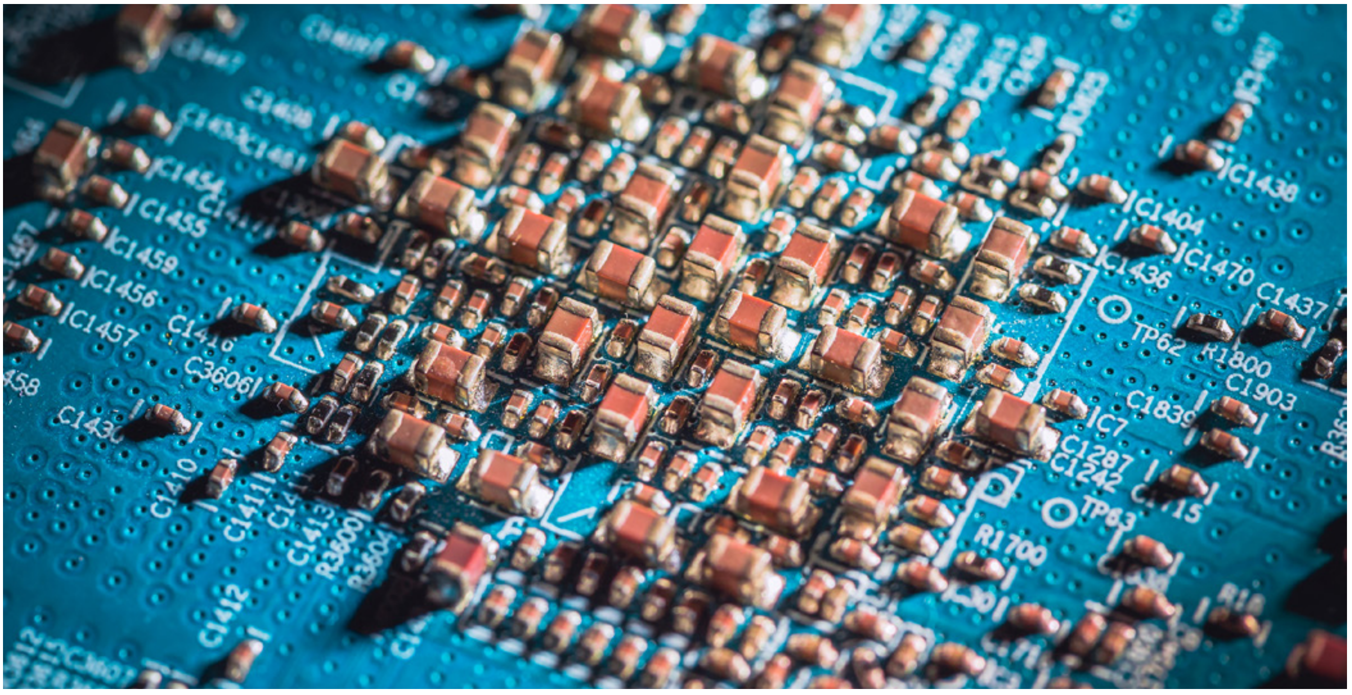


Vern Solberg is a distinguished member of the Global Electronics Association Raymond E. Pritchard Hall of Fame and has served as chair or vice chair of many committees, developing technical standards and implementation guidelines, including the IPC-7090 series, which focuses on design for manufacturing and reliability for electronic assemblies. He's a longtime contributor to Design007 Magazine, and he conducted a half-day tutorial at APEX EXPO 2026, where he addressed 2D, 2.5D, and 3D packaging and ultra-high density hybrid bond interconnect. I caught up with Vern at the show and asked about his pivot from addressing more standard design challenges to his focus on high-density circuits.

Vern Solberg

Marcy LaRont: Vern, it's always good to see you. You are an icon in our industry. Tell us how you got started in the electronics industry and found yourself in this wonderful place.

Vern Solberg: Coming out of school, I was trained to be an industrial designer, but I couldn't find work, as the term "industrial design engineering" wasn't terribly meaningful at that time. So, I found a job as a draftsman with ITT, a communications company at Vandenberg Air Force Base. It just so happened they had an R&D facility in Palo Alto that needed help. One of the guys who was working on a new technology asked if I knew how to design printed circuit boards. I did not. He became my mentor in PCB design, and that really kicked off my career.



It is always interesting how we end up doing the exact thing that becomes central to our careers.
I eventually moved to Southern California for a job where they needed someone with a mechanical background, which I had. At that point, my work went beyond just PCB design to more of a full mechanical system design, including the PCBs. At my next company, they needed somebody who could design PCBs for volume manufacturing. This company was getting into automation and needed all its PCBs redesigned accordingly. Eventually, I moved back to Northern California (South Bay Area) and pretty much stayed there. As one's career progresses, though, you end up in management, and I wanted to be back on the engineering side. Finally, I started my own small design company, NuGrafix Group, in Los Gatos, focusing on surface mount technology, which was becoming an important technology in the design and fabrication of PCBs.

I was ahead of the curve, as the industry mainstream was still puzzled about what surface mount actually was. (Originally, we had called this type of connection technology "hybrid technology.")

For my customers, everything needed to be miniaturized. Instead of designing boards at a 2:1 scale, we were designing at a 10:1 scale. At the time, the disk drive industry was emerging from Silicon Valley, and they wanted to make their boards smaller. The only way they could do that was to use surface-mount components, which had become more widely available. At one point, I was designing boards for several disk drive companies in Silicon Valley.

You have been involved in design for manufacturability since the beginning of SMT in the U.S., talking about design and fab. That has to be interesting.
If you don't design for manufacturing, you get instant feedback when you go out on the production line and see the expressions on people's faces. I had designers who worked under my management who were afraid to go on the production floor, but I'd tell them, "Just be casual about it. They will give you valuable feedback on your design, and if there is a problem, they will let you know, and that is a good thing."

SMT is your expertise, which you have expanded now to include microelectronics. You've also recently published a book about design guidelines for surface-mount microelectronics technology. When did you decide to evolve the surface mount discussion to include microelectronics, HDI, UHDI, and even substrates?
When the industry began to move over to surface mount technology, suddenly everybody was very interested in why they had made the components

one quarter-size of what they had been before. Of course, it was to scale things down and make the board much smaller. Placing and soldering the SMT components directly onto the PCB's surface was perfect for automation, which had become much more widely used in assembly. When the industry moved over to SMT, it had everything to do with the silicon driving the ability to fit more dies on a wafer. Again, we are seeing silicon driving PCB technology in a different direction with HDI and UHDI, and now, package substrates.

Your tutorial this week is called "PCB Design Engineer's Introduction to High Density Semiconductor Packaging Technologies." I've also noticed that your column has shifted away from standard PCB design challenges. What will you cover in the tutorial, and what do you hope that designers will take away?

I begin with a simple review of legacy components, showing where surface mount and miniaturization started and eventually evolving to variations of ball grid array (BGA) semiconductor packaging. As we have experienced over the past couple of decades, we have continued to see semiconductor companies add more functionality to the die elements, increasing bond pads or bumps per die, and consequently, the packages for those die elements follow suit, getting smaller and their terminals increasingly closer together, a challenge for both PCB designers and manufacturers.

"To support the industry's goal of onshoring key technical skills, we need to learn to design and fabricate very high-density package substrates and interposers, beyond just PCBs."

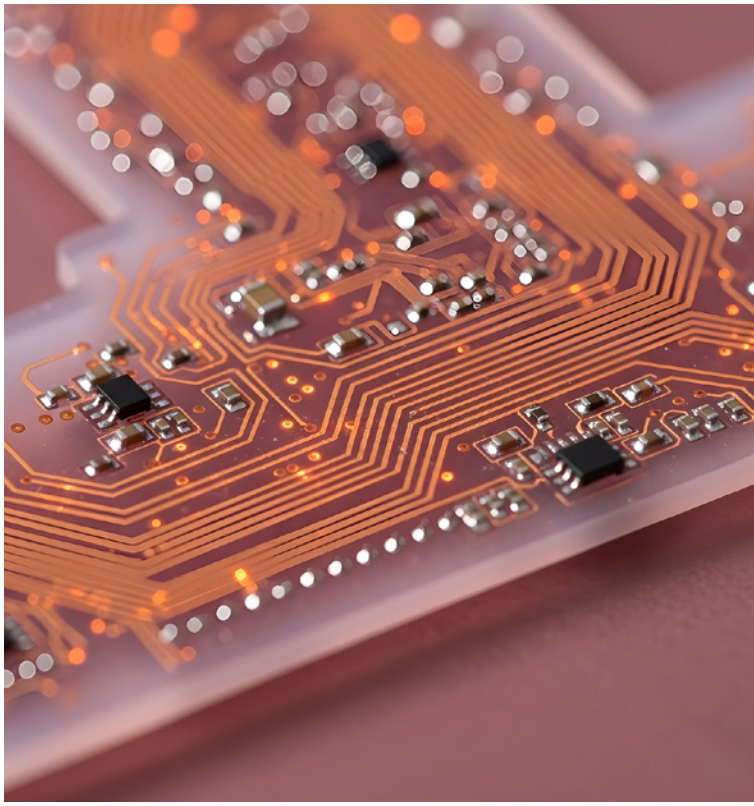
Manufacturing methods had to change to keep up with the semiconductor package technology that was being implemented. Here we are again, seeing the next evolution of this process to accommodate a whole host of new components with a terminal pitch not even considered possible five years ago. It's a matter of scale. To support the industry's goal of onshoring key technical skills, we need to learn to design and fabricate very high-density package substrates and interposers, beyond just PCBs. The interposer will furnish a great deal of interconnect between these fine-pitch semiconductor die elements mounted on its surface. The power, ground, and in/out interface is then routed through plated via holes and spread out to a wider spaced array terminal pattern on the opposite surface. The wider pitch terminal pattern will better accommodate termination to the next level package substrate or circuit board.

The traditional monolithic, system-on-chip (SoC) fabrication put all the functions into one single piece of silicon. With the introduction of multicore processor development, the semiconductor core(s) and their related support functions were integrated onto a single silicon base; the semiconductor die outline became excessively large, difficult to process, and yields became unacceptable. To improve process efficiency and overall manufacturing yield, developers of the newer generation, multi-core processors opted to separate the supporting functions—memory, I/O, specialized accelerators, digital signal processors, as examples. These now singulated functions have been classified as chiplets, configured to mount onto the interposer in proximity with the core processor's die. We used to call these configurations multichip modules (MCMs). Chiplets are just another evolution step in semiconductor package technology we now refer to as heterogeneous packaging.

So, for the PCB design engineer developing the interposer or package substrate, the software tools are available; it's really just a matter of scaling and applying the design rules for chiplet land patterns and what is classified as ultra high density (UHDI) circuit routing strategies. The bottom line is that a significant amount of time and resources are invested in developing and fabricating these high-performance processors. Using what the industry is referring to as the "LEGO" approach for system-on-silicon (SoC) applications, developers can avoid discarding significant portions of the silicon-based product. So, instead of getting 50% yield, maybe you get close to 95+% yield.

You've been in the design game a long time. So, you believe that the new PCB designer will be involved in the design of substrates or interposers?
Absolutely. As I mentioned, it's a matter of scaling. PCB designers have been connecting parts for decades, and they will get the same data you would use for a packaged component, but now they're shown as bare dies with preformed terminals. You simply have to adapt the existing CAD tools to assist in developing a new component library and establish the design criteria for circuit routing. The design engineer may still be using multiple layers but with significantly different materials—maybe silicon or glass as a base. Folks at Georgia Tech, for example, have been working on developing the process for using glass as an interposer base for over 20 years. Although silicon remains a primary base material for interposer applications, the polymer glass materials are evolving technology that is just beginning to hit the mainstream, much like we saw with the transition to SMT back in the 1980s.

In bringing up the next generation of PCB designers, we talk about them needing to know more than just the electronic design piece. How much do today's PCB designers need to know about the engineering disciplines, even if they aren't experts in them?
The design engineer will certainly need to understand base materials and how they impact product performance. We have a lot of good materials today and new ones being developed for higher operating temperatures, less flexing, and less loss. The problem with circuit boards has always been that the materials they are made with are not physically compatible with the uncased silicon they are being mounted to. When it comes to material properties, there is quite a disparity between the behavior of the silicon and the epoxy-glass based circuit board. But material manufacturers (primarily in Japan) have created new formulations where the materials coefficient of thermal expansion



(CTE) rate is very close to the silicon itself. When employing semi-additive circuit forming technology, the commercial PCB fabricators will be a viable supply source for UHDI package substrates.

You do a lot on the education front, working with young engineers. What do you tell young people these days?
I tell them not to rush it just because somebody's putting pressure on them; do the best they can to make it right the first time. Rushing almost always leads to redesigns which cost companies a lot of money. Sometimes it cannot be helped, of course. There will always be revisions that may have nothing to do with the design engineer, so patience is part of it.

And I always tell them to remember that somebody has to manufacture their design. You're going to design it once, then you can watch the company churn hundreds, or even thousands of your design off the production line. There is quite a lot of satisfaction in seeing that.

Vern, it's good to catch up with you. Thank you.
Thank you, Marcy. **I-CONNECT007**

Advanced Materials Roundtable

LISTEN NOW!

BY MARCY LARONT, I-CONNECT007

Driven largely by the need for advanced chip technology for super compute ability and AI applications, low Dk, low-loss resin systems, and heavy copper laminate are attracting significant attention and global resources. There are numerous unavoidable challenges in this market that will impact manufacturers and the supply chain, and they may be hitting critical mass sooner than OEMs and fabricators think.

In this roundtable discussion, moderator Marcy LaRont speaks with topic experts: Mark Goodwin, COO of Ventec International; John Fix, senior manager of Glass Fiber Marketing at Nittobo America's Glass Fiber Research and Marketing division; and Ed Kelly, principal of Four Peaks Innovation and an expert consultant to material manufacturers such as Nanya. They discuss some of these dynamic material trends and what is driving them, as well as examining the current state of global supply for glass and copper, and what that implies. What are manufacturers and OEMs to do? Listen and find out.

PARTICIPANT BIOS

Mark Goodwin, COO

Mark Goodwin is Chief Operating Officer at Ventec International Group, an established leader in technology and supply chain solutions, dedicated to innovative solutions in the electrical and electronic materials manufacturing industry. With over a decade of steering operations at Ventec International Group and extensive experience across the global laminate and PCB supply chain, he works closely with OEMs, PCB fabricators, and industry partners to support strategic material planning and long term supply chain resilience.

Ed Kelley, Technology and Marketing Executive

Ed Kelley is a Technology and Marketing Executive with more than 38 years of experience in printed circuits and electronic materials development and commercialization with broad functional experience in Research & Development, Process Engineering, Operations, Product Management & Marketing, New Product Introduction, Applications Engineering & Business Development. He holds extensive global experience in North America, Asia, and Europe, including 9 years living in Asia. A results-driven leader with a proven track record, Ed is the author of the base material chapters of the *Printed Circuits Handbook*, was Chair of the iNEMI PCB & Laminate Technology Integration Group from 2020-2023, is a contributor to the iNEMI PCB & Laminate Roadmap, and 2024 recipient of iNEMI's Dedicated Service Award.

John Fix, Senior Manager of Glass Fiber Marketing

John Fix is the Senior Manager of Glass Fiber Marketing at Nittobo America Inc., specializing in the planning, coordination, and direction of research and marketing initiatives for Glass Fiber products. With a long tenure on the supply side of electronics manufacturing, he is skilled in setting and achieving sales and growth goals, and developing and implementing strategic plans.

Designing for the Future of Manufacturing Reality: *Surface Finish*

BY MATT STEVENSON, ASC SUNSTONE CIRCUITS

When designing the complex boards that many electronic devices require to operate, designers should consider manufacturability at every step. This is my last article focused on designing for the always-evolving manufacturing reality.

The previous installment focused on the solder mask and legend process. The last phase of production involves applying surface finish to protect copper from oxidation and facilitate soldering components onto the board. Choosing the right surface finish has always been important. If you are creating intricate designs with a wide variety of components, like for an ultra-high density interconnect (UHDI) board, surface finish is a critical last step.

