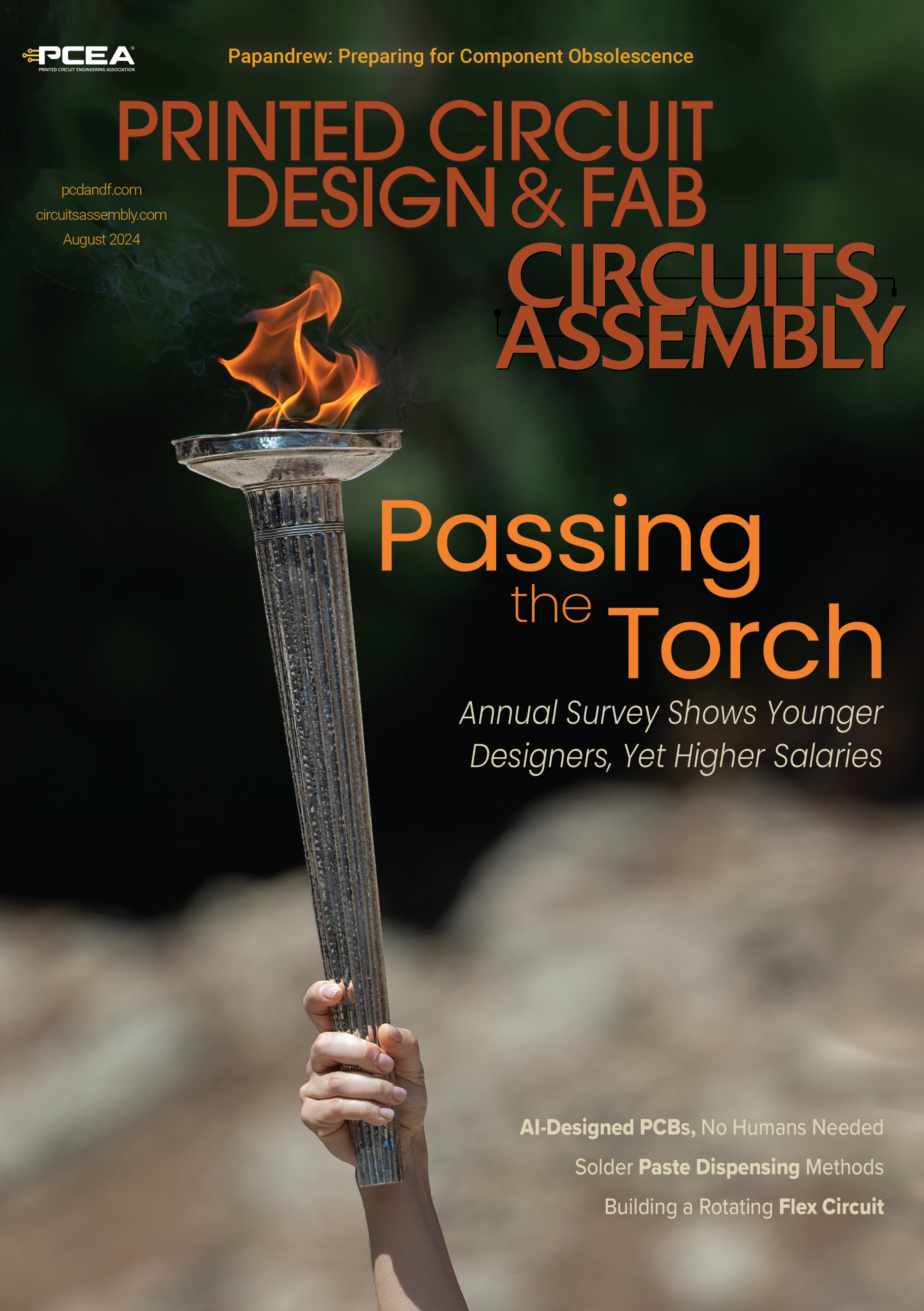


PCEA
PRINTED CIRCUIT ENGINEERING ASSOCIATION

Papandrew: Preparing for Component Obsolescence

PRINTED CIRCUIT DESIGN & FAB
CIRCUITS ASSEMBLY

pcdandf.com
circuitsassembly.com
August 2024

Passing the Torch

Annual Survey Shows Younger Designers, Yet Higher Salaries

AI-Designed PCBs, No Humans Needed

Solder **Paste Dispensing** Methods

Building a Rotating **Flex Circuit**


Xray
START
STOP
FISCHERSCOPE® X-RAY XDAL®

AUGUST 2024 • VOL. 41 • NO. 8

PRINTED CIRCUIT DESIGN & FAB CIRCUITS ASSEMBLY

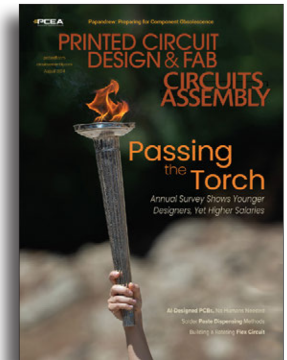
FIRST PERSON

MONEY MATTERS

TECH TALK

DEPARTMENTS

FEATURES



ON PCB CHAT (PCBCHAT.COM)

PCB Chat

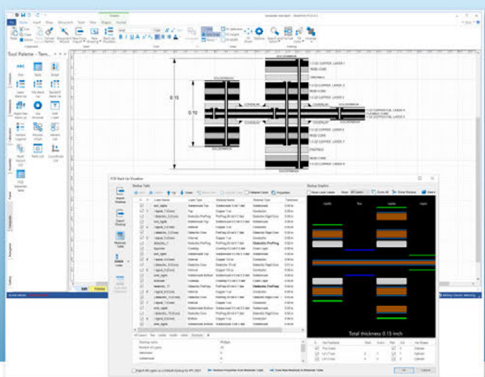
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

BluePrint·PCB®

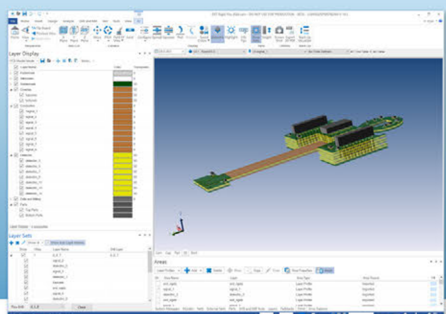
CAM350®

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.

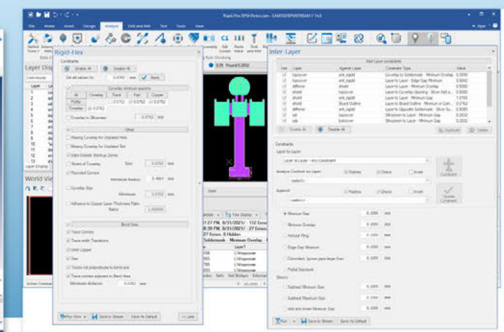
- **Import and Visualize Flex, Rigid-Flex and Embedded Component Designs**
- **3D Visualization to Validate PCB Construction and Component Assembly**
- **Manage Variable Stackup Zones for Rigid-Flex Designs**
- **Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation**
- **Use 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

PCEA
Printed Circuit Engineering Association

PCEA
PO BOX 807
AMESBURY, MA 01913

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

MANAGEMENT

PRESIDENT
Mike Buetow 617-327-4702 | mike@pcea.net

VICE PRESIDENT, SALES & MARKETING
Frances Stewart 770-361-7826 | frances@pcea.net

PCD&F/CIRCUITS ASSEMBLY EDITORIAL

MANAGING EDITOR
Tyler Hanes 205-258-0067 | tyler@pcea.net

COLUMNISTS AND ADVISORS

Peter Bigelow, Robert Boguski, John D. Borneman, John Burkhert, Jr., Stephen Chavez, Geoffrey Hazelett, Mark Finstad, Jake Kulp, Nick Koop, Alun Morgan, Susan Mucha, Greg Papandrew, Chrys Shea, Jan Vardaman, Gene Weiner

PRODUCTION

ART DIRECTOR & PRODUCTION
blueprint4MARKETING, Inc. | production@pcea.net
Nathan Hoeller | nathan@pcea.net

SALES

VICE PRESIDENT, SALES & MARKETING
Frances Stewart 770-361-7826 | frances@pcea.net

SENIOR SALES EXECUTIVE
Will Bruwer 404-313-1539 | will@pcea.net

REPRINTS

sales@pcea.net

EVENTS/TRADE SHOWS

EXHIBIT SALES
Frances Stewart 770-361-7826 | frances@pcea.net

TECHNICAL CONFERENCE
Mike Buetow 617-327-4702 | mike@pcea.net

EVENTS MANAGEMENT
Jacqueline Bress 404-955-7675 | jacqueline@pcea.net

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.

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

DATABASE
Directory of EMS Companies .... circuitsassembly.com

There's One Problem AI Can't Resolve: Implementation

SPEAKING, AS WE were last month, about artificial intelligence and its adoption into electronics design and manufacturing, we observed that a current obstacle to implementation is the use by vendors of customer data in order to build their models.

And while vendors insist the data are aggregated and anonymized, said customers, naturally, have been generally circumspect over the perceived cost of the lessons they have learned – often painstakingly – being used to enable competitors, not to mention ultimately paying those same vendors for the courtesy.

To that I will add the musings of Neil Thompson, who is the director of the future tech research project at MIT's Computer Science and Artificial Intelligence Laboratory.

Thompson argues that AI systems must not just be capable of performing "human" tasks but also must overcome the costs of implementation, including redesigning processes and methodologies. "There are a lot of places where … humans are a more cost-efficient way," he says.

Over the past couple years, via our PCB Chat podcast and webinars, our editors have spoken with a growing number of AI experts across the spectrum of the electronics supply chain. They include:

- Sebastian Schaal, cofounder of Luminovo, a Munich-based startup working on applying machine learning to electronics development and production processes;
- Leah Slaughter, then vice president of supply chain and artificial intelligence programs at a multinational EMS company, on how AI is being used to improve electronics supply chain management;
- Matthias Wagner, chief executive and cofounder of Flux, a developer of a browser-based PCB design platform and programmable simulator;
- Andrew Scheuermann, cofounder and CEO of Arch Systems, a developer of software tools that collect raw machine data and use predictive analytics to calculate manufacturing key performance indicators (KPIs);
- Tomide Adesanmi, founder of Circuit Mind, an AI-driven component selection and schematic capture platform;
- Arif Virani and Bart Piwowar, then chief operating officer and CTO, respectively, of Darwin AI, a

manufacturer of innovative inspection systems (now acquired by Apple);

- Taylor Hogan, Cadence Design Systems; Kyle Miller, Ph.D., Zuken; and David Wiens, Siemens Digital Industries, major ECAD vendors that have invested in AI-driven features.

And see our interview this month with Sergiy Nesterenko, founder and CEO of Quilter, an AI software company whose goal is to accelerate hardware development by fully automating circuit board design.

The suggestions that AI can reduce time spent coding are well-accepted and, indeed, one would hope the effort that goes into setting constraints could be automated (assuming they are correctly prompted – precise communication is key to effective AI use) as well. But it all turns on specificity. As Scheuermann told us earlier this year, "What we have to do in manufacturing is combine these generic AIs that are being built with the specific data of each manufacturer's problem, which needs to be kept private and secure only to them, so they can combine these things in a controlled environment and then they can get the best use of AI combining general AI and their specific data for their problems." While Scheuermann was couching his opinion around manufacturing, it applies all the same to design.

As the business models continue to take shape, vendors will have to think long and hard about how to overcome user hesitancy. To that end, we are excited to announce our keynote speaker for PCB West in October is Charles Pfeil. Few understand the nature of printed circuit design tools better than Pfeil. He was the original product architect of Expedition PCB, and an inventor of Team PCB, XtremePCB, XtremeAR, and the Sketch Router. If anyone knows the nature of engineers when it comes to implementing new tools and features, it's Pfeil.

Pfeil's talk, "AI Roadmap for PCB Layout: New Opportunities for PCB Designers," will home in on the impediments to adoption and how to overcome them. His solution involves balancing the growing capabilities of AI with the preservation of proprietary design knowledge and methodologies. (His talk will take place October 9 at 11 a.m. at the Santa Clara [CA] Convention Center.)

As PCEA continues to work on the industry roadmap for implementing AI, Pfeil's talk is sure to spur thoughtful action toward a workable solution for a conundrum even AI hasn't been able to solve.

mike@pcea.net

@mikebuetow

P.S. Our thoughts are with the families and colleagues of George Dudnikov, an engineer and executive at several leading PCB fabricators, and Jim McElroy, the first CEO of iNEMI, both of whom passed away this summer.

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

NCAB Acquires PCB Units of EPI Components Trade, DVS Global

BASSANO DEL GRAPPA, ITALY – NCAB Group in July announced the acquisition of the PCB division of DVS Global, an Italian electronics provider. The PCB distributor will pay SEK200 million ($18.6 million), with a possible further earnout of a maximum of SEK45 million ($4.2 million).

DVS was founded in 2003 and also has offices in Switzerland, Hong Kong and China. Annual sales of the PCB division in 2023 amounted to SEK230 million ($21.4 million), with sales primarily directed toward Italian customers in the industrial and automotive sectors.

NCAB is acquiring the PCB division of the company, while the previous owners will focus on the remaining division, Teknokit. The deal is expected to close in September and the sellers will stay during a transition period.

NCAB Group said it expects the acquisition to be accretive to earnings in 2025.

"DVS is an important addition to NCAB Italy," said Peter Kruk, CEO, NCAB. "The business concept is similar to NCAB's and the company has a good reputation and a strong market position, being located close to the areas that have a high density of electronics industries in Lombardy and Veneto. Adding DVS to our existing operations in Italy, including the acquisition of Prevent in 2021 and BBC in 2022, is making NCAB a major player in the PCB market in Italy."

"Since its foundation, DVS has built its reputation on excellence of customer service, a stable and well managed supply chain and loyal enthusiastic employees," said sellers Denis Vigo and Paolo Facco. "We have been looking for a way to expand our business and bring more opportunities to our customer base. We are excited to join forces with NCAB and are confident this will be an interesting and rewarding step both for our customers and employees."

NCAB Group also has acquired the former printed circuit board unit of EPI Components Trade, based in Lireserbrucke, Austria, along with its employees and customers.

The unit's annual revenues are roughly SEK 35 million ($3.34 million), mainly from industrial customers, and a profitability in line with NCAB's.

The unit has four employees including owner, Christian Brugger, who will stay on, NCAB said. Sourcing comes mainly from Slovenia. Synergies are expected in the areas of suppliers, payment terms and logistics.

In a press release, Benjamin Klingenberg, vice president, NCAB Europe, said, "I am very excited about this acquisition in the important Austrian market. In the past, Austria was served from the German NCAB operation and with this move we get boots on the ground with an organization and management to fully focus on our customers there. With Christian and his team, we will win a lot of PCB know-how and customer relationships."

Added Brugger: "I am pleased to become part of the NCAB Group with our employees, who have more than 20 years of experience in PCB sales in Austria. With the entire NCAB portfolio, we can offer our customers a wider range of service which will give us new business opportunities and make us able to expand our customer relations."

ZDT JV Signs Agreement to Develop Thai PCB Workforce

BANGKOK – Peng Shen Technology, a joint venture between Saha Group and Zhen Ding Technology, has initiated a collaborative project with Thai educational institutions to develop local talent for the country's PCB manufacturing industry.

The company has signed cooperation agreements with five leading Thai educational institutions with a stated goal to increase Thailand's share of the global PCB production and elevate its position to the forefront of the industry.

The company said the collaboration aims to achieve five key objectives:

- Collaboration on developing national industry policies
- Collaboration on training and developing PCB personnel
- Collaboration on teaching PCB-related vocational subjects and training industry experts
- Collaboration on student internships, both domestic and international
- Collaboration on factory visits, career guidance, and job promotion

Peng Shen said it is confident that this cooperation will help drive the organization's mission to develop science and technology that benefits humanity, while also restoring and preserving the environment, making the world a better place.

Representatives from the company and educational institutions attended a signing ceremony in June with Siriporn Pittayasophon, senior strategist and acting director of Thailand's National Higher Education, Science, Research, and Innovation Policy Council, serving as the guest of honor.

Pittayasophon said MHESI has a strategy to promote and develop human resources in semiconductors and advanced electronics to prepare for driving the Thai economy toward a sustainable future, and the announced initiative is expected to reassure investors and significantly contribute to developing high-skilled personnel that meets the needs of the country's advanced electronics industry.

Vorayos Thongtan, president of Saha Pathana Inter-Holding, said the rapid advancement and growth of current technologies, such as AI, electric vehicles and the Internet of Things have increased the complexity and demand for

PCBs. He said China is the world's leading PCB production base, followed by Taiwan, South Korea, and Japan, while Thailand is recognized as the leading PCB manufacturing base in ASEAN, attracting foreign PCB investors due to the expansion of production bases and continuous relocation of manufacturing plants to Thailand.

Peng Shen Technology marks the beginning of ZDT Group's expansion into ASEAN. The company is currently constructing a factory in Saha Group Industrial Park in Kabinburi, which will impact Thailand's potential to become a leading PCB production country in the world, Thongtan said, so preparing infrastructure to support manufacturers must go hand in hand with developing skilled PCB personnel.

The collaboration with five leading educational institutions, namely King Mongkut's Institute of Technology Ladkrabang (KMITL), King Mongkut's University of Technology North Bangkok (KMUTNB), Kasetsart University, National Institute of Development Administration (NIDA), and Sirindhorn International Institute of Technology (SIIT), Thammasat University, is a significant step toward a shared future and serves as a declaration of Thailand's commitment to becoming a global PCB production hub, Thongtan said.

APCT Relaunches as AdvancedPCB

AURORA, CO – Advanced Printed Circuit Technology, Advanced Circuits and San Diego PCB Design have combined to relaunch as AdvancedPCB.

"Since the founding of APCT in 1977, we've expanded operations and expertise through more than 10 acquisitions. At the same time, we've consistently teamed up with global leaders in PCB design, manufacturing, and innovation," said Peter Austin, CEO, AdvancedPCB. "This new brand represents our commitment to powering the next generation of technology, from healthcare to aerospace and defense to consumer products.

"The semiconductor industry continues to grow quickly in North America and across the globe. American-made AdvancedPCB is well positioned to service this growth," Austin added. "Our products and solutions extend through global markets, providing enterprises in myriad sectors outside of North America with the same range of security, reliability, and quality standards that customers within the United States have come to expect."

AdvancedPCB operates six PCB manufacturing sites and four PCB design sites across the United States, employing nearly 1,000 people and already serving as the second-largest PCB manufacturer in North America. With an expansive factory network and design capabilities, AdvancedPCB said it can manufacture a wide range of custom circuit boards, from simple prototypes to complex production, while providing high-quality technology and rapid turn times.

AdvancedPCB also launched a new website, which allows users to quickly find products and design services for the next wave of advanced technology. PCB enthusiasts and designers can create custom designs and get instant quotes.

Alba PCB Group Acquires WM Eltar

TARNOWSKIE GORY, POLAND – Alba PCB Group has acquired fellow printed circuit board fabricator WM Eltar, based here outside of Krakow.

Terms were not disclosed.

WM Eltar specializes in the production of single- and double-sided PCBs, aluminum and flex PCBs. Alba produces PCBs and offers R&D and engineering in Italy and China.

"The two companies share that the advantages of the transaction lie in the synergy of corporate values that start with European production," said Alberto Bacchin, president, Alba PCB Group. "The strength of local production led by experienced professionals such as Dorota Łozowska, lies above all in the complementarity of offers and markets and makes it possible to optimally meet customers' needs and ensure a stable and competitive presence on the local and international market."

"This is a crucial moment in the development of WM Eltar," said Dorota Łozowska, CEO, WM Eltar. "Joining Alba PCB Group will allow us to significantly expand the offer for Polish customers and support them in business development. Thanks to the state-of-the-art technology of the Italian factory and a professional service from Asia, with a well-established and structured presence in China, we will be able to offer a complete PCB portfolio and professional service."

Wistron to Invest $25M in Latest Vietnam Campus

DONG VAN WARD, VIETNAM – Wistron is investing $24.5 million in a new campus for electronics assembly in northern Vietnam, local media are reporting, citing government filings.

Equipment installation for the 49,000 sq. m. campus in the Dong Van III industrial park is scheduled for November, with production beginning shortly thereafter.

According to reports, the plant will have an annual capacity of more than four million LCD displays, docking stations, webcams and other Internet-related devices.

Wistron is also investing $45 million in another site in Ha Nam. The ODM, ranked among the top five on the CIRCUITS ASSEMBLY Top 50, has invested more than $360 million in the country since 2020 as it rebalances capacity with India and China.

Variosystems Acquires Swiss EMS Firm

STEINACH, SWITZERLAND – Variosystems in June announced the acquisition of the EMS division of the Swiss electronics components manufacturer Schurter.

With the acquisition of Schurter Solutions, Variosystems said it has strengthened its European supply chain by adding a manufacturing site in Romania, and enhanced its engineering and rapid-prototyping capabilities via a second VARIOincubator hub in Mendrisio, Switzerland. The facility in Mendrisio will support and develop customers in Northern Italy, France and Switzerland, the company said.

The two added locations in Mendrisio and Gruiu, Romania, double Variosystems' production capacity in Europe. The manufacturing site in Romania will focus on high-volume customer programs that require highest cost competitiveness. The site in Mendrisio will focus on engineering, rapid-prototyping and low-volume production services.

"With the acquisition of Schurter Solutions, we expand our expertise in the development and rapid prototyping of electronics solutions and strengthen our supply chain in Europe. By doing this, we accelerate our transformation into a global contract development and manufacturing organization (CDMO) and support our European customers in their nearshoring initiatives," said Variosystems CEO Stephan Sonderegger. "We are delighted to be able to offer existing Schurter Solutions customers direct access to our integrated electronics service platform and global supply chain network, giving them a competitive advantage."

By handing over its electronics manufacturing, Schurter said it is ensuring long-term job security and pooling resources for the ongoing development of its core business.

"Schurter Solutions has established itself as a first-class address for all-inclusive solutions and is now moving to Variosystems, where it will be in the best of hands," said Lars Brickenkamp, CEO, Schurter, "Together, we are setting the course for the successful further development and will remain closely connected as a partner in the future."

Variosystems' VARIOincubator sites provide services for development and industrialization of electronic solutions, and following the acquisitions of Solve GmbH in Buchs, Switzerland, in 2020 and Kubeg AG, in Zizers, Switzerland, in 2023, the integration of the Mendrisio site is the third major step in the company's VARIOincubator initiative.

With Schurter Solutions' location in Romania, Variosystems said it has completed its supply chain in Europe. In addition to its production site in Croatia, which specializes in box-build assembly services, the hub in Romania will focus on printed circuit board assembly, permitting the company to offer an integrated supply chain between the two locations for cost-competitive box-build assembly.

