

JUNE 2021

IConnect007

PCB007

MAGAZINE

You Asked...



...We Answered!



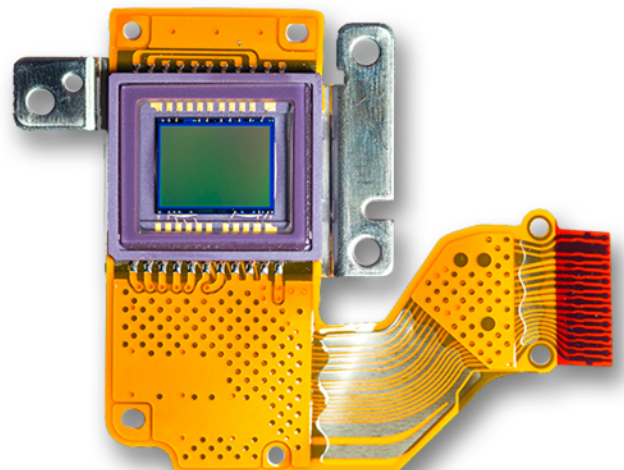
Online Training Workshop Series:

Flexible Circuit Technology

with Joe Fjelstad

This free on-demand workshop series is a comprehensive look into the structures, applications, materials, and manufacturing processes of flexible printed circuits.

START WATCHING



PRESENTED BY

I-Connect007
GOOD FOR THE INDUSTRY

SPONSORED BY



American Standard Circuits
Creative Innovations In Flex, Digital & Microwave Circuits

I-007eBooks

Distinctly different.

Our books are written by recognized industry experts.
At around 8,000 words, they are unique in that they are able
to be incredibly focused on a specific slice of technology.

“I-007ebooks are like water in the desert
...it's up to you to drink it in order to survive!”

Stephen V. Chavez
PCEA Chairman, MIT, CID+



VISIT OUR LIBRARY



PCB007

M A G A Z I N E

Just Ask...

This month, with our “Just Ask...” issue, we bring in the experts. We opened a virtual Q&A session, you asked the questions, and the experts answered them. Part of continuous improvement methodologies, after all, is to leverage the expertise of others.

FEATURE QUESTIONS

14

**Supply Chain—
An Industry Veteran’s
Perspective**

Answered by Richard Crowe



16

**How long is the copper
shortage anticipated to last?**

Answered by Michael Coll



18

**Is it possible to use another
material as a substitute for
heavy copper with different
characteristics and the
same performance?**

Answered by Kim Sauer



20

**Beyond solder, what’s the
next best interconnection
between the components
and the PCB?**

Answered by Charles Bauer



22

Also Answered by Joe Fjelstad



24

FEATURE QUESTIONS

**What channels are most PCB
assemblers using to educate
themselves on the latest process
improvements and equipment?**

Answered by Leo Lambert



26

**How can PCB fabricators
achieve finer lines and spaces using
current moderately priced substrates?**

Answered by Eran Lipp and Yaad Eliya



28

**What marketing techniques are
best suited to engage with PCB
assembly engineers in 2021/2022?**

Answered by Dan Beaulieu



FEATURE COLUMNS

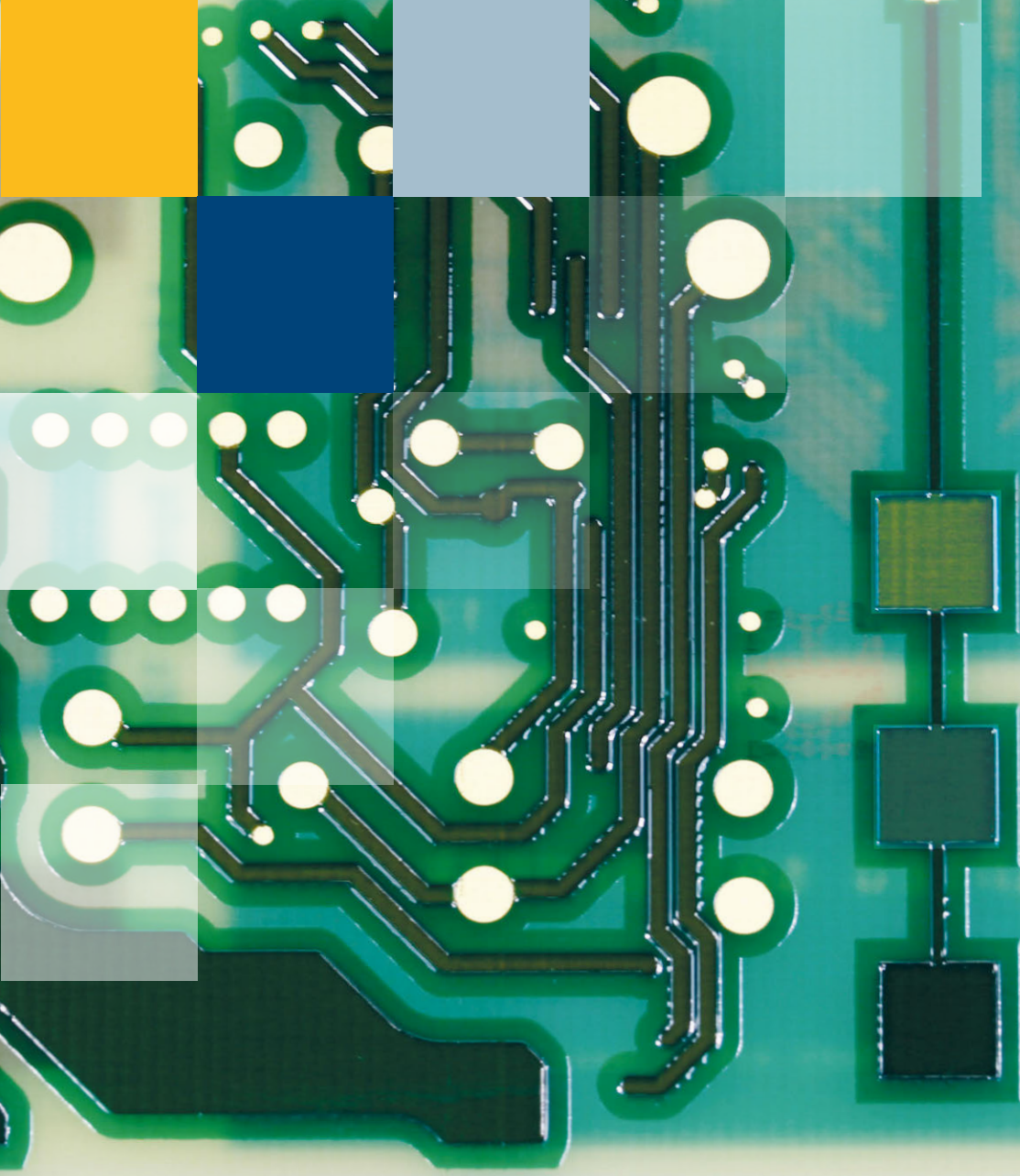
10

**Responding to Questions About the
Future of Electronics Manufacturing**
by Dr. John Mitchell

32

**A Simple Question Can Save
You Time and Money**
by John Steinar Johnsen





inkjet

printed

soldermask

1,0 kV

2,5 kV

3,5 kV



INKJET DEPOSITION OF SOLDER MASK RESISTS FOR PCB PRODUCTION

Only add material where it's required!

Inkjet printing considerably reduces capital equipment investments, associated labor, floor space, chemicals usage and related handling and disposal costs. Because inkjet is digital and additive, it is highly flexible, thus minimizing manufacturing turn-around time.

SUSS MicroTec's award winning novel inkjet-printed solder mask solution for PCBs enables drop-on-demand inkjet printing. Avoid costly photo processes and leapfrog to the digital future of additive solder mask today!

Always be one step ahead — with solutions from SUSS MicroTec.

Scan QR code for more aspects of inkjet solder mask coating!

SUSS MicroTec
info@suss.com
www.suss.com/pcb



PCB007

M A G A Z I N E



40

ARTICLE

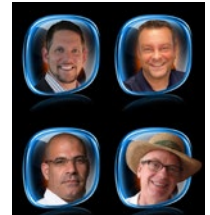
- 40 TLPS Z-Axis Interconnect Solutions for Thermal Transfer and Electrical Connection in PCBs**
by Catherine Sheerer and Gary Legerton



56

COLUMNS

- 8 Just Ask... and More Tailor-Made Content**
by Nolan Johnson



- 56 DIG: The Next Generation**
by George Milad
- 62 Meet Mr. Henry and Don't Blow a Fuse**
by Todd Kolmodin
- 68 Leadership 101—The Laws of Navigation, Addition, and Solid Ground**
by Steve Williams



68

SHORTS

- 9 I-Connect007 'Just Ask' Q&A Compilation 2020 Edition**
- 30 Book Review: Dan's Biz Bookshelf—Winning the War for Talent**
by Dan Beaulieu
- 72 Driving in the Snow is a Team Effort for AI Sensors**

DEPARTMENTS

- 77 Career Opportunities**
- 90 Educational Resource Center**
- 91 Advertiser Index & Masthead**



HIGHLIGHTS

- 38 EIN007 Industry News**
- 60 MilAero007**
- 66 PCB007 Suppliers**
- 74 Top 10 from PCB007**



WORLD-CLASS SUPPLIERS

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

CALL 949.587.3200 FOR MORE INFORMATION



RF MANUFACTURING MADE EASY

Astra® MT77 is a low dielectric constant with ultra-low loss performance material with easy PCB manufacturing.

- ▶ Stable Dk of 3.0 over temperature and frequency
- ▶ Ultra-low loss material Df 0.0017
- ▶ Laminates and preregs provides full design flexibility
- ▶ Easy standard PCB processing



FEATURES


 HIGH THERMAL
RELIABILITY


 SUPERIOR ELECTRICAL
PERFORMANCE


 RADIO FREQUENCY
& MICROWAVE

MARKETS


 RADIO FREQUENCY
& MICROWAVE


 AUTOMOTIVE &
TRANSPORTATION


 AEROSPACE
& DEFENSE


 5G
APPLICATIONS

✉ info@isola-group.com
☎ 800-537-7656
🌐 isola-group.com

isola®

EMD PERFORMANCE MATERIALS

Ormet® Sintering Paste: Fewer Multiple Lamination Cycles

Ormet® Transient Liquid Phase Sintering (TLPS) via fill pastes enable anywhere and any-layer vias in high-performance complex printed circuit boards without any need for copper plating. TLPS-filled z-axis interconnect layers can be fabricated in parallel with individual x-y trace layers, or PCB sub-constructions of multiple layers with installed PTH and laminated in a single cycle.



Just Ask... and More Tailor-Made Content

Nolan's Notes

by Nolan Johnson, I-CONNECT007

The year of $X = X_c - 1$ continuous improvement rolls ever onward; this June issue marks the halfway point. Our dive into this theme has uncovered some insightful and sometimes poignant stories. I'm personally touched by all the industry folks we've talked to—they continue to be lifelong learners, continue to grow, and tackle knowledge and expertise as a continuous improvement exercise.

On the editorial team, we're already planning the October magazine content. July, August, and September issues are in their respective stages of development and planning. For us, we're over 75% of the way through steering the continuous improvement discus-

sion and, I have to say, there's still plenty to talk about on this topic!

This month, though, we take a slightly different approach, with our "Just Ask..." issue. If you subscribe to our Daily Newsletter then you've seen our periodic newsletter feature, "Just Ask..." In that feature, we conducted a virtual Q&A session with a specific industry expert and published the results. For this issue, we flipped the script a bit, in that we asked you for questions and then we went out to find the experts who could provide the answers. The questions ranged from global and strategic, to tactical and technical. The "Just Ask..." issue allows us all to benefit from questions and



answers that our colleagues shared. Part of continuous improvement methodologies is to leverage the expertise of others, after all.

This brings up another opportunity to leverage the expertise of others. If you haven't already subscribed to I-Connect007's Educational Resource Center, inside the *my I-Connect007* portal, I-007e, you really should. You will find so much useful information with which to cultivate your expertise. Our topic-focused books, webinars, workshops and roundtables offer plenty of strong content for those seeking further education. In fact, some of our readers have shared with us that our materials are used as company desk references and employee training materials. Some universities, we've learned, are using our technical library as supplemen-

tal texts for electronics courses.

Of course, if you're an expert on a topic, and your company or institution wishes to publish on that topic, we invite you to talk with us about how to get started.

The process of building up this issue was fun for us. We all hope you find value in the "Just Ask..." conversations. Let's keep this ball rolling; if you find this issue inspired questions or ideas for you, we'd like to hear from you. **PCB007**

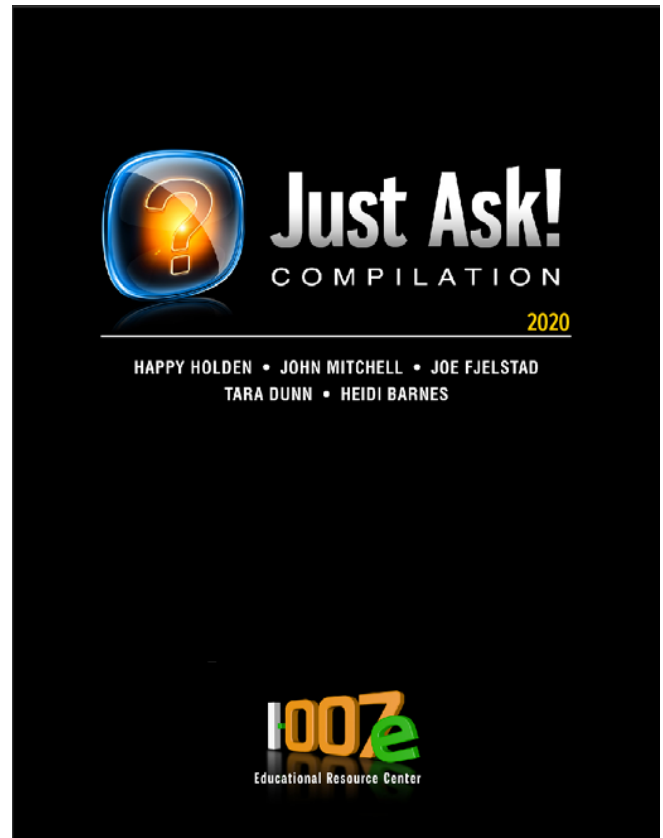


Nolan Johnson is managing editor of *PCB007 Magazine*. Nolan brings 30 years of career experience focused almost entirely on electronics design and manufacturing. To contact Johnson, [click here](#).

I-Connect007 'Just Ask' Q&A **Compilation** 2020 Edition

Throughout 2020, we asked our readers to send in their questions for Happy Holden, John Mitchell, Joe Fjelstad, Tara Dunn, and Heidi Barnes. We gathered an array of questions for these industry experts.

These five luminaries have seen it all. Happy, also known as "Mr. HDI," pioneered many of today's PCB design and fabrication processes while working at Hewlett-Packard. John Mitchell is president and CEO of IPC, and he co-founded the Alpine Electronics research company that introduced navigation systems into the U.S. Joe Fjelstad founded Silicon Pipe and Verdant Electronics, and he has over 185 patents issued or pending related to electronic interconnect and packaging technologies. Tara Dunn is the VP of marketing and business development for Averatek and founder of Omni PCB, and she specializes in additive processes, flex and rigid-flex, and RF/microwave applications. And Heidi Barnes is a signal integrity engineer with Keysight Technologies. She was named 2017 DesignCon Engineer of the Year, and Heidi also won NASA's coveted Silver Snoopy Award.



[Click to download the *Just Ask!* Compilation.](#)



COLUMN

One World, One Industry

John Mitchell, IPC



Responding to Questions About **the Future** of Electronics Manufacturing

One World, One Industry

Feature Column by Dr. John Mitchell, IPC PRESIDENT AND CEO

The questions I receive most frequently depend upon the individual or group asking the questions. For this column, I will focus on the different questions coming from the manufacturing industry and media.

The industry often asks questions about business, such as:

- How long will the supply chain pressures last?
- What government impacts do you expect?
- What can we do to solve our workforce issues?

The media often asks about the future, such as:

- What is the next thing that will take us to new capabilities?
- How can global manufacturing companies continue to meet the challenges presented to them?

Here are a few high-level answers to those questions.

How long will the supply chain pressures last which have been exacerbated by COVID-19?

This question is very dependent upon the industry segment that is asking, but right now there is a cascading effect that will likely last through the end of the year, if not into mid-2022. A shortage in one segment of the supply chain can often lead to additional shortages as substitutes are sought and modifications are made to try to meet demand.

The steep nature of the recession, followed by bounce-back, wreaked havoc with supply chains in several areas—largely because they did exactly what they were supposed to do. When an industry drops demand significantly (like the automotive industry did in going from 900,000 vehicles manufactured in a month to 6,000), the prudent thing to do is to move your processes to a different segment where you can make sales. This was done. What has never happened before was the meteoric rise back to higher levels of sales in such a short period of time. Sadly, I expect the aftershocks to continue to ripple for a while.

CAN YOU PLAN FOR THE FUTURE IF YOUR SYSTEMS KEEP YOU IN THE PAST?

See why the top flex PCB fabricators rely on ESI to help them innovate and stay ahead.

Leverage CapStone's unique technologies to deliver breakthrough productivity.

- Cut per-panel processing costs by up to 30%
- Double your flex PCB production throughput
- Lower total cost of ownership

Learn More Here

"Stepping Up To Laser Processing for Flex"



CapStone™



ESI®

www.ESI.com



What government impacts do you expect on the manufacturing industry?

Every time there is a change in a major nation's government, this question comes up. In 2013, the question was about how Xi Jinping would change things now that he was in charge in China. Today, what changes can you expect under the Biden administration? These are tricky ones to answer because, obviously, we are not in the inner circle of the administrations of any country. The best we can offer is based on prior actions and continual inquiries to those who work with and influence these leaders. If the latest actions are any indication, I expect the Biden administration to continue to attempt to invest in strengthening the U.S. industrial base. Because of this, we at IPC are looking to make sure the electronics manufacturing ecosystem is not lost in the glare of spotlight focused on the semiconductor industry.

What can we do to solve our workforce issues?

The world is changing rapidly, and nowhere do we understand that better than in the electronics industry. The shortage of skilled workers has been an issue for a decade or more. IPC is assisting with this challenge through our workforce education programs. These programs are designed by industry experts and build in collaboration with organizations around the world. The IPC Education Team utilizes the latest educational techniques to ensure the maximum level of learning retention, while simultaneously reducing training times as much as possible. These programs help organizations address specific skill gaps in their existing workforce, upskill existing workers and, perhaps more importantly, provide a solid foundation for those who are not yet, but can rapidly become, a contributing part of the manufacturing workforce.

What is the future 'next thing' that will take us to new capabilities?

We are finding ways to improve and use the latest technologies in our manufacturing facilities. The drive to modernize and transform factories will lead to more innovation. There are tremendous capabilities that are available today, but the challenge is that manufacturing is a capital-intensive business. You don't just toss away a half million-dollar piece of equipment because it is last year's model—like you might with a cellphone.

Finding ways to leverage the best techniques and tools is the key to accelerating to the future. Once you have those, you next need people who know how to use them and who understand the fundamentals behind why they are performing the tasks they are in the larger manufacturing picture.

How can global manufacturing companies continue to meet the challenges presented to them?

The answer to this question varies with the time and the issues the industry in a particular region is facing. Over the past couple of years, my answers have centered around strengthening your local/regional industrial base. If you have 100% reliance upon any other region—you are at risk. This doesn't mean you need to have every capability 100% sourced locally, but you do need to have options should relations break down. This also provides more local know-how, which is always a good thing. **PCB007**



Any additional questions not addressed here? Feel free to contact me at johnmitchell@ipc.org.

Dr. John Mitchell is president and CEO of IPC.

To read past columns or contact him, [click here](#).

*Welcome to the Summit
Interconnect Family!*

WELCOME TO THE SUMMIT

SUMMIT ACQUIRES EAGLE ELECTRONICS

READ THE PRESS RELEASE HERE



summit-pcb.com



eagle-elec.com



Just Ask Dick Crowe

Burkle North America



Supply Chain: An Industry Veteran's Perspective

In earlier times, economists said the consumer should always opt to purchase items from the lowest-priced supplier, regardless of origin. This policy always benefitted the consumer. Tariffs and regulations were designed to enforce fair and equal access in an economic world that once was a great deal simpler than today.

Now, the world economy is a great deal more complex and intertwined than ever before. The theory that developing countries could prosper and provide jobs to help ensure peace and to rebuild economies after war was a major part of the thinking after World War II. Europe and Asia rebuilt their economies, providing workers with income and a bright and successful future. The United States prospered as well with a segmentation of staples that balanced trade and access to lower-cost goods. The list of items that fit this theory included toys, consumer goods (like electronic items), and high value luxury items like exotic, high quality cars.

In more recent times, American industry has been burdened with regulations that increased cost, but have been ignored by other nations. The printed circuit board industry struggled to maintain a world manufacturing position and nearly collapsed in the beginning of the current century. I remember well a TMRC presentation by Bill Loeb who correctly said that there was “no money in the box (meaning the

computer) but in the software that operated the box.” His presentation, “The Winds of Change,” was correct; computer company after computer company moved their manufacturing “offshore.” IBM is a good example of a successful computer product—the Think Pad—that was sold off to Lenovo, a Chinese company. Other companies followed—Dell, Compeq, Gateway, and more. Highly trained engineers and workers lost their jobs and were told to re-educate themselves into other types of employment. This is just one of the reasons why our economic policy should address this offshore/onshore issue.

But there are other aspects of trading with a single source supplier. The recent worldwide pandemic illustrated the fragile nature of “putting all of one’s eggs in one basket.” How dangerous is it for electronics companies to design in the United States, but build elsewhere?

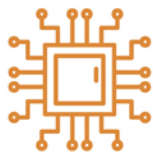
Asian suppliers responded quickly to the “pandemic shutdowns” and it seemed as if no company was endangered by a lack of supply. However, medical, personal protection equipment, and medicine were purchased offshore and there was a supply issue for some time. Now we see a shortage in semiconductor chips used in cars and trucks. CBS’s news show, *60 Minutes*, recently interviewed U.S. Secretary of State Antony Blinken, where he discussed this subject and highlighted the



COPPER FOIL

North America's only electrodeposited copper foil manufacture is beginning a new era as part of the Nippon Denkai, Ltd. family.

DENKAI AMERICA IS THE LEADER in the manufacture of high-quality electrodeposited copper foils for printed circuit board (PCB), industrial, and energy storage applications. With the strength of domestic manufacturing, and backed by a global presence, Denkai America delivers both conventional cladding and application specific copper foils required by technology leading companies. We are committed to:



Continued dedication
to the PCB industry



American made ED
copper foil



Unparalleled
domestic support



New access to
technology and
innovation



803-425-7900



denkaiamerica.com



sales@denkaiamerica.com

danger of using China as the sole supplier of many components.