Surface finish for UHDI boards requires extreme flatness and high solderability for fine-pitch components. Applying surface finish to the solder pads can affect solderability during assembly to a limited extent, but today's surface finishes are designed with solderability in mind and are necessary to sustain the productive life of the board.

Evaluating Surface Finishes

When selecting the right PCB surface finish for their project, designers find themselves weighing a variety of factors. Designers should consider the following when they establish their surface selection criteria:

- Cost
- Restriction of hazardous substances (RoHS)
- Flatness of the board surface
- Solder joint integrity
- Shelf life
- Wire bondable (if applicable)

With more boards having to accommodate larger components and more leads over multiple edges or surfaces, flatness is a more critical manufacturability consideration and surface finish plays a large role.

There are five commonly used surface finishes, plus several more that are used for specialty applications.

Hot Air Solder Leveling (HASL)

HASL has a long and successful history, and is still relied upon across many industries, most notably defense. It has become associated with the adage, "Nothing sells quite like solder." HASL is cost-effective, durable in harsh environments, and very reliable. It also has a long shelf life (the time the board can sit in inventory before assembly) and provides excellent solderability.

But it has drawbacks. It often leaves a lack of

flatness on the board that can be problematic during assembly, especially for IC components. The standard HASL surface finish includes lead, which is not compatible with RoHS requirements. Note that a lead-free HASL option is available. Designers should also consider the number of surface-mount components their board must accommodate before choosing HASL. If there are more than eight pads per side or fine pitch on an integrated circuit with many leads, flatness becomes an issue.

Immersion Silver

Immersion silver is a thin deposit that closely follows the contours of the copper and offers designers and manufacturers a very flat surface finish. It will be relatively level and have a good grain structure that the silver follows, creating exceptional flatness and planarity.

When we compare HASL to immersion silver, we are essentially contrasting old-school with new school. Both create the best possible solder joints. The true advantage of immersion silver is its flatness and ability to handle fine-pitch components. It is also cost-effective compared to other options and in full compliance with RoHS standards.

Like your grandmother's silver flatware, boards treated with immersion silver surface finish can be susceptible to heat, humidity, or even UV light. They will tarnish if left out in the elements. Boards with immersion silver surface finish that aren't going directly to assembly should be stored and packaged as you would a moisture-sensitive component.

Electroless Nickel Immersion Gold (ENIG)

ENIG is a two-layer coating. The first layer is nickel electrolessly deposited atop the copper, with a layer of gold immersion-plated on top of the nickel. This surface finish includes two layers because nickel is a metal that, in its pure form (like it is here), passivates (becomes unreactive) in oxygen. The layer of gold protects the nickel from oxidation.

The solder joint forms between the tin in the solder and the nickel on the surface finish. This intermetallic is different from the copper-tin finish, meaning it has pluses and minuses, but ENIG produces solid solder joints for most applications. Like immersion silver, ENIG creates a flat surface—it's a bit thicker than immersion silver. As you can imagine, cost is a big issue with ENIG as gold is among the most expensive raw materials on the planet, and the electroless immersion process is more costly than other surface finish application processes.

Electroless Nickel Electroless Palladium Immersion Gold (ENEPIG)

ENEPIG is the same surface finish as ENIG, but with a layer of palladium between the nickel and the gold. ENEPIG allows gold leads to be wire-bonded to the pads. Only specialized applications require wire bonding. ENEPIG is typically called out in the design specs, and entails the same cost


MicroCraft's Product Range
Built for What's Next
EMMA
Moving Probe Tester
Moving Probe Testers
Fast and reliable,
Emma testers are trusted worldwide
for precision and speed
E8M6151G-AL
8-Probe Loader & Unloader
E8S6151G
8-Probe High Accuracy
CraftPix
On-Demand Inkjet Printers
Whether prototyping or
full production, the CraftPix lineup
offers inkjet systems to meet
your throughput needs
CPA8165ST
Stacker Type Loader & Unloader
C8K7265T
8-Head Inkjet Printer
Sarra
Automated TDR & VNA Testers
Built for today's high-speed designs,
Sarra adds advanced impedance
and signal integrity testing
to your toolkit
H2P45
2-Probe Variable Pitch TDR Tester
S4V54
4-Probe VNA Tester
MicroCraft
www.usamicrocraft.com sales@usamicrocraft.com

Surface Finish Selection Tips

Every electronic device project has unique priorities that will impact decisions made by PCB designers. You often have to balance budgetary constraints with design constraints and functional requirements. Consider:

- If containing production costs is your focus, HASL is probably your best bet. It is less expensive than other surface finishes, has good solderability, can sit on the shelf for up to a year, and is durable.
- More complex boards—those with high-density interconnects—will likely have fine pitch components. We often use ENIG in these cases because it offers great solderability, resists oxidation, produces superior flatness, and ensures reliable joints. It's a sound choice for boards operating in harsh environmental conditions.
- Immersion silver is a solid choice for high-frequency applications because it delivers excellent signal integrity and limited signal loss.
- For devices operating in harsh environments, ENEPIG offers solid protection against corrosion and oxidation.

considerations as ENIG, plus those associated with adding a palladium layer. That additional layer can also create solderability issues that may impact board durability and operability.

We recommend using ENEPIG surface finish only on the parts of the board where it is necessary, and ENIG on all other solderable surfaces. This can help contain costs, improve manufacturability, and avoid issues with board function.

Immersion Tin

As the name suggests, this method is similar to immersion silver. The result is a very flat and thin surface finish that is cost-effective, with a relatively long shelf life. This surface finish has low insertion friction, making it a viable surface finish where you have press-fit connectors. Though "nothing solders like solder," the introduction of tin makes this a very solderable surface finish with a copper-tin intermetallic bond.

Immersion tin is more vulnerable to environmental conditions than immersion silver surface finish. Boards treated with immersion tin should be used promptly or stored with environmental concerns in mind.

These five methods represent the most common surface finishes, but there are others used for more specialized applications or to keep production costs down. Some of the more noteworthy finishes include:

- Electrolytic nickel with hard or soft gold
- Electroless palladium immersion gold (EPIG)
- Organic solderability preservative (OSP)

I-CONNECT007

For a deeper dive into the wide range of surface finish options, listen to the **Designing for Reality: Surface Finish** episode of **On the Line With… .**

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.

BY MANFRED HUSCHKA, MANFRED HUSCHKA MANAGEMENT CONSULTING (SHENZHEN) LTD

Is China Plus One Still Happening in the PCB Industry?

For much of the past five years, China Plus One has been shorthand for supply-chain diversification: reducing dependency on mainland China by adding manufacturing capacity elsewhere in Asia. In the PCB industry, however, in early 2026, it is more nuanced. It looks less like a clean geographic shift and more like a layered, capital-intensive rebalancing of global capacity, one that still leaves China deeply embedded at the center.

From Decoupling to 'Webs of Connection'

At the 56th World Economic Forum in Davos, Canadian Prime Minister Mark Carney succinctly captured this changing mindset in a Q&A session on de-risking from China. He warned that excluding major nodes from global networks is not risk management but a strategic mistake, and that supply chains are increasingly webs of connection, not binary choices.

That framing resonates strongly with today's PCB ecosystem. While investments across Southeast Asia are accelerating, very few industry players are fully exiting China. Instead, they are redistributing risk and cost across multiple geographies while keeping advanced capabilities anchored in China's mature manufacturing ecosystem.

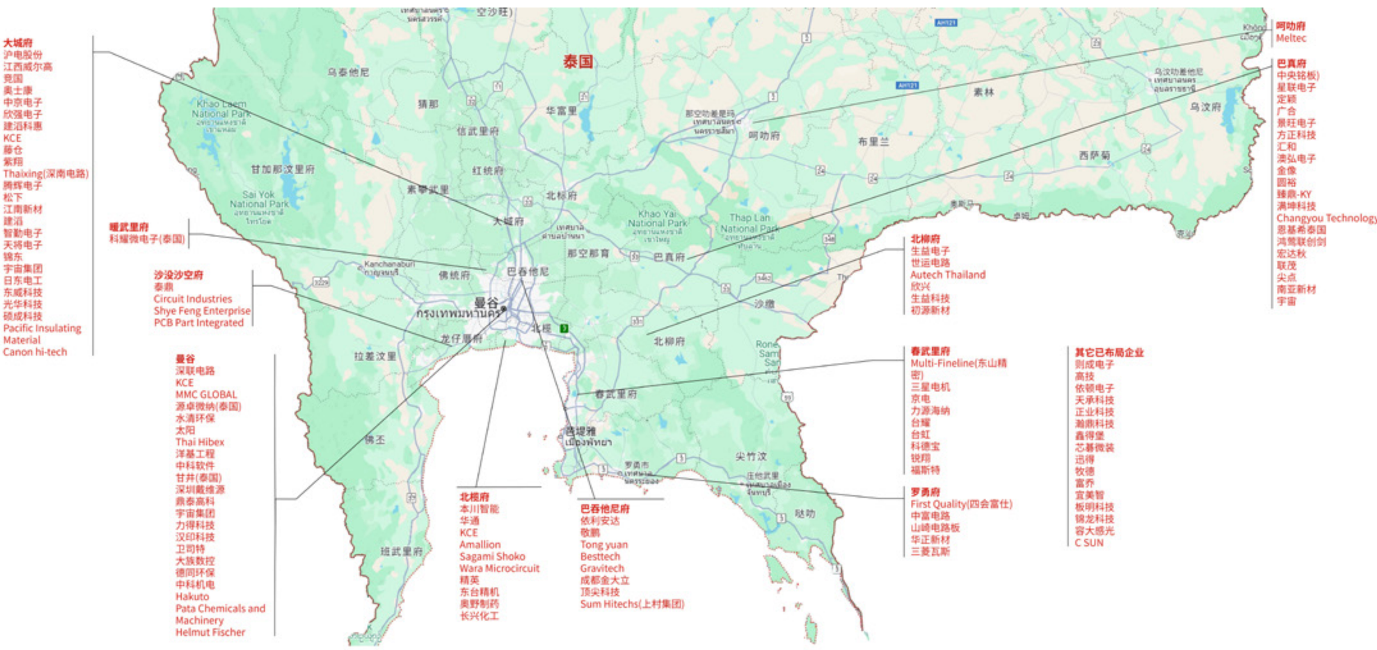
The Core Market Driver: AI Servers

Before examining regional investment patterns, it is essential to understand the dominant market driver reshaping the PCB industry: AI servers.

According to Goldman Sachs research published in early 2026,[1,2] the rapid escalation of AI server capabilities is triggering a supercycle in the PCB and copper-clad laminate (CCL) industries. As AI servers transition from node-level to cluster-level architectures, requirements for CCL, PCBs, and interconnects are rising sharply, with 800G to 1.6T networking becoming an industry standard.

Goldman Sachs highlights several structural implications:

- Material upgrades, with platforms moving from Panasonic M7/M8 toward higher-grade, faster materials often referred to as M9-class and beyond



This map illustrates how PCB shops are located in clusters in Thailand.


mks
Atotech
Run Leaner.
Last Longer.
Lower Cost of Ownership with MKS' Atotech systems approach –
Our equipment combined with our DFS and After Sales.
Combine MKS' Atotech equipment with the Digital Factory Suite (DFS) and an After Sales maintenance contract to establish a closed-loop system that converts real-time data into operational efficiency and prompt service. The DFS adds real-time condition monitoring, predictive maintenance, traceability, and remote support, delivering up to 50 % fewer unexpected downtime events and lowering maintenance costs by up to 40 %. Our after-sales programs maintain optimal performance with scheduled inspections, original equipment manufacturer (OEM) parts, retrofits, and 24 / 7 support, extending equipment lifetime, maximizing availability, and making service costs more predictable. Since our Uniplate® and vPlate® lines are IIoT-ready, DFS can connect from day one to stabilize processes and optimize energy usage. Reduce your total Cost of Ownership across your line by using our integrated systems approach, which combines equipment, software, and service!
To find out more about how to lower CoO across your line, scan the QR Code to the right.
info@atotech.com
www.atotech.com

- Sharp growth in value density, with Goldman Sachs projecting the AI server PCB market to expand from approximately $3 billion in 2024 to over $25 billion by 2027, and CCL demand growing even faster off a smaller base
- Higher-layer PCB designs, driven by integrated AI rack architectures such as NVIDIA's VR200 and VR300 platforms
- Architectural shifts, where PCB backplanes and midplanes increasingly replace traditional copper cabling
- Supply-side constraints, giving pricing power to suppliers capable of producing high-end CCL and advanced PCBs

This front-loaded investment cycle is reshaping competitive dynamics across the PCB value chain.

Southeast Asia: Capacity Moving at a Cost

Across Thailand, Vietnam, and Malaysia, PCB investment announcements continue to accelerate:

- Victory Giant Technology acquired APCB Thailand in 2024 rather than building a greenfield site and broke ground in March 2025 on a $260 million PCB plant in Bac Ninh, Vietnam, scheduled to begin operations in Q4 2026.
- Meiko is investing approximately $225 million in Vietnam to manufacture advanced 3D-structured boards for Samsung's AI-enabled smartphones.
- Zhen Ding has budgeted more than $1.58 billion in 2026 capital expenditure, including high-end capacity in Thailand and Taiwan, with roughly 10 plants under construction.
- Unimicron plans over $800 million in 2026 CapEx, largely for advanced ABF substrate production for AI chips and server platforms, and has acquired land in Thailand for potential multi-factory expansion.
- Gold Circuit Electronics (GCE), NVIDIA's main PCB supplier for switch trays, is operating its first factory in Thailand at full capacity.
- Unitech PCB, Kingboard, Compeq, and Tripod Technology have all announced or ramped new capacity across Thailand and Vietnam.
- TTM Technologies officially opened its first PCB manufacturing facility in Penang, Malaysia, in April 2024, with a $200 million investment. The site is designed for a future Phase-2 expansion of approximately 25% capacity.
- AT&S has begun high-volume manufacturing at its Malaysia campus, supplying high-end IC substrates for AMD's data center processors.

On paper, this resembles a textbook China Plus One strategy. In practice, the economics remain challenging.

Morgan Stanley estimates that PCB production costs in Thailand are approximately 20% higher than in mainland China, driven by cross-border logistics, equipment debugging costs, and differences in labor efficiency. While hyperscalers such as NVIDIA and Amazon are currently willing to absorb this premium to secure supply-chain resilience, these structural disadvantages remain a meaningful headwind for others.

Structural Constraints Outside China

Prismark analysis indicates that many new PCB entrants outside China are proceeding cautiously. Despite achieving volume production, numerous suppliers face:

- Weak end-market demand outside AI-related segments
- Low utilization rates
- High material and logistics costs
- Shortages of trained engineers and experienced production talent

Labor dynamics further complicate ramp-ups. Thai labor practices tend to emphasize balanced working patterns, while PCB suppliers from mainland China and Taiwan often operate more aggressively during scale-up phases. Demand for Chinese-speaking engineers and technical managers remains particularly acute, and once incentives and efficiency adjustments are factored in, effective labor costs in Thailand can be several times those in China.

Material supply is another structural bottleneck. Local production of laminates and specialty chemicals remains limited, forcing reliance on imported advanced materials such as low-CTE fiberglass fabrics, ultra-thin copper foils, specialty resins, and prepregs. Equipment support is similarly constrained, with high service costs due to reliance on overseas engineering teams.

The result is a widening gap between Tier-1 suppliers with access to materials, equipment, and capital, and newer entrants struggling to achieve sustainable margins.

China: Still Investing, Still Advancing

Crucially, diversification has not slowed advanced investment within mainland China itself:

- In December 2025, PCBAIR in Shenzhen launched pilot production of advanced 8-layer glass core PCBs using through-glass via (TGV) technology for AI accelerators, high-speed servers, and optical transceivers.
- Jiangmen Yixiang Industrial broke ground in January 2026 on a project targeting annual PCB output of 1 million square meters.
- Kinwong recently completed the roof-capping of its $415 million Zhuhai AI server and high-end HDI project, which is expected to add more than 800,000 square meters of high-end HDI capacity by late 2027.
- TTM Technologies also announced an additional $200 million China capacity expansion during its February 2026 earnings call, on top of previously allocated investment for data-center computing applications.