"The integration of Schurter Solutions enhances our production services in Europe and makes Variosystems an ideal partner for customers that require a cost competitive supply chain in that region," said Stefan Walther, COO, Variosystems.

Starteam Acquires Italian PCB Factory

HONG KONG – Starteam Global has acquired CTB Circuiti Stampati and its PCB manufacturing facility located in the Italian town of Flero.

The PCB manufacturer said the new factory has been rebranded as Flero Starteam, and amplifies its manufacturing capability and market presence in Europe. Starteam said Flero Starteam will concentrate on innovation through prototyping, sample production, specialized high-mix, low-volume manufacturing, quick turn, and customized requirements.

"This expansion is a milestone for Starteam, reinforcing the company's presence in the heart of Europe and highlighting its dedication to serving its European customers in the most local and flexible way," the company said in a release.

The Flero facility's technology will initially focus on manufacturing one- to two-layer PCBs, including PTFE base material (Teflon) for high-frequency applications, with plans to expand capabilities up to six layers by 2025, Starteam said.

Uwe Schneider, formerly automotive technical director at the Jiangyou Starteam factory in China, has been appointed general manager. Roberta Tonini, the former owner of CTB, will also take on the general manager role.

Nano Dimension to Acquire Desktop Metal

WALTHAM, MA – Nano Dimension entered into a definitive agreement to acquire all outstanding shares of Desktop Metal for around $183 million.

Under the agreement, Nano Dimension will acquire all outstanding shares of the 3-D printing system provider in an all-cash transaction for $5.50 per share, subject to possible downward adjustments to $4.07 per share, for a total price of approximately $183 million, possibly down to $135 million in total with possible adjustments.

The transaction is expected to close in the fourth quarter of 2024, subject to the satisfaction of customary closing conditions.

"Our combination with Desktop Metal is another step in Nano Dimension's evolution to become the leader in digital manufacturing, with capabilities in mass manufacturing for critical industrial applications," said Yoav Stern, CEO, Nano Dimension. "We're excited to join forces with an excellent group of technology leaders, all of whom share our vision for transforming manufacturing to Digital Industry 4.0. I look forward to working with Ric Fulop and his team to drive value for all our stakeholders, including creating opportunities for our employees as part of a larger, more diversified global innovative company, driving customer support and generating long-term growing value for shareholders as we focus on profitable growth."

"We're excited to bring together our pioneering, complementary product portfolios that will further enhance our ability to serve our customers in high-growth industries with a more complete offering of digital manufacturing technologies for metal, electronics, casting, polymer, micro-polymer and ceramics applications," said Ric Fulop, Desktop Metal CEO and cofounder. "We look forward to working with Nano Dimension to join two great companies and their devoted teams that can serve our stakeholders to the maximum extent possible."

Desktop Metal maintains platforms focused on industrial-volume scale applications of metal and polymer with proprietary materials, software, and sintering solutions, and the combined company will be the first AM provider covering the full gamut of customer needs from prototyping to production across a range of critical and high-performance medical and electronics applications in industrial and high-performance materials, Nano Dimension said in a release.

The company said Nano Dimension and Desktop Metal's portfolio will be focused on high-tech, high-margin solutions and are supported by an installed machine base of over 8,000 systems, representing significant opportunities for recurring revenue generation from a larger services and consumables offering. The combined company will benefit from a significantly enhanced scale and a diversified profile with 2023 combined revenue of $246 million.

Kitron to Build New Production Facility

ARENDAL, NORWAY – Kitron is planning a 7,500 sq. m. production facility here, with plans for completion during the first half of 2026.

"There is growth and a high level of activity at Kitron in Arendal, and we believe the activity will increase further in the years to come," said Heine Østby, managing director of Kitron Norway. "We are simply outgrowing the current premises. Now we get the opportunity to help design the perfect building for our needs. This gives us the opportunity to grow further here in Arendal, where Kitron has its roots back to the 1960s. We are extremely happy about that."

Kitron's operations in the area are currently divided between a facility of 7,300 sq. m., in Kilsund, Norway, and 4,000 sq. m. of rented premises at Stoa. When the new facility is ready, the company will move its Stoa operations to the new site, while the plant in Kilsund will continue as is.

Garner Osborne Acquired by Entrepreneur

NEWBURY, UK – Garner Osborne, a UK-based specialist in PCB manufacture and assembly, has been acquired by entrepreneur Dean Curran through Novastone Capital Advisors' Entrepreneurship Through Acquisition program.

Operating for more than 33 years, Garner Osborne delivers PCBs that include rapid prototyping, small- to medium-sized volumes, and large-volume offshore manufacture. The transaction provided an exit for founder and former CEO Mike Garner and his business partner Nigel Priest, with Curran stepping into the business as CEO and co-owner. Garner will remain a minority stakeholder in the company.

"I am grateful to Mike for trusting me to continue his legacy," said Curran. "I have been hugely impressed by the operational excellence and extremely high-quality PCB products that Garner Osborne manufactures, supplying a diverse range of customer sectors. The business has great potential for growth."

Curran said NCA's Entrepreneurship Through Acquisition program played a pivotal role in finalizing the acquisition.

"This deal would have been impossible without NCA's support, in particular Albert Farreras and Isaac Leyne from NCA's European M&A team, and Noora Haapaniemi from NCA's legal department," he said. "Having a team that I could turn to throughout the process really proved this program is unparalleled in comparison to anything else in the search fund world."

Lava Considers Spinoff of Manufacturing Arm

NOIDA, INDIA – Lava International is reportedly considering a spinoff of its production arm into a separate company, which may generate business from other handset brands and help meet targets under the production-linked incentive scheme for smartphones.

The contract manufacturer, which has not been able to meet its PLI smartphone targets since the scheme began in 2020-21, is hopeful of a turnaround in its manufacturing fortunes with the upcoming incentive package for component production, head of manufacturing Sanjeev Agarwal told *The Economic Times*.

"Lava's performance in the ongoing PLI scheme is an internal concern, and we are working on plans to bridge the gap," he said.

Agarwal said the company could spin off the manufacturing arm to make it independent, but has not yet decided.

He said the company's failure to meet smartphone PLI targets starts with a higher base that made incremental targets more difficult for Lava than others who started their manufacturing journey with the onset of the scheme. The company also could not benefit from the Indian government's push toward Chinese smartphone brands working with PLI-eligible domestic firms for meeting local demand and exports, as rival brands have policies in place to not partner with manufacturers who also operate their own brand, Agarwal said.

Lava also plans to start localizing more components after starting with printed circuit board assembly, ET reported.

"Now we are seeing which other components we need to focus on from the bill of materials. This will be based upon technology and capital requirements – which ones to do in-house and which components to outsource, that's what we are working on," Agarwal said.

FIT Launches In-House Circuit Assembly Services

MILPITAS, CA – Flex Interconnect Technologies launched its own circuit board assembly service in June, bolstering support for product development initiatives throughout North America.

"We are thrilled to now offer in-house assembly," said Chetan Shah, president and CEO, FIT. "We recognize the growing demand in the electronics market for quickturn, low-volume circuit assembly with exceptional quality and high yields."

While reshoring has brought assembly work back to the United States, higher volume production orders take up existing assembly capacity, the company said. Product development teams can struggle to find low-volume solutions to meet their go-to-market timelines, and by focusing on low-volume projects with quickturn timelines, FIT said its offering fills a current hole in the market.

The company said it will be able to offer faster quote turnarounds, shorter lead times and more competitive pricing.

"Our assembly service will quickly be a valuable tool for our customers," said Shah. "Our quickturn capabilities will allow them to bring their ideas to market more quickly than ever."

Chinese EDA Startup Slashes Workforce

NANJING, CHINA – X-Epic, a Chinese developer of electronic design automation tools, is rumored to be laying off 50% of its workers after struggling to gain ground in the EDA space.

The *South China Morning Pres*s reported that the firm is struggling in its mission to make up ground on the US firms that dominate the EDA space: Cadence, Synopsys and Mentor Graphics.

X-Epic was founded in March 2020 and raised $121.7 million while developing a suite of chip design software EDA tools covering areas such as digital chip verification, hardware simulation, system debugging, and cloud-based verification.

SCMP reported that progress has been slow for X-Epic and other Chinese EDA companies, which saw their business dip in the first quarter, with the biggest Chinese firm – Empyrean – seeing first quarter profits drop 64% year over year.

PCD&F

Apple has been granted a US patent for a thin system-in-package with a shielded stepped mold.

BH will supply flexible printed circuit boards for the camera modules on **Samsung's** new foldable phones.

Gold Circuit Electronics (Thailand) Co. in July signed a deal to acquire **CESE 2 (Thailand) Co.** for approximately NT$1.14 billion ($35 million).

LQDX signed a sales and licensing agreement with **Kansai Denshi Industries** for manufacturing support for next generation IC substrates and advanced PCBs.

Mobenus will move to Namchun Industrial Complex in South Korea by next year to mass-produce high-density circuit boards.

Schweizer Electronic and **Zollner Elektronik** announced a strategic partnership to increase efficiency in power embedding.

Soochow University researchers unveiled a new method to fabricate stretchable and breathable electronics.

Ventec announced its membership in the **Printed Circuit Board Association of America.**

Victory Giant has completed registration for the construction of a printed circuit board manufacturing campus in Vietnam, with a planned investment of up to $260 million.

CA

Absolute EMS partnered with **Ventiva** to build its Ionic Cooling Engine (ICE).

Benchmark is constructing a new state-of-the-art facility in Guadalajara.

Bentec appointed **Accurex Solutions** manufacturers' representative for its Prey UPI in India.

BTU International appointed **E-Tronix** representative in North Dakota, South Dakota, Minnesota, Iowa, Wisconsin and Illinois.

CAES was awarded a $172 million contract from **Raytheon** to support a major international missile program.

CE3S will distribute **Nitto Kohki** Delvo electric screwdrivers.

Circuitronics installed a **VJ Electronix** XQuik II Plus component counter.

Continental AG's 10,000 sq. m. factory expansion in China's Jingyue High-tech Industrial Development Zone has commenced operations.

Critical Manufacturing announced a strategic alliance to combine **Loftware's** labeling expertise with its advanced MES technology.

Dixon Technologies' acquisition of up to 56% of shares in **Ismartu India Private Ltd.** was approved by the Competition Commission of India.

Ducommun rejected an unsolicited revised nonbinding indication of interest from **Albion River**, a private direct investment firm, to acquire all outstanding shares of Ducommun for $65 per share in cash.

ECM PCB Stator Tech and **East West Manufacturing** announced a strategic partnership to advance adoption of next-generation products incorporating PCB Stator electric motors.

EVS International appointed **Circuit Assembly Products SE** representative for the southern states of Alabama, Georgia, Tennessee and Mississippi.

FANUC America opened a new 650,000 sq. ft. robotics and automation facility in Auburn Hills, MI.

Gen3 shipped a next-generation CM Contaminometer process ionic contamination tester to an aerospace company in Germany.

Genie Electronics implemented **Cetec ERP's** enterprise resource planning system.

Heraeus Electronics announced its participation in the FastLane project, an EU-funded initiative aimed at revolutionizing the European value chain for Silicon Carbide (SiC)-based power electronics.

Incap UK upgraded its SMT line with the installation of an automated solder screen printer and two advanced placement modules.

Inspectis appointed **Microscope World** as its North American distributor.

Kasdon Holdings, a UK-based PCB assembler, is planning an expansion at its facility in Wolverhampton.

Kitron received orders with a value of over NOK500 million ($46.8 million) from **Kongsberg Defence & Aerospace.**

Latham Industries expanded to a new 10,000 sq. ft. facility in Phoenix, AZ.

Luminovo and **Sluicebox** have formed a partnership to offer product carbon footprint estimates for an actual printed circuit board assembly.

Murray Percival Company will represent **Bimos** ESD seating solutions in Michigan, Ohio, Indiana, Kentucky, Illinois, Wisconsin, and Western Pennsylvania.

Myotek will receive nearly $1 million from Michigan's Going PRO Talent Fund for training its electronics assembly employees.

Nordes EMS installed a **PBT Works** SuperSwash III PCBA cleaning system at its technology park in Wroclaw, Poland.

OVES Enterprise announced that it is expanding its services to include printed circuit board production.

Pemtron appointed **American Tec** its official distributor in China.

Samsung Electronics' wafer foundry in Taylor, TX, initially set to mass-produce 4nm process technology, is reportedly shifting to 2nm processes.

Scanfil will produce electronic assemblies for **Framery** at its factory in Pärnu, Estonia.

StenTech appointed **SilvaCo** sales representative for Southern California, Southern Nevada and Arizona.

Surface Technology International has secured a £11.3 million ($12.3 million), three-year asset-based lending facility for its EMS operations in the UK.

TRI opened a new office in Guadalajara, Mexico.

TVS Electronics launched electronics manufacturing services capability at its facility in Tumakuru, India.

Vicor secured a preliminary injunction that prevents **Foxconn** subsidiaries from pursuing arbitration against Vicor before the **China International Economic and Trade Arbitration Commission.** Foxconn had sought rulings from CIETAC based on fine print in its purchase orders, claiming entitlement to royalty-free licenses for Vicor's patents.

ViTrox Americas opened a new demo lab in Hutto, TX, and named **Hitronix Technology** sales channel partner in Brazil.

PCDF


Dan Lisowsky


Jon DeGaynor


Benjamin Smith


Bobby Johnson


Marc Ladle

George Dudnikov, an engineer and executive with Hadco, Sanmina and most recently Victory Giant, passed away in June at 66.

ECM PCB Stator appointed **Dan Lisowsky** vice president of its Industrial OEM Products Division.

Methode Electronics appointed **Jon DeGaynor** president and CEO.

SelecTech appointed **Benjamin Smith** director of sales.

Summit Interconnect appointed **Bobby Johnson** vice president of human resources.

Ventec Giga Solutions appointed **Marc Ladle** project manager.

CA

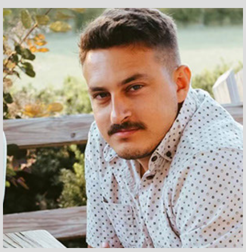

Scott Law


Brett Petrill


Rebeca Obregon-Jimenez


Florin Larion


Craig Gates


Devon Piatkin


Brian Porter


Robert Graumann


Anette Mullis

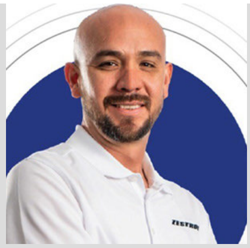

Eliud Herrera

Mark Sullivan

Jake Kurth

Shuichi Imai

Johannes Vorwerk



Daniel Chacon

ACDi appointed **Scott Law** plant manager.

Amtech appointed **Brett Petrill** director of operations.

Farnell named **Rebeca Obregon-Jimenez** president.

Intervala appointed **Florin Larion** director of operational excellence.

Keytronic signed a consulting arrangement with **Craig Gates**, who retired as president and CEO on June 30.

MaRC Technologies added **Devon Piatkin** to its sales team to cover the Oregon and SW Washington territory.

Niche Electronics appointed **Brian Porter** to sales/business development.

Parmi USA named **Juan Arango** head of SMT and business development.

Pro-Active Engineering appointed **Robert Graumann** CEO.

Scanfil appointed **Anette Mullis** chief people officer.

Selectronic Assembly appointed **Eliud Herrera** production manager.

Tagarno appointed **Mark Sullivan** sales representative for the American market and promoted **Jake Kurth** to vice president of technical sales.

Yamaha Robotics SMT Section appointed **Shuichi Imai** sales general manager for Europe and **Johannes Vorwerk** sales manager for Germany.

Zestron appointed **Daniel Chacon** to its LATAM sales team.

PCB West Keynote to Offer Path for Implementing AI in Electronic Design

Charles Pfeil

PEACHTREE CITY, GA – Electronics design software architect Charles Pfeil will keynote the PCB West conference and exhibition in October, PCEA announced.

Pfeil's talk, "AI Roadmap for PCB Layout: New Opportunities for PCB Designers," illustrates the capabilities and methodologies that will be needed for AI success with complex PCB designs, with a focus on constraints, place and route. Success, he says, lies in balancing the growing capabilities of AI with the preservation of proprietary design knowledge and methodologies. His talk will take place Oct. 9 at 11 a.m. at the Santa Clara (CA) Convention Center.

"AI functionality for effective place and route of simple designs is currently available; however, using AI for place and route of complex designs is still in the first stages. But the ultimate goal of AI-driven PCB design tools is to replicate and potentially surpass the capabilities of experienced human designers, particularly for complex designs," Pfeil notes.

"Communicating PCB design expertise and decisions to an AI program involves several steps, including knowledge extraction, representation, and the development of an appropriate system to leverage this knowledge. But significant challenges remain, particularly in capturing and applying the nuanced expertise of experienced designers and the unique methodologies of different companies."

Over his 50-year career in the PCB industry, Pfeil has been a designer, owner of a service bureau, and had engineering management and product definition roles at Racal-Redac, ASI, Cadence, PADS, VeriBest, Mentor Graphics, and Altium. Most of his career was at Mentor Graphics, where he was a software architect focused on the advanced development of PCB design tools. He was the original product architect of Expedition PCB, and an inventor of Team PCB, XtremePCB, XtremeAR, and the Sketch Router. He is also the author of *BGA Breakouts and Routing* and *High-Speed Constraint Values and PCB Layout Methods.*

"Charles Pfeil is synonymous with electronics design automation," said Mike Buetow, president of PCEA and conference chairman, PCB West 2024. "He has guided the development of two platforms in wide use today, and his presentation focuses on both the fundamental issues that are obstacles to AI implementation for complex designs, and the viable paths forward. Those interested in knowing how AI might change their jobs in a positive manner should not miss this keynote address."

PCB West is the leading technical conference and exhibition in North America for electronics design, fabrication, assembly, and test engineers. It will take place Oct. 8-11 at the Santa Clara (CA) Convention Center. The event includes a one-day exhibition on Oct. 9.

Susy Webb to Present Webinar on PCB Design Techniques

PEACHTREE CITY, GA – Susy Webb will present a new all-day webinar titled "Effective PCB Design Techniques" on Sept. 10, sponsored by PCEA.


Susy Webb

Webb, a senior designer with 40 years of experience across a variety of industry sectors, will share ideas for setup, planning, and procedures that lead to efficient board design.

The all-day online class will take place from 9 a.m. to 6 p.m. EST, with a one-hour break from 12 p.m. to 1 p.m. EST.

"Years ago, someone new to designing printed circuit boards would have had access to other designers who worked with them in the same office, or what we often call 'mentors,'" says Webb. "Today's lack of dedicated designers, coupled with the trend toward electrical engineers designing their own boards – some quite infrequently – has created a gap in understanding of the entire design processes. This includes a lack of knowledge of the reasons why certain things need to be done, and how to implement those ideas within each new PCB design."

This class was designed to discuss many of the tasks that are needed to create an effective PCB, including choosing quality parts, finding the rise time of signals, understanding where the energy flows within a PCB, deciding the orientation of parts, setting up routing and fanout grids, and the routing layers to use for different signals and busses.

Also discussed will be procedures that seasoned designers use to make their design work efficient.

For more details: https://attendee.gotowebinar.com/register/4213104697774443360.

2024-25 Board of Directors Ballots Out

PEACHTREE CITY, GA – The ballot for the next PCEA board of directors term is available to members for voting. All PCEA members in good standing are eligible to vote for the board of directors.

In accordance with PCEA Bylaws, section 9.3, directors shall be elected at the annual meeting of the membership for two-year terms. In accordance with the association bylaws, candidates for the board of directors were solicited from the active membership earlier this year, and the PCEA Board's Nominating Task Force has nominated candidates for each director position.

PCEA members may vote for up to 12 individuals for the board. The following link may be used to access the ballot: https://www.surveymonkey.com/r/pceaboard2024. The deadline for voting is Aug. 2.

To see the candidates' biographies, please click here.

The new board will be formally announced at the next PCEA annual meeting, scheduled for Oct. 8 from 9 to 10 a.m. at the Santa Clara (CA) Convention Center, in conjunction with PCB West.

PCEA CURRENT EVENTS

CHAPTER NEWS

General. The PCEA annual meeting will be held Oct. 8 from 9 to 10 a.m. during PCB West at the Santa Clara (CA) Convention Center. An update on all association programs, plus the annual awards, will be presented.

Abstracts for next year's PCB East technical conference are due Sept. 6. The conference, the largest of its kind in New England, will take place April 29 to May 2, 2025, in Boxborough, MA. The event includes a one-day exhibition on April 30. Submit abstracts here.

Portland, OR. The chapter will meet on Aug. 15 for a webinar on reverse engineering PCBs featuring Bill Loving of ScanCad International. Contact Stephen Schmidt at stschmidt@pcea.net for details.

Richmond, VA. We have interest in starting a local chapter here. Those interested should email mike@pcea.net.

San Diego. Our next meeting is scheduled for Aug. 20 from 11:30 a.m. to 1:30 p.m. The scheduled speaker is Mark Finstad, director of engineering at Flexible Circuits Technologies. Contact Luke Hausherr at lucashausherr@gmail.com for details.


MACROFAB
Fast-Track Your PCB and PCBA Prototyping
Quick Turnaround, High Quality, Assembled in USA
LEARN MORE

Dips and Dumps: Electronics Industry Wary of Demand, China

BANNOCKBURN, IL – Sentiment among electronics manufacturers fell in June but remained above its long-term average, according to IPC's monthly supply chain report. Respondents noted concerns over possible dumping practices by Chinese companies.

The industry outlook remains strongly positive, though it has slightly softened over the two months from a record high in March.

The results are based upon the findings of an IPC survey conducted between May 15 and May 31.

Some 68% of manufacturers expressed "somewhat" or "extreme" concern over the possibility of Chinese manufacturers dumping components including PCBs at artificially low prices over the next year to stimulate its economy. Twenty-nine percent of respondents indicated dumping concerns from manufacturers operating in Asia Pacific.

Regarding outlook for the next six months, electronics manufacturers expect both labor and material costs to come down slightly, while material costs are expected to remain relatively stable. Although backlogs and ease of recruitment are likely to remain challenging, manufacturers expect profit margins and capacity utilization to rise.

The survey revealed industry demand fell to its lowest level this year, with demand fundamentals falling for the third consecutive month, down 3.1% to the lowest level in 2024. The Shipments index fell one point, while the New Orders and Backlog indices each dropped two points. Capacity Utilization fell eight points and dropped into contractionary territory.

The Labor Costs index dropped two points but the Material Costs index increased six points, the latter to its highest level since August 2023.