Another issue concerning our offshore trade balance is the number of container ships sitting offshore in Southern California. I routinely see 20–30 cargo ships every week out on the water. Not only is the economic impact more visibly striking, but the delays in unloading these ships creates a supply chain problem.

Politics aside, I believe there is a slippery slope in the West if we don't encourage businesses to invest and bring some manufacturing home, particularly those that are high technology and/or military related. We are in an economic contest that we cannot afford to lose. I believe in free and fair trade where we all play and abide by the same rules. Those that don't should not enjoy the benefits of our market.

This article hasn't even addressed the issue of industrial espionage, reverse engineering, disregard for intellectual property, spying, and the alleged implanting of devices that in some fashion are able to keep track and disseminate private and confidential information. Proper business ethics and regard for human values are paramount.

Certainly I am not qualified to address this complex subject with the same vast knowledge of others, but at my age, I have seen how we as a country have placed profit over the issue of fair trade. We must address excessive regulations, tax issues, and investment encouragement to allow our manufacturing industry to compete fairly and more competitively. **PCB007**

Richard Crowe is chairman of Burkle North America.



Just Ask Michael Coll

Denka America



Q How long is the copper shortage anticipated to last?

A I think that this is going to be a tough question for anyone to answer. We do not really see a “shortage” in the market, but rather a tightening of supply as copper prices continue to increase. ED foil is made from scrap copper wire, which is still plentiful in the U.S. I cannot comment on the availability

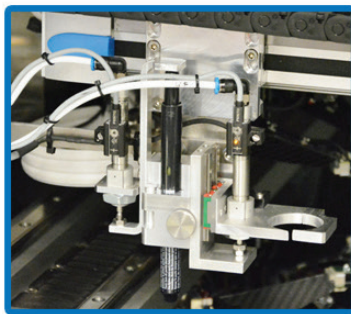
in other regions. The U.S. exports scrap wire to Asia so the ongoing delays in ocean freight may be contributing to some additional delays in the foreign markets. This may impact copper cathode availability as well. **PCB007**

Michael Coll is COO at Denka America Inc.

Unrivalled Test Speed with Full Automation



Introducing the newly designed atg A8a with 8 test probes and a new high speed “lights out” automation for unrivalled throughput.



Highlights:

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

Watch video

Get more info

atg Luther & Maelzer GmbH

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





Just Ask Kim Sauer

Ventec International Group



Q Is it possible to use another material as a substitute for heavy copper with different characteristics and the same performance?

(For example: Instead of using FR-4 with 4-ounce base copper, use a material with other characteristics with 1- or 2-ounce copper.)

A The simple answer is a guarded yes. Copper is usually chosen for its superior all-around characteristics compared to other metals: product designers appreciate qualities such as its low resistivity, easy formability, broad compatibility with other processes and materials, stability over time and temperature, low cost, and relatively easy availability, to name a few. In practice, alternatives are unlikely to deliver the advantages designers are usually looking for: better performance, smaller size, lower power, lower cost.

If you want to design your PCB with 1- or 2-ounce copper instead of 4 ounces, the obvious solution is to use wider conductors. The price is that a larger PCB area is required, which may not be an acceptable compromise.

Obviously, copper is not the only conductor available to electronics designers. An alternative may be sought to overcome various issues, either commercial or technical. These may include price and availability, or there may be a requirement for certain mechanical properties or corrosion resistance. In some cases, the appearance of any visible conductors may be a consideration.

Regarding the design of PCB traces, in particular, the first aspect to consider is the resistivity, ρ (Greek, rho), of the chosen conductor.

This is a constant for any given material and equivalent to the resistance of a sample having unit cross-sectional area and length. Mathematically, $\rho = RA/l$ (R = resistance; A = cross-sectional area; l = length), expressed in MKS units as Ohm-meter (sometimes Ohm-centimeter). The resistivity of copper is 1.77×10^{-6} ohm-centimeter.

From this, $R = \rho l/A$, which suggests that you could replace a 4-ounce copper conductor with 1-ounce copper of four times the width. Clearly, this would occupy more space on the PCB. You could consider changing the conductive material. However, copper is difficult to beat when everything is considered, such as cost, environmental characteristics, and compatibility with other electronic materials and manufacturing processes.

Aluminum is sometimes used in applications such as large inverters and power converters in e-mobility and green energy applications. Aluminum, at around half the price or less per kilo than copper, can be more economical when very large conductors are required. However, the resistivity is higher than that of copper, hence demanding a larger cross-section—effectively, thicker or wider traces—to achieve the same electrical performance.

You also need to consider the effect of

INNOVATION + PRECISION + RELIABILITY



IN-HOUSE CUSTOMER LAB

Available to assist every Chemcut customer with:

- R&D process development
- Producing an initial prototype
- Small pilot production runs
- Feasibility studies
- Producing scale-up data
- Operator training and education



Contact Matt



To learn more about our lab, contact Matt Erlichman
at merlichman@chemcut.net

www.chemcut.net

 **CHEMCUT**
BOUNDLESS INNOVATION | UNBEATABLE PRECISION

temperature on resistance. Reducing the temperature allows the conductor to carry more current in relation to the cross-sectional area. However, there is self-heating proportional to I^2R . It is normal to use a lookup table of acceptable temperature increase related to current and conductor area.

The use of special thermally conductive dielectrics such as Ventec VT-5A2 or VT-4A2H could help to reduce the temperature and increase your traces' maximum current capability. Going a step further, IMS materials could

also extend the window by helping with heat dissipation. However, this is not always a suitable solution.

A serious shortage of copper, for any reason, could change things by driving up prices. Then you need to consider the cost and practicalities of introducing the alternative material to your manufacturing processes, and how soon the copper price may return to normal. **PCB007**

Kim Sauer is global marketing communications of Ventec International Group.



Just Ask Chuck Bauer

TechLead Corporation



Q Beyond solder, what's the next best interconnection between the components and the PCB?

A I would like to add a brief comment regarding one additional option available, albeit on a limited basis and for specific application areas. Herb Neuhaus and I developed a technique based on nano-particle thermo-compression to mount optical components with very high I/O counts on pitches as fine as 12–15 microns. In our program we employed various nano-Ag compositions developed/targeted for die attach applications, the most useful of which was a formulation from Nihon Superior out of Osaka, Japan. They did do some customization of the particle size mix and distribution, but the process is quite stable in these high pin count applications.

While our approach was developed and deployed specifically for mounting bare chips

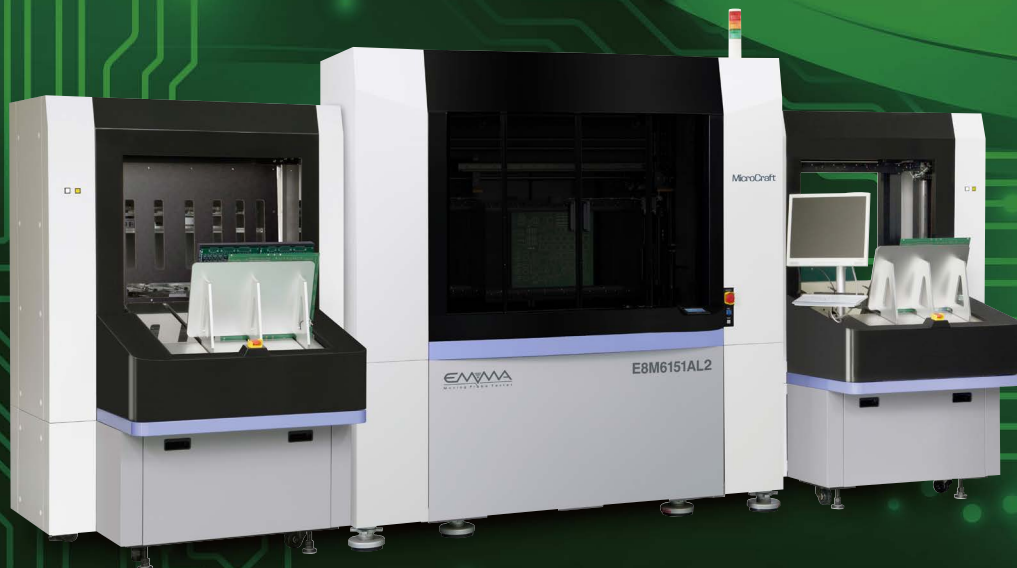
onto ceramic substrates in these applications, Thomas Brunschweiler at IBM Zurich picked up on the idea and extended it to PWB assembly employing Cu nano-particle formulations developed in Europe. I have not spoken with Thomas recently, but he did present a keynote talk at the Pan Pacific Microelectronics Symposium in 2017 in which he briefly described their work. I believe that the work was completed in late 2018 or early 2019, but I have not followed up or searched for any publications. I do know that the work was part of a doctoral dissertation for a student being advised by Thomas. **PCB007**

Charles Bauer works for TechLead Corporation in Portland, Oregon.

Fully Automated Unattended Operation

Over 1,900
Machines Installed
Worldwide
(as of March 2020)

EMMA Series



EMMA
Moving Probe Tester

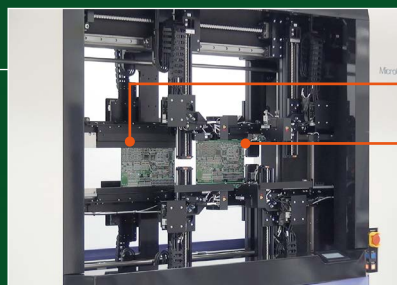
10,000
Test Points/mins.

Load up to
200 pcs

E8M6151AL2

Increased Throughput

Transfer speed reduced by **37%**

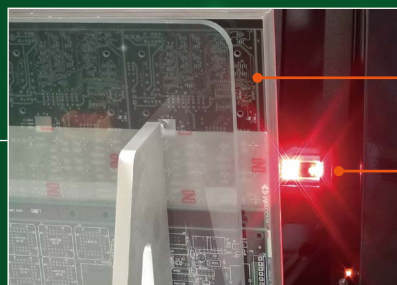


Ready for next test

Being tested

Built-in Scanner for Seamless Job Changeover

Improved Efficiency for
high-mix boards



Boards stacked
vertically for easy
operator access

Bar & QR
Code Reader



USA MicroCraft, Inc.

1247 N Tustin Ave, Anaheim, CA 92807
TEL (714) 482-2550

www.usamicrocraft.com
sales@usamicrocraft.com

Visit Our Website

Watch Video



Just Ask Joe Fjelstad

Verdant Electronics



Q Beyond solder, what's the next best interconnection between the components and the PCB?

A This is a relatively simple question but one which conjures up some technically diverse potential answers, each of them necessarily weighted by several qualifying questions to make certain they will meet the end product's requirements. This is a seemingly small but very important detail and thus precludes the notion of being able to offer up a universal "next best" solution.

That said, there are several historical choices for making electrical connections between components and PCBs without relying on traditional solder. The most common is arguably a conductive adhesive, both isotropic and anisotropic types, which have been in use in electronics for many years. Their limitations are that they are generally less conductive and do not generally provide the kind of bond strength that solder does; thus, they may not be as resistant to shock and vibration as solder. The components are often "glob topped" to hold them in place and this, unfortunately, makes rework problematic and makes them more susceptible to overheating in operation.

More recent years have seen the rise of low temperature sinterable alloys enabled by nanotechnology. Included among these is a nano-copper "solder" from Kuprion. These show promise and are finding some traction, but wide acceptance is not yet here.

Another historical method that has been largely reserved for connectors is press fitting of leads into plated through-holes. When intelligently designed and assembled, press-fit connections can be quite reliable and provide a gas-tight seal between lead and component. Obviously, the method precludes the use of most SMT devices.

Socketing of components is another method

of interconnecting components to a printed circuit without solder but sockets add expense and typically are soldered to the board themselves.

A method I have been promoting for more than a dozen years through my company, Verdant Electronics, is solder alloy-free electronic (SAFE) assembly which seeks to obviate the need for solder. This objective is accomplished using a novel reverse approach to manufacturing where components are first attached to a carrier with planar leads facing up and circuits are built up by directly plating copper to lead terminations using familiar printed circuit manufacturing processes. The elimination of solder precludes the need for numerous process steps.

Concerns have been expressed about the need for rework and repair, but it is arguable that the need for rework and repair is evidence of a process that has intrinsic control issues, making rework and repair a necessity. My credo has evolved to, "If we seek to make the best products possible, we must first do all the right things and then do all those things right." For more detail, please read my May 2021 column and/or my free I-007 eBook, *Solderless Assembly For Electronics—The SAFE Approach*. **PCB007**

Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, [click here](#). Download your free copy of Fjelstad's book *Flexible Circuit Technology, 4th Edition*, and watch his in-depth workshop series "Flexible Circuit Technology."

Excellon

*Manufacturers of precision Micro-Machining
Solutions for over 50 Years*



COBRA II Hybrid Laser

*UV and CO2 lasers
Blind via, flex cutting, cavities*



HS-4L

*Low-cost, 4-station precision drilling
or routing system with large panel option*



HS-1L DS

*Single-station, dual spindle high-speed
drilling/routing with Intelli-Drill Vision*



136L

*High-Speed, precise accuracy
drilling/routing with Intelli-Drill Vision*



154L

*Large panel size precision drilling/routing
Single station with vision options*



HS-2L

*High-speed, 2-station drill or router
w/populated board routing option*

**Move into the future with the latest innovations
in Fabrication Technology from the Industry Leader!**



Just Ask Leo Lambert

EPTAC Corporation



Q What channels are most PCB assemblers using to educate themselves on the latest process improvements and equipment?

A The opportunity for education on process improvements requires an open mind as to the avenues available and the support of management to offer those opportunities to their employees. Sounds simplistic enough, however, the resources need to be sought out for the appropriate programs as to whether they are virtual, online, self-paced, specific classes, on-site, or attended at various locational training centers.

Process improvements come in many forms, whether it's learning about newly acquired equipment, or understanding the software and hardware operational specifications of the equipment. Process improvements also include the introduction of new materials, components, and designs while also working with the engineering staff as to what is needed to provide the ability to manufacture a quality product within a given period of time. At times, the industry identifies these as manufacturability meetings where all aspects of a product and process are discussed. Basically, the goal is to review cost and return on investment on the value of the product, which includes materials, equipment, staff and space to create a process to manufacture the product, be it in-house or subcontracted.

Once the design is complete and ready for release to manufacturing, industrial standards are adapted to control the manufacturing technology. Additionally, this includes defining how the contracts and master drawings are to be written to include the manufacturing speci-

fications to be used.

At this point, PCB assemblers have a variety of options for training their employees in both skills and knowledge-based programs. IPC has many online programs available to introduce the operators to the electronic assembly operations, covering items from component identification to assembly techniques and quality requirements. These documents are adapted for internal usage for skills and knowledge development and internal training programs.

Many of the manufacturers are mandated by contract that the employees building their products are to be certified to various manufacturing specifications. They want the employees building their product to know what they are doing prior to initiating the contract.

These certification classes are available through various training centers around the globe and are offered as in-house programs, on-line programs with an instructor, or at a training site with an instructor. These instructor-based programs are the most efficient methods of imparting the information, as they offer the opportunity to have question and answer periods during the sessions to explain, clarify, and emphasize certain points of importance.

Increasing the knowledge base of the employees is mandatory in the ever-changing electronics assembly environment. **PCB007**

Leo Lambert is vice president of EPTAC Corp.

WORLD-CLASS SUPPLIERS

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

CALL 949.587.3200 FOR MORE INFORMATION



REVOLUTIONARY LAMINATING & BONDING TECHNOLOGY

X-PRESS

RFX



230N

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

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

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

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

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

230N - Next-generation inductive bonding machine



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



Just Ask!



Yaad Eliya and Eran Lipp
PCB Technologies



Q How can PCB fabricators achieve finer lines and spaces using current moderately-priced substrates?

A Indeed, reaching fine line technology below $\sim 2/2$ mil L/S is a nontrivial task, since reliable and robust etching of under-2-mil lines requires well controlled copper thickness and small variations within a board as well as lot-to-lot. High-volume PCB manufacturers, specifically those in the Far East, have the production volume and technological need to go as low as even $5\text{-}\mu\text{m}$ lines, and choose to transfer their process into PVD-based (physical vapor deposition) copper coating. In this process, which is known as semi-additive processing (SAP), copper thickness is well controlled in a process borrowed from the semiconductor industry. After etching of the thin PVD-coated copper, the final copper thickness can be reached by electroplating.

This method, although reliable, involves significant investments that may not be justifiable for low-volume or specialty manufacturers like PCB Technologies. For those companies, the demand for sub-1-mil technology is not that pressing. To reach the 1-mil line/space ballpark, subtractive approaches are adequate, although they pose several limitations:

1. Keeping copper thickness to the lowest values possible. Copper thickness above $\sim 20\text{ }\mu\text{m}$ will not enable subtractive

processing. Average thickness between lots should be maintained to variations below $1\text{ }\mu\text{m}$.

2. Keeping copper thickness variation to values below about $\pm 1\text{ }\mu\text{m}$ across the board.
3. Applying appropriate thin resists for line etching, resist thickness, etc., should match the required etched thickness. The resist should be thick enough to protect the desired lines, yet thin enough to enable stable lithography and prevent high aspect ratio during etching.

To support these requirements, special attention should be given to copper plating stages. Lowering variation at these processes may require thinking about agitation, throwing power, and anode shapes as well as usage of current thieves. In addition, one should be aware of copper etching processes along the process flow. Any copper etching process will adversely affect the total variation and will result in poor L/S definition. **PCB007**

Eran Lipp is head of R&D and
Yaad Eliya is CTO at PCB Technologies.

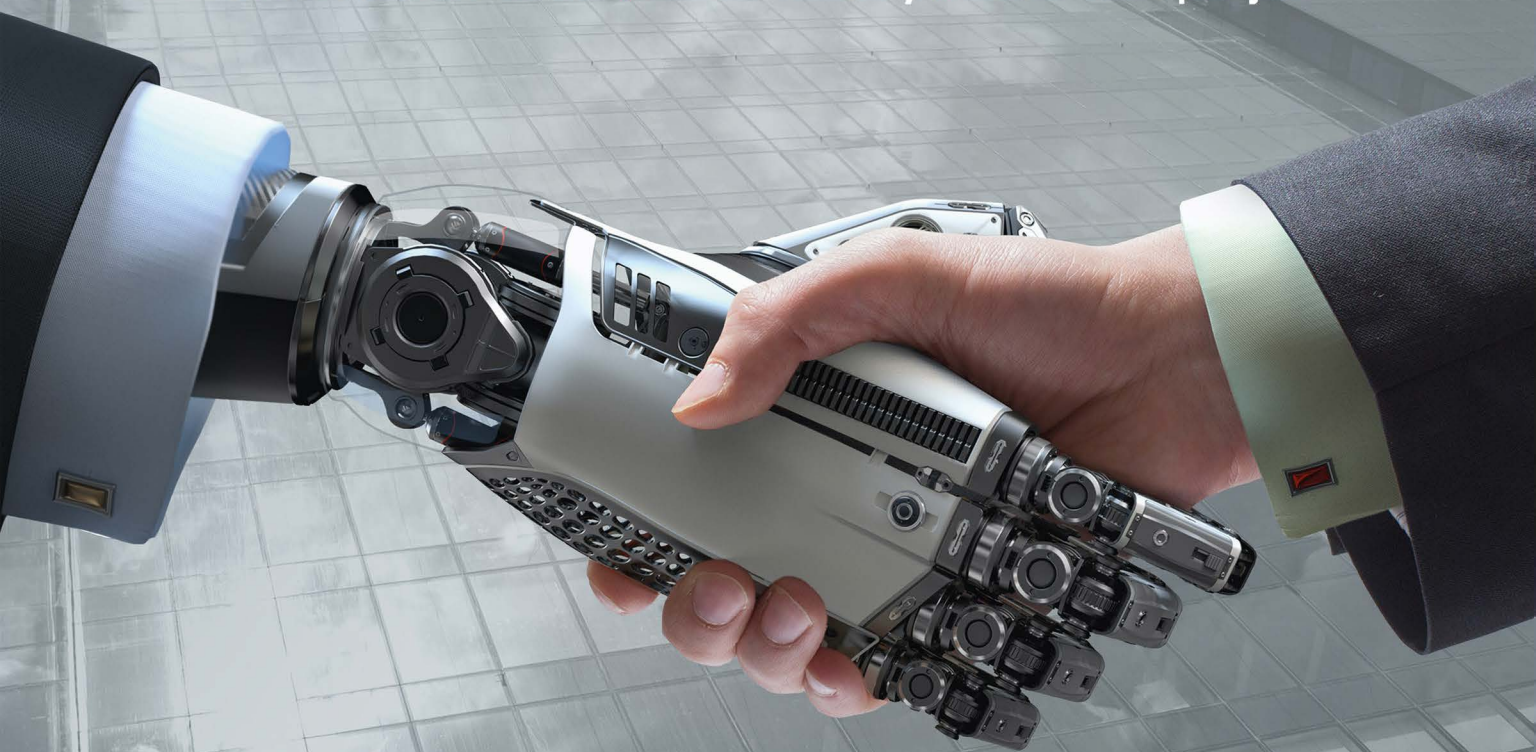


Market leader in
PCB CAM and Pre-CAM Software
Laser Photoplotters
Direct Imaging Systems
www.ucamco.com

iamcam

Intelligence Aided Manufacturing

Artificial intelligence | Next level of automation | Global web access
Fewer human errors | Faster delivery | Lower cost per job



AI automation shortens
delivery times

Free your brain resources
for complex jobs

iamcam.ucamco.com



Just Ask Dan Beaulieu

DB Management Group



Q What marketing techniques are best suited to engage with PCB assembly engineers in 2021/2022?

A Marketing your company is like planting seeds in your garden—with time and care, your business will grow and flourish. The most successful marketers generally find that a strategy combining consistent branding and content marketing is the most effective way to engage with their target market.

Engineers today are starved for information. Whether you are providing young engineers with very basic PCB 101 information or appealing to experienced engineers with information about cutting edge technology, engineers and designers are always looking for accurate technical information.

Here are some of the most effective ways to engage with engineers:

Technical articles and white papers: Nothing shouts “technology expert” louder than a groundbreaking paper or article about your specific technology. Be sure to present them at industry trade shows and conferences as well.

Branding ads: Even if your company is already well-known, daily branding to your target market reinforces your strength and position in the context of people’s daily work lives. Spend time developing your messaging. Avoid tropes like, “on time delivery” and “excellent quality”; create unique, memorable messaging that is consistent with your brand identity.

Monthly technical columns: Publish a technical column in a respected industry publica-

tion. Writing a good, solid, interesting, and informative monthly column highlighting your expertise in a specific technology will put you at the top of the list of experts in your field.

