PCEA
PRINTED CIRCUIT ENGINEERING ASSOCIATION

Chavez: Automation the Answer to Workforce Deficits

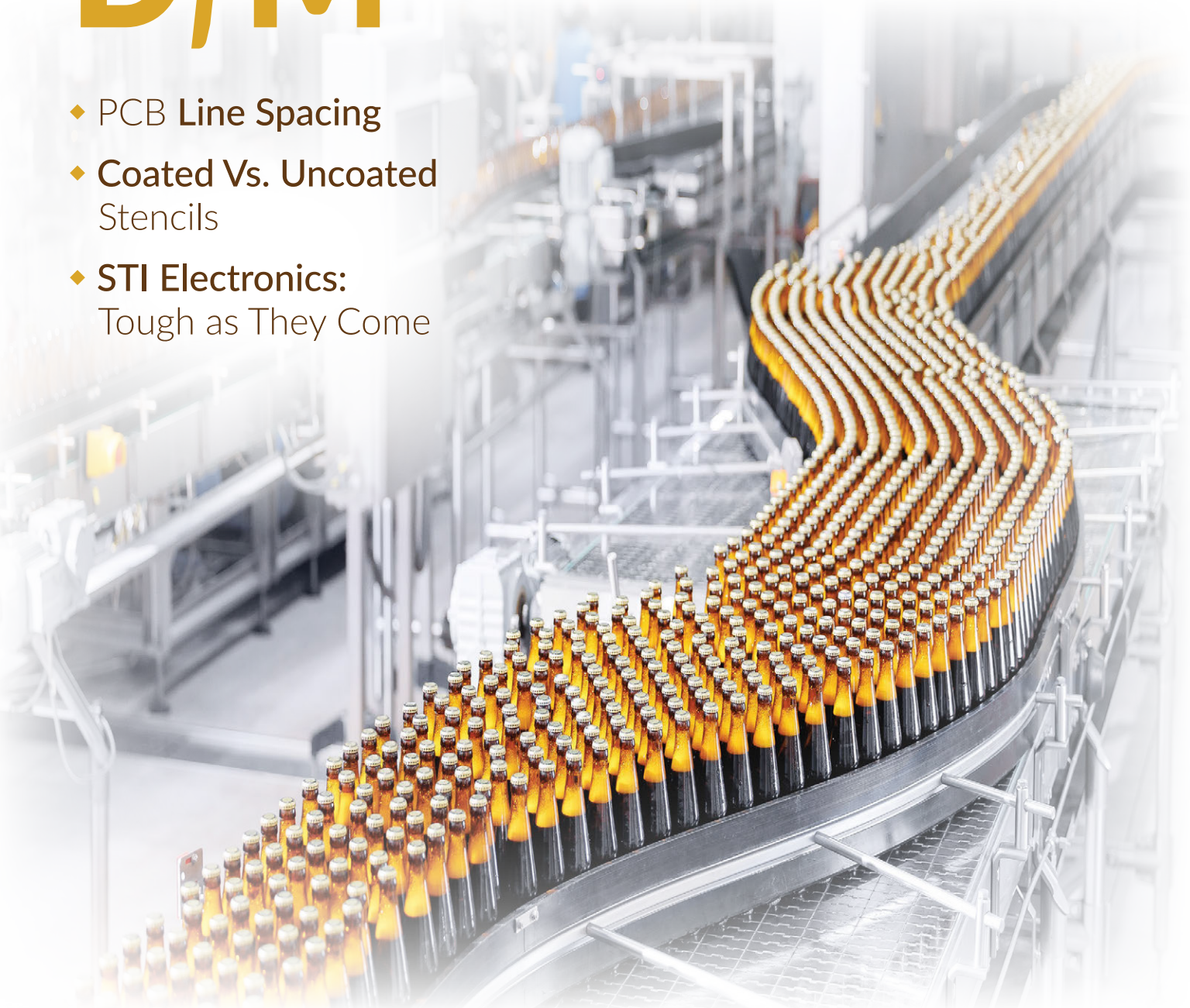
PRINTED CIRCUIT DESIGN & FAB

CIRCUITS ASSEMBLY

pcdandf.com
circuitsassembly.com
January 2025

DfM *from* Prototype *to* Ramp

- PCB **Line Spacing**
- **Coated Vs. Uncoated** Stencils
- **STI Electronics:** Tough as They Come

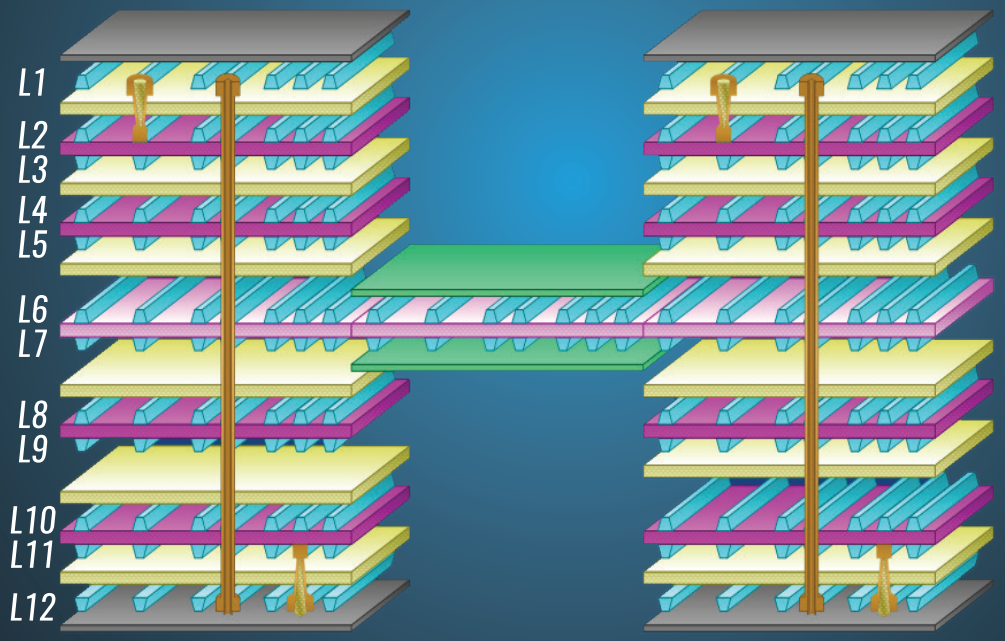


RUSHPCB

20 YEARS OF MANUFACTURING EXPERIENCE WITH BLIND AND BURIED VIAS, RIGID FLEX & FLEX PCB ELECTRONIC DESIGN, & QUICK TURN CIRCUIT BOARDS, ASSEMBLY & FULL TURNKEY

Multilayer PCB Stackup

12 Layer Flex Rigid, 0.030" Thick Rigid, 0.008" Thick Flex
Blind Via L1-L2 & L11-L12, 0.003" Hole, Resin Filled,
0.002"/0.002" Trace/Space, Controlled Impedance



Manufactured on August 2016

When you're in a rush call RUSH PCB

Affordable Pricing ⋆ Satisfaction Guarantee ⋆ Fast Response ⋆ Free Quote ⋆ Fast Shipping
HDI Boards ⋆ Stacked Micro Vias ⋆ 2 mil line & Space ⋆ Flex & Rigid Flex

Located in the Silicon Valley
PCB up to 12 Layers in 3 days
PCB Assembly Same Day Turn
Full Turnkey Service

Certified: ISO9001/ITAR/UL

www.rushpcb.com | Sales@rushpcb.com | 1 408 496 6013
Rush PCB Inc, 2149-20 O'Toole Ave, San Jose, CA 95131, U.S.A

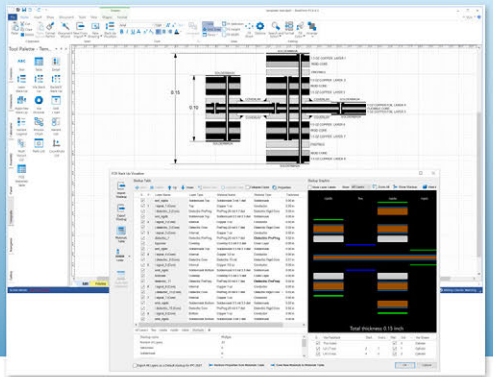
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

BluePrint-PCB®

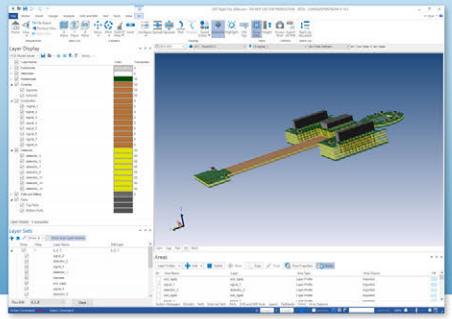
CAM350®

DownStream's CAM350 and BluePrint-PCB support importation and visualization of PCB designs containing Flex, Rigid Flex or Embedded components. Visualize designs in both 2D and 3D, and easily document complex Flex or Rigid- Flex Stack-Ups for submission to PCB Fabricators.

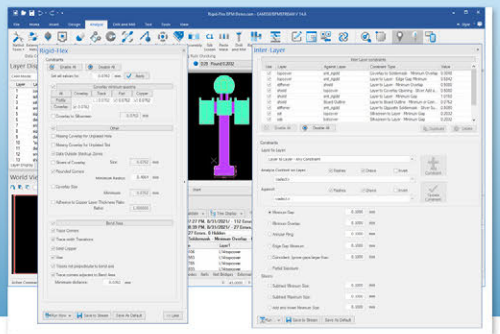
- **Import and Visualize Flex, Rigid-Flex and Embedded Component Designs**
- **3D Visualization to Validate PCB Construction and Component Assembly**
- **Manage Variable Stackup Zones for Rigid-Flex Designs**
- **Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation**
- **Use DFM analysis to analyze a flex or rigid-flex design for potential fabrication or bend related defects**



Use Stack Up Visualizer and Blueprint's Rigid-Flex Stackup template to easily manage and document rigid-flex stackups.



A rigid-flex design in 3D. Shown with layers spread to improve visualization of the layer stackup.



Use Rigid-Flex and Inter-layer DFM analysis to analyze flex and rigid-flex designs.

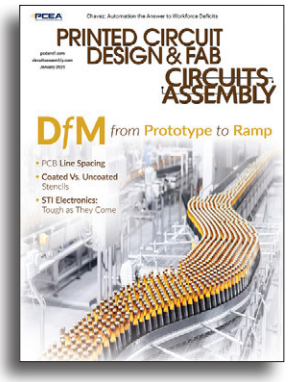
DownStream Technologies

For more information visit downstreamtech.com or call (508) 970-0670

JANUARY 2025 • VOL. 42 • NO. 1

PRINTED CIRCUIT DESIGN & FAB CIRCUITS ASSEMBLY

FEATURES



ON PCB CHAT (PCBCHAT.COM)

PCB Chat

NEW US BRANCH OFFICE IN WATERTOWN, CT!

Your new supplier for development and manufacturing processes!

INFOTECH automation

Exceeding your expectations!

PROCESS AUTOMATION

- Die Bonding
- Flip-Chip Bonding
- Sintering
- Dispensing
- Testing
- Sorting

ADVANTAGES

- Reliable
- Accurate
- Flexible
- Customizable
- Fast
- Proven

SCALABLE PROCESSES

Development

to

Production

MD&M WEST

February 4-6, 2025, Anaheim Convention Center
Exhibit Halls A-E — 2867 — Swiss Pavilion
Anaheim, California

Swiss Engineering

www.infotech-automation.com

PCEA®
Printed Circuit Engineering Association

PCEA
PO BOX 807
AMESBURY, MA 01913

PRINTED CIRCUIT DESIGN & FAB
pcdandf.com
circuitsassembly.com
CIRCUITS ASSEMBLY

PCEA.NET

THE PRINTED CIRCUIT ENGINEERING ASSOCIATION, INC. BRANDS:

PUBLICATION
PCD&F/Circuits Assembly ..... digital.pcea.net

WEBSITES
PCD&F ..... pcdandf.com
Circuits Assembly ..... circuitsassembly.com

NEWSLETTER
PCB Update ..... pcbupdate.com

PODCASTS
PCB Chat ..... pcbchat.com

EVENTS
PCB West ..... pcbwest.com
PCB East ..... pcbeast.com

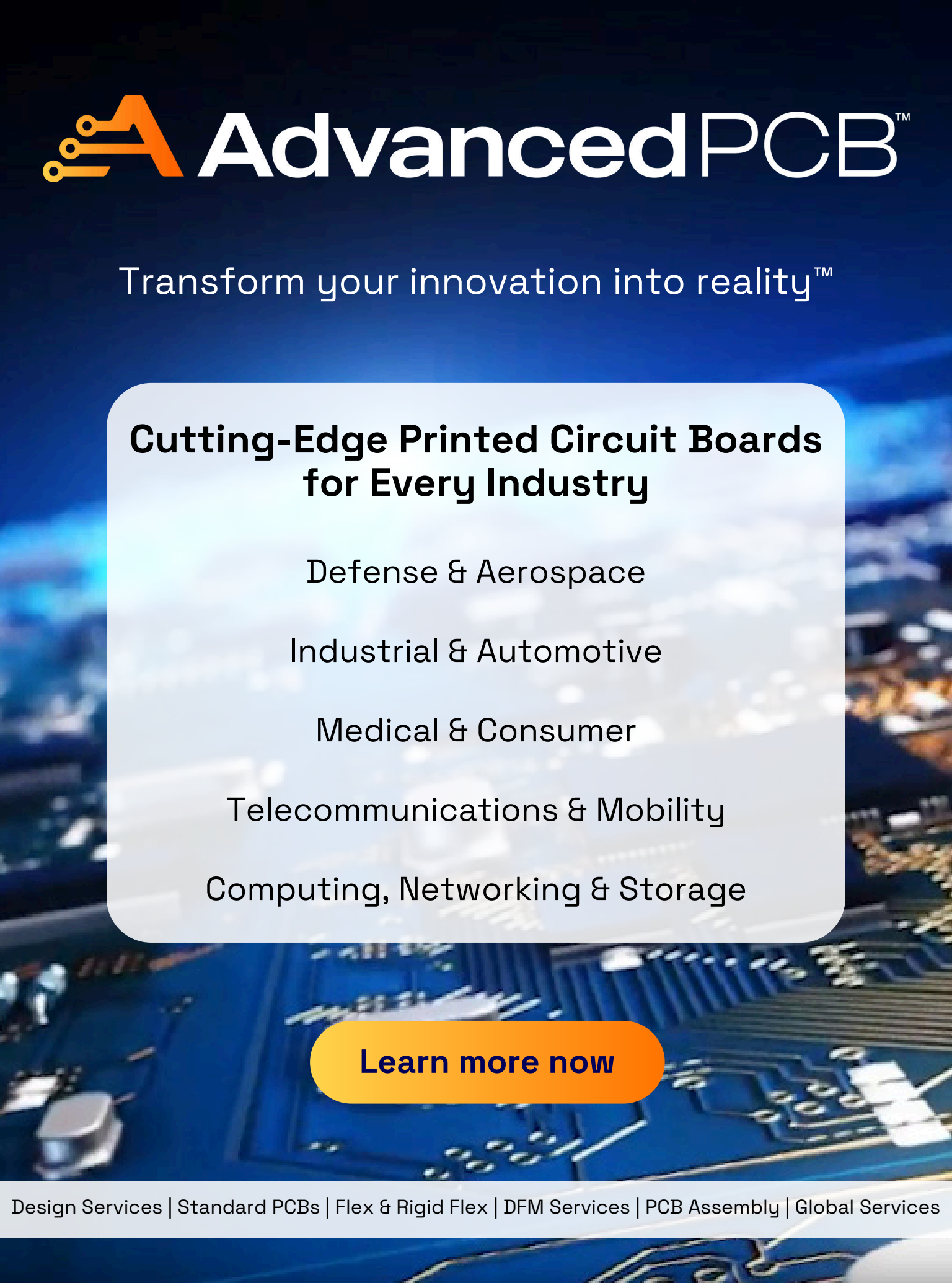
EDUCATION
PCB2Day pcb2day.com
PCEA Training ..... pceatraining.net
Printed Circuit University ..... printedcircuituniversity.com

AWARDS PROGRAMS
Service Excellence Awards ..... circuitsassembly.com
NPI Awards ..... circuitsassembly.com
pcdandf.com

SUBSCRIPTIONS

PRINTED CIRCUIT DESIGN & FAB/CIRCUITS ASSEMBLY is distributed without charge to qualified subscribers. To subscribe, visit pcdandf.com or circuitsassembly.com and click on Subscribe. For changes or cancellations to existing subscriptions: subscriptions@pcea.net

PRINTED CIRCUIT DESIGN & FAB/CIRCUITS ASSEMBLY is published monthly by Printed Circuit Engineering Association, Inc., PO Box 807 Amesbury, MA 01913. ISSN 1939-5442. GST 124513185/ Agreement #1419617.


AdvancedPCB™
Transform your innovation into reality™
Cutting-Edge Printed Circuit Boards for Every Industry
Defense & Aerospace
Industrial & Automotive
Medical & Consumer
Telecommunications & Mobility
Computing, Networking & Storage
Learn more now
Design Services | Standard PCBs | Flex & Rigid Flex | DFM Services | PCB Assembly | Global Services

Foiled Again: Copper Plant Closing Respins Old Supply Chain Concerns

THE MANY BILLIONS of dollars in direct funding or incentives the US Chips Act has bestowed on domestic manufacturers continue to obscure the basic adage that a chain is only as strong as its weakest link.

That truism was tested, again, in late November as Denkai America, the US-based subsidiary of the Japanese materials company, announced its intentions to close and liquidate its copper foil manufacturing plant in Augusta, South Carolina, the last such factory in the United States.

The news comes just two years after its parent company, Nippon Denkai, heralded the site as the future of foil production, with tens of millions of dollars in investments planned and hundreds of new jobs expected. The company cited financial problems stemming from semiconductor shortages, a decrease in exports of batteries manufactured domestically, lower demand for smartphones, and other factors for the decision.

Industry trade groups recognize the potential issues ahead. In a statement for PCD&F/CIRCUITS ASSEMBLY, the Printed Circuit Board Association of America said, "Single points of failure in the microelectronic supply chain will have consequences for PCB manufacturers and related industries. With no more US copper foil vendors, PCB manufacturers will have to qualify other vendors or find alternative materials. This will likely disrupt delivery of certain DoD programs of record as well as commercial customers supporting the nation's critical infrastructure. DoD Industrial Base Policy needs to double down on domestic single-source/single-point of failures in the microelectronics supply chain."

The impact is made all the more problematic because of the timing: The incoming Trump administration has made clear its intention to renegotiate – again – international trade agreements, including the USMCA free trade agreement that was just ratified in 2020.

To wit, while Circuit Foil, the materials subsidiary of Solus Advanced Materials, has a distribution center in Canada, President Trump has said he plans to impose a 25% tariff against the neighbors to the north. And as PCD&F/CIRCUITS ASSEMBLY contributing editor Dr. Hayao Nakahara notes, "The final objective of PCB users is price." So while there are alternate suppliers just over the border, the ripple effects mean there is no guarantee of supply chain continuity.

Much of the US defense strategy of late has centered on keeping advanced technology onshore through export controls or bans, or incentivizing key companies to develop domestic capacity (read: the Chips Act). But the focus, as usual, has been on blue chip, media-savvy firms like Intel, Samsung and TSMC. In a classic case of too little, too late,

the Department of Commerce has revived the President's Export Council Subcommittee on Export Administration (PECSEA), a panel of outside experts who advise export controls and scarce commodities in the interest of advancing national security interests.

The US does have other tools to help prevent the loss of critical suppliers to the Defense Department. The Committee on Foreign Investment in the United States, or CFIUS, was established to review transactions involving foreign investment in the US, and has the authority to negotiate or impose conditions to mitigate national security risk arising from a covered transaction. If the government determines the loss of a company would have a detrimental effect on the defense supply chain, it can force the business to stay open. Whether the powers that be recognize the significance of Denkai's closing, however, is unclear.

So here we are, in 2025, with wars ongoing in Eastern Europe and the Middle East, the Chinese playing wargames off the coast of Taiwan, and the US down to one (or soon to be fewer) domestic manufacturers of vital materials such as dry-film photoresist and copper foil, and equipment like etchers, laser drills and mechanical drills.

Quoting Nakahara again, "My gut feeling is that more than 50% of PCBs produced in the US go to military applications. If you consider major PCB makers in the US, more than 60% of their output go to military applications."

Engineers are risk-averse, politicians even more so. As we revisit this sad story, what's their risk tolerance for a broken supply chain, particularly with the nation's defense at stake?

mike@pcea.net
@mikebuetow

P.S. One person who could be counted on to be shouting from the mountaintops about this looming crisis was Harvey Miller. Sadly, Harvey passed away on Nov. 27, at the age of 102. He was truly a beloved and vital figure for his work, especially in the area of market research on the PVB industry; his passion for searching for understanding what the data are actually saying; and his willingness to slay so many of our industry's sacred cows. We will all miss him.

MIKE BUETOW is president of PCEA (pcea.net); mike@pcea.net.

Nippon Denkai to Close Last Remaining US Copper Foil Plant

AUGUSTA, GA – Nippon Denkai has decided to close and liquidate its copper foil manufacturing plant here, the last such factory in the United States.

Denkai America Inc., the US-based subsidiary of the Japanese company, will close just two years after its parent company heralded the site as the future of foil production, with tens of millions of dollars in investments planned and hundreds of new jobs expected.

The company cited financial problems stemming from semiconductor shortages, decreased exports of domestically manufactured batteries, lower demand for smartphones, and other factors for the decision. The continued losses, coupled with loan payments coming due and dried-up sources of new capital, have rendered the company insolvent.

Denkai America announced in 2022 the planned investment of some $150 million in the plant to make copper foil. It took combined losses in the 2023 and 2024 fiscal years of about $23 million tied to the plant.

For its fiscal year ending March 2025, decreased sales and a rapid increase of copper prices, plus an impairment loss related to the Augusta plant, led to first-half consolidated ordinary losses of approximately 1.5 billion yen ($9.9 million) and a consolidated net loss of approximately 5 billion yen ($33 million).

In addition to the deterioration of its performance in and after the fiscal year ended March 2023, the company's cash flow gradually deteriorated as a result of proceeding with the reconstruction of manufacturing facilities in the American subsidiary's previous plant in Camden, SC, and investments in foil manufacturing facilities in the Augusta plant.

Attempts to find additional funding have stalled, Denkai said in a statement.

In November, the company filed a petition for commencement of civil rehabilitation proceedings. Under the supervision of the Tokyo District Court, Nippon Denkai will continue its business while seeking a buyer. Sumitomo Mitsui Banking is providing up to 2 billion yen as DIP financing for the company's future fund-raising, and Nippon Denkai is being delisted from the Tokyo Stock Exchange.

EU Set to Approve Synopsys-Ansys Deal; UK Watchdog Raises Concerns

BRUSSELS – Synopsys' $35 billion acquisition of Ansys is reportedly set to win conditional EU antitrust approval after the company pledged to address competition concerns, while the UK debates permitting the deal to go through.

Reuters reported that the European Commission, which acts as the EU's antitrust enforcer, is set to approve the deal after Synopsys offered to sell its optical design tool maker Optical Solutions Group and Ansys PowerArtist.

The company is expected to offer the same remedies to the UK's Competition and Markets Authority, which also raised concerns about the potential for reduced competition in the semiconductor chip design and light simulation markets, Reuters reported.

Synopsys and Ansys announced a $35 billion merger in January, with the goal of combining Synopsys' semiconductor electronic design automation with Ansys' broad simulation and analysis portfolio to create a design leader.

The CMA said its concerns about the merger include the reduction of choice among customers who use the two companies' products, which could lead to a loss of innovation, lower quality software and/or higher prices which may then be passed onto UK businesses and consumers.

The CMA's investigation, which examined the impact of the merger across a range of semiconductor chip design and other software markets, looked at the extent to which Synopsys and Ansys currently compete or may do so in the future. The CMA found that while the companies' products are largely complementary, the deal could reduce competition in the supply of three software areas where Synopsys and Ansys have strong market positions and compete closely.

The CMA said its concerns relate to global register transfer level power consumption analysis, which is used to check how much power a chip consumes and requires to function, as well as global optics software and photonics software, which are used to design and model light-related products.

"Synopsys and Ansys are important suppliers of semiconductor chip design and light simulation software, and we're concerned that this deal could reduce innovation and lead to higher prices for these products in the UK," said Naomi Burgoyne, senior director of mergers at the CMA. "Millions of businesses and consumers in the UK use products that rely on these companies' software every day, whether that's consumer electronic devices, medical equipment, modern vehicles or even AI."

The companies now have the opportunity to offer solutions to address the CMA concerns, otherwise the deal will be referred to a more in-depth investigation.

DBG Technology to Acquire All Circuits

MEUNG-SUR-LOIRE, FRANCE – All Circuits, one of France's largest EMS providers, is set to be acquired by China's DBG Technology at the beginning of 2025.

With the acquisition, All Circuits will add China to its manufacturing footprint, as well as India and Vietnam, which, alongside Mexico, are seen as winners in the move to reduce dependency on China as the factory of the world, the

company said.

DBG, which also has a facility in Bangladesh, will gain additional capacity in France, Tunisia and Mexico.

“We’re excited to have manufacturing in China for our customers that are selling products in China, as well as having a solid footprint elsewhere in Asia where demand is strong,” said Bruno Racault, president, All Circuits. “This along with our own sites in Europe, North Africa and North America will deliver outstanding global manufacturing choices to our current and future customers.”

The acquisition discussions could result in the signing of a final agreement this spring, All Circuits said in a release. This agreement would create a combined production capacity of more than 150 SMT lines, design offices, and complementary sector specializations (primarily in telecommunications for DBG Technology and automotive for All Circuits).

Under the integration, All Circuits’ current management team would retain operational control of the company, which would continue to operate independently under its current name.

“This proposed partnership represents a major milestone in our company’s history, offering solutions across international markets to produce closer to our clients’ needs while maintaining and strengthening our position in Europe,” said Racault. “It will enhance our ability to innovate and serve our customers with the same level of excellence and commitment in an ever-evolving global context. All Circuits would like to express its gratitude to IEE S.A. and its shareholder for their strategic cooperation over the past 10 years, which has significantly contributed to the company’s rapid growth. All Circuits will continue to serve IEE S.A. as one of its key customers in the global electronics manufacturing sector.”

AI-Based Design Startup Raises $6.8M in Seed Funding

MONTREAL – Cadstrom, an AI startup that aims to allow electronic design with minimal respins, announced that it has raised $6.8 million in seed funding.

Founded in 2023 by Margot Blouin and Scott Bright, Cadstrom builds AI tools enabling electrical engineers to create devices that work correctly on their first attempt. Its proprietary Sigma Engine uses a first-principles understanding of physics, electronics and generative AI to identify mistakes while automatically validating complex designs – cutting development costs, accelerating GTM timeline and shortening design cycles up to 66% by eliminating respins, the company said.