Making Memories

Trends in the US electronics equipment market (shipments only)

	% CHANGE			
	MAR.	APR.[r]	MAY[p]	YTD
Computers and electronics products	-0.2	0.0	-0.1	1.6
Computers	-0.1	3.8	-0.1	25.5
Storage devices	-5.1	1.7	0.7	8.0
Other peripheral equipment	-5.8	5.4	0.6	6.2
Nondefense communications equipment	0.6	0.0	-2.4	-0.1
Defense communications equipment	3.6	1.5	1.5	-2.4
A/V equipment	1.6	-7.1	-3.3	21.8
Components[1]	-1.1	1.0	1.5	7.1
Nondefense search and navigation equipment	2.3	2.2	-0.8	2.2
Defense search and navigation equipment	0.6	0.4	0.7	6.0
Electromedical, measurement and control	0.2	-0.3	-0.5	-0.8

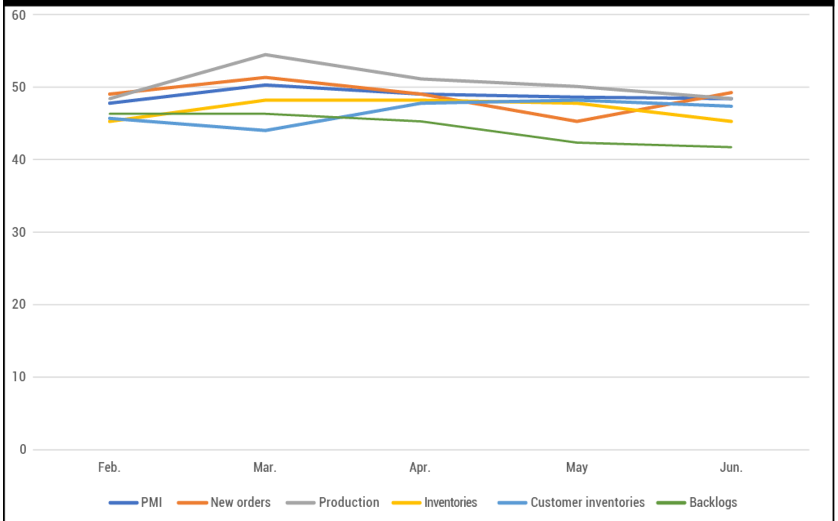
[r]Revised. [p]Preliminary. [1]Includes semiconductors. Seasonally adjusted.
Source: US Department of Commerce Census Bureau, July 3, 2024

Key Components

	FEB.	MAR.	APR.	MAY	JUN.
EMS book-to-bill[1,3]	1.22	1.31	1.42	1.36	1.32
Semiconductors[2,3]	16.3%	15.2%	15.8%	19.3%	18.3%
PCB book-to-bill[1,3]	1.07	1.13	1.06	1.06	0.95
Component sales sentiment[4]	100.8%	106.9%	124.1%	112.3%	98.9%

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

US Manufacturing Indices



Source: Institute for Supply Management, July 1, 2024

Hot Takes

Sales of **PCB and MCM design software** rose 2.8% year-over-year in the first quarter to $378.9 million. The four-quarter moving average, which compares the most recent four quarters to the prior four, for PCB and MCM rose

13.2% (ESD Alliance).

AI server shipments in the second quarter will increase by nearly 20% sequentially, and the annual shipment forecast has increased to 1.67 million units, representing 41.5% growth year-over-year. (TrendForce)

Second quarter **PC shipments** rose 1.9% from a year ago to 60.6 million units, the third consecutive quarter of year-over-year growth. (Gartner)

Global **semiconductor manufacturing equipment sales** by OEMs are forecast to set a record, reaching $109 billion in 2024, growing 3.4% year-on-year. (SEMI)

Traditional **PC shipments** reached 64.9 million units in the second quarter, up 3% year-over-year growth. Excluding China, worldwide shipments grew more than 5%. (IDC)

Global **semiconductor sales** hit $49.1 billion during May, an increase of 19.3% year-over-year and 4.1% sequentially. (SIA)

The **Indian EMS industry** is expected to touch almost $300 billion by 2026-27. (Dixon Technologies)

The **electronics industry in India** has urged its government to consider reducing tariffs on inputs in the upcoming budget to bolster local manufacturing and enhance competitiveness against China and Vietnam. (India Cellular and Electronics Association)

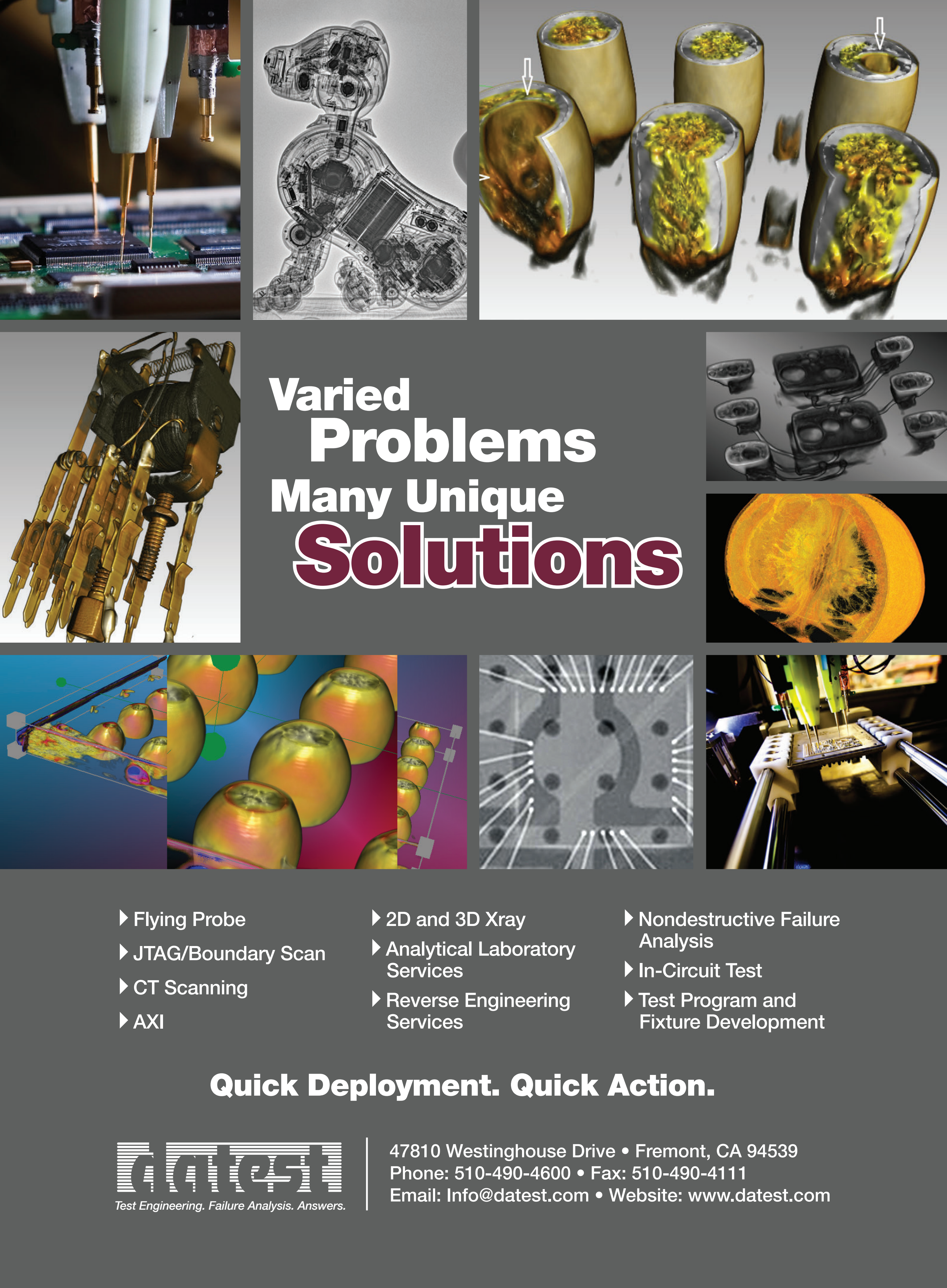
Global **smartphone shipments** increased 6.5% year-over-year to 285.4 million units in the second, the fourth consecutive quarter of shipment growth. (IDC)

Worldwide **IT spending** is expected to total $5.26 trillion in 2024, up 7.5% from 2023. (Gartner)

Total **copper-clad laminate sales** in 2023 were $13 billion, including prepreg but excluding mass laminate – down 16% from 2022. Some 650 million sq. m. of rigid laminate materials were produced, including paper, composite, FR-4, and specialty laminates/other, a slight decrease from 2022. (Prismark Partners)

North American PCB bookings in June were down 2.7% year-over-year and up 3.2% sequentially. PCB shipments were up 1% over last year and 9.1% sequentially. (IPC)

North American EMS shipments in June fell 2.4% from last year and 3.3% sequentially. Bookings dropped 3.5% and 2.8% respectfully. (IPC)


Varied
Problems
Many Unique
Solutions
Flying Probe
JTAG/Boundary Scan
CT Scanning
AXI
2D and 3D Xray
Analytical Laboratory Services
Reverse Engineering Services
Nondestructive Failure Analysis
In-Circuit Test
Test Program and Fixture Development
Quick Deployment. Quick Action.
datest
Test Engineering. Failure Analysis. Answers.
47810 Westinghouse Drive • Fremont, CA 94539
Phone: 510-490-4600 • Fax: 510-490-4111
Email: Info@datest.com • Website: www.datest.com

The 3 'Cs' of Success

The next generation of workers needs to understand how to be successful.

WHAT WE DESPERATELY need, or more specifically, what the West desperately needs is a generation of workers who understand and believe in the three critical "Cs" – commitment, concern and consistency.

The next generation – or generations – entering the workforce by and large offer impressive credentials. Tech savvy is one such attribute. It is especially important for an industry like ours to have employees who understand the technologies the industry produces. The current generation has also demonstrated an uncanny ability to multitask while using technology, which bodes well for the factory floor needs of the future. Regrettably, that is where the impressive credentials all too often end.

Alas, the next generation of shop floor employees too often demonstrates some not-so-impressive traits. Among these are poor communication skills – both writing and verbal – lackadaisical adherence to work schedule demands, and a lack of interest in what is happening up and down the process flow. Regrettably, these unimpressive habits transcend their impressive skills.

What is missing in this equation? The three personal traits that historically have separated the successful from the not so: commitment, concern and consistency!

Commitment should be the easiest trait to find, yet it rarely is. Commitment covers many facets of daily life, from showing up to work on time, each day, every day, regardless of external issues that get in the way. Everyone wakes up sometimes feeling tired or achy from an activity they did over the weekend. We also, from time to time, have an activity that keeps us out later than normal, but seasoned employees know that when the bell rings, they must be onsite and ready to work. Legitimate sick days are understandable, but too many next-generation employees call in "sick" when they just don't feel like working.

Commitment is knowing that to progress, to earn more, to get promoted – to have a solid career – takes sucking it up and demonstrating your commitment to do the *best* you can, day in and day out. Commitment is looking yourself in the mirror and knowing you are taking the steps, no matter how challenging, to better yourself.

As important as commitment is, so is concern – concern that what you are doing is helping make a better product (or not making defective product) and concern that you are respectfully communicating to those around you in a way they understand.

Too many next-generation employees do the bare-bones minimum to complete their task, without caring whether the task has been done correctly. These workers do not understand their marginal quality negatively impacts the entire process flow. Concern is caring about how well you do your job, and ensuring you are not causing problems for those who next touch the product, or ultimately, the customer. Concern is doing your job well while taking a 360° look around to make sure what you do is not causing problems for others. Concern is showing interest in not just your job, but the job your coworkers are doing, so together, high-quality product is delivered to the customer!

Having commitment and concern is fine, but only if both are consistently deployed.

Consistency may be the most difficult for younger people. Consistency embodies commitment and concern, but it adds one other characteristic: responsibility. Consistency is doing it right the first time, every time. Consistency is getting a quality job done even when you "just don't feel like it." Consistency is like being that home run hitter who works tirelessly at hitting just because they want to be consistently good.

Commitment, concern and consistency are what separate those who succeed from those who founder, regardless of vocation, industry or location. These three traits are how the West became a manufacturing and economic powerhouse. Likewise, employees with these traits are also why Asia has become an economic powerhouse today.

It is imperative that we in the West follow the lead of other global powers and engage with the next generation, explaining what it takes for success in the manufacturing environment. Perhaps more important, teachers, be they in vocational or traditional high schools, must understand that manufacturing can be a phenomenal career path, especially for those who have the commitment, concern and consistency to succeed.

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

WHO'S EXHIBITING

AdvancedPCB
AGC Multi Material America, Inc.
All Flex Solutions
Allfavor Technology
AllSpice.io
Altair
Ameliorate Electronics Solutions LLC
Arlon EMD Specialty Materials
ASC Sunstone Circuits
Bay Area Circuits
Best Technology Co., Limited
Bittele Electronics Inc.
Boardera Software Inc.
Briocean Technology Co., Ltd.
Cadence Design Systems
Cicor Group
Cofactr
Depthtech
Direct PCB
DownStream Technologies
DYCONEX AG
Element Materials Technology
Elephantech Inc.
Elgris Technologies, Inc.
EMA Design Automation
EMSolutions Inc.
ESTECO
Fischer Technology, Inc.
Flexible Circuit Technologies
Founder PCB USA
Freedom CAD Services
FTG Corporation
Fujipoly America Corp.
Glory Faith Electronics (Suzhou) Co., Ltd.
GS Swiss PCB
Haixiang Electronics (HK) Limited
Hardware AI
Hitek Technology (HK) Co., Ltd.
HZO
IBE Electronics Inc.
Imagineering, Inc.
Insight Electronics Group Inc.
InstaDeep
IPC-2581 Consortium
Ironwood Electronics
Isola USA Corp.
JETPCB USA
JITX
Jove PCB Enterprise Limited
KiCAD EDA
Kinwong Electronics
Leader Tech Inc.
MaRCTech2, Inc.
MFS Technology
MVinix Corporation
NCAB Group USA
Newgrange Design
Notion Systems GmbH
Oak-Mitsui Technologies
OKI Circuit Technology Co. Ltd.
Optiprint
PalPilot International Corp.
Panasonic Electronic Materials
PCB Technologies, Inc.
PCBWay
PFC Flexible Circuits Ltd.
Pinsheng Electronics Co., Ltd.
Polar Instruments, Inc.
Printed Circuit Board Association of America (PCBAA)
Printed Circuit Engineering Association (PCEA)
Quantic Ohmega – Ticer
Risho/Sanko-Giken
Rogers Corporation
San-ei Kagaku Co. Ltd.
Schmid Systems Inc.
Screaming Circuits
SEP Co., Ltd.
Siemens
Sierra Circuits
SiliconExpert
Sourceability
STARTEAM Global
Summit Interconnect
Sundell Technology Company Ltd.
Sunshine Global Circuits
Suntech Circuits Inc.
Surface Mount Technology Association (SMTA)
SVtronics
Trilogy-Net
Trylene
Ultra Librarian
Vayo Technology
Ventec International Group
Victory Giant Technology
Winsource Electronics
XDry Corporation
Zhen Ding Tech Group
Zuken, USA, Inc.

Engineering *Tomorrow's* Electronics

pcbwest.com

Component Obsolescence Management

Ways to avoid redesign due to discontinued parts.

ELECTRONICS FOUND INSIDE infrastructure items such as buses, trains, signage, control units – anything that requires a printed circuit board assembly – are usually expected to have a long lifespan of continuous operation.

Each of those electronic systems and their corresponding subsystems can be affected by component obsolescence, however, especially when the components placed into them today have a purchase availability of only about 10 years.

This is because changes in design, improved component speed and efficiency, a drop in demand, etc. lead to component replacement and can make older components obsolete.

That's why component obsolescence management is so important for OEMs.

Yet many governments and other larger customers of OEMs are not thinking about obsolescence. They assume the electronics they purchase will last more than 10 years. But OEMs must plan for the electronic components in their products today to be discontinued within five to 10 years of the original design.

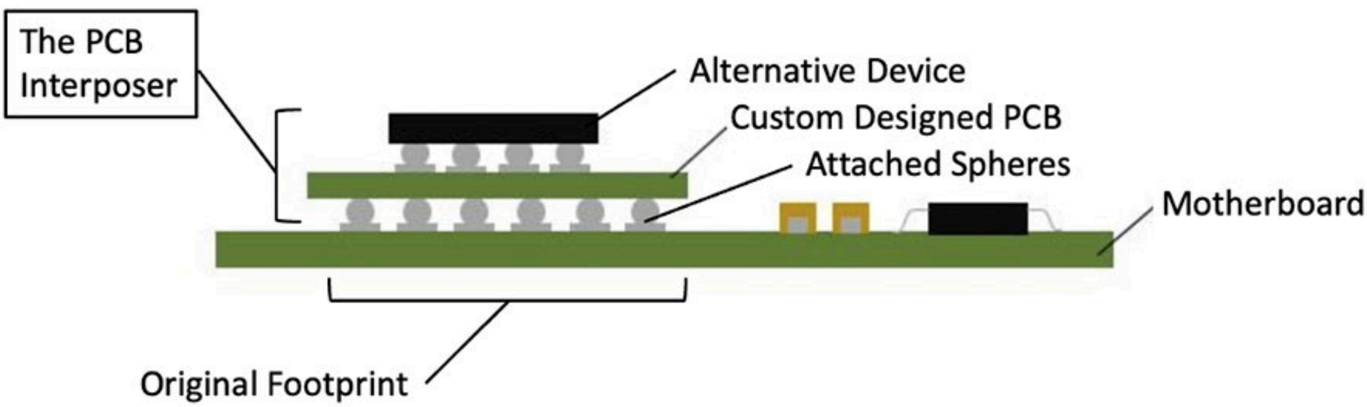


Figure 1. A diagram of a PCB interposer, or daughterboard.

How do they do that? Through careful obsolescence management.

Component obsolescence management is a newer discipline, requiring a strategic approach that identifies which parts of every product may become obsolete. And it involves continual monitoring of the availability of key components.

When an OEM becomes aware that a component manufacturer is going to declare a particular device obsolete, it must either make a lifetime buy or redesign the part.

Many OEMs make a lifetime buy, as redesigning and revalidating a particular product can be very expensive. A good plan involves significant amounts of electronic components in stock to cover its production and service needs. The smarter plan involves managing that inventory cost through long-term maintenance contracts.

A redesign of product might be necessary if a lifetime buy of a component that is about to become obsolete is not available. A costly redesign at the system level may not be needed, however, if the redesign can be performed at the component level with a PCB interposer or adapter.

An interposer provides a flexible, cost-effective, and efficient solution to the challenge of semiconductor obsolescence or unavailability.

When a semiconductor component becomes obsolete or has a lead time beyond 52 weeks, finding a direct replacement that fits the existing system's footprint can be challenging.

Figure 2. PCB interposers can provide a solution to semiconductor obsolescence or unavailability.

An interposer (also known as a daughterboard) is a small electronic assembly that has a custom-designed PCB with components that mimic the function(s) of the original IC. That PCB contains the I/O interconnect that emulates, or is an upgrade of, the performance of the original IC.

More importantly, it is manufactured with the same footprint as the original IC, can be shipped in tape-and-reel or in a JEDEC tray, and attaches using conventional SMT assembly processes.

The PCB interposer acts as a bridge between the new component and the old system, permitting integration of modern semiconductors into legacy systems without the need for extensive redesign.

By permitting the integration of newer semiconductor technologies into older systems, adapters and interposers ensure these systems continue to function and remain relevant, despite the original components being phased out or not immediately available.

Implementing an adapter is typically faster than undertaking a full system redesign. This speed is crucial for industries where time to market and system uptime are critical factors, as interposers eliminate the need for extensive technical re-evaluation and redesign, simplifying the upgrade process.

Interposers also offer PCB buyers and engineers more options. They are often more economical than redesigning the entire system or product to accommodate new semiconductor components. This approach saves time and money.

Custom-made interposers also prevent costly last-minute or risky gray-market buys.

Obsolescence is a risk – a risk most customers don't understand or consider as part of their quoting process. OEMs don't have that luxury. Obsolescence management is key to both product longevity and customer satisfaction.

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); greg@directpcb.com.


ENGINEERING TOMORROW'S ELECTRONICS
PCB
EAST 2025
Conference & Exhibition
Conference: April 29 – May 2
Exhibition: April 30
Boxboro Regency Hotel & Conference Center, MA
2025
Information coming soon at pcbeast.com

Meeting the Challenge of EMS Workforce Shortages

Employee recruitment and retention require an active company effort.

IT'S NO SECRET that a tight labor market is an issue in all segments of the electronics manufacturing services (EMS) industry. A large part of this is the result of manufacturing offshoring trends. When I entered the workforce, I had friends and relatives who worked or had worked in manufacturing-related careers, influencing my choice to work for an EMS company. That isn't the case today.

A few years ago, I participated in a local manufacturing awareness day targeted at high school students participating in their schools' robotics programs. I put together a short video that illustrated manufacturing processes and careers in the EMS industry. Even students interested in engineering careers weren't thinking about manufacturing engineering or hardware engineering. Most were focused on software engineering with an eye on getting jobs at Google or some other highly visible tech employer.

This underscores the value of educational outreach programs at the high school and community college level, when young people are making career decisions. Similarly, this also underscores the need for considering the lack of workplace familiarity that entry-level employees are likely to have in their first manufacturing job. One of my clients, Electronic Design & Manufacturing, is conducting that outreach and education successfully in its community, and I thought it might be a good example to share.

The team at EDM has been working with Central Virginia Community College (CVCC) on paid college-level internships for over a decade. In 2022, the EMS provider also began offering unpaid high school internships to students from the mechatronics program at the Lynchburg Regional Governor's XLR8 STEM Academy at CVCC. High school students receive educational credit for time spent in the internship and are dual enrolled in high school and CVCC programs. Under terms of the partnership, they are not allowed to work on direct manufacturing tasks but do get exposure to the skills and technical education required for a range of positions.

As an example, one semester's interns worked with the test engineering team developing test fixtures. They focused on test capability enhancement projects that had lower priorities for adoption due to resource limitations. As part of their initial assignment, they optimized the connection methodology to enable the fixture to automatically connect with pogo pins. Their assignment included using Fusion360 to create a template for 3-D printing of required parts. Their work on the first fixture cut test time in half. They were assigned to more advanced projects for the rest of their internship.

These projects help train the interns on a range of engineering tools, while providing lessons learned (and course credit) on the engineering strategies they explore. At the same time, their work enables more enhancement projects to be completed, benefiting this EMS provider and its customers. As an added benefit, because the company is 100% employee-owned, the students also get introduced to the benefits of the employee owner model vs. more typical employment options.