Technical expert pages on your website: Your website should be where people come to learn about technology. There should be individual pages for each of your technologies. Promote your engineers as “experts” in their specific technology. Include links to your columns, articles, and white papers. Include live discussion resources. Make your website a true resource for engineers to come and learn.

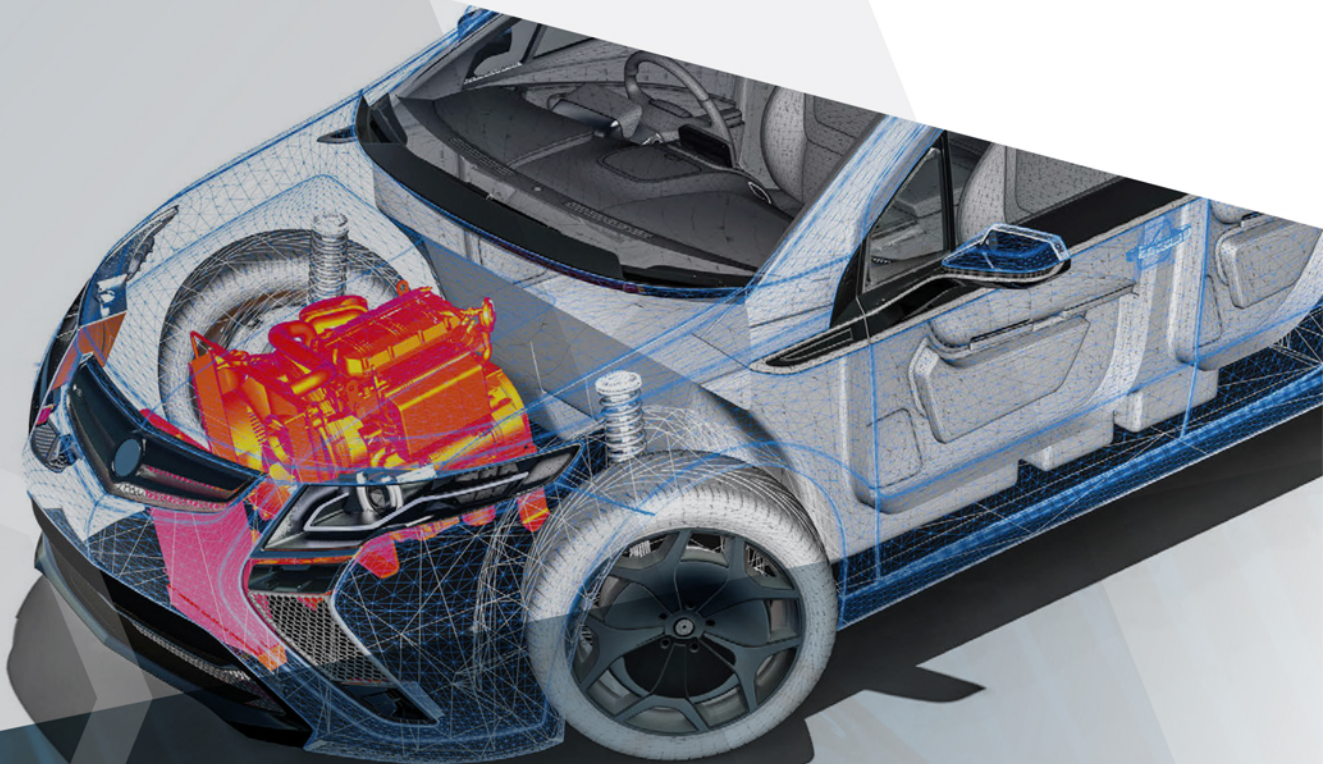
Webinars: Using Zoom and other conferencing tools, many companies are providing informative webinars to audiences eager to learn. How much time, money, and effort would it take to show customers your presentation the old-fashioned way, one at a time?

I-007eBooks: The most effective way to promote your company as the true expert in your field is through I-Connect007’s book program. Writing and publishing one of these targeted technology books will put your company and your experts at the very top of the technology ladder. Nothing gives your company more credibility than a third-party published book, especially with I-Connect007. They put their mighty editing and promotional powers behind the packaging and distribution of your book. Everything about this unique program



ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam 

Automotive electronics technologies are evolving at an increasing rate. Paying attention to the properties of materials at the substrate level is the first step towards achieving the most stringent performance targets of today's automotive manufacturers. autolam offers the solutions demanded by the diverse and unique requirements of automotive applications today and in the future.



venteclaminates.com

is designed to get your name in front of the right customers. One of my clients has written and published three of these informative books and, as a result, they are marketing to a database of over 11,000 qualified, content-hungry, grateful engineers. These are all engineers who signed up for the free downloads of these books and voluntarily gave up their contact information. My client is now considered the expert and industry leader for each of the technologies their books cover. Best of all, they are now furnishing these 11,000 engineers with sought-after technical information. Their business has

grown a whopping 75% since they wrote their first I-007 eBook two years ago.

The best and most efficient way a PCB company can market to engineers is by finding ways to show off their technological knowledge, experience, and prowess. Try it for yourself... it works. **PCB007**

For more information on the above-mentioned programs, visit I-007e.com.

Dan Beaulieu is president of D.B. Management Group. To contact Beaulieu, or read past columns [click here](#).

BOOK REVIEW

Dan's Biz Bookshelf: Winning the War for Talent

by Dan Beaulieu

Winning the War for Talent: Recruit, Retain, and Develop the Talent Your Business Needs to Survive and Thrive

In the first chapter of his book on hiring, author Chris Czarnik discusses the differences between boomers and millennials: boomers wanted security while millennials want purpose and social consciousness. Boomers wanted to support their family while millennials want work that they are passionate about. Back when the boomers were first looking for jobs, they were at the severe disadvantage of there being too many of them and too few jobs; today, there are too many jobs and too few millennials.

These details are combined with what the author calls the "Silver Tsunami." Boomers are aging out, which has caused the current issues of bringing new young people not only into our industries but into our companies as well. This is what we are facing today, and this book answers what we can do about it.

Czarnik rightfully points out that our HR departments must be in "sales mode"; they must be out there selling to young people about joining their companies. They have to make positions in their companies and overall industries appealing to the people they are trying to hire. He suggests that we create a "Target Employee Profile" describing exactly the right people we want to hire.

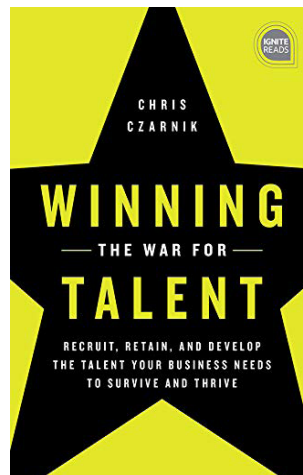
This profile must include not only the job description but the things the ideal candidate is interested in outside of the company. Companies must develop a career profile to show the right candidates not only what the job will entail but where it will lead as well.

The author completes the book with a detailed description of what must be done to develop the next generation of leaders for the future.

This is a very important and timely book. It is perfect for the time we are experiencing as we struggle to replace our aging work force with a new wave of vibrant, educated, talented, and passionate young people.

This book is a valuable handbook for building the future of your company. Buy it and read it. It will be the best investment you've made this year.

[Find on Amazon.com.](#)





Meetings & Courses

January 22-27

Conference & Exhibition

January 25-27

San Diego Convention Center | CA

Achieve **Digital Transcendence**

Get ready to transcend the norm as you transform through education, innovation and collaboration at North America's largest gathering of electronics industry professionals.

See you in San Diego at IPC APEX EXPO 2022!

IPCAPEXEXPO.ORG | [#IPCAPEXEXPO](https://twitter.com/IPCAPEXEXPO)



COLUMN

The PCB Norsemen:

John Steinar “Josse” Johnsen, Elmatica



A Simple Question Can Save You Time and Money

The PCB Norsemen

Feature Column by John Steinar Johnsen, ELMATICA

A PCB is a component like any other electronic component, except for the fact that it is not on the shelf waiting for you to buy it, but will be produced from scratch when you order it. In my eyes, it's the most important component, as it is the carrier or foundation in any electrical product connecting most other components. With this responsibility on “its” shoulders you might want to go that extra round before production, to make sure all data is correct and the design feasible. Poor design might lead to poor connection and failures—not the result you want.

Questions about PCB design are a “never-ending story.” Strangely, after years of PCB production and development, lots of the questions and issues are still the same, just only affected by the changes related to PCB technology.

Rather Too Much Than Too Little

PCB designers and engineers who are not frequently designing might find it not only helpful, but crucial, to seek help one time too many rather than too little.

The consequences and costs of not asking

can be high. My experience is that the earlier involvement, the better the product. When possible, involve the right team from when the idea for a product or printed circuit is hatched. When you have the right knowledge of PCBs onboard, you can make decisions based on knowledge and experience. This helps make sure the PCB process is as flawless as possible.

Get Another Set of Eyes

Design reviews allow others to check the features and function of the PCB design and inspect the interconnection of the various circuits. Don't assume that everything is fine until you've had someone double-check your work. When a qualified official design review is performed, you might discover errors early in the process—ones that you might not have discovered until later in the production process, or even worse, not discovered at all.

Communicate and Look Outside Your Box

PCB design is a long and collaborative process, but when engineers get so focused on



pluritec



Surface Preparation Machines for Printed Circuit Boards

PUMEX/PUMIFLEX

Mechanical cleaning (Brush - Pumice or Aluminum Oxide)



MECETCHBOND

Chemical cleaning, supports all industry supplied or in-house created blends.



**Don't settle for less – make IS your trusted partner. 40 years of experience
with mechanical and chemical surface preparation ...and counting!**



www.pluritec.com | info@pluritec.com

their part of the puzzle and forget to share information, errors and frustration might develop. Designers should be careful to communicate constantly, and above all, not just share thoughts and improvements with each other, but include every member of the project.

If you can avoid these common mistakes of narrowing down the involvement, you might be able to shave time and money off of your PCB design, and generate a more consistently high-quality project with a clear path from design to production.

These initial basic phases are usually related to function, technology, choice of materials, organization of layers and stackup, and electrical design requirements. The cost factor is not to be forgotten either. I have mentioned all this earlier in various contexts, and I am sure I will mention it hundreds of times in the future.

Search for Mistakes Like the Eagle Searches for Prey

However, here's a small sidestep focusing on the fact that when you as a designer have placed the last connection on a design, successfully ran a design rule check (DRC) without any error messages, and your design is still not optimized for production.

I have explained this numerous times. At this phase in the project, it's time to bring out the eagle eyes and study your masterpiece. Search for improvements and mistakes, as the eagle searches for its prey, to clean up your layout. This clean-up aims to improve the design for production and thus also improve the board itself and provide a better production yield. Better yields save time and money.

As a designer, it is important to know the most critical processes, and understand the consequences of your choices at each step, as these are crucial to obtain the best possible conditions for a successful PCB production. It will benefit and generate a more consistently high-quality project.

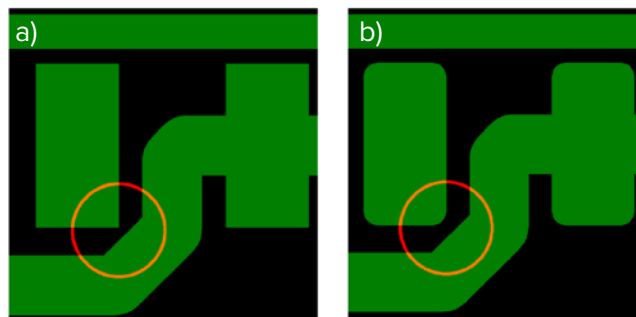


Figure 1: a) Rectangular pad;
b) Same pad with rounded corners.

Here Are Some of My Related Tips

Some of these choices may be in your CAD software library. For example, how you have defined SMT pads in your pad-stack? I recommend that square/rectangular pads shall be defined with rounded corners (Figure 1). It is electrically good, as well as beneficial for PCB production, assembly, and soldering.

What to Look Out For

Improved Routing—How?

Improved routing by increasing the distance between conductors, vias, and pads where possible. Move the connections so they are centered (Figure 2). Figure 3 shows more samples where it is possible to increase distance between pads and traces, also between differential pairs. In all the samples, traces are too close to the mechanical drilled hole.

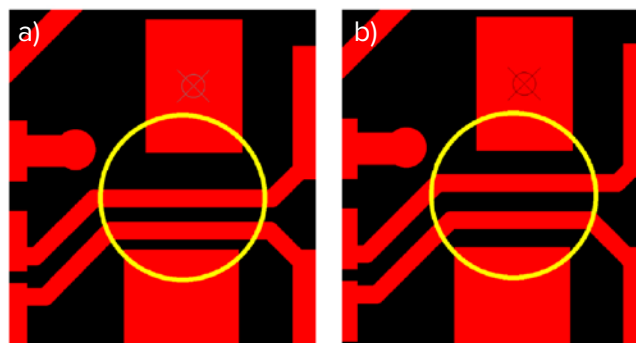


Figure 2: a) Before moving the traces;
b) After moving the traces.

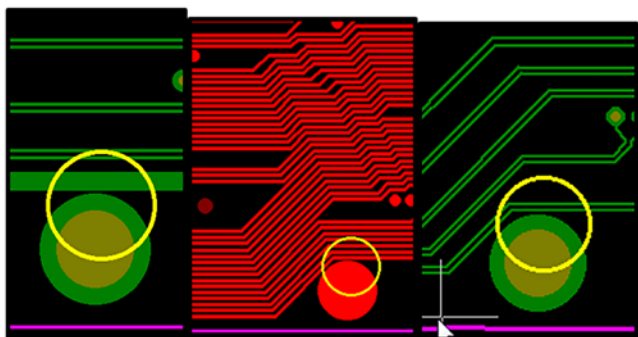


Figure 3: Samples where it's possible to increase distance between pads and traces, also between differential pairs.

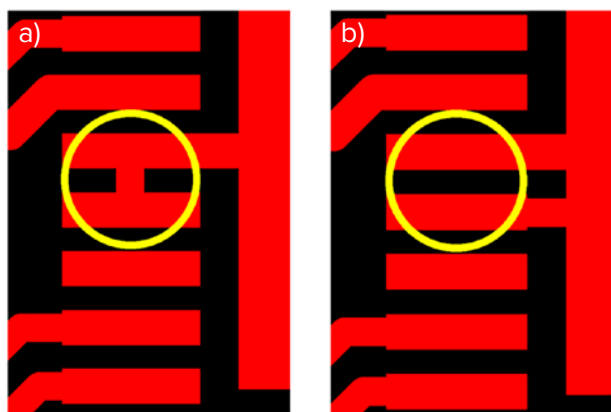


Figure 4: a) Do not short pads like this; b) This is the way to do it.

How Do You Connect Neighboring Pads?

It should not look like a short circuit on the finished board (Figure 4).

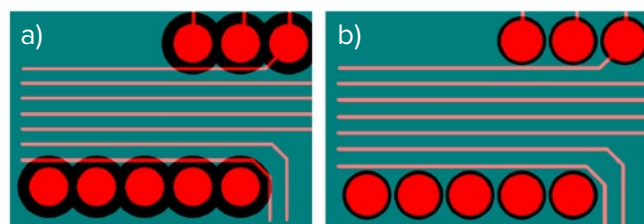


Figure 6: a) Signals are missing the reference plane; b) Signals have the reference plane.

Prevent Unwanted Flow of Solder Paste During Soldering/Vias That Are Not Covered With A Solder Mask

Move the via further away from the SMD pad, so you ensure that there is room for a solder mask in between the SMD pad and via pad. An alternative is to reduce the solder mask opening to be slightly bigger than the drilled via hole. Consult the IPC document IPC-4761 Design Guide for Protection of Printed Board Via Structures, to find the solution for your design. Or just ask.

Critical Signals That Require a Reference Plan

Make sure the connections are sufficiently far from neighboring pads so that they really get the desired reference copper plane below or above them (Figure 6).

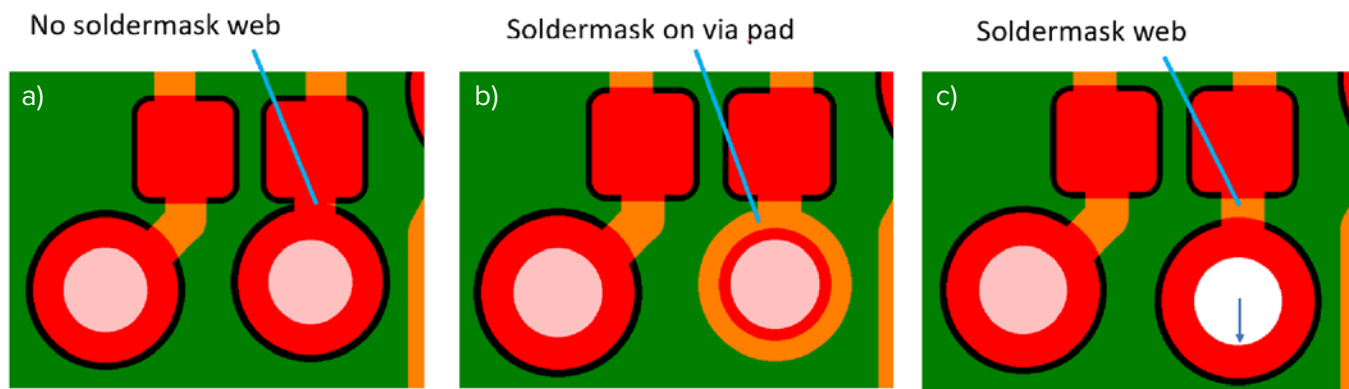


Figure 5: a) No solder mask web between SMT pad and via-pad. Solder paste will flow into via hole; b) Solder mask opening has been reduced to cover via pad, but is slightly bigger than the via hole; c) Via hole has been moved away from the SMT pad, so solder mask web can be kept.

Uneven—Unbalanced Copper Distribution Can Give Bow and Twist

Layer-to-layer copper coverage: Within several layers causing low pressure areas through the bonding process of the board. In Figure 7, all features in red are balancing patterns on the inner layers. On this 14-layer board there were a total of eight nearly similar layers.

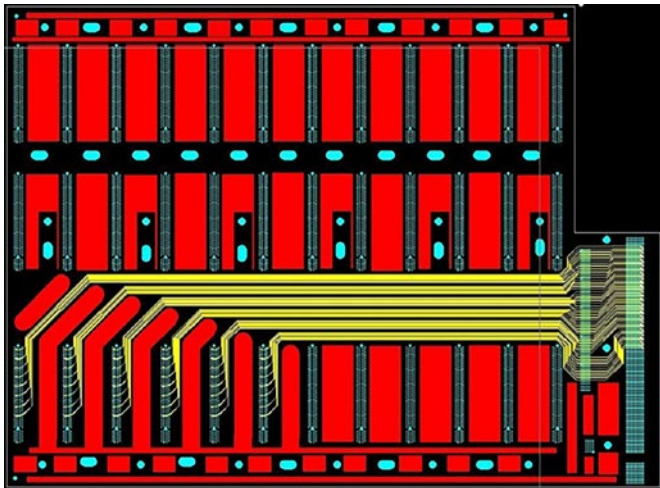


Figure 7: All features in red are balancing patterns on the inner layers.

Same Net Spacing

Electrically nothing is wrong with this (Figure 8), but during automatic optical inspection (AOI) this will cause problems and delay in production. In the sample, the same net distance is below 45 μm .

Unwanted Angles—Causing Acid Traps

In these sharp angles, chemicals from the processes can remain and cause reduced or broken connections over time (Figure 9).



Figure 8: In the sample, the same net distance is below 45 μm .

Some Parameters Can Be Set and Taken Care of During Post-Processing

Do you keep or remove unused pads and vias? There is no rule without exceptions, but my general recommendation is to keep pads on all layers for through-hole components and screw holes.

Allow unused via pads to be removed if there is no risk for low pressure area. Do not remove all unused via pads in local areas where there is a high proportion of vias, such as typical BGA areas. BGA areas may require higher temperatures during soldering. This can promote delamination.

Teardrops or Snowmen?

My general recommendation is to enable this feature.

I have been working with PCBs for decades. In the past year, I have held numerous seminars and webinars, talked to customers, improved designs, scrapped designs, helped students, made hundreds of drawings, explained the basics, experienced advanced technology and materials, and pushed the PCB design to its limits.



Figure 9: a) The original; b) The routing has changed a bit and the SMT corners have also been rounded. A possible problem solved.

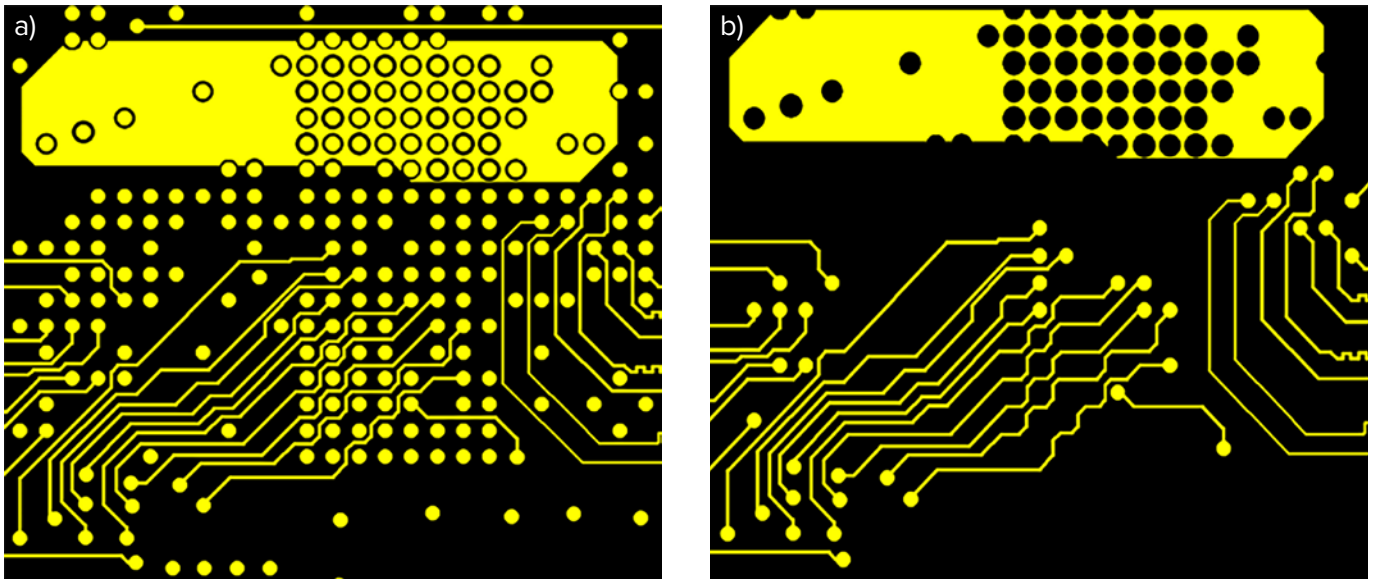


Figure 10: a) All via pads are kept; b) All non-functional via pads are gone.