WUS Printed Circuit: CoWoP and mSAP as a Technology Incubation Platform

A particularly notable example of China's continued push into advanced interconnect technology is WUS Printed Circuit's newly announced high-density optical-electrical PCB project in Jiangsu province, with a planned investment of up to $300 million.

Rather than a conventional capacity expansion, WUS has positioned the project as a technology incubation platform spanning R&D, pilot production, process validation, and eventual commercial deployment. To support this, the company has established a wholly owned subsidiary dedicated to the development of chip-on-wafer-on-platform (CoWoP) and modified semi-additive process (mSAP) manufacturing technologies.

CoWoP, often described as "CoWoS without a substrate," represents a significant architectural departure from traditional packaging flows. In this approach, silicon dies and interposers are directly bonded onto a reinforced platform PCB, eliminating the need for conventional organic package substrates and BGA processes. By collapsing multiple packaging layers into a single integrated structure, CoWoP aims to improve signal integrity, power distribution efficiency, and overall system integration—key requirements for next-generation AI servers and high-bandwidth computing platforms (Figure 1).

Complementing this, WUS is deploying mSAP processes to support ultra-fine line widths and spacing required for high-density interconnect and optical-electrical integration. The project targets the development of optical-copper integrated PCB architectures, enabling tighter signal routing, reduced transmission loss, and improved thermal and electrical performance in high-speed modules.

What Is CoWoP? (Chip-on-Wafer-on-Platform)

CoWoP (chip-on-wafer-on-platform) is an advanced interconnect and packaging architecture in which silicon dies and interposers are directly bonded onto a reinforced platform PCB, rather than being assembled onto a conventional organic package substrate.

By eliminating traditional substrate and BGA layers, CoWoP shortens the electrical path between chips, improving signal integrity, power delivery efficiency, and overall system integration. The approach is designed to support the extreme bandwidth, low latency, and high power density requirements of next-generation AI servers and high-performance computing systems.

CoWoP is often described as *"CoWoS without a substrate,"* reflecting its goal of collapsing multiple packaging layers into a single, highly integrated structure.

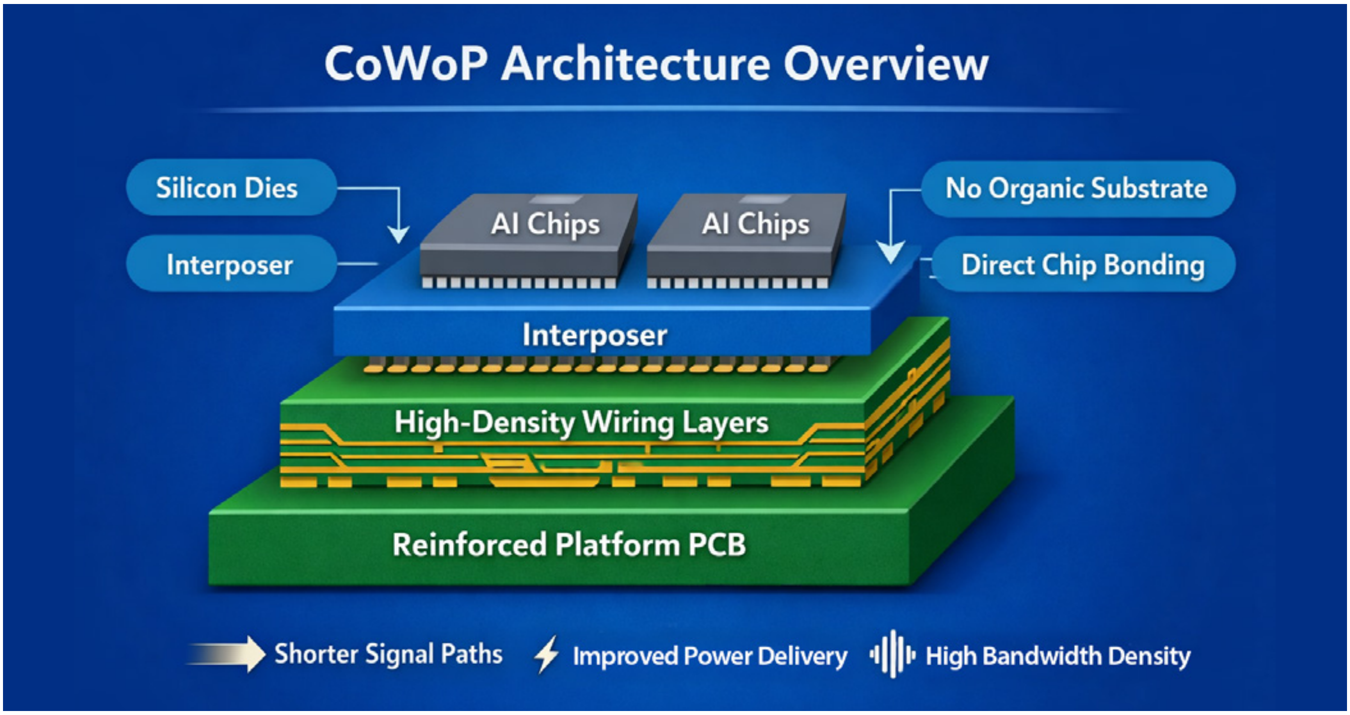


Figure 1: CoWoP architecture overview.

The investment is structured in two phases:

1. With approximately $100 million, phase one focuses on leased pilot production lines for CoWoP and mSAP technologies, supporting early-stage development and qualification.

2. The potential additional $200 million is explicitly conditional on technical validation results and market conditions. WUS has stated that the second phase may be canceled if performance targets are not met.

If fully implemented, the project is expected to add approximately 1.3 million units of high-density optical-electrical PCBs per year, with projected annual revenue of around $285 million. However, WUS has emphasized that mass production will only commence once processes meet full industrial qualification standards, underscoring a disciplined, technology-first investment strategy.

China remains the focal point for advanced materials, process development, and ecosystem density, particularly as AI packaging complexity continues to rise.

CoWoP vs. CoWoS vs. Traditional Substrate-Based Packaging

Traditional substrate-based packaging relies on organic substrates and BGA interconnects to connect silicon devices to system boards, offering mature manufacturing but facing increasing limits in bandwidth, warpage control, and power density. CoWoS (Chip-on-Wafer-on-Substrate) improves performance by integrating silicon interposers and high-bandwidth memory on an advanced package substrate, but remains complex and cost-intensive. CoWoP (Chip-on-Wafer-on-Platform) removes the substrate layer entirely, directly bonding silicon dies and interposers onto a reinforced platform PCB. This approach reduces interconnect length, lowers signal loss, simplifies stack height, and shifts more integration responsibility into the PCB domain—positioning advanced PCB fabrication as a central enabler of future AI system architectures.

Materials: The New Competitive Battleground

AI packaging scale is now colliding directly with upstream materials realities.

Low-CTE fiberglass cloth and high-frequency HVLP copper foil are experiencing structural supply tightness, with allocation behavior becoming increasingly common and lead times extending well beyond historical norms. Some CCL suppliers

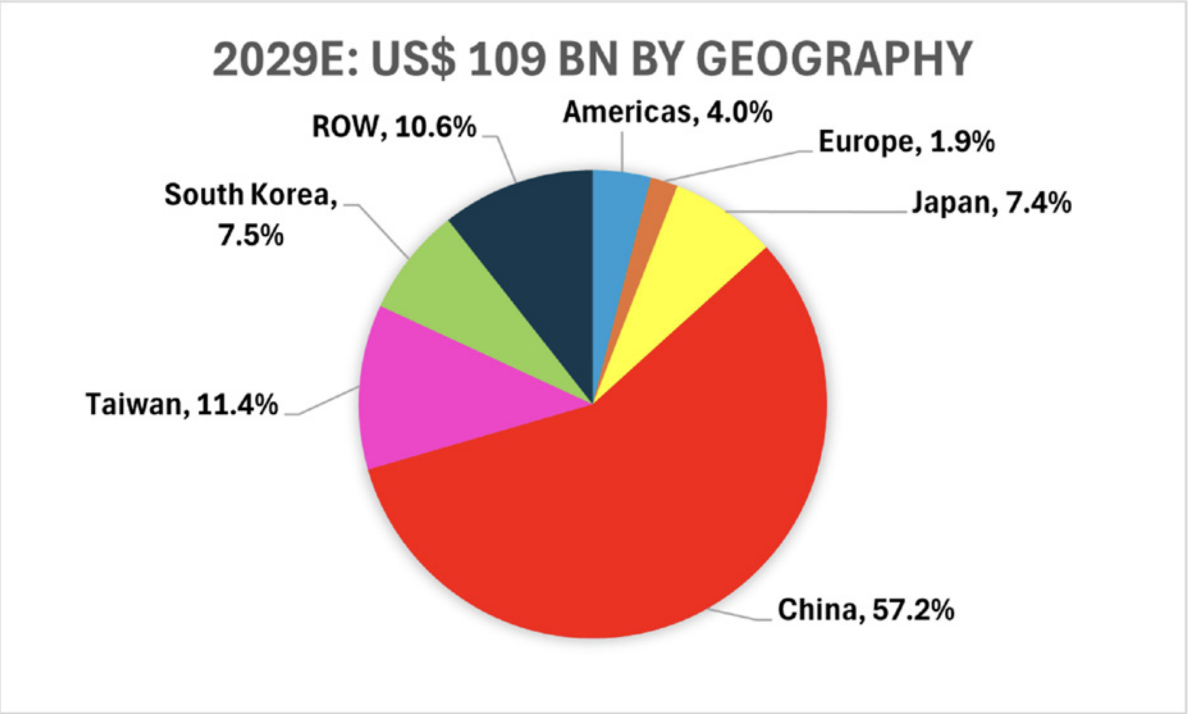


Figure 2: 2029E: Global PCB production by geography. (Source: PRISMARK 2026 data)

have begun freezing order changes to manage speculative AI-driven demand. Industry indications suggest that supply-demand imbalances for key materials may persist through 2027.

As a result, competition is shifting from pure capacity expansion toward control over critical materials and supply-chain resilience. Suppliers with long-term material access and strong bargaining positions are best placed to protect margins.

Is China Plus One Still Happening?

Yes, but not in the way the slogan implies.

The PCB industry is experiencing "China Plus Many," layered on top of "China Still." Southeast Asia is absorbing incremental capacity for risk mitigation and customer localization, while China remains indispensable for advanced technology, materials, engineering talent, and scale.

PRISMARK's latest projections reflect this reality. While global PCB output is expected to grow strongly through 2029, the combined China-and-Taiwan share of global production is now forecast to increase, rather than decline as previously expected.

In this environment, success will favor suppliers capable of operating across an interconnected global web—not those betting on a clean break from it.

Conclusion: "China Plus Many," Not "China Plus One"

Rather than a clean geographic decoupling, the PCB industry is evolving toward China Plus Many, layered on top of China Still. Southeast Asia is absorbing incremental capacity for diversification and localization, while China remains indispensable for advanced materials, engineering talent, and next-generation process innovation. Recent PRISMARK projections reflect this reality, with the combined China-and-Taiwan share of global PCB output now expected to increase rather than decline.

In this environment, long-term success will favor suppliers that can operate across an interconnected global web, not those betting on a simple or rapid exit from China. **I-CONNECT007**

References

1. "Don't just focus on memory! AI infrastructure is igniting a supercycle in PCB and CCL. Goldman Sachs: PCB and CCL are entering a super-cycle," by Chen Yanqi, Commercial Times, Jan. 22, 2026.
2. "AI behemouth! Goldman Sachs predicts explosive growth for CCLS, specifically those few the whole chain follows the rise," China Times, Feb. 1, 2026.

Manfred Huschka

THE END OF AN ERA

DoD Proposes MIL-PRF-31032 Cancellation

BY STEVE WILLIAMS, THE RIGHT APPROACH CONSULTING

The Defense Logistics Agency has initiated formal proceedings to cancel the military's primary performance specification for printed circuit boards, a move that could reshape how the U.S. defense industrial base qualifies and sources one of its most critical electronic components.

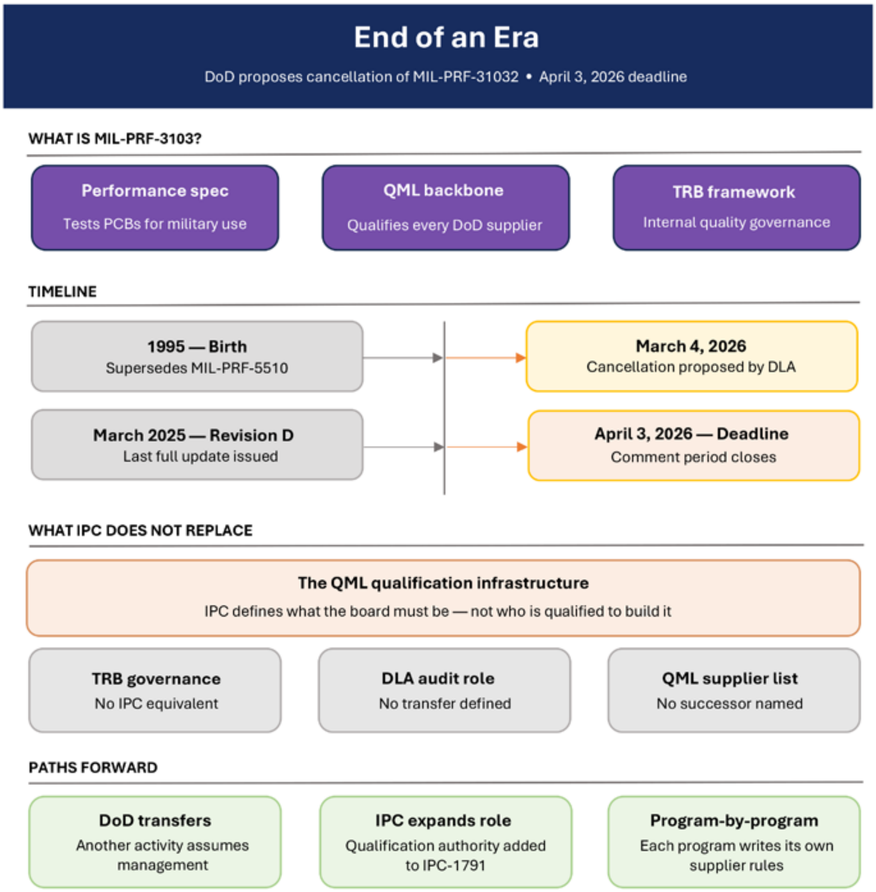
On March 4, 2026, DLA Weapons Support issued a memorandum to military and industry coordination activities announcing that MIL-PRF-31032, along with its six associated specification sheets, has been proposed for cancellation. A 30-day comment period was allotted, with concurrence or comments due by April 3, 2026. DLA Weapons Support has stated it will cease management of these documents in April 2026, and that any activity wishing to preserve the specification must formally accept transfer of management responsibility from DLA Weapons Support before that deadline.

The notice is terse and bureaucratic in tone, but its implications are anything but routine.

What MIL-PRF-31032 Actually Does

MIL-PRF-31032, *Printed Circuit Board/Printed Wiring Board*, General Specification For, has been the cornerstone of DoD's printed wiring board (PWB) qualification framework since it first superseded MIL-PRF-55110 in 1995. The specification establishes general performance requirements for all PCBs and PWBs procured for military use, and its associated specification (slash) sheets (1 through 6) address specific board technologies ranging from rigid multilayer boards to flexible and rigid-flex configurations used in high-frequency applications.

Critically, MIL-PRF-31032 is not a design guideline or quality standard; it is a performance specification tied directly to the Qualified Manufacturers List (QML) program. Every printed board produced for the DoD must come from a manufacturer certified under this specification. Certification requires successful completion of a rigorous qualification process administered by the DLA Land and Maritime Sourcing and Qualifications Division, including verification of the manufacturer's Quality Manage-



ment Plan by a Technical Review Board, demonstrated manufacturing capability, and ongoing lot acceptance testing. Manufacturers who achieve certification are listed on the QML, which defense contractors and military OEMs rely on when selecting suppliers for new programs.