This EMS provider also offers paid internships to local university students from time to time. The most recent intern has completed a degree in electrical engineering and is now working full-time at the company as a sales engineer.

The issue of new employee unfamiliarity with a manufacturing environment is also well addressed. This EMS provider has a robust 90-day onboarding program for entry-level employees, utilizing a staffing agency for initial recruitment and offering full-time positions to employees successfully completing the 90-day onboarding period. In the first month, training includes use of IPC's Electronics Assembly for Operators program. The company's facility includes a Center for Excellence training center which includes all the equipment needed to train employees on hand tools, soldering stations and ESD-safe mechanical assembly, plus online access to instruction via IPC's platform and remote instructors.

Training is divided into manageable segments, so employees spend the balance of their time applying what they are learning. The remaining two months are focused on guided skills training for whatever role the employee has been hired into. Oversight is reduced in the last month as employees become proficient in their skills. Full-time employees receive continuing skills training based on a comprehensive skills matrix. Employee competencies and completion of required courses for their assigned roles are tracked by date and recertification or refresher training may be scheduled as required.

While these examples are not unique in the EMS industry, they illustrate best practices in dealing with the challenges of a tight labor market. Educational partnerships, effective onboarding programs and continuing education processes are critical in attracting and retaining a skilled workforce within the EMS industry.

SUSAN MUCHA is president of Powell-Mucha Consulting Inc. (powell-muchaconsulting.com), a consulting firm providing strategic planning, training and market positioning support to EMS companies and author of *Find It. Book It. Grow It. A Robust Process for Account Acquisition in Electronics Manufacturing Services.* She can be reached at smucha@powell-muchaconsulting.com.

Designing for Board Thickness

Small differences can have big consequences.

FOR AS LONG as there have been printed circuit boards, the nominal thickness seems to have been set at 0.062" – or in Latin, 1.5748mm, but call it 1.6mm for short. In practical terms, the standard dielectric materials available support this board thickness while providing anything from two to 20 layers. I imagine four layers is still the most common use case.

Larger boards will need more connections and require more stiffness. To manage connectivity and flatness requirements, standard PCB thickness targets ratchet up to 2.4mm and 3.2mm. On the low side we find 1.0mm and go down to 0.8mm. All these targets are related to using so-called gold fingers as a printed edge connector.

It's about connectivity and solderability. This was handed down from the backplane and daughtercard configurations found in our tower computer systems. The motherboard has expansion sockets and the memory cards come with fingers to plug and play. As a result of this variety, many connector vendors that market to plated through-hole technology users will offer different pin lengths that fit the range of board thickness options.

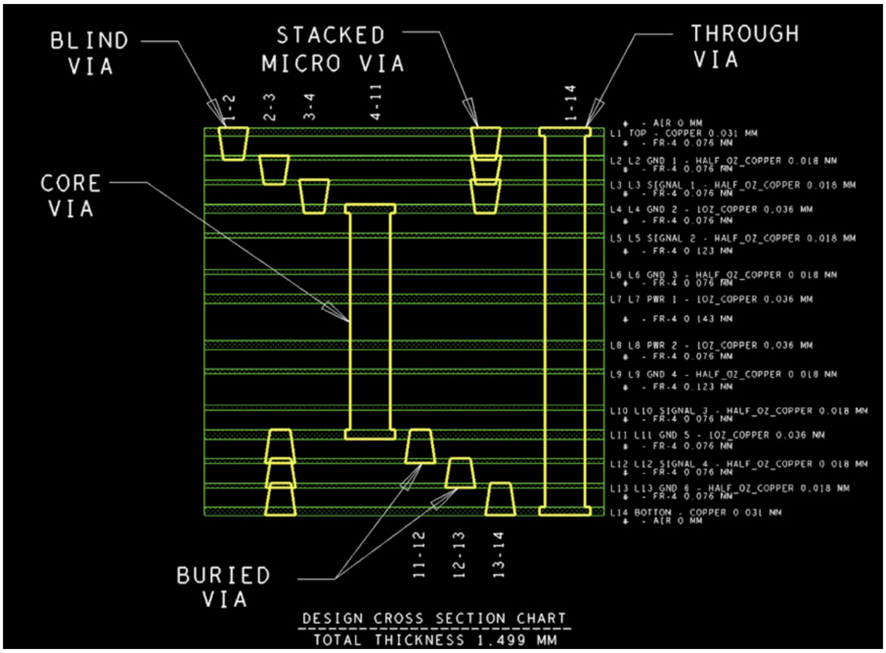


Figure 1. A 14-layer board where the stack-up would be designated as a 3-N-3 as a general case to describe the number of lamination sequences and a 3-8-3 for a more specific case to designate the total number of layers.

One edge connector vendor has an offering that accommodates boards between 1.37mm and 1.8mm. You have to imagine some flexibility is due when the PCB thickness variance is considered. This is probably this column's most important takeaway. The construction of the board with all the material and process variables gives rise to a +/-10% process window in terms of PCB thickness.

Tolerance stacks up quickly. Doing the math for a standard 1.6mm board gives us 1.6mm minus 0.16mm, or 1.44mm on the thin side and up to 1.76mm on the high end. The broad thickness range for the connector mentioned above is just right for this application. The problem is not all the mechanical engineering world is fully aware of this limitation – not by a long shot. While thinner boards tighten up the process, thick ones need more than the standard allotment for the Z-axis.

Case study: Two possible thermal solutions; one winner. We had a QFN component that ran really hot and would fail. It had a good-sized ground paddle in the middle where the plugged vias were not up to the task. The mechanical designer decided to cut a slot in the board under the ground paddle. Then he would position the board over a pedestal that passed through the slot. He wanted a very thin layer of thermal grease on the top of the pedestal rather than a thermally conductive pad.

He also wanted a limit on the board thickness of +/-0.10mm rather than the +/-0.16mm I was calling out. I said he couldn't have a tight tolerance on the stackup. The height of the pedestal was going to be predicated on hemming in the PCB fabrication process. You can do that with an aluminum billet but not with a cured strata of copper, glass and resin. We taped out a board with his specs and let the chips fall where they may. Of course, there was DfM pushback on the thickness callout.

We also did a parallel project in which I added DuPont CB100 thermal fill to all the ground paddle vias; not just the ones under the problem child but anywhere they would put a check on thermal issues. Finally, the day came when the two boards faced off and measurements were taken.

This was a case where the EE effort won out over the ME version by a good margin. The thermal vias were not affected by variations in PCB thickness. They were a bit of a cost concern, but the project moved forward with the thermal problem solved. The moral of that story is that we can't put tight controls on multilayer PCBs. If you want to be in the game, you must be the bearer of this cold, hard fact.

Is it "in to be thin?" There is a lot to be said for thinner boards. They have a singular attribute where they can be depaneled with lasers rather than traditional milling or V-score. As far as I know, the maximum thickness for the laser treatment is 0.5mm. Makers of laser ablating machines don't brag about their maximum cutting depth.

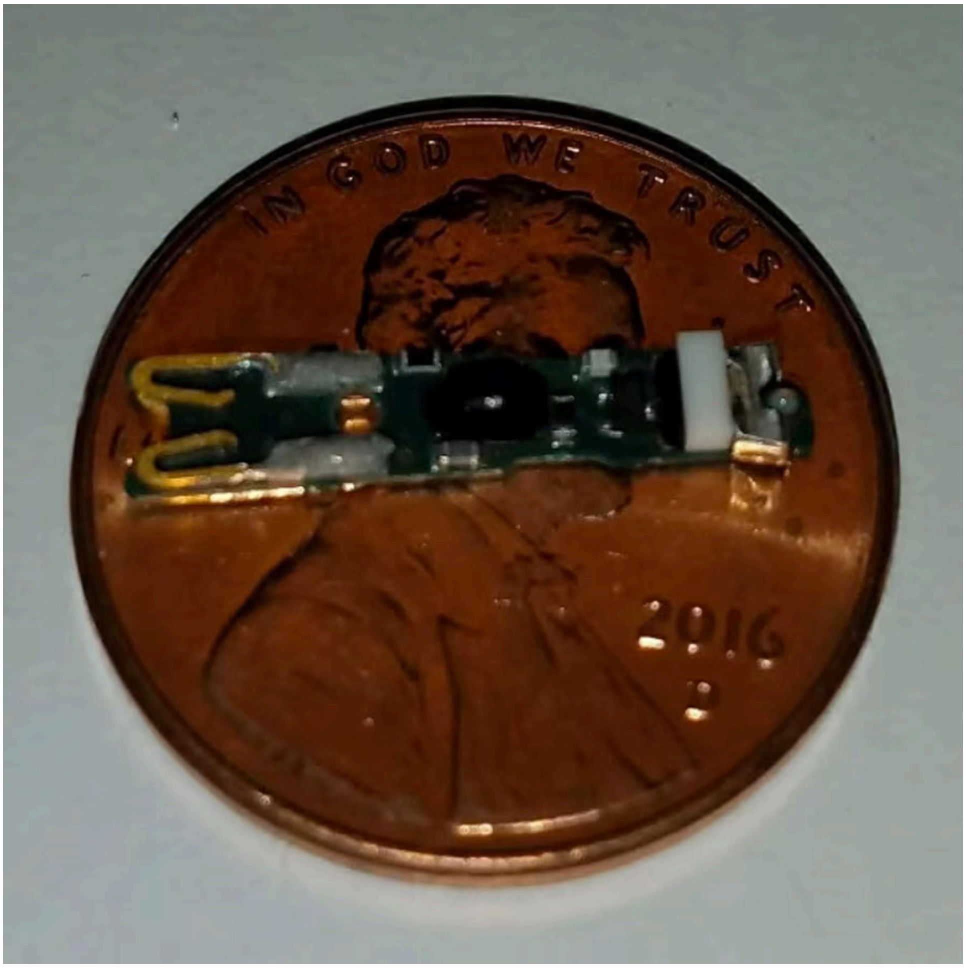


Figure 2. This must be the smallest PCB I've ever met. Let's just say that some rules were bent in the making of this board.

What I learned about it is from working with a specialty PCB vendor that focused on optical solutions. It seems to be about the number of passes to cut through the dielectric materials and any metal that may be present. One of the bugs encountered in laser depanelization is when the laser scorches the board edge. It's called carbonization (along with some other not-so-nice words).

One nice thing you can do for these thinner boards is provide sufficient mounting hardware locations so that board flexing does not stress the solder joints. In this case, three generous slots provide mounting locations for these quirky printed circuit boards.

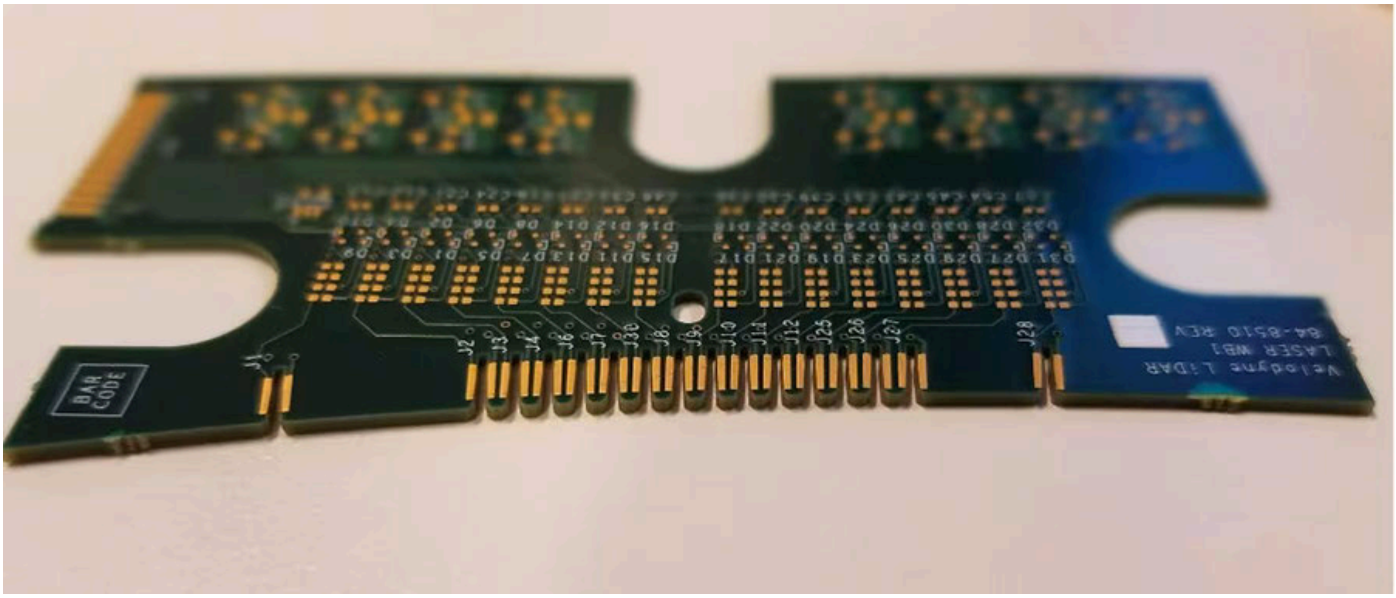


Figure 3. Laser depanelization is a must to get this intricate shape.

What we gained other than a fancy outline was the ability to handle finer edge-to-copper minimums. That may sound like a small win, until you consider the orthogonal board-to-board fusion we had going on there. Groovy little cutouts accept ridiculous little boards that shoot lasers everywhere while an equal number of detectors spin in harmony to catch the rebounding light off the local terrain. These lidar sensors put the fabricators to the test. All my round boards came from this platform.

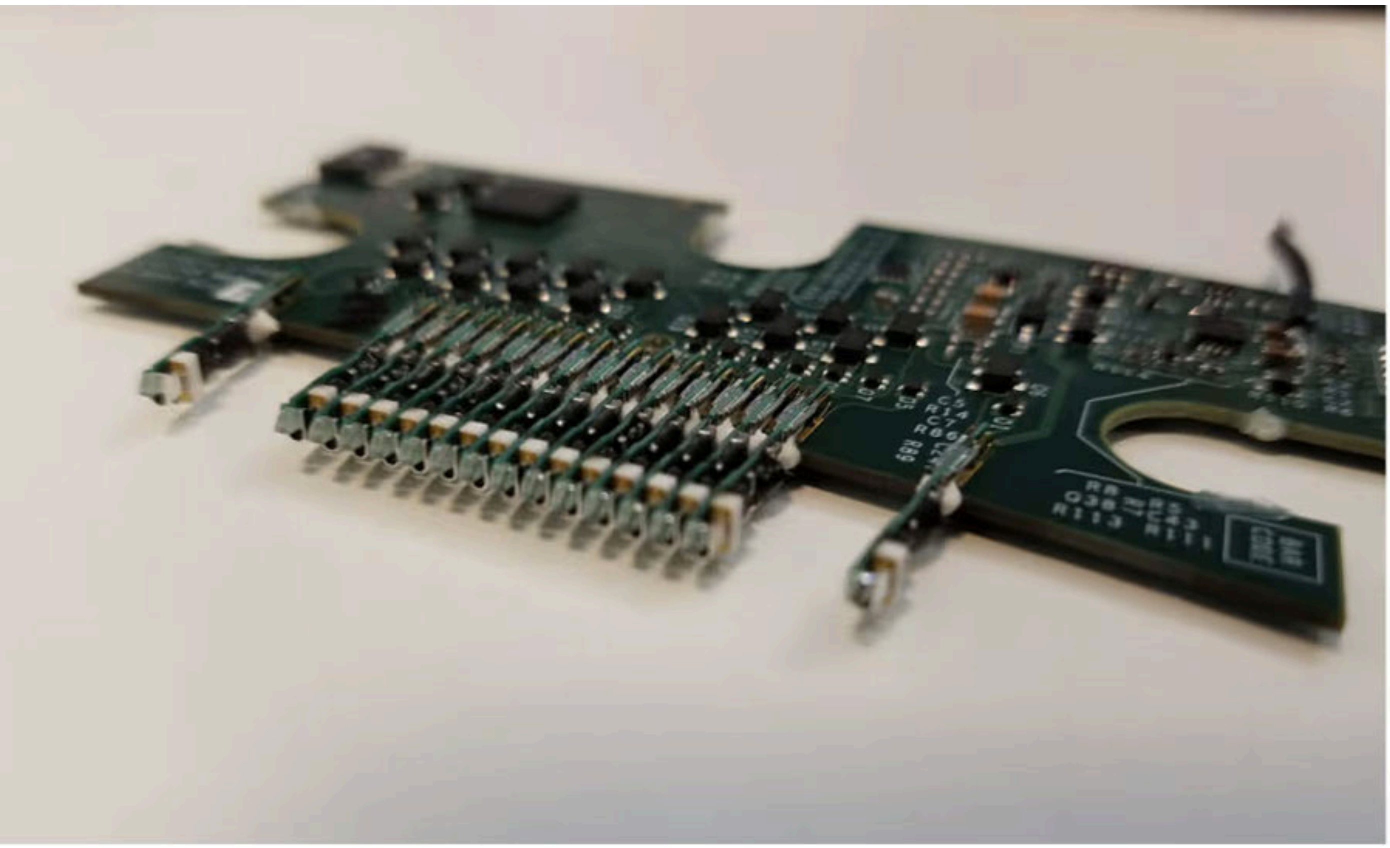


Figure 4. The tiny boards meet the little board in an unusual connector-less fashion. Giving sensory perception to a machine can require some out of the box thinking.

To wind this up, there are tradeoffs to consider for each PCB project. Working out the routing scheme through fan-out studies and experience gets you to the optimum layer count. The electrical and mechanical constraints add the spice to the mix where the z-axis gaps between metal layers are established. Simple things like metal loading have a direct effect on the geometry. Having some traces on a layer is not the same as a copper pour in terms of the outcome.

Those factors add up to the total board thickness. Solder mask and other treatments add small but potentially significant differences. A board could swell just from a humid environment. That's why assemblers routinely prebake boards prior to assembly. No surprise that there are a lot of variables when it comes to PCB thickness.

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.

PCBs' Green Evolution

The humble printed circuit board continues to change to meet new demands.

POWER IS NOTHING without control. It's not a quote by a famous politician or social commentator, or even Mark Twain. It's an advertising slogan for car tires. But it's also an apt description of the opportunities for our industry that are now happening as part of the green energy transition.

Electrification is one of today's dominant megatrends. The "old way" of releasing energy from traditional fuels by explosions and burning is giving way to alternatives like electromagnetic and photovoltaic conversion, as well as chemical processes inside batteries and fuel cells. Taking the utmost care of every joule is critical to maximize the harvest from the scarce ambient energy sources and to minimize waste throughout the conversion system, distribution infrastructure, storage and – ultimately – the load.

Exercising that care demands control. This is where more power electronics are being employed to ensure efficient and precise conversion as we accelerate the pace of electrification; changing traditional mechanical, hydraulic, and fossil-fueled tools and vehicles that we have all become accustomed to using into electrical equivalents that can be powered from clean and sustainable energy. Replacing conventional boilers with electric heat pumps for heating buildings is one example.

The pace of electrification in the automotive industry is another. Automotive electrification has been going on for most of the past three decades, offloading crank-driven loads from the engine, saving weight, reducing cost, and improving reliability. Electrification also enables the latest software-defined features for safety, infotainment and driver assistance. The ultimate step is the change to electric traction, bringing yet more power electronics into the drivetrains of every new vehicle on the road.

I noted recently figures from TrendForce Global Automotive Reports showed that plug-in hybrid electric vehicle (PHEV) sales are vastly outstripping battery electric vehicles (BEVs), growing more than 10 times faster in the first quarter of this year at 48% versus 4%. PHEVs deliver some advantages over BEVs, such as freedom from range anxiety, despite having a battery that is not only significantly smaller and lighter, but also less expensive and easier to replace at its end of life. On the other hand, they are more electrically complex, placing greater demands on the electronics industry to step up and deliver solutions.

Of course, this also brings valuable opportunities for component suppliers, including PCBs. A modern PHEV is reckoned to contain over 1 sq. m. of printed circuit board in total; that's more than double the quantity in a typical conventional vehicle after over 30 years spent electrifying auxiliary loads and adding smart features and infotainment. This is evidence that the PCB, our favorite component, has a leading role in making the green energy revolution happen.

New techniques are needed for managing the high power levels coming into and out of the circuits built on these substrates, and to dissipate the heat associated with the various loss elements. Technologies collectively known as metal in board (MiB) can extend the PCB designer's toolkit, enabling greatly increased electrical and thermal power-handling capability. With these, we can build boards capable of carrying current as high as 1000A and raising power density beyond $50W/cm^2$.

On the other hand, techniques like pedestals, inlays, encapsulated bus bars, and embedded wires and strips can be difficult to combine with some conventional processes and may demand specialist knowhow. MiB elements need to be designed into the PCB at an early stage, which increases opportunities for PCB designers to add value at the system design level, including completing all power and thermal calculations to select the right technology and optimize the selected materials and dimensions.

In addition to embedding metal, embedding silicon components such as power semiconductors promises to further enhance thermal performance and therefore add to the PCB's power capabilities. Embedding components in drilled or fabricated recesses in the PCB surface makes sense from several perspectives, including reducing the number of parts to be surface mounted – thereby streamlining final assembly. In addition, reliability and ruggedness can be improved.

We have been working toward embedding passive components for more than 20 years. Now, as markets focus intently on power density and efficiency, emerging techniques can put power semiconductors such as inverter bridges inside the PCB to reduce thermal resistance and save switching losses by lowering overall inductance in the system, in addition to reducing system complexity, enhancing reliability, and simplifying surface-mount assembly.

In addition, new thermally conductive resins and resin-coated foils, generically known as bond-ply materials, are entering the market. Increasing the thermal conductivity of non-IMS substrates makes these a great fit with embedded semiconductors and MiB, as well as lighting, power conversion, motor drives, and high-performance computing like cloud AI accelerators. As glassless systems, these materials come without Dk issues and are used in low-loss applications.

Away from the search for greater power, other interesting developments could make tomorrow's PCBs more benign toward the environment. New plant-based substrate materials to replace fiberglass now enable biodegradable PCBs to mitigate e-waste problems, enhancing recyclability and reducing landfill.

While the PCB's underlying technical role – to support the connectivity between the different elements of the system – remains consistent, its properties continue to adapt and evolve to satisfy the changing priorities of our time.

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

Custom Tailored or Off the Rack?

Much like a bespoke suit, PCBs require individual tailoring to meet a customer's needs.

A PRINTED CIRCUIT board is not a commodity and many in the electronics supply chain do not understand that. Buyers wanting "champagne at beer prices" leads them to press fabricators to devalue their efforts and work. Additionally, I have spoken with designers who have no idea how boards are made, but then turn around and speak with an air of authority that the fabricator should just "dial in the fabrication process" to etch a narrower trace on their board.