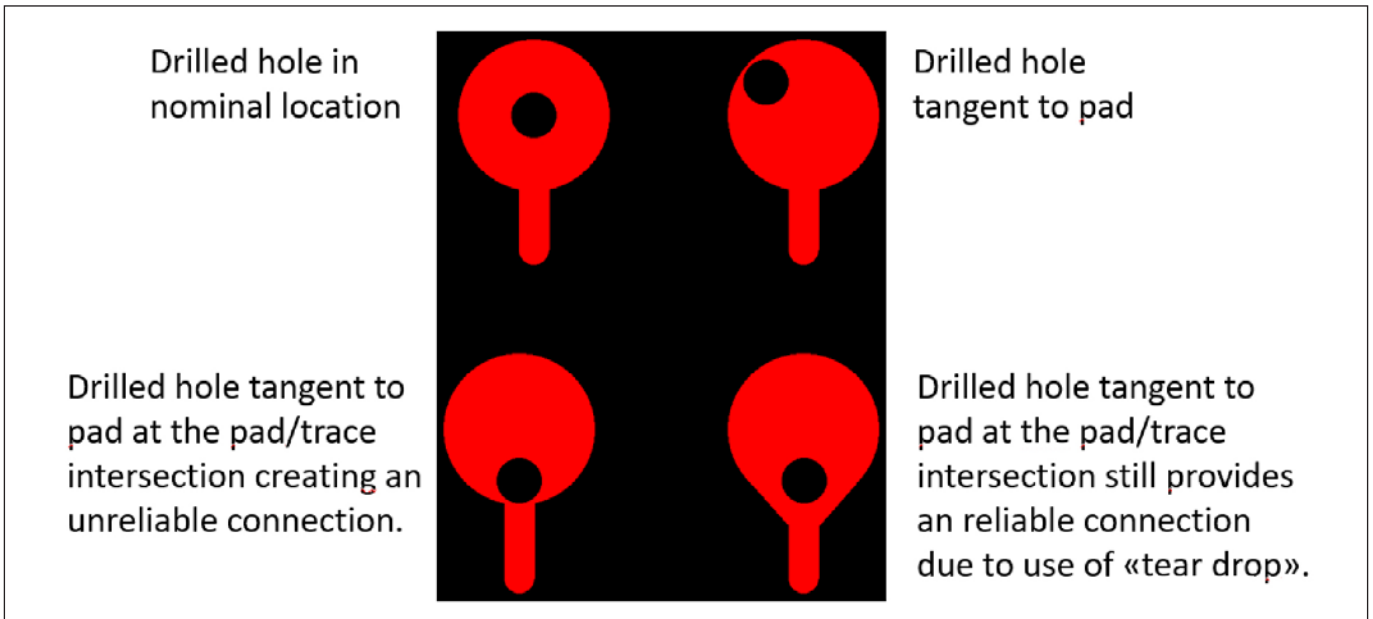


Figure 11: Enable this feature: teardrop or snowmen.

However, even as advanced as the technology might currently be, sometimes it's great to get back to the basics, ask the simple questions, and before thinking about the cool and advanced features, make sure that what you are thinking of designing, actually is designable. Just ask, and you will get an answer. And make sure the source is reliable, because that's what you expect from your PCB. **PCB007**



John Steinar Johnsen is senior technical advisor at Elmatica. To read past columns or contact The PCB Norsemen, [click here](#).

ein Electronics Industry News and Market Highlights



AIStorm's AI-in-Imager Solutions Use Tower Semiconductor's Hi-K VIA Capacitor Memory ▶

AIStorm and Tower Semiconductor announced that AIStorm's new AI-in-imager products will feature AIStorm's electron multiplication architecture and Tower's Hi-K VIA capacitor memory, instead of digital calculations, to perform AI computation at the pixel level.

WIMI Hologram Cloud's AI, AR Builds 5G Holographic Cloud Ecosystem ▶

MobiusTrend, the fintech market research organization, recently released a research report "WIMI Hologram Cloud's AI and AR Builds a 5G Holographic Cloud Ecosystem, Apple and Facebook Accelerate the Development of AR Devices on the Consumer Market."

Yageo, Hon Hai to Form Semiconductor Joint Venture, XSemi ▶

Yageo Group and Hon Hai Technology Group announced to enter into a joint venture agreement to form XSemi Corporation. The goal of this newly established joint venture is to extend the businesses into the semiconductor industry, including product development and sales.

Researchers Develop New Graphene-based Sensor Technology for Wearable Medical Devices ▶

Researchers at AMBER, the SFI Centre for Advanced Materials and BioEngineering Research, and from Trinity's School of Physics, have developed next-generation, graphene-based sensing technology using their innovative G-Putty material.

Fujitsu Signs Strategic Collaboration Agreement with AWS ▶

Fujitsu announced a Strategic Collaboration Agreement with Amazon Web Services, Inc. (AWS), an Amazon.com company, to accelerate the digital transformation (DX) of the mobility industry.

GENERA, Henkel to Drive End-use Production in Additive Manufacturing ▶

GENERA recently joined Henkel's Open Materials Platform with the intention of providing additive manufacturing solutions at production scale.

PsiQuantum, GLOBALFOUNDRIES to Build World's First Full-scale Quantum Computer ▶

PsiQuantum, the leading quantum computing company focused on delivering a 1 million-plus qubit quantum computer, and GLOBALFOUNDRIES (GF), a global leader in feature-rich semiconductor manufacturing, announced a major breakthrough in their partnership to build the world's first full-scale commercial quantum computer.

Silicon Wafer Shipments Edge Higher in Q1 2021 to Set New Record ▶

Worldwide silicon wafer area shipments increased 4% to 3,337 million square inches in the first quarter of 2021 compared to the fourth quarter of 2020, topping the previous historical high set in the third quarter of 2018, according to the SEMI Silicon Manufacturers Group (SMG) in its quarterly analysis of the silicon wafer industry.



STRIVE FOR **EXCELLENCE**

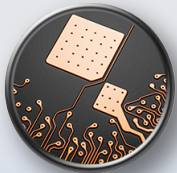
www.emctw.com | EMC.INFS@mail.emctw.com

Enable 800GbE Ethernet Speed with EM-892K and EM-892K2 Materials

Industry's First Halogen-free Extreme Low Loss (ELL) Laminates



 World's largest producer of environmentally-friendly advanced laminate materials



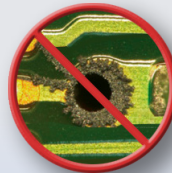
Key Material Features:

EM-892K @70%RC:
Dk: 2.84/ Df: 0.0017 @10GHz
EM-892K2 @70%RC:
Dk: 2.76/ Df: 0.0013 @10GHz
with Cavity Resonator Method



Lead-Free Soldering Test:

PASS LF-260°C/ 10X
with 32L 0.6mm hole pitch



Anti-CAF Test:

Pass 100V/ 85°C/ 85% RH
with 32L 0.6mm hole pitch
for 1K Hours



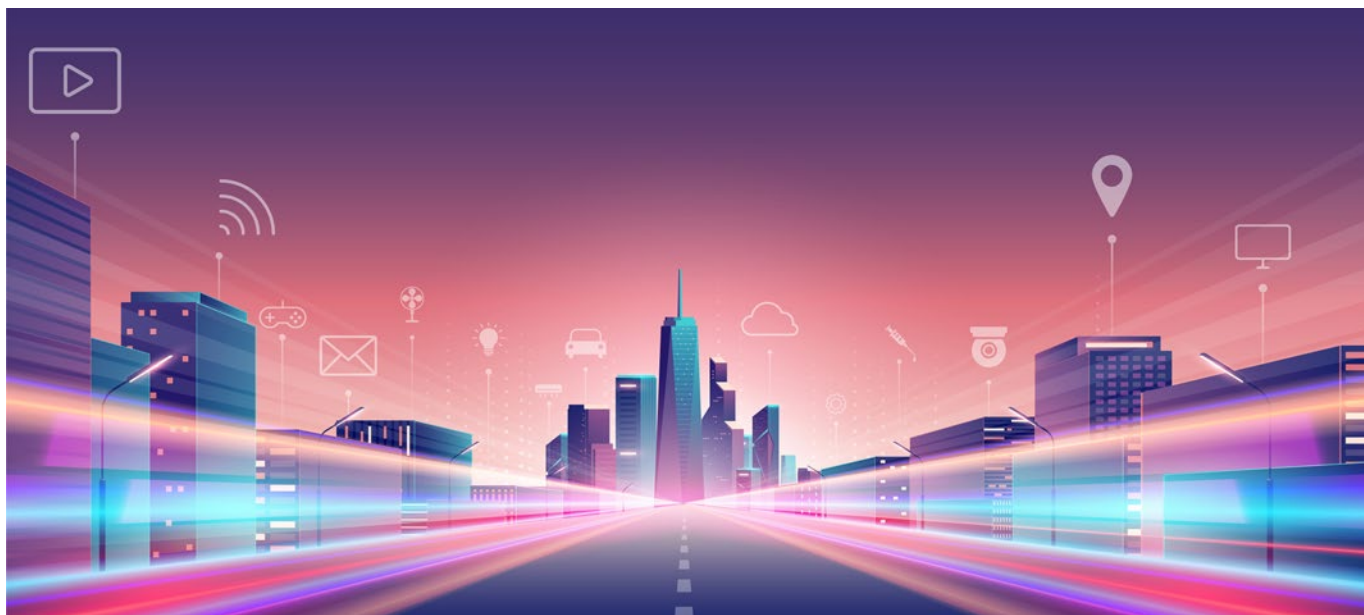
IST Test:

Pass Room temperature
to 150°C for 3K Cycles
with 32L 0.6mm hole pitch



TECHNICA, U.S.A.

North American Master Distributor
1-800-909-8697 • www.technica.com



TLPS Z-Axis Interconnect Solutions for **Thermal Transfer** and **Electrical Connection** in PCBs

Article by Catherine Shearer
and Gary Legerton

EMD ELECTRONICS, SAN DIEGO, CALIFORNIA

Abstract

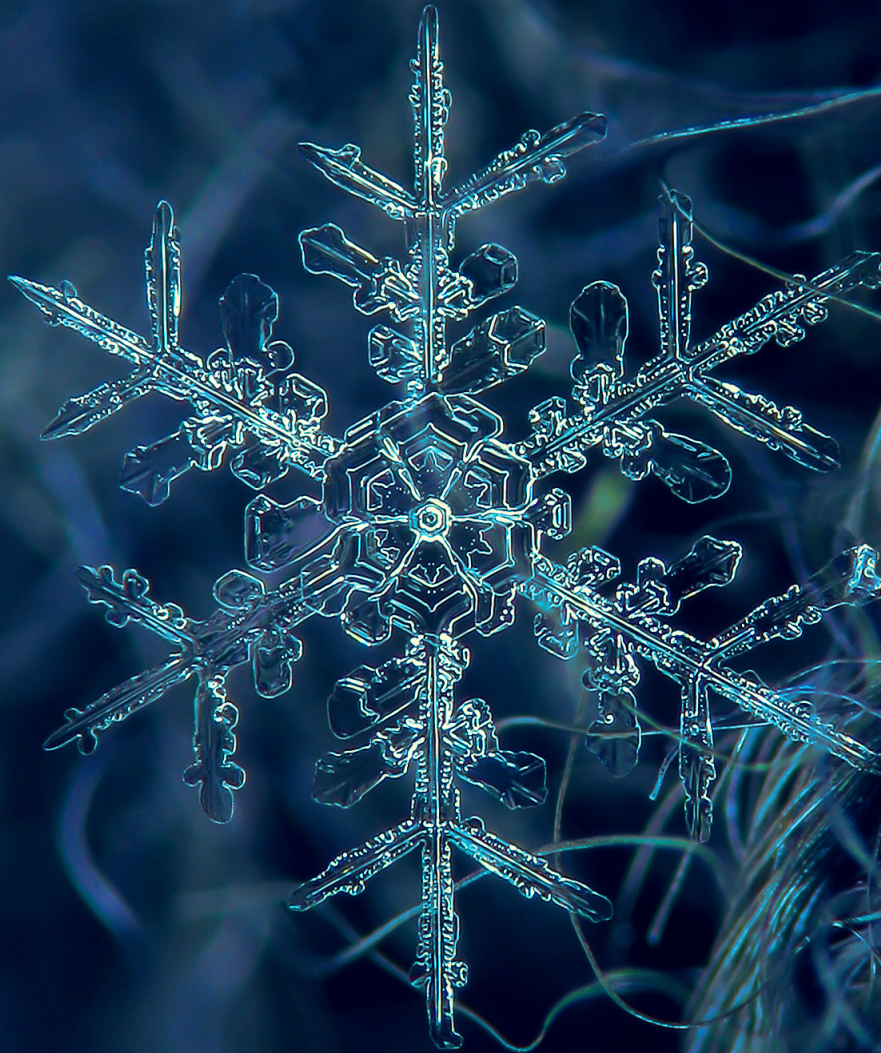
The electronic packaging industry is undergoing a revolutionary convergence between the printed circuit board segment and the semiconductor packaging segment. New, streamlined and hybrid package architectures are emerging to meet future product requirements—particularly for mobile electronics and infrastructure to support industry megatrends like 5G. There are new challenges for forming electrical interconnections between different types of package elements while maintaining high volume manufacturability and reliability.

In particular, the use of high speed and high frequency dielectric materials complicates the PCB fabrication process. Generally, these low-loss and low Dk materials are not amenable to multiple lamination cycles due to the nature of their chemistry; however, conventional PCB fabrication techniques that circumvent

this problem by using plated through-holes introduce undesirable resonance structures and consume precious real estate that could otherwise be used for routing.

Transient liquid phase sintering (TLPS) paste vias can be used to either augment or replace sequentially formed plated microvia interconnects, which necessitate multiple lamination cycles, as well as plated through-holes (PTHs) with their attendant loss of routing density and lossy stubs. TLPS-filled Z-axis interconnect layers can be fabricated in parallel with individual X-Y trace layers or PCB subconstructions of multiple layers with PTH, interleaved, and laminated in a single cycle. The circuit layers thus electrically joined through the Z-axis can be of the same or different materials, complexity, and native construction. The adhesive surrounding the TLPS interconnects and mechanically joining the circuit layers can be prepreg or film adhesive and be selected for its adhesive, dielectric, and mechanical characteristics. With an appropriate adhesive layer, the TLPS Z-axis interconnect concept is extendable to applications outside of PCB

Uniquely suited solutions for an increasingly demanding world.



Outsourcing can help you remove your CAM bottleneck. We deliver the highest quality PCB CAM engineering services to customers around the world.

Find out how we can help you >



www.entelechyglobal.com

construction including area array assembly and thermal transfer.

TLPS pastes, which metallurgically bond to circuit pads, offer both high performance and versatility of installation that is conducive to high manufacturing volumes. Because sintering pastes can be formulated with a variety of particle sizes and flow behavior, this technology can provide a spectrum solution to applications from filled microvias in either a printed circuit board or semiconductor package scale, to printed bumps for interconnection of sub-assemblies, to thermal interfaces with embedded heat sinks.

This paper will present the two most common implementation flows for the installation of the TLPS paste Z-axis interconnects in mixed mode PCB constructions.

Introduction

The production of high-layer-count PCBs for the telecommunications, semiconductor test, and high-end computing industries requires an advanced set of fabrication techniques due to the complexity of their design. Frequently, manufacture of these PCBs requires the use of blind and buried vias, high-aspect-ratio plated through-holes and backdrilling to meet their design specifications. The technical roadmap for industries using high-layer-count PCBs indicates that future PCB designs will be increasingly complex and there is a drive toward both higher interconnect density and maintaining signal integrity at high frequency.

The reliability demands on high layer count boards are also increasing due to the increasing power dissipation requirements of

telecom, high end computing, and military applications. This increase in thermal exposure over the life span of the PCB exacerbates the need for robust electrical interconnect structures throughout the board beyond just withstand of assembly operations.

With the drive to 0.3 mm pitch components, and the desire to route multiple circuit traces between copper pads, fabricators are being pushed to use smaller drill sizes and to plate ever higher aspect ratio holes. The high-aspect-ratio PTHs in these boards increase the cost of the final board as they are tricky to drill, as well as expensive and difficult to plate consistently.

One solution to the difficulty of plating the high-aspect-ratio PTH in these PCBs is to break the PCB into subassemblies or “cores” with manageable-sized PTHs. By breaking the PCB into cores of reasonable thickness, the PTHs of these cores can be plated in high quality, yield, and throughput. These cores may then be interconnected using paste interconnects to form a monolithic PCB. Figure 1 is a close-up view of a single TLPS paste intercon-

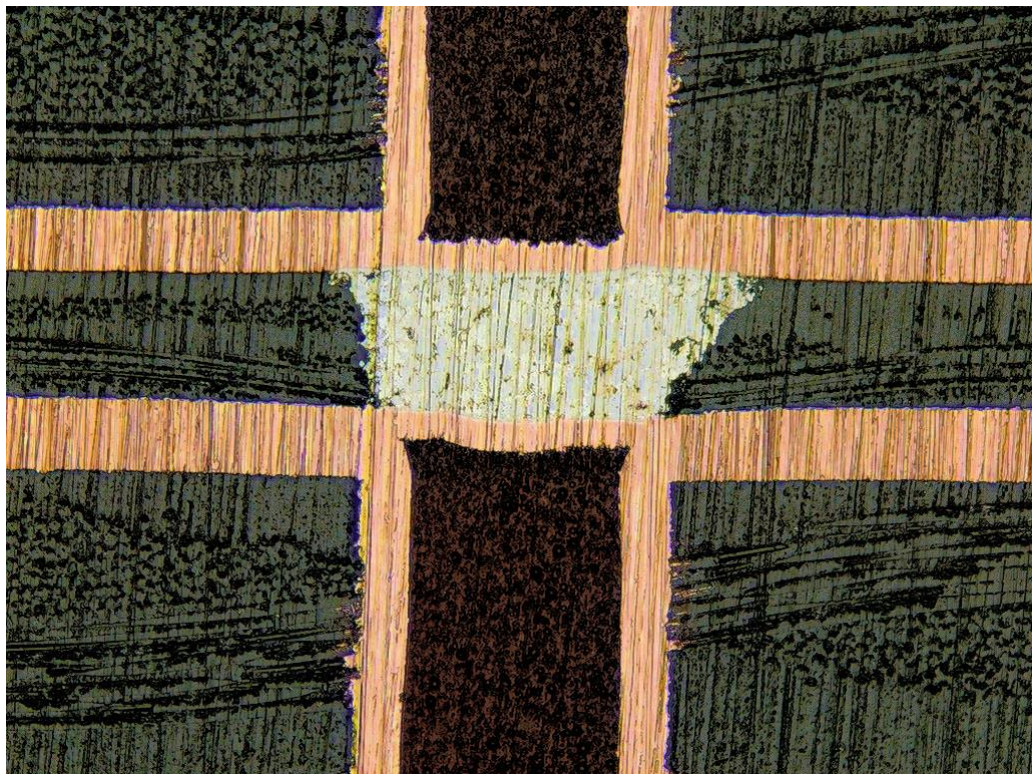


Figure 1: TLPS paste Z-axis interconnect after 6X lead-free reflows.

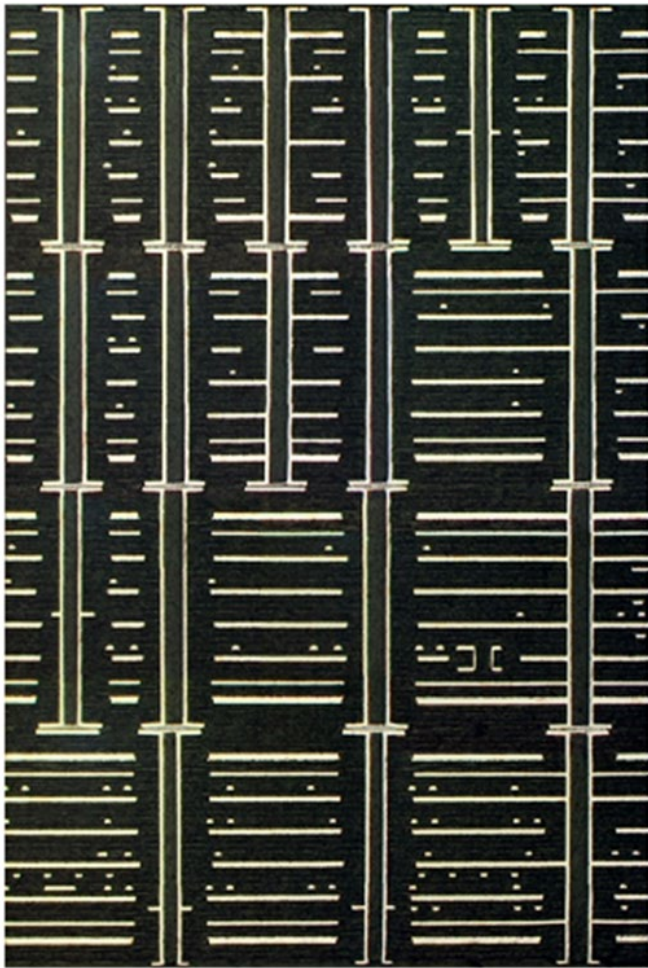


Figure 2: A high-layer-count PCB broken into four cores and joined with sintering paste interposers.
(Source: i3 Electronics)

nect between PTHs in two PCB subconstructions, and Figure 2 is a large-view cross-section of a 72-layer PCB that has been divided into four subconstructions, and joined with TLPS paste vias layers.

One of the important considerations when employing a subassembly core-to-core manufacturing strategy is determining an effective material for interconnecting the cores. Electrical performance, reliability, manufacturability, and cost are all critical factors.

Interposer sheets consisting of copper-paste-filled vias drilled or laser ablated into prepreg are one option for interconnection of the cores during a conventional lamination cycle. The problem with this solution is that non-sintering copper-filled epoxy pastes used in these

vias do not offer adequate electrical or reliability performance.

Sintering conductive pastes are a proven alternative technology solution that enables high performance and reliability in a core-to-core interconnection strategy for the manufacture of high-layer-count, high-speed and high-frequency PCBs.

Sintering Conductive Paste vs. Conventional Conductive Paste

Conventional conductive epoxy materials are thermosetting or thermoplastic resins loaded with metal powders that provide electrical conductivity through percolation-based conductive pathways. They can be either isotropic or anisotropic depending on the level of the conductive particle loading and the method of application. The main deficiencies of these passively loaded polymers occur with temperature and humidity fluctuations. These fluctuations cause loss of electrical continuity due to the oxidation of the contact pads and particles as well as the expansion of the polymer. In addition, no metallurgical connection is formed with the pads when using conventional conductive pastes. The conventional conductive paste materials have high interfacial resistance to the copper pads, have low current carrying capability, and cannot maintain an electrical shift below 10% when subjected to common reliability test requirements.