“We founded Cadstrom to address the reality that most electronics designs require several time-consuming revisions, and the problem is getting worse as electronics, AI, and connectivity becomes increasingly ubiquitous. Our vision is one where every engineer can achieve robust designs on their first build and get to market faster,” said Blouin. “This funding represents a critical milestone for our team, and we’re excited to put the capital to work to further develop our automated verification tools that detect and address the most common issues that electrical engineers face.”

The round was led by Bison Ventures and included participation from Innovation Endeavors and AI2 Incubator, which previously invested $650,000 in a pre-seed round.

“Cadstrom’s approach is bringing unprecedented progress to the Testing, Inspection, and Certification market,” said Tom Biegala, founding partner, Bison Ventures. “[Blouin and Scott Bright] built the product they desperately needed as hardware engineers: By improving one of the most tedious parts of PCB development, Cadstrom helps engineers be more productive and bring validated hardware to market at remarkable speeds. This is in stark contrast to tools seeking to replace engineers with ineffective AI.”

Cofactr Raises $17M in Series A Funding

NEW YORK – Cofactr, a provider of a supply chain and logistics management platform for hardware manufacturers, has raised $17.2 million in a series A investment round.

Bain Capital led the Series A round, with existing investors Y Combinator, Floating Point Ventures, Broom Technologies and DNX Ventures also participating. The company said it will use the funding to grow its go-to-market efforts and its suite of supply-chain risk management and process tools. It also plans to introduce additional product categories, with multiple applications slated to launch each year.

Cofactr’s platform integrates procurement automation, supply chain data intelligence and logistics infrastructure to streamline the manufacturing process for hardware teams, allowing them to source, quote, procure, manage, kit and ship electronic components through the same platform.

“Traditional supply chain management has left serious gaps for innovative companies navigating the electronics and mechanical spaces. We’re filling them by creating a seamless link between product lifecycle management (PLM), enterprise resource planning and manufacturing execution systems (MES),” said Matthew Haber, CEO and cofounder of Cofactr.

“For these companies, it’s not agility or rigor – it’s both. We’re giving oversight departments the control visibility, and processes they require while giving product engineers the tools they need to get products to market fast,” added Phillip Gulley, the company’s chief strategy officer and cofounder.

Cofactr said its platform is currently used by more than 50 customers, including the robotics division of the world’s largest e-commerce marketplace, the hardware division of the world’s largest social media company and the world’s leading self-driving car manufacturer.

“In mission-critical industries such as aerospace, defense, automotive and robotics, electronic components represent 70% of the bill of materials, yet existing procurement and supply chain software is generic and not built for the speed and requirements of electronics,” said Ajay Agarwal, partner at Bain Capital Ventures. “Cofactr is the first modern AI solution for end-to-end electronics procurement and logistics that meets the needs of engineers, procurement teams and suppliers.”

GreenSource Plans UHDI PCB Facility in NH

CHARLESTOWN, NH – GreenSource Fabrication has received additional funding for its planned 98,000 sq. ft. UHDI PCB manufacturing facility here.

The New Hampshire Business Finance Authority (NHBFA) board unanimously approved a bond of up to $50 million to support the project, with the approval also involving the US Department of Defense as a funding partner, the New Hampshire Business Review reported.

GreenSource, a subsidiary of Whelen Engineering, launched in 2019 and previously received a $46.2 million grant from the DoD to enhance its capabilities in defense-sector UHDI PCB manufacturing.

The company said the funds will be allocated toward advanced manufacturing equipment for the new facility to boost production quality, support national security initiatives and diversify its client base.

With the expansion, GreenSource plans to retain 142 existing jobs and create an additional 50-100 jobs. In addition, Whelen and GreenSource will collaborate with New Hampshire high schools and colleges to recruit talent and provide internships and full-time opportunities.

“This facility represents a monumental investment in the future of American manufacturing,” said Ashish Jain, CEO of GreenSource Fabrication. “Our partnership with the NHBFA and the DoD underscores our shared vision for innovation, security and sustainability.”

Ventec Chooses Thailand for PCB Material Manufacturing Facility

SUZHOU, CHINA – Ventec has selected Thailand as the location for a $17 million PCB material manufacturing facility.

The company said the new facility will be built on 8.4 acres in the Hi-Tech Industrial Estate north of Bangkok in Ayutthaya Province, reinforcing its commitment to global supply chain resiliency by expanding its manufacturing footprint beyond China and Taiwan, as well as customer proximity in the developing Southeast Asia PCB manufacturing hub.

Ventec announced in June 2023 that it would establish a manufacturing base in Southeast Asia to diversify geographical risk and enhance support for growing customer demand. Construction is scheduled to commence in the second quarter of 2025, and full production will start in the first quarter of 2026. The facility will offer cutting-edge production capabilities across two phases, achieving a capacity of 150,000 sheets per month in the first phase, the company said.

Ventec said it selected Thailand for its new manufacturing facility due to several factors, including the diversification

of geographical risk by expanding beyond existing hubs in China and Taiwan and the country's emergence as a leading PCB manufacturing hub outside China.

Additionally, Thailand offers a robust logistics infrastructure with reliable international shipping routes, which ensures fast delivery times, and the country's strong support for the electronics manufacturing sector provides a favorable environment for efficiency and cost-effectiveness, benefiting customers across all regions, the company said.

"Now is the right time to invest in establishing an additional manufacturing presence in Thailand to support the expanding Southeast Asian electronics industry, which is experiencing significant growth across consumer, industrial, and automotive sectors," said CEO Jason Chung. "This new factory will enhance our supply chain security promise to our global PCB and OEM customer base as we plan to manufacture the complete portfolio of advanced high-reliability and high-performance materials."

"The addition of our Thailand factory strategically positions Ventec to better serve our customers worldwide," said Mark Goodwin, COO, EMEA & Americas. "Thailand's robust logistics infrastructure enables faster shipping times, and exceptional supply chain reliability. As Thailand strengthens its position as a leading PCB manufacturing hub, we can deliver unrivaled value to our customers by ensuring their supply chains remain agile and competitive, whilst also addressing the geopolitical and geographical diversity of manufacturing footprint requirements of some key customers and market sectors. The new facility underscores Ventec's leadership in delivering high-reliability and high-performance PCB materials for applications in automotive, aerospace, communications, and more."

OKI Improves Heat Dissipation with Stepped Copper Coin Insertion

TOKYO – OKI Circuit Technology has developed multilayer PCB technology with stepped copper coin insertion, which reportedly achieves 55 times better heat dissipation than conventional PCBs.

The stepped copper coin is offered in two types, circular and rectangular, to suit the shape of the electronic component mounted on the PCB, and OKI said it is working to develop mass-production technologies with the aim of PCBs with the new technology into markets for compact devices or devices used in outer space or other environments where air cooling technology cannot be used.

In 2015, OKI developed proprietary design and mass production technologies for multilayer PCBs based on copper coin insertion. This innovation embeds cylindrical copper coins with high thermal conductivity into PCB through-holes and bonds them to heat-generating electronic components to dissipate heat to the substrate's underside.

The use of copper coins that transfer heat to the underside of the PCB achieves high heat dissipation efficiency in cases where it is not possible to mount heat-dissipating devices directly to electronic components due to the device's dimensional constraints or where heat-dissipating devices cannot be mounted to certain functional surfaces (e.g., image sensors or light-emitting components). This time, OKI said it has made further progress with this technology and succeeded in developing the stepped copper coin that achieves approximately twice the heat dissipation efficiency.

The newly developed stepped copper coin features a larger heat-dissipating area relative to the bonding surface with the heat-generating electronic component to improve heat conduction efficiency. Additionally, aiming for an optimal heat dissipation structure to suit the shape of each electronic component, a rectangular type featuring a rectangular component installation surface and rectangular heat-dissipating surface has been developed, in addition to the conventional circular type, to increase the contact area and heat-dissipating area for rectangular electronic components to improve heat dissipation efficiency.

Both types can be customized to obtain an optimal heat dissipation structure for the given shapes of the mounted component, the PCB itself and the substrate thickness, OKI said.

Meiko Plans $300M PCB Fab Expansion in Hanoi

HANOI – Meiko Electronics reportedly plans to invest $300 million to build a new PCB fabrication facility here.

The new investment by Meiko will build its fourth PCB plant in the city and will raise its total investment in the area to $800 million, *The Investor* reported.

The new construction is scheduled to be completed in March 2027, with operation beginning the following month. The project, located in Thach That-Quoc Oai Industrial Park, now employs 3,770 people and is set to increase the workforce to 4,170 with the expansion.

With the expansion, the company's PCB fabrication in Hanoi will reach an annual capacity of 2.16 million sq. m. of plated through-hole PCBs, 540,000 sq. m. of flex PCBs, and 2.2 billion electronics manufacturing service (EMS) items.

Icape Group Acquires UK PCB Distributor, Minority Stake in Green PCB Developer

FONTENAY-AUX-ROSES, FRANCE – Icape Group has announced the acquisition of 100% of the capital of ALR Services, a British PCB distributor.

With the acquisition, Icape expands its business in the UK while integrating a local structure renowned for its expertise and commitment to continuous improvement, the company said, while ALR Services now has access to a greater range of services and the full purchasing power of Icape.

"We are delighted to integrate ALR Services, which gives us a footprint in the United Kingdom, a highly competitive market for PCB distribution players," said Icape Group CEO Yann Duigou. "As with all Icape Group acquisitions, we identified strong synergies and concrete cross-selling opportunities between our two organizations, which should

translate into increased post-integration revenue. In addition, ALR Services' local teams serve a portfolio of 300 active customers, not addressed by the Icape Group, to whom they will now be able to offer the full range of services offered by our global platform."

With a portfolio of 300 EMS and OEM customers in the defense and automotive industries, ALR Services expects to achieve revenue of £2.5 million ($3.2 million) in 2024.

Icape will also acquire a minority shareholding in Jiva Materials, a UK-based developer of what it calls the world's first fully biodegradable PCB substrate.

Soluboard, Jiva's PCB material, is designed to dissolve in water at end-of-life, paving the way for easier recycling and significantly reducing the environmental impact of electronic products.

Icape's PCB factory in Malmö, Sweden, will be one of the key fabrication facilities in Europe for the development of the Soluboard PCB moving forward.

"This collaboration marks a major milestone for Jiva," said Steve Driver, CEO of Jiva Materials. "We've earned the trust of prominent brands in the electronics industry, and now, with Icape as a partner, we are ready to accelerate the industrialization of Soluboard with PCB manufacturing facilities in Europe."

"As a global company based in France, we are thrilled to see our partnership with Jiva unfold in Europe," said Duigou. "Icape is committed to retaining technological expertise in Europe, and this collaboration will not only enhance our sustainability focus but also solidify our leadership in eco-friendly technology."

Gentex Announces Acquisition of Voxx International

ZEELAND, MI – Automotive electronics manufacturer Gentex has announced the acquisition of all remaining shares of Voxx International for around $120 million.

Gentex previously held around 30% of Voxx's shares and agreed to acquire the rest of the company's shares at $7.50 per share in an all-cash transaction. The acquisition of Voxx will result in an increase in annual revenue of $350 million to $400 million, Gentex said in a release.

Voxx produces automotive and consumer electronics, supplying in-vehicle entertainment and security systems to the automotive industry and consumer audio systems under the Klipsch, Onkyo and Integra brands.

Gentex said the transaction will give it full access to Voxx's biometric technology, which will provide further product applications into the automotive, aerospace and medical markets, as well as provide its expertise in high-volume manufacturing to Voxx's audio electronics team to help its expansion in the consumer and connected home space.

Sero Acquires European EMS EPSa

ROHRBACH, GERMANY – Sero EMS has announced the acquisition of EPSa-Elektronik & Präzisionsbau Saalfeld, a German EMS provider, and EPSa Děčín, its Czech subsidiary. Financial terms were not disclosed.

With the addition of EPSa's facilities and personnel in Germany and Czechia, Sero strengthens its development capabilities, enhancing its expertise in software and hardware development, mechanical engineering, rapid prototyping and high-mix, low-volume production, the company said in a release. EPSa's specialized skill set will enhance Sero's ability to better serve complex customer needs and offer tailored solutions with a focus on MedTech equipment and high-end industrial applications.

"EPSa is a fantastic addition to the Sero EMS family," said Jan-Frederik Kalee, CEO, Sero EMS Group. "Their technical capabilities and commitment to quality align perfectly with our strategic goals. Together, we are positioned to offer enhanced, end-to-end solutions that meet the growing complexity of customer demands across industries."

"We as EPSa are excited about the synergies and high cultural fit with the group," said Ali Sahin, managing director, EPSa. "It will allow us to develop more differentiating value-add services toward existing and new customers. Taking over the responsibility for the group CTO role is an honor and motivation emphasizing the strong level of integration between the group companies. Looking forward to joining and growing a superior SERO EMS Group team."

Eagle Electronics Building OH Manufacturing Facility

SOLON, OH – Eagle Electronics has raised $14 million to build a new manufacturing facility here in partnership with CO-AX Technology.

The EMS provider and producer of cellular modules said it plans to use the funding, led by the OHIO Fund with participation from Asymmetric Capital Partners, to build an advanced electronics manufacturing plant, leveraging cutting-edge surface-mount technology manufacturing equipment.

"This is a major moment for Eagle and our vision of onshoring and securing America's critical technologies," said TJ Dembinski, co-founder and CEO of Eagle Electronics. "Our goal is to set a new standard for electronics manufacturing in the US, combining the most advanced automated manufacturing and testing technology with rigorous hardware and software cybersecurity testing throughout the supply chain. This offering will allow Eagle to onshore critical knowledge and essential technologies, all while continuing to cement the United States' leadership in the chip industry."

The new operation is also expected to create numerous jobs in Ohio, further establishing the state as a key player in the US semiconductor industry.

"Eagle is further proof that Ohio is becoming one of the world's leading semiconductor manufacturing hubs," said

Ohio Lt. Gov. Jon Husted. “With the addition of Eagle, we’re bringing more jobs in this sector to Northeast Ohio, reinforcing our commitment to the ‘Made in Ohio’ strategy.”

Eagle said it is already seeing demand for American-made cellular modules, with Boston-based Cherish contracting with Eagle to source components for its health and safety monitoring products manufactured in the communities where its customers live.

“Our customers trust us to invite our products into their homes to monitor the health and safety of loved ones,” said Cherish CEO Sumit Nagpal. “Our work with Eagle helps deepen this trust with, in this case, cellular modules manufactured in the USA. This is good for us, our customers, and the communities we serve.”

“Our partnership with Cherish Health demonstrates the market’s confidence in our ability to deliver high-quality, state-of-the-art cellular modules,” added Mark Kvamme, Eagle’s cofounder and chairman. “We look forward to expanding our customer base as we ramp production and continue to innovate in the cellular module and electronics manufacturing spaces.”

SEMI Publishes Recommendations for EU Semiconductor Policies

BRUSSELS, BELGIUM – SEMI has published recommendations for the European Union to bolster its semiconductor ecosystem, urging new commissioners to implement legislation to advance the semiconductor sector given its role as a key driver of technological advancement, economic growth and ensuring Europe’s future competitiveness on the global stage.

“In light of multiple technological disruptions such as AI and autonomous vehicles poised to drive industry growth over the years ahead, SEMI Europe presents its recommendations for a successful long-term strategy for the European semiconductor industry,” said Laith Altimime, president of SEMI Europe. “The European Chips Act has generated strong momentum, and our recommendations highlight urgent actions to continue the progress made. Policymakers must build on this momentum by strengthening the legislative framework and advancing toward a ‘Chips Act 2.0.’”

SEMI Europe’s key recommendations for policymakers to consider include:

- Strengthening the semiconductor ecosystem by optimizing funding across the EU, member states, and private sectors to achieve the EU’s goal of a 20% global market share by 2030, advancing toward a comprehensive “European Chips Act 2.0”
- Enhancing economic security by aligning protective measures with proactive initiatives to improve global competitiveness, streamlining export controls, reinforcing intellectual property protection, and reducing administrative burden
- Integrating semiconductors in the EU’s Green Deal by developing a policy framework that balances innovation and sustainability

- Ensuring responsible and sustainable use of essential chemicals, while supporting research for alternatives and maintaining the competitiveness of the European semiconductor supply chain
- Addressing the increasing talent gap by fostering collaboration between industry and education, and reforming immigration legislation to attract skilled talent globally
- Aligning funding with industry needs through the next Multi-Annual Financial Framework while fast-tracking the Chips for Europe Initiative and Important Projects of Common European Interest (IPCEI).

The SEMI Europe recommendations underscore the importance of the European Commission in prioritizing strategic investments and establishing robust policy frameworks that promote collaboration across sectors, advance sustainable innovation, and cultivate a skilled workforce, thereby ensuring Europe's competitive standing within the global semiconductor industry.

"SEMI Europe is committed to engaging with stakeholders to advocate for a comprehensive 'Chips Act 2.0' initiative that will address gaps in the current framework," said Altimime. "This initiative will help solidify Europe's position in the global semiconductor sector, boost competitiveness, and enhance security."

Wistron Set to Open Vietnam Manufacturing Plant

HA NAM PROVINCE, VIETNAM – Wistron reportedly plans to begin operations in January at its new $24.5 million manufacturing plant here.

The manufacturer's Victory II facility, which was scheduled to have equipment installed by the end of December, will employ 530 staff once completed, and will have an annual production capacity of 3 million LCD panels, 5.9 million notebooks, 375,000 desktop computers, 1.2 million VoIP phones, 300,000 motherboards and additional products, *The Investor* reported.

The company previously invested $363.9 million in its Victory I facility, also located in Ha Nam Province, with operations beginning in December 2021 to manufacture notebooks, motherboards, desktop computers, and camera modules.

The construction of the second phase of Victory I is currently underway, with completion planned for April. After the expansion, the facility will employ 14,470, with an annual output of 4 million notebooks, 6 million LCD panels, 1.5 million docking stations, 1 million motherboards, 2 million desktop computers, 375,000 webcams and other products, according to *The Investor*.

Delta, Cal-Comp Sign MoU to Advance EMS Automation

BANGKOK – Delta Electronics and Cal-Comp Electronics are partnering to advance the EMS industry by combining Delta's automation expertise with Cal-Comp's manufacturing capabilities.

The memorandum of understanding between the companies reflects a shared commitment to advancing efficiency, innovation, and sustainability in the EMS industry, while formalizing previous successful projects between Delta and Cal-Comp, the companies said in a release.

The partnership will permit the combination of Cal-Comp's CCET 4.0+ platform, AI solutions, and data center run-through manufacturing process with Delta's DIAEAP-IMM solution, which will bolster parameter consistency, optimize mold flow and significantly reduce production errors.

"Cal-Comp and Delta have both been deeply root in Thailand for more than 35 years and both have done well and succeed in its professional field," said Cal-Comp COO Khongsit Choukitcharoen. "Over the years, Cal-Comp has been developing internally to enhance its automated smart manufacturing process and digital management in aiming to maintain its competitiveness among the competitors in the EMS industry. Delta has great achievements in energy conservation and carbon reduction, automobiles products, and AI-enabled solutions in recent years. Therefore, we are exciting to enter a strategic collaboration with Delta to explore and to enhance the capabilities in smarter and greener manufacturing processes and to further drive the development of electronics industry in Thailand to the next milestone."

"We are excited to formalize our relationship with Cal-Comp through this MOU, which represents a significant step forward in advancing smart manufacturing and Industry 4.0+ practices within the EMS industry," said Victor Cheng, CEO, Delta Electronics. "By integrating Delta's cutting-edge automation and AI-enabled solutions into Cal-Comp's manufacturing processes, we aim to enhance operational efficiency, optimize production workflows, and drive sustainable innovation. The MOU allows for collaboration between Delta and Cal-Comp, focusing on supporting smarter and greener factory operations."

PCD&F

Compeq opened a new PCB fabrication facility in Thailand as it expands its presence in the country to open a new key export base.

India's **CSIR-National Metallurgical Laboratory** announced the transfer of its PCB recycling technology to **Novasensa**, which will process valuable metals and materials from waste PCBs.

Sweden's **DP Patterning** developed in-house flex PCB manufacturing capabilities using a dry-phase patterning process that eliminates the need for harsh chemicals.

Czech company **Gatema PCB** has added a new press center to produce PCBs, featuring a **Lauffer** press machine, a **Monolam** MLP 25M high-temperature press and a **Cedatec** optical pinless bonding machine.

Hindalco Industries is investing $10 billion in PCB recycling with the goal of tackling India's growing e-waste problem.

ISU Petasys has pulled out of its plan to acquire carbon nanotube maker **JEIO**.

Kaynes Technology India has launched projects in PCB fabrication and semiconductor assembly and test to expand its presence in the electronics industry.

Mint Innovation, an Australian company scaling up a hydrometallurgical technology to recover precious metals from circuit boards, began construction on a $20 million refinery in Texas.

PCBWay launched a range of competitions for electronics and mechanical designers to encourage engagement in open-source innovation projects.

Sierra Circuits is moving its administrative headquarters into a 10,000 sq. ft. industrial building near one of its plants in Sunnyvale, CA.

Würth Elektronik opened a new prototype lab in Munich.

CA

Absolute EMS signed a strategic partnership with **PTEC Solutions** to provide assembly services and receive PTEC's expertise in design, cable manufacturing, and mechanical assembly.

Aimtron Electronics opened a Texas-based US subsidiary on Jan. 1.

Altix, a French specialist in photolithography and direct imaging equipment, announced its successful merger with **MGI Digital Technology**.

APAG Elektronik, a Swiss automotive electronics company, opened a new 4,000 sq. ft. research and development facility in Windsor, ON.

Apple plans to invest $1 billion in a manufacturing plant in Indonesia to produce components for smartphones and other products.

Asteelflash expanded its SMT assembly capacity at its Bedford, UK, plant with eight **Fuji** NXT III placement machines.

ASYS Group opened a new Swiss subsidiary, **ASYS Automation Switzerland**.

BAE Systems and **Boeing** will provide EW systems for the US F-15E jet fighter-bomber in a $54.2 million order.

Bittele Electronics upgraded its printed circuit board facilities to better serve the AI industry.

BYD's contract manufacturing business now assembles over 30% of all iPads and is taking a bigger role in manufacturing iPhone parts such as the titanium frame.

Cicor will receive a $96.3 million takeover bid from US investment firm **One Equity Partners.**

Dixon Technologies and Chinese smartphone maker **Vivo** are forming a joint venture to manufacture phones in India.

Essemtec opened a new demo center in Newton, MA.

Foxconn may expand its Tamil Nadu, India, facility to add more than 20,000 jobs and broaden its manufacturing capabilities in the country beyond just assembling iPhones.

Green Circuits installed a **Nordson** DAGE Quadra 5 Pro x-ray system.

GOCL Corp. is venturing into high-growth sectors like electronics manufacturing services and metal cladding.

GPV completed a major refurbishment of a former mechanics factory in Thailand, transforming the facility into a 22,000 sq. m. hub for electronics manufacturing.

Hensoldt added a second PCBA production line at its Ulm, Germany, facility, with a third planned for 2025.

HMD Global, which makes phones under the **Nokia** brand, will move its manufacturing center from China to India.

HyRel Technologies announced the sale of its first Versacell robotic solder dip and BGA reballing system to a top-five defense contractor.

JST, a Japanese electronics manufacturer, will build a 300,000 sq. ft. manufacturing and design center in Guntersville, AL.

Kitron announced a $5 million order from a major defense manufacturer to provide custom electronics to the **US Army** and is expanding its facility in Jönköping, Sweden, to add around 6,000 sq. m. of production space.

L3Harris completed critical hardware checks and simulation tests for the **US Navy's** F/A-18 fighter jet using 3-D printed models, paving the way for further prototype development.

Luxshare Precision is expanding its production capacity in Vietnam by significantly enlarging its Nghe An facility.

Microchip Technology will close its wafer manufacturing factory in Arizona after grappling with slowing orders for its automotive chips.

Murata Electronics will invest THB62 billion ($1.8 billion) to build a new advanced capacitor factory in Thailand.

Padget Electronics, a subsidiary of **Dixon Technologies**, is set to commence mass production of **Google** Pixel smartphones at its Noida, India, facility.

Sat Nusapersada chose **Siemens'** Process Preparation Software to boost its NPI and SMT line efficiency.

Schneider Electric will build a PCB manufacturing plant in Karnataka, India, creating an additional 5,000 jobs in the coming years.

Shunsin Technology, a **Foxconn** subsidiary, will build a TWD2.11 billion ($65 million) factory in Vietnam.

SKP Resources seeks to diversify its customer base while expanding its printed circuit board assembly injection molding and engineering capabilities.

Spectrum AMT was named manufacturing partner for **D-Orbit USA.**

Tata Electronics is in talks with major OEMs such as **Microsoft**, **Dell** and **HP** to diversify its client base and position the company as a major force in the global EMS landscape.

Texas Instruments is expanding its in-house manufacturing capacity and its AI capabilities for embedded and edge designs.

UK Circuits installed a new reflow oven and unloader equipment for its SMT line.

ViTrox appointed as representatives **MaRCTex** in Texas, Louisiana, Oklahoma and Arkansas; **ASC International** in Minnesota, North Dakota and South Dakota; and **Gemaddis** for the French market.

Volex will not pursue an acquisition of **TT Electronics** after expressing interest in the potential takeover earlier in November.

PCDF


Julien Lederman


Laura Russell


Michael Lopez

AdvancedPCB appointed **Gregory Halvorson** CEO.

GreenSource Fabrication named **Megan Tata** strategic account manager.

Nano Dimension appointed **Julien Lederman** interim CEO.

Rogers Corp. appointed **Laura Russell** chief financial officer.

Summit Interconnect named **Michael Lopez** vice president and general manager, Anaheim.

CA


Mark McClellan


Gareth Cuthbert


Katie Xu Yifan


Ryan Alstad


Jana Kieckhafer


Doug Henke


Kevin Krumm


Ross Berntson


Brian Oleary


Haley Reid

Dagmara Charyk

Kristin Schuetter

Amber Patton

Ag Express Electronics appointed **Mark McClellan** COO.

Altus named **Gareth Cuthbert** application and service engineer.

ASMPT promoted **Katie Xu Yifan** to EVP and CFO.

E-tronix named **Ryan Alstad** solutions engineer and added **Jana Kieckhafer** and **Doug Henke** to its sales team.

Flex appointed **Kevin Krumm** CFO.

Indium appointed **Ross Berntson** CEO and promoted **Brian O'Leary** to director of global accounts.

Kyzen promoted **Haley Reid** to analytical lab manager.

MicroCare appointed **Dagmara Charyk** regulatory affairs manager.

Mouser Electronics promoted **Kristin Schuetter** to senior vice president of products.

TestEquity named **Amber Patton** account manager.

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

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

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

Reliable answers. Reliable PCBs.

NCAB GROUP

PCB East 2025 Technical Conference Announced

PEACHTREE CITY, GA – PCEA this month will open registration for the technical program for PCB East 2025, featuring more than 75 hours of in-depth electronics engineering training.

Rick Hartley, Susy Webb, Dan Beeker, Tomas Chester and Zach Peterson are among the headliners of this year's conference. It will be held April 29 to May 2 at the Boxboro Regency Hotel and Conference Center in Boxborough, MA. It features classes for every level of experience, from novice to expert.

The scope of classes ranges from the basics of design engineering, documentation, libraries and circuit grounding to more advanced fare such as RF and mixed signal design, DDR5 routing, board stackups, simulation, controlling EMI, and power delivery system design.