I am not sure what most buyers and designers have in mind of how printed circuit boards are manufactured, but a good analogy helps many to understand that boards are custom tailored, like a fine bespoke suit, and not a garment off the rack.

Now I realize a bespoke suit may be an analogy many wouldn't understand without a bit of context. Bespoke means made specifically for one customer. To be sure, in the modern era of business casual and even casual attire in certain work environments, suits are a lot less common.

Suits can be acquired from many different establishments these days: thrift stores, men's specialty stores, department stores, even online retailers! Depending on the type of establishment, services are generally commensurate with pricing. At a thrift store, given its nature, you are likely looking at a secondhand garment, one already customized for a different body. Not something that really fits with the analogy of PCB sourcing, but it would be very interesting to be able to reuse old PCBs in such a way.

At a department or men's specialty store, you are likely to encounter the most common type of suit, one commonly called "off the rack," as these are all premade. And by far, this is the most common type of suit seen today. Components, such as the slacks and jacket, hang on the rack, hence the name, and are generally sorted by shoulder size for the jacket and waist size for the pants. The suits can have various modifications to further customize them – usually small modifications to improve the fit. Generally these customizations come in the form of a simple hemming of the pants and adjusting of sleeve length, but they sometimes get more involved in further after-manufactured tailoring. The process for purchasing the suit involves a quick measurement of the shoulders and waist to determine the jacket and pants to try on. Once a selection is made, then customization measurements are made for the tailor to adjust the premade garment. Thus provides the name of these types of suits as "off the rack" as they are made before the customer even arrives.

The process for a bespoke suit and some made-to-pattern suits is one where measurements of one's body are taken by the tailor, and the fabric and style are chosen all the way down to the buttons. Then the cloth is cut from a roll and either sewn to a modified pattern or to a specific pattern based on the measurements. The significant difference here is everything is customizable, and nothing is precut or sewn until after measurements and selections are made. Each bespoke suit is tailored from the start to specifically fit your body. After it is sewn, there might be another fitting or two to correct any small issues or make last minute changes, but this is a suit designed from conception to fit you. Just as a printed circuit board is custom made for each revision, so too is a bespoke suit.

To give context, rough prices for these suits range from a thrift store suit costing $50-$100 and a department store suit around $200-$500. A bespoke suit may cost in the range of >$2000. This is because the process of making the bespoke suit values the artistry and customization of the tailor. More time, energy, and customization are involved, with direct communication with the customer needed to make a suit based entirely on what they want. An off-the-rack suit is modified after the fact and is sewn in a factory months or years before the customer walks in for a fitting.

Many in the PCB industry, especially buyers, try to commoditize the bespoke nature of the printed circuit boards. If the boards are commodities, then they have less value, and the price can be driven down. A commodity, after all, is a product that is identical regardless of who produces it, but there are significant differences in quality and performance between different factories. One is likely to see differences in boards between two lines in the same PCB factory!

The same schematic handed to 10 different designers, who each in turn have the designs fabricated at 10 fabricators, will result in 100 different boards. One might think that the boards become a commodity when broaching into high-volume manufacturing, but building a significant number of boards doesn't change the fact that they are still only for one customer. The PCBs aren't going to several different customers to make different products, but to that one customer. The idea that printed circuit boards are a commodity does a disservice to designers and fabricators.

Perhaps one day we will achieve design building blocks that we can pull and connect for pre-designed sections of printed circuit boards, but given their unique nature and the ever-evolving needs of designers, I doubt that will be in the near future.

Components like resistors and capacitors may be interchangeable commodities, but the printed circuit board is the last thing completed in the design and the first thing needed by the assembler to start placing components. I suspect some in our industry don't value the skill and craftsmanship necessary for high-quality front-end engineering and the factory skills that exist among PCB fabricators today. So the next time someone asks you about the PCB industry, you can tell them it is like ordering a bespoke suit.

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.

Clock Spring Flex Designs

Rotating flex circuits are buildable, but will require additional considerations.

I HAVE AN application where I need to make an electrical connection to a rotating shaft. The shaft rotates approximately 180° in each direction. Can a flex circuit wrapped around the shaft like a clock spring accomplish this? Are there any guidelines to follow or ways to predict life expectancy?

While clock spring flex designs are not super common, they are certainly not rare either. I have had multiple successful clock spring flex designs over the years. Most of these initial designs probably would not have performed as desired in the "as received from customer" state, but with a little bit of guidance we tweaked the designs and they all ended up performing well.

Many significant variables must be considered, along with a few lesser features, and all need to be weighed to get to a successful design. A few of the big factors are:

- How many total cycles will the flex realistically need to function?
- Is the coil constrained by an outer housing as it unwinds?
- How fast is the winding and unwinding?
- How many signals are being transported to the shaft?
- Are there impedance concerns?

How many cycles over the flex lifetime? Clearly this is a dynamic application, so all the rules for dynamic flex apply. The best performance is with a single layer design using rolled annealed (RA) or hyper-annealed copper foil no thicker than 1 oz. The foil grain direction should follow the longitudinal axis of the flex. Base and cover insulation and adhesive layers should be as balanced as possible. The minimum bend ratio (bend radius to circuit thickness, calculated in the "wound up" configuration) should be >150:1. No variations should be made to *anything* in the dynamic area of the flex, period.

This means:

- No changes in materials
- No cover or base openings

- No variations in overall circuit width
- No changes in trace widths or direction.

The circuit should be completely uniform through the entire flexing area. Any of the features listed above could cause a stress concentration point, the death knell for a dynamic flex. Insofar as life expectancy, design as well as possible, and then test to verify. A well-designed flex should function through hundreds of thousands, if not millions, of flexing cycles.

Figure 1. Clock spring flex-circuit designs can help facilitate electrical connections to rotating features in an electromechanical assembly.

Is the flex coil constrained by anything? The flex must be able to coil in and out without contacting any surfaces other than maybe along the edge of the flex. If there is an outer diametrical constraint where the flex makes contact and "bottoms out" before the shaft reaches the full 180° of range, there will be problems. If this is the case, another turn or two may be added into the coil length to minimize the amount that each turn in the clock spring is moving.

How fast is the shaft rotating? If the shaft rotates fairly quickly, stress may concentrate at the ends of the clock spring. Consider strain relief where the uniform run of the flex transitions to the connection. Options for strain relief would be semi-rigid epoxy or staggered lengths of thin polyimide near the last 0.5-1″ of length adjacent to the connection zone.

How many signals are there? The number of signals on the flex dictates the number of layers needed. As stated, a single layer design will give the best overall performance and longest life. If the design allows widening the flex to get all traces on one layer, do it.

Impedance requirements? If there are controlled impedance or electrical noise issues, shielding is needed. This, of course, will add thickness to the circuit, which is exactly what you *don't* want. But if the circuit will not function without a shield on one or both sides, use shielding film instead of copper planes. Shielding films typically add less than 25µm per layer, thus the impact on overall thickness is minimal. Also, consider whether the shield is needed on both sides of the circuit. Since each turn of the "spring" will be near the other adjacent turns, there may be some noise benefit from those adjacent shields.

Final considerations. As you calculate the total overall length of the clock spring, keep in mind that the more turns there are, the less each must move to accommodate the 360° of motion. The downside is that more turns means filling up the cavity where the flex needs to be able to move without restriction. Additional length will also add cost, as the circuit footprint increases. While reducing the number of turns may save some money, it increases the risk the flex will become taut in the "wound up" configuration. Balance the length so that the flex will be in a relatively relaxed condition in both the "wound up" and the "unwound" positions.

Due to all the potential pitfalls you may encounter with a clock spring design, I *highly* recommend a chat with the flex supplier to ensure that all bases are covered. This will give the best chance of success on the first spin.

MARK FINSTAD is director of engineering at Flexible Circuit Technologies (flexiblecircuit.com); mark.finstad@flexiblecircuit.com. He and co-"Flexpert" **NICK KOOP** (nick.koop@ttmtech.com) welcome your suggestions. They will speak at PCB West at the Santa Clara Convention Center in October.

PCB Design. No Humans Required.

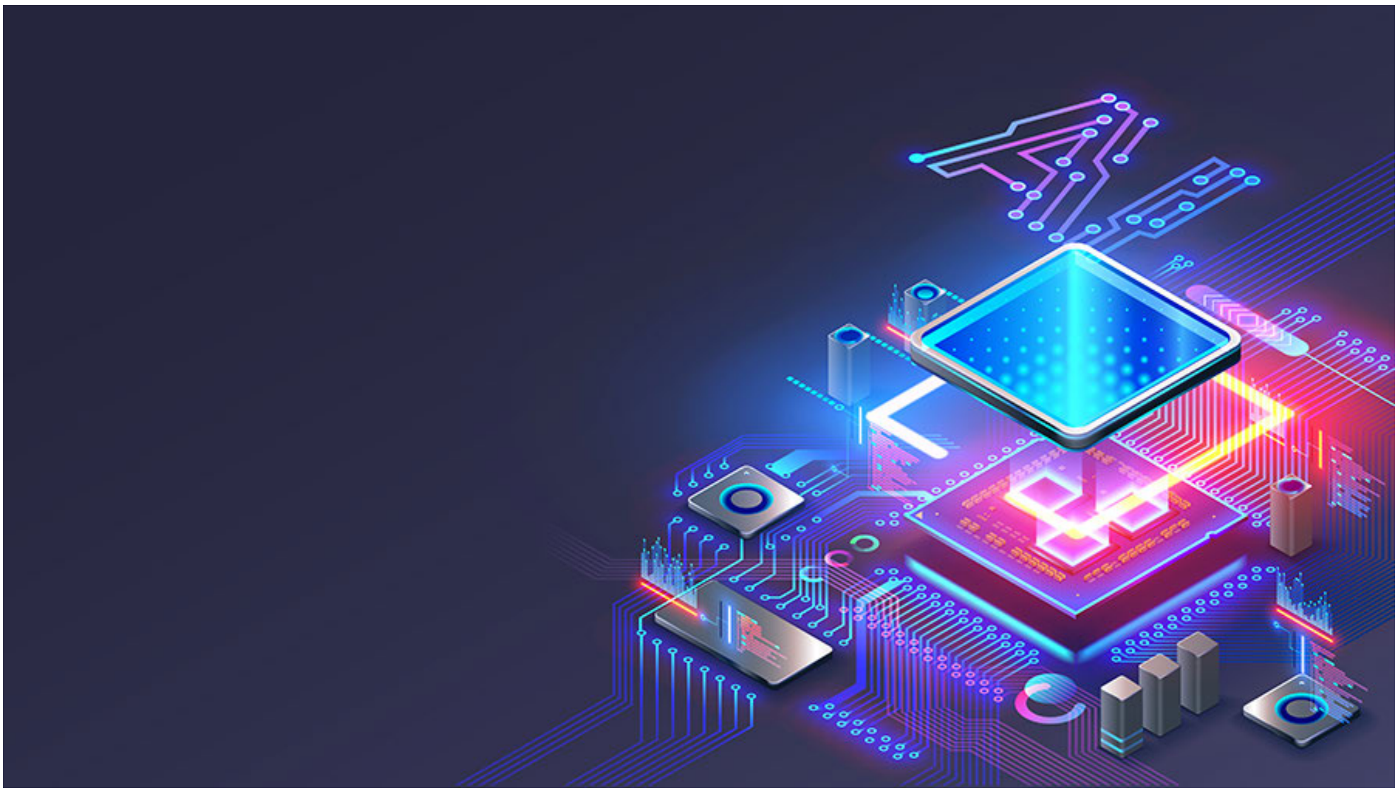
An electronics startup is developing AI-driven design software that lessens manual intervention.

by **MIKE BUETOW**

The role of artificial intelligence in PCB design is a growing topic of debate throughout the industry, with discussions ranging from the previous inadequacies of autorouting, the best methods for training it, and its potential to replace human designers.

We spoke in June with Sergiy Nesterenko, founder and CEO of a new software company called Quilter, whose goal is to accelerate hardware development by fully automating circuit board design. The former SpaceX engineer discussed why he thinks the margins designers build in are excessive and how Quilter's AI-driven, physics-based platform can resolve and even violate some "human" rules while still generating superior printed circuit boards.

The following has been lightly edited for grammar and clarity.



Quilter sees the hard constraints of reinforcement learning as superior to supervised learning AI training methodologies.

Mike Buetow: As we dive further into the world of artificial intelligence in electronics design, we have to come to grips with the fundamental hangup many designers have with regard to turning over routing and parts placement of their boards to the CAD tool. Now, some designers point out that the autorouters have been part of their tools for decades, and they are all built on some form of machine learning. But those same autorouters are often ignored in practice because they don't complete the designs to the satisfaction of the users. That's a fact that often gets glossed over or completely ignored by developers of AI-based platforms, yet you have acknowledged this head-on, noting in the past the problems inherent in autoroutes. Can you explain your thinking here?

Sergiy Nesterenko, founder and CEO, Quilter.

Sergiy Nesterenko: I think from my experience most people just don't use autorouters, right? When you go to my previous company, SpaceX, and you look at any flight board, it's a bunch of people who are choosing the stackups, who are placing components, who are routing boards and who are thinking about the physics of everything – and of course that's time consuming. For a complicated board, that's easily weeks or maybe even months of work, and that's a problem. I think the motivation to automate this is basically to just go faster and to build better boards. It's a clear need if it were to exist, and the question becomes, why doesn't it yet exist? Why hasn't it been built and why is everything that has been built not good enough?

Where I think autorouters kind of fall apart today is in maybe three major areas. The first is that they typically don't tackle the entire job. Typically, you are still responsible as the user to configure the stackup. You're typically responsible to figure out which fabricator you're going to use and enter in the manufacturing rules. If you have any special considerations like differential pairs or impedance control, you have to encode that into basically geometry rules yourself. You typically have to do the placement yourself, and then even when the autorouter presents a layout, you have to make sure it's actually going to work. That's a lot of configuration and setup, and a lot of things that simply aren't tackled by the autorouter as opposed to someone doing the full layout. The second problem is that with a board of any complexity autorouters usually just don't complete the job. It's very common to click on the autoroute button to see it get to, say 90% of a design, even if the design is only a couple hundred components. And then you'd think, "Oh, I only have 10% of the work left to do." And you quickly find that the autorouter just didn't leave you room throughout the last 10% of the traces, and you have to delete everything and restart.

The third problem I already alluded to, which is that you then have to review the autorouter's work. Even if it completed everything, you have to look at every single trace and ask yourself, "Is the signal integrity here going to be good? Is the crosstalk here going to be too much? Did it consider my high current traces? Did it consider so and so forth?" And that is arguably just as difficult as doing the routing yourself and following the rules that you've built up over time.

So the place that we're coming from is that in order to achieve full automation, in order to really solve this problem, we have to solve all of those. We have to tackle the whole problem. Start with the schematic ideally and do the whole thing: identify the manufacturers, respect all of their rules and tolerances, do the placement, do the stackups, and do the full routing with all of the physics included. You have to complete the job, right? A board that's routed to 99.9% is a broken board. Anything short of 100% is not acceptable. And you have to account for the physics. You have to show the user you've considered signal integrity and crosstalk and all of that stuff such that the schematic still fundamentally works.

MB: What does that schematic to layout process look like? Is there any human interaction needed here under the Quilter platform?

SN: One of the opinions we take is that we shouldn't try to get the user away from the tools they already know and love. For example, if you're used to Altium, we are not trying to come in and say, "Hey, stop using Altium and learn this new platform." That would be a big lift for many organizations. Instead we ask you to basically take your Altium files as an input, drag and drop them to Quilter, we do our work and we return the Altium file to you so you can continue with reviews and generate manufacturing files the same way you would have. From the user's perspective, what we really expect is that you've finished your schematic completely. At this point, we still ask the user to define the footprints, and the reason there is that a lot of companies have their own footprint libraries that are not externally shared, and so they want to have their own control over those. At that point you have a board file where the netlist and the footprints have been loaded, and then perhaps you've defined the outline of the board and maybe replaced some connectors where you know they have to go in a specific spot. Everything else stays off the board, and that's what you're inputting into Quilter. A good model for this is if you were to hire an external layout resource, like a contractor to do your layout. Think about what you would email them, and just instead of emailing that person, you're dragging and dropping that into Quilter's website.

MB: That's one of the more novel aspects to this from what I understand. The user is going to pull the files from the native environment into Quilter. Will it still look like, for example, the Altium environment inside of Quilter. Does Quilter have its own user interface?

SN: Quilter has its own user interface, and it is not meant to be complicated. The main thing that Quilter has to accomplish in the conversation with the user is to understand all of the schematic, and in particular, all of the signal types that are happening throughout the schematic. One of our areas of growth is to understand where you have high currents. Where do you have high-speed? Where do you have differential pairs? Where do you have impedance control? Because that information isn't necessarily encoded in the schematic. But after that point, Quilter's aim is to do the layout for you. The inspiration we use internally is that doing the layout for your circuit board should feel like compiling your code. If you think about it, a schematic is a little bit like a programming language, and then a circuit board is trying to make that schematic happen in real life without screwing it up. To us, that should feel more like compiling your schematic than actually doing the layout yourself, and so the interface for Quilter is simple. For that reason, it doesn't necessarily look like Altium or KiCad or any of the other tools.

MB: You mentioned KiCad there as another one of the CAD formats that you work with. Are there any others of note?

SN: Altium and KiCad are the main ones we support today. We actually had initially built out support for Eagle, but that seems to be shutting down for the most part, so we're not supporting that anymore. Cadence Allegro is most likely to be the next one we add.

MB: Do you have to develop VAR agreements or any type of technology transfer or shared agreements with any of the CAD vendors in order to develop this?

SN: Thankfully, legally speaking, reading and writing common file formats is not something that we have to go get agreements for. But we like to work with the companies because it's easier for us, and in general they've been supportive. Altium has been very supportive of us doing this so far and have extended some support from their engineers. We haven't worked with somebody at Cadence yet, but if somebody's willing to, we'd be happy to.

MB: Let's talk about the types of boards that you have used as proof of concepts and that you've successfully routed today. Let's just start at the hardest part. What's the most complex board that you know of that's been then that's been designed using Quilter? Meaning how many layers, how many pins, DDR memory, or anything like that.

SN: I think of complexity roughly in two dimensions. One is just kind of the raw combinators in geometry. So that's roughly how many pins, how dense is the board, and so on and so forth. The other is physics, like what are the most interesting signal types that you've captured.

Starting on the physics side, the most interesting board we've done to date, or at least the ones that we've personally brought up within Quilter, is one of the open-source projects from OpenMV. They have a really cool set of computer vision boards where they basically have an onboard camera, I think it was an STM 32 that we did, an SD card, USB, things of that nature, running onboard live computer vision for basic face detection or basic neural network on board or whatever else. That's an example of something we have done on the slightly faster side; I think that was a 500MHz processor. We're pushing in the next probably six to eight weeks to release support for much faster differential pairs than impedance control. That'll allow us to get into the multi-gigahertz territory for signal integrity, which will be really fun. I'm really excited for that.

On the combinatoric side, the biggest boards that we've done are unfortunately not public. They've been for companies, and I can't reveal them, but you can imagine big test boards. If you imagine a board that's maybe 15″x30″ or something of that nature with maybe 400 or 500 components, close to 2,000 pins and nothing super high-speed. Nothing more than a couple of amps but still just a lot of parts and a lot of things to connect. I believe both of these were four-layer boards. I think we've never gone so far above six layers.

MB: How long did it take for the tool to route that board?

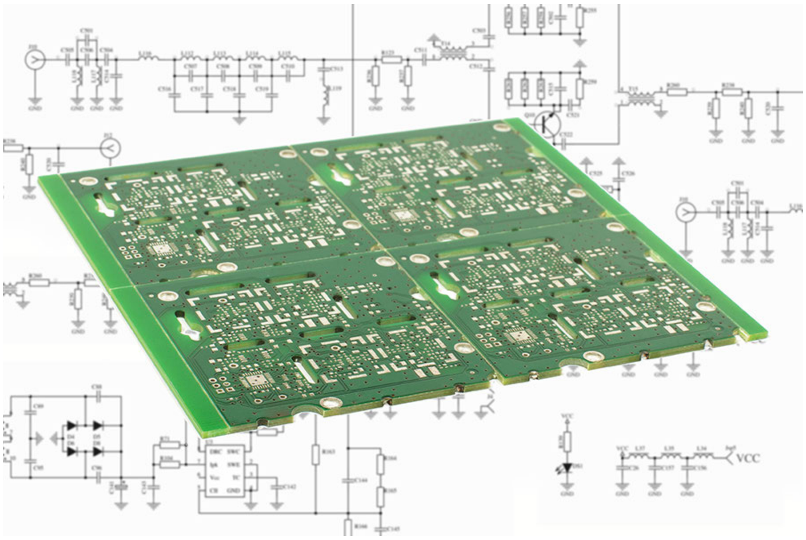
SN: We've obviously improved drastically since we've done both of those, but both of those should be running in about two to three hours by now.

MB: Start to finish, two to three hours? In your previous life, just setting up the constraints probably

took two to three days, right?

SN: Yeah, definitely. The company we did the big board for, the engineer that we worked with, estimated that that whole board would have taken them about a week to do, and that's very much our focus. The major pain that I felt while at SpaceX with designing these boards is that layout is a bottleneck. You can't put five people on it and get it done in a day, and so you're waiting for your first prototype for five days or for 10 days and you're going to have to do multiple of those iterations before you eventually actually go into production. Since it comes at the end, it becomes a critical path, and so the ability to shorten that is very valuable for a fast-moving company.

The other thing that Quilter does that's interesting is because it's done in software, we don't just lay out one board, we lay out hundreds, and so we can have a two-layer, four-layer, six-layer variant across multiple different manufacturers, multiple stackups, multiple design rules, and multiple tradeoffs, all in that two-to-three-hour time. I think that is what empowers engineers to prototype much more quickly using Quilter.



Quilter wants to make the process of going from schematic capture to a manufacturable board as fast and automated as possible.

MB: Let's talk a bit about the intelligence part of the AI. How does Quilter learn?

SN: The way that we approach this problem is by using a small subfield of AI called reinforcement learning. Most people will have heard of this technique from learning about how DeepMind defeated the world champion at Go about five years ago now, and the idea there was that they created an implementation of the game Go within a computer and then they taught the AI to sit on both sides of the board and basically play against itself. Then whenever the neural network won a game, it would be changed in such a way as to make those moves more probable, and whenever it lost a game, it would change in such a way that would make those moves less probable. After hundreds of millions of games, the neural networks found strategies for playing the game of Go much better than the human, and famously, of course, defeated the world champion.