The reliability and performance limitations of conventional conductive pastes are surmounted by sintering conductive paste materials. Sintering conductive pastes combine small particles of solder materials with small particles of solderable metals in a self-inerting flux. During lamination of the subassembly cores into a single PCB, the solder particles melt, wet the solderable particles, and form an interconnected metallic network. The metallic network formed has a melting point higher than the original solder alloys and is therefore stable through subsequent thermal excursions such as lead-free assembly operations. Because the

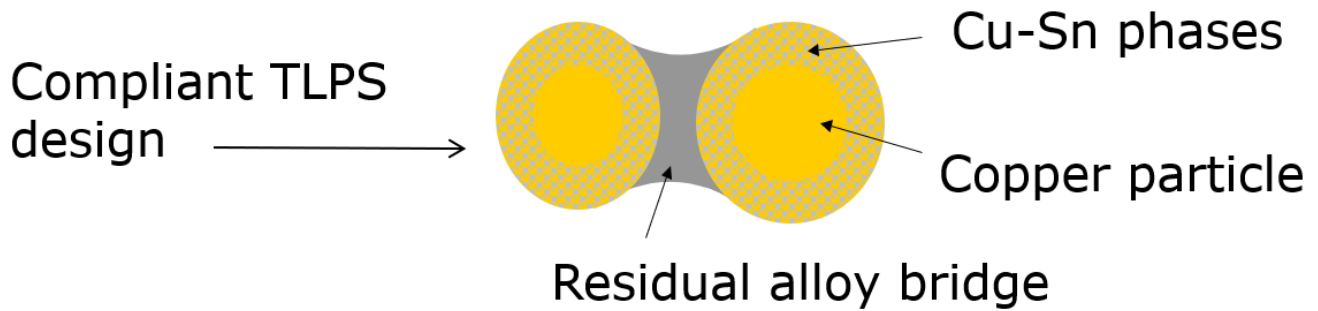


Figure 3: Conceptual view of a sintered TLPS interconnect.

solder particles also wet to the copper caps on the PTHs, the caps become permanently metallurgically bonded to the metal network of the sintered paste interconnect. Therefore, during lamination, the sintering conductive paste delivers a continuous and robust thermal and electrical conduction through sintered metal joints. Figure 3 illustrates the sintering paste concept of metallurgically interconnected particles.^[1]

In Figure 1, the copper plating over the PTHs on either side of the sintering paste via can be clearly seen to be wetted by the solder alloy in the paste in the cross-section to form a continuous metallurgical pathway. By design, processing of sintered conductive pastes is done at temperatures compatible with standard printed circuit board materials and common lamination cycle conditions. Once sintered, as seen in the cross-section view, there are a variety of metallic phases in the metallic network, however, all the phases present in the sintered material are stable in thermal exposures well above the initial process temperature and can thus withstand assembly or other subsequent thermal processing. The metal matrix does not remelt, nor is it damaged during lead-free solder reflow as can be seen by again referring to the cross-section in Figure 1, which has been exposed to six such solder reflow cycles.

These sintering conductive pastes overcome several of the disadvantages of copper paste materials. Of particular importance in vertical interconnection, they provide a low and stable electrical resistivity on the order of 30 μW -

cm. This stable electrical performance in both normal use and under environmental and thermal cycling is the result of the alloyed metallurgical web with a compliant design that accommodates the large shift in CTE in the Z-axis as the laminate is heated above its T_g , which preserves the via integrity. Furthermore, the sintered conductive paste has a high thermal conductivity of nearly 20 W/mK,^[2] an order of magnitude better than most conventional conductive adhesives, which prevents thermal runaway. More critically, because the metallurgical joint is contiguous from pad-to-pad, the high bulk thermal conductivity is not squandered on poor interfacial transfer between the pad and the deposit. The low bulk and interfacial electrical and thermal resistance are also critical to current-carrying capacity and signal integrity.

Transient Liquid Phase Sintering

Sintering is a term that has become more familiar in the electronics packaging industry with the advent of nanomaterials into joining applications, but the term sintering encompasses many variations. Sintering is a process in which adjacent surfaces of metal powder particles are bonded by heating. Nano-phase sintering relies on the very high surface activity of the nano-sized particles to cause rapid interdiffusion without the formation of a liquid state. Liquid phase sintering is a form of sintering during which solid powder particles coexist with a liquid phase. Densification and homogenization of the mixture occur as the met-



Profit. On Demand.

Gardien has two test and inspection solutions that will improve your bottom line.



On Demand

Global test and inspection service where and when you need it.

[SEE NOW →](#)



Integrate

We manage and improve your QA department

[HERE'S HOW →](#)

als diffuse into one another and potentially form new alloy and/or intermetallic species.

In transient liquid phase sintering (TLPS) of powders, the liquid phase only exists for a short period of time as a result of the homogenization of the metals to form a mixture of solid alloy and/or intermetallic species. The liquid phase has a very high solubility in the surrounding solid phase, thus diffusing rapidly into the solid to form the metallic reaction products. Diffusional homogenization creates the final composition without the need to heat the mixture above its equilibrium melting point. After cooling, subsequent temperature excursions, even beyond the original (LMP) alloy melt temperature, do not reproduce the original melt signature of the mixture. This is the “signature” of a typical low-temperature transient liquid phase sintered metal mixture. This signature can be seen in Figure 4 where the paste is initially sintered at 190°C and then subsequently cycled to 280°C multiple times to simulate multiple reflow cycles. As can be seen in the DSC scan, the original endothermic alloy melt is largely gone after the initial cycle

and there is some continued interdiffusion in the first simulated reflow cycle, but the microstructure composition is very stable through the remaining reflow cycles.

TLPS technology is used to produce conductive compositions that include metal powder(s), solder alloy powder(s), and a self-inerting flux system. TLPS compositions are used to form interconnects by creating a patterned deposition of the TLPS composition, and then sintering the metallic components in the composition by heating to relatively low temperature. During heating, the self-inerting flux cleans the metal powders, allowing TLPS to occur. After heating, the self-inerting flux chemically binds the resultant metal oxides, rendering them harmless. For this reason, these compositions provide good electrical and thermal conductivity with little opportunity for conductivity deterioration due to oxidation, corrosion, or thermal expansion and contraction. TLPS compositions do not require nanoparticles, precious metals or polymer binders to create robust interconnects under standard lamination conditions.

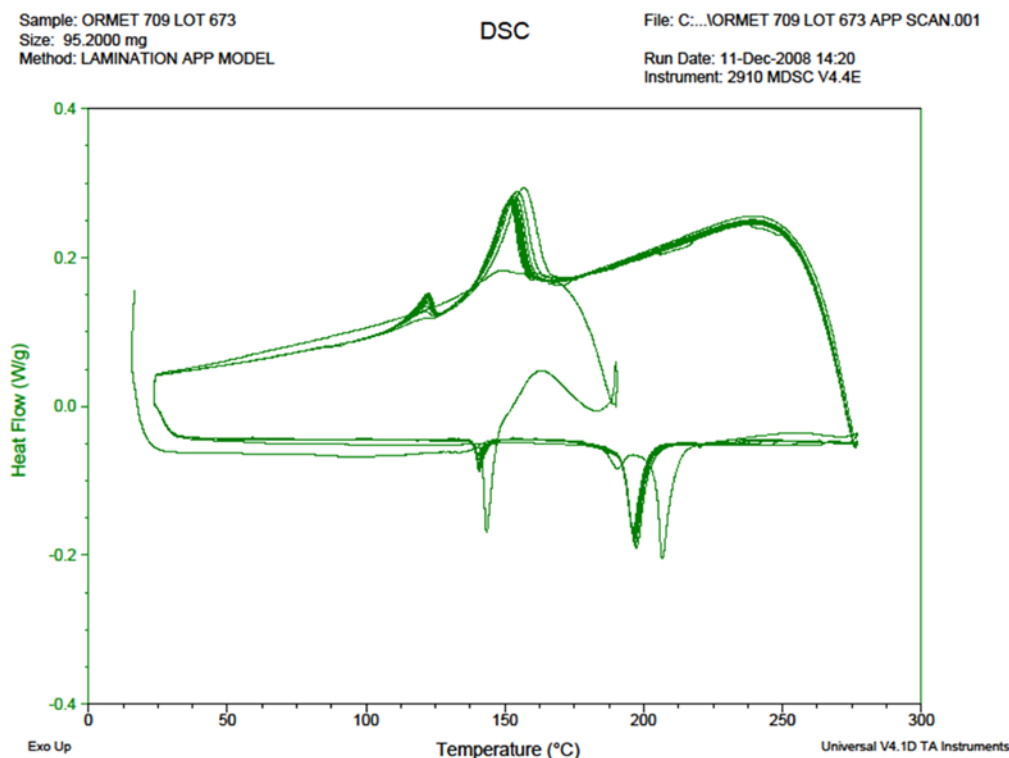


Figure 4: DSC signature of a TLPS reaction.

The microstructure of processed TLPS compositions looks like a network of particles of metal, each bearing one or more “shells” of the newly formed alloy/intermetallic compositions, which are in turn interconnected by the non-reactive portion of the original solder alloy. Open areas of the metallic network structure are generally filled with the self-inerting flux in small pockets. The interconnected metal forms 84 to nearly 100 volume percent of the sintered paste depending on the specific layup and lamination conditions.

Reaction between the metal and the reactive element(s) of the solder alloy may result in either partial or complete incorporation of the metal particles into the newly formed alloy and/or intermetallic species. The number and nature of the new alloy and/or intermetallic species that form is dependent on the selection of metallic constituents in the TLPS composition, their relative proportions, the particle size distribution, and the process temperature. The composition of the residual components of the original solder alloy is likewise dependent on these factors; therefore, cross-sections of the processed TLPS compositions will exhibit a number of distinct phases that vary in

proportion according to both the formulation and process conditions.

Transient liquid phase sintering (TLPS) pastes have been successfully used for over two decades in printed circuit board (PCB) interconnects—printed jumper wire and Z-axis layer-to-layer connections—that are subjected to subsequent solder reflow. In these applications, TLPS pastes are sintered during common PCB fabrication processes such as lamination, and will not remelt in subsequent assembly operations. Billions of interconnects have been made with TLPS pastes in these PCB applications. Often, the PCBs fabricated with TLPS interconnects are for extremely complex, reliability-critical, and relatively niche applications such as supercomputing and aerospace. The increasing demands from the cloud compute and 5G telecom markets are removing the “niche” categorization from that characterization.

Cost Analysis vs. Conventional PCB Fabrication Methods

The value proposition for the use of blind and buried vias in the fabrication of high-layer-count PCBs depends heavily on the specif-

Table 1: Construction-specific cost benefits of TLPS paste Z-axis interconnects.

Situation	Benefits
High density interconnect: Anywhere paste vias used to join high-density double-sided cores in a single lamination (e.g., handset)	Substantially reduce sequential-build-up cycle time and lamination steps
Moderate layer count: replace PTH with anywhere vias on some layers (e.g., server)	<ul style="list-style-type: none">• Improve routing density and reduce layer count• Reduce lamination cycles
High layer count: break into several PCB subs with interconnecting paste via layers (e.g., high performance compute)	<ul style="list-style-type: none">• Eliminate low yielding high-aspect-ratio PTH• Eliminate lossy PTH stubs without backdrill
Mixed material PCB: join dissimilar material cores using interconnecting paste via layers (e.g., rigid-flex)	Eliminate complex PTH preparation in dissimilar materials
Mixed function PCB: join digital and analog cores using interconnecting paste via layers (e.g., 5G infrastructure)	Fabricate each according to best design practices and material sets and then join
Mixed density PCB: join high- and low-density subs using interconnecting paste via layers (e.g., probe cards)	Build high density layers independently and join to low-cost cores

ic construction of the PCB and the constituent materials. It is generally most useful to study the issue from the perspective of total cost of ownership through the entire fabrication process—including yield. Blind and buried vias can increase the interconnect density of complex PCBs by enabling anywhere placement of vias without the need for multiple, semi-additive, sequential build-up and lamination cycles. As layer counts, PTH aspect ratios, and interconnect density requirements increase, yield losses increase dramatically in conventionally fabricated PTH PCBs. In high frequency applications, the use of TLPS vias can eliminate the risky back-drilling operation to truncate lossy stubs. Table 1 explores situations in which the use of TLPS paste vias for Z-axis interconnect can provide a potential cost/performance benefit.

Implementation of Sintered Paste Interconnections

There are a variety of ways in which sintering paste interconnections can be implemented depending on the specific needs of the application and the associated cost-benefit analysis. Flexibility of design and implementation flow is one of the advantages of paste interconnect technology. Because the TLPS paste forms sintered interconnects during standard prepreg lamination conditions, the number of

subassemblies, the method of application of the TLPS paste, the manufacturing flow, and the specific configuration are all at the discretion of the manufacturer.

A generalized concept of how the sintered interconnections might be installed is depicted in Figure 5.

Step 1: Subassembly cores are manufactured using standard PTH manufacturing methods. The number of subassemblies, and the number of layers within each subassembly, is at the discretion of the PCB fabricator/designer. The PTHs are filled and cap-plated to form a land for the TLPS paste interconnects. Generally, following standard industry best practices, an adhesion-promotion treatment is applied to the outer surfaces of the cores to ensure good bonding with the prepreg during lamination of the cores.

Step 2: Prepreg and a release sheet, generally PET film, are tack-laminated to one side of the mating pairs of cores. This is the first process that is atypical to standard PCB manufacturing, but can be performed using standard PCB fabrication tools such as a lamination press or dry-film laminator. The prepreg may consist of one or more layers depending on the thickness of the copper on the surface.

STEP-1: Adhesive material selection



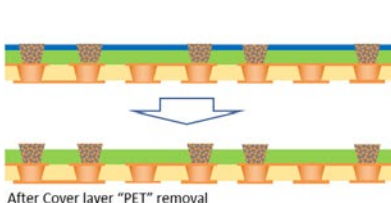
STEP-2 Tacking to circuit core



STEP-3 Laser ablation of blind vias



STEP-4: Via filling



STEP-5 : Laminate

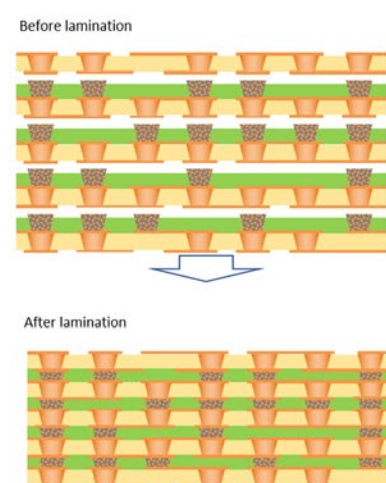


Figure 5: Generic installation process for TLPS pastes Z-axis interconnects.

Focused on Solutions

We are setting new standards for PCB manufacturing.

Schmoll Maschinen MDI digital imaging machines provide a high-volume and stable processing solution for manufacturers looking for performance at the highest level. For unbeatable exposure times, look no further.



schmoll  **optronics**

Schmoll MDI Micromirror-Digital-Imaging
Tandem Table Machine

- + Direct imaging of inner layers, outer layers and solder mask
- + High resolution with L/S down to 0.5 mil
- + Tandem table concept (2x 24" x 36") and big panel capability (1x 54" x 36")

For more information call us:
+1 (336) 660-2701

1991
2021  **BURKLE**
NORTH AMERICA, INC.

burkleamerica.com

Step 3: Via holes are laser ablated through the PET film and prepreg using the copper cap plate as a stop for the laser. This is the second point of differentiation from standard PCB fabrication practices. Although it is common to laser ablate microvias into C-staged laminate for sequential-build-up plating, tuning the laser parameters to cleanly cut the PET, the underlying B-staged prepreg (or film adhesive), and removing any organic residue from the pad surface takes some learning. The use of low- or no-flow prepregs, thinner glass styles, and a combination of a CO₂ laser to do the majority of the ablation, followed by a quick burst of a UV laser to desmear the bottom of the hole, is highly recommended. If a laser is not available, or if the feature sizes and registration tolerances are forgiving, the via holes could be achieved by punching, or mechanically drilling prior to attachment of the prepreg to the core.

Step 4: TLPS paste is then deposited into the vias using the PET film as a conformal stencil mask. The use of automated stencil or screen printers improves the consistency of the filling process and an “open” stencil that frames the active area of the circuit layer can be used as a “parking lot” for the paste between prints. Although automated stencil printers are not commonly found at most PCB fabricator shops,

screen printers are relatively common from the days of printing solder mask and legend ink. Once the vias have been filled and topped off, the PET film is carefully removed by peeling, leaving slight protrusions of paste on the surface. The subassemblies, now bearing prepreg and paste-filled vias, are placed in a standard box oven for 30 minutes at 90air environment) to evaporate the solvent and stabilize the protruding paste for handling in layup of the lamination book. The protruding paste stubs are compressed into the vias during lamination resulting in sintered interconnects with very high metal density.

Step 5: The subassemblies bearing the interconnect layers are aligned in a lamination book with the usual stiffeners, pressure distribution materials, and release layers. Lamination is performed under the standard lamination conditions recommended for the prepreg selected and the TLPS pastes as the prepreg adhesive cures. When the PCB is removed from the lamination press, all of the electrical interconnections have been formed and the PCB is complete.

There are an enormous number of potential variations to the process and ultimate construction of the PCB. Figures 6, 7, and 8 depict some of the more common variations, with typical dimensions and increasing level of reliance on the TLPS Z-axis interconnect layers as an alter-

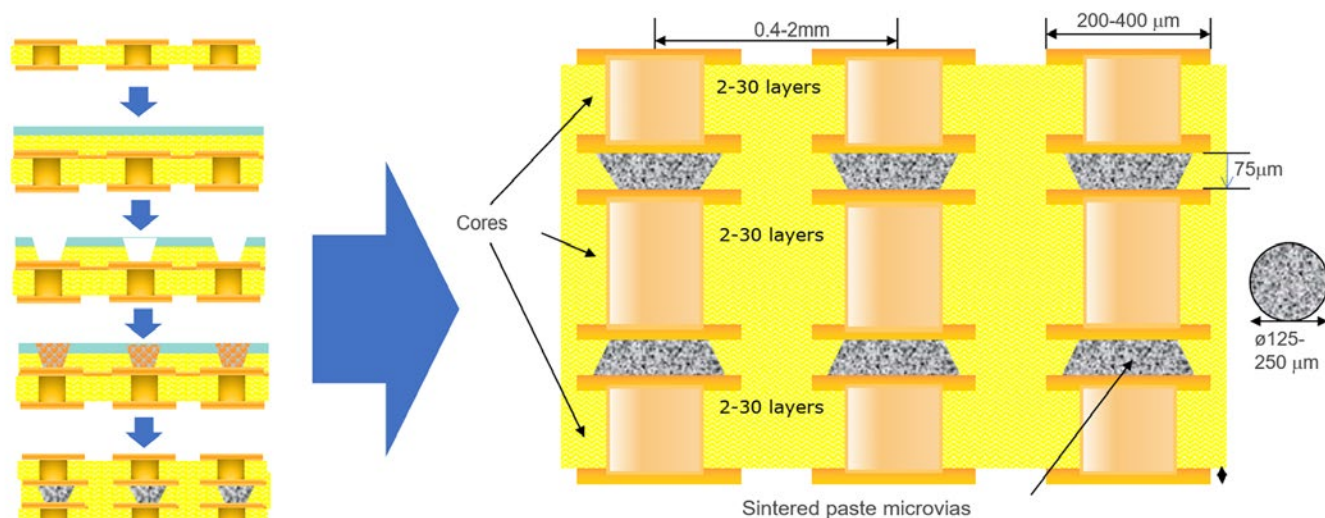


Figure 6: “Core-to-core” implementation method for installing TLPS paste vias in high-layer-count PCBs.

native to conventional fabrication techniques.

Core-to-core joining is the most common implementation and most closely resembles the generic process flow. This method could be used to form alternating layers of copper and TLPS vias, but most commonly PCB subassemblies with multiple layers interconnected by PTH are joined as large “chunks” of the overall PCB structure. This method is also popularly used to interconnect cores of dissimilar types, whether in dielectric material, circuit density or analog/digital design rules. The core-to-core process flow is depicted in Figure 6.

Figure 7 depicts a variant in which TLPS vias are mixed with conventional PTH cores and sequential-build-up HDI with plated microvias. This process flow takes advantage of the ability to fabricate the interior cores cost effective-

ly and limit the expensive plated-HDI process just to high density outer layers. TLPS paste is more resistive than copper so this type of mix-and-match can provide a means to maximize signal performance in the sensitive outer layers.

A third popular variant is to go full-Z with TLPS vias and no PTH or plated microvias. In this process flow, each layer undergoes just two lamination cycles no matter how many layers are in the final PCB construction. The first lamination cycle is used to create the double-sided 2-layer core with sintered TLPS connecting the opposing faces. The standard process flow is then used to interconnect the TLPS-based cores together in a single lamination cycle. This concept is depicted in Figure 8. As with all the other process flows depicted, there are many possible variations on this theme.

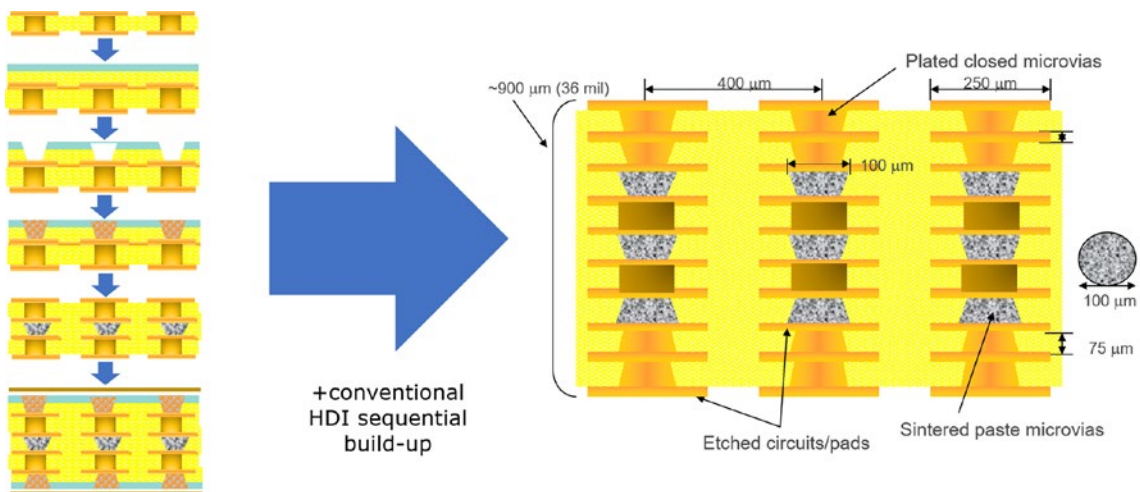


Figure 7: Mixed mode implementation method for installing TLPS paste vias in high-layer-count PCBs.