More than half the presentations are new to the conference, including ones on maximizing routing channels, successful design release, differential pair design for 112 Gbps and faster systems, EMI shielding, flex design, UHDI design, controlling for tin whiskers, and AI and other upcoming changes in PCB design.

"PCB East continues to grow as the leading place for learning about electronics design and manufacturing on the East Coast," said Mike Buetow, conference director, PCB East. "This year's event offers an array of experts in printed circuit design engineering and manufacturing, and a new emphasis on assembly."

Registration for both the technical conference and the exhibition takes place at pcbeast.com. Those who sign up by March 17 can take advantage of the early bird special discounts for the conference.

The PCEA Conferences Task Group developed the program from more than 60 abstracts submitted. The task group, chaired by Troy Hopkins, is made up of 10 industry veterans with more than 200 years of cumulative experience in the printed circuit industry.

PCB East is the largest technical conference and exhibition for the electronics design, fabrication and assembly industry in New England.

The one-day exhibition takes place April 30 at the Boxboro Regency in Boxborough, MA, while the four-day

technical conference takes place April 29 to May 2. For details, visit pcbeast.com.

Mizzou Chapter Kicking off with DfM Talk

PEACHTREE CITY, GA – The new PCEA Missouri chapter will sponsor a talk on DfA by Dale Lee, staff DfX process engineer at Plexus, on Feb. 18 from 11 a.m. to 1 p.m. CST.

Today's SMT and PTH assembly processes are driven by component packaging technology and functional design requirements. These requirements, combined with the growth in automobile electrification, space commercialization, wearable/disposable medical electronics and no touchup/rework allowed assembly designs, are driving manufacturing/test toward "build right" and increased reliability requirements.

With this transition, DfM needs to migrate away from general assembly design guidelines primarily driven by general assembly equipment/process capability toward component/printed board-specific driven assembly. understanding the requirements and tolerances of each solder connection on a single device location on the assembly. Manufacturing now has to be able to report design issues not only from assembly-level effects/process customization but also to be able to communicate with designers on potential functional design mitigation strategies and/or cost impacts.

Manufacturing by design is now required to match the assembly/test processes to the functional requirements of the design and address when traditional material limitations, industry standards and assembly tolerances are not adequately addressed with traditional DfM reviews.

The presenter, Dale Lee, is a principal DfX engineer with Plexus, primarily involved with printed board/printed board assembly DfX analysis, root cause failure analysis and definition/correlation of design, process and tooling impacts on assembly processes and manufacturing yields including impacts of new technologies.

Lee has been involved in PTH/surface mount design, package and process development and production for 30-plus years in various technical and managerial positions. Activities have included research, development and implementation of advanced manufacturing technologies and interconnect techniques, component package design and development, PCB and PCBA design support, DfM/DfX analysis of flex, rigid-flex and rigid PCB/PCBAs including supply chain, process qualification and new process introduction for domestic and foreign low, medium and high-volume production applications for consumer products, industrial, aerospace, defense and medical devices.

Lee has authored multiple technical articles and papers and is a frequent instructor/presenter globally on topics including PCB and SMT design, assembly, cleaning, and DfM/DfX. He is a past recipient of the Surface Mount Technology Association's Excellence in Leadership award and IPC's President's Award. He has been or is very involved with multiple industry associations, industry standards development, and symposium/conference technical development committees.

To register: https://attendee.gotowebinar.com/register/506313452873561692

PCEA Issues Call for Abstracts for PCB West 2025

PEACHTREE CITY, GA – Abstracts are sought for the PCB West 2025 technical conference. The conference, the largest of its kind in Silicon Valley, focuses on training and best practices for printed circuit board designers and design engineers, electronics hardware engineers, fabricators and assemblers.

The four-day technical conference will take place Sept. 30 to Oct. 3 at the Santa Clara (CA) Convention Center. The event includes a one-day exhibition on Oct. 1.

Papers and presentations of the following durations are sought for the technical conference: one-hour lectures and presentations; two-hour workshops; and half-day (3.5-hour) and full-day seminars.

Preference is given to presentations of two hours in length or more, and no presentations of less than one hour will be considered.

Abstracts of 150-500 words and speaker biographies should be submitted to PCEA. Papers and presentations must be noncommercial in nature and should focus on technology, techniques or methodologies related to printed circuit board design, fabrication, assembly, test, components or packaging, and additive manufacturing.

Submit abstracts at pcbwest.com by Jan. 24, 2025. No emailed abstracts will be accepted. Submitters will be notified by early April if their abstract has been accepted. Presentations are due Sept. 5, 2025.

Speaker benefits: Presenters of accepted abstract(s) for the 2025 program receive the following event benefits.

- Complimentary access to the online proceedings
- Complimentary pass to the technical conference
- Invitation to the Speaker Reception

For more information about PCB West, visit pcbwest.com or contact conference director Mike Buetow.

PCEA to Hold Conference and Exhibition in Detroit in June

PEACHTREE CITY, GA – Printed Circuit Engineering Association (PCEA) will host a new technical conference and exhibition in June in Detroit for printed circuit designers, design engineers, fabricators and assemblers.

PCB Detroit will take place Jun. 2-3 on the campus of Wayne State University. It will include two days of technical sessions, plus a tabletop exhibition on Jun. 2.

Wayne State University is Michigan's third-largest university, and a licensee of the PCEA Training Certified Printed Circuit Designer curriculum.

The conference will feature presentations on controlling noise, low layer count board design, using AI in hardware and PCB design, flex and rigid-flex materials and DfM, and HDI via design, among others.

For information on attending, please contact pcbdetroit@pcea.net. For information on exhibiting, please contact Frances Stewart.

PCEA CURRENT EVENTS

ASSOCIATION NEWS

Designer education classes. Certified Professional Circuit Designer training and certification classes will be held in successive weeks starting Jan. 20, Feb. 28 and May 9, respectively. Each class is 40 hours long and includes a copy of *Printed Circuit Engineering Professional,* a 400-page handbook on circuit board design, and the optional certification exam.

The deadline to register for the class beginning Jan. 20 is Jan. 3. The deadline for the February class is Feb. 3.

The live, instructor-led classes cover the gamut of printed circuit design engineering, from layout, place and route to specifications and materials to manufacturing methods. Schematic capture, signal integrity and EMI/EMC are also part of the comprehensive program.

There are no prerequisite requirements to enroll. Upcoming classes will be held online. All courses are led by experienced instructors.

For information about the course overview, class format, and materials to prepare for the class, visit pceatraining.net/course-overview. To enroll, visit pceatraining.net/registration for the next available class or email pceatraining@pcea.net for additional information.

CHAPTER NEWS

Portland, OR. During our November meeting, we had an interesting discussion about electronic data transfer, including the Gerber X2 and Gerber X3 formats. We are looking for input from designers who use an EDA tool that exports Gerber X2 or Gerber X3, and from fabricators and assemblers whose CAM system imports X2 or X3, and how often (if ever) they receive those files from customers. (Here is a link to the Ucamco website as a reference for the latest iterations of the Gerber format.) Share your comments with Stephan Schmidt, stschmidt@pcea.net.

Our next meeting is Jan. 16 at 12 p.m. PST. Our guest speaker is Tony Lentz of FCT Solder, who will present on low-temperature solders and processes.

Finally, congratulations to Geoffrey Hazelett, who has been elected vice chair for our chapter.

SIEMENS

Introducing:
Next generation electronic systems design

Xpedition • HyperLynx • PADS Professional • Valor

MARKET **WATCH**

Taiwanese PCB Industry Enters 'Wait-and-See' Mode

TAOYUAN, TAIWAN – The Taiwanese PCB industry grew 9.6% year-over-year in the third quarter, reaching NT227 billion ($7 billion), the Taiwan Printed Circuit Association reported in December.

The association said the quarter was mainly driven by the peak season effect, the mild recovery of mainstream terminal products, as well as the specification upgrades of AI infrastructure (such as servers and network communication equipment) and the boost from the low-orbit satellite market.

Looking ahead to the fourth quarter, lower-than-expected sales of Apple phones and the uncertainty of global markets are driving a strong wait-and-see sentiment, TPCA said. Terminal and downstream enterprises remain conservative in stocking, and the overall growth momentum is restricted.

Taiwanese enterprises are expected to have global output value in the fourth quarter of NT$209 billion ($6.4 billion), a decline of 1.2% compared with the same period last year. The estimated total output value in 2024 will still reach NT$808 billion ($24.8 billion), however, an increase of 5% compared with 2023.

Overall, global production of circuit boards of Taiwanese companies is forecast to expand 5.7% in 2025, with total output reaching NT$854 billion ($26.2 billion).

For the quarter, Taiwanese PCB sales were mainly distributed among communications (37%), computers (21.8%), semiconductors (17.7%), automobiles (11.4%), consumer electronics (8%) and others (4.1%). Among them, the semiconductor sector witnessed the most prominent growth, driven by demand for AI chips and high-bandwidth memory HBM. Communication products output continued to increase due to demand for network communication products extended by low-orbit satellites and AI servers. Although sales in the PC market were lower than expected, demand for AI servers and general servers supported growth in the computer application category. Rapid inventory replenishment in the previous two years, combined with weakened demand and the base effect, has slowed automotive growth. Meanwhile, consumer products sales remained sluggish.

In terms of PCBs, multilayer boards (30%), flex circuits (24%), HDI boards (19.6%) and IC substrates (17.7%) made up the largest segments. Demand for IC substrates remained strong. Driven by demand for servers, ADAS for high-frequency applications and low-orbit satellites, HDI boards also showed good growth. Multilayer boards leaned on AI demand for growth, and network communication products required for the extension of infrastructure drove growth for multilayer boards due to architecture or usage upgrades. In contrast, flexible circuits demand was relatively weak in the quarter.

Enter the New Year

Trends in the US electronics equipment market (shipments only)

	% CHANGE			
	AUG.	SEPT.[r]	OCT.[p]	YTD
Computers and electronics products	0.0	0.9	0.6	1.0
Computers	-1.0	0.2	2.9	20.3
Storage devices	0.2	0.9	0.2	4.9
Other peripheral equipment	3.6	-2.7	-0.6	6.9
Nondefense communications equipment	-1.1	2.0	2.7	1.2
Defense communications equipment	2.6	-0.5	-2.5	-3.1
A/V equipment	-1.2	0.2	1.7	-2.7
Components[1]	2.2	0.0	-0.9	4.8
Nondefense search and navigation equipment	0.2	4.3	-0.1	4.0
Defense search and navigation equipment	1.3	-1.3	1.2	5.2
Electromedical, measurement and control	-1.3	1.0	0.9	-1.6

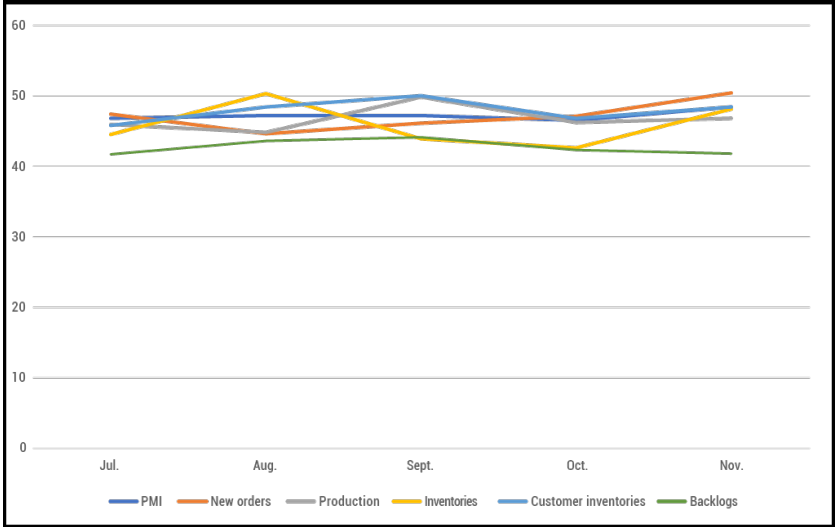
[r]Revised. [p]Preliminary. [1]Includes semiconductors. Seasonally adjusted.
Source: US Department of Commerce Census Bureau, December 4, 2024

Key Components

	JUL.	AUG.	SEPT.	OCT.	NOV.
EMS book-to-bill[1,3]	1.21	1.27	1.26	1.26	1.26
Semiconductors[2,3]	18.7%	20.6%	23.3%	TBA	TBA
PCB book-to-bill[1,3]	0.99	0.99	1.05	1.09	1.15
Component sales sentiment[4]	103.4%	108.4%	98.8%	97.1%	109%

Sources: [1]IPC (N. America), [2]SIA, [3]3-month moving average, [4]ECIA

US Manufacturing Indices



Source: Institute for Supply Management, December 2, 2024

Hot Takes

Global sales of **semiconductor manufacturing equipment** by OEMs are forecast to set an industry record, reaching $113 billion in 2024 and growing 6.5% year-over-year. China, Taiwan and Korea are expected to remain

the top three destinations for equipment spending through 2026. (SEMI)

Some 46% of component suppliers expect **electronics parts sales** to grow in the first quarter, while 7% expect a decline. (ECIA)

Global demand for **AI and high-performance computing** will continue to rise, driving the **semiconductor market** to grow over 15% in 2025. (IDC)

Notebook shipments are projected to grow at a compound annual growth rate of 4% from 2024 to 2029. (DigiTimes)

The global market for **personal computing devices,** including PCs and tablets, is set to grow 3.8% in 2024, reaching 403.5 million units. (IDC)

The **hardcopy peripheral market** rebounded in the third quarter, rising 3.8% year-over-year to approximately 20.3 million units shipped, following five consecutive quarters of YoY declines. (IDC)

Global semiconductor revenues surged in the third quarter, climbing 25% year-over-year to $177.8 billion. (Omdia)

Brazil's electrical and electronics industry is entering 2025 cautiously optimistic, with electronic components projected to grow 29% after a double-digit rise to $37.4 billion in 2024. (Abinee)

India's government wants to achieve $500 billion in **electronics production value** by 2030 with a plan to get global giants to build scale in the country by focusing on the domestic market and exports.

Worldwide **smartphone shipments** are forecast to grow 6.2% year-over-year in 2024 to 1.24 billion units. (IDC)

Prices of critical metals such as gallium, germanium and antimony used in electronic components and gadgets have begun to spike as China's trade restrictions with the US begin to bite. (Bloomberg)

China's IC output increased 8.7% year-over-year to 37.6 billion units in November. (China National Bureau of Statistics)

North American PCB shipments in November rose 4.7% year-over-year and fell 1.4% sequentially. Bookings were up 29.1% versus last year and down 13.7% from October. (IPC)

North American EMS shipments jumped 10.6% in November from a year ago and 4% from October. Bookings increased 8.1% year-over-year and slid 2.7% from the previous month. (IPC)

Affordable, Reliable Compliance

J-STD Storage of PCBs & MSD Components

HXD2-1106-01
Our largest heated dry cabinet

DRY CABINET STORAGE

XDry humidity-controlled storage cabinets are self-contained, low-maintenance storage systems. There is no need for nitrogen or compressed air, just plug into the wall and they are ready to go! XDry storage cabinets satisfy standards such as IPC/JEDEC J-STD-033 and IPC/JEDEC J-STD-020. The cabinet environment can be maintained from ambient to 1% RH, providing constant & stable storage at precisely desired RH levels.

XDry cabinets are the ideal solution for printed circuit board and moisture-sensitive component storage.

HXD1-703-02
Small footprint, but high capacity

XD1-302-02
Competitively Priced

XDRY CORP.

Corporate Office
3407 County Road 427
Anna, Texas 75409 USA

Phone: 214.296.4868

www.XDry.com

sales@XDry.com

ROI

The Global Economy is Here to Stay

Worldwide collaboration still has a place in the printed circuit board supply chain.

AS TIME MARCHES on and technology, as well as economic and geopolitical events, change and evolve, one thing is as true today as it was decades ago: It is *still* a small world, after all!

Being in a technology-driven industry, I often marvel at how small refinements to materials, processes and equipment, together, over time, are catalysts that enable truly significant technological advances. One person's – or team's – brilliant and possibly radical idea can become reality only when the little things are done in the way of refining existing materials, tweaking known processes and tuning the equipment necessary for its manufacturing. In almost all cases I can think of, the hype over so-called "game-changing" materials or processes, was for naught; the game did not change.

Over time, economies – and especially the geopolitical environment(s) – behave strikingly similarly to how technologies develop and become refined. The little things that fly under the radar tend to collectively impact events far more than the hype surrounding a global leader's words or actions. As bold or brash as they may seem at the time, more often than not, the result is far from what was promised, expected or feared: The hype was for naught and the game was not changed.

For over half a century, industry has quietly been evolving from the inefficiency of different companies in far-flung parts of the world trying to create and develop the same things to the far more efficient approach of working together to more quickly create and develop new technologies and products. Companies have opened research and development (R&D) sites in many regions of the world to get the best minds collaborating on next-gen products. The same is true with joint ventures where two or more companies leverage their respective strengths: working together to deliver a better product faster and more economically for all. The printed circuit and EMS industries could be considered poster children for how global collaboration can result in highly cost-effective, high-technology products that benefit consumers and manufacturers alike.

Despite the proven efficiency of an interconnected economy, some think the time has arrived to revert to an earlier model, before different companies – and countries – worked together to more efficiently develop and build products. And some proponents of this are hyping their thoughts in bold and brash ways that attract headlines and much hoopla, especially in social media.

Well, call me old fashioned – or just old enough to have started my career in the infancy of globalization – but I'm not buying the brash hype that reverting will be beneficial and replace globalization's efficiency.

Some claim the global economy has made it too easy to lose control of intellectual property or controlled unclassified information (CUI), etc. Well, it doesn't matter where product is developed, produced or used; there are ways to protect what is important without closing borders. The same can be said about the cost-effectiveness of where product is made. Having fewer eggs in any one basket may be a more effective way to mitigate risk than moving everything into one facility, regardless of where it is located. In short, risk has always been, and always will be, one of the considerations when determining what R&D or manufacturing initiatives take place. Small, quiet moves to protect what is important have always been more successful in achieving long-term success than hyped and brash "game-changing" ideas.

Others cannot get the supply chain issues of recent years out of their minds. While Covid impacted everyone everywhere, supply chain problems were not caused by a fundamental issue with the globalization of industry, but rather a singular event that is now, thankfully, mostly behind us.

Meanwhile, I am willing to bet that as the hype and headlines criticizing the global economy circulate, cooler – and quieter – minds are at work refining processes, tightening security and utilizing the best available resources to make an efficient global economy and related global logistics even more so.

Which brings me back to the fact that regardless of headlines or hype, brash and bold comments alike, it is *still* a small world, after all! Regardless of where you are located, businesses, economies and people need each other more than not. Despite the current environment where so many are focused on headlines rather than results, don't write off the global economy for electronics – or any industry – quite yet.

On the current state and future of globalization, I am siding with history and decades of experience that show how globalization has only made business, technology and industry more efficient and cost-effective for all.

PETER BIGELOW is president of FTG Circuits Haverhill; (imipcb.com); pbigelow@imipcb.com. His column appears monthly.

BOARD **BUYING**

PCB Buyer and Supplier – A Love-Hate Relationship

Ask questions now to save heartburn later.

I HAVE BEEN selling PCBs for over 30 years and have numerous longtime customers with whom I've developed close friendships.

But as with any relationship, there are ups and downs. Given the dynamics involved in being a buyer under pressure to obtain good quality boards at the best possible price, or in my case, being a supplier expected to ensure those perfect PCBs are delivered on time and with no problems, things can sometimes get testy.

Frustrations can arise, especially when the supplier needs to maintain a profit margin either through raising prices or increasing the quantity of boards purchased, while the buyer is expected to constantly cut costs without hurting production.

When a customer relies heavily on a particular supplier and that supplier faces circumstances beyond its control that affect pricing, delivery or quality, the relationship can turn contentious and must be carefully managed to avoid damaging your supply chain.

Here are several love-hate scenarios common to our industry:

Too many engineering questions (EQs) from supplier. Every buyer would like to place a purchase order with no questions asked and get on-time delivery and perfect boards. But the reality is that the PCB is custom-made. Quality is directly proportional to the number of EQs received. Discouraging questions lead to dangerous assumptions and set up the supplier for failure. Buyers should be willing to help their suppliers. Sure, time is of the essence. But it is better to have correct boards arrive late than on time and wrong.

When a supplier communicates bad news. No one likes bad news. But a supplier delaying bad news is worse. Buyers need to encourage suppliers to communicate bad news immediately. Buyers need to be thankful for this heads-up from the supplier, as it gives them an opportunity to lessen the possible blow to the production schedule. Likewise, the supplier must constantly follow up with the buyer until the situation is resolved. No buyer I know of has ever died from hearing bad news about an order. Poor communication sure kills business, however. I've known quite a few buyers I call "screamers" who discourage all bad news. They deny themselves the opportunity to be proactive instead of reactive.

Buyers who do not give feedback. The phrase "no news is good news" does not apply to our industry. While I don't think buyers should discourage bad news about their orders, I do like it when buyers feel free to openly communicate their concerns. If they tell me what they need, I will do all I can to meet those needs. I can also feel free to tell buyers what I need from them to get the job done. Dialogue is great for strengthening relationships. Of course, it is best to communicate in a manner conducive to business. I prefer a buyer who is blunt, however, even to the point of rudeness, over one who clams up until it's too late to fix a problem.

Suppliers who do not make suggestions. The days of having physical drawings and artwork on hand for review are gone. In today's digital world, many buyers, through no fault of their own, do not understand the complexities of making a circuit board. Unfortunately, many buyers don't know what they are buying, or how they could have bought differently and made a better deal for their company, both in cost and for better assembly throughput. I can't tell you how many times a previous supplier left one of my buyers hanging. Is it a sales technique, product ignorance, or plain laziness? Regardless, the supplier needs to be asking questions and volunteering solutions even when – *especially* when – buyers are unaware of what they're facing.

Slow payment by buyers. I have put customers on shipping or credit hold, only to have buyers' bosses call me fuming that I dare hold the order. I always ask why it's unacceptable to deliver an order weeks late, but it seems to be okay to make payment weeks late. Suppliers should never have to apologize for asking for timely payment, especially when quality goods were delivered on time. Also, a customer who pays timely is more easily accommodated by the supplier when their buyer is demanding.

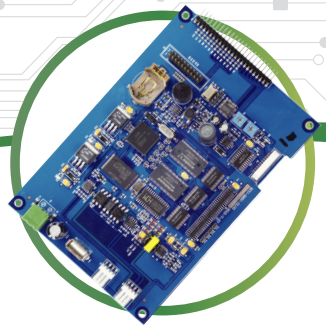
The relationship between buyer and supplier is dynamic and can sometimes become tense. That tension can be managed through consistent communication. Encouraging an open dialogue with clear expectations in good *and* bad times builds trust, as does a supplier steadily offering on-time delivery of quality product. When buyer and supplier collaborate, they can find solutions to the many issues that will arise, and this strengthens the partnership and the supply chain.

GREG PAPANDREW has more than 25 years' experience selling PCBs directly for various fabricators and as founder of a leading distributor. He is cofounder of DirectPCB (directpcb.com) and can be reached at greg@directpcb.com.

PCBWay

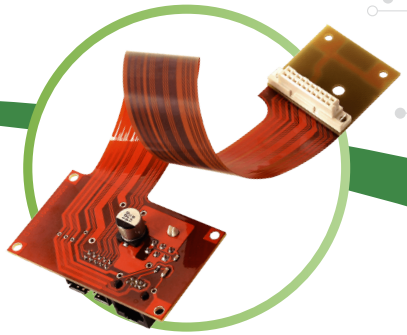
FULL-SERVICE ELECTRONICS MANUFACTURER

PCB Capabilities:



PCB Assembly
From $29

- Improved assembly precision
- Short turnaround for assembly
- Flexible support



Flex, Rigid-Flex PCB
From $46.74

- Up to 16 layers Flex circuits
- Polymide Flex, PET, High Frequency
- 0.025-≥0.4 mm FPC thickness

PCB Prototype
From $5/10 pcs

- Multiple color options
- 24-hour production time
- Worldwide delivery

www.pcbway.com

service@pcbway.com

Talk to Every Prospect

You never know what you'll find in the bushes.

A SIMPLE CONCEPT: Business development should qualify each lead or prospect to determine if the electronics manufacturing services company's resources will be spent pursuing that prospective client.

The decision of "fit" should be the job of the senior business development professional, not the CEO, GM, COO or CFO. Elements of fit include the end product's viability, the prospect's financial standing, the tooling and capital equipment required to complete the job, the floor and warehouse space needed (in the case of a box-build program) and other elements of the deal.

How does a company know the fit is right unless properly vetted by a biz dev professional?

Case in point: When I was vice president of sales and marketing for a Florida-based mid-tier EMS, I received an intriguing call from an inventor in South Florida. I agreed to meet him at our factory to understand the product and requirements. After giving explicit directions to our visitors' entrance, my admin called to say he arrived at the employees' door. As an ITAR facility, specific protocols must be followed for proper vetting and sign-in. I asked the admin to let my guest in and escort him to the conference room after he executed the proper procedure and signed the appropriate paperwork.

Then came the panicked call: "He's urinating in the bushes! What do I do?"

The obvious response was, "Well, don't shake his hand." He washed up and came to the conference room full of apologies.

Prints and a breadboard also came, and after a few minutes of my inquiries, it was clear this was not a fit for us, as were many potential deals in my 10 years there.

But how would anyone know unless you devoted a little time to vet each deal? Many of my counterparts bragged that they could discern which products are winners and losers with virtually no vetting. Where are my BS cards when I need them?

Conversely, the risk of proceeding is well-vetted when a product fits into an EMS's desired market, has a robust design, solid test verification, and a good chance to scale by filling a need. The company behind the product should also have a war chest of money and offer reasonable terms and conditions. Most CEOs don't have the time to vet these deals. Most COOs have too many day-to-day issues to deal with each opportunity. Most CFOs have too little

experience in the go/no-go decision process. And most GMs have some bias in the decision.

If the vice president of biz dev can wear the hats of both a businessperson and a salesperson, trust them to either make such a decision or, if conflicted, trust they'll ask for help on how to proceed.

Not every decision I made to walk away or invest was the correct one. New data emerge as time passes. Provided you have a high hit rate of good decisions over poor ones, however, the EMS should prosper.

And be vigilant when a "go forward" decision is made only to be proved wrong. In those situations, take quick action to fix detrimental issues or kill the deal.

JAKE KULP is founder of JHK Technical Solutions, where he assists OEMs and EMS companies with optimizing demand creation offerings and deciding when and where to outsource manufacturing. He previously spent nearly 40 years in executive roles in sales and business development at MC Assembly, Suntron, FlexTek, EMS, and AMP Inc. He can be reached at jkulp@cox.net.

Measure PCB Surface Finishes ENIG, EPIG, EPAG & ENEPIG

- XRF with Automated X-Y Table
  - Increased Throughput
  - Improved Reliability
- IPC 4552 B & IPC 4556

Measuring Made Easy®

www.fischer-technology.com | info@fischer-technology.com | 860.683.0781

fischer®

DESIGNER'S **NOTEBOOK**

So You're a Designer. Now What?