In short, MIL-PRF-31032 is not a paperwork formality. It is the structural backbone of how the DoD ensures its PWB supply base is capable, verified, and trustworthy.

Why This Matters

The DLA has been understaffed for years, as most of the seasoned auditors have retired, and replacing the decades of tribal knowledge with new personnel has been difficult, if not problematic. Many of my clients have had their qualifications and ongoing certifications delayed for months due to this qualified resource shortage. The proposed cancellation comes amid domestic PCB manufacturing being a recurring concern for supply chain resilience and national security.

The United States has watched its share of global PCB production erode substantially over decades, with most commercial board fabrication now concentrated in Asia. The QML program, anchored by MIL-PRF-31032, has been one of the key mechanisms for ensuring that a known-good, U.S.-based defense supplier base is maintained and continually verified.

Without the specification or without a successor framework of equivalent rigor, several important questions arise. Will the QML for PWBs still exist, and who will manage it? What performance benchmarks will procurement activities use when writing contracts for military PCBs? How will legacy programs still citing MIL-PRF-31032 or MIL-PRF-55110 (which predates 31032 and remains in use on certain older platforms) be affected? Perhaps most urgently, what happens to manufacturers who have invested substantially in achieving and maintaining QML certification if the specification underpinning that certification is withdrawn?

The 30-Day Clock

The memorandum, signed by Muhammad Akbar, chief of the Active Devices Branch, makes clear that the comment period was short and the consequences of silence were significant. Lack of response by April 3, 2026, could have been construed as concurrence with cancellation. Military review activities were instructed to route comments through their custodians in time for consolidated departmental replies, meaning internal DoD coordination timelines are effectively even tighter than the 30-day window suggests.

For industry, this was the critical window to act. Manufacturers currently on the QML, defense prime contractors who specify QML-certified boards in their programs, and industry associations with equities in defense electronics have already submitted formal comments. The specification sheets—covering rigid multilayer, rigid single- and double-layer, flexible, rigid-flex, and high-frequency board technologies—impact a wide range of programs across all services.

What Comes Next

The memorandum leaves open one meaningful pathway: Any activity that determines there is a continued need to maintain MIL-PRF-31032 or its specification sheets may request a transfer of management responsibility from DLA Weapons Support. This suggests the cancellation is not necessarily permanent or irreversible, but it would require a willing and capable government entity to step forward and assume ownership.

Whether that happens will depend largely on how much of the defense establishment treats this notice as a bureaucratic formality vs. an urgent supply chain risk. Given the strategic importance of qualified domestic PCB manufacturing to virtually every weapons system in the U.S. inventory, the answer to that question deserves careful attention from program managers, acquisition professionals, and defense industry leaders alike.

One Forward Path: The IPC Equivalent Framework

Many military specifications have slowly been transitioning to IPC specifications for quite a while, so this might be the logical move forward. The IPC-6010 series already serves as the commercial counterpart to MIL-PRF-31032, and the mapping is fairly direct.

IPC-6013 Class 3 is generally accepted by government agencies as a commercial off-the-shelf equivalent to MIL-PRF-31032 for flexible boards. The same Class 3 performance tier logic applies across the 6010 series.

The Global Electronics Association also anticipated the defense use case with IPC-6012DS, a Space and Military Avionics Applications Addendum to IPC-6012D, which specifically addresses requirements for space and military avionics environments. It supplements or replaces identified requirements of IPC-6012D for rigid printed boards that must survive the vibration, ground testing, and thermal cyclic environments of space and military avionics.

So, the performance specification side of the transition is largely covered. The harder problem is everything else MIL-PRF-31032 does beyond just specifying board performance.

The Bottom Line

The performance requirements transfer to IPC cleanly. The qualification infrastructure—the QML, the TRB governance model, the DLA oversight role—does not transfer automatically and would require deliberate design by whoever accepts management responsibility for these documents, or by a DoD acquisition policy decision to formally adopt IPC standards with an accompanying qualification authority. Now the waiting begins. **I-CONNECT007**

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

MIL-PRF-31032 Sheet	IPC Equivalent
/1 & /2 (Rigid boards)	IPC-6012 (Qualification & Performance for Rigid PCBs)
/3 & /4 (Flexible/Rigid-flex)	IPC-6013 (Flexible Printed Boards)
/5 & /6 (High Frequency)	IPC-6018 (Microwave/RF Printed Boards)

Defense Speak Interpreted: Hypersonics Report Back After Six Years of Silence ▶

It's been six years since my Defense Speaks column about hypersonic weapons. Back then, these weapons were the most sought-after technology as there was little defense for them. They were the cornerstone of the "strike any location on earth within one hour" scenario. Of course, the war in Ukraine, and now the action in Iran, have grabbed the weapons headlines, but hypersonics still play a role and development continues. Here is the update.

Boeing-built Space Launch System Core Stage Powers First Crewed Artemis Mission around the Moon ▶

NASA's Space Launch System rocket, powered by the Boeing-built core stage, lifted off at 6:35 p.m. ET on April 1, 2026. Eight and a half minutes into flight, the core stage successfully completed its mission and separated from the upper stage of the rocket, enabling NASA's Orion spacecraft, Integrity, to carry humankind around the moon.

Renesas' Radiation-Hardened ICs Take Flight on NASA's Artemis II Crewed Lunar Mission ▶

Renesas Electronics Corporation, a premier supplier of advanced semiconductor solutions, announced its radiation-hardened (rad-hard) ICs are being used in NASA's Artemis II mission, which successfully launched from the Kennedy Space Center in Florida on April 1.

JRC, SKY Perfect JSAT, Sharp Partner on Defense Multi-Orbit Comms Project ▶

Japan Radio Co., Ltd., SKY Perfect JSAT Corporation, and Sharp Corporation will collaborate to develop and demonstrate a next-generation communications system in support of the Ministry of Defense (Japan) program titled "Development and Demonstration of Resilience-Enhancing Technologies for Multi-Orbit Communications Systems," for which JRC was selected on Feb. 4, 2026.

Learning with Leo: Why Risk-based Auditing Is Reshaping Cable Assembly Quality ▶

Effective product development requires early and active participation from all stakeholders. The design process must include input from supply chain, manufacturing engineering, purchasing, quality, and production. A design cannot be considered complete until it has been evaluated against the full manufacturing process, whether executed by an OEM or a subcontractor. Without this collaboration, designs are often released that are not manufacturable in real-world conditions.

Redwire Imaging Tech Supports Artemis II Mission ▶

Redwire Corporation, a global leader in space and defense technology solutions, announced that its advanced optical imaging and sun sensor technology will launch on board the Orion spacecraft as part of NASA's Artemis II mission, the first crewed mission for the Artemis program. Through contracts with Lockheed Martin, NASA's prime contractor for Orion, Redwire is responsible for the production and testing of the Orion Camera System for Artemis missions I-V.

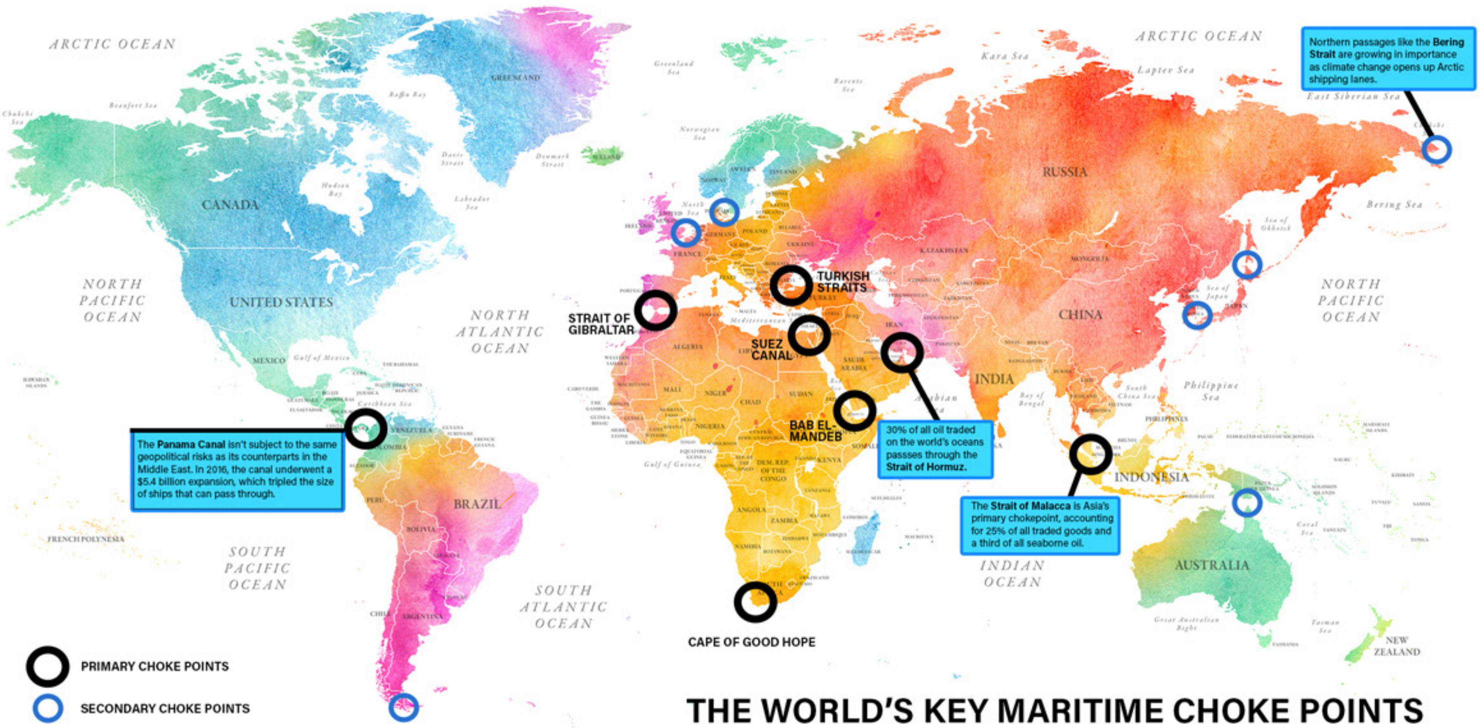
Commerce, Energy Departments Partner to Ensure Affordable Energy and Power America's AI Future ▶

The U.S. Department of Commerce (DOC), alongside the U.S. Department of Energy (DOE), announced a unique public-private partnership with SoftBank and AEP Ohio to redevelop DOE land, modernize energy infrastructure, and develop advanced computing in Southern Ohio. As part of the partnership, SB Energy, a SoftBank Group company, is planning to build 10 gigawatts (GW) of new power generation—including 9.2 GW of natural gas generation—that will connect to the local grid and provide power to a new 10 GW data center development at the Portsmouth Site in Pike County, Ohio at no cost to American families. These collective efforts will deliver lower electricity costs across the region, create thousands of American jobs, and strengthen America's national security.

EMI Promotes David Vue to Lead Military and Aerospace Division ▶

Express Manufacturing, Inc. (EMI), a global electronics manufacturing services (EMS) provider, announced the promotion of David Vue to Military and Aerospace Division Manager. Vue joined EMI in 2016 and has progressed through several roles of increasing responsibility, including associate manufacturing engineer, manufacturing engineer, and senior manufacturing engineer.

Rebuilding America's Military Stockpiles Begins With Microelectronics

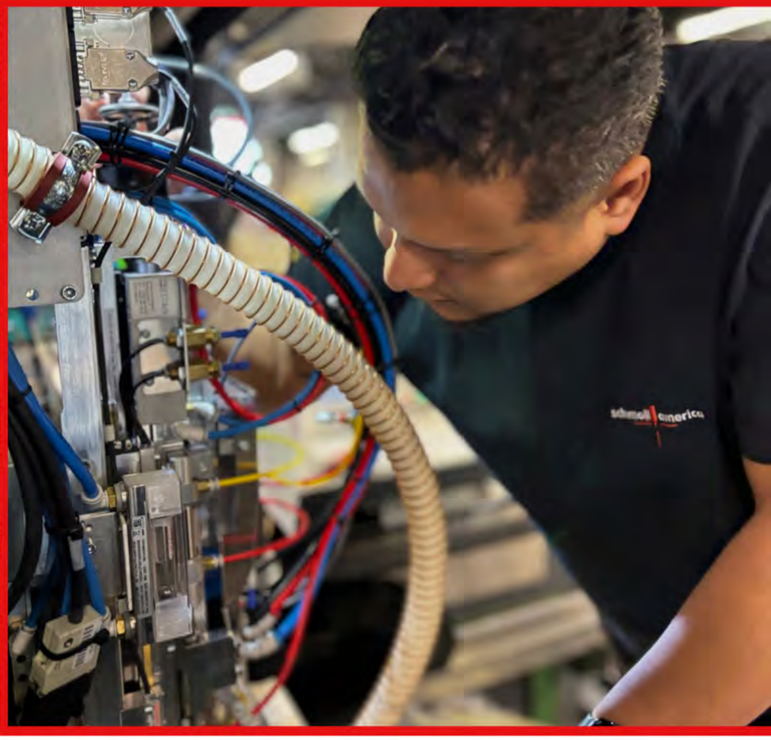


BY SHANE WHITESIDE, PCBAA

Current world events demonstrate the fragility of long-distance supply chains transiting multiple zones of conflict. The U.S. military is currently drawing down supplies of key munitions and other electronic systems at unprecedented rates.[1] Every one of those systems is powered by printed circuit boards.

The American PCB industry has kept pace with peacetime demand for the defense industry, but will now be called upon to increase production to a wartime footing at rates not seen in decades. The challenge is capacity. The companies America relies on to produce PCBs for defense applications continue to experience internal capacity erosion as HDI/UHDI demand increases. This trend is expected to accelerate as defense procurement of electronics increases. However, over the past three decades, America has been shuttering PCB factories, not building them. The result is very few PCB companies that are ready to expand production.

This problem has been addressed for other elements of the technology stack. The U.S. government made major investments in semiconductors, and rare earth and critical minerals. These investments are an important first step, but pale in comparison to the aggressive industrial policies of governments in Asia, where governments understand the importance of PCB manufacturing and



are investing billions in their capacity. China has sought dominance in PCB manufacturing for many years and has invested accordingly. In addition to China, Thailand, Vietnam, Malaysia, South Korea, and Japan are also investing significant amounts in PCB production.[2-6] The U.S. has not.

"For the past five years, PCBAA has sounded the alarm about foreign dependency. The U.S. can no longer ignore this threat to our national and economic security."

While the U.S. debates policy, our competition deploys capital. This imbalance puts production of American weapons systems and supporting material at risk. It will only be a matter of time before the supply of trusted and secure PCBs becomes a supply chain bottleneck.

For the past five years, PCBAA has sounded the alarm about foreign dependency. The U.S. can no longer ignore this threat to our national and economic security. To scale up, replenish the electronics needed for defense, and be ready for future conflicts, Congress needs to support American manufacturing through legislation like H.R. 3597, the Protecting Circuit Boards and Substrates (PCBs) Act, which calls for incentives to reinvigorate this industry.