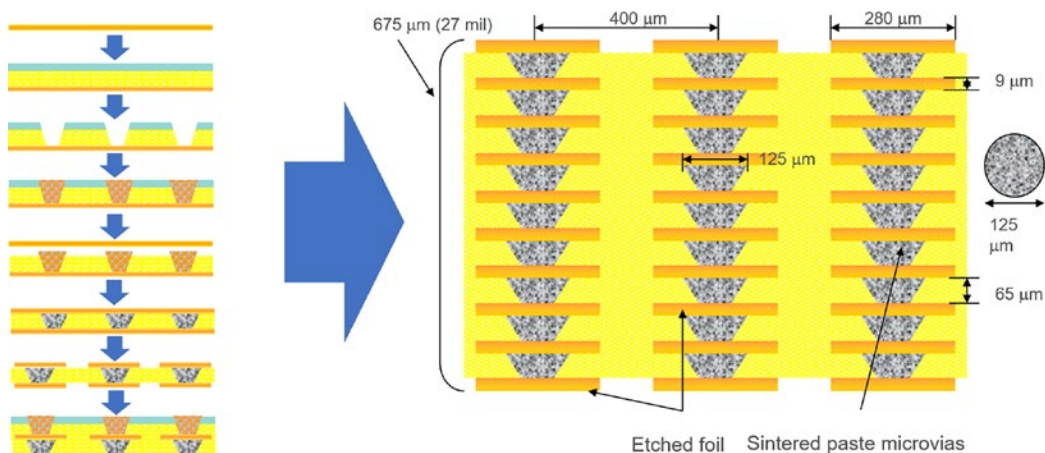


Figure 8: Implementation method for anywhere TLPS paste vias as exclusive Z-axis interconnect method.

The versatility and design flexibility are substantial advantages of the TLPS paste via process; however, absence from common industry design tools presents a challenge in exploring potential stack-ups. Partnerships are being formed to create design patches that enable designers to explore constructions that contain paste vias alongside conventional techniques.

Performance of TLPS Z-axis Interconnect vs. Conventional Fabrication Methods

Performance of TLPS paste Z-axis interconnects relative to plated interconnects is obviously a topic of interest. An early test vehicle by i3 Electronics investigated the relative performance of purely PTH constructions and those broken by a layer of TLPS vias. S-parameter measurements were made on a net that contained only PTH versus one that contained four sintered interconnects. Figure 9 demonstrates that the use of sintered interconnects connecting PTH bearing subs in place of a continuous PTH does not significantly degrade the signal performance. Below 10 GHz the difference between a copper barrel structure and a sintered via structure is negligible, and above 10 GHz there is only a slight degradation due to the additional via length. Overall, the performance is similar to a solid copper barrel.^[3]

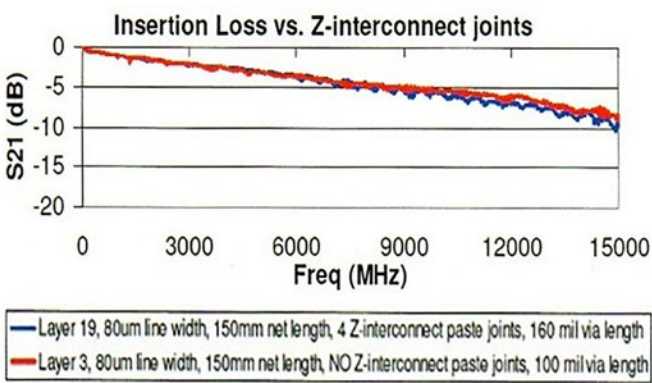


Figure 9: Insertion loss of nets with and without TLPS interconnects. (Source: i3 Electronics)

Full-Z test vehicles in which the copper interconnects were entirely replaced with TLPS vias layers also demonstrated equivalent performance to the plated copper interconnect control in third party testing.^[4,5]

In recent work to support the growing high frequency market, a dedicated test coupon was designed by Insulectro^[6] to test the signal integrity and current-carrying capability performance of various TLPS paste via structures relative to conventional PTH and plated-microvia fabrication methods. The coupon contains 10 nets including a copper microstrip baseline, and nine comparison nets for signal integrity performance and current-carrying capacity. The coupon design is detailed in Figure 10.

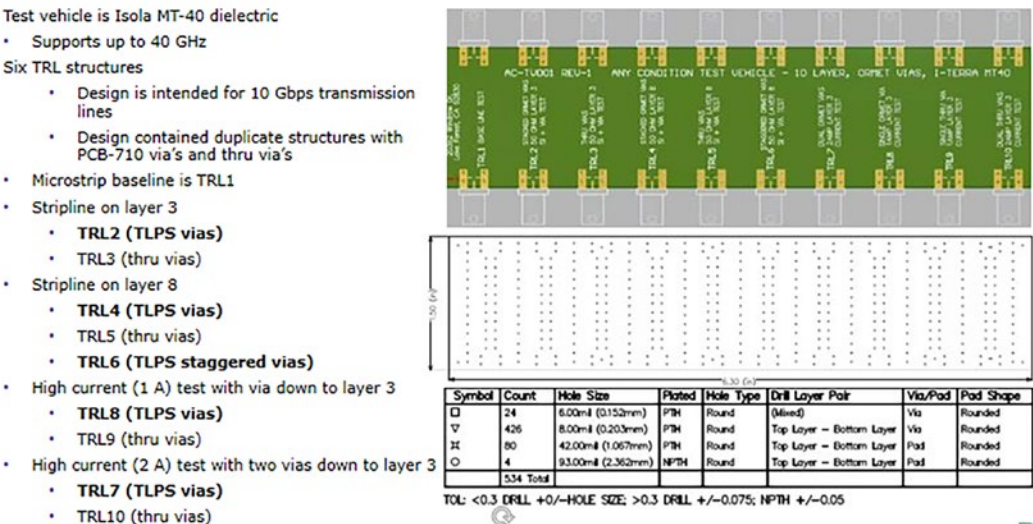


Figure 10: Dedicated high frequency test coupon for plated copper vs. TLPS paste interconnects.



ROUNDTABLES

Lively and insightful discussions from industry experts. **Watch now!**



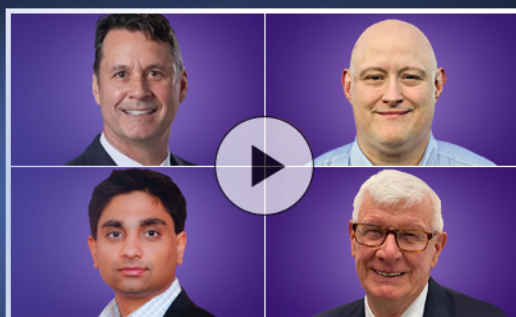
**App Notes
and Fab Notes**



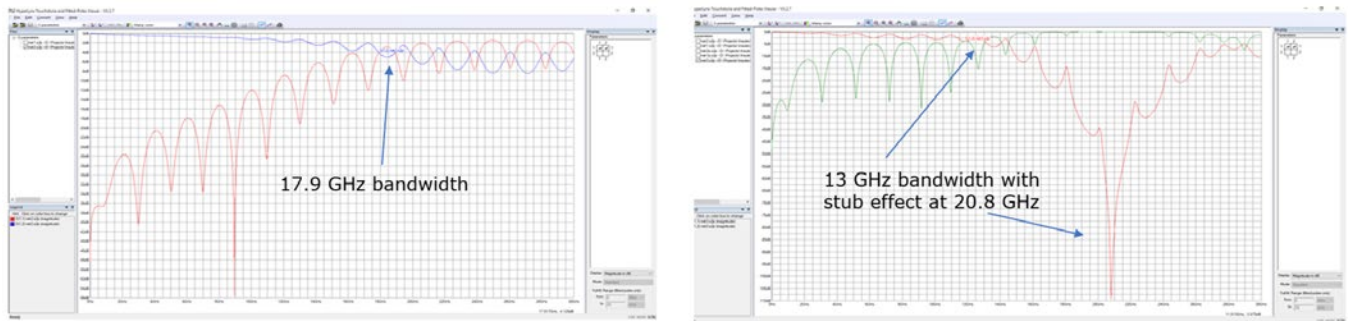
**Process Ionic Contamination
Test (PICT) Standard**



**Achieving Operational Excellence
in Electronics Manufacturing**



**Use of IMS Thermal Materials
in Multilayer Stackups**



Layer 1 to layer 3 in a 10-layer test coupon

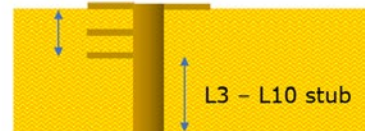
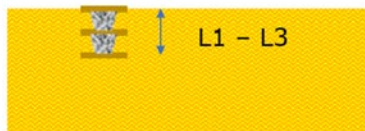


Figure 11: TLPS paste vias provide high frequency bandwidth capability without the need to backdrill.

Although a detailed analysis is beyond the scope of this paper, the impact of eliminating a lossy PTH stub is clearly seen in the comparison of TRL2 and TRL 3 in Figure 11. Although such a stub could be eliminated by backdrilling, the use of TLPS vias to make this interconnection eliminates the need for this process and preserves routing area in layers 3-8. Nets TRL4, TRL5 and TRL 6 can be seen in cross-

section in Figure 12 and the relative signal integrity performance is shown in Table 2. Even when the Z-axis interconnect extends almost through the entire layer count of the coupon, the high frequency performance of the net is improved over the PTH. Finally, the two sets of current-carrying capacity test nets showed no degradation with current levels of 4.5 amps—well in excess of the design margin—for both

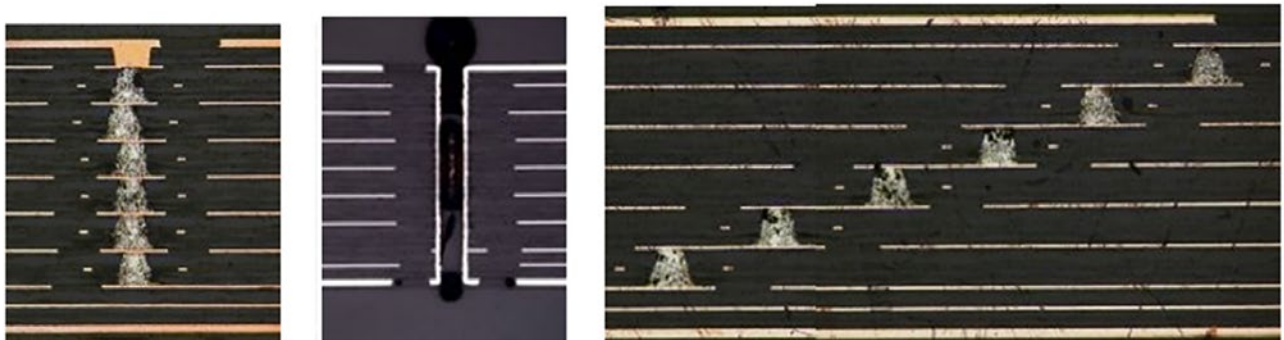


Figure 12: Comparison of layer 2 to layer 8 interconnection in a 10-layer coupon using: stacked TLPS paste vias, PTH and staggered TLPS paste vias (note: not all TLPS vias are centered in cross-section plane).

Table 2: Signal integrity performance of TLPS vias vs. PTH

Net	Type	BW* (GHz)	Loss (dB)	Line impedance (Ω)
TRL4	Stacked TLPS	13.5	-5	60
TRL5	PTH	9.5	-8	60
TRL6	Staggered TLPS	12.5	-8	60

*S-parameter performance: through loss S21 and return loss S11

the plated and paste options, which attests to the good thermal dissipation and low electrical resistivity of each of these nets.

Conclusion

The market for high-layer-count PCBs has traditionally been highly specialized to address the needs of high-end computing, military, telecom, and semiconductor test applications. Using conventional drill and plate process steps to build these PCBs is becoming unsupportable and has driven adoption of TLPS paste vias to overcome issues in high-aspect-ratio and high-density, leading-edge designs for over two decades. Creating PCBs from several subassemblies that can be independently fabricated and joined using sintering pastes is an attractive alternative manufacturing strategy that leverages existing manufacturing flow. Both the technical performance and the cost benefit of this approach have been favorably evaluated by independent sources.

As the trend to higher PCB complexity challenges the use of conventional fabrication strategies in a greater array of applications, the design versatility and process flow flexibility of replacing one or more layers of conventional Z-axis interconnect with TLPS paste becomes more compelling in mass market applications. Coupled with now-established manufacturing flows and a solid track record of reliable performance, software patches under development for common design tools will enable wider adoption of TLPS paste interconnect as the benefit can be readily determined at the design phase. PCB007

This article/paper was presented at IPC APEX EXPO 2021 and was published in the Proceedings.

References

1. U.S. Patent No: 8,840,700
2. R. E. Taylor, H. Groot, and J. Ferrier, TPRL Report 1584, "Thermophysical Properties of Conductive Ink."
3. Das, Rabindra, Egitto, Frank, Laufer, Joh, Antesberger, Tim and Markovich, Voya, "Z-Axis Interconnections for Next Generation Packaging," Advancing Microelectronics, Vol. 38 No. 6, pp 12-19.
4. Carver, Chase; Seatrand, Norman; Welte, Robert; "PWB Z Interconnect Technology—Electrical Performance," IMAPS Proceedings, Oct 2014, San Diego, CA ISBN: 979-0-9909028-0-5
5. Laufer, John; Knadle, Kevin; "Z-Interconnect Technology—a Reliable, Cost Efficient Solution for High Density, High Performance Electronic Packaging," IMAPS Proceedings, Oct 2014, San Diego, CA ISBN: 979-0-9909028-0-5
6. Coupon design is the property of Insulectro, Inc, Lake Forest, CA, 92630. Coupon build performed by Gorilla Circuits, Inc. General design features and results shared with permission.



Catherine Shearer is head of conductive paste R&D, EMD Electronics.



Gary Legerton is an applications engineer at EMD Electronics.



American Standard
Circuits



On Demand: Free Training Video Series

FLEXIBLE CIRCUIT TECHNOLOGY

with Expert Joe Fjelstad

[WATCH NOW](#)

DIG: The Next Generation

The Plating Forum

by George Milad, UYEMURA

DIG stands for “Direct Immersion Gold.” The acronym is used to specify direct deposition of gold on copper as a surface finish. It is a metallic solderable finish. At assembly, DIG forms a Cu/Sn intermetallic with the gold layer dissipating into the bulk solder. DIG has been around for at least 15 years.

Gold will readily immerse on copper based on their respective positions in the EMF series. The reaction is driven by +1.22 volts. As in all immersion reactions, the reaction will continue as long as the substrate is available to the displacement reaction. As the substrate is covered by the depositing species, it becomes less available, rendering the reaction self-limiting.

The original formulation of DIG produced a relatively thin gold layer that had a reddish hue to it from the partial diffusion of copper. Copper diffusion to the surface continued with time; after a year of storage at ambient conditions the deposit color became increasingly reddish, almost brown. Checking the solderability of the discolored surface using wetting

balance methods showed excellent wetting and no signs of soldering degradation.

At that time, DIG did not seem to offer any breakthroughs as other established finishes like OSP, immersion silver, and immersion tin. These surface finishes were well-established, their limitations were well understood, and they were deployed extensively in PCB manufacturing. DIG was a more costly finish and created apprehension as it was clear that the copper would diffuse through the immersion grain boundaries into the gold, altering the as-received surface.

A new generation of DIG was developed to meet the needs for wire bonding and high frequency signal propagation. High frequency RF signal loss is associated with thicker nickel deposits. The new DIG process uses a reduction-assisted immersion gold (RAIG). The use of an RAIG gold allowed for the deposition of a thicker gold layer up to 0.3 μm (12 μins) that prevented the diffusion of copper to the surface.



Electronics Coatings for the Digital Era



ELECTRA

ELECTRAJET® EMJ110 **Inkjet Soldermask**

- SM840 and UL V-0 approved
- High resistance to ENIG, Tin and lead-free processes
- Suitable for rigid & flexible substrates
- Optimised ink characteristics for sharp printing and excellent cosmetics
- Wide jetting window to maximise print strategy options
- Universal product suitable for a range of print heads including Konica Minolta 1024i Series and Dimatix Samba™
- Fully additive process with substantial environmental improvements in energy usage and VOC emissions

EMJ110 is a high performance, high reliability soldermask specifically designed for application by inkjet. Using over 35 years' experience in developing and supporting soldermask products, Electra has rigorously formulated EMJ110 to meet or exceed industrial standards and PCB manufacturing process demands.



T +44 (0)1732 811118
info@electrapolymers.com
www.electrapolymers.com

During deposition the immersion gold initiates on the copper surface and that triggers the electroless deposition, which will continue to deposit as the copper substrate gets plated over and is no longer available to sustain the immersion reaction. The electroless reaction does not require substrate contribution as it is driven by a reducing agent in the electrolyte. The electroless deposit is non-granular and not porous.

A new generation of DIG was developed to meet the needs for wire bonding and high frequency signal propagation.

Unlike the original immersion DIG, this new mode of deposition can produce a thicker layer that is pore-free (no grain boundaries) thus limiting the migration of copper. The deposit has the lemon-yellow color that is expected of a gold deposit in contrast with the reddish yellow of the original DIG. At 0.2 to 0.3 μm , the deposit shows no signs of copper diffusion to the surface.

Hofstetter PCB AG in Küssnacht/Switzerland is a supplier of various coatings in microelectronics. Hofstetter supplies a complete spectrum of surface finish plating to the microelectronics industry. They offer some of the newer finishes like EPIG (electroless palladium immersion gold), ISIG (immersion silver immersion gold), and DIG (direct immersion gold). Their R&D department invests in studying the properties of some of the newer finishes coming to the market, exploring solutions to meet the ever-changing demands of their customers.

Hofstetter has conducted a comprehensive study on the gold wire bondability of the new DIG process.

They studied the consistency and strength of the gold wire bonds under different tem-

peratures, namely 90–130°C for the wedge bond and 150°C for the first (ball) bond. They included room temperature bonding in the study. Room temperature bonding has a narrow operating window compared to a heated table. Success at room temperature bonding was achieved by the optimization of the DIG surface topography and the US (ultrasound) bonding force. The study concluded that the new DIG at 8–12 μins (0.2–0.3 μm) thickness is capable of producing wire bonds consistently and reproducibly with optimized bonding parameters.

The DIG surface at 8–12 μins exhibits a distinct fine roughness, which is very evenly formed over the entire surface. The new DIG surface requires minimum processing steps. Deposition requires a standard cleaner and micro-etch followed by the gold bath. This contrasts with ENEPIG which requires cleaner, micro-etch, catalyst, electroless nickel, electroless palladium, and immersion gold.

The new DIG is a surface finish that offers advantages over the traditional finishes like OSP, immersion silver, and immersion tin. OSP is a fragile, organic solderability preservative that cannot be used as a contacting surface. Immersion silver is a good contacting surface but was susceptible to creep corrosion and occasionally formed voids at the IMC surface. Immersion tin will form a Cu/Sn intermetallic during storage and may lose some of its wetting properties at assembly.

The new DIG, which is commercially available today, is solderable and aluminum and gold wire bondable, and it is ideally suited for plating small features with limited spacing. The absence of nickel in the surface finish makes it well-suited for high frequency applications. **PCB007**



George Milad is the national accounts manager for technology at Uyemura. To read past columns or contact Milad, [click here](#).

Need help finding the right people for your team?

DB Management knows that finding the right people can be like looking for a needle in a haystack. We can streamline your hiring process with pre-qualified candidates.

Engineering • Management • Sales & Marketing



[Click here to get started](#)

☎ 207-649-0879 ✉ danbbeaulieu@aol.com



MilAero007 Highlights



PRIDE Industries: A Nonprofit EMS and Staffing Firm Moves Into Mil-aero ►

PRIDE Industries is a contract manufacturing provider with a twist: The company provides training and coaching for job seekers with disabilities, including service-disabled veterans. If they don't have openings in their Sacramento, Calif., facility, they may have a job for you in one of 15 other states.

Medical, Defense Face Shortage of Multilayer Ceramic Capacitors ►

Industrial, medical, and military demand for high-quality, high-voltage multilayer ceramic capacitors (MLCCs) has been hit hard by a shift in production by the world's largest MLCC manufacturers who are focusing on a seemingly insatiable demand for smaller, lower voltage—and in some way—lower-performance MLCCs.

Defense Speak Interpreted: Defense on Legacy Weapons Systems ►

As “Defense Speak Interpreted” readers have surmised, the weapons systems of yesterday, today, and tomorrow are under review, both with President Biden and with the Congress now in control by Democrats. But “weapons systems of yesterday”? In the fast-paced consumer electronics world, “legacy” never comes up.

Industry CEOs Urge Action to Improve Electronics Manufacturing Ecosystem ►

More than 50 CEOs urged Commerce Secretary Gina Raimondo to take concrete steps to address challenges confronting the entire U.S. electronics supply chain.

Arlon Takes ‘AIM’ in Hiring OEM Global Marketing Manager ►

Arlon is increasing the company's focus in their primary markets: aerospace, industrial and military (AIM). When EMC acquired Arlon from CriticalPoint Capital, EMC's top-level management made it clear that Arlon was to take the lead for the company in promoting products (EMC and Arlon) into Arlon's AIM markets.

TT Electronics UK Facility Achieves AS9100D Certification ►

TT Electronics, a global provider of engineered electronics for performance critical applications, announced that its Eastleigh, UK, facility has achieved AS9100D certification for the manufacture of systems for the aerospace industry.

NASA's Ingenuity Mars Helicopter Completes First One-Way Trip ►

NASA's Ingenuity Mars Helicopter completed its fifth flight on the Red Planet with its first one-way journey from Wright Brothers Field to an airfield 423 feet (129 meters) to the south.

CAE to Accelerate the Design, Development of Jault Air Mobility's eVTOL Aircraft ►

CAE announced that it has been selected by Jault Air Mobility to lead the design and development of the Jault Aircraft Systems Integration Lab for the company's new all-electric vertical take-off and landing aircraft, the Journey aircraft.

Your circuit boards delivered...

**1
DAY
LATE**

**2
DAYS
LATE**

**3
DAYS
LATE**

WHAT'S THE COST TO YOU WHEN YOUR BOARDS ARE DELIVERED LATE?

With Prototron, you don't have to ask that question. Serving customers for over three decades with a 98% on-time delivery record, we understand that providing you with high-quality PCBs on time, and right the first time, is critical.