PCB designers can have many different career pathways.

THE ROUTE A PCB designer takes through the job market can lead to a number of different outcomes.

A board designer comes in with a knowledge set that helps transform an abstract schematic into physical electronics. A lot of that has to do with knowing what happens downstream from the day the artwork was created. The steps involved with PCB fabrication and assembly are complex, even for the simplest of jobs.

Take a factory tour and notice the rooms full of different machinery. Material is cut to shape and drilled in a kinetic energy field that makes so much noise. Vast plating lines do serial dunking in different vats of bubbling hazards. These are the CAD data manifested in copper for the first time.

Another room is more Zen, with tons of pressure and high temperature being applied. It's a slow process, and the presses are very expensive. This is usually a small factory's bottleneck. Down the hall, a brightly lit room full of automated optical inspection equipment ensures compliance. What we do on the monitors plays out across the factory.

Physical design, building blocks and fitting them together. Crossover points in the machining and drilling are seen over and over in a machine shop. Creating the drawings and CNC data for the single board and the "step and repeat" assembly panel is work that must be done in every PCB fab shop.

Why stop there? If you have a penchant for creating virtual widgets using CAD tools, you could continue to learn about 3-D modeling and parlay that into generating the heatsinks, shields, base plates and miscellaneous brackets for your team.

Generating the documentation for piece parts uses the same dimensions and other conventions as the PCB. Metal parts get anodized or otherwise protected from the elements. These material callouts are standard; pick a color.

Figure 1. The factory floor is where it all comes together.

In the age of 3-D printing, we can prototype quite a few things. Getting into finite element analysis and creating intricate geometry with a calculated breaking point requires an engineering degree, even if the latest software can get you there. Material science covers a lot of interesting ground. You can start simple and grow into it like anything else.

Think small to earn big on the chip team. It seems that the deeper you go into the PHY stack, the nicer the cars parked in front of the office. The chip team has a lot of sway in most chip companies. Getting closer to the IC by designing the substrates or modules can be a step toward the silicon. That's still on the periphery but close enough that you might get in on the action with the right company.

Forward-thinking companies with strong financials are the ones that can provide tuition assistance. I gave Qualcomm almost seven years and was grateful for the chance to go back to school part-time on its dime. Maybe they can buy your startup too. All I'm saying is that it pays to miniaturize your CAD outlook if you get the opportunity.

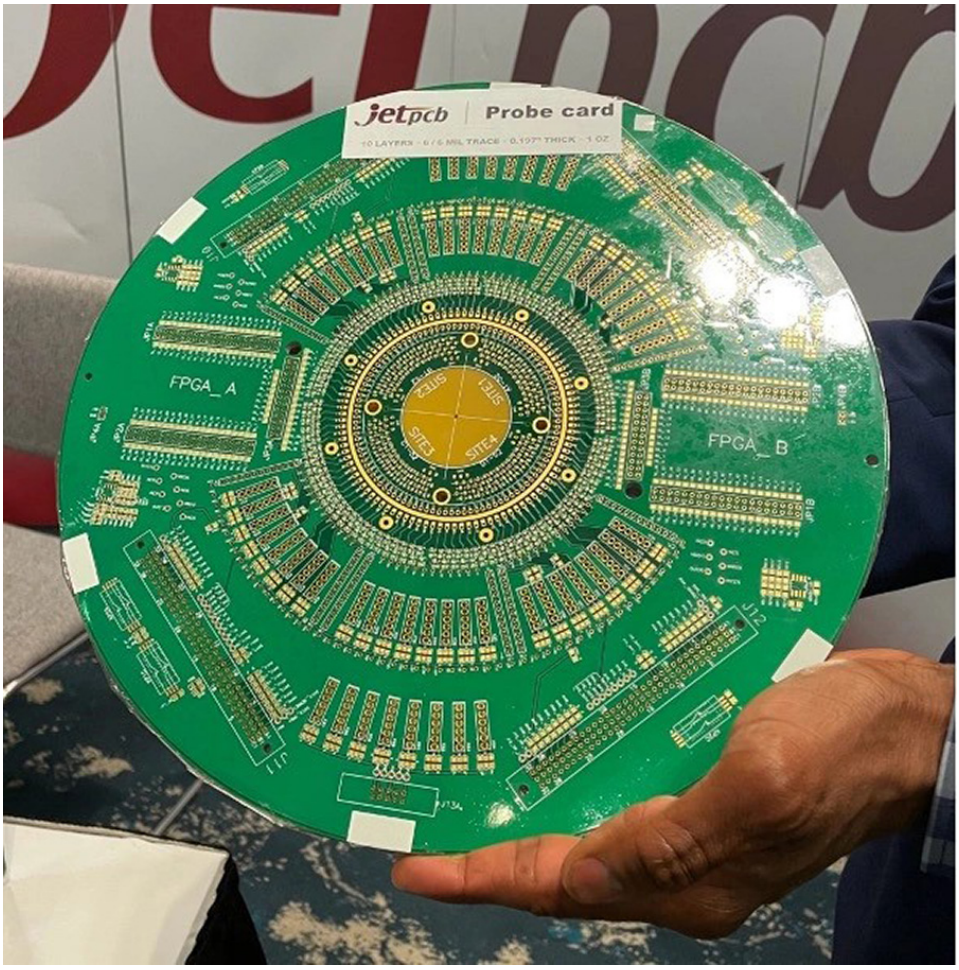


Figure 2. Get the chip team's attention with probe cards that help them quantify the device's throughput.

Circuit boards are at the end of that range, but the concepts do carry over. You're mainly stitching together IP blocks with vanishingly small features. Of course, this job has increased in complexity over the years. They still need new people to join their ranks as the OG experts retire.

CAM tools expert, the conduit to the copper. If you have been submitting PCB artwork to a fab shop for any length of time, you've probably gotten a notice that they have a question about the data you provided. It may be something like asking if an indicated location is acceptable for the vendor's logo and date code. That's an easy and expected question.

No matter the gravity of the question, cosmetic or catastrophic, someone must be the fab shop's backstop, and that's the CAM person. Every data package that comes into the fab house goes through the CAM evaluation. Their tools and experience can turn the average artwork into an actual phototool. Don't think for a minute that the data you send them doesn't get a workover before it hits the factory floor.

When it comes to the worse-than-average submission, they get to walk the customer through their options that fall within that shop's production capabilities. We've all been there when a connector or similar item has its own geometric demands. You can and should include those tight tolerances in the package if that's what the datasheet says. Then, tape out the board and see what happens.

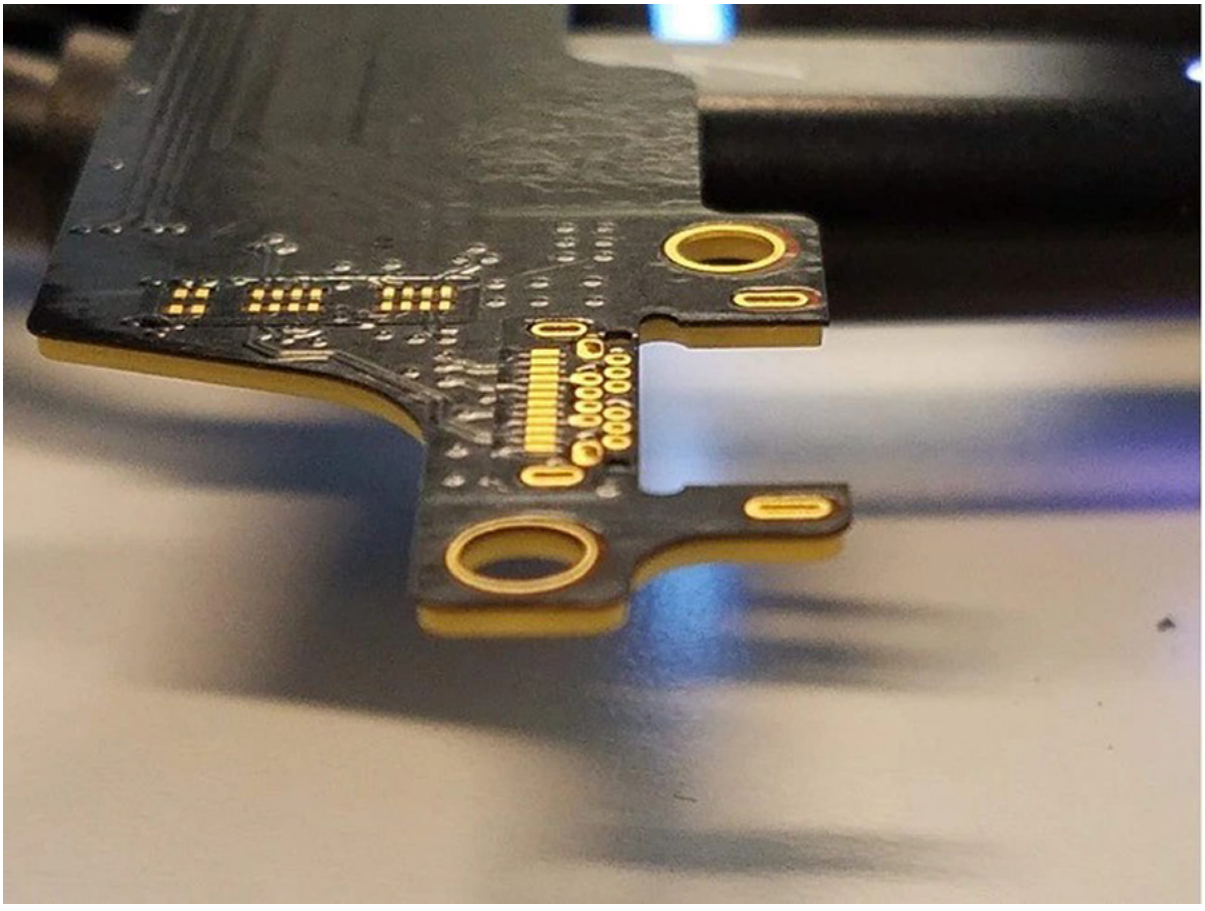


Figure 3. USB-C connectors are hard enough to incorporate without going on to a flex circuit where tolerances are always a bit more “squirrely.”

Nonsense! It’s the PCB designer’s job to make sure the information in the datasheet aligns with the capabilities of their approved vendors. We can’t always know the pain point, so it’s best to get ahead of these things as a designer and check with the local CAM department. When we don’t, we get that call, often on a Friday around 6:30 p.m. local time. Anyway, you could find yourself making that call if you wind up working in computer-aided manufacturing.

Starting or managing a business unit. Being a contractor exposes you to different types of design work as you move from one assignment to another. It’s like a lateral move, but it’s also slightly upward when you start a new project. I was doing that kind of hourly work when my manager from the previous company asked me to consult for their new employer.

I did my contract work on edge routers from 7:00 a.m. to 3:30 p.m. and went up the road from 4 to 8 p.m. to design what was called WiMAX equipment. The six-month contract for the routers ended, and I went full-time on the wireless team. That assignment lasted for about a year before I got “headhunted” and returned to life as a corporate employee doing those ceramic substrates.

Variety is king since most employers feel better with someone who has equivalent experience. The web has made it easier to market yourself for this niche and grow a decentralized business that can keep the lights on ’round the clock. That is an important marketing draw for a service bureau.

Working around the clock for PCB design jobs. You have to leverage geography and let your partners work while you rest. You can go solo, but then you'd miss the opportunities where the timeline is compressed for the size of the job. Corporations outsource the layout when they do not have the resources in-house. You may be called upon only during the off-hours in your location if they have a captive designer for that shift.

Being in business for yourself gives you a lot more to do. When my customer was late paying the invoice, I emailed the CEO explaining that I was charging him the usual rate while composing this email since I didn't have a collections department. He told accounts payable to cut me a check for everything owed, even those invoices that were not yet due. It was a good drive home that night since I had evidence that I was actually working all those hours.

If that kind of insecurity concerns you, and it should, you might be able to rise within a company into a CAD management position. For me, the extent of that was working with onsite contractors to schedule and course-correct their work. To be honest with you and myself, I'm not cut out for that kind of work. I'm rather obsessive/compulsive about the boards I design and don't think it's fair to impose that level of scrutiny on others.

CAD management. Managing full-time employees is a commitment to their well-being. You shine only to the extent that they shine, and you own whatever blunders come down the road. As first-level management of creative people, it's important to have empathy and patience with the inevitable mistakes we all make from time to time.

Setting expectations around the job's uncertainty is key. We know there will be iterations. We know it will be most intense as the schedule winds down to go time. The people will be prepared to succeed if good guardrails are in place that pave the path to a product launch. These guardrails are repeatable processes that close the loop between getting information and echoing it back to the sender with clear data. Everything is checked off; if not, an erratum is created to note where we pushed the limits.

Processes would ensure that we've learned from our previous mistakes and incorporate those improvements as we proceed. Although the inherent uncertainty resolves only in the 11th hour, that is the way to get the very latest technology into the product that goes out to the world at the appropriate hour. Balancing long-term improvement with bursts of design activity is one of the things that keeps a manager on their toes.

Learning PCB design is more than using the tools. We're tightly wrapped around the mechanical attributes while fleshing out the electrical universe. Straddling these two camps puts us in touch with groups such as:

- Electrical and mechanical engineers who define success for the PCB
- Component engineers who evaluate the chosen parts for usability
- Signal and power integrity (SI/PI), where they enter their own virtual worlds to ensure performance
- Procurement folks who can goad the vendors into supplying what's needed at a time and cost that makes sense to the bottom line
- Fabrication and assembly groups, where our knowledge of their processes informs almost everything we do as designers

- Test and reliability, who put the product into different corners of the specification where maximum and minimum conditions collide – just to see what happens.

By the time you've done this a few dozen times, you can start to think about how you might like to expand your responsibilities while leveraging what you already know. Until someone showed me a PCB design on a computer screen, I didn't know I wanted to do this. Maybe the best use of your knowledge as a designer is to share your passion and teach others how to get into design or become better in areas they want to improve. This is something I wrote in answer to a question from one of you. Stay curious, my friends.

JOHN BURKHERT JR. is a career PCB designer experienced in military, telecom, consumer hardware and, lately, the automotive industry. Originally, he was an RF specialist but is compelled to flip the bit now and then to fill the need for high-speed digital design. He enjoys playing bass and racing bikes when he's not writing about or performing PCB layout. His column is produced by Cadence Design Systems and runs monthly.

ENGINEERING *TOMORROW'S* ELECTRONICS

PCB EAST 2025

Conference & Exhibition

Conference: April 29 – May 2

Exhibition: April 30

Boxboro Regency Hotel & Conference Center, MA

WHO'S EXHIBITING

AB Electronics LLC
ACDi
Accurate Circuit Engineering
All Flex Solutions, Inc.
Arena AI
ASC Sunstone Circuits
Chester Electronic Design
Cofactr Inc.
Component Dynamics, LLC
DirectPCB
DownStream Technologies, Inc.
Electronic Interconnect
EMA Design Automation
Fischer Technology Inc.
Flexible Circuit Technologies
Freedom CAD Services
GS Swiss PCB AG
Imagineering, Inc.
InstaDeep
JBC Tools USA Inc.
Newgrange Design
NextPCB
OKI Circuit Technology Co., Ltd.
PalPilot International Corp.
PCB Technologies – USA
Polar Instruments, Inc.
Printed Circuit Engineering Association (PCEA)
Quantic Ohmega-Ticer
Screaming Circuits
Siemens
SiliconExpert
Surface Mount Technology Association (SMTA)
Syntronic Production Services
Trylene Inc.
Winonics

Registration Details Coming Soon!

DESIGN **BEST PRACTICES**

Addressing the Global Talent Shortage

Changing dynamics in the workforce are driving the need for an intuitive user experience. Are the tools up to the task?

THE DEMAND FOR electronics is surging globally, driven by the rapid pace of technological advancement and innovations in sectors like automotive, telecommunications, healthcare and consumer electronics. PCB design, the backbone of electronics hardware, powers everything from smartphones and medical devices to industrial machinery and aerospace systems. PCB design plays a critical role in enabling technological growth.

As the demand for advanced, high-performance electronics devices grows, so does the need for skilled engineers and PCB designers who can navigate the complexities of creating reliable and innovative circuits and board designs. The changing demographics of engineers and PCB designers reaching retirement, the growing talent gap and the limited pipeline supply of new talent have stirred up many discussions and debates on how to address this need.

The reality is fewer people will be available to do the work on both the engineering and designer sides. In the US, at least one in every three posted engineering jobs will go unfulfilled, per a Boston Consulting Group's analysis published in 2023.

Impacts on PCB design teams and companies. While attending industry conferences, lecturing at college campuses and participating in many social media activities in 2024, several discussions I had seemed to converge on the talent gap and expertise shortage. The negative effect is already felt throughout the industry. Engineering teams are already stressed by having to do more with less while under reduced budgets, limited resources and shortened project schedules.

The talent gap is not merely a logistical concern; it has business implications. Companies are finding it increasingly difficult to meet market demands, leading to delays in product development and heightened competition for the few skilled professionals in the field. Moreover, the global nature of the electronics industry means that companies worldwide are competing for the same limited pool of experts.

Addressing this talent gap is critical for maintaining the pace of innovation. Without skilled designers, companies face increased time to market, higher production costs and potential reliability issues. Strategies to attract, train and retain talent in PCB design are urgently needed to ensure the industry remains agile and competitive in the face of growing challenges.

Let's specifically dive into the talent shortage by looking at two primary challenges in this context:

1. **Skills gap.** Attrition is one of the main factors contributing to the growing skills gap. With an aging workforce that is either retiring or leaving the industry, demand for trained specialists is at an all-time high and there is simply not enough available talent to go around.

   Many professionals entering the market lack expertise. Most college graduates do not have sufficient knowledge or skills to be instantly productive. The industry continues to outpace educational programs, leaving a disparity between the skills graduates possess and the technical demands of companies. These college graduates typically lack the training, understanding and skill set necessary to successfully design even the most basic PCB. Yes, these are college graduates, but they require time-consuming training specifically regarding the core principles of PCB design and EDA tool usage to compensate for necessary knowledge that was not provided by their educational institutions. The sooner they can get up to speed and become productive contributors within their company, the faster their individual value increases, which has a direct positive impact to their company's financial bottom line.

   The second prong of this issue is related to veteran professionals re-entering the job market. Many of these professionals have experience but may lack specific knowledge of advanced technologies and domains such as flex, RF, high-speed, mixed-signal, HDI and UHDI. They also lack familiarity with advanced EDA tools, which offer features like automation and AI. They might also lack exposure and experience with cloud-based environments for collaboration.

   This means companies may struggle to innovate or meet project deadlines due to insufficient expertise. This could lead to increased training costs, reliance on external vendors, product quality risks increasing or delayed product launches, affecting competitiveness.

2. **Competition for talent.** Experienced engineers are in high demand, leading to fierce competition among firms, including startups, established companies and even nontraditional players in tech, like automakers. The global talent pool is limited, and companies are vying for the same small group of highly skilled professionals.

   This means employers face escalating salaries, greater turnover and difficulties retaining talent. Establishing and maintaining attractiveness as a workplace is critical in such a competitive environment.

Together, these challenges require companies to rethink hiring strategies, invest in upskilling and collaborate with educational institutions to bridge the skills gap in the long term. But how can the issue be solved in the short term?

The shortage of talent has not only pushed engineering teams, but it has also pushed the EDA tool vendors to upgrade their solutions to compensate and fill the skills gap. Making EDA tools that are more intuitive and easier to master with shorter learning curves is the solution. Intuitive is the key word here, and what I mean by intuitive is this: Today's tools, whether they are utilized daily or seldom, need to be instinctive for users to understand and use without prior or consistent experience, or without specialized, in-depth training.

Software evolution is the key to bridging the skills gap. For PCB designers, professional development has two aspects: mastering the craft (skillset) of printed circuit engineering and mastering the ECAD tool(s). These go hand in

hand.

After three decades in the field, I can honestly say this is not an easy journey from either perspective. Obtaining education and knowledge on the core principles of printed circuit engineering, including the ever-evolving industry's best practices, is an ongoing quest. Because of the lack of colleges and universities providing the necessary PCB design curriculum I needed, I was forced to rely heavily on industry conferences, tech summits and webinars to fill the educational gap. Acquiring the necessary knowledge and experience is only half the battle, however.

Mastering an ECAD tool is an entirely separate challenge. It simply takes time in the form of many hours engaged with the tool. Many engineering tools are not user-friendly, with steep and lengthy learning curves.

When I entered the industry, it was acceptable and even tolerated for EDA tools to have lengthy learning curves. It was a challenge to get up to speed quickly, to say the least. It was not a priority for the EDA vendors to focus on ease of use because most tools were used by a specialist with in-depth training and experience. Times have changed, and EDA tools need to keep pace with other software designed to facilitate a smooth and frictionless user experience.

Software you don't use regularly is especially challenging. You may go through initial training on a particular tool, but if not used regularly, trying to recall exact process steps or how to manipulate the interface takes up much time, regardless of whether you received formal training.

As I grew in my profession, maintaining the highest optimization potential was always a goal, yet not always achievable due to the learning curve of new tools. I spent valuable project time going through this repeated learning process. It may not seem like a long time per project, but it adds up – and the cost is real!

It's especially tough when getting up to speed on a tool outside of one's main layout tool, such as simulation, analysis or DfM tools. Like many today, I took on responsibility and tasks outside my realm due to the lack of in-house expertise. With such specialized tools that I did not use regularly, mastery was a challenge. For example, when getting refamiliarized with a user interface, figuring out how to manipulate specific features, functions and the next logical process steps can sometimes be very tough.

At the start of a new project, I would often find myself putting my own project tasks on hold for a short time to assist a fellow team member who was new to the tool, or to retrain an individual on the software to get them up to speed due to the lack of the tool's intuitiveness. This training/retraining process happens regularly, taking up the time of both designer and team manager, and is an inefficient way of functioning, especially when we talk about the "true cost" of engineering.

This happens frequently, especially in large enterprise companies, where engineers in large, specific ecosystems may only use a tool once or twice throughout the year. The daily repetitiveness and knowledge retention of wielding a specific tool are simply not there. This leads to a repeated learning process, with an indirect cost to the employer.

This non-productivity has a direct effect on innovation as well, potentially reducing it. A designer's direct value to the company as a high return on investment (ROI) employee is lessened during those non-productive times due to the

time being directed at lengthy learning or relearning of a tool, rather than dedicated to innovation.

Today's tools for the next generation of electronic systems design need to address this steep and lengthy learning curve. They need to empower users and lower the barriers to adoption. Engineers cannot be burdened with nonproductive time and tasks.

Shifting from seeking experience to high ROI. In the struggle to acquire engineers, most companies typically search for specialists familiar with a particular ecosystem or software experience. As mentioned however, this pool of experienced talent has shrunk. Per PCD&F's 2024 designer and design engineer salary survey regarding "job satisfaction," on a scale of 1-7, with one being completely dissatisfied in their position and seven being highly satisfied, 65% of respondents rated their satisfaction as a 5 or higher, with 21% giving the highest rating of 7. That means many are content and not so eager to jump ship.

So, companies must adapt. We see this from engineering teams that do more with less simply because they have no other choice. Engineers are upskilling and cross-pollinating between disciplines. This can be challenging, but with intuitive software, companies can shift their focus to obtaining candidates that are aligned with the company culture and values, and more importantly – are trainable, eager to learn and adaptable, versus trying to acquire that hard-to-find specialist trained on a specific EDA tool.

As I finished my industry conference tour of 2024, which consisted of attending PCB West, PCB East, PCB Carolina and a few tech summits, I can say the industry is characterized by inexperienced engineers. So, with intuitive software tools, companies don't have to struggle to bridge the gap. By utilizing such software tools, they can onboard an engineer with less experience or upskill existing engineers, maintain high productivity and, of course, be more innovative.

The global talent shortage in the PCB design industry is a pressing issue that threatens innovation, time-to-market goals and competitiveness. As advanced technologies demand increasingly sophisticated designs, the scarcity of skilled professionals has created challenges beyond traditional hiring practices.

The industry cannot rely solely on experienced specialists to bridge the gap; instead, it must cultivate adaptable talent and create environments that empower engineers through advanced, user-friendly software solutions.

STEPHEN CHAVEZ is a senior printed circuit engineer with three decades' experience. In his current role as a senior product marketing manager with Siemens EDA, his focus is on developing methodologies that assist customers in adopting a strategy for resilience and integrating the design-to-source Intelligence insights from Supplyframe into design for resilience. He is an IPC Certified Master Instructor Trainer (MIT) for PCB design, IPC CID+, and a Certified Printed Circuit Designer (CPCD). He is chairman of the Printed Circuit Engineering Association (PCEA); stephen.chavez@siemens.com.

New Technologies, New Opportunities

Will 2025 bring new solutions to familiar challenges?

THE DESIRE TO solve problems is ingrained in human nature. But we know that our solutions often create new problems. It's a cycle that will likely never end.

So if you want to know what's coming in 2025, the short answer is nothing we didn't already know. Important trends will include, of course, more AI spreading from cloud to edge, as well as developments in commercial space exploration and sustainable technologies. Each of these presents exciting opportunities while also being the subject of dire warnings if things were to go wrong.

Elon Musk recently suggested that the chances of AI turning out badly are about 10-20%. That sounds alarmingly high. Without delving more deeply into his comment, a logical, if slightly facetious, response is that the chances of AI working in our favor must then be 80-90%. That sounds more encouraging. In practice, however, we must be prepared for good and bad outcomes. Legislation is beginning to arrive as technology acts proposed in the US and Europe put forward restrictions on the use of the AI, including prohibiting undesirable practices like deception, social scoring, biometric categorization and untargeted facial recognition.

It may already be too late to hold back the coming changes. It's a fact that AI has quickly become deeply infused into our lives and has fundamentally reshaped our interactions with technology. Our expectations have risen considerably as products and services have become smarter, faster and more intuitive with AI. We now expect personalized recommendations in entertainment and shopping, AI-powered assistance with domestic tasks, accurately curated content on our social media platforms, intelligent tools to enhance our productivity at work and many more. These AI-powered innovations that support our digital lives are now accepted as the norm and, in some ways, even taken for granted.

Among the tools redefining our expectations, AI PCs enhance creative and professional tasks by automating repetitive processes, accelerating data analysis and providing AI-driven content creation. They also benefit from adaptive features that intelligently allocate system resources and improve battery life. Shipments are rising sharply and are predicted to increase from just over 20 million in 2023 to exceed 100 million units in 2025 as more users embrace AI-powered tools and AI is no longer seen as a luxury but a standard feature in modern technology. These machines, defined by the addition of a neural processing unit (NPU) alongside the typical combination of CPU and graphics processor, are expected to account for 43% of all PC shipments in 2025 and dominate by 2026.

Demand for neural processing accelerators will profoundly affect the semiconductor industry, which is predicted to

bounce back from declining sales in 2023 to achieve double-digit growth in 2025. We will see the effects of this in the PCB industry, too, as demand for IC substrates rises to become a significant proportion of the overall market. Among the stringent technical demands of this sector, the substrates are typically extremely thin, often down to as few as 10 microns. This is equivalent to only two filaments of ordinary glass fiber. Right now, however, only a tiny percentage of IC substrate manufacturing, and IC packaging in general, happens outside of Asia. This shows there is still much work to be done if the West is to significantly increase its indigenous chip-making capability and establish the resilient and sustainable supply chains that are one objective of the US Chips Act and comparable European legislation.