Times of conflict remind us that the Arsenal of Democracy is more than a catchphrase. We can't predict what conflicts the future will bring, but we must make investments to ensure we are ready. Click here for more information. **I-CONNECT007**

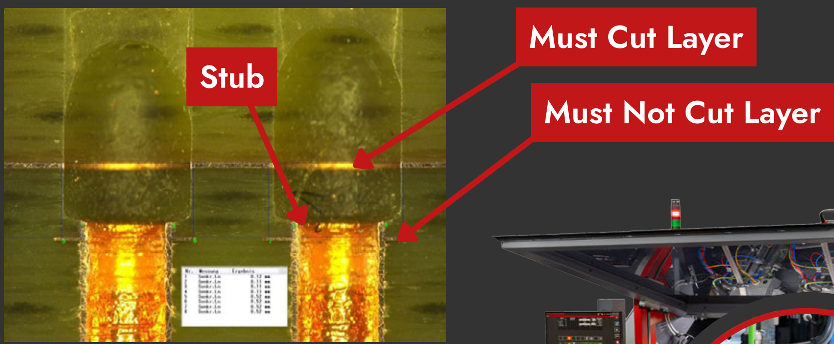
References

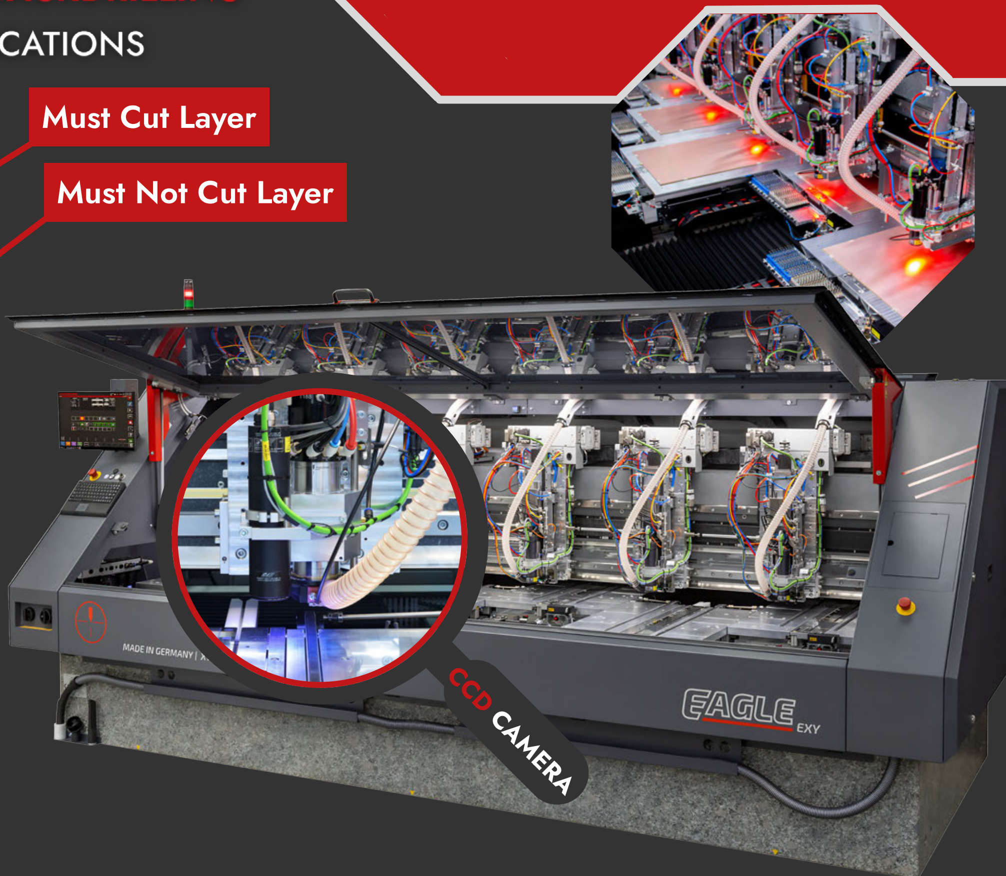
1. "US has burned through 'years' of munitions since start of Iran war," by Abigail Hauslohner, Steff Chavez, Lauren Fedor, and James Politi, *Financial Times*, March 12, 2026.
2. "Thailand approves $2 billion PCB manufacturing investment by joint venture led by Taiwan's ZDT," Reuters, Jan. 18, 2026.
3. "Thailand and Vietnam to Emerge as World-Class PCB Hub," NEPCON Thailand.
4. "Malaysia Strengthens Position in Global PCB Supply Chain With Semiconductor Investments Surpassing USD 25 Billion by 2035," Nexdigm Market Research.
5. "South Korea Printed Circuit Board Market Size & Share Analysis—Growth Trends and Forecast (2026-2031)," Mordor Intelligence.
6. "TPCA Analyzes New Competitive Landscape of PCB Manufacturing in China, Japan, and South Korea," I-Connect007, Dec. 11, 2025.

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



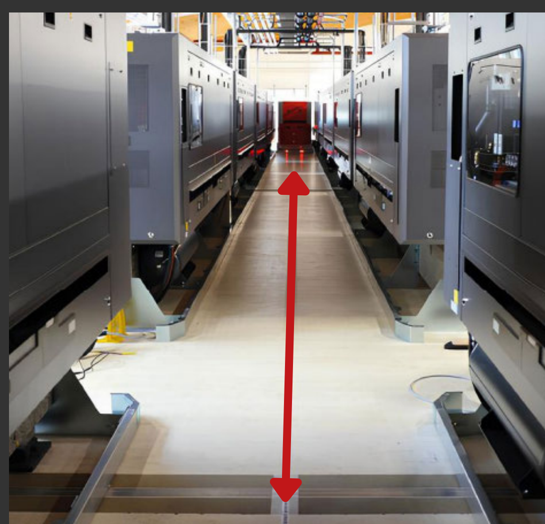


Back Drilling: stop of depth drilled hole before must-not cut layer within stub length tolerance

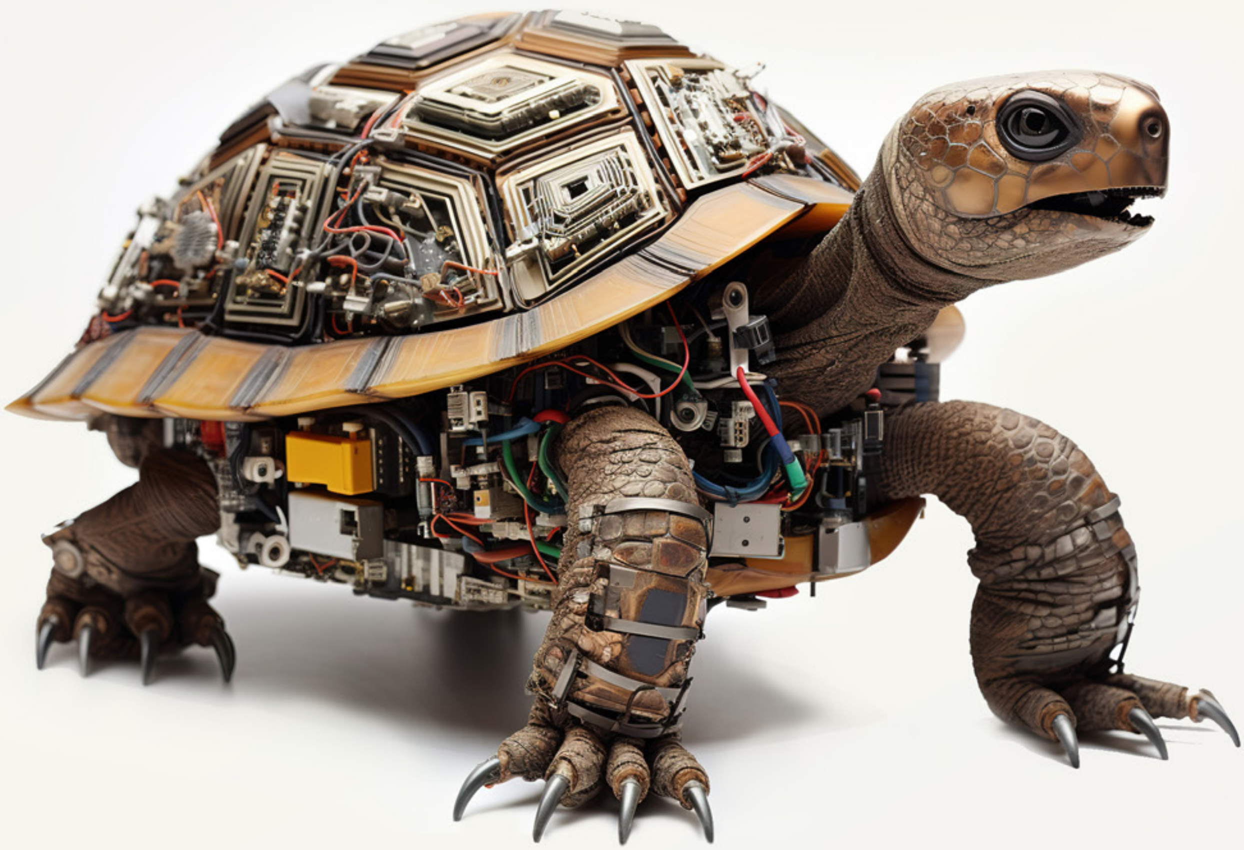
Depth Drilling: precise and reliable drilling to defined inner layers, enabling high-quality blind vias.

AUTOMATED COLLECT CLEANING (ACC)

AUTO LINE SHUTTLE (ALS) AUTOMATION



When the Industry Moves Faster Than the Standards



BY **DON BALL,** CHEMCUT

As a supplier of wet processing equipment, we have rules and standards we must adhere to, including both regional and national electrical codes and safety and environmental regulations, as well as myriad other standards to make the equipment safe to use. Things are a little different when it comes to rules and standards for manufacturing PCBs, though, because technical advances and requirements change so quickly that standards can't keep up. Because I've worked on several IPC standards subcommittees in the past, I know it takes a while to get everyone to agree on what the standards should be and then actually set those standards. Nowadays, by the time this has been accomplished, the industry has moved on.

Because of this, most suppliers are forced to be reactive rather than proactive. We hear about new requirements and processes mostly from our

customers that are trying to meet the demands of their customers. The typical sequence of events: A potential customer approaches a PCB manufacturer and says, "This is what we want to do. Can you do it?" The PCB manufacturer is understandably reluctant to give a potential customer a flat-out no if they are not sure, so they contact their suppliers and ask, "Can we do this with what we have from you?" Most suppliers are equally reluctant to tell an existing or potential customer no and are thus forced to determine what needs to be done to meet their customers' requirements.

"Partnerships imply exclusivity, and while that may be an advantage for a supplier of consumables (plating chemistries, etch resists, etc.) to a very large customer, it might not be for a supplier of capital equipment."

To cope with rapid development and change within the industry, it is invaluable to maintain a process development lab with a full-sized develop-etch-strip line (DES) that can also serve as a strip-etch-strip line (SES) or as a photochemical machining line (PCM) for etching steel, by just changing process chemistries. There are also a couple of smaller process lines that allow us to do development work with a much-reduced volume of chemistry.

Having an onsite process lab allows us to relatively quickly determine whether a new process can be run in existing equipment or if changes need to be made so that it can. This is immensely helpful when trying to cope with rapidly changing manufacturing requirements with no hard-set standards. It enables rapid response to customer demands, which, in turn, helps retain existing customers or attract new ones. In cases where there is no obvious solution or fix, it is possible to team with the customer to develop the equipment and chemistry needed to accomplish the required task at hand. The downside, of course, is that maintaining and staffing an R&D center is not inexpensive, and those costs must be added to the cost of the equipment. Still, the ability to team up with customers to quickly respond to new challenges has proven advantageous in the long run, even though it may have cost some short-term sales.

A Note About Partnerships

Forming partnerships with customers can be tricky, however. Partnerships imply exclusivity, and while that may be an advantage for a supplier of consumables (plating chemistries, etch resists, etc.) to a very large customer, it might not be for a supplier of capital equipment.

Many, many years ago (even before my time, if you can believe it), we had a development agreement with a very large computer company to develop equipment to etch printer bands for then state-of-the-art high-speed printers, which were unbelievably primitive by today's standards. Careful reading of the agreement revealed that any equipment developed for this process would be exclusively for that computer company and could not be sold to anyone else. Our lawyer caught that and got the exclusivity clause removed. That was fortunate because that equipment line became our mainstay throughout the 1970s, '80s, and '90s, and is still a major component of our sales today.

Even though we prefer not to use the term "partnership," so as to avoid the implied exclusivity, things can still get a little awkward, especially given the prevalence of non-disclosure agreements (NDAs) today. Even a nondescript boilerplate NDA that prevents you from disclosing that you are working with that company and what they are working on to anyone can be somewhat problematic.

Keeping Things Close to the Vest

For instance, a few years ago, two start-up solar cell companies contacted us about a new process

for solar cell manufacture that necessitated the cleaning of manufacturing residues off the back of the cell without anything touching the front side of the cell. The process was new to us as well, so we had no pre-existing wisdom to offer. Being start-ups, they had no access to equipment or chemistry needed to run tests. We offered to rent them both lab time and staff (me) to run tests under their supervision.

Our problem was that each company was only marginally aware of the other and only suspected that the other was working on the same process. The goal of both companies was to develop a process and sell it or the entire company to the highest bidder. NDAs were signed that allowed us to use any specialty equipment developed for this process for anything not related to this specific solar cell process, but the process developed during these tests belonged to the customer.

Company A came in first and, after a few visits and testing between them, we were able to develop a viable, patentable process. (It also didn't require any extensive equipment redesign on our part). Company B came in a few months later, and we provided them with the same initial set-up as we had for Company A. It soon became very frustrating as we could see they were clearly going down the wrong path, and we couldn't even give them hints that might point them in the right direction. Fortunately, a moral dilemma was avoided when company B decided the whole thing was a bad idea and disappeared from the scene, never to be heard from again.

So, even if forming customer/supplier teams to investigate new processes can be complicated, it is still a worthwhile endeavor and one way to cope with ever-changing process demands without any hard standards. Suppliers have probably seen more good, bad, and ugly ideas than any manufacturer can possibly imagine. Keeping good relations with your various suppliers can help you as standards change and also help the suppliers keep up with rapid changes in manufacturing PCBs.

I-CONNECT007

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

BY RICHARD NICHOLS, GREENSOURCE ENGINEERING

The Journey From Dilution *to Zero Liquid Discharge*

If you're familiar with the PCB industry, and a little long in the tooth like me, you may remember the cry, "The water board is here!" (or an equivalent authority). This was the signal for a frantic but regularly rehearsed exercise to turn on all the rinses. This anecdote demonstrates that in the early days of PCB production, prevailing practices revolved around a "dilution is the solution" mentality, in which manufacturers used copious amounts of water to dilute contaminants before discharging them into regulated municipal wastewater systems or natural water bodies.

This approach, while superficially addressing immediate discharge requirements, failed to consider the long-term environmental consequences of such practices, leading to growing awareness of pollution and its effects on public health and ecosystems. As the mg/L unit changed to mg/day, the tightening of industrial regulations to align with environmental concerns became paramount. These more stringent regulations led directly to more technological approaches to wastewater treatment.

In addition to the water regulations making significant wastewater investment obligatory, the increasing water costs fueled a water-saving mindset. This enforced investment without a viable ROI was coupled with potentially poor rinsing, which then needed to be addressed. That solution was water recycling. Initially, the industry stumbled through a series of inefficient, expensive solutions and ultimately could not produce water that supported the more sensitive, chemically based processes, such as electroless final finishing.

This misery was compounded by many of the chemical suppliers, who, as a precautionary measure, stipulated that water regenerated by reverse osmosis was not acceptable for use with their products. These growing pains left a distinctly anti-recycling mentality within the PCB and substrate industries. This remains prominent because it is deeply ingrained in most manufacturers' thinking.

In short, the cost, thinly veiled by quality, has always been the prohibitive disincentive to wastewater treatment and recycling.

But time marches on, technology improves, and

the global market for ZLD systems is currently anticipated to expand significantly. This trend is largely a response to increasingly stringent regulations governing wastewater discharge, compelling industries to implement advanced water treatment technologies to minimize liquid waste and maximize water reuse.

Unfortunately, ZLD has become a buzzword in the industry with misleading connotations. At first glance, it's self-explanatory, but in practice, it is often minimum liquid discharge (MLD), whereby the waste is rerouted to cooling or fume scrubbing or even shipped to a specialist company to process. Designing and maintaining ZLD systems can be intricate, necessitating expert knowledge and skilled labor. Water management will inevitably become part of the process rather than a backroom, low-skill activity.