That's very much how we approach this problem. A lot of people think that we would have taken a whole bunch of human-designed boards and learned on them or something like that. I think that's a mistake for a number of reasons, even if you were able to get that to work well. What we think of it as is that the neural network and other algorithms are playing this game of layout. Where do I put the components? What stackup do I choose? How do I connect everything? Then, once we have that kind of candidate locked in, we can then run a bunch of physics and heuristic evaluations to figure out basically, is this board going to work, yes or no, and if it's going to work, the agent becomes more probable to make a design like that, and if it doesn't, then it won't, and so it's really playing the game of layout being graded by physics.

MB: I want to yank on that thread where you noted that you think it's a mistake to try to take the human decision-making process and apply it. What is it about that that you feel is misguided?

SN: There's two main problems with it. And that's not to say that humans haven't developed great strategies; they surely have and many of those are useful as shortcuts in this process. Many things that we'll do will look kind of human because we take inspiration. But from a first principles perspective, the biggest reason that I think it's not a good idea to build a company on learning on existing board designs is that humans aren't necessarily the greatest at designing circuit boards. Many boards fail to function on their first bringup, many boards fail EMI on their first test, so already you have an issue that you are learning from a corpus of data where mistakes are probable. The second issue is that humans have to reserve a lot of margin. If you're looking at a board that's going to take you a month to do in layout, your worst-case scenario is coming up against that deadline, finding out that you couldn't quite fit everything; the components don't fit, or the routes don't fit, or you have to make it bigger or change the stackup, and all of a sudden you have to undo and add two or three more weeks of work. To avoid that, humans will start with a board that's a little bit too big for what's strictly necessary, a little bit too many layers for what's strictly necessary, with more margin on clearance and trace widths and separations than strictly necessary, just to make sure that it works and is on time the first time around.

If you learn from that dataset, that's the best you'll ever do, whereas if you learn from first principles physics, you can eventually make boards that are better than humans can. You can eventually eliminate layers, you can make it smaller, you can make it more performing. You can violate some of the rules that humans have off-the-cuff but that the physics says is actually going to work and pass. The potential to make boards that are better than what humans can do is the main reason for me.

The second reason is simply that there's not a lot of good open-source designs. The best boards are locked behind key at Apple or SpaceX or whatever, and they won't want anybody training on those boards, and even if you could collect them all there's maybe thousands or tens of thousands of them, which is typically not enough to train machine learning systems. Those are the two main reasons that we go after reinforcement learning rather than supervised learning.

MB: One of the things that appears that's really differentiating you, I think, if you go and look at many of the AI-based platforms that use actual designs as teaching tools, and what you're saying is that it's not worth the effort to try to extract from the design what the intent of the designer is, because even if you could do that, that intent might be fundamentally flawed.

SN: I think it's a matter of what level of intent you're talking about. It's very critical for us and for anybody else to understand that the designer expects a 4A current on a certain net or 2GHz impedance-controlled signal or whatever else. That's really important. You have to respect that rule, you have to verify with physics that you did it. Otherwise, the board just simply won't work. But I think the mistake that most other AI approaches take is that they are looking at the fact that the designer followed, say, the "three widths" rule for isolating crosstalk or something like that, and they're optimizing for making designs to just replicate that and you're effectively replicating a heuristic that humans have found really solve the real problem, which is "I want to make sure my cross-stack has good enough isolation between these two pairs."

MB: Obviously, it's one thing to say that a net is functional. But it's another thing to say that this is actually a buildable design. Because when we talk about whether something can work, as you noted earlier, it all comes down to the physics of the manufacturing plant – their ability to actually make it in a way that it's functional. So how do you determine if the board itself is manufacturable? Do you try to train on the constraints of the fabricator, as well?

SN: One of the convenient things in reinforcement learning is that you can explicitly force constraints onto your agent that cannot be violated. I'll give a really simple example. Very obviously, you can't have two components colliding on a board because then you couldn't solder them and they couldn't work. Instead of allowing the neural network to place components in such a way that they're colliding and letting it learn that they shouldn't collide, in reinforcement learning you can simply make it impossible for them to collide. And that's convenient because then you don't have to spend all that time for the network to learn that very, very basic fact. The same thing is possible with the manufacturing tolerances.

The way that we approach this is within Quilter, there are currently about 18 or 20 different combinations of stackup design rules that are sourced for manufacturers, and different manufacturers of course have different tolerances. So those constraints are hard constraints that the agent must respect. Quilter is effectively trying a generic stackup, something that's like four layers, 10-mil traces, 10-mil spacing – something that basically any manufacturer can do, and if that works, fantastic, but if your board has very, very small components, very small pins, that's not going to work, so then Quilter tries everything down to like a 3-mil trace with 3-mil clearance that only some manufacturers are confident in doing, and then we'll identify that for this board, you're going to need this manufacturer that can handle that kind of tolerance. Instead of training on those constraints, we just take them as actual constraints.

MB: So then if somebody wanted to do package-on-package?

SN: That's not something that we've considered to this point, to be honest.

MB: I'm just thinking in terms of the example of two components colliding, but your point is well taken. I think that the bigger question that I have is simply that as the manufacturing capabilities change, how do you ensure that the platform itself stays up to date?

SN: For us, that literally looks like a laundry list of what are all the manufacturers we're aware of, what are their tolerances, and frankly as manufacturers improve their tolerances and can support smaller features with lower clearances, that's music to my ears. That makes the combinator problem much easier and then we are more successful with more boards. For us, keeping up with manufacturers is just a matter of adding another entry to a database that says this new manufacturer exists or this new capability exists with this new kind of stackup and these new tolerances.

MB: I really do see a difference in terms of the way that you're fundamentally approaching this versus some of the other AI-based tools out there, either by startups or even by some of the very large ECAD companies. Let's talk about the company itself for a moment. How many staff do you have and what are their roles?

SN: As you mentioned, we've been doing this for a few years now, and until now we've been mostly in R&D mode. I think most people don't appreciate that this is really an ancient problem. The first papers I've ever seen of people doing actually placement and routing at the same time in an algorithmic way date back to the early '60s, so people have been researching this for six decades without really a practical solution in the market. So just to acknowledge, this is very, very difficult and we fully expect that, and we've built a company in that way. Between full-time and part-time and contractors and everybody involved, I think we're about 18 or so right now. The vast majority of that is engineering, so we've obviously got a lot of folks on what we think of as the compiler; all the algorithms and modeling required to represent boards, to manipulate boards, to input them from Altium, export them to Altium, even classic algorithms that do some amount of exploration in the design space, and of course reinforcement learning folks who do the training of models where that's appropriate. We have a team that's focused on the application itself, so that's the website, the servers, ingesting files, talking with the user, validating the input files, things of that nature. And we're starting to build up a product and design function as we're reaching from pure R&D into serving users.

MB: In your past role at SpaceX, you were an electronics engineer. Were you responsible for board design in any way?

SN: Not for flight boards, but for test boards. In particular, most of my time there was spent on radiation effects, basically ionizing radiation, "please don't kill the rocket." And most of that work is around good parts choices, schematic choices, subschematic, subsystems, redundancy, things of that nature, and I built a fair amount of boards myself to support that kind of testing and had input on the design of the actual flight boards to make sure their reliability was sufficient to survive flight.

MB: Was there a Eureka moment when you said, "Hey, look I want to go off on my own and create this product because it's a problem that I see us running into here and I hear from other companies about the same?"

SN: Ironically, the Eureka moment came five years before I left. I sat down to do my first board and was learning Altium for the first time, and I asked myself, "Why am I doing this? A computer should do this." I tried an autorouter, the autorouter failed, and I said, "OK, I guess I have to do this." I built the board, got it reviewed by a few other senior engineers to make sure it was all okay, got it brought up, and it caught on fire in my hands.

I then just took it as fact that this is how PCB design goes and I better get better at it, and it wasn't until four or five years later that I saw that it's possible for me to start a company and raise venture capital and all these things, and I married that with, what did I really suffer from in my job? What is the big bottleneck that I saw within our departments? And those two came together in the "Alright, let's start this" moment.

MB: How do you go from that to neural networks and machine learning? Did you take classes in coding?

SN: My undergrad was all in the sciences. I triple majored in math, physics and chemistry just to get a really good base for basic understanding. At SpaceX, a lot of my job was programming because I had to do a lot of statistics. So it wasn't enough to just screen parts and see how they do in radiation environments; I had to ultimately be accountable to the customer, in particular, the biggest ones at time were the US Air Force and NASA, that their payloads were safe with us. And I'd have to figure out from all the different inputs of the entire environment and the entire rocket, what is the final kind of probability of success that we're accountable for? That was a lot of statistics and simulations. Around that time, 2015 or 2016, early neural network applications were starting to come up, and it kind of became self-evident to me that that was going to be important. I just took my own time on the side to learn all the math behind neural networks and back propagation, and write it out from scratch, and I kept playing with them for the next six or seven years before getting to Quilter and being here. So partially self-taught, partially school, partially experience on the job.

MB: A triple major? You kept your advisor pretty busy, I guess.

SN: Yeah, I had to go to three different advisors to make sure that I could complete all three majors. That was fun.

MB: You announced a $10 million funding earlier this year. Can you share details about the company valuation?

SN: We haven't made the valuation public yet.

MB: Do you feel pressure to succeed given that kind of funding?

SN: Honestly, I personally feel pressure from all directions. I started the company because I wanted to see this work and so I feel accountable to myself to make it happen. Of course, the investors who were willing and brave enough to bet on us, and there's many other than the Series A round. We did a few rounds before that, and so I want to do right by them. Most importantly, probably, are my family and my employees. We've got dozens of people here who are supporting us and it's not just the people working at the company who have to carry the burden and take a risk. It's all our families as well that have to support us, and that's very, very meaningful. At the end of the day, working at a startup is much harder in a lot of ways, it takes a lot more hours, takes a lot more time, and typically for less pay, unless it actually works out. So I really appreciate the risk that everybody's taking with us and that's what gets me up every morning bright and early to make it happen.

MB: There's a lot of a lot riding on all of this and I keep going back to this idea that the surveys that we've done suggest that maybe only between 5% and maybe about 20% of boards are autorouted at all. Most of them are hand routed and hundreds of thousands of hours each year are spent doing everything manually. Congratulations for convincing others of your vision and getting this far.

Let's talk a bit about your roadmap. If we talk two years down the road from now, where will you be at that point?

SN: I roughly view our core technical compiler roadmap on the same two axes identified as the complexity earlier on. One is just combinatorics and solving boards and the second is physics. As we improve the geometry engine and the exploration of different geometries of the boards, that's what eventually allows us to go from 300, 400, 500 components and 1,000-2,000 pins to eventually tens of thousands of pins and eventually hopefully the big monster boards and the networking applications that are hundreds of thousands of pins. That's one thing that I will obviously be pushing.

Simultaneously, it's also very critical for us to push on the physics side. I'll give an example. Today, if you log into Quilter, you'll only see support for the very basic thermal considerations of currents. Within the next four or six to eight weeks, we're hoping to push support for impedance control and differential pairs, which gives us ability to do some more high-speed – specifically high-speed digital is what we're thinking about the most right now. Of course, over time we have to add support for things like switching converters and power supplies and antennas, analog circuits, ESD, and so on and so forth. I hope that within two years we're talking about some very serious boards. Some boards that are FPGAs or CPUs with DDR4, that would have taken a human a month or a month and a half to do, that we're able to do fully autonomously the physics guarantees that the user would need to trust the layouts that hopefully work on the first try. Maybe we'll have some demonstration sooner of that, and that's certainly my hope, but we'll see.

MB: What does the typical user look like today and are you able to aggregate any of the data based on how they're using the platform?

SN: To be very clear, we don't train from human boards. Anybody submitting boards to us, that's not the input that we take, because people are obviously very concerned with the privacy of their boards and the fact their boards aren't being reused for other vendors. So that's very important to state.

For the kind of users we see right now, we're in an open beta for a reason, we welcome everyone to come try it out. It's not going to be a great application for everyone yet, just because we can't do a lot of things that we want to do, but we see everybody from people actually trying to do serious big board designs that are thousands of pins to people that are adjacent. Think about firmware engineers, mechanical engineers, test engineers who need a design to de-risk and work on their prototypes but maybe aren't building the flight design or the main production design that's going to go into a big product. And of course, all the way down to hobbyists. I saw some keyboards come through in the last few days, I've seen some basic radios and things of that nature, lots of Raspberry Pi and Arduino applications, which are fun.

But I think today where the product is the most sticky is within what I think of as supporting boards. If you think a SpaceX flight computer, we can't do it reliably yet. The person designing that, we can't really help. But if you think about a person who is de-risking some of the components on it and is doing environmental testing and building boards for that use case, or somebody who's building a bed-of-nails for that board, or somebody who's building a test track for that board, those people we can help and save significant time. That's where we see the best applications so far.

MB: Tell me about the environment. Right now it's cloud-based. Do you anticipate that this will ever be available on the desktop?

SN: That's a tough question. The desktop is unfortunately much more constrained than cloud for compute resources. As you might imagine, we drastically benefit from having access to thousands of CPUs and tens or hundreds of GPUs. So the product would have to be very sophisticated to fit all the computing you need into a single desktop. I wouldn't rule it out, but what I think is more likely to happen, if we got to the point where we could be efficient enough for a single computer, is that we would then still be on the cloud but then using all that extra compute to make boards that are actually better than human. If we could take a high production run of a million boards and save a dollar per board in manufacturing, that would be very, very significant and worth spending the compute on. I suspect that is more likely where things will end up. To answer your question briefly, we don't currently expect a single desktop application.

MB: You did mention earlier about working with different fabricators, and I want to point out one of the incentives that Quilter recently added is a PCB ordering service which its users can leverage for free prototypes of their AI-designed circuit boards. Tell me more about this Fab for Free program.

SN: Obviously as an early-stage company with something that is so difficult, we are very incentivized to learn. We want to get as many people as possible creating designs, manufacturing those designs, telling us how it went, giving us feedback for what to build better, and so we're supporting that. For professional use cases where people have their fabricators and contracts already set up, this is obviously probably not that helpful, but for people who are running a board as a hobbyist or at home, we don't want fab to be a barrier to trying it out, especially if you're willing to talk with us about it or share it with other users of Quilter and let us publish a blog about it or something like that. We're happy to find the build of the PCBA and to work with you on it.

MB: Which fabricators are you working with for those?

SN: So far, we are tailored to produce files for JLC PCB, OSH Park and PCBWay, but in general, we also run compilations for rules that basically any fab should be able to handle for simpler designs, and so I wouldn't rule out any of the others either.

MB: And you're looking at this essentially like a loss leader in order to help learn more about developing your process?

SN: Yeah. I really welcome folks trying it out and just giving us good feedback. If you believe this is how things should work in five or 10 years, come help us make it so.

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

Young Guns

The annual snapshot of the industry finds the PCB design workforce is – at last – skewing younger.

by TYLER HANES

Is the PCB design workforce finally getting younger? After years of consternation from the PCB industry – and the country as a whole – about the daunting task of replacing retired or soon-to-be retired workers, those worries may be coming to a head as the workforce begins to skew younger.

That's the suggestion of the annual PCD&F/CIRCUITS ASSEMBLY Designers Salary Survey.

After last year's survey found the most respondents in the 61-70 age group, this year's found the largest number of respondents from the 26-30 age range at 19% **(Figure 1)**, edging out the 61-70-year-old group by two percentage points. Moreover, the overall response of designers and design engineers ages 40 and under was a tick higher than those ages 41-60. Balance is coming.

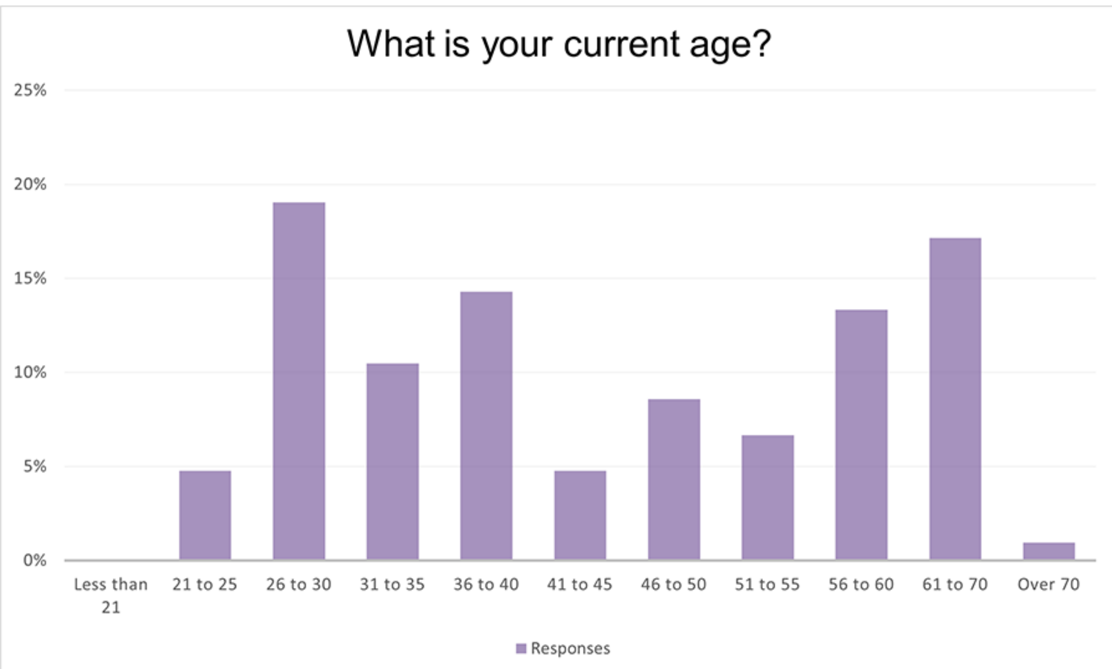


Figure 1. Designers' ages skewed younger in this year's survey.

This year's survey ran from May to July, and included responses from more than 100 designers, answering questions related to job titles and functions, ages, years of experience, education, location, types of projects, annual salaries and sales, job satisfaction and challenges, ECAD tools used, and years left in the field. Previous survey data are included here only, but due to the unscientific nature of the surveying, they should be used for general comparison only.

Table 1. Respondent Demographics

Annual Salary	Total	Graduate Degree	4 Yr. Degree	2 Yr. Degree or Less	≤10 Yrs. Experience	11-20 Yrs. Exp.	20+ Yrs. Exp.	≤30 Yrs. Old	31-50 Yrs. Old	50+ Yrs. Old	N. America	Europe	Asia	Other
>$10,000	4.7%	7.1%	7.3%	0.0%	11.6%	5.6%	0.0%	20.0%	0.0%	0.0%	0.0%	0.0%	50.0%	44.4%
$10,001 to $15,000	2.8%	7.1%	3.6%	0.0%	4.7%	0.0%	0.0%	0.0%	7.3%	0.0%	0.0%	25.0%	0.0%	11.1%
$15,001 to $20,000	0.9%	7.1%	1.8%	0.0%	2.3%	0.0%	0.0%	0.0%	2.4%	0.0%	0.0%	0.0%	50.0%	0.0%
$20,001 to $25,000	0.9%	0.0%	1.8%	0.0%	0.0%	5.6%	0.0%	4.0%	0.0%	0.0%	0.0%	0.0%	0.0%	11.1%
$25,001 to $30,000	0.9%	0.0%	0.0%	0.0%	2.3%	0.0%	0.0%	4.0%	0.0%	0.0%	0.0%	12.5%	0.0%	0.0%
$30,001 to $40,000	0.9%	0.0%	1.8%	0.0%	2.3%	0.0%	0.0%	4.0%	0.0%	0.0%	0.0%	12.5%	0.0%	0.0%
$50,001 to $60,000	2.8%	0.0%	1.8%	5.4%	2.3%	5.6%	2.2%	4.0%	4.9%	0.0%	1.2%	25.0%	0.0%	0.0%
$60,001 to $70,000	6.6%	0.0%	5.5%	10.8%	9.3%	0.0%	6.7%	4.0%	4.9%	10.0%	5.8%	12.5%	0.0%	11.1%
$70,001 to $80,000	6.6%	7.1%	7.3%	5.4%	4.7%	5.6%	8.9%	8.0%	7.3%	5.0%	4.7%	12.5%	0.0%	22.2%
$80,001 to $90,000	8.5%	14.3%	5.5%	10.8%	16.3%	0.0%	4.4%	12.0%	7.3%	7.5%	10.5%	0.0%	0.0%	0.0%
$90,001 to $100,000	11.3%	14.3%	12.7%	8.1%	14.0%	11.1%	8.9%	16.0%	7.3%	12.5%	14.0%	0.0%	0.0%	0.0%
$100,001 to $110,000	10.4%	7.1%	12.7%	8.1%	11.6%	11.1%	8.9%	8.0%	14.6%	7.5%	12.8%	0.0%	0.0%	0.0%
$110,001 to $120,000	11.3%	7.1%	14.5%	8.1%	9.3%	22.2%	8.9%	8.0%	17.0%	7.5%	14.0%	0.0%	0.0%	0.0%
$120,001 to $130,000	5.7%	7.1%	3.6%	8.1%	7.0%	0.0%	6.7%	4.0%	4.9%	7.5%	7.0%	0.0%	0.0%	0.0%
$130,001 to $140,000	4.7%	0.0%	5.5%	5.4%	2.3%	0.0%	8.9%	4.0%	2.4%	7.5%	5.8%	0.0%	0.0%	0.0%
$140,001 to $150,000	6.6%	7.1%	7.3%	5.4%	0.0%	11.1%	11.1%	0.0%	4.9%	12.5%	8.1%	0.0%	0.0%	0.0%
>$150,000	14.2%	14.3%	7.3%	24.3%	0.0%	22.2%	24.4%	0.0%	14.6%	22.5%	16.3%	0.0%	0.0%	0.0%

With the composite ages of respondents decreasing from last year, experience in the industry mostly followed along the same path. Respondents with more than 30 years of experience still held on to a narrow lead in the category, representing 27% of responses **(Figure 2)**, but nearly another quarter of designers (23%) reported five years or less and another 18% said they have six to 10 years of experience.