With the coming change of administration in the US, tariffs may become the main tool discouraging overseas sourcing of chip-packaging components and services. On the other hand, the entire packaging value chain is fundamentally globalized, and this may continue to be the most cost-effective option compared to starting up domestic production from today's almost non-existent base.

Alongside AI, the commercialization of space is similarly contentious but unstoppable, given the benefits available and the sheer fascination associated with such unfamiliar and unquantified opportunities. Among these, direct satellite-to-cellular communication services are about to begin and can help extend connectivity into remote locations that have not yet been reached by conventional terrestrial cellular networks.

Moreover, we could soon be mining asteroids for scarce minerals, using autonomous, space-roving refineries that capture and process ores remotely before sending them back to Earth. And humans' return to the moon – currently being bankrolled and promoted by well-known tech billionaires – is expected to grow what we now know as the cislunar economy, encompassing activities like resource extraction, infrastructure development, scientific research and even space tourism. Ultimately, these could provide the starting point to seek economic growth and project human life even further beyond Earth.

As far as the development and adoption of sustainable technologies is concerned, almost all organizations harbor objectives in this area right now, and the drive to establish more sustainable practices will continue. Investors and ethical consumers demand it, and it is vital that we continue seeking new techniques that enable us to use less energy and reduce the consumption of materials such as rare minerals to produce the things we need. The world has finite resources to share between its large and still-growing population, and this challenge will likely become tougher.

Electrification, as a supporting principle of the drive for sustainability, is founded on the assumption that we can generate enough electricity from renewable sources to meet our needs. Although we have made great progress in renewable energy – in technical terms like improving PV-cell efficiency, as well as rapidly growing the installed generating capacity – electrification will not be the only answer to the challenges we face.

We must continue to be inventive and solve problems – those we inherited and those we will inevitably create ourselves as we seek to improve our lives and protect the planet. We can only keep trying to find our way forward, through 2025 and beyond.

We can anticipate an effective technological solution. It's what engineers do. On the other hand, we may need to

consider changing our always-on, energy-hungry lifestyles to ensure reliable, sustainable access to services we all need.

ALUN MORGAN is technology ambassador at Ventec International Group (venteclaminates.com); alun.morgan@ventec-europe.com. His column runs monthly.

Electronics Design Training

Developed by
Mike Creeden
Stephen Chavez
Gary Ferrari
Rick Hartley
Susy Webb

The **printed circuit design** industry's choice for comprehensive **training and certification.**

This 40-hour **live curriculum** was **developed by leading experts** and includes a 400-page handbook containing information on everything from parts libraries to high-speed design.

PCEA TRAINING

pceatraining.net

Register Today!

Check the website for additional classes to be added throughout the year.

UPCOMING CLASSES:

Jan. 20, 27, Feb. 3, 10 & 24
Feb. 28, Mar. 7, 14, 28 & Apr. 4
May 9, 16, 23, 30 & Jun. 6

ROUGHLY **SPEAKING**

3-D Vision

Three EMS companies show why additive manufacturing isn't just the provenance of the fab shop.

THE PRINTED CIRCUIT board industry isn't historically known for sudden innovation and change, but 3-D printing technology has been successfully adopted by many along the PCB supply chain. While there are those in the industry pushing for adoption of full 3-D printed circuit boards, capability and demand remain extremely niche. On the other hand, demand is quietly growing on the PCB assembly side for 3-D printing of fixtures for solder application, assembly and testing.

Three PCB assembly operations I recently visited are using 3-D printers to solve manufacturing and testing issues. The ability to quickly build a test fixture, assembly fixture or even custom shielding components for conformal coating means these facilities can deliver assembled boards with custom solutions traditionally reserved for high-volume production. This results in faster time to market because the assemblers are no longer waiting for a third party to fabricate a molded plastic or piece of metal for their application. In many cases, the metal material and fabrication costs aren't economical, but a resin 3-D printed solution is, bringing up potential solutions that were previously easily dismissed.

I was graciously given a tour of Fusion EMS and appreciated seeing engineers on the manufacturing floor. What caught my eye in the corner where functional tests were performed was a table set up with two 3-D printers. The engineers shared that their facility could assemble and test a large order with a very tight deadline for delivery because they had been tinkering with a 3-D printer. On behalf of the customer, they were performing quality testing, which meant connecting power and testing the functionality of the assembled board. A near-impossible task to do individually and manually, but through experimentation with the 3-D printer on the manufacturing floor coupled with their engineering resources, they were able to quickly spin up custom test fixtures. As the success of that shipment was due to the quick thinking of the engineering manager and use of the 3-D printer, they proudly showed me an even nicer second 3-D printer they specified because management saw its worth.

Similarly, PCB Assembly Express also has a 3-D printer, laser etcher and aluminum CNC machine. With almost instant access to its in-house engineered custom fixturing and stencils, it can quickly adapt to modern challenges on the manufacturing floor. PCB Assembly Express leverages one of its sister companies, Pentalogix, to provide a development team capable of making custom software solutions to improve data processing, communication with equipment and DfA/DfM checking – another example of utilizing resources traditionally beyond the scope of a standard PCB assembler.

In November, Axiom Electronics hosted our local in-person PCEA chapter meeting, and I was able to speak with Rob Rowland about Axiom's use of 3-D printing. Axiom uses 3-D printing at several different stages of the assembly process, with solutions being created and used by its various types of engineers to solve problems in tooling, process and test. It was pointed out that even things as simple as customized trays for parts and components add to the company's efficiency and quality.

While it wasn't the case with all these assemblers, several members of their respective teams started with 3-D printing as hobbyists, then leveraged that self-gained knowledge into their professional work. Those pushing the industry forward, it seems, aren't all coming from traditional places. Mechanical engineers interested in 3-D printing and metal fabrication probably wouldn't think of a PCB assembly as an environment to do such things, but here we are.

In these cases, it was more than just the existence of the 3-D printers; it was also a company culture of supporting engineers with engineering tools on the manufacturing floor. This enabled these engineers to proactively address situations with a creative and cross-disciplined problem-solving mindset. Modern PCB assembly operations are pulling in engineering resources beyond process engineering to create and adopt engineered solutions through 3-D printing and CNC machining to be faster, more accurate and of higher quality. What other interdisciplinary technologies and capabilities can we incorporate into the PCB industry to be better tomorrow? Feel free to share your thoughts with me.

GEOFFREY HAZELETT is a contributing editor to PCD&F/CIRCUITS ASSEMBLY. He is a technical sales specialist with more than 10 years' experience in software quality engineering and sales of signal integrity software. He has a bachelor's degree in electrical engineering; geoffrey@pcea.net.

PCB LINE SPACING

Clearance and Creepage in PCB Design

An overview of standards and guidelines for high-voltage applications.

by ANDREW GONZALES

PCB line spacing is relevant to both safety and function when it comes to traces. Comprised of copper, fuses and insulation, a trace is a conductive connection between different components on a PCB. Numerous factors determine the effectiveness of tracing, including PCB trace widths.

By connecting various signals, the trace is an important connection with its own benefits and avoidable drawbacks.

Traces permit signals to travel between different components. They are necessary but prone to issues, as they are susceptible to potential errors that could lead to a malfunction.

But for products or circuit boards using high voltages, safety standards are extremely relevant. As line spacing between different conductive components increases, so do spacing standards. If the breakdown voltage is exceeded when safety standards aren't adhered to, arcing could occur along the board, presenting issues between the traces.

Clearance and Creepage

Clearance and creepage are now more than important safety practices; they're internationally required safety standards. Ultimately, both creepage and clearance are used to prevent arcing. Adhering to those standards is essential for safety and functionality in high-voltage circuitry; otherwise, the PCB and its product may face experience issues.

Creepage considerations are important in all applications and voltages. PCB designers often overlook the need for creepage at lower DC-based voltages, which can sometimes get them in trouble. It's especially important around connectors and exposed pads placed too close together.

Clearance: A bird's eye distance between conductive parts. The shortest distance in the air between conductive parts is known as clearance **(Figure 1)**. Clearance can be evaluated by measuring the shortest path that doesn't pass through insulation, parts or sleeving. Environmental effects are extremely relevant to clearance; humidity can increase the chance of arcing, and moisture and humidity could lead to arc flashes. Dust or other small particles can also cause issues between conductive components.

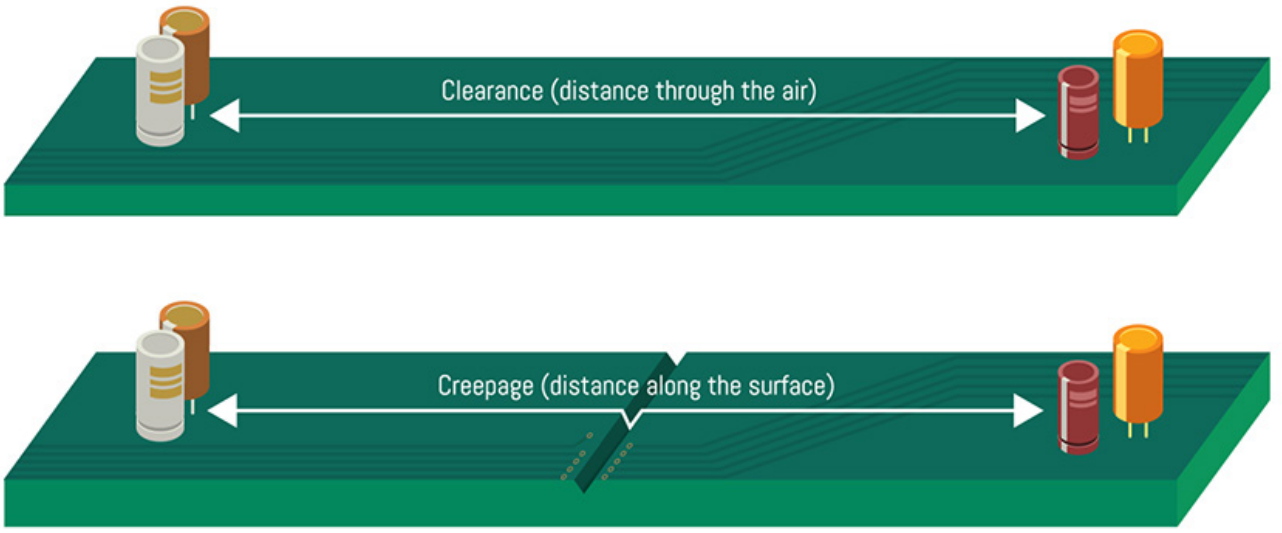


Figure 1. A comparison of clearance and creepage.

Creepage: The surface distance between conductive parts. Creepage is the shortest distance between conductive parts, except they must be along an insulated surface. They also have similar requirements and concerns, with the difference being the insulation of the creepage instead of the air that clearance passes through.

The only difference in these is regarding the surface along which current travels. Creepage goes through an insulated surface, and clearance passes through air. When factoring in the individual specifications, clearance and creepage distance are typically very similar. The only instance where there would be a difference concerns high-voltage circuits that may require different clearances and components.

Several safety regulations pertain to creepage and clearance. They provide direction on operating equipment under ideal conditions and temperatures, and detail how to manufacture industry-acceptable equipment and parts in a safe environment. They include:

- **IPC-2221, *Generic Standard on Printed Board Design.*** IPC-2221 details numerous requirements designed to prevent overheating and thermal management issues **(Table 1)**. They ensure a proper trace width is calculated to keep the relevant temperature below its threshold. This is a widely accepted standard that not only lays out best practices, but also establishes material specifications.

Table 1. Electrical Conductor Spacing

Voltage Between Conductors (DC or AC Peaks)	Minimum Spacing						
	Bare Board				Assembly		
	B1	B2	B3	B4	A5	A6	A7
0-15	0.05mm [0.00197in]	0.1mm [0.0039in]	0.1mm [0.0039in]	0.05mm [0.00197in]	0.13mm [0.00512in]	0.13mm [0.00512in]	0.13mm [0.00512in]
16-30	0.05mm [0.00197in]	0.1mm [0.0039in]	0.1mm [0.0039in]	0.05mm [0.00197in]	0.13mm [0.00512in]	0.25mm [0.00984in]	0.13mm [0.00512in]
31-50	0.1mm [0.0039in]	0.6mm [0.024in]	0.6mm [0.024in]	0.13mm [0.00512in]	0.13mm [0.00512in]	0.4mm [0.016in]	0.13mm [0.00512in]
51-100	0.1mm [0.0039in]	0.6mm [0.024in]	1.5mm [0.0591in]	0.13mm [0.00512in]	0.13mm [0.00512in]	0.5mm [0.020in]	0.13mm [0.00512in]
101-150	0.2mm [0.0079in]	0.6mm [0.024in]	3.2mm [0.126in]	0.4mm [0.016in]	0.4mm [0.016in]	0.8mm [0.031in]	0.4mm [0.016in]
151-170	0.2mm [0.0079in]	1.25mm [0.0492in]	3.2mm [0.126in]	0.4mm [0.016in]	0.4mm [0.016in]	0.8mm [0.031in]	0.4mm [0.016in]
171-250	0.2mm [0.0079in]	1.25mm [0.0492in]	6.4mm [0.252in]	0.4mm [0.016in]	0.4mm [0.016in]	0.8mm [0.031in]	0.4mm [0.016in]
251-300	0.2mm [0.0079in]	1.25mm [0.0492in]	12.5mm [0.4921in]	0.4mm [0.016in]	0.4mm [0.016in]	0.8mm [0.031in]	0.8mm [0.031in]
301-500	0.25mm [0.00984in]	2.5mm [0.0984in]	12.5mm [0.4921in]	0.8mm [0.031in]	0.8mm [0.031in]	1.5mm [0.0591in]	0.8mm [0.031in]
> 500 See para. 6.3 for calc.	0.0025mm /volt	0.005mm /volt	0.025mm /volt	0.00305mm /volt	0.00305mm /volt	0.00305mm /volt	0.00305mm /volt

B1 - Internal Conductors, B2 - External Conductors, uncoated, sea level to 3050 m [10,007 feet], B3 - External Conductors, uncoated, over 3050 m [10,007 feet], B4 - External Conductors, with permanent polymer coating (any elevation), A5 - External Conductors, with conformal coating over assembly (any elevation), A6 - External Component lead/termination, uncoated, sea level to 3050 m [10,007 feet], A7 - External Component lead termination, with conformal coating (any elevation), Source: IPC-2221B conductor spacing requirements

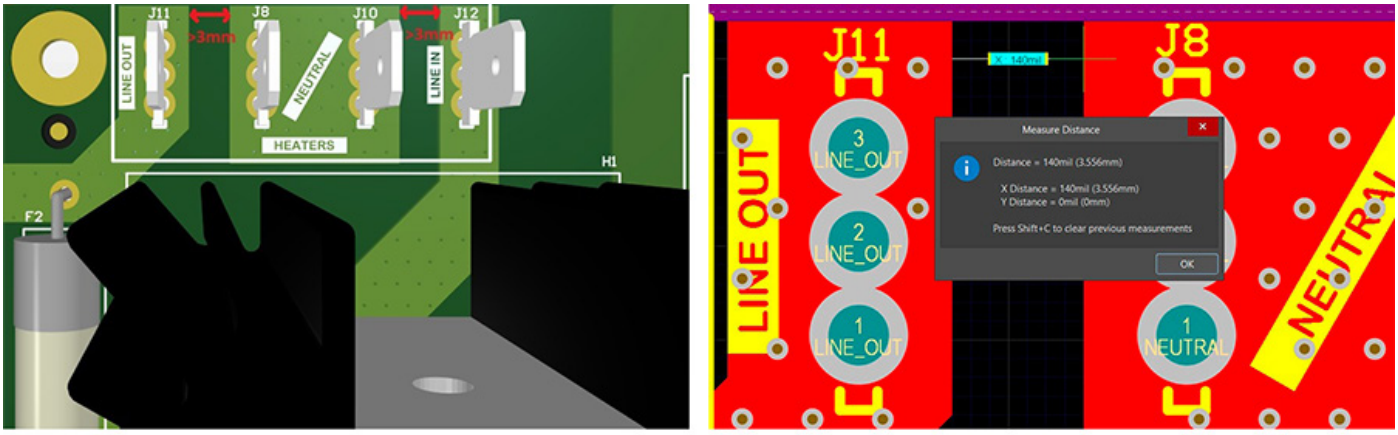
- **IPC-2152, *Standard for Determining Current Carrying Capacity in Printed Board Design.*** IPC-2152 provides guidance on determining the appropriate conductor sizes on the finished printed circuit board as a function of the current carrying capacity required and the acceptable conductor temperature rise.
- **IPC-9592, *Requirements for Power Conversion Devices for the Computer and Telecommunications Industries.*** This dictates the safety standards specifically for power conversion devices. By establishing a standard power for computers and other communication devices, this creates a safety net by which engineers can analyze spacing requirements based on voltage and whether it's an external or internal layer.
- **UL 61010-1, *Standard for Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use; Part 1: General Requirements.*** This safety standard regards the appropriate distances for clearance and creepage for laboratory and industrial equipment. Poorly measured distances pose a massive risk for equipment used for measuring and controlling processes like pH controllers or temperature. Suggested measurements are laid out in BS EN 61010-1:2010 ("Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – General Requirements"), including those found in **Table 2**.

Table 2. Clearances and Creepage Distances for Mains Circuits of Overvoltage Category II up to 300V

Voltage line-to-neutral a.c. r.m.s. or d.c.	Values for Clearance	Values for Creepage Distance								
		Printed wiring board material		Other Insulating Material						
		Pollution Degree 1	Pollution Degree 2	Pollution Degree 1	Pollution Degree 2			Pollution Degree 3		
		All material groups	Material group I, II, IIIa	All material groups	Material group I	Material group II	Material group III	Material group I	Material group II	Material group III
V	mm	mm	mm	mm	mm	mm	mm	mm	mm	mm
≤150	0.5	0.5	0.5	0.5	0.8	1.1	1.6	2	2.2	2.5
>150≤300	1.5	1.5	1.5	1.5	1.5	2.1	3	3.8	4.1	4.7

Notes: a) The values in Table 2 are for basic insulation and supplementary insulation. Values for reinforced insulation shall be twice the values for basic insulation. b) Minimum clearance for basic insulation, supplementary insulation and reinforced insulation for pollution degree 3 is 0.8mm. c) If the equipment is rated to operate at an altitude greater than 2,000m, the clearances need to be multiplied by the factors in Table 3 of the standards document. Source: UL 61010-1

Maintaining these standards isn't just important; it leads to increased functionality and longer use for the components and the PCB itself. However, this doesn't address or solve every potential issue with creepage or clearance.



A B

Figure 2. A design with 240VAC input power and >3mm creepage spacing.

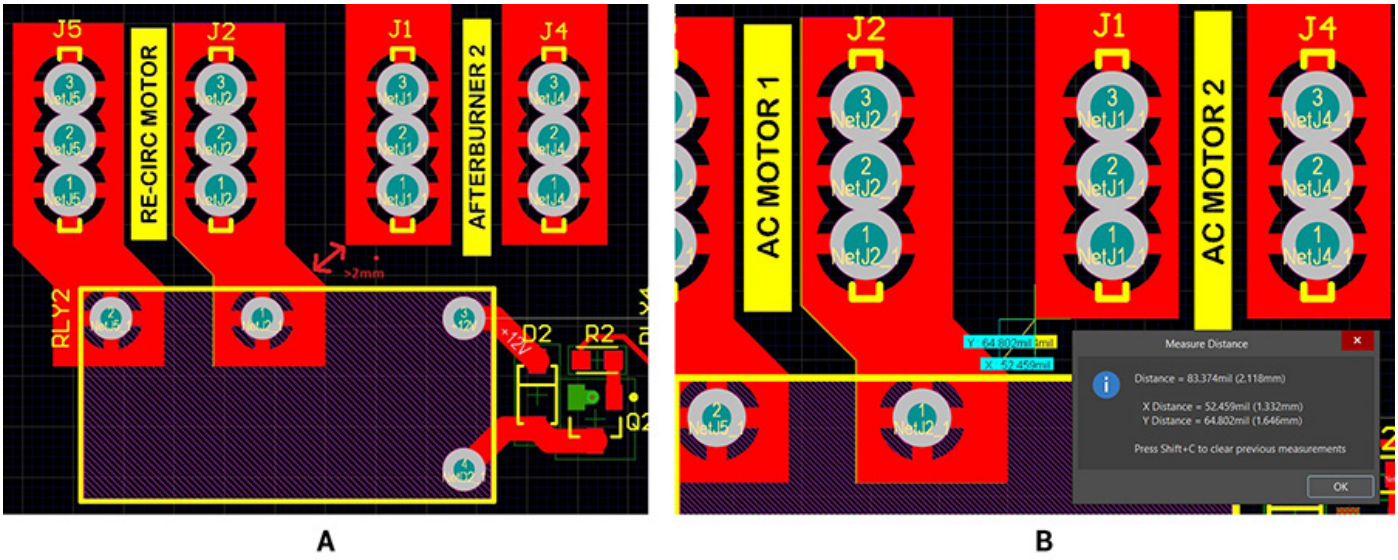


Figure 3. A design with multiple 120VAC motors separated by >2mm creepage spacing.

Calculating the necessary clearance and creepage is just the first step in a complicated process. Temperature, air pollution, moisture, humidity and voltage are all variables that affect the PCB. Guidelines to help avoid concerns and issues when determining creepage and clearance specifics include:

1. Constantly monitor spacing between any traces and other components.
2. Maintain the "3W principle," where the spacing between lines must be no less than three times the width of the line itself.
3. Determine where high voltage will occur and use larger spaces and distances to prevent issues like arcing.
4. Use insulation when necessary, especially when there is a connection between high- and low-voltage components.
5. Use curves, not angles, to utilize the shortest possible trace lengths.

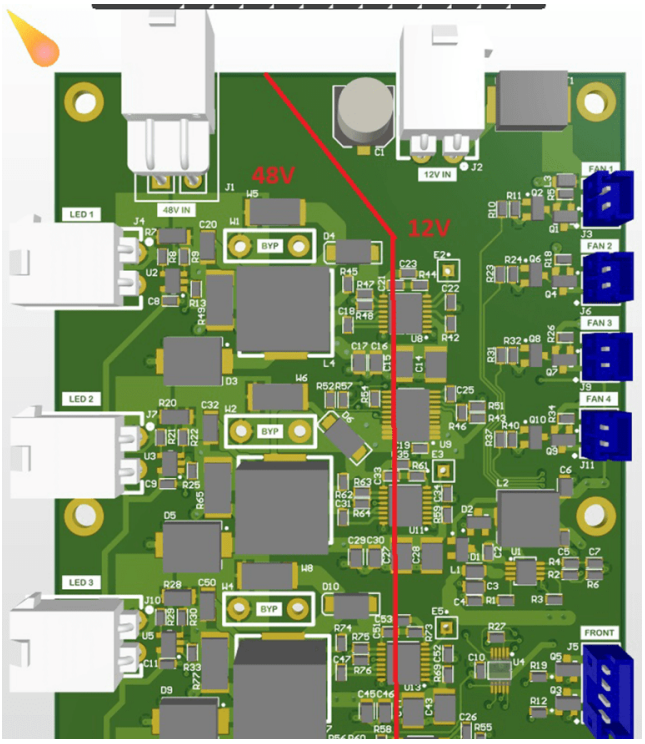


Figure 4. A top-down view of a design containing 48VDC. Creepage should apply to every design, even lower voltages such as 48VDC.

Optimal Materials and Circuits

More demands will be required of the materials and components for high-voltage PCBs and connections. They'll be more susceptible to voltage fluctuations, exposure to different environments and humidities, and potential overvoltage as the product degrades That's why it's incredibly important to use circuits specifically recommended and designed for clearance and creepage **(Figure 5)**.

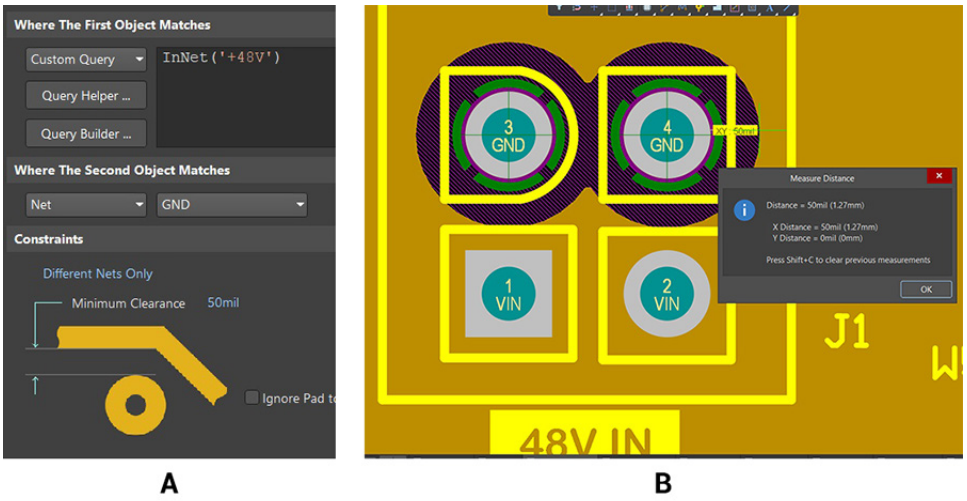


Figure 5. Design rules are set up (left) with the result (right) shown on traces, planes and polygon pours; found on same 48V input power connector (50 mils or >1mm between 48V and ground).

High voltage. High-voltage circuits operating at a minimum of 50V will need longer distances for both clearance and creepage to prevent electrical issues and breakdowns.

Medical devices. Medical devices often have more fail-safes than other industrial equipment. Because of the devices' importance and significance, both kinds of distances need to be strictly adhered to and specified. Components, use and interaction with patients are all factors for clearance and creepage in medical equipment.

Industrial equipment. OSHA and other governing bodies have put strict specifications on industrial equipment and devices, especially those that account for particulates like dust, heat and vibration/movement.

Power electronics. The higher the voltage and power of a device, the higher the risk of problems. When dealing with high currents and voltage, longer clearance and creepage distances are typically required to avoid issues with arcing, insulation and voltage faults.

Every component needs to be carefully selected for materials. High operating voltages will be a factor, so not only does there need to be accurate spacing, but 2oz minimum traces or vias of copper and resin and glass levels can provide the durability and reliability required for those high-voltage projects and products.

PCB Practices and Standards

Standards and practices are becoming unique to the job or project they're used for, and that's expected to continue as technology evolves. Maintain these practices and utilize these materials, and you're one step closer to stable, high-voltage PCBs.

ANDREW GONZALES is vice president of engineering at San Francisco Circuits (sfcircuits.com); sales@sfcircuits.com.

ENGINEERING *TOMORROW'S* ELECTRONICS

PCB WEST 2025

Conference & Exhibition

SAVE THE DATE

Conference: Sept. 30 - Oct. 3

Exhibition: October 1

Santa Clara Convention Center

2025

DFM/**DFA**

Maximizing Yields with Minimal Iterations

DfM practices for preventing common fab and assembly defects.

by AKBER ROY

Printed circuit board (PCB) design is an area of engineering that blends art and science, and a significant portion of the science side falls into manufacturing. Formally, designers know this as design for manufacturing, or DfM, where alignment is achieved between the circuit board's features and the capabilities of the manufacturing process.