GreenSource Engineering (GSE) is a pioneer in ZLD, having built a closed-loop system that recycles process water for reuse in production. A correctly sized system will solve your water consumption issue and environmental obligations whilst maintaining sufficient dilution requirements to ensure good yields.

For an effective new ZLD system in the field, the concept must start with through-factory water design (which is crucial for the ramp-up phase of production), a rinsing concept that considers drag-out and dilution factors, and separation of both concentrate and dilute treatment streams. An overview is represented in Figure 1.

In Figure 1, the concentrated stream is represented by the red number 1, and the dilute side is represented by the red number 2. The dilute side, in grey, will be discussed in a future article.

Experience has shown that approximately 30% of the used process water (UPW) is from chemistry maintenance and low-flow rinses. The primary function of the concentrate side is to carry out rough filtration and be the first step in the removal of bulk dissolved and suspended solids.

This step is present in all traditional water treatment systems and is simply an updated technological equivalent. The follow-up function for the concentrated side is to extract membrane-treated water, which is mixed with the dilute UPW and processed by the dilute side to start generating process water.

This step simultaneously reduces the volume

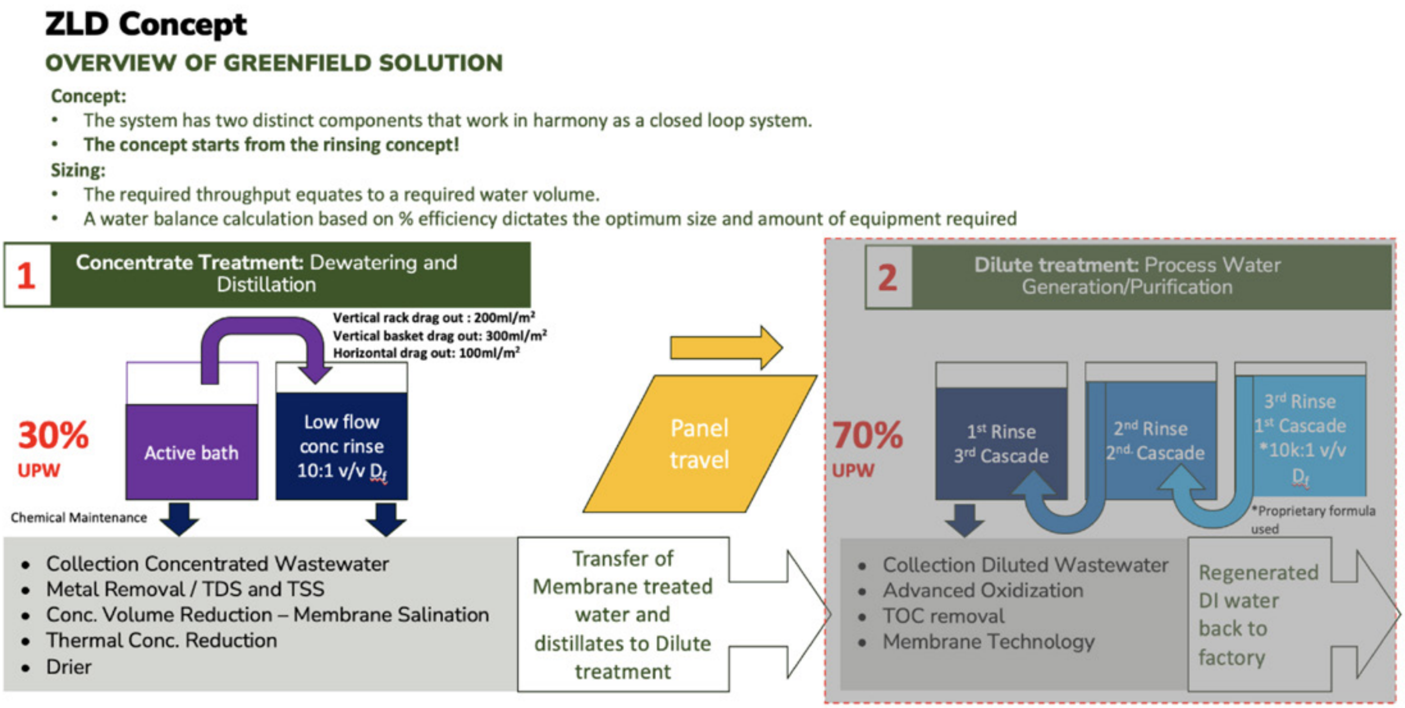


Figure 1: An overview of the major components to design a ZLD, closed-loop, water recycling system.


Panasonic
Multi-layer & Flex Laminates
peters
Printed Circuit Coatings
CIRCUIT FOIL
Electro-deposited Copper Foils
KB
KINGBOARD
Rigid & Multi-layer CCL
PERFECT POINT
PRECISION CARBIDE TOOLS
Drills & Routers
JINZHOU
3.407 CpK (±3 mil)
APEX
ALUMINUM
COOLSTRIKE
LUBRICATED BACKUP MATERIAL
Drills Entry & Backup
IRON MOUNTAIN
PCB Press Plates & Tooling
EuroAssist
PRODUCTS FOR PCB LAMINATION
PCB Lamination Assist Products
Hakuto
Dry Film Resist Lamination
AE
ADVANCED ENGINEERING
PCB Automation Solutions
and more....
MATRIX
CIRCUIT BOARD MATERIALS
Supplier to the North American Circuit Board Industry since 1977
www.matrixusa.us
1-800-668-5447
TORONTO • SANTA ANA • SANTA CLARA • CHICAGO • MINNESOTA • OSAKA

Figure 2: An overview of the major components that define a ZLD, closed-loop, water recycling system in a fab.

of the concentrated effluent. In turn, it reduces the investment required for the evaporation and drying steps, which are physical steps that are very robust when compared with membrane technology, and that can be a significant CapEx with a significant OpEx if not managed well. To be a ZLD system, dry waste must be created to be removed offsite for landfill. A moisture content of below 10% is specified in India.

Sizing a ZLD system is probably the most critical engineering step. GSE has a proprietary calculation sheet for this and works with customers directly regarding their daily water usage.

Case Study

For a modern wastewater treatment system capable of handling the demands of a PCB fabrication facility with a water consumption of approximately 500,000

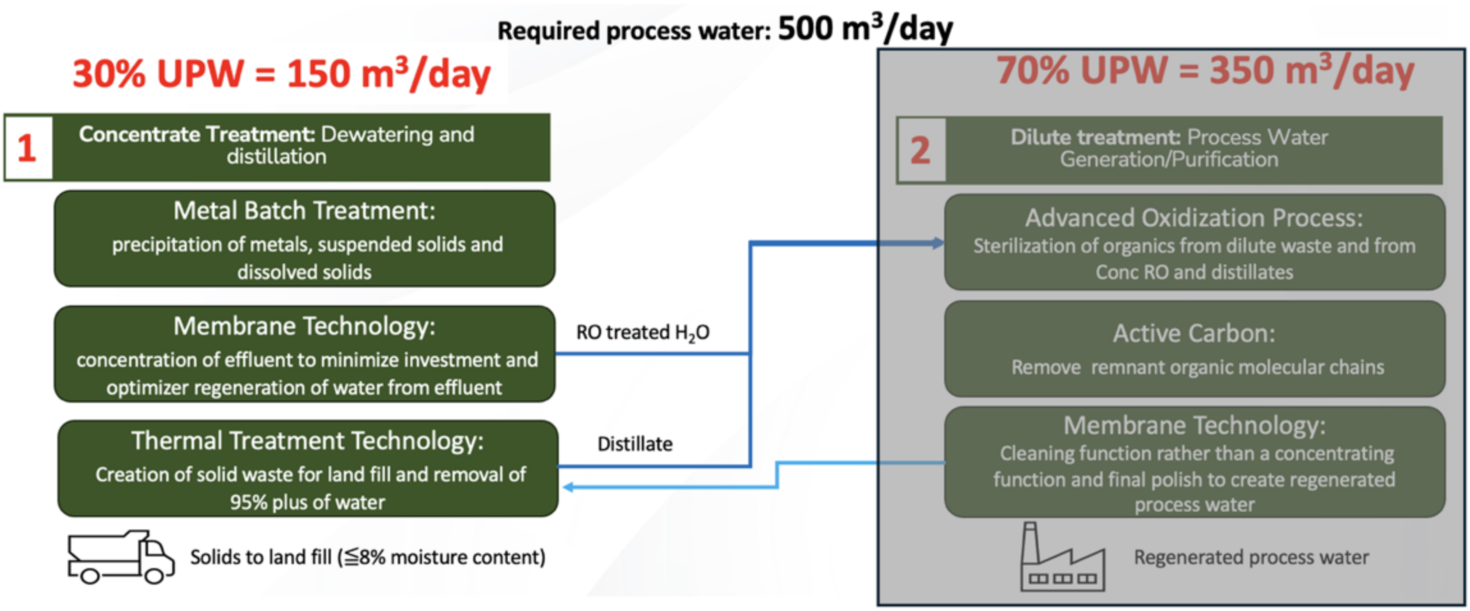


Figure 3: The planning of a 500m3/day ZLD system.

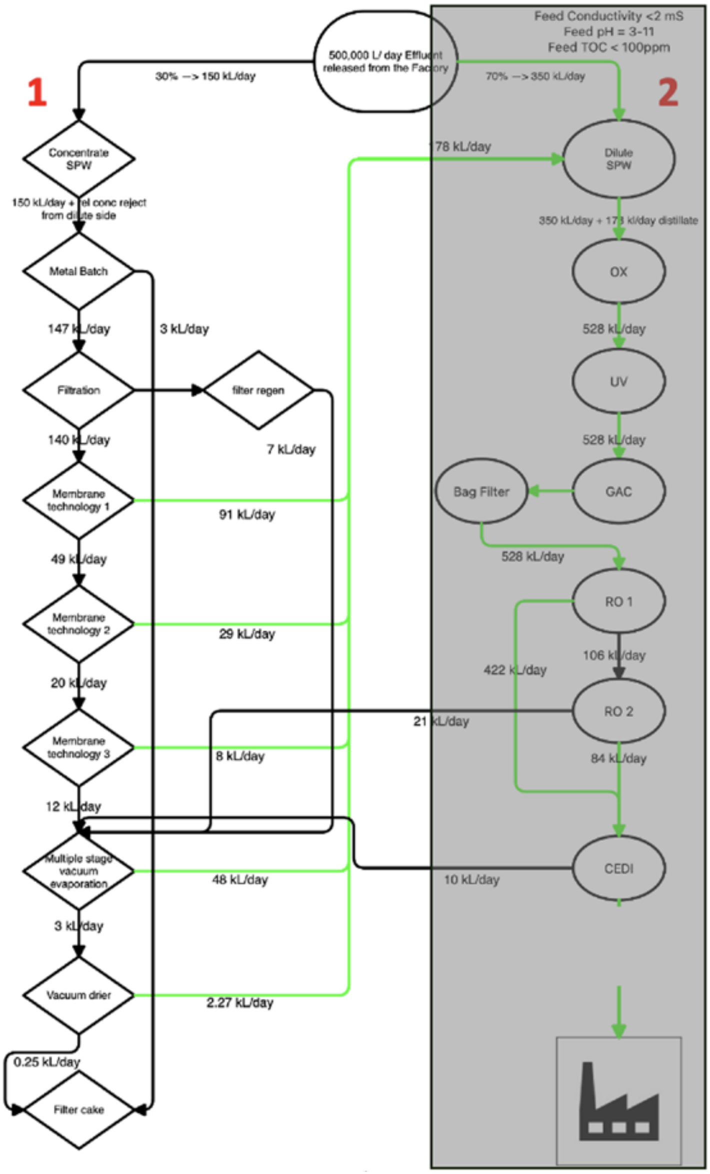


Figure 4: A flow diagram to visually evaluate the sizing of a closed-loop, water recycling, ZLD plant.

liters (or 500 cubic meters) per day, the investment required can vary significantly based on the complexity and technology employed. A traditional wastewater system designed for such a capacity will cost approximately $2.5 million. This figure includes costs for equipment, installation, regulatory compliance, and system integration. This is a significant investment for a finite process.

In contrast, a comparable ZLD system for the same facility could require an investment of approximately $3 million to $4 million due to the additional technologies and processes needed to achieve zero discharge.

It is worth noting that a ZLD system is not a water-saving scheme for a fab; it is an insurance of good rinsing, which has a significant impact on yield.

The layout for a system designed to regenerate 500 m3/day can be expected to follow the same modeling as previously discussed in this article, i.e., 30% will be concentrated UPW and 70% will be diluted UPW.

A water balance flow chart is created to verify the quote calculation and ensure the equipment is correctly sized.

Conclusion

As industries increasingly prioritize water sustainability, compliance with environmental regulations, and efficient resource management, ZLD systems are expected to be integral to future industrial water management strategies. The ongoing advancements in technology and growing regulation will further drive the adoption of these systems, ensuring a more sustainable industrial future.

The ZLD systems market is witnessing notable innovations. There is a swift uptake of state-of-the-art membrane technologies, including reverse osmosis and ultrafiltration, to enhance treatment efficiency. Compact and modular ZLD systems are being developed for small and medium-sized manufacturing facilities, providing flexibility and scalability. The incorporation of digital monitoring and automation technologies is enhancing operational efficiency and reducing downtime.

Critically, for the concentrated treatment stream, technological advancement and increased investment are being directed toward more energy-efficient evaporation and crystallization technologies, helping to mitigate some of the energy concerns.

Stay tuned for the next installment of this article series to get the full picture. **I-CONNECT007**

Richard Nichols is technical marketing director at GreenSource Engineering.

SELECTING THE RIGHT Laser Source

BY SIMON KHESIN WITH STEFAN RUNG, SCHMOLL MASCHINEN

When I first joined Schmoll Maschinen, I brought experience from almost every PCB process, except for laser. As I immersed myself in laser processing, I realized why it can seem so daunting to a newcomer. The complexity arises from three intersecting factors:

- **A vast variety of laser sources:** CO_2, UV-nano, green-pico, UV-pico, IR-pico, and others
- **A diverse range of applications:** Drilling, cutting, ablation, and more
- **An extensive list of materials:** These have vastly different absorption rates

Choosing the right machine or laser source is rarely trivial. Even for experienced engineers, answering "Which source is best?" requires examining the business's specific goals. Stefan Rung and I have written this article to help simplify that decision-making process, covering the most common scenarios from the perspective of someone building their knowledge from the ground up.

Intro to Laser Sources

In laser material processing, the laser source is only half the story. The other half is time. When we talk about nano, pico, and femto, we are discussing pulse duration, meaning the amount of time the laser beam actually touches the material. Understanding this distinction is the difference between a burned via and a surgical via (Figure 1).

1. **Microsecond (10^{-6} s): The Industrial Workhorse (CO_2)**

Most CO_2 lasers used in PCB drilling operate in the microsecond range.

ARK™

AQUA REGEN KINETICS

GREENSOURCE
ENGINEERING

A closed loop Zero Liquid Discharge (ZLD) water recycling process.
Another example of GreenSource Engineering's core competencies.