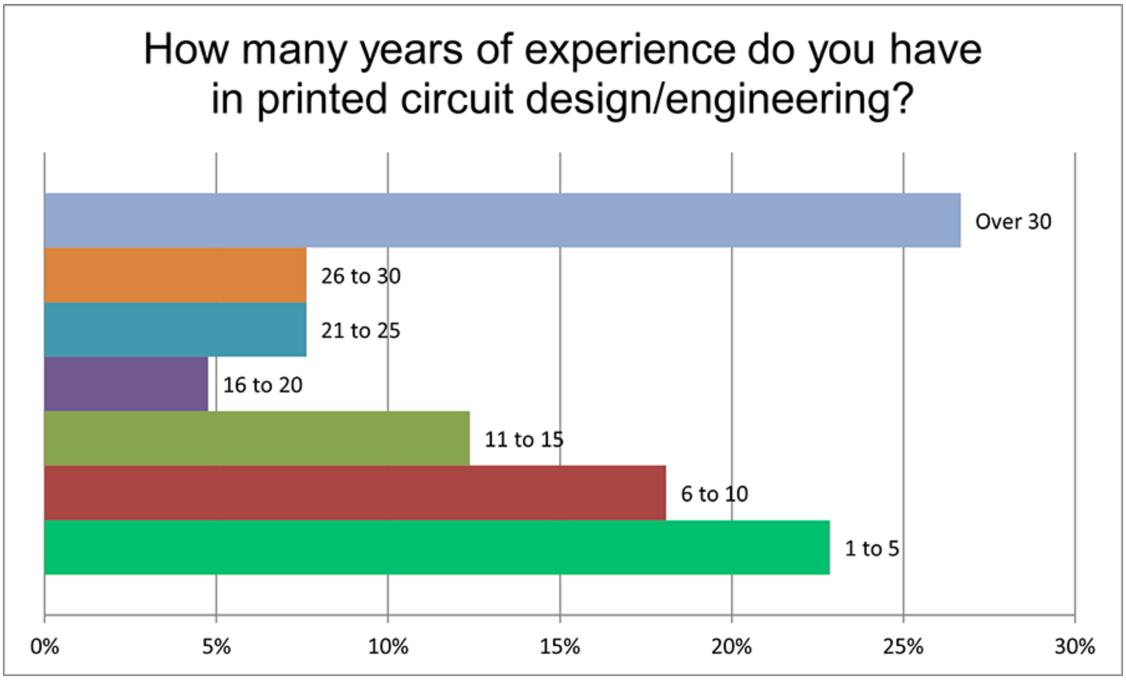


Figure 2. Designers and design engineers with more than 30 years of experience represent the largest segment of those responding.

Senior PCB designers made up the highest percentage of respondents at 18%, with PCB designer (17%), senior PCB design engineer (15%), electrical engineer (9.5%) and senior electrical engineer filling out the top five **(Table 2)**. Some 77% of designers reported that they have not changed jobs over the past 12 months and 92% said they have retained their job during that time.

Table 2. Job Titles

PCB design engineer	8.6%
Senior PCB design engineer	15.2%
PCB designer	17.1%
Senior PCB designer	18.1%
Electrical engineer	9.5%
Senior electrical engineer	9.5%
PCB design manager	7.6%
Hardware engineer	5.7%
Senior hardware engineer	2.9%
Principal engineer	2.9%
Electronics technician	1.0%
Technical director	1.0%
Other	1.0%

Percentages may not equal 100% due to rounding. n=105

Salaries and bonuses. Even with the younger ages represented, designers' salaries have remained stable over the past year. More than half of respondents, 52%, reported a salary of more than $100,000, compared to 55% last year, and more than three-quarters of respondents said their salary increased from last year. Many respondents still have some room to grow in their organizations, with 55% yet to hit the top of their employer's salary range for their position.

Figure 3. More than half of respondents reported a salary above $100,000 per year.

Of those receiving raises, 31% reported a pay increase of 1-3%, 22% reported an increase of 4-6%, 7% saw their salaries rise by 7-10%, and 18% saw pay bumps of more than 10% **(Figure 4)**.

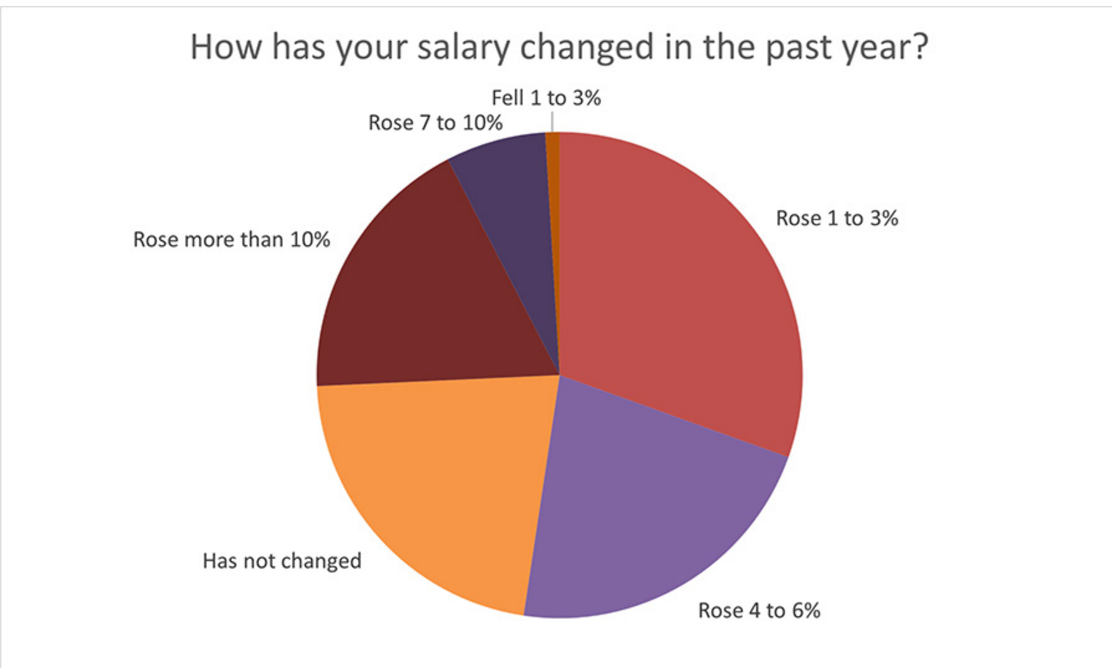


Figure 4. Most designers reported a raise from their employer in the past year.

Most employees (59%) also received an annual bonus from their employer, with 40% of those bonuses being 1-3% of their current salary and 60% receiving bonuses of 4% or more.

Job satisfaction. The bonuses and pay raises could be a major factor behind the majority's satisfaction with their jobs. 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 **(Figure 5)**.

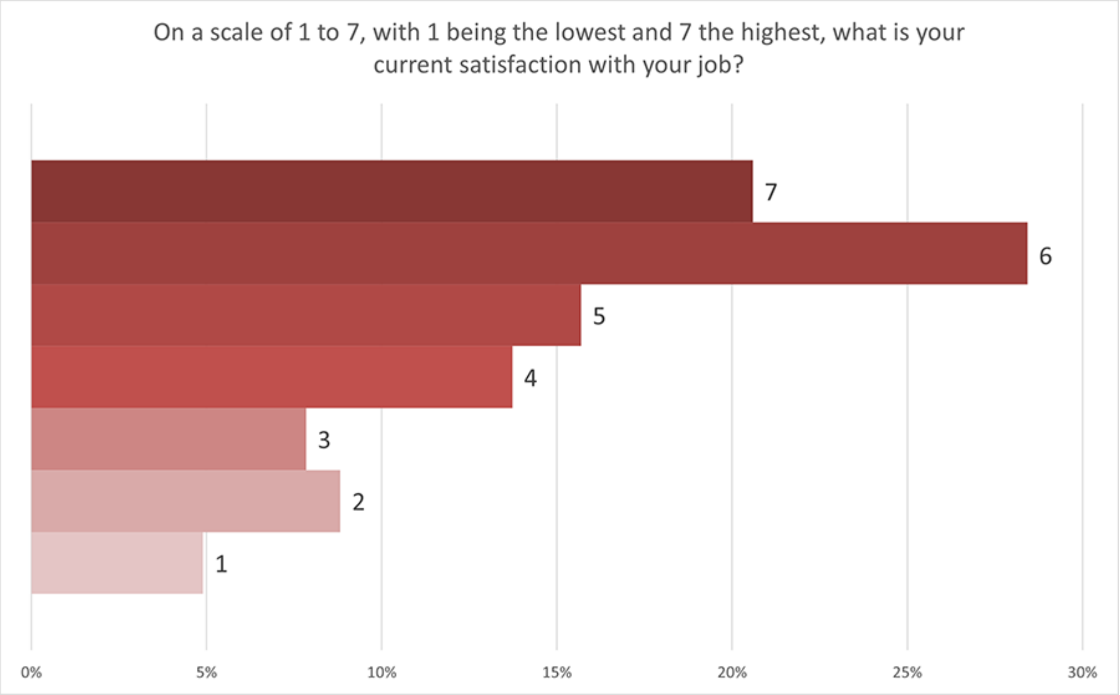


Figure 5. More than 64% of respondents said they are satisfied with their job.

That satisfaction, however, did not translate into long-term ambitions in respondents' current positions, perhaps due to the younger workforce being more open to moving to new opportunities as they arise. Some 43% of designers said they plan to be in their current position for the next one to five years, and 24% said they planned to stay for another six to 10 years. Those who reported plans to stay in their position for the long haul – 31 or more years –amounted to 8% of respondents.

Education and certification. Nearly half (46%) of respondents said they have a bachelor's of science, while 16% have a one- or two-year associate's degree and 12% attended some college but did not get a degree. Of those with a BS, 74% of the degrees are in electrical engineering.

Fewer than half (43%) of respondents said they have a design certification, and of those who are certified, 76% have earned IPC CID, and 16% have PCEA's Certified Printed Circuit Designer (CPCD) certification.

Most designers said they produce between one and 10 new designs per year, with 37% reporting one to five new designs and 23% saying they build six to 10 new designs in an average year **(Figure 6)**. More than 55% said they produce one to five respins per year, with 17% reporting six to 10, and 9.5% reporting 11-15 respins. Some 16% of designers said the number of respins has risen in the past 12-18 months.

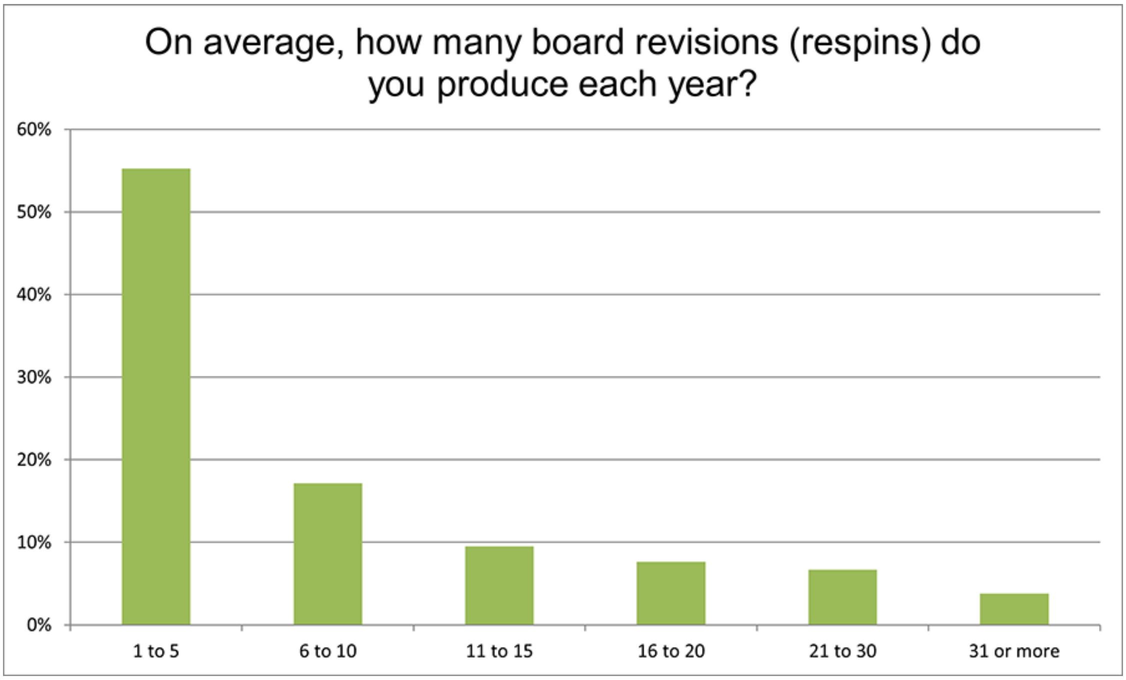


Figure 6. Most designers said they produce between one and 10 new designs per year.

Job responsibilities. With the industry continuing to expand into new technologies and techniques, the variety of projects completed by designers is also growing. Still, four in five respondents said they engineer, design, and/or lay out double-sided PCBs. Another 81% of respondents produce four- to six-layer PCB designs, and more than 66% of designers said they produce seven- to 10-layer PCBs, with 63% using BGAs and 60% designing high-speed products **(Table 3)**.

Table 3. Types of Projects/Technologies

Single-sided PCBs	51.4%
Double-sided PCBs	80.0%
4 to 6 layers	81.0%
7 to 10 layers	66.7%
12 plus layer PCBs	54.3%
Flex or rigid-flex PCBs	54.3%
FPGAs/PLDs	44.8%
Microvias/HDI	54.3%
RF/microwave circuitry	49.5%
High-speed design	60.0%
BGAs	62.9%
ASICs/ICs	31.4%
Embedded systems	36.2%
Chip-scale packages	12.4%
Chiplets/heterogenous packages/MCMs	5.7%
Package-on-Package (PoP)	5.7%
Processor and memory (DDR)	41.9%
SoCs	27.6%
SiPs	14.3%
Optoelectronics	27.6%
Enclosures	16.2%

Fewer than 20% of designers said their time is spent exclusively on design, while 29% said they spend three quarters or more of their time designing and 20% said they design 51-75% of their time.

Increased responsibility among designers is the highest point of concern among respondents, with 56% reporting that workload is one of the biggest challenges they face in the coming year and 36% being worried about keeping up with new technologies **(Figure 7)**.

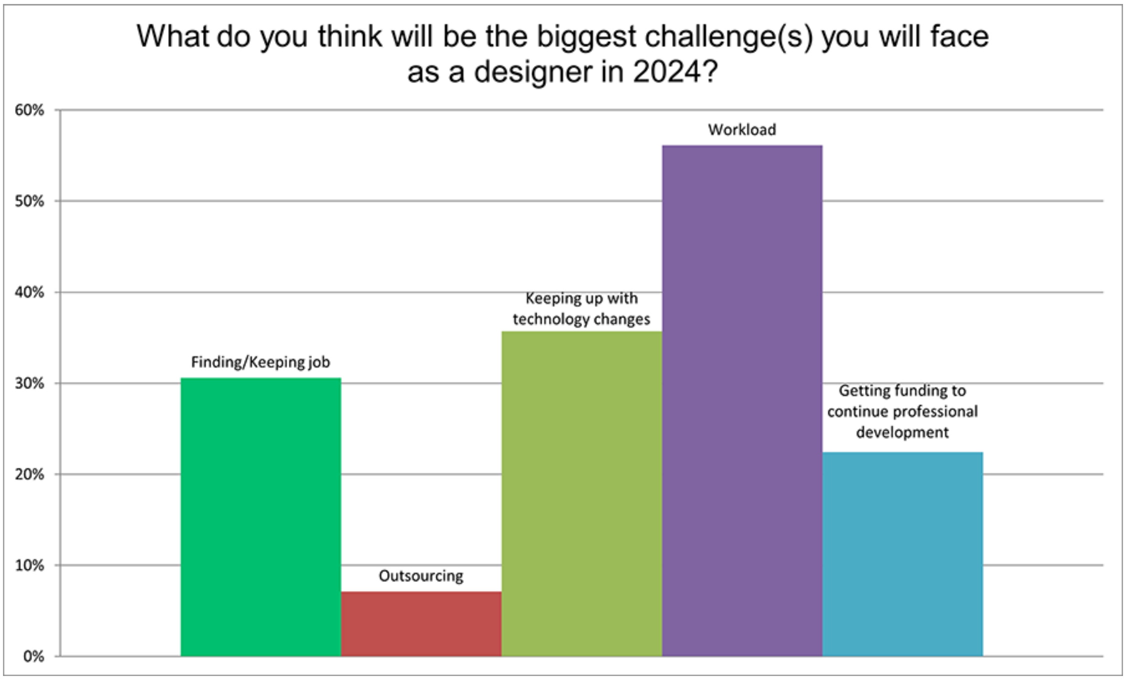


Figure 7. More than half of respondents say workload is an issue.

Other findings. Some 41% of respondents said telecommuting is a benefit offered by their company, down from 48% last year. The most common benefits included health insurance (91%), dental insurance (82%), and a 401(k) plan (76%). More than 28% of respondents said their company offers a stock repurchasing plan, 24% reported a pension/retirement plan, and 17% have profit-sharing benefits.

Of educational opportunities offered by their companies, 55% of respondents said they have the opportunity for on-the-job training, 50% said they can receive tuition reimbursement, 41% said they are able to attend classes and conferences, and 37% said they receive support for industry certification and mentoring.

Among the respondents, Altium Designer was the most commonly reported ECAD software in use, with 57% of respondents reporting they use the tool at least weekly, while Cadence Allegro came in second with 25%, and Cadence OrCAD and Siemens Xpedition tied for third with 22%.

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

'A Legacy of Innovation'

Indium Corporation celebrates its 90th anniversary with a look at its past and future innovations.

by TYLER HANES

A sense of curiosity and discovery drove the founding of Indium Corp. in the early 1930s, and that continued drive to innovate has carried the company to success over the past 90 years of its existence – leading it to become a premier manufacturer and supplier for the PCB, semiconductor and thermal management industries.

After reaching the 90th anniversary milestone this year, the company is celebrating with the theme of "A Legacy of Innovation Fueled by Curiosity," meant to reflect the technological breakthroughs and the people who have fueled them.

Aside from its headquarters in Clinton, NY, Indium Corp. has 15 other facilities in India, China, Germany, Malaysia, Singapore, South Korea, the UK and the US.

Indium was founded in 1934 as The Indium Corporation of America by William S. Murray, who sought to identify and enable the first commercial applications for indium metal, and the company's metal was included in Studebaker automobile bumpers in 1942.

In 1952, Indium developed a commercially viable process for the manufacture of precision solder preforms, which enabled mass production of alloy-junction transistors. In the late '70s, it began development of solder pastes, which soon emerged as a core pillar of the company.

In 1989, the company participated in experiments aboard multiple space shuttle missions, and entering the 1990s, it developed a number of innovations, including a no-clean solder paste that would quickly become an industry standard.

In the 2000s came advances in solder alloy technology, thermal interface materials, and semiconductor packaging materials innovations, and today, the company has grown to have 16 facilities in India, China, Germany, Malaysia, Singapore, South Korea, the UK and the US, which all help carry out Indium's mission of supplying the global electronics supply chain.

President and COO Ross Bernston, who joined the company as an engineer in the 1990s, said the company credits its growth and advances to its continued engagement with the global industry, as well as its commitment to R&D, which has allowed Indium to foresee industry trends and respond with innovative solutions – including new products to address the growing trend of miniaturization.

He said Indium has also used its 60-plus years of experience in manufacturing gallium-based liquid metals to develop high-performance liquid metal-based TIMs, which are supporting growing needs in high-performance computing for AI and IoT.

"Technological advancements are continuing at a rapid pace," Bernston said. "For example, the current advent of AI technology has created an even greater, more immediate need for thermal interface materials."

More than anything, Indium Corp. places a priority on its engagement with customers through its technical support and consultation, Bernston said, and providing that level of engagement and support is the No. 1 piece of advice the company would offer for those who are looking to build their own long-lasting enterprise.

"We pride ourselves on being more than a materials supplier or manufacturer but a strategic partner to increase our customers' productivity and profitability through the design, application, and service of our advanced materials," he said.

With governments and companies taking steps to reduce their carbon footprints, Indium has also worked to adopt energy-saving processes and sustainable practices to remain competitive in the international marketplace.

"Our product portfolio and company practices are a direct reflection of our own commitment to environmental sustainability," Bernston said.

Some of the more recent developments on that front include its low-temperature alloy technologies, which reduce energy consumption by enabling lower processing temperatures in preform soldering and by permitting reflow ovens to run at lower temperatures.

Looking to the future also includes educating and training the next generation of workers, and Bernston said Indium has contributed to that effort by supporting schools and community organizations that are introducing students to the opportunities in STEM fields.

The company estimates that it impacts more than 5,000 students annually through those efforts, and its college internship program has launched the careers of more than 100 college students in STEM fields since 2012.

Bernston said Indium also offers an apprenticeship program and assistance to support employees learning new skills and pursuing degree programs.

While it's taking the time to look back on 90 years of history, Indium is also looking to its future to make sure that it is able to continue its legacy of innovation and discovery.

"I am very excited for what the future holds as I truly believe the best days for Indium Corporation are still ahead of us," Bernston said. "We have some of the most brilliant minds in the electronics industry and, by preserving the special culture we've built here, I believe we will continue to attract the very best talent to lead us into the future and continue our legacy of innovation."

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

SOLDER **DISPENSING**

Stretching Solder Capabilities

Solder paste dispensing can solve the challenges posed by differing component sizes.

by **TIMOTHY O'NEILL**

While stencil printing has long been the cornerstone of applying solder paste, it faces limitations with extremely small or large components. This challenge underscores the need for adaptable and innovative solder paste application methods. Here we delve into the realm of advanced dispensing technologies, exploring solutions that can help overcome common production challenges.

Stencil printing, despite its efficiency, struggles with the dual extremes of component sizes – area array and bottom termination components continue to shrink, while connectors and other components may increase in size. This variance stretches the capabilities of stencil printing. Dispensing offers the advantage of infinite flexibility, capable of producing both minuscule and substantial deposits while seamlessly integrating into the existing SMT process.

For example, a 25-gauge dispensing needle has an inner diameter of 10 mils, or 0.25mm, and a length of 0.25″, or 6.35mm. If that tip were treated as a stencil aperture and its area ratio calculated, it would be 0.10, which is considered impossible on a stencil by a factor of five! Without the pressure of the dispenser and finer solder particles in the paste, nothing would get deposited onto the PCB. **Figure 1** shows a typical 25-gauge needle producing a 15-mil, or 0.375mm, deposit with Type 6 solder paste.

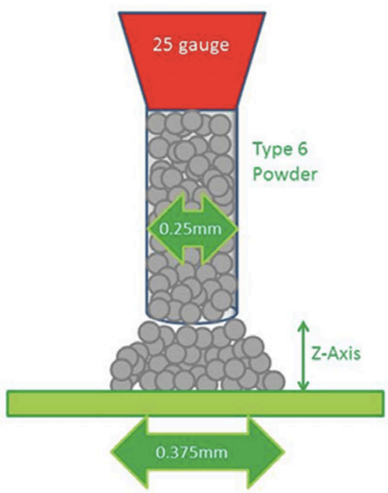


Figure 1. A 25-gauge needle dispensing Type 6 solder paste.