Prototron Circuits
America's Board Source

www.prototron.com



Meet Mr. Henry and Don't Blow a Fuse

Testing Todd

by Todd Kolmodin, GARDIEN SERVICES USA

Volts, amperes, and ohms: in electrical test, these terms are very familiar. Combinations of these electrical elements make up the standard electrical test of the average PCB. Standard electrical test uses the IPC specification 9252 as a baseline. Depending on the performance class, different parameters are used to screen the board. There are many machines out there that can perform these basic tests, and this has been the way of life for decades.

However, life changes. These design guys and gals do not stop the train of evolution. Packages become smaller, PCBs become smaller, and newer electrical characteristics come in to play. Now we must deal with farads and Mr. Henry—after Joseph Henry. See what I did there?

Today, new requirements bring buried capacitance and inductance. Although today's flying probe machines are familiar with capacitance via the indirect test and capacitive discharge analysis, direct measurement of the

actual values become more challenging. If the design drawing is provided and locations for direct capacitive measurements are given (with values), the machines of today can provide this test much akin to buried resistance.

However, inductance becomes more of a challenge. Most fixture testers are ill-equipped to test anything other than the standard ET parameters using volts, amps, and ohms. Mr. Henry introduces a variable that these metering systems simply cannot measure. Buried or active inductance is not anything new to PCBs but the ability to properly measure the inductance is. In the past, simple primary and secondary passive coils are laid out in a PCB resembling nested circles or blocks. Resistance of the primary and secondary circuits can easily be measured but if there is a short in either the primary or secondary circuits to itself (remembering the circular patterns) the change of resistance in the primary or second-

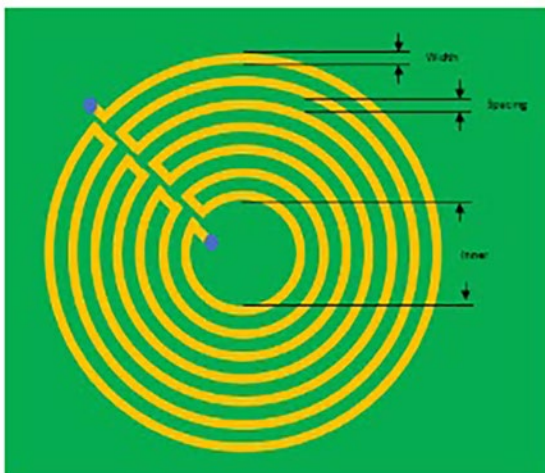


Figure 1: Circular coil.

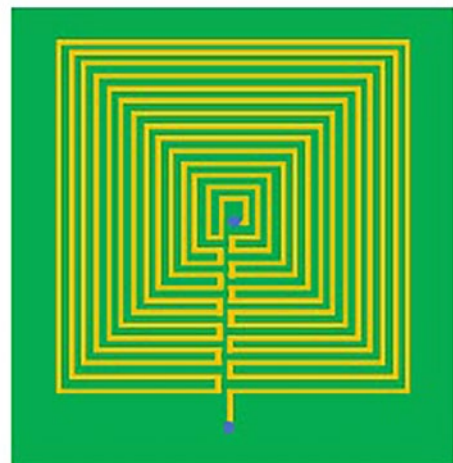


Figure 2: Square coil.

WHAT'S THE CARBON FOOTPRINT OF YOUR PTH LINE?

ELECTROLESS COPPER

VS

BLACKHOLE / SHADOW
DIRECT METALLIZATION

15+ horizontal or
vertical steps



~6 horizontal steps
89% Reduction in operator labor

Heavy metals, chelators,
toxic chemicals



Little to no waste
processing required
69% Reduced waste treatment

High temperature,
high water usage



Low Temp, Minimal Rinsing
57% Reduced power usage
76% Reduced water usage

Requires constant dosing to
maintain plating



Non-dynamic
process chemistry
Stable quality, start-stop ease

Palladium metal



Carbon or Graphite
Zero Precious Metal Costs



ary circuit within itself is not large enough to cause a continuity failure.

Therefore, the actual inductance of the coil should be measured to ensure a breakdown of the primary, secondary, or both is not present. Some newer flying probes have developed software modules that can measure this inductance. Just like buried resistance, the inductance (in Henry's) is pre-programmed within the netlist and the flying probe measures the inductance of the coil circuit just like any other network. However, this does require an upgraded metering system beyond the standard out-of-box configuration. If you are interested in automated inductance testing, consult your preferred equipment vendor, and discuss what options may be available to you.

Another caveat of the electrical test arena is the presence of high-resistance shorts, sometimes referred to as high-Z or micro-shorts. The problem these phenomena bring to detection is that they can be present momentarily at the time of test but due to current flow of the test machine being used, they can "blow" like a fuse. Although electrical testers use a minimum of current flow during test, these micro-shorts can be extremely small, like slivers or whiskers. It takes a minimum of current to blow them open. A technician may attempt to verify the short reported, only to find that it has disappeared, and the retest of the PCB will pass. Unfortunately, a ticking time-bomb remains. The board now passed and moves on to assembly and finally into service. However, deep inside the board a process called electro-chemical migration may be taking place. Due to the internal burn of the "blown fuse," there are burn deposits left behind. Over time, this process can migrate metal ions from one electrode to the other—in this case between the once shorted tracks or traces. In the worst case, this can cause an electrical path to return and a high resistance short may reappear. This can cause the final product to have degraded functionality in the field or fail altogether.

To combat this, extremely low current should

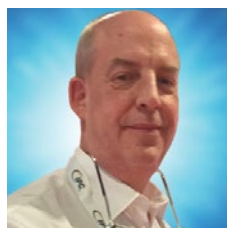
Current Limited to (μA)	Voltage (V)	Short Threshold (Mohms)
5	10	154
10	20	310
15	30	466
20	40	500
25	50	500

Table 1: Micro-short detection.

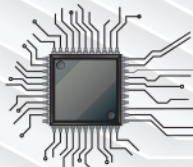
be applied to the circuits while still having the ability to detect the high resistance short. Many times, these shorts will register well above the standard ET parameter for isolation such as 10 or 20 megohm. Using a "micro-short" algorithm some test machines can identify these defects without destroying them. Keeping the current to the lowest possible limit while still looking for the high resistance fault is the key. Using standard methodology, the current may exceed the resistance threshold of the short and it heats up and finally burns. This can happen almost immediately so this test should be done prior to standard ET screening. That way the micro-shorts will be identified immediately before higher current is applied.

Doing a micro-short test after standard ET defeats the purpose of the test, as any damage may have already occurred. One must also remember that these shorts can exhibit very high resistance and a standard "off the shelf" ohmmeter may not have the resolution to verify the high resistance present. So, in micro-short detection one must not assume the defect is fake because an ohmmeter does not detect it. If you don't have the correct verification equipment, retest the board. If it fails again, believe the machine, fail the board, and have it examined. Second guessing a fault such as this only results in disaster.

Be safe! **PCB007**



Todd Kolmodin is VP of quality for Gardien Services USA and an expert in electrical test and reliability issues. To read past columns or contact Kolmodin, [click here](#).

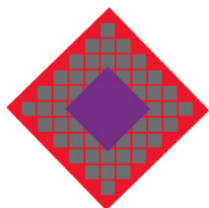


MivaTek

Global

****Worldwide patents pending**

Major New Technology Release



NextGen LE

New Light Engine Technology

Miva Technologies NextGen Light Engine is loaded with new features in support of its technical road-map. The NextGen is now available in all models.

- **Quad-wave Technology:** Images everything - in a single machine.
- **DART™ Compliant:** Scaling and line-width control tools that operate at 10X resolution for the most precise and repeatable total process.
- **New Vision System:** Miva's new vision system sets the stage for its line-width measurement and Gerber driven metrology.
- **Increased camera FOV:** Expanded FOV improves automation operation as well as eases panel alignment.
- **Resolutions:** NextGen 15/30-micron available allows for high speeds and resolutions.

High Power LED for 2x+ throughput:

Light Engines	1	2	3	4
30Mj Resist	46	83	100	124
50Mj Resist	33	62	75	95
DI SM	25	47	57	73

****18x24 panels per hour, both sides, 2400DT**



Digitally Adaptive Rasterization Technology

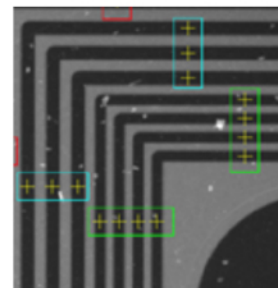
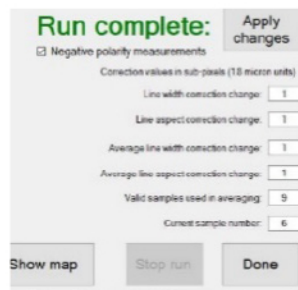
Digitally Adaptive Rasterization Technology

DART™ changes everything. Rasterization at 10X imaging resolution permits on the fly digital adjustments to line-width and scaling at 10X system resolution for hyper-precise feature/scale control.

- **TRUE Zone:** Pre-rip scaling capabilities allow for true zone scaling in linear or non-linear algorithms at 10X system resolution.
- **Feature Tuning:** Globally adjust feature sizes to include changes in develop or etch conditions in sub-pixel increments.

DART Optimization Suite

- **FA Tool:** Closed loop digital line-width compensation to include develop and etch process variation for first article or continuous.
- **Process Control Tools:** Topographical mapping of develop and etch process variations for improved process control and AB etch uniformity.
- **Coming Soon:** Integrated AB Registration Confirmation



Learn all about **NextGen**

MivaCon: Virtual Technology Conference June 15-16, 2021

Signup by emailing us at MivaCon@MivaTek.Global



Supplier Highlights



Insulectro Promotes Industry Veteran Michelle Walsh to Vice President of Product Management ►

Insulectro, the largest distributor of materials for use in manufacture of printed circuit board and printed electronics, has promoted Michelle Walsh to vice president of product management.

Mycronic to Acquire atg Luther & Maelzer ►

Mycronic is to acquire atg Luther & Maelzer GmbH (atg), a leading global developer, manufacturer and supplier of advanced equipment for electrical testing of PCBs and substrate. The acquisition strengthens and broadens Mycronic's offering and creates a platform in the field of electrical testing.

Atotech Reports Q1 2021 Results, Raises 2021 Full Year Guidance ►

Atotech, a leading specialty chemicals technology company and a market leader in advanced electroplating solutions, reported its financial results for the first quarter of 2021 and raised its revenue and adjusted EBITDA guidance for full year 2021.

MKS Instruments Enters Agreement to Acquire Photon Control ►

MKS Instruments, Inc. announced that it has entered into a definitive agreement pursuant to which MKS will acquire Photon Control Inc. for CAD\$3.60 per share, in an all-cash transaction valued at approximately CAD\$387 million, with an estimated enterprise value of CAD\$343 million.

VARIOPRINT Converts to Peters Solder Resist ►

In the course of a quality campaign, VARIO-PRINT AG relies on the ELPEPCB® Elpemer® AS 2467 solder resist system. The Swiss company will use this solder resist in the future for a large part of their printed circuit board types.

Rogers Corporation Reports First Quarter 2021 Results ►

Rogers Corporation announced financial results for the first quarter of 2021. "Rogers delivered strong first quarter sales and earnings, driven by the continued execution of our growth strategy and operational excellence initiatives," stated Bruce D. Hoechner, Rogers' president and CEO.

KLA Foundation Pledges \$550,000 to COVID-19 Relief in India ►

KLA Corporation announced the KLA Foundation is expanding its COVID-19 relief efforts in India with a donation of \$550,000 to help combat the second wave of coronavirus infections currently taxing the country's healthcare system amidst a national vaccine shortage.

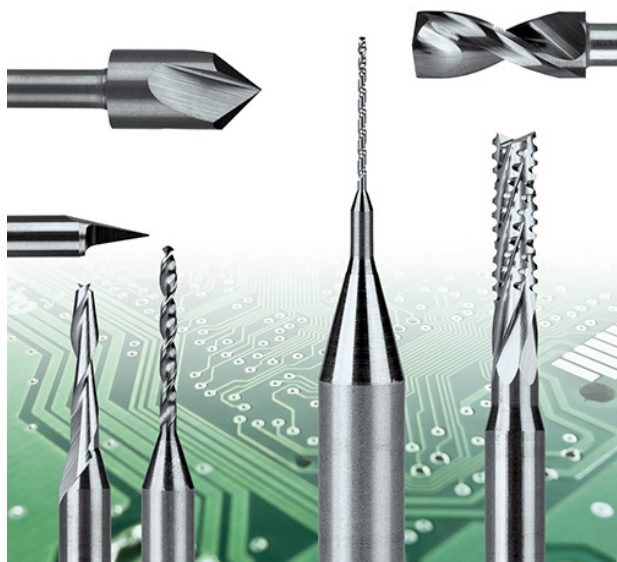
WAGO Introduces Michael Cuff as New Business Development Specialist ►

Michael Cuff has been hired by WAGO for the position of Business Development—PCB and Electronic Interconnect Specialist. A lifelong resident of Massachusetts, Michael studied electrical engineering, earning his bachelor's degree from Wentworth Institute of Technology in Boston and his master's at Tufts in Medford.

WORLD-CLASS SUPPLIERS

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

CALL 949.587.3200 FOR MORE INFORMATION



KEMMER PRÄZISION

The Wait is Almost Over! Drill Line Coming Early 2021

The Insulectro team is pleased to have received outstanding performance feedback on both quality and tool life with the solid carbide Kemmer router and endmill lines. Our router inventory consists of both Chipbreaker and Diamond cut tools with both Fishtail and Drill Point end styles. Our Endmill offerings will be in single and two flute. The rollout of the Kemmer drill line will be in early 2021.



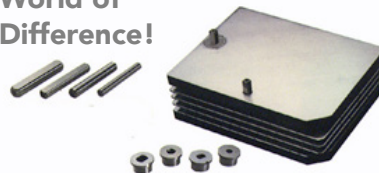
Lamination Assist For Flex & Rigid PCBs



Providing innovative lamination processing solutions including TRIPAK (PACOTHERM™), a releasable Press Pad that is specifically engineered to control the variables associated with the lamination processes.



Let us Make a
World of
Difference!



TADCO PRODUCTS: the most precise Lamination Tooling in the industry.

- Lamination Plates,
- Precision Pins & Bushings
- Release Agents
- Pin Cleaners
- Brushes

Precision is our specialty!



Denkai America is a leader in the manufacture of high-quality electrodeposited copper foils for printed circuit board (PCB), industrial, and energy storage applications. With the strength of domestic manufacturing, and backed by a global presence, Denkai America delivers both conventional cladding and application specific copper foils.



Leadership 101— The **Laws** of Navigation, Addition, and Solid Ground

The Right Approach

by Steve Williams, THE RIGHT APPROACH CONSULTING

Introduction

“Good leadership always makes a difference; unfortunately, so does bad leadership.” This leadership truth continues as we will be talking about the fourth and fifth of the 21 Irrefutable Laws of Leadership.

- a. The skipper of the SS Minnow
- b. The Love Boat’s Captain Stubing
- c. The captain of the Titanic
- d. None of the above

“Anyone can steer the ship, but it takes a leader to chart the course.” -

—John C. Maxwell

Navigation

What does navigation have to do with leadership? As it turns out, quite a bit. Leadership is not about blindly following the leader as much as it is about the leader charting the proper course for his or her followers. Think of it this way: You’re sitting in a boat when suddenly you enter a monster storm. Who do you want as the captain?

Followers need leaders who can effectively navigate for them; but navigation is not just about controlling the direction. Navigators see the big picture—the whole trip, if you will—before they leave the dock. They have a vision of how they will reach their destination, they understand what it will take to get there, they know who they’ll need on their team to be suc-



THE NEXT BIG THING IS HERE

Taiyo's ink jet solder mask is the ultimate innovation in solder mask application by eliminating waste and increasing productivity. Remove the mess of traditional lacquer applications and apply coverage only where needed. Enjoy less handling, zero developing, and improve registration yields. Created for both rigid and flex printed circuit boards.



BY TAIYO

- 🕒 IJSR-4000 JM Series JM02DG & JM03G
- 🕒 Superior adhesion to laminate with dual cure (UV/Thermal) Process
- 🕒 Precision additive process. ZERO waste!

Clean. Coat. Cure.

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.



Follow us on social media



cessful, and they recognize the obstacles long before they appear over the horizon.

The key to a leader becoming a great navigator is to plan ahead:

Predetermine a course of action

Lay out your goals

Aadjust your priorities

Notify key personnel

Allow time for acceptance

Head into action

Expect problems

Always point to the successes

Daily review your plan

Addition

The law of addition focuses on advancing others, not ourselves—remember “Servant Leadership” from the Law of Influence? Leadership is an act of service to others and the true leader focuses on creating value for others. The best place to serve is where we can add the most value to others. Leaders add value to others by valuing others and relating to what others value. True leaders ask, “How can I help you?” instead of “How can you help me?”

If we think back on the successes we have achieved, more times than not there was someone who influenced and inspired us in some way. Great leaders play an important role in the lives of those they lead, and this relationship is another example of what separates a manager from a leader. John Maxwell teaches us three simple truths on how to add value to others:

1. Truly value others.

I have had many mentors in my career who have believed in me even through my many

mistakes in my journey from being a manager to a leader. These mentors had, and continue to have, a profound influence on the person I am today.

2. Make ourselves more valuable to others.

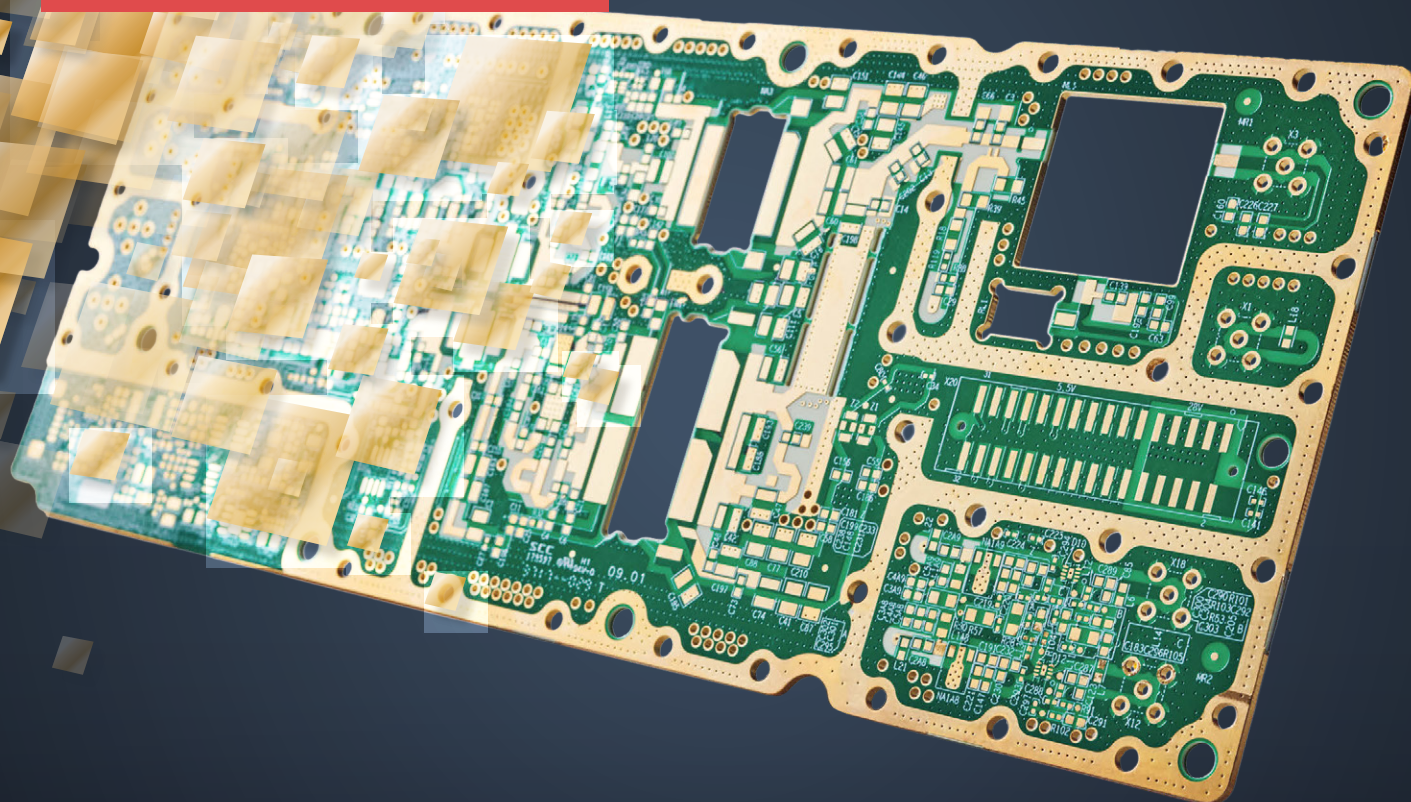
It took me a long time to learn that developing a strong bench strength is not only the path to success for my followers, but also for me. Until a manager learns to not be threatened by the talent in their team, they will never progress to being a leader.

3. Knowing and relating to what others value.

Early in my career I valued experience over education but eventually realized that to get the jobs I wanted and to eventually start my own company, my undergrad degree would not be enough, so I went back to university and earned my MBA. Years later, when I hired one the most talented “engineers” in the plastics industry, we had a career path discussion. He had very little advancement opportunity at his prior company because he did not have a college degree. Being able to relate to his aspirations, we developed a plan for him to get into an engineering program. He earned his BS, which led to a promotion about a year after graduating.



The new industry benchmark – Our true universal and economic PCB final finish



The desire to develop the best, most innovative industry solutions drives and defines Atotech. This is apparent in our contemporary electroless palladium and ultra-low corrosion mixed reaction immersion gold offerings: **PD-Core®** and **Aurotech® G-Bond 2**. Both offer considerable savings through their low precious metal content, best in class stability and ease of operation. These dynamic, all-in-one solutions can be used in series or individually. What's more, not only are they appropriate for ENIG and ENEPIG applications, but, when used in conjunction, are suitable for plating Pd/Au directly over copper PCB substrates.

For more information scan the QR-Code to the right





Solid Ground

Trust is the foundation of leadership, and the minute a leader loses this they also lose their influence. Leaders cannot break trust with people and expect to keep influencing them. Some leaders (actually managers) use their power and influence to force people into doing what they want them to do. And while this may work in the short-term, this strategy always backfires as followers begin to lose respect and loyalty for the leader.

Character and integrity are big parts of trust and solid ground. John Maxwell compares trust to change in a leader's pocket. Each time they make good leadership decisions, they earn more change, but each time they make poor decisions, they must pay out some of that change to the people. If leaders keep making poor decisions, they will end up without change in their pockets or, in other words, nobody will trust them any more as leaders.