GREENSOURCE
ENGINEERING

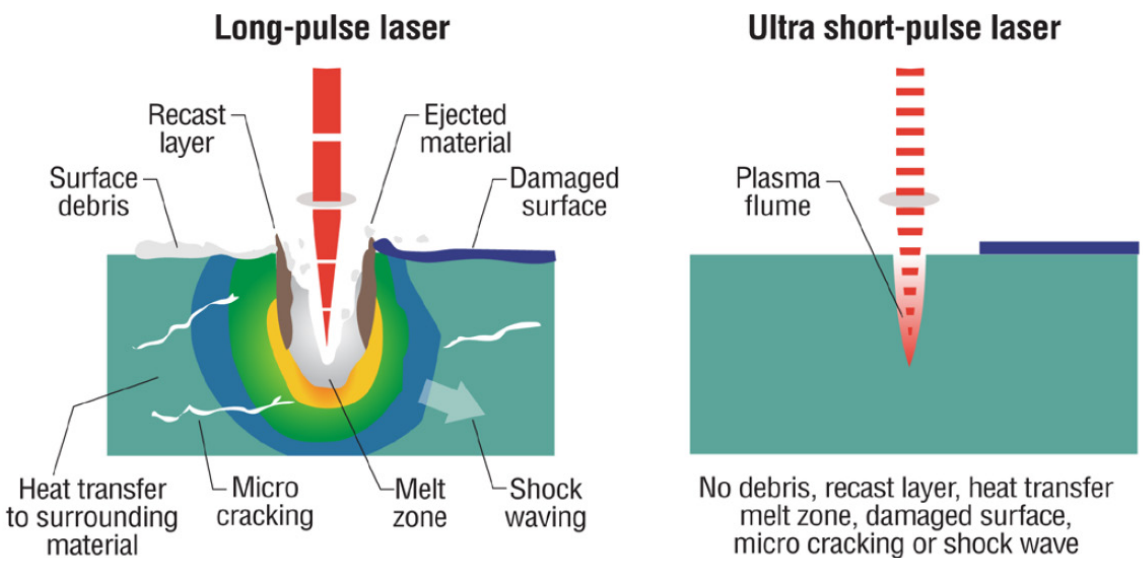


Figure 1: Comparison of long pulse and short pulse lasers.

Mechanism: This is an almost purely thermal process. The pulse is long enough to vibrate the molecules of the dielectric (resin/glass) until they reach their boiling point and vaporize.
Result: High volume removal. It is the fastest way to drill a via, but because the pulse is long, heat has time to spread. This creates a heat-affected zone (HAZ) and requires a copper "stop" layer because the laser cannot easily ablate copper.

2. **Nanosecond Lasers (10^{-9} s): The Thermal Process**

Nanosecond pulses are the traditional solid-state workhorse of the industry.

Mechanism: Photothermal. The laser heats the material until it reaches its boiling point, at which point it evaporates.
Result: Because the pulse is relatively long in molecular terms, heat spreads into the surrounding resin and glass. This creates a HAZ, which can lead to charring or carbonization on the sidewalls.

3. **Picosecond Lasers (10^{-12} s): The Cold Ablation**

Picosecond pulses are significantly faster; they are so fast that they change the physics of the cut.

Mechanism: Mostly photolytic. The pulse duration is shorter than the "thermalization time" of the material. The laser breaks molecular bonds before heat can travel to the next molecule.
Result: Often called cold ablation. The edges are incredibly clean, there is almost no HAZ, and carbonization is nearly eliminated.

4. **Femtosecond Lasers (10^{-15} s): The Ultrafast Frontier**

Femtosecond pulses represent the current peak of industrial laser technology.

Mechanism: Non-linear absorption. Energy is delivered so instantaneously that even materials normally transparent to light are forced to absorb the energy and vaporize.
Result: Perfection. Precision is at the molecular level, allowing for features so small and clean that they are difficult to see even under a high-powered microscope.

In current PCB manufacturing, CO_2 and UV-nano remain the industry standards for high-volume production. However, as designs push toward the miniaturization of features and the use of sensitive materials, picosecond (pico) technology is increasingly adopted to achieve cold ablation results. While femtosecond lasers represent the pinnacle of precision, they remain a rare case in the PCB industry, reserved for highly specialized R&D or ultra-high-end semiconductor packaging.

Which Wavelength Should I Choose?

While the terms microsecond, nanosecond, or picosecond describe the pulse duration, the color

of the light is the second main parameter making the choice of the correct laser source important. Laser sources have discrete wavelengths, which have a huge impact on the desired application. The wavelength, i.e., the "color" of the light, determines the absorption behavior of each material. We've learned this from our daily lives: If materials absorbed different wavelengths homogeneously, our visual impression of the world would be a greyscale between black and white.

However, for most of us, the world is colorful. Due to the reddish appearance of copper, we see that red is predominantly reflected at the surface while the other colors of our spectral eyesight are absorbed more strongly. From the perspective of a laser application, we prefer to use wavelengths outside the red spectrum to deposit laser energy for removal purposes, such as green or blue light. On the other hand, we can use the reddish spectrum to prevent copper from absorbing the laser energy and being damaged by it. By selecting the appropriate laser wavelengths, we can directly influence the targeted removal and protection of different materials. The most common laser wavelengths utilized in the PCB industry are:

- 355 nm (UV)
- 532 nm (Green)
- 1064 nm (IR)
- 9600 nm (IR/CO_2)

For each application that requires laser processing, selecting the laser wavelength is important. This makes it mandatory to consider the entire application spectrum of current and future materials before deciding the laser wavelength.

Drilling Methods: Navigating the Possibilities

Laser drilling involves several distinct methods for creating holes. The choice depends on the target diameter, the material stackup, and the required pulse diameter.

- **Trepanning:** The laser beam follows a spiral path to cut out the hole. This is used when the desired hole is larger than the beam diameter (Figure 2).
- **Pulsing:** Also known as "percussion drilling," where the beam diameter matches the desired hole diameter, allowing the via to be created in one or more pulses without moving the beam (Figure 3)

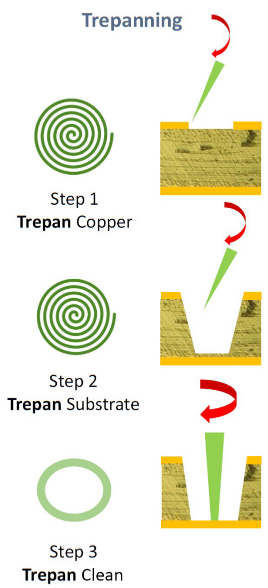


Figure 2: Trepanning process.

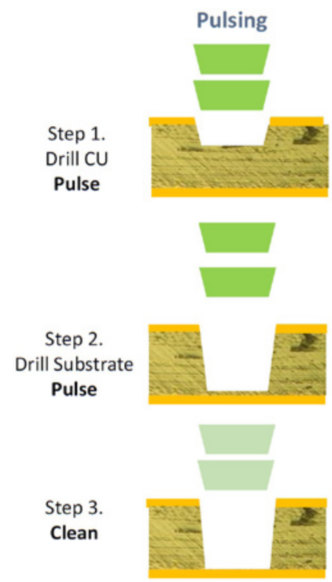


Figure 3: Pulsing process.

- **The Combi Method (Figure 4):** A multi-step process typically involving:

1. UV-nano or pico-green to trepan (ablate) the top copper layer.
2. CO_2 to pulse and remove the dielectric material.
3. Optional: A de-focused UV or pico-green pass for "bottom cleaning" to ensure a reliable connection to the inner layer.

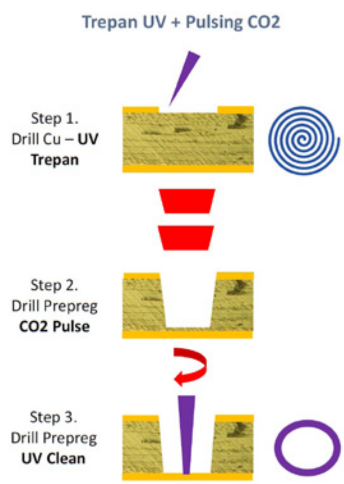


Figure 4: Combi process.

There are two methods of direct CO_2 drilling, which differ from one another only in the first step.

- **Direct copper pulsing:** Uses only a CO_2 laser. Since CO_2 cannot ablate shiny copper, the copper surface is chemically treated (blackened or browned) to increase absorption

and thin it down so the CO_2 can penetrate it (Figure 5).

- **Conformal mask:** Copper is either pre-etched or the process is used on panels where no copper exists on the top layer (common in build-up processes) (Figure 6).

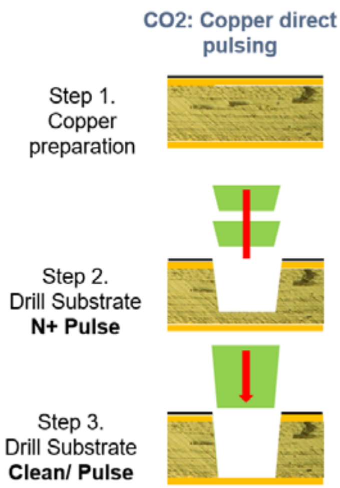


Figure 5: CO_2 copper direct pulsing process.

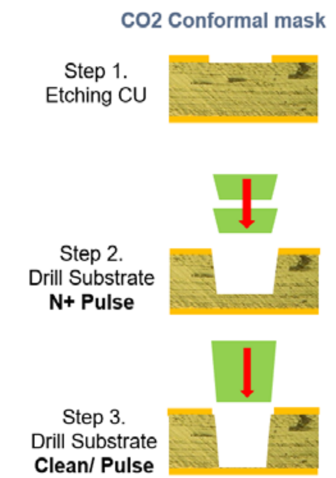


Figure 6: CO_2 conformal mask process.

When Should You Use Which Method?

For prototyping and mid-size production: The Combi process is the gold standard for flexibility. It combines the high CO_2 productivity (which naturally stops at the target copper pad) with the precision of UV or pico lasers (which can ablate copper and achieve smaller diameters).

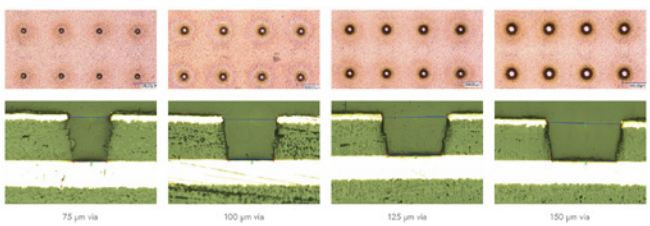


Figure 7: Blind vias drilled by Combi process.

For mass production: Standard mass-market HDI often utilizes pure CO_2 configurations to maximize throughput on pre-defined stack-ups.

For ultra-small vias (<75 microns): CO_2 becomes physically limited at these sizes. One must switch to UV-nano, pico-green, or pico-UV. These sources offer the precision needed for microvias but require careful energy management and process fine-tuning (Figure 8):

- **Risk of low energy:** Not reaching the target copper pad (open circuit)
- **Risk of high energy:** Penetrating through the target copper (damage to the inner layer)

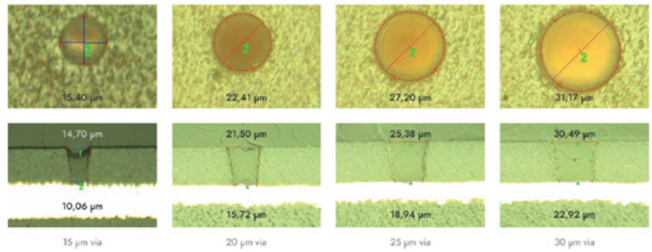


Figure 8: Good quality ultra-small blind vias drilled by ultra-short pulse lasers.

Cutting Applications

Cutting performance is strictly material dependent. While any laser can cut, the best match ensures the cleanest edge and fastest speed:

- **Flex materials:** UV nano is a classical solution or pico-green lasers for minimizing the HAZ/carbonization and achieving the clean edge
- **PTFE:** CO_2 or pico-green lasers have the best absorption threshold for PTFE
- **Ceramics:** Pico is often the best match for brittle ceramic substrates

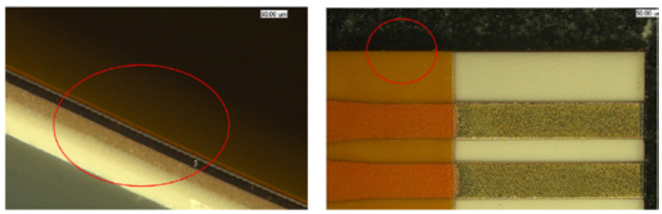


Figure 9: Cutting of flex materials by ultra short pulse lasers.

Ablation (Depth Routing by Laser)

Depth routing with a laser is used, for example, to reach contact pads on inner layers.

- **Constant thickness:** If the layer to be removed is uniform and you need to protect the layer beneath, an ultra-short pulse laser is ideal for cold ablation.
- **Variable thickness:** If the layer to be removed is dielectric and its thickness is inconsistent, a CO_2 or IR laser is better, as these sources are reflective of copper, which acts as a natural safety stop.

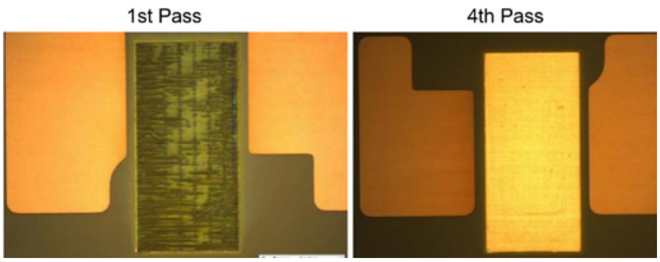


Figure 10: Ablation by ultra short pulse laser to reach the contact pad on inner layer.

Combination of the Applications

In high-volume environments, machines are usually dedicated to one task. However, in prototype and mid-series shops, the goal is often versatility.

When making a decision, you must prioritize:

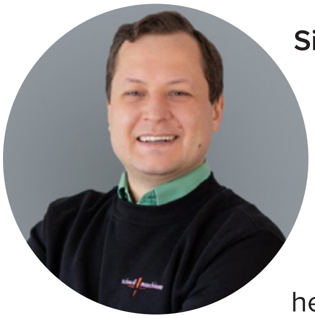
1. Budget and space: Can one machine do the work of two?
2. Technological margin: Do you buy for today's needs or tomorrow's requirements?
3. The "good enough" factor: A pico-green + CO_2 combination is the most flexible machine configuration, but is it overkill for your specific product mix?

Summary

The choice of a laser source is a business decision as much as a technical one. The answer lies in your priorities, so ask yourself:

- Should I spend more now for a technological margin in the future?
- Which specific PCBs bring the highest margin to my company, and which laser do those boards require?
- Universal machine or dedicated specialist?

We have examined the most common and critical use cases for laser micro-machining. However, in a real-world manufacturing environment, variables are far more diverse. Every project presents a unique combination of materials, specialized applications, and demanding process requirements, such as extreme aspect ratios (AR), specific taper control, or unique blind hole geometries. Furthermore, the success of the laser process often depends on the synergy between pre- and post-processing steps, which must be carefully calibrated to ensure final product integrity. **I-CONNECT007**



Simon Khesin is the key account manager at Schmoll Maschinen GmbH. Stefan Run is technical director for laser systems at Schmoll Maschinen GmbH in Germany, where he focuses on the development and industrial implementation of advanced laser processing technologies.

To read past columns, click here.

BY MARCY LARONT, I-CONNECT007

Jiva Soluboard

Getting the Attention It Deserves

Jiva is a newer company that bridges the divide between PCB fabrication and product circularity or sustainability. Jiva Soluboard is the first fully recyclable laminate material ever created for PCB fabrication, and it's not going unnoticed. Stephen Driver, CEO of Jiva, gave us an update at APEX EXPO, including an exciting certification achievement in February.

Marcy LaRont: Steve, we last chatted at productronica, but I have been following Jiva and Soluboard for about two years now. How did Jiva come about?

Steve Driver: The original story started with Jack Herring, who invented the product as part of a dissertation project. When he and I met, he had a proof-of-concept and asked me to help him commercialize the product. In 2021, we took it to a minimum viable product with a single-sided mouse for Microsoft, and they validated its functionality, motivating us to continue. We've evolved the product since then. In 2022–23, we developed a plated through-hole version of the material, and more recently, we have been focused on obtaining UL certification for it. We are now in the validation and certification phase and working toward a successful multilayer build. We achieved UL94 V-1 last year. We are very active in the Information and Communication Technology (ICT) sector, where these applications require UL V-O, and I am pleased to announce that we achieved this in February. That's a big milestone for us and potentially a game-changer.

Congratulations! What other products are you currently working on?

I can't say too much because of our non-disclosure agreements, but we are working on some computer peripheral products with several OEMs. We were part of the Amazon accelerator, for example, which was great for us. They want to stay engaged with us, and, of course, several big companies are interested in our progress toward multilayer capability and certification. We are working on a four-layer design that we hope to start on in late Q2 or Q3. OEMs in the LED markets in the UK and greater Asia are also interested in what we have to offer. We also have some industrial applications specifically related to materials handling, and this year we will go to scale. It's all very exciting for us.