While leveraging the manufacturer's knowledge to identify design problems that could lead to defects is commonplace, this can drive unnecessary design iterations. Before transitioning a product to manufacturing, certain design practices can help prevent common fabrication and assembly defects.

This is not just about PCB prototyping. The designer's goal should be to implement practices that help scale a product to volume production at high assembly yield. Some of these design practices start as early as library creation, while others can be applied after a preproduction review by CAM. Let's dig in and see what can be done to maximize assembly yield.

DfM Practices that Prevent Fab Defects

PCB manufacturers publish their processing capabilities primarily to ensure that designers adhere to these important constraints. CAD tools can help designers avoid most of the simplest fabrication defects when implemented as design rules. On the fabrication side, line spacing, etch clearances, drill wander, warpage and delamination affect bare board yield and long-term reliability.

Issues like warpage and delamination happen during buildup and are under the control of the fabricator. Other issues under the control of the PCB designer include:

- **Plating voids.** Failure to allow for adequate copper plating in through-holes could leave behind plating voids, particularly on high-aspect-ratio vias **(Figure 1)**.

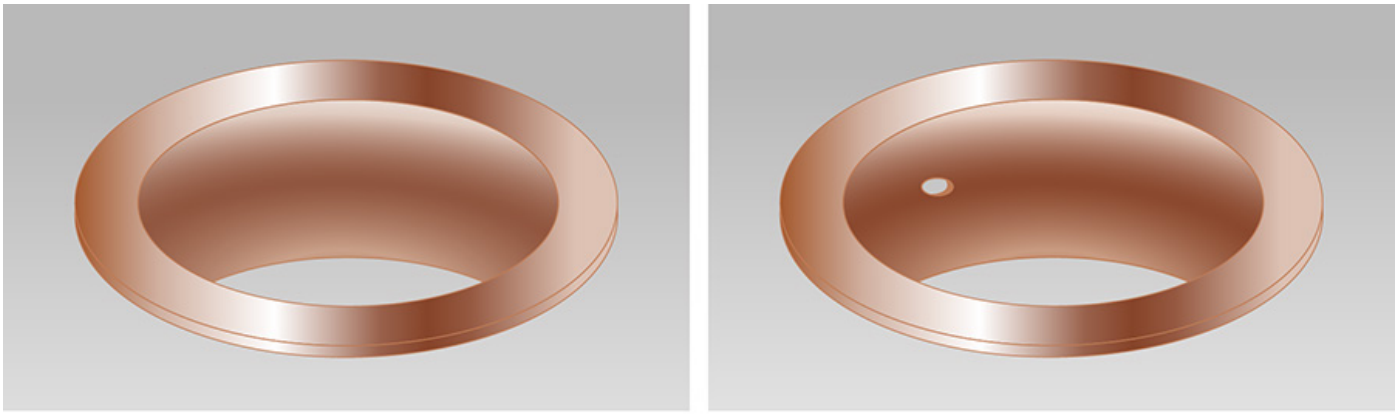


Figure 1. Acceptable plating voids in Class 2 and 3 electronics. 1a shows no plating voids, while 1b. has one plating void. (Source: IPC-A-600)

- **Insufficient copper-to-edge clearance.** Insufficient clearance potentially leaves copper exposed through the edge of the PCB after depaneling, which may cause electrical shorts or even scrap the PCB **(Figure 2)**.

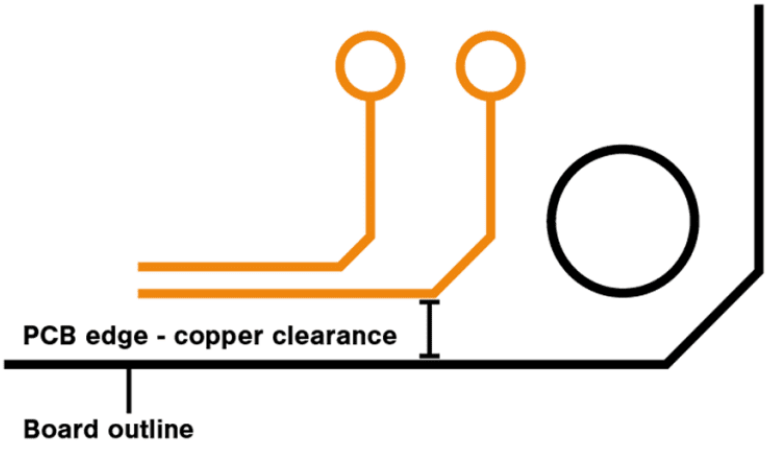


Figure 2. Proper copper-to-edge clearance prevents electrical shorts. (Source: IPC-A-600)

- **Solder mask slivers.** When placement is very dense, a solder mask web with a width less than 4 mils (3 mils being the minimum) can lift off the PCB surface later in the assembly process, creating the risk of solder bridging **(Figure 3)**.

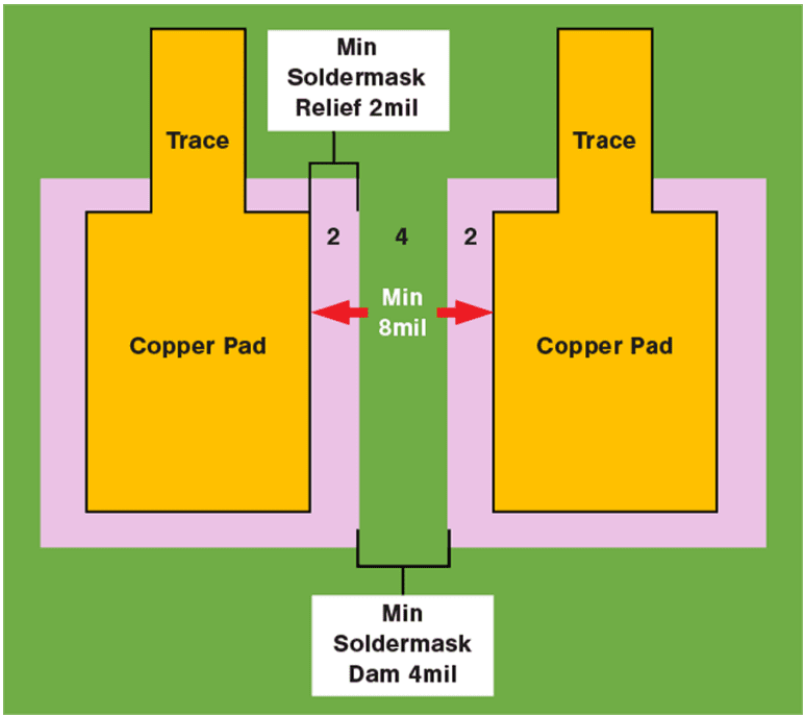


Figure 3. Solder mask coating between SMD pads prevents solder paste from flowing across parts. (Source: IPC-A-610)

- **Shorts/opens.** Etching or drilling problems near closely spaced copper features can leave behind open or short circuits **(Figure 4)**.

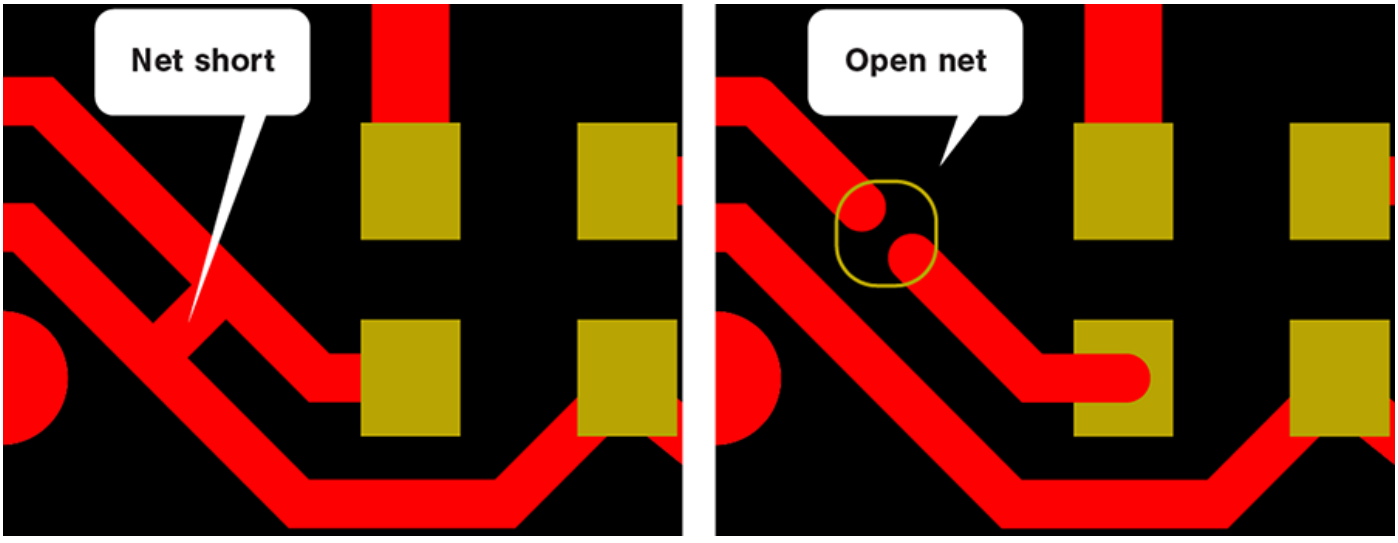


Figure 4. Shorts (a) or open (b) circuits can occur near closely spaced copper features.

- **Annular ring breakout.** Small pads on plated through-holes may not be able to account for drill wander, the result being a drill hit that severs a pad from a trace **(Figure 5)**.

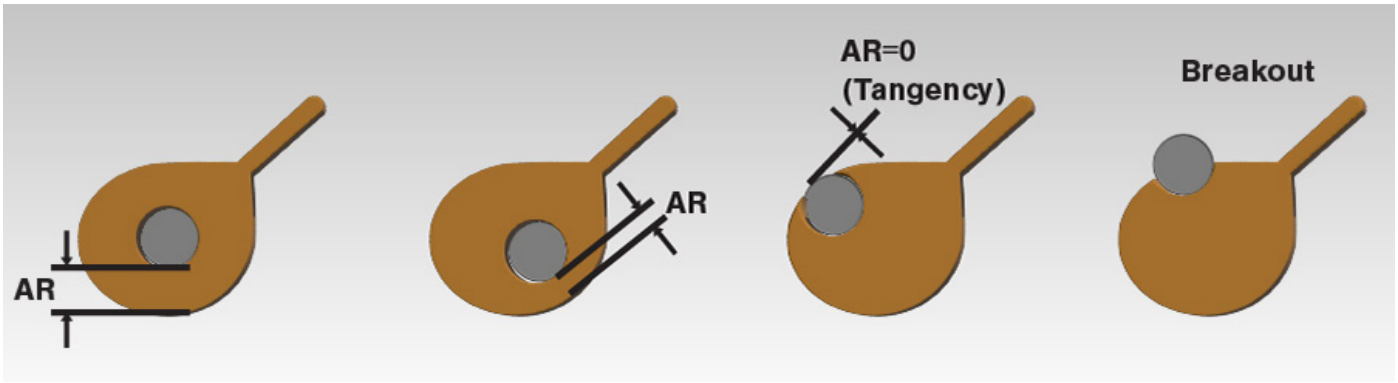


Figure 5. Drill wander can sever a pad from a trace.

Within this list, designers should apply the required clearance and feature size rules to ensure accurate board fabrication without creating defects. Fabricators can assist with this, and the information is most often contained in a capability statement on the fabricator's website.

In PCB manufacturing of products targeting Class 2 or Class 3 compliance, the most important consideration is drill wander resulting in annular ring breakout. The annular ring on a plated through-hole is defined as the copper pad that encircles a drilled and plated hole, and it must maintain some minimum value after drilling. For Class 3 products, no breakout is allowed, and some minimum annular ring must be maintained. This may require the use of "teardrops" and oversizing the pad on the plated through-hole. Both factors are simple design choices that produce large payoffs in terms of reliability.

DfA Addresses Assembly Defects

Fabrication is, of course, just the first part of the manufacturing process. A poorly assembled board could require rework or scrapping. To ensure minimal defects, fabricators often rely on third-party assemblers to manage the component placement and processes. Manufacturers can only do so much, however, which puts a heavy responsibility on the PCB designer to ensure defect-free assembly. While the designer can't control solder processing, QA/QC or the reliability of rework procedures, some basic design practices can help reduce defect rates.

To start, some common defects that arise during assembly are:

- Shifting, lifting or tombstoning of SMD components **(Figure 6)**. An imbalance of heat dissipation across component leads is the prime cause. As a remedy, add a thermal relief connection across discrete SMD pads to slow the rate of heat dissipation.

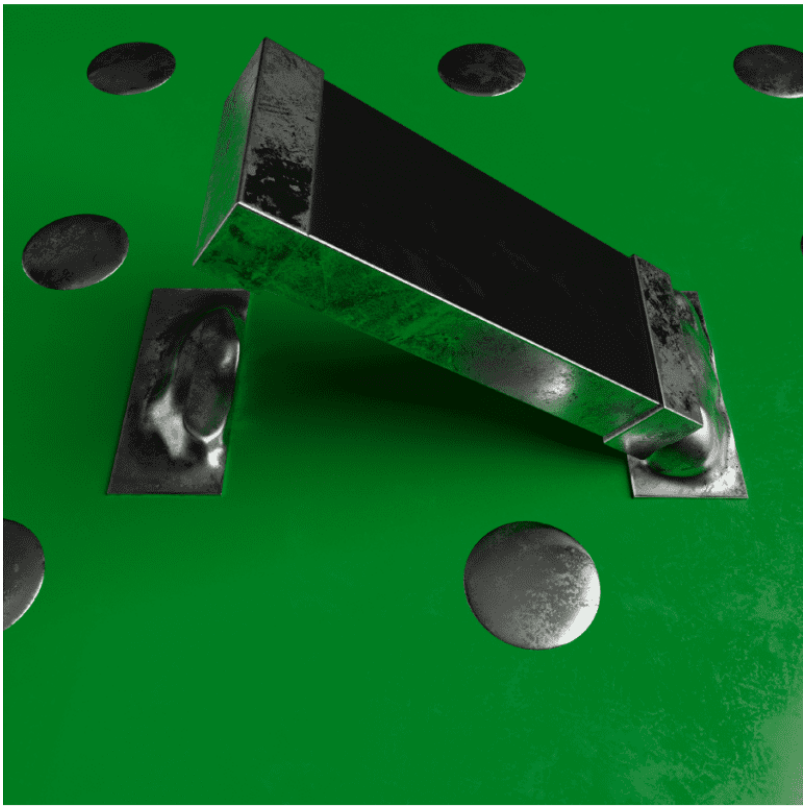


Figure 6. An example of tombstoning. (Source: IPC-A-610)

- Cold joints, particularly on through-hole leads
- Leadless component defects, particularly in BGA packages
- Reduced reliability from leftover flux residues
- Short circuits (solder bridging, solder balling, etc.) **(Figure 7)**.

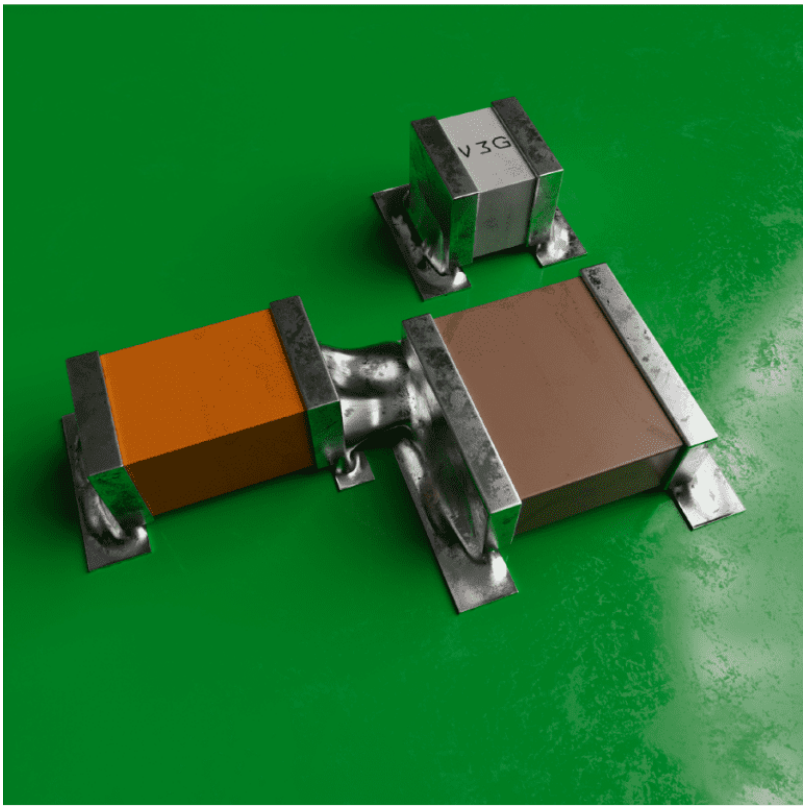


Figure 7. Solder bridging can occur with very dense placement. (Source: IPC-A-610)

A designer can influence the quality of custom PCB assembly in these major areas. Of course, the assembler must run its process correctly and with minimal variability, but the design should also implement practices that help ensure a good process will produce a high yield.

Table 1 outlines some areas where an assembler must implement quality control and where the designer can implement best practices to increase yield. In this section, we'll focus on the design side.

Table 1. Roles in Quality Control

Designer's Role	Assembler's Role
Create footprints to IPC density standards	Control solder paste dispense rate and clean stencils
Determine mask expansions to control exposed pad symmetry	Ensure uniformity in reflow temperature
Apply negative paste mask expansion where required (e.g., on QFN pads)	Clean assemblies to remove flux residues and other debris
Apply thermal reliefs where necessary	Inspect to identify rework needs

Thermal Reliefs

Thermal reliefs on SMD pads and through-hole pins are among the most common DfA guidelines in PCB assembly. This is most common in tombstoning, specifically for resistors and capacitors, due to their multi-sided termination and low weight. The idea that thermal reliefs are an absolute requirement on all SMD pads, however, in practice means they are treated as a universal design rule in some CAD tools, meaning they are applied in a PCB by default.

The user then must selectively disable the rule to remove the thermals.

Figure 8 shows two typical thermal reliefs. In this case, the thermal reliefs are applied on the copper pour side of a passive surface mount component with a spoked connection between a pad and the copper pour. The thermal relief applied on these pads is intended to reduce the copper available to conduct heat during soldering.

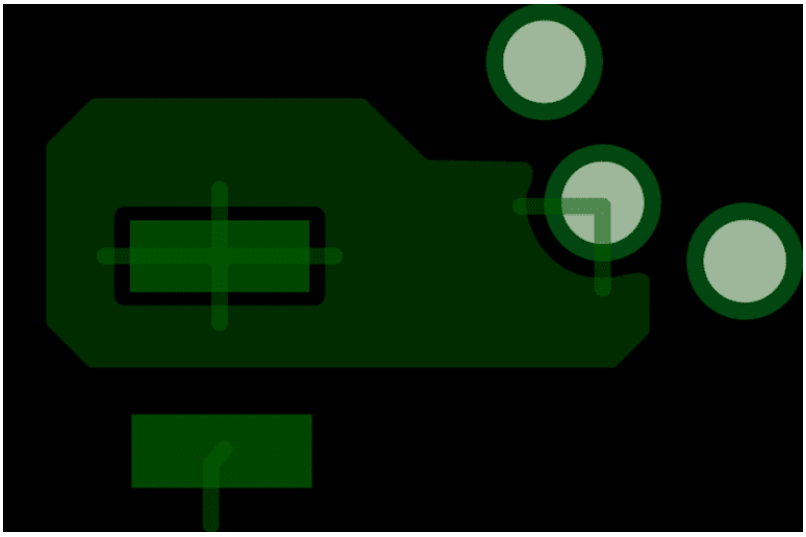


Figure 8. Thermal reliefs on an SMD component and through-hole component shown in OrCAD.

In addition to thermal reliefs on SMD pads, there are through-hole pin thermal reliefs, specifically on pins that connect to very large copper pours or planes. These thermal reliefs on through-hole pins perform the same function and are intended to address the potential for a cold joint.

Should these be applied everywhere, and are they effective? It is best to apply thermal reliefs on specific SMD pad connections that are at risk for tombstoning or on through-hole pin connections at risk of cold joints:

- SMD pad thermal release should be applied judiciously on large copper pours; small copper pours can omit thermal relief
- If the board is hand soldered, SMD pads are more likely to need thermal relief
- In both cases, the same idea applies to through-hole pins connecting to internal planes or large copper pours.

First, the challenge is determining what constitutes a “large” copper pour. No set of rules or standards dictates exactly how large a copper region needs to be before a thermal relief should be applied. If the board design is submitted to the contract assembler prior to fabrication, ask it to examine planes/pours for potential SMD defects. Assemblers can often advise when and where to place thermals on these large regions of copper.

Component Shift

Tombstoning is a form of component shifting, but shifting can also happen on parts that are not at risk of tombstones.

These factors are typically related to a paste dispense region and pad size definitions, some of which are under the designer's control:

- Pads sizes and shapes on SMD parts
- Solder mask openings on SMD pads
- Paste mask openings on SMD pads
- Consistent dispensing of solder paste.

Only the first three of these points can be controlled by the designer. The PCB designer will often define these features in their parts library or they will implement design rules to apply these features (or both).

With these pad mask features, the fabricator wants to ensure the pad position on each side of a component is consistent and over deposition has not occurred. The component package needs symmetry to prevent components from floating once the solder paste melts. As shown in **Figures 9-10**, IPC has set acceptance criteria for part skew/ overhang.

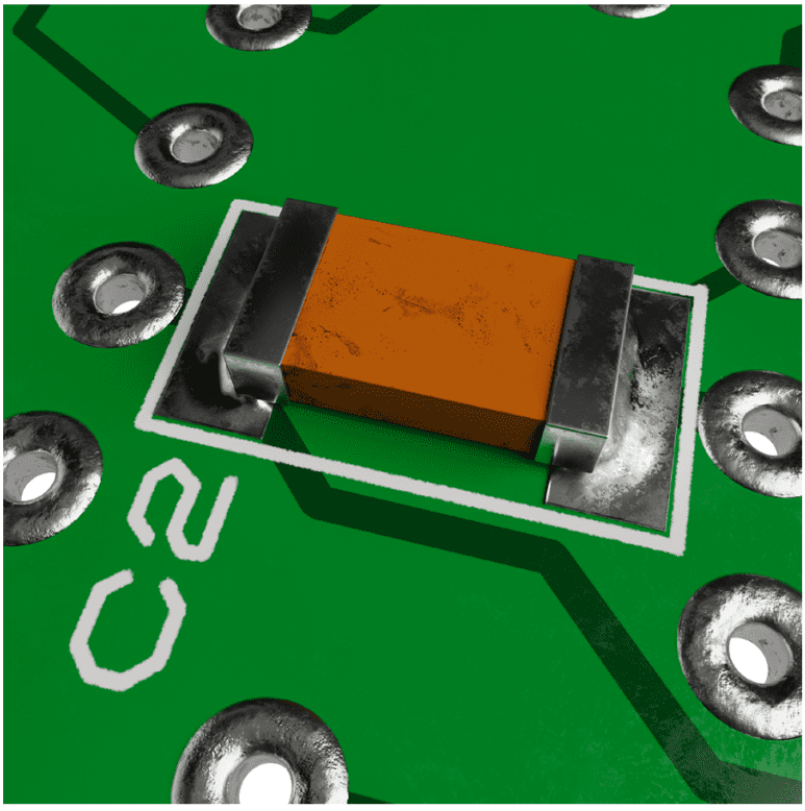


Figure 9. Ideal solder joint. (Source: IPC-A-610)

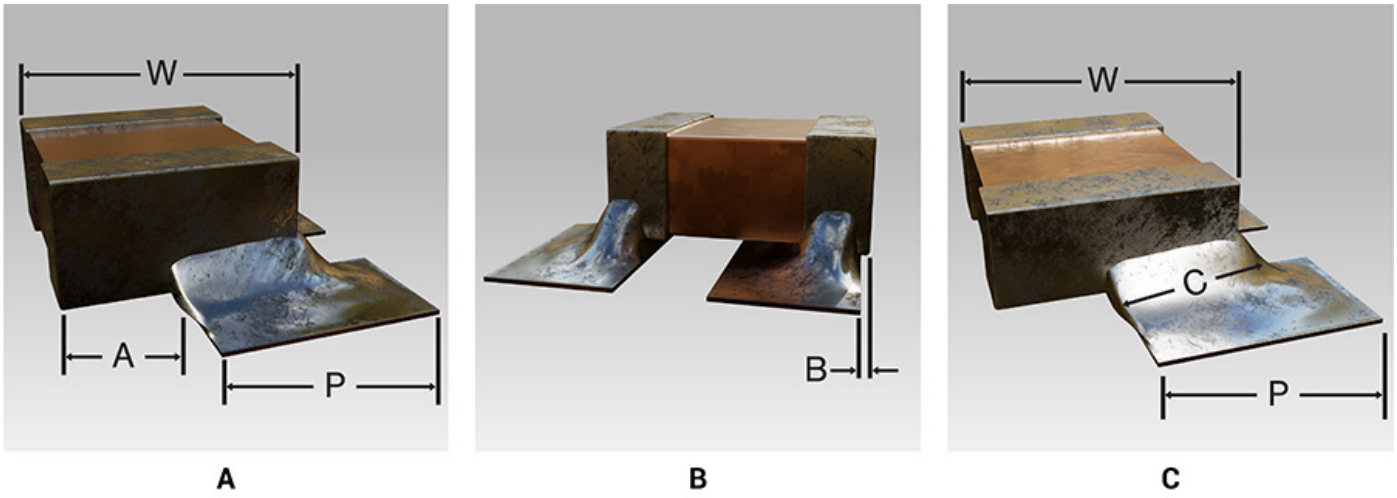


Figure 10. Acceptable and unacceptable examples of part skew. (Source: IPC-A-610)

Clean the PCBAs

According to industry experts and surveys, approximately 80% of commercially manufactured circuit boards use solder with no-clean flux. No-clean flux formulations are varied, and are intended to eliminate the need to remove flux residues from the soldered PCB assembly. Residues from these flux formulations are supposed to be fully insulating, inert thin films, and in regular operation are not expected to create a risk of electrochemical reaction that could lead to failure **(Figure 11)**.