Dispensing techniques. Dispensing technology encompasses several methods, each with its unique advantages and limitations. Table 1 summarizes the advantages and disadvantages of three main classes of dispensing equipment used for solder paste deposition.

Table 1. Dispensing Equipment Types

Dispenser Type	Advantages	Disadvantages
Timed Pulse Pneumatic	Low cost More repeatable than manual application Low maintenance	May damage paste Limitations to dot size
Positive Displacement	Smaller dot size Excellent repeatability Flexible	High cost Increased maintenance Increased cycle time
Jetting	Noncontact Smallest dot size Fastest cycle time Flexible	High cost New technology Costly materials

Despite the apparent benefits, dispensing's primary drawback is its cycle time, which is significantly slower compared to the thousands of deposits a squeegee can achieve in one stroke during stencil printing. When substrate or component configuration and solder paste volume requirements do not permit stencil printing, however, dispensing solder paste may be the best option, either by itself or in addition to printing. Some stencil printer models even have internal paste (and adhesive) dispensers as an option, as seen in Figure 2.

Figure 2. A solder paste dispenser mounted inside a stencil printer.

Optimizing dispensing for efficiency and quality. Achieving success with dispensing technologies requires careful consideration of equipment selection and process adjustments. The solder paste being dispensed consists of two primary ingredients: solder alloy powder and flux medium. Because solder and flux densities are so different, keeping the suspension homogenous during dispensing is challenging.

Additionally, solder powder is particulate material, which increases the probability that dispensers will occasionally clog. To avoid separation or clogging with timed pulse pneumatic and positive displacement dispensers, several key guidelines should be carefully followed:

- Use the shortest tip length possible.

- Keep air pressure as low as possible; <40psi (2.75bar) for timed pulse dispensing and for <10psi (0.7bar) positive displacement, and do not cycle the air pressure on/off.
- Control the z-axis height relative to the substrate. This is critical for dot-to-dot size repeatability and reproducibility.
- Store solder paste under refrigeration, tip down, before use. If warmed to ambient temperature, do not re-refrigerate.
- Once paste has begun clogging the dispense tip, it should be scrapped.

Modifications in powder mesh size and metal content may also be necessary as the dispensing tip's inner diameter decreases, necessitating collaboration with material suppliers to ensure compatibility and performance.

Solder paste jetting advancements. Solder paste jetting has gained popularity due to its flexibility and precision, permitting easy adjustments in paste deposit locations and volumes. The latest jetting technology has not quite matched the speed of printing but has narrowed the gap considerably in comparison to its predecessors. For high-mix/low-volume or prototype manufacturers, it is a technology worth investigating.

Additionally, paste jetting valves are now being fitted to solder paste inspection equipment to provide assemblers with a "no-touch" corrective method for addressing insufficient paste deposits identified by the SPI system **(Figure 3)**. Paste can also be jetted onto PCB features requiring additional solder paste, potentially eliminating the need for stepped stencils, expensive preforms, or unconventional print techniques.

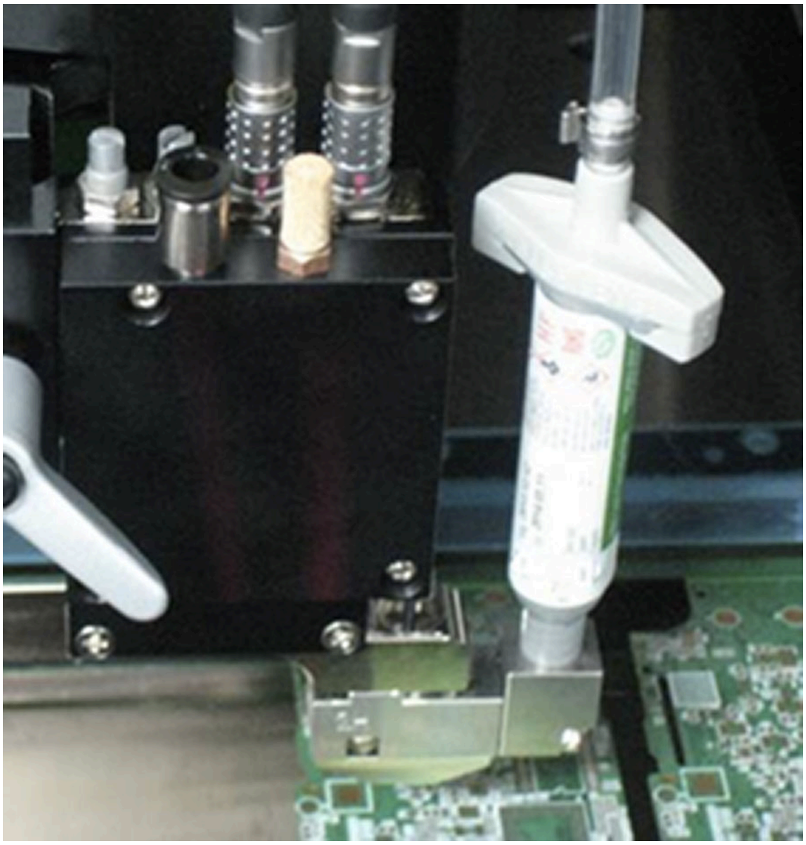


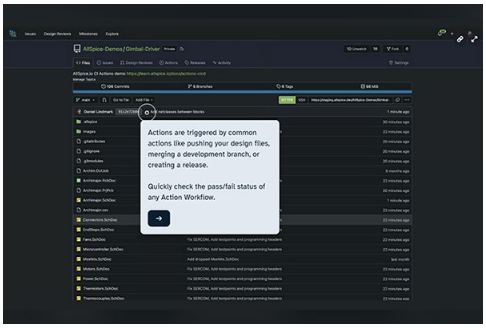
Figure 3. A jet dispenser on an SPI system touches up insufficient paste deposits and then reinspects to ensure proper volume.

Strategic considerations for dispensing success. When it comes to solder paste deposition, stencil printing is almost always the first choice because it is the industry standard. Material availability and a vast resource and knowledge base within the technical community also make printing the preferred option. But when printing alone cannot supply small enough or large enough amounts of solder paste, dispensing is often the answer.

Critical to the success of dispensing methods is collaboration among assemblers, material suppliers and equipment manufacturers. Understanding the inherent properties of solder paste and the technical nuances of dispensing equipment can significantly impact the efficiency and quality of PCB assembly processes.

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

PCD&F



ALLSPICE ACTIONS AUTOMATION ENGINE

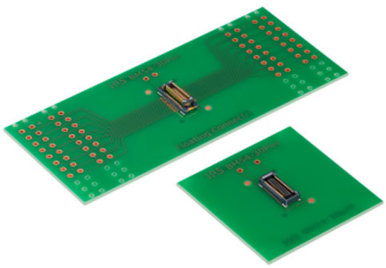
Allspice Actions enables hardware teams to automate manual checks, validations and releases of electronics designs. Provides framework to build workflows that run tests automatically. Automates almost any manual steps a hardware engineer faces, such as verifying library parts, checking for design errors and packaging releases. Common use cases include generating bill of materials artifacts, automating static checks and reviews, checking part availability, calculating cost of goods sold and verifying parts in APL.

AllSpice
allspice.io

ALTAIR HYPERWORKS 2024 SIMULATION

HyperWorks 2024 design and simulation software delivers significant advancements in AI-powered engineering and business, mechanical and electronics systems design, and simulation-driven design and optimization. Deepens AI capabilities, accelerates design exploration and iterations with generative design capabilities in Altair HyperMesh, conducts thermal analysis in Altair SimLab through “what-if” studies using real-world data from Altair physicsAI models to make design of experiments more efficient, rapidly predicts behavior with generative design in Altair Inspire to maximize efficiency in both design and manufacturing while ensuring performance, and improves training times.

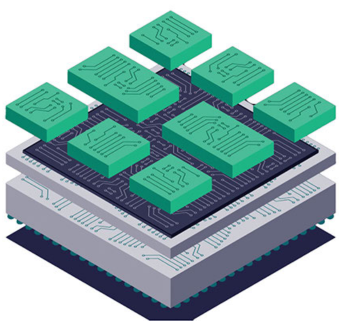
Altair
altair.com



HIROSE BM54 SERIES CONNECTOR

BM54 series board-to-board connector features the world's smallest width class for its category and is for use in automotive applications. Has a small form factor, a 0.4mm pitch, and a stacking height of 3-4.5mm. Is for PCBs with multiple connector sets and offers a wide floating range of ±0.4mm in the x-y direction. Features a two-point contact design to ensure contact reliability and rugged performance in harsh automotive conditions. Has a heat resistance of 125°C, supports PCI-ex Gen4 (16Gbps) and MIPI D-PHY Ver. 2.1 and is said to be suitable for automotive front cameras, rear/side cameras, displays, millimeter wave radar, and LiDAR systems.

Hirose
hirose.com

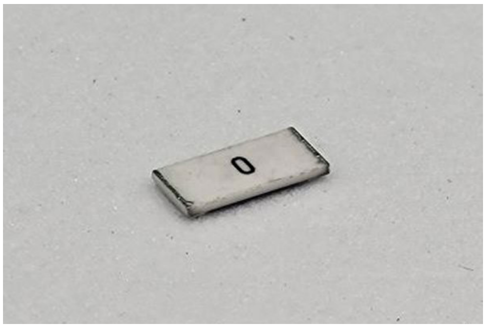


SIEMENS INNOVATOR3D IC PACKAGE ANALYSIS

Innovator3D IC provides a consolidated cockpit for constructing a digital twin, featuring a unified data model for design planning, prototyping and predictive analysis of the complete semiconductor package assembly. Unifies power, signal, thermal and mechanical stress analysis tools to enable "what-if" exploration while identifying, avoiding and solving challenges prior to detailed design implementation. Is integrated with Xcelerator portfolio but has open architecture that supports integration with third-party point solutions. Supports industry standard formats such as 3Dblox, LEF/DEF, Oasis and interface IP protocols (such as UCIe and BoW). Is capable of planning and prototyping all leading and emerging semiconductor integration methodologies and platforms including interposers (organic, silicon or glass), ABF build-up, RDL based with chips first or last including support for Deca Technologies adaptive patterning process. Is also certified for panel-level packaging (PLP), embedded or raised silicon bridges as well as system-in-package (SiP) and modules.

Siemens Digital Industries Software

sw.siemens.com

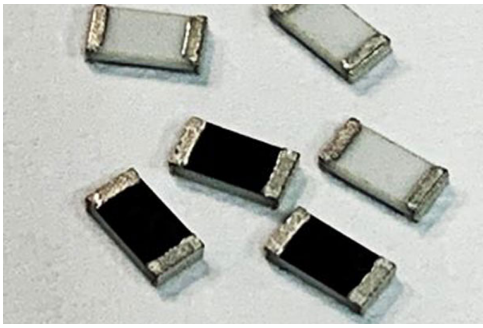


STACKPOLE CSFA SMD JUMPERS

CSFA series automotive-grade high-current SMD jumpers are capable of handling currents up to 70.7A with overload current ratings reaching up to 158A. Is suitable for a range of high-current applications, providing enhanced reliability and reduced sensitivity to surge current events, and is AEC-Q200 qualified, anti-sulfur per EIA-977 and RoHS-compliant without exemption.

Stackpole Electronics

seielect.com

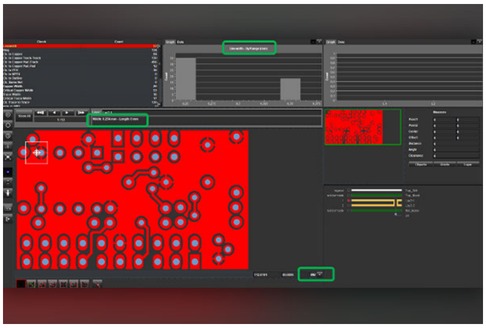


STACKPOLE HVCR CHIP RESISTORS

HVCR is a high-precision, low-VCR chip resistor with high stability. Is 100% screened to ensure VCR is within spec, and is available in resistance values from 10M and 100M, with precise VCR of less than -5ppm for the 1206 size and less than -2 ppm for the 2512 size.

Stackpole Electronics

seielect.com

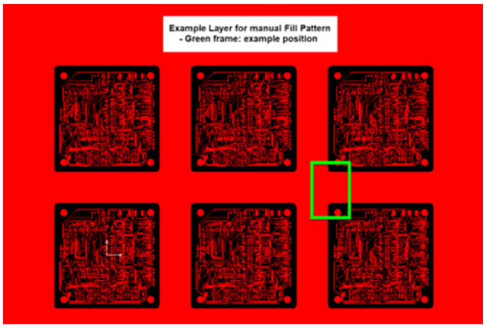


UCAMCO INTEGR8TOR V2024.06 PRE-CAM

Integr8tor v2024.06 includes extended QED Copper Areas section for top and bottom, extended Drill Tools section with a wall (or barrel surface) area for each tool individually, improved via-in-pad paste layer analysis, a QED section listing copper clearance information based on layer netlist rather than job netlist, and minimum clearance legend to pad for solder mask-defined pads. Also features checkpoint unit conversion, archive contents exclusion, reuse of incoming ODB++ pad type information, and configurable step names for DPMX import and ODB++ export.

Ucamco

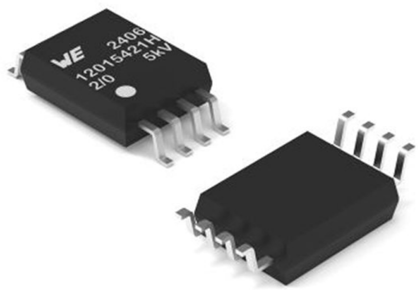
ucamco.com



UCAMCO UCAMX V2024.06 CAM

UcamX v2024.06 now features the ability for a visual check before clipping, get a copper coverage percentage by filling with tracks, and calculate copper areas for all sub jobs. Also features ATG Output that supports data sets with up to 150 layers, a new feature to edit contour text, new extended view options, extended Board Analyzer by count of drilled slot, and Gerber X2/X3 output now fully supports layer, aperture and object attributes.

Ucamco
ucamco.com



WÜRTH 2-CHANNEL DIGITAL ISOLATOR

Digital isolators series now features a two-channel version in a SOIC 8WB package (5.85 x 11.5 x 2.8mm). Has an isolation voltage of 5kVRMS, which is important for devices operating in environments exposed to high voltages and EMC interference. Is suited for applications up to a data rate of 150Mbps and features high common mode transient immunity (CMTI) to system noise of ±150kV/μs.

Würth Elektronik
we-online.com

CA

INDIUM AU-BASED PRECISION DIE-ATTACH PREFORMS

Indium's high-reliability gold-based PDA preforms offer a high level of precision to reduce defects, control bondline thickness, and deliver high-yield performance and reliability in critical die-attach applications. Features include highly accurate thickness control, precise edge quality, optimized cleanliness, and default waffle pack method. Are available in the following alloys: primary – 80Au/20Sn, 79Au/21Sn; development – 78Au/22Sn, 77Au/23Sn, 76Au/24Sn, 75Au/25Sn, 88Au/12Ge, 96.8Au/3.2Si, 82Au/18In.

Indium Corp.

indium.com

INDIUM PICOSHOT NC-6M SOLDER PASTE

PicoShot NC-6M is a no-clean, halogen-free, Type 6 powder-size material specifically formulated to be compatible with Mycronic jetting systems and Mycronic's "small dot" ejector. Is chemically compatible with Indium8.9HF series of solder pastes and is said to have been extensively tested to provide fine-dot jetting performance. Delivers dot volume of 1.6nl/dot, 230µm diameter, precision deposit (x,y targeting), long usage (stencil life more than 8 hr.), and minimal satellites.

Indium Corp.

indium.com

KYOCERA AVX KAM SERIES AUTOMOTIVE MLCCS

KAM Series automotive MLCCs now include 20 new components. Come in extensive range of case sizes, capacitance values and voltage ratings optimized for use in small, densely packed automotive circuits with fast processing speeds. Are compliant with AEC-Q200 Stress Test Qualification for Passive Components standard, manufactured in IATF, QS9000, and VDA 6.4-approved facilities, and tested in quality assurance laboratories using methods such as 100% visual inspection, increased sampling for accelerated wave soldering, and lot-by-lot reliability testing. Are RoHS-compliant and designed to be operated below the rated voltage. Come in four case sizes (0201, 0402, 0603, and 0805) with thicknesses extending from 0.33mm to 1.45mm, NiSn terminations, two dielectrics (X7R and X7T), four voltage ratings (4, 6.3, 10, and 16V), capacitance values extending from 0.1–22µF (±10%), and 7" reels loaded with paper or embossed plastic tape.

Kyocera AVX
kyocera-avx.com

MASTER BOND SUPREME 11AOHTLP EPOXY

Supreme 11AOHTLP is a two-component epoxy featuring thermal conductivity and electrical insulation. Features a working life of 60-90 min. at room temperature for a 100g batch, making it suited for demanding situations where more time for fixturing is needed. Has reliable electrical insulation properties with a volume resistivity exceeding 10^{14}ohm-cm at 75°F. Also exhibits thermal conductivity of 3-5BTU·in/(ft²·hr·°F) [0.5-0.7W/m/K]. Provides high bond strength properties with a lap shear strength of 3,200-3,400psi, tensile strength of 7,000-8,000psi, compressive strength of 20,000-22,000psi and T-peel strength of 15-20pli. Is designed to withstand thermal cycling and offers a wide service temperature range from -112° to +400°F (-80° to 204°C). Cures at room temperature in two to three days, and faster with heat, in 2-3 hrs. at 200°F. Is said to bond to a wide variety of substrates including metals, ceramics, glass, rubbers, and many plastics.

Master Bond
masterbond.com

MEK ISO-SPECTOR S3 SPI

ISO-Spector S3 series of 5-D solder paste inspection systems offers a comprehensive suite of advanced features. Is a progression from the Mek S2 SPI system, utilizing multiple projector technology to calculate solder height information. Uses a 360° ring light to accurately extract the outline of solder paste on PCBs and ensure precise inspections. Z Axis auto-focus technology measures PCB warpage during inspection and automatically adjusts camera height. Offers live SPC data for printing trends and distribution histograms to enable swift diagnosis of printing issues, is fully compatible with the Mek Catch system and facilitates closed-loop integration with stencil printers and pick-and-place machines. Features custom resolution options, plus an adjustable touchscreen that can be moved up to 10° upwards and 25° downwards for ergonomics. Can create inspection programs from Gerber data in as few as 3 min. and uses offline programming software to achieve part number registration using mount data. Accommodates PCBs up to 510 x 510mm, with ample clearance on both top and bottom sides.

Marantz Electronics

marantz-electronics.com

In Case You Missed It

Additive Manufacturing

"A Novel Method of Incorporating CNT into Additive Manufacturing Electronics Dielectric Material"

Authors: Daniel Slep and Fan Yang

Abstract: Flexible hybrid electronics parts that are additively manufactured offer greater flexibility in geometry, complexity, and variety or customizability. In this work, UV-curable dielectric materials for additive electronics and incorporated carbon nanotubes (CNTs) within a formulated UV/LED curable matrix were used. It is shown that inkjet printing CNT mixture in specific manner with a commercial UV-curable dielectric improves mechanical and thermal properties of the final dielectric compared to the dielectric without the CNT mixture, including a significant decrease in coefficient of thermal expansion, while keeping excellent electrical properties. (*Journal of Surface Mount Technology*, July 2024, https://doi.org/10.37665/5ng0w157)

Quantum Computing

"Time-tronics: From Temporal Printed Circuit Board to Quantum Computer"

Authors: Krzysztof Giergiel, *et. al.*

Abstract: Time crystalline structures can be created in periodically driven systems. They are temporal lattices which can reveal different condensed matter behaviors ranging from Anderson localization in time to temporal analogues of many-body localization or topological insulators. The potential practical applications of time crystalline structures have yet to be explored, however. Here, the authors pave the way for time-tronics where temporal lattices are like printed circuit boards for realization of a broad range of quantum devices. The elements of these devices can correspond to structures of dimensions higher than three and can be arbitrarily connected and reconfigured at any moment. Moreover, the authors' approach allows construction of a quantum computer, enabling quantum gate operations for all possible pairs of qubits. The authors' findings indicate that the limitations faced in building devices using conventional spatial crystals can be overcome by adopting crystalline structures in time. (*arXiv*, June 2024, https://doi.org/10.48550/arxiv.2406.06387)

Reverse Engineering

"Synthetic Data for Semantic Segmentation: A Path to Reverse Engineering in Printed Circuit Boards"

Authors: Adrian Phoulady, *et. al.*

Abstract: The authors present an innovative solution to the challenge of part obsolescence in microelectronics, focusing on the semantic segmentation of PCB x-ray images using deep learning. Addressing the scarcity of annotated datasets, the authors developed a novel method to synthesize x-ray images of PCBs, employing virtual images with predefined geometries and inherent labeling to eliminate the need for manual annotation. The authors' approach involves creating realistic synthetic images that mimic actual x-ray projections, enhanced by incorporating noise profiles derived from real x-ray images. Two deep learning networks, based on the U-Net architecture with a VGG-16 backbone, were trained exclusively on these synthetic datasets to segment PCB junctions and traces. The results demonstrate the effectiveness of this synthetic data-driven approach, with the networks achieving high Jaccard indices on real PCB x-ray images. This study not only offers a scalable and cost-effective alternative for dataset generation in microelectronics but also highlights the potential of synthetic data in training models for complex image analysis tasks, suggesting broad applications in various domains where data scarcity is a concern. (*Electronics*, June 2024, https://doi.org/10.3390/electronics13122353)

Wearable Electronics

"Permeable, Three-Dimensional Integrated Electronic Skins with Stretchable Hybrid Liquid Metal Solders"

Authors: Qiuna Zhuang, *et. al.*

Abstract: The development of wearable and on-skin electronics requires high-density stretchable electronic systems that can conform to soft tissue, operate continuously and provide long-term biocompatibility. Most stretchable electronic systems have low-density integration and are wired with external printed circuit boards, which limits functionality, deteriorates user experience and impedes long-term usability. Here, the authors report an intrinsically permeable, 3-D integrated electronic skin. The system combines high-density inorganic electronic components with organic stretchable fibrous substrates using 3-D patterned, multilayered liquid metal circuits and stretchable hybrid liquid metal solder. The electronic skin exhibits high softness, durability, fabric-like permeability to air and moisture and sufficient biocompatibility for on-skin attachment for a week. The authors use the platform to create wireless, battery-powered and battery-free skin-attached bioelectronic systems that offer complex system-level functions, including the stable sensing of biosignals, signal processing and analysis, electrostimulation and wireless communication. (*Nature Electronics*, July 2024, https://doi.org/10.1038/s41928-024-01189-x)