What do you mean by going to scale in this example?

For production volume, that would be 200,000 to 300,000 square meters annually on the first line, with plans for the second line to produce 1 million square meters annually. The first line is 600 meters wide, and it's a continuous process. The production line will be 1,500 meters wide, increasing capacity by three times.

That's not big by standard laminate standards, but it's a significant amount of laminate to produce.

Especially when you're talking about a brand-new product and lines. It sounds like, surprisingly, you've had solid reception from the OEM sector. How has the education process been?

Yes, we've been up against all the naysayers. Many in the U.S. and the UK told me, "Steve, that will never work," but it is working. We have many good people on board and on our advisory panel, including Alun Morgan, Emma Hudson, and Murali Sethumadhavan, who was formerly Innovations Director with Rogers. Our strategy is similar to Rogers', and, like most laminate suppliers, we are OEM-driven. We are framing Soluboard as an advanced material, meaning it's a specialty laminate that needs to be spec'd in on the drawing by the designers and OEMs.

Whilst our customers might be PCB fab shops and CEMs, the specifier must be an OEM. All our work has been OEM-driven educational programs, just like here at the Global Electronics Association. We're trying to influence and educate, which is one of the hardest parts. We've been in a different country every month for the past two years, presenting at various seminars, conferences, and workshops.

It's a really difficult challenge because there isn't an IPC spec sheet or a material slash sheet yet for Soluboard. We are a true disruptor. We've got to get the specs caught up, so there's a lot of work to be done. We are relying on AABUS (As Agreed Between User and Supplier) relationships. It's built on a lot of trust now.

Because Jiva is a young company still in early development phases for your product or products, how are things on the investment side?

Yes, we are also investment-driven. Our customers, investors, and stakeholders want us to achieve commercial traction as soon as possible, which will require double-sided boards before we spend much more money developing a multilayer product. But it's good, and exciting. We do believe that people genuinely want this product.

Steve, you are, no doubt, following the stories developing on the glass fiber shortage and copper and base laminate price increases affecting the supply chain. What does that mean for Jiva?

Yes, I have, and I believe this is our moment. The focus of all the laminate guys is on the very high end: the AI servers, the 40- to 100- -layer boards, and military and aerospace. Those markets can take those price increases.

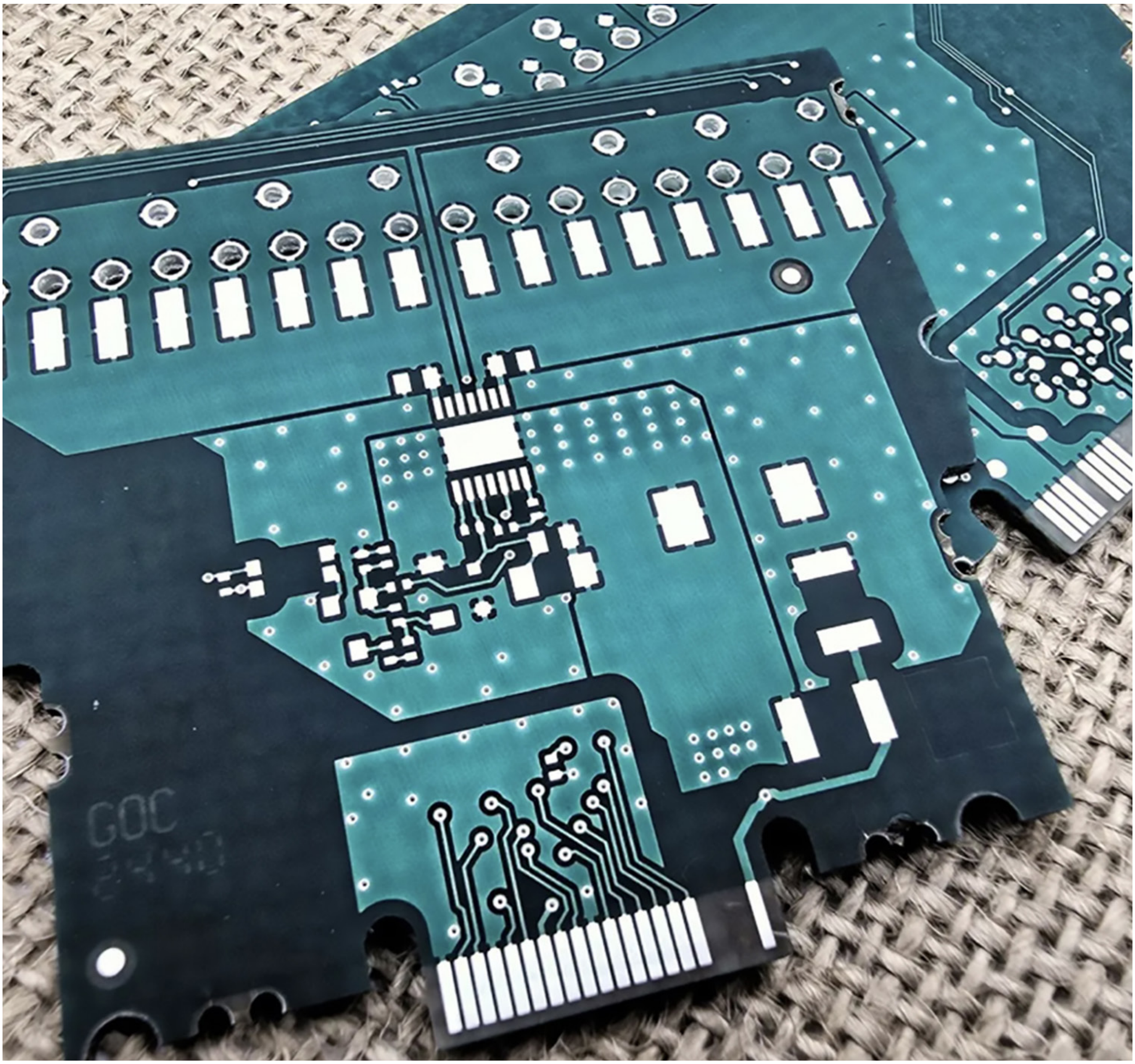
The guys at the bottom of that food chain—the consumer market, two-layer boards—is the market we are looking to serve, and our time is absolutely now. We always said we would be able to enter the market at parity with standard FR-4 pricing. We be-

lieve we can now be lower-priced than FR-4. With Jiva materials, there is no green premium, but there could be green savings. That's where we think we will be.

There have been roughly 12 huge announcements from the different glass suppliers saying, "We are no longer supporting the low-end material." They are focusing their efforts on the product that returns the highest margin. But there is still a lot of rigid product that's not as sexy, still needs to be made, and needs to be made in volume. They won't be able to push big price increases on the consumer market. The average shopper will look for alternatives, and hopefully, the alternative will be green. There is a lot of pressure on OEMs to be greener. In Europe, there is a requirement for an Environmental Product Declaration (EPD), which is essentially a product passport that reports a product's environmental credentials and defined waste streams.

Yes, the legislative requirements for sustainability and reporting in Europe have become extensive.

I recently gave a paper on the life of a disruptor. This industry was massively disrupted in 2006 with



lead-free directives. If a similar directive comes in now about carbon emissions, we might have a better chance at success. We aren't there yet. At present, the Scope 1, 2, and 3 directives are just about data collection, whereby manufacturers are just being asked to report their emissions.

Of course, we know that's a precursor to legislated limits at some future point.

Exactly, it's a matter of time, but I believe it's coming. Working with OEMs, many now include sustainability or green statements in their mission statements or annual reports. They are really committing to it. They've got a dedicated team of people working on sustainability initiatives. They're the folks we want to work with, where we can see there's a commitment.

They are the ones willing to experiment, and they're being forgiving. When it doesn't work, they're giving us feedback because they want to see it be successful. That is how we've been able to improve the material.

We are in a nice position. We've got about 40 live projects, partnering with good OEMs, and some of them very large and well-known. One of those could change the trajectory of Jiva very quickly if they engage at scale.

"Every decision is important, and there is much accountability. I am confident Soluboard has a future in our industry."

It sounds like your whole team has a lot to be proud of and to look forward to. Your work is important and must be incredibly gratifying. So, what's next?

Well, it's time to ask for more money. Our investors have been very happy and patient. We've raised around $5 million to date, and I am quite proud of what we've achieved with a relatively modest investment. I've been counseled on what to ask for now; in this next fundraise we are trying to raise exactly what we need and what will sustain us until we have good commercial traction. We have a lot of work to do and many OEMs in support of it.

You mentioned giving a presentation on being a disruptor. In my opinion, we need many more of them. Give me a glimpse of that journey.

I have always enjoyed stepping up when it was said that something couldn't be done. I have always liked doing the stuff other people can't do. You look for those niches that make you tough and resilient. I say yes to everything. I'm not so motivated by money as I am by getting things done and making a difference.

When I initially started working with Jack and Jiva, I didn't want the job as CEO. I was very happy being COO and working on the product. The company needed my hands-on experience and organizational skills. I like getting things done. When the investors asked me to take the role of CEO two years ago, I was delighted that I had gotten the product to a good position, built a team, and put systems in place. It now needed me to step up and take the business to the next phase.

I ran my own board shop for 30 years with my own money, sink or swim. You make your decisions; sometimes you screw it up, and you rebuild, or you don't. But this is different. I am now operating with investor money and someone else's dream. Every decision is important, and there is much accountability. I am confident Soluboard has a future in our industry.

I love this industry. My wife always asks me why I'm still doing this, but I have a love affair with this industry. I just wish I were a little younger with what's going on with AI. It's an exciting time.

Steve, it has been great talking with you. Good luck with everything you are striving for at Jiva. I hope it is very successful.

Thank you, Marcy. It's been a pleasure. **I-CONNECT007**

IN COMPLEX SYSTEMS, DESIGN RULES AREN'T OPTIONAL

BY MICHAEL CARANO, GLOBAL ELECTRONICS ASSOCIATION

There is no question that the electronics industry, especially in circuit board design and fabrication, advanced packaging, and innovation throughout the value chain, has seen a significant transformation, whether it be in materials, system architecture, HDI and ultra HDI, semiconductors, or chiplets. AI and high-performance computing (HPC) are driving change across several fronts, including material properties, assembly techniques (think hybrid bonding), and power management. Innovation has triggered the need for a better understanding of:

- Higher power on smaller traces
- Signal integrity and magnetic effects at higher frequencies
- New processing technologies (dielectric film, semi- and fully additive)
- Optical materials
- Innovation in cooling techniques

The silicon-to-systems approach must not be taken lightly. Essentially, winging it to make this all work, because our collective experiences show us the way, reminds me of the song that says, "Party like it's 1999." Well, it's not 1999, or 2023, for that matter. So much has changed in rapid-fire succession as Moore's Law has reached its limits, and chiplets and larger format substrates are becoming the norm. All this, however, comes with its own set of circumstances. Regardless, let's look at things from a rulebook perspective, without negating opportunities for innovation.

Rulebooks and Design Rules

With the increasing complexity of electronic systems, it is more critical than ever to ensure close cooperation among the end user, the designer, the fabricators, and the teams assembling these packages. You see multiple chips mounted onto a silicon interposer, the interposer soldered to a substrate, and the entire package assembled onto a board. All this has to function as the device/system was intended. What does all this mean? First, even with this level of complexity and need for higher performance, you must be able to fabricate these packages with high yield. More than one executive in this industry has said, "It's all about yields."

Design for excellence (DFX) requires a set of rules and cooperation between all parties in the project. Since many of these projects involve several companies (one firm handling the board/substrate design, another the interposer, another

"That's why design rules (and understanding each party's constraints) are parameters that, when followed, will ensure an accurate design."

the fab of the board and substrate), it is easy for things to get lost in translation.

I have watched designs handed to the fabricator with significant flaws, whether they concern material selection and signal integrity, lines and spaces too fine for the fabricator to succeed, or stacked microvias without understanding the potential for microvia interfacial fracture (MVIA).

If there were no set rules, the process would be a free-for-all. Designers often break these rules in the name of efficiency and optimum electrical performance. This often leads to product failure and multiple and costly respins.

That's why design rules (and understanding each party's constraints) are parameters that, when followed, will ensure an accurate design. It requires checking the design process step by step with all parties involved: fabricators, assembly, and the silicon fab for the interposer. It includes electrical rule checks (ERC) for the schematic and design rule checks (DRC) for the PCB and substrate. These rules serve a specific purpose. Think of them as gates that one must pass prior to moving to the next step. For example, the designer works with data to create a schematic. Once that is completed, ERC checks are required to validate whether the design is electrically correct. The ERC needs to give a clean report. If there is anything off, stop and fix the issue right away, as they can't be fixed later in the process.

When designers were pushing the software to deliver the most efficient design, was there a discussion with the board and substrate fabricators about whether the design could be manufactured reliably? Perhaps the OEM has communicated in the Statement of Work that the entire package (Z-axis) can only be so high, including the substrate thickness, the height of the chip or chips on the interposer, the interposer-to-substrate distance, the protective lid for the package, and the thickness added by the solder. Imagine if the designers completed the tedious task of routing signals, designing the stackup, providing the drill file, etc., only to learn that the substrate as designed is too thick to fit in the socket on the product board.

This is precisely why the designer must be cognizant of the constraints up front before going too deep into the design process. The fabricator and assembly firm should be involved from the outset.

A Few Caveats

If the rules are not set or are set incorrectly, you should understand how the board/substrate package will be used. Is this a package for AI, or an ultra HDI design for very high-frequency applications? Is it needed for high-power? The design rules will differ depending on what the circuit is supposed to do.

It's important that issues arising early in the design phase not be ignored. Signal integrity models, for example, can predict signal loss at various frequencies. What if the modeling shows issues with overshoot and undershoot? Address it immediately, not when the design is transferred to the fabricator for manufacturing.

Finally, check the design rules at each step and follow them throughout the design process. This will save heartache later.

Finally, don't underestimate the need for constant communication and collaboration with all parties involved, including the OEM, the EMS provider, PCB fabricator, and the substrate and chips/interposer teams. I cannot stress the need for transparency enough, and for the parties involved in the fabrication process to communicate their manufacturing capabilities and constraints to the design team. There are no trade-offs here. **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.

CAREER OPPORTUNITIES

CAREER OPPORTUNITIES

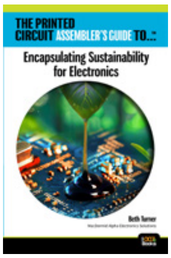
CAREER OPPORTUNITIES

CAREER OPPORTUNITIES

CAREER OPPORTUNITIES

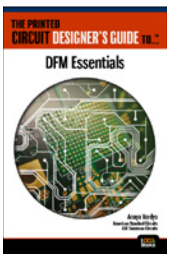
EDUCATIONAL RESOURCES

I-007eBOOKS



Encapsulating Sustainability for Electronics

by Beth Turner, MacDermid Alpha Electronics Solutions
This book discusses the growing demand for sustainable solutions in the market and highlights examples of bio-based resins and the demand from emerging technologies. Read it now!

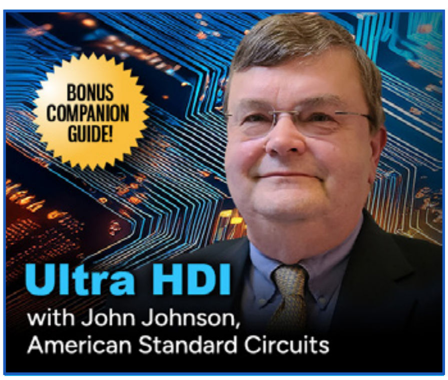


DFM Essentials

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

ON THE LINE WITH... I-Connect007



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 © CHOCKDEELAW

I-CONNECT007 MAGAZINE®
is published by IPC Publishing Group, Inc.,
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

April 2026, Volume 1, Number 4
I-CONNECT007 MAGAZINE is published monthly,
by IPC Publishing Group, Inc., dba I-Connect007.

ADVERTISER INDEX

Problems solved!