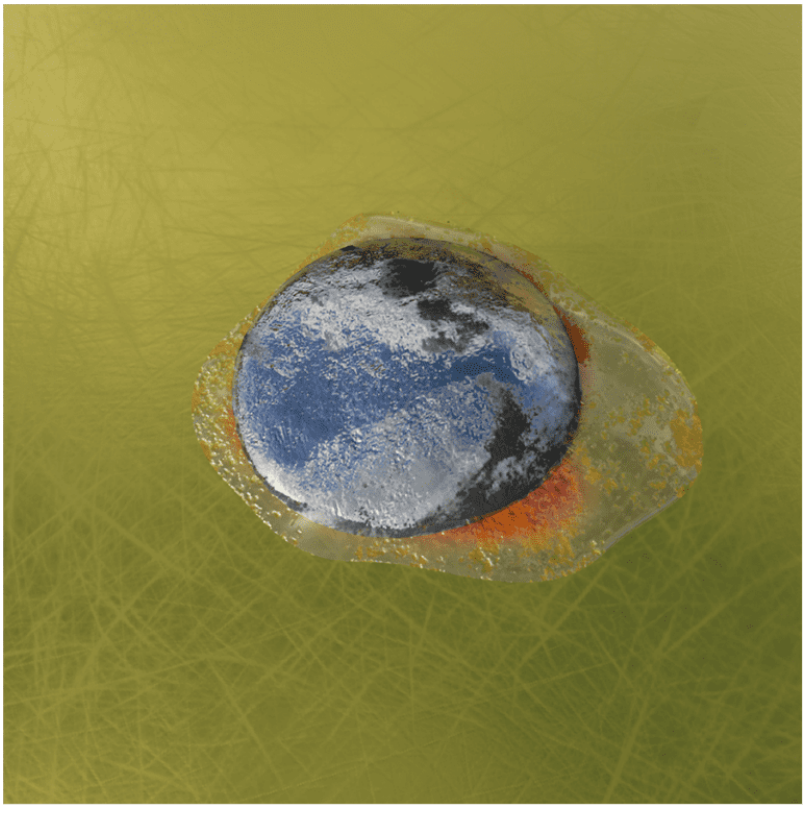


Figure 11. Unacceptable flux residue.

In use, however, some products will be exposed to humid air and temperature cycling, each of which can cause the reactivation of no-clean flux residues. The result is a gradual increase in surface insulation resistance (SIR) on the PCB, possibly driving an electrochemical reaction between conductors.

The PCB designer is not responsible for implementing a cleaning process during assembly, but they can demand it be used as part of the contract assembler's process. If there is a potential for exposure to extreme environments, and a no-clean solder paste is specified, ensure to include a requirement to remove flux residues from the PCBA. If the product is being marketed for its long-term reliability, cleaning is a simple step that helps ensure the target product lifetime can be realized.

Collaborate with Manufacturing

While designers can address many of the above points in some ways, a printed circuit board manufacturer must implement control over its processes to ensure high yield. With early collaboration, a designer can implement some of these design practices as constraints in the CAD platform, and a final DfA review can identify any outstanding risk factors before production.

AKBER ROY is chief executive of Rush PCB Inc., a printed circuit design, fabrication and assembly company (rushpcb.com); roy@rushpcb.com.

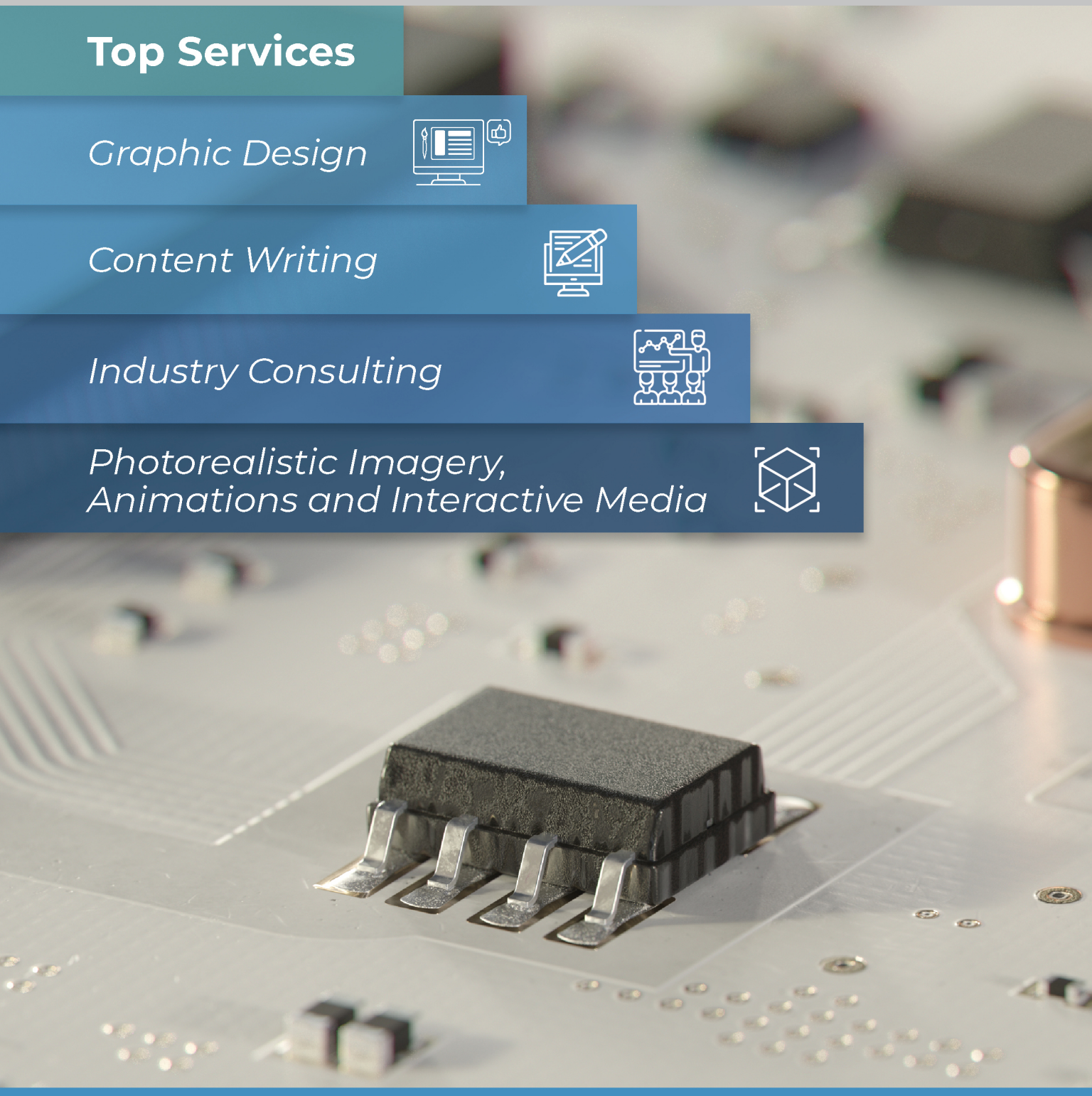
360CIRCUITS

Full-Service
Marketing Agency

::: WITH A PASSION FOR ELECTRONICS :::

Top Services

- *Graphic Design*
- *Content Writing*
- *Industry Consulting*
- *Photorealistic Imagery, Animations and Interactive Media*



www.360circuits.com

Comparing Coated vs. Uncoated Stencils

While cheaper to use, uncoated stencils can have a detrimental effect on printing performance.

by TIMOTHY O'NEILL

In a recent study focused on optimizing solder paste transfer efficiency, the initial phase used factory-applied coated stencils to isolate and understand the effects of solder alloy powder size on print performance. This approach ensured precise data by minimizing stencil-induced variations.

Our follow-up study explores the implications of using uncoated stencils, which are used in many manufacturing settings.

Factory-applied nanocoated stencils are designed to enhance the release of solder paste, while uncoated stencils are still widely used due to their cost-effectiveness. The choice between these stencils can play a significant role in determining the quality and efficiency of solder paste application.

Testing Protocol

Our in-house applications lab conducted extensive testing to compare the performances of coated and uncoated stencils. The stencil supplier provided stencils with and without nanocoating, using the same materials, machines and operators. Utilizing a controlled environment, the same engineers and consistent equipment, we aimed to minimize variables other than the stencil coating and paste type. The tests employed type 4 (T4) and type 5 (T5) solder pastes and examined their interactions with both stencil types across a variety of aperture area ratios.

Print quality criteria. Two key metrics were used to assess print quality: transfer efficiency (TE) and coefficient of variation (CV). A robust print process is defined by a TE of at least 80% and a CV of 10% or less. These benchmarks help ensure consistency and reliability in solder paste application. We focused on print features with area ratios (AR) of 0.50 to 0.75, so when using a 4-mil (100µm) foil, we looked at feature sizes from 8 to 12 mils (200-300µm). The area ratios and theoretical aperture volumes (in cubic mils) are shown in **Table 1**.

Table 1. Details for Each Pad Size Analyzed

Pad Size (mil)	Area Ratio	Ap Vol Circle (cu mil)	Ap Vol Square (cu mil)
8	0.50	201	256
9	0.56	254	324
10	0.63	314	400
11	0.69	380	484
12	0.75	452	576

Influence of pad definitions. In both the original and supplemental studies, pad definition was found to have a significant impact on print quality – solder mask-defined pads (SMD) demonstrated lower deposit variation with less overall volume, whereas non-solder mask-defined pads (NSMD) produced greater deposit variation but with greater volume. In fine feature printing found on 0201 or 01005 chips, LGAs, BTCs or other leadless packages, limiting paste deposit variation is a greater priority than applying higher paste volume. Stencil aperture designs can be modified to increase deposit volume, whereas inconsistent deposits are an uncontrolled liability.

Figure 1 illustrates the effect of SMD and NSMD pads.

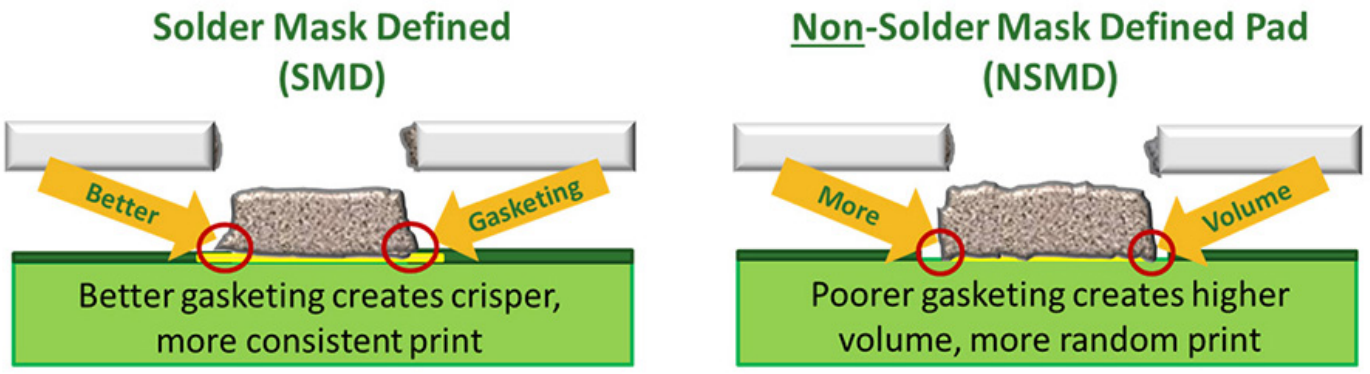


Figure 1. Effect of pad definition on solder paste print quality.

For this study, we used SMD pads for consistency and to observe results under the best-case scenario.

Results and Analysis

Our results revealed distinct differences between coated and uncoated stencils in their ability to meet the desired TE and CV benchmarks. Coated stencils generally achieved higher TE across a broader range of aperture ratios. Similarly, the influence of solder paste types (T4 vs. T5) was evaluated, indicating that coated stencils paired with T4 paste provided the best combination for achieving high TE and low CV, as illustrated in **Figure 2** and **Figure 3**.

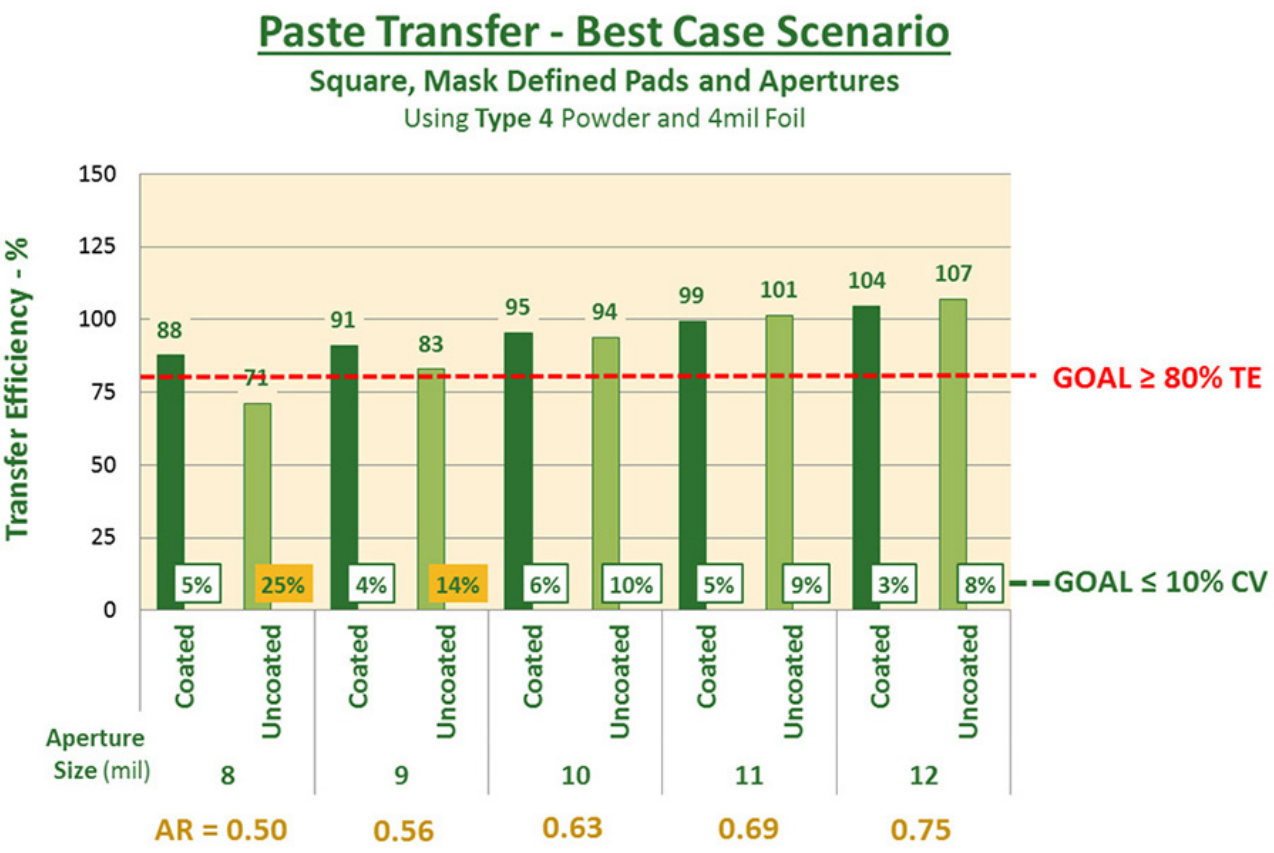


Figure 2. Transfer efficiency and CV of type 4 powder at various area ratios (AR).

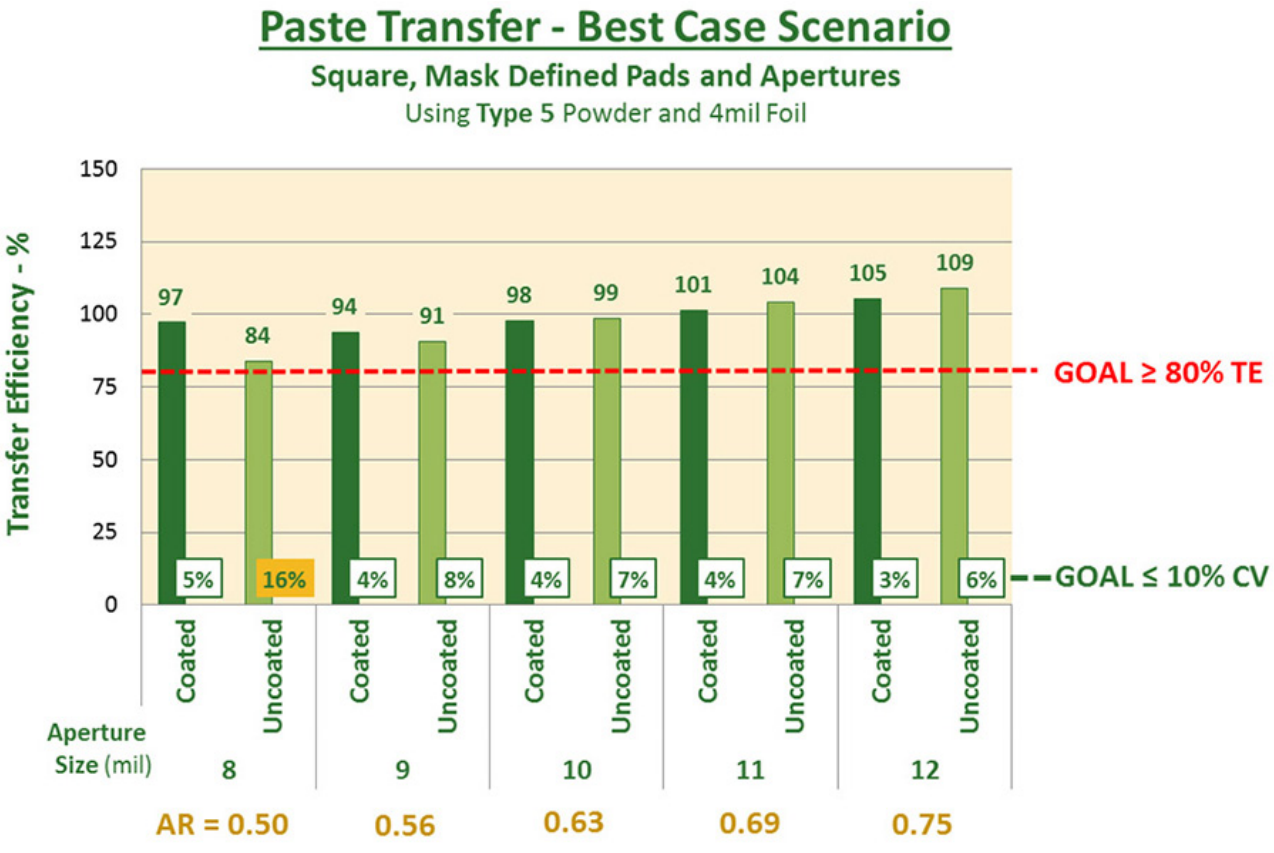


Figure 3. Transfer efficiency and CV of type 5 powder at various area ratios (AR).

Comparing CV results. When using T4 paste, we can achieve the goal of a CV of 10% or less on an AR of 0.50 using a coated stencil. We cannot, however, reach our goal with an uncoated stencil until the AR is 0.63. T5 solder paste was successful at an AR of 0.50 with a coated stencil, but not until an AR of 0.56 with an uncoated stencil. Hence, whether using T4 or T5 paste, a coated stencil yields better CV results.

Comparing TE results. With an uncoated stencil, Type 4 paste did not achieve either the 80% TE cutoff or the 10% CV cutoff on ARs of 0.50. It barely achieved the TE benchmark on the 0.56 AR but with an unacceptably high CV. The Type 5 paste met the ≥80% TE criteria with all ARs on both stencils, but failed to meet the CV benchmark at 0.50 without the help of the nanocoating. Again, whether using T4 or T5 paste, a coated stencil yielded better results.

When overlaying the results, T4 pastes printed using nanocoated stencils met the criteria for print quality at all ARs tested, and performed nearly identically with T5 pastes at ARs of 0.63 and up. Further, even when an uncoated stencil met the benchmark, it still demonstrated about twice the variation of its coated counterpart.

Another observation on the nanocoating influence on TE: it provides a boost in the low AR ranges, but not in the higher ones. This means we can consistently expect more paste on our fine feature prints while keeping our larger prints stable.

Conclusions and Recommendations

While coated stencils are more expensive, their cost must be weighed against their benefits to the printing process, particularly for high-volume or high-precision applications. By reducing print variations and improving TE, coated stencils can lower defect rates and reduce rework costs.

TIMOTHY O'NEILL is director of product management at AIM Solder (aimsolder.com); toneill@aimsolder.com.

Why Become a **PCEA Corporate Member?**

- ✓ Information, opportunities & marketing channels to **highlight your brands, products & services** to buyers & users worldwide
- ✓ Inclusion in the **PCEA Member Directory,** a product & service guide for buyers & users
- ✓ **Free advertising** of your products & services to more than 50,000 design engineers, fabricators & assemblers
- ✓ **Discounts** on trade shows, technical conferences, online learning platforms & webinars

PCEA®
PRINTED CIRCUIT ENGINEERING ASSOCIATION

Join today:

pcea.net/pcea-membership

Offering price points to meet every company's budget and needs!

‘It’s Got to Work’

STI Electronics prides itself on reliability and training the next generation of workers.

by TYLER HANES

With its home just a few miles from a major military and aerospace testing center, STI Electronics has built a reputation for providing electronics that continue to work through the toughest conditions.

Now in its 43rd year of operation, STI moved to Madison, AL, from California in 1993. The company settled into its current location – just down the road from Redstone Arsenal, a US Army base that houses NASA’s Marshall Space Flight Center and more than 75 federal agencies – in 2008.

Figure 1. STI Electronics moved into its current facility in Madison, AL, in 2008.

With its primary business coming from many of those agencies and the Department of Defense, reliability is where the company has claimed its niche in the industry, said president and CEO David Raby.

He said the company may not win a pricing battle, but the priority for products in those industries is that they need to

work reliably, and that's where STI shines the most.

"In the end, it's got to work," he said. "And that's what we put our reputation on. We build the highest reliability products."

STI has two SMT lines offering placement capability down to 01005 in size. It has two Juki FX-3 and KE2080-L pick-and-place machines, inline and secondary AOI to ensure accurate placement, a 12-zone oven and a Juki selective solder machine to automate through-hole soldering for higher-volume production. When PCD&F/CIRCUITS ASSEMBLY visited in December, the company was in the process of putting together a third SMT line to expand its production capabilities.

All processes in the facility adhere to AS9100 certification, with manufacturing and inspection following IPC J-STD-001 (Space Addendum). All completed assemblies are washed and visually inspected after each process, and BGA devices are 100% x-ray inspected. The company also offers flying probe testing to detect opens and shorts on the assembly and provide statistical analysis for device tolerances.

With the company already following the strict requirements of aerospace and defense, STI recently expanded into life-critical industrial products, earning ISO 14385 certification to build medical devices.

Raby said many of the requirements for that certification were similar to those for AS9100, with the differences primarily coming in record-keeping, so adding that certification made sense to expand the business and continue its growth.

"We're doing more medical than we've ever done before, and we're looking at that as part of our growth," Raby said. "It just fits right in with what we've been doing."

Some of the medical work also makes use of another capability that sets STI apart from its peers: a 2,400 sq. ft. clean room.

That's an investment that is still growing in use, but is beginning to bear fruit for STI as it branches into new industries, Raby said.

"We don't have anything that's built completely in there, but we do have some things that parts of them are built there before they move out onto the floor," he said.

The company also has an analytical lab to look at any problems that may crop up during production or predict those problems before they can rear their heads.

"It aids us tremendously in our manufacturing," Raby said.

Along with analyzing its in-house products, STI offers its analytics lab as an outside service for other companies that need to inspect their own products for faults, which provides another source of business and has permitted the lab to mostly pay for itself, Raby said.

‘The Best Trainers in the World’

STI currently has 60 employees and is on the lookout for more, but like most manufacturers, finding workers who are willing and able to do the job can be a challenge, he said.

“Our biggest holdup is just finding qualified people,” he said.

One major advantage the company has over others who are looking for workers is its status as an IPC-authorized training center, offering assembly and solder training courses from a highly trained and accomplished staff, Raby said.

Figure 2. STI’s training staff offers courses in IPC and NASA certifications, as well as its own lesson plans.

“I’m always bragging that we have the best-trained workforce in the world because it’s literally a walk down the hallway to the best trainers in the world,” he said.

The company has six full-time instructors, five in Alabama and one in Indiana, with training classrooms in Madison and Houston, TX. More than 1,000 students come to the Madison facility every year, and instructors also spend a lot of time on the road, traveling to companies to provide training in their facilities, said Diana Bradford, VP of operations/training resources.

STI is also the only private company that offers NASA certifications, with NASA-STD-8739.4 – Crimp Cable &

Harness and NASA-STD-8739.1 – Polymeric Application on Electronic Assemblies available for onsite training in Madison.

The company can't offer every certification that can be earned in a government facility, but its close relationship with the agency and proximity to its largest research center put STI in a unique position among other private training centers, Raby said.

"We've got a long history with NASA, especially with Marshall Space Flight Center here, and they have put a lot of trust in Diana and our training staff," he said. "We're limited in what we can do, but we can do more than anyone else."

TYLER HANES is managing editor of PCD&F/CIRCUITS ASSEMBLY; tyler@pcea.net.

PCB Chat

pcbchat.com

Recent Chats:

- **SMT Assembly**
  with DANIEL STANPHILL, AURORA BOARDWORKS; SEAN KINCAID, K & F ELECTRONICS; and ELIAS MALFAVON, METAL ETCH SERVICES
- **PCB Industry Legislation**
  with DAVID SCHILD, PCBAA

The PCB Podcast

SEEING IS BELIEVING

First Person Plural

Three road trips underscore the importance of connecting.

RECENTLY I TRAVELED to the Atlanta area on business. The meeting was outside of Atlanta, in a suburb, a fair distance from both the airport and downtown. Given that distance, and Metro Atlanta traffic, I had a perfect chance to observe life from the back of the ridesharing car while the Lyft driver navigated and fulminated about 8 mph traffic and the consequent decline of civilization.

It's remarkable what you can observe in 90 freeway minutes while not driving. The molasses pace of "rush hour" illuminates a new world beyond the dashboard, much of it disagreeable. Like billboards. Scads of billboards. A throwback to the visual blight of the pre-Earth Day, zero-regulation, strip mall '60s, in the eyes of this Southern California-raised resident. Easily 100 billboards graced the shoulder and assaulted the senses between Hartsfield Airport and my hotel. Lest I doubted the evidence of my own eyes, the return journey from hotel to airport, 48 hours later, confirmed that my triple and quintuple takes weren't a mirage. Their sheer number paid throwback tribute to the nonregulatory state. Something we had in California when Lyndon Johnson was president, which we the people had the good sense to banish.

Strikingly, the majority of Georgia highway billboards advertised exactly one service: personal injury lawyers. Someone emerging from hibernation into the artificial light might reasonably conclude this was the region's sole industry. Attorneys' ads sequenced like roadside Alice in Litigation Land playing cards do nothing to dissuade this impression. I can attest that other industries populate the area, as my mission was to evaluate a machine we may acquire. Our target machine was not on a billboard. Reassuringly, if we do complete the acquisition, and if said unit fails, and if that failure is from negligence, and I feel sufficiently aggrieved, I know where to find a lawyer and exact my share. Just pull over on the highway, jot down a number, and see you in court the next day. So goes the Free Market for lawsuits in that part of the world. Who needs manufacturing? Adversarial relations pay better, judging from the roadside attractions.

But what about that market? Isn't the high ambulance-chasing quotient evidence of a taker – as opposed to a maker – society? And in conservative, no Guv'mint Georgia, no less. Whose laissez is truly faire? What is going on here?

I'm confused.

Shockingly, a superficial analysis plus years of education and experience compels the visiting observer to deduce that this is all about money and making it by paths of least resistance. This is distinct from more familiar Silicon Valley commerce, my world, which, as everyone knows and accepts, is about saving the world, because that's what superior

intellects do. And they said so. Q.E.D.

It's different in Georgia.

Perceptions matter. Maybe the market for personal injury attorneys is just subtler, more subterranean, more cosmopolitan and sophisticated in the San Francisco Bay Area, where I now live. Litigation in 140 characters.

Another trip. Another car ride. Another hotel. This one in Sacramento, closer to home. I can drive myself, which in Thanksgiving traffic is just as bad as Atlanta's, only more festive. Another 90-minute ride, this time taking three hours. The miles run off, and I talk to myself with gathering frequency, muttering oaths. No accidents are seen; just more congestion on the roads than the roads will accommodate. Traveling metaphorically blocked arteries to get to a place where sustenance blocks real arteries. Not an attorney billboard in sight.

Holidays are for family gatherings, so one's thoughts in the moment naturally gravitate to murder. November's chill heightens the effect. It is good such gatherings happen infrequently; otherwise, we'd be sick of politeness. Thanksgiving obligations mercifully concluded, and before opinion provokes action, I take my leave of family and check in to a nearby hotel. It's one of those modern, hip hotels, the kind playing endless cocktail party ambient music in the lobby as you hand over your credentials to a black-clad front desk attendant of severe demeanor, who disapprovingly beckons you in the direction of your room you should feel privileged to occupy for a night.

Without asking, I'm awarded a cavernous corner suite with the acreage of a mid-range mausoleum, with furniture and furnishings obviously spaced to accommodate wheelchair-bound guests. Or a handball court. Either they like my frequent guest membership and have granted me an upgrade, or the hotel is sparsely populated this Thanksgiving night, with a dearth of mobility-hindered patrons checking in to sleep off overcooked turkey. The sitting room in the suite has an echo. Digestive noises reverberate off the walls like stones.

It's an Edward Hopper kind of hotel. Full of Nighthawks: lonely people sitting in the bar and restaurant, maintaining deliberate distance from their nearest neighbor. Small satellites by choice. On a night for family, the enforced solitude is piercing. We carry on in mute ritual, bearing the great weight of aloneness.

Yet another hotel. This time it's a relief stop after an all-day trade show, all day being "on," pitching the advantages of flying probe, boundary scan and imaging to retired engineers seeking momentary company. When empathy ends and the "on" is disabled – usually around the first glass of wine at Happy Hour – the day's accumulated fatigue kicks in. Traveling home the usual 60-mile distance could be hazardous to one's health. Better to spend the night close to work, sleep well, recharge and not be a menace to society more than any other day.

This hotel chain has a polished way of annoying patrons. One's rest is disturbed by text messages. "Hi, I'm Crystal," she says, five minutes after check-in. "How is your stay so far?" "And be sure to select five out of five stars on the survey when you check out, so we can (game the system) get your feedback about our service." "Let me know if I can answer any questions." I wonder if she's real or AI.

Well, since you asked, there is one. Why does your hotel chain shaft its best customers? Until eight months ago, I was

in your highest rewards category. I lost that category on March 31 because of insufficient stays. Why do you penalize your best customers and demand that they requalify each year? In my case, I couldn't meet your stay criteria because I was home for three years, caring for my dying wife. Thanks for the recognition. Once we reach a new milestone, we should be permitted to stay there, regardless of stays. And remind me why, after all these years of loyalty, I should keep patronizing your chain?

Silence.

Three vignettes. So what's the point? What do these stories have to do with your company and our industry?

Fair question. My answer:

1. The world is not Silicon Valley-centric. Other parts of the country contain more lawyers than engineers.
2. Holidays are a very lonely time to spend in a hotel. Introspection is hard when you're lonely. The feeling becomes sharper in winter.
3. Big companies are in it for them, not you. The customer is a means to an end. Full stop.

Someone once said that it takes a village and was ridiculed for it. It turns out that person was partially right. It took me 20 years to realize that community, accompaniment and teamwork are essential. It is not nonsense. We need others. I learned this truth the hard way.

Which brings me to the moral of the story. Businesses are teams. Some teams work better than others. Some teams learn from their mistakes and get better; others don't. Some never learn. Those go out of business, some slower, some faster.

Lawyers don't add to GDP. They transfer wealth from one party to another. Nothing is added to the aggregate wealth of the country. That isn't growth. It's parasitism. It's telling when one realizes that a large portion of American legislators are lawyers, while a large portion of foreign rulers – in China, for example – are engineers.

Lonely people make bad leaders. Stuff gets done in groups. People support each other best in groups. Lone wolves are just that. Use the first person plural when describing your business and how you do business.

That's "we."

Give credit to the people who helped you to be successful. Never forget them. Say "thank you" once in a while, while looking them in the eye, and mean it. Without them, you are nothing.

Be grateful for every day. You don't know how many more you'll have. Gratitude is contagious.

Customer service needs to be meaningful. Deliver your service well, for a fair price, and follow it up with sincere, solicitous interest in the customer's welfare. Don't stack the deck with some worthless survey. Show you care; back it with follow-up. Admit your mistakes and be honest. Set expectations correctly. Be clear in your communication,

delivering bad news or good. People will respect you for that and will likely give you more business.

It took me 40 years to realize these things. They were right in front of me the whole time. I was a slow learner. Still am.

Three hotel visits in three different places give time for introspection.

For a short while at least, we remain a country and a business environment that believes in, encourages and seldom stigmatizes second, third and fourth acts.

So act. You're welcome.

ROBERT BOGUSKI is president of Datest Corp. (datest.com); rboguski@datest.com. His column runs bimonthly.

Optimizing Legacy Products

Respinning a board to eliminate PTH parts can pay off in production.

WHEN AN ELECTRONICS manufacturing services (EMS) provider is involved in developing a new product, it is easy to provide design for excellence (DfX) recommendations that align with Lean manufacturing philosophy.

Addressing manufacturability or testability issues associated with legacy products is much more challenging, however. The reasons printed circuit board layout or product design choices may not be optimal are many. Design cycles have been compressed and design resources have been cut at many OEMs. When layout or product development is outsourced to a third-party design team without manufacturing familiarity, the result may be a PCB assembly (PCBA) that meets form, fit, function and cost recommendations, but ignores industry standard design rules, sole sources much of the bill of materials (BoM) or requires unnecessary processing.

SigmaTron's new product introduction (NPI) process evaluates these issues as new projects are onboarded through a combination of an engineering team review and automated checks through a third-party design tool. When improvement opportunities are identified, the team can provide redesign or PCB layout services to correct the issues.

How does this align with Lean manufacturing philosophy? One recent through-hole to SMT conversion is a good illustration. A legacy, mixed-technology PCBA was experiencing material cost and availability issues because it had many through-hole parts that were either going end-of-life (EOL) or sustaining manufacturer price hikes. The OEM was open to converting the bulk of the through-hole parts to SMT. Connectors and a sensor needed to remain unchanged for the PCBA to function within its larger product and remain compatible with the existing test fixture, but the bulk of the layout could be optimized for manufacturability.

The design for manufacturability (DfM) analysis showed several issues:

- Due to tight clearances, through-hole components required manual placement, increasing assembly costs and creating defect opportunities
- SMT components had been placed on both sides of the PCBA with tight edge and component clearances, creating defect opportunities and requiring selective solder pallets in wave solder
- Top- and bottom-side components required two passes through the SMT line to place all parts, increasing the number of thermal cycles the PCBAs were exposed to.

The PCBA was redesigned to replace the problematic through-hole parts with SMT packages. The new layout placed all SMT parts on the top side, eliminating the need for a bottom-side placement pass and the concomitant thermal

cycle in reflow. It also enabled automated placement of the bulk of the components, eliminating much of the variation present in manual processes. This improved yields significantly. The single-sided design also eliminated the need for selective solder pallets in wave soldering, which lowered tooling costs and eliminated the pallet load and unload steps. Additionally, the SMT part replacements were cheaper and more widely available than their through-hole counterparts.

With product sizes shrinking, it is not uncommon to see edge clearance and component spacing design rules violated, particularly when a third-part design team performs the layout without equipment constraint information set up in their design tools. As shown in the example, that can impact the ability to automate production processes and create defect opportunities.

Companies stick with legacy designs that are not optimized for Lean environments for many reasons. Insufficient volume or regulatory issues may not justify the redesign cost. The design may have been appropriate at the originally specified volumes, and no analysis over whether the volumes have grown to justify redesign to better automate production has taken place. Product managers may not understand design issues' cost or quality impact on the PCBAs they manage. As this example illustrates, however, assembly costs and defect drivers in designs that don't incorporate Lean principles to enhance manufacturability can be significant.

The adage, "What costs a dollar in design, costs $10 in production and $100 in the field," illustrates the cost escalation driven by the failure to correct design issues. Considering the cost and quality benefits of a PCBA optimized for Lean manufacturing can be valuable in assessing whether the redesign of a legacy product is justified. When products are outsourced, collaborating with an EMS provider's engineering team is a good way to achieve the lowest total cost of ownership.

MARK BELLOT is a senior design engineer with SigmaTron International (sigmatronintl.com); mark.bellot@sigmatronintl.com.

Printed Circuit University

Stay on the forefront of technology

with the most **comprehensive database**

of online **courses and webinars**

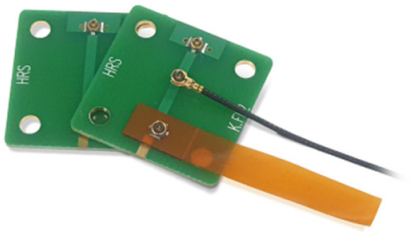
for the **printed circuit engineering** community.

Check it out
TODAY

printedcircuituniversity.com

PCU

PCD&F

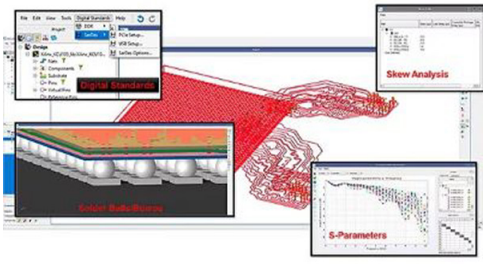


HIROSE K.FL2 MICRO RF CONNECTOR

K.FL2 series of micro RF connectors feature a profile of 1.2mm when mated. Come in two versions, an RF cable mating type and an FPC-to-board mating type, both supporting high-density mounting to save PCB real estate. Are said to deliver high-frequency performance and have low voltage standing wave ratios (VSWR) of 1.3 Max: 0-3GHz; 1.5 Max: 3-6GHz; 1.6 Max: 6-8GHz. Are compatible with pick-and-place mounting and are used in consumer applications, including laptop computers, smartphones, tablets, VR/AR glasses, routers and more. Use a tapered design for easy mating with significantly reduced insertion force, a polarity key for identification of the mounting direction and a clear tactile click to confirm proper mating. Robust, integrated plug design is said to prevent plug deformation during un-mating and nickel barrier at the shell and contact prevents solder wicking and flux penetration.

Hirose Electric

hirose.com



KEYSIGHT EDA 2025 SOFTWARE SUITE

EDA 2025 software suite uses AI, machine learning and Python integrations to reduce design time for complex RF and chiplet products. Enhances data manipulation, integration and control of simulators, for building efficient workflows seamlessly across multiple tools. Features AI-enhanced workflows and high-performance computing to reduce time-to-insight, as well as end-to-end component models and measurements that conform to digital standards for simulating fast digital interconnects.

Keysight Technologies

keysight.com

KYOCERA AVX CR RESISTOR SERIES

CR series of high-power chip resistors now includes high-power 0603 size package said to enable the miniaturization of RF power amplifiers and improve thermal management, with a high-thermal-conductivity substrate and maximized heat sink grounding area. Rated for 0.3pF and up to 2.6W, and features proprietary thin-film resistive elements, aluminum nitride substrates and silver terminals. Features two standard resistive values (100Ω and 50Ω), a power handling capability of up to 250W, rated operating temperatures from -55° to +150°C, and resistive tolerance as tight as ±2%.

Kyocera AVX

kyocera-avx.com

RS DESIGNSPARK PCB DESIGN SOFTWARE V.12

DesignSpark v.12 PCB design tool introduces features for free and paid subscriptions. New functions include pre-plot checks for correct layer selection, edit component options useful for high-pin footprints like BGAs, an angular precision setting for design units, dual units for the measurement tool and the ability to export components, including referenced symbols, from any library. Also now features a segment mode for shape and track editing, translate-to-PCB options designed to improve reference design conversion to PCB, the ability to save project-specific libraries, new component bin options like sort and filter for high-component-count designs, design rule check on current view and additional DRC options, like maximum vias allowed in each net, maximum stub route length allowed and check PCB-only nets.

RS

rs-online.com

SCHMID INFINITYLINE H+ ELECTROLESS CU LINE

InfinityLine H+ electroless copper system is designed for high-performance advanced packaging applications using mSAP and SAP processes. Is said to achieve highly uniform copper deposition and features enhancements such as an ozone module for pristine surface preparation, an advanced electroless copper module design and a three-point transport system that eliminates contact with active areas. Uses a horizontal process to treat each panel automatically under identical process conditions using efficient flood boxes that target treatment areas, enhancing the uniformity of copper thickness and providing superior performance in rinsing and drying substrate panels. Offers a smarter and more sustainable production solution through the Schmid Watch 3.0 HMI and process control system, which leverages the decentralized H+ bus system for active and passive components.

Schmid

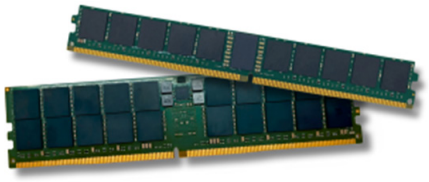
schmid-group.com

SCHMID TGV ETCHER

The acidic-based through glass via (TGV) etching system is said to work with all common glass types with CTEs ranging from 3 to 9. Reportedly delivers six to eight times higher productivity than traditional alkaline etching methods and features a 35% reduction in power consumption to maximize efficiency while fostering sustainable

manufacturing practices. Eliminates need for cassettes, reducing material costs and simplifying operation, and accommodates substrate sizes ranging from 100mm x 100mm to 600mm x 600mm. Comes in lab and high-volume manufacturing versions to meet requirements for research and development or large-scale production.

Schmid
schmid-group.com



UNIGEN DDR5 MEMORY MODULES

DDR5 family of high-performance memory modules come in registered DIMM (RDIMM) capacities up to 64GB at 5600MT/s speeds in both standard height and very low profile (VLP) form factors. Are 18.75mm in height and permit vertical DIMM placement in 1U blade servers and other embedded systems.

Unigen
unigen.com

CA

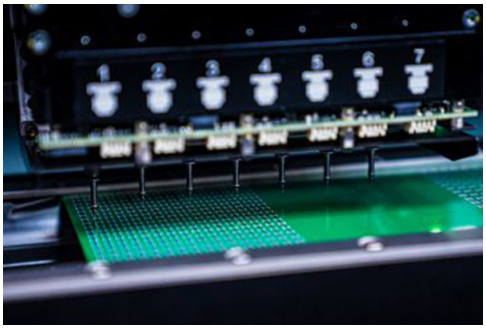
INDIUM CW-807RS FLUX-CORED WIRE

CW-807RS halide- and halogen-free flux-cored wire improves wetting speeds and cycle times for electronics assembly and robot soldering applications. Is a no-clean formula designed to maximize solderability of a halogen-

free cored wire solder flux and is an upgraded version of Indium's CW-807 formula. Provides faster wetting speeds, clear residues for electronics assemblers that use high (>385°C) tip temperatures and tend to experience charring and tip build-up and improves cycle times in robotic soldering applications.

Indium Corp.

indium.com



MYCRONIC MYPRO A40SX AND A40LX PLACEMENT MACHINES

A40SX and A40LX placement machines are equipped with new MX7 high-speed heads. Can place components as large as 45mm x 45mm x 15mm or 150mm x 40mm x 15mm and as small as 0.4mm x 0.2mm (01005). Feature top speeds of 29,000cph (SX) and 18,000cph (LX). Also feature up to 224 feeder positions, can place a full range of components in one machine, allow on-the-fly changeovers, can handle short tapes and nonstandard carriers and enable electrical verification of resistors, capacitors, diodes and transistors with full traceability. Also includes a newly designed GUI to simplify training and pick-and-place operations.

Mycronic

mycronic.com

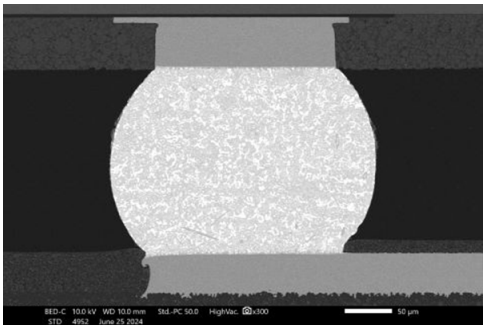
SAKI 3XI-M200V3 AXI

3Xi-M200v3 x-ray automated inspection system is part of the 3Xi-M200 AXI series, achieving a reported three times the resolution of its predecessors while reducing inspection times by up to 50%. Features a large, highly sensitive

detector that captures high-contrast images of ultra-thin materials and enhanced CT computation capabilities to improve resolution while cutting inspection times in half, permitting precise separation of multilayer components within complex power modules. Is designed to optimize overall production efficiency and features dedicated operator judgment software tailored for power module inspection to streamline final inspection processes. Is fully compliant with SEMI SECS/GEM standards and supports automation and workforce reduction initiatives.

Saki Corp.

sakicorp.com

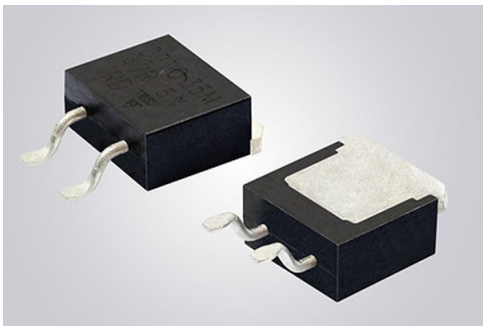


SHENMAO PF606-P SOLDER PASTE

PF606-P lead-free solder paste is designed specifically for the reverse hybrid assembly process, which uses SAC solder paste and BGA components with LTS balls. Is said to offer printability and solderability, with a specially designed flux for voiding control, minimizing void formation and avoiding short defects caused by ball volume expansion due to voids.

Shenmao America

shenmao.com



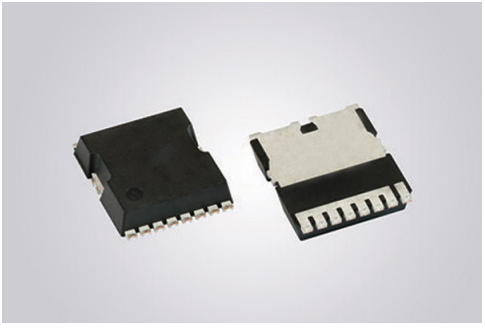
VISHAY D2TO35M RESISTOR

D2TO35M automotive grade surface-mount thick film power resistor combines multi-pulse capabilities with high power dissipation of 35W at 25°C. Is said to provide higher power dissipation and reliability for multiple and

repetitive pulse use than standard devices in the TO-263 (D2PAK) package, and offers a maximum drift of 2% after 1000 pulses at 25°C and a drift less than 5% after 100,000 pulses. Features turn-on time of 500ms, turn-off time of 11 sec. and pulse energy of 18.9J, withstands 1,000 cycles of rapid change of temperature (RCT) testing and offers a load life of 1000 hr. Also features a resistance range from 10Ω to 10kΩ – with tolerances of ±2%, ±5% and ±10% – thermal resistance of 4.28°C/W, TCR of 150±ppm/°C, and an operating temperature range from -55° to +175°C. Is RoHS compliant and solder reflow secure at 270°C/10 sec.

Vishay Intertechnology

vishay.com

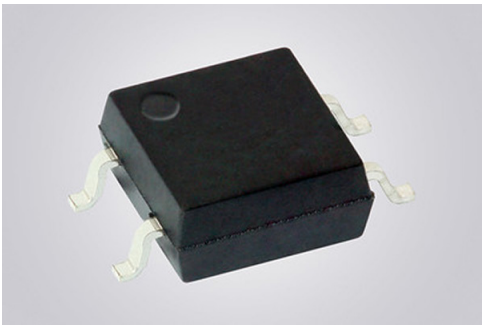


VISHAY SILICONIX SIJK140E MOSFET

Siliconix SiJK140E 40V MOSFET is said to reduce on-resistance by 32% while offering 58% lower on-resistance than 40V MOSFETs in the TO-263-7L. Minimizes power losses from conduction to increase efficiency while improving thermal performance with a low RthJC of 0.21°C/W typical. Allows designers to utilize one device instead of two in parallel to achieve the same low on-resistance, improving reliability and mean time between failures. Features a bond-wireless (BWL) design that minimizes parasitic inductance while maximizing current capability, and offers a continuous drain current up to 795A for increased power density while providing a robust SOA capability. Occupies an area of 120 mm^2 in the PowerPAK 10×12 package, saving 27% PCB space compared to the TO-263-7L while offering a 50% lower profile. Is for synchronous rectification, hot swap switching and OR-ing functionality, with typical applications being motor drive controls, power tools, welding equipment, plasma cutting machines, battery management systems, robotics and 3-D printers. Offers a threshold voltage of 2.4Vgs, is RoHS-compliant and halogen-free, and is 100% Rg and UIS tested.

Vishay Intertechnology

vishay.com



VISHAY VOR1060M4 SOLID-STATE RELAY

VOR1060M4 solid-state relay offers a 600V load voltage and isolation voltage of 3750VRMS in the low-profile SOP-4 package. Features a turn-on time of 0.3ms typical and low leakage current of 2nA. Is designed to provide isolated switching in inverters, motor controls and battery management systems (BMS) in energy storage systems (ESS); industrial motor drives, power tools and controls; security and automation systems and instrumentation. Is RoHS-compliant, halogen-free and Vishay Green.

Vishay Intertechnology

vishay.com

In Case You Missed It

Electromigration

"Comparison of Electromigration in Tin-Bismuth Planar and Bottom Terminated Component Solder Joints"

Authors: Prabjit Singh, *et. al.*

Abstract: Electromigration monitoring of bottom terminated component (BTC) solder joints is limited to electrical resistance measurements of the solder balls. Tracking the microstructural evolution such as bismuth segregation in tin-bismuth solder balls is typically performed via metallurgical cross-sectioning, a destructive technique. Once cross-sectioned, the solder ball is not available for further electromigration current stressing. A novel planar solder geometry has been invented and developed that allows real-time, nondestructive monitoring of solder microstructure, while the progress of electromigration can be concurrently tracked via electrical resistance means. Planar solder joints are easy to fabricate in a typical metallurgical laboratory. If the electromigration behavior of the planar and the BTC solder joints happen to be similar, the planar solder joint approach could greatly aid in the quick development of solder alloys by comparing rates of electromigration and metallurgical changes in planar solders of various compositions. In this work, the electromigration rates and behavior of eutectic SnBi alloy in planar and in BTC solder joints were compared and shown to be similar. This important finding opens the use of planar solder joints for the quick and low-cost development of low-temperature solder alloys. (*Journal of Surface Mount Technology*, November 2024, https://doi.org/10.37665/aha4dx58)

Flex Circuits

"Pressure-Constrained Sonication Activation of Flexible Printed Metal Circuit"

Authors: Lingxiao Cao, *et. al.*

Abstract: Metal micro/nanoparticle ink-based printed circuits have shown promise for promoting the scalable application of flexible electronics due to enabling superhigh metallic conductivity with cost-effective mass production. It is challenging to activate printed metal-particle patterns to approach the intrinsic conductivity without damaging the flexible substrate, however, especially for high-melting-point metals. Here, the authors report on a pressure-constrained sonication activation (PCSA) method of printed flexible circuits for more than dozens of metal (covering melting points from room temperature to 3422°C) and even nonmetallic inks, which is integrated with the

large-scale roll-to-roll process. The PCSA-induced synergistic heat-softening and vibration-bonding effect of particles can enable multilayer circuit interconnection and join electronic components onto printed circuits without solder within 1 sec. at room temperature. The authors demonstrate PCSA-based applications of 3-D flexible origami electronics, erasable and foldable double-sided electroluminescent displays, and custom-designed and large-area electronic textiles, thus indicating its potential for universality in flexible electronics. (*Nature Communications*, September 2024, https://doi.org/10.1038/s41467-024-52873-7)

Sustainability

"Leaftronics: Natural Lignocellulose Scaffolds for Sustainable Electronics"

Authors: Rakesh R. Nair, *et. al.*

Abstract: The global rise in electronic waste is driven by the persistent use of glass, epoxy, and plastic substrates owing to their cost, stability, flexibility and transparency. This underscores the need for biodegradable alternatives with similar properties. This work shows that leaf-derived lignocellulose scaffolds can stabilize bio-sourced, solution-processed polymers by acting as natural sequestering media. Such reinforced films, even when based on gelatin (Tg ~60°C), can endure processes over 200°C. The authors demonstrate dip-coated ethyl cellulose films for commercially viable reflow soldered circuitry. The films offer high flexibility, more than 80% transparency, and surface roughness below 5.5nm. Advanced OPDs and OECTs fabricated on these films perform comparably to those on glass and the low material cost and simple fabrication process yields a minimal carbon footprint of 1.6kgCO_2/m^2. This work thus opens a vista of possibilities for biodegradable polymers heretofore considered unsuitable for making temperature-stable substrates for state-of-the-art electronics applications. (*Science Advances*, November 2024, https://doi.org/10.1126/sciadv.adq3276)

Wearables

"Revolutionizing Wearable Technology: Advanced Fabrication Techniques for Body-Conformable Electronics"

Authors: Ruilai Wei, *et. al.*

Abstract: With the increasing demand for wearable electronic products, there is a pressing need to develop electronic devices that seamlessly conform to the contours of the human body while delivering excellent performance and reliability. Traditional rigid electronic fabrication technologies fall short of meeting these requirements, necessitating the exploration of advanced flexible fabrication technologies that offer new possibilities for designing and fabricating flexible and stretchable electronic products, particularly in wearable devices. Over time, the continuous development of innovative fabrication techniques has ushered in significant improvements in the design freedom, lightweight, seamless integration, and multifunctionality of wearable electronics. Here, the authors provide a comprehensive

overview of the advancements facilitated by advanced fabrication technology in wearable electronics. It specifically focuses on key fabrication methods, including printed electronics fabrication, soft transfer, 3-D structure fabrication and deformation fabrication. By highlighting these advancements, it sheds light on the challenges and prospects for further development in wearable electronics fabrication technologies. The introduction of advanced fabrication technologies has revolutionized the landscape of wearable/conformable electronics, expanding their application domains, streamlining system complexity associated with customization, manufacturing and production, and opening up new avenues for innovation and development of body-conformable electronics. (*NPJ Flexible Electronics*, December 2024, https://doi.org/10.1038/s41528-024-00370-8)