Character earns respect. Without character it is very difficult to have respect for others and earn respect from your followers. Remember a manager does things right, a leader does the right thing. So how do leaders earn respect? By making sound decisions, admitting their mistakes, adding value to others, and putting others ahead of themselves. **PCB007**



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

Driving in the Snow is a Team Effort for AI Sensors

A major challenge for fully autonomous vehicles is navigating bad weather. Snow especially confounds crucial sensor data that helps a vehicle gauge depth, find obstacles and keep on the correct side of the yellow line, assuming it is visible. Averaging more than 200 inches of snow every winter, Michigan's Keweenaw Peninsula is the perfect place to push autonomous vehicle tech to its limits. In two papers presented at SPIE Defense +

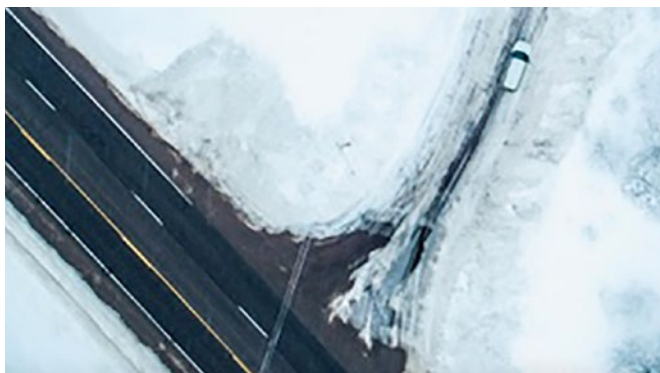
Commercial Sensing 2021, researchers from Michigan Technological University discuss solutions for snowy driving scenarios that could help bring self-driving options to snowy cities.

Major automakers and research universities are still tweaking self-driving technology and algorithms. Occasionally accidents occur, either due to a misjudgment by the car's artificial intelligence (AI) or a human driver's misuse of self-driving features.

Drivable path detection using CNN sensor fusion for autonomous driving in the snow

A companion video to the SPIE research from Rawashdeh's lab shows how the artificial intelligence (AI) network segments the image area into drivable (green) and non-drivable. The AI processes — and fuses — each sensor's data despite the snowy roads and seemingly random tire tracks, while also accounting for crossing and oncoming traffic.

(Source: Michigan Tech)



BUILD A BETTER WORKFORCE

- Electronics Assembly for Operators
 - IPC-A-610 for Operators
 - IPC-J-STD-001 for Operators
- Wire Harness Assembly for Operators



2/3 of electronic
industry companies
have difficulty finding
production workers.¹

vs



Upskill Your
Employees with
IPC Electronics
Workforce Training

IPC Electronics Workforce Training helps you bridge the skills gap. If you need to train assembly operators or wire harness assembly operators, these courses deliver consistent training and are easily scalable.

See our current course listing on training.ipc.org.

Courses can be offered directly to employees or integrated into your training programs.

¹ IPC. (2017). Findings on the Skills Gap in U.S. Electronics Manufacturing.



Editor Picks from PCB007

1 Arlon President Discusses Acquisition by EMC ►

Arlon President Brad Foster updates Nolan Johnson on EMC's acquisition of Arlon in December 2020. Foster shares the basic structure of the agreement, the long-term stability built into the merger, and outlines how this brings EMC, Arlon and Technica together as a team.



Brad Foster

3 EIPC Technical Snapshot Webinar: Aspects of Additive Manufacturing ►

Introduced and moderated by EIPC board member Martyn Gaudion, managing director at Polar Instruments, the eighth in the series featured a well-balanced selection of presentations with the collective focus of additive manufacturing in the electronics industry.



Martyn Gaudion

2 Catching Up With Gardien's Niraj Patel ►

I have always been fascinated by this company, especially two particular services they offer. These are two services everyone in our business needs to know about, which is why I recently reached out to Niraj Patel, Gardien's vice president in charge of North America.



Niraj Patel

4 One World, One Industry: New Thought Leaders Program Brings Industry Insight ►

As the electronics industry undergoes dramatic change, it is essential for IPC to obtain advice and counsel from experts throughout the industry that help us navigate these changes. To this end, IPC created the Thought Leaders Program, a select group of experts who will generate ideas and insights.



John Mitchell

5 Newly Appointed IPC APEX EXPO Technical Program Committee Calls for IPC APEX EXPO 2022 Participation ►

The newly appointed IPC APEX EXPO Technical Program Committee is inviting engineers, researchers, academics, technical experts, and industry leaders to submit technical conference abstracts for IPC APEX EXPO 2022 to be held at the San Diego Convention Center.

6 Lenthor Engineering Adds ESI's GEODE CO2 Microvia Drilling System ►

Lenthor Engineering, Inc., a California-based designer, manufacturer and assembler of flex and rigid-flex printed circuit boards, has purchased ESI's Geode CO2 microvia drilling system, including the optional automated loading and unloading stations.



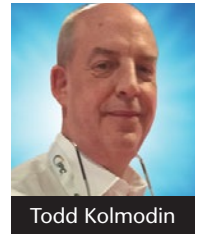
7 IPC: Driving Our Industry for 64 Years ►

There are many IPC committees, most of them focused on standards. Many IPC members take part in the committees—after all, electronic design and manufacturing standards is one of the key reasons IPC was founded. The results of the committees' efforts represent a significant part of the value the IPC has contributed to the industry since it was founded in 1957.



8 Testing Todd: Keeping the Tools Sharp ►

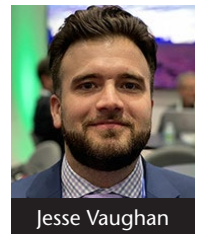
Quality is not just an action; it is a way of life. We can say we are quality conscious but as the days pass the discipline can fade. The tools become worn, dull and, finally, discarded. So, we must revisit the tool shed periodically to make sure our tools are razor sharp and at the ready.



Todd Kolmodin

9 Operational Excellence from a Design Services Manager ►

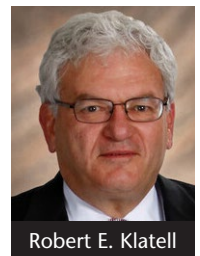
Jesse Vaughan discusses the key aspects of continuous improvement he sees in his role as manager of design services at ACDi, and the importance of carving out communication channels—both internally and externally to help build operational excellence.



Jesse Vaughan

10 TTM Announces Retirement of Board Member, Appointment of New Chairman ►

TTM Technologies, Inc., a leading global printed circuit board and radio frequency components and assemblies manufacturer, announced that Robert E. Klatell has retired from his position as chairman of the board and board member of TTM in accordance with TTM's Corporate Governance Policy that proscribes a mandatory retirement for directors at the age of 75.



Robert E. Klatell

For the Latest PCB News and Information, Visit: PCB007.com

Career Opportunities



Is your team growing?

Find industry-experienced candidates at I-Connect007.

For just \$750, your 200-word, full-column ad will appear in the "career opportunities" section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, and suppliers.

In addition, your ad will be featured in at least one of our newsletters, and your posting will appear on our jobConnect007.com board, which is also promoted in every newsletter.

Potential candidates can click on your ad and submit a resume directly to the email address you provide or be directed to the URL of your choice. If you wish to continue beyond the first month, the price is the same per month.

No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:

Barb Hockaday at barb@iconnect007.com or +1 916.365.1727 (-8 GMT PST)

I-Connect007
GOOD FOR THE INDUSTRY



Career Opportunities



Technical Writer (Full-time, Remote)

SEO company seeking a technical writer in the area of PCB design & manufacturing. We provide Search Engine Optimization and Thought Leadership services for well-managed, ethical companies. Our team produces high-level content for companies who are leaders in their industry. We are seeking writers who are well-rounded researchers with a particular interest in electrical engineering and impeccable writing skills.

Ideal candidates are seasoned writers with practical experience in electronic systems development in one or more of the following areas:

- Circuit simulation
- Schematic capture
- PCB layout
- Design analysis
- PCB fabrication
- PCB assembly
- PCB testing

Qualifications:

- Technical writing experience with a proven track record of independent research and content development. Experience in data sheet, report, or white-paper writing preferred.
- Electronic technician or engineering degree, ideally in electrical engineering, computer science, or mechanical engineering.
- Industry experience in PCB design, testing, or manufacturing.
- Punctuality, professionalism, and excellent time management skills.
- A reliable internet connection and computer

Salary & Benefits:

This is a full-time telecommuting position with a starting salary range of \$62,000 to \$68,000 annually. Benefits PTO, sick time, 401K, health and dental coverage, and more!

To see the full job description and apply, please click the link below.

[apply now](#)



Multiple Positions

Innovative Circuits, a quick-turn, high mix, low-volume PCB manufacturer located in Alpharetta, Georgia, is growing and looking for talented individuals to join the team.

Front End Engineering Manager

Oversee CAM, programming/production engineering and quoting departments. Ideal candidates will have 15 years' experience working in a printed circuit board front-end department with flex and rigid flex circuit board construction.

Process Engineer

Responsible for the implementation and maintenance of chemical and/or mechanical processes used to produce flex circuits, rigid flex and rigid printed circuit boards.

Third Shift Production Manager

Oversee third shift productions workers, product schedule and reporting.

Wet Lab Tech

Perform all lab analysis using burettes, pipettes, pH/ion meters, atomic absorption spectrophotometer, laboratory balance, hydrometers, hull cells, CVS, and all other lab-related equipment.

CAM Operator

Inspect, modify, and contribute to the initial development of producing flex circuits, rigid flex and rigid printed circuit boards based upon customer requirements and data files.

Quality Inspector

Responsible for verifying that the product meets customer requirements prior to shipping.

Wastewater Technician

Operate, monitor, maintain and troubleshoot the wastewater treatment facility and its processes.

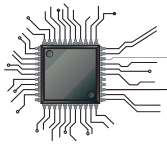
Production Worker

Machine operator and light chemistry in a PCB manufacturing environment.

Please visit the link below to view our opportunities and apply.

[apply now](#)

Career Opportunities



MivaTek

Global

Product Manager

MivaTek Global is preparing for a major market and product offering expansion. Miva's new NG3 and DART technologies have been released to expand the capabilities of Miva's industry-leading LED DMD direct write systems in PCB and Microelectronics. MivaTek Global is looking for a technology leader that can be involved guiding this major development.

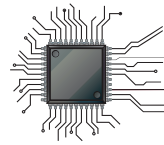
The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptations to Miva products
- **Design:** Be the voice of the customer for new product development
- **Quality:** Verify and standardize product performance testing and implementation
- **Training:** Conduct virtual and on-site training
- **Travel:** Product testing at customer and factory locations

Use your 8 plus years of experience in either the PCB or Microelectronic industry to make a difference with the leader in LED DMD direct imaging technology. Direct imaging, CAM, AOI, or drilling experience is a plus but not required.

For consideration, send your resume to N.Hogan@MivaTek.Global. For more information on the company see www.MivaTek.Global or www.Mivatec.com.

apply now



MivaTek

Global

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now

Career Opportunities



A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in America include:

Project Manager – Northfield, MN

Candidate will provide timely cost estimation and project budget definition, be responsible for maintaining customer relations, participate in meetings, etc.

[apply now](#)

Program Manager – Specialty Films

Candidate will work with our Specialty Films in the Aerospace, Medical, and Commercial Aviation markets providing timely cost estimation and project budget definition, maintaining customer relations, participate in meetings, etc.

[apply now](#)

Business Development Manager – North America

Candidate will provide leadership in the planning, design and implementation of customers' specific business plans and will provide vision, penetration strategies and tactics to executive managers in order to develop and drive external and internal senior-level relationships.

[apply now](#)



A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in Europe include:

Business Development Manager – France

Seeking out-of-the-box thinkers to help us take the ordinary to the extraordinary by cultivating current customer relationships and developing new business opportunities with our European team, based in France.

[apply now](#)

Business Development Manager – Germany

Seeking out-of-the-box thinkers to help us take the ordinary to the extraordinary by cultivating current customer relationships and developing new business opportunities with our European team, based in Germany.

[apply now](#)

Career Opportunities



ventec
INTERNATIONAL GROUP
騰輝電子

Technical Support/ Sales Engineer, UK

We are looking to expand our UK technical & sales support team. As a technical support/sales engineer (home office/Leamington Spa) you will assist potential and current customers in appreciating the benefits of using--and optimizing the use of--Ventec materials in their printed circuit board manufacturing processes, and so enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. You will provide a two-way channel of technical communication between Ventec's production facilities and UK/European customers.

Skills and abilities required for the role

- HNC, HND, degree or equivalent in a technical/scientific discipline
- Sales experience/negotiating skills
- Printed circuit board industry experience an advantage
- Good written & verbal communications skills
- Ability to work in an organized, proactive and enthusiastic way
- Ability to work well both in a team and independently
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on Offer

- Excellent salary and benefits commensurate with experience

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits.

Please forward your resume to
anthony.jackson@ventec-europe.com

[apply now](#)

INSULECTRO



Are You Our Next Superstar?!

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

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

View our opportunities at
[Insulectro Careers \(jobvite.com\)](https://www.insulectro.com/careers)

[apply now](#)

Career Opportunities



Packaging Engineer

Job description: The Packaging Engineer designs and deploys product packaging to ensure product integrity under varying shipping conditions. This individual is responsible for testing, analyzing, and selecting materials for packaging based on durability, function, ease of use and cost effectiveness. The Packaging Engineer helps ensure that packaging complies with all regulatory requirements.

Requirements: Bachelor's degree in engineering, packaging science and at least one year of related work experience. An equivalent combination of education and related work experience may be considered. Demonstrable skills with computer-aided design (CAD) software and other relevant programs.

Indium Corporation is a premier materials refiner, smelter, manufacturer, and supplier to the global electronics, semiconductor, thin-film, and thermal management markets. Products include solders and fluxes; brazes; thermal interface materials; sputtering targets; indium, gallium, germanium, and tin metals and inorganic compounds; and NanoFoil®. Founded in 1934, the company has global technical support and factories located in China, India, Malaysia, Singapore, South Korea, the United Kingdom, and the USA. Indium Corporation is an Equal Opportunity/Affirmative Action and Minority/Female/Disability/Protected Veteran Employer. We provide a drug-free work environment and a full benefits package.

[apply now](#)

SIEMENS

Siemens EDA Sr. Applications Engineer

Support consultative sales efforts at world's leading semiconductor and electronic equipment manufacturers. You will be responsible for securing EM Analysis & Simulation technical wins with the industry-leading HyperLynx Analysis product family as part of the Xpedition Enterprise design flow.

Will deliver technical presentations, conduct product demonstrations and benchmarks, and participate in the development of account sales strategies leading to market share gains.

- PCB design competency required
- BEE, MSEE preferred
- Prior experience with Signal Integrity, Power Integrity, EM & SPICE circuit analysis tools
- Experience with HyperLynx, Ansys, Keysight and/or Sigrity
- A minimum of 5 years' hands-on experience with EM Analysis & Simulation, printed circuit board design, engineering technology or similar field
- Moderate domestic travel required
- Possess passion to learn and perform at the cutting edge of technology
- Desire to broaden exposure to the business aspects of the technical design world
- Possess a demonstrated ability to build strong rapport and credibility with customer organizations while maintaining an internal network of contacts
- Enjoy contributing to the success of a phenomenal team

***Qualified applicants will not require employer-sponsored work authorization now or in the future for employment in the United States. Qualified Applicants must be legally authorized for employment in the United States.*

[apply now](#)

Career Opportunities



U.S. CIRCUIT

Plating Supervisor

Escondido, Calif.-based PCB fabricator U.S. Circuit is now hiring for the position of plating supervisor. Candidate must have a minimum of five years' experience working in a wet process environment. Must have good communication skills, bilingual is a plus. Must have working knowledge of a plating lab and hands-on experience running an electrolytic plating line. Responsibilities include, but are not limited to, scheduling work, enforcing safety rules, scheduling/maintaining equipment and maintenance of records.

Competitive benefits package. Pay will be commensurate with experience.

Mail to:
mfariba@uscircuit.com

[apply now](#)



Indium Corporation: Field Sales Representative

Field Sales Representative serves as lead sales contact and customer advocate to maintain existing sales and to drive new qualifications and sales of products and services through effective account management and coordination of efforts throughout Indium Corporation's Metals, Compounds, Solar and Reclaim (MCSR) organization. This position is ideal for a sales- and customer-focused individual with an engineering degree.

- Develop, cultivate, and follow-up with prospective and existing customers to generate orders
- Develop an in-depth expertise of product offerings
- Work to gain insight into customer activities for future R&D developments
- Respond to customer requests for product data, specifications, and service information
- Identify customer requirements, priorities, and opportunities
- Build strong, trusting relationships with key decision-makers and influencers at target accounts
- Gather competitive insight, including pricing, delivery, and performance information
- Visit customer facilities to observe manufacturing processes and exchange information
- Promote industry recognition of Indium Corporation, its products, and its services
- Be a key member of overall team, including worldwide sales organization, product management, operations, engineering, R&D, etc.
- Submit required paperwork in timely manner
- Work within established budget, while increasing market share
- Perform other duties and projects as assigned

Click below for more details on job responsibilities and requirements.

[apply now](#)

Career Opportunities

Now Hiring

Director of Process Engineering

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a director of process engineering.

Job Summary:

The director of process engineering leads all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering processes within the plant.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Provides guidance to process engineers in the development of process control plans and the application of advanced quality tools.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating them into the manufacturing operations.
- Strong communication skills to establish priorities, work schedules, allocate resources, complete required information to customers, support quality system, enforce company policies and procedures, and utilize resources to provide the greatest efficiency to meet production objectives.

Education and Experience:

- Master's degree in chemical engineering or engineering is preferred.
- 10+ years process engineering experience in an electronics manufacturing environment, including 5 years in the PCB or similar manufacturing environment.
- 7+ years of process engineering management experience, including 5 years of experience with direct responsibility for meeting production throughput and quality goals.

[apply now](#)

Now Hiring

Process Engineering Manager

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a process engineering manager.

Job Summary:

The process engineering manager coordinates all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering team and leading this team to meet product requirements in support of the production plan.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating into the manufacturing operations

Education and Experience:

- Bachelor's degree in chemical engineering or engineering is preferred.
- 7+ years process engineering experience in an electronics manufacturing environment, including 3 years in the PCB or similar manufacturing environment.
- 5+ years of process engineering management experience, including 3 years of experience with direct responsibility for meeting production throughput and quality goals.

[apply now](#)

Career Opportunities



SMT Operator Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for a **surface-mount technology (SMT) operator** to join their growing team in Hatboro, PA!

The **SMT operator** will be part of a collaborative team and operate the latest Manncorp equipment in our brand-new demonstration center.

Duties and Responsibilities:

- Set up and operate automated SMT assembly equipment
- Prepare component kits for manufacturing
- Perform visual inspection of SMT assembly
- Participate in directing the expansion and further development of our SMT capabilities
- Some mechanical assembly of lighting fixtures
- Assist Manncorp sales with customer demos

Requirements and Qualifications:

- Prior experience with SMT equipment or equivalent technical degree preferred; will consider recent graduates or those new to the industry
- Windows computer knowledge required
- Strong mechanical and electrical troubleshooting skills
- Experience programming machinery or demonstrated willingness to learn
- Positive self-starter attitude with a good work ethic
- Ability to work with minimal supervision
- Ability to lift up to 50 lbs. repetitively

We Offer:

- Competitive pay
- Medical and dental insurance
- Retirement fund matching
- Continued training as the industry develops

[apply now](#)



SMT Field Technician Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for an additional SMT Field Technician to join our existing East Coast team and install and support our wide array of SMT equipment.

Duties and Responsibilities:

- Manage on-site equipment installation and customer training
- Provide post-installation service and support, including troubleshooting and diagnosing technical problems by phone, email, or on-site visit
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Participate in the ongoing development and improvement of both our machines and the customer experience we offer

Requirements and Qualifications:

- Prior experience with SMT equipment, or equivalent technical degree
- Proven strong mechanical and electrical troubleshooting skills
- Proficiency in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Travel and overnight stays
- Ability to arrange and schedule service trips

We Offer:

- Health and dental insurance
- Retirement fund matching
- Continuing training as the industry develops

[apply now](#)

Career Opportunities



Sales Account Manager

Sales Account Management at Lenthor Engineering is a direct sales position responsible for creating and growing a base of customers that purchase flexible and rigid flexible printed circuits. The account manager is in charge of finding customers, qualifying the customer to Lenthor Engineering and promoting Lenthor Engineering's capabilities to the customer. Leads are sometimes referred to the account manager from marketing resources including trade shows, advertising, industry referrals and website hits. Experience with military printed circuit boards (PCBs) is a definite plus.

Responsibilities

- Marketing research to identify target customers
- Identifying the person(s) responsible for purchasing flexible circuits
- Exploring the customer's needs that fit our capabilities in terms of:
 - Market and product
 - Circuit types used
 - Competitive influences
 - Philosophies and finance
 - Quoting and closing orders
 - Providing ongoing service to the customer
 - Develop long-term customer strategies to increase business

Qualifications

- 5-10 years of proven work experience
- Excellent technical skills

Salary negotiable and dependent on experience. Full range of benefits.

Lenthor Engineering, Inc. is a leader in flex and rigid-flex PWB design, fabrication and assembly with over 30 years of experience meeting and exceeding our customers' expectations.

Contact Oscar Akbar at: hr@lenthor.com

[apply now](#)



Senior Process Engineer

Job Description

Responsible for developing and optimizing Lenthor's manufacturing processes from start up to implementation, reducing cost, improving sustainability and continuous improvement.

Position Duties

- Senior process engineer's role is to monitor process performance through tracking and enhance through continuous improvement initiatives. Process engineer implements continuous improvement programs to drive up yields.
- Participate in the evaluation of processes, new equipment, facility improvements and procedures.
- Improve process capability, yields, costs and production volume while maintaining safety and improving quality standards.
- Work with customers in developing cost-effective production processes.
- Engage suppliers in quality improvements and process control issues as required.
- Generate process control plan for manufacturing processes, and identify opportunities for capability or process improvement.
- Participate in FMEA activities as required.
- Create detailed plans for IQ, OQ, PQ and maintain validated status as required.
- Participate in existing change control mechanisms such as ECOs and PCRs.
- Perform defect reduction analysis and activities.

Qualifications

- BS degree in engineering
- 5-10 years of proven work experience
- Excellent technical skills

Salary negotiable and dependent on experience. Full range of benefits.

Lenthor Engineering, Inc. is the leader in Flex and Rigid-Flex PWB design, fabrication and assembly with over 30 years of experience meeting and exceeding our customers' expectations.

Contact Oscar Akbar at: hr@lenthor.com

[apply now](#)

Career Opportunities



ventec
INTERNATIONAL GROUP
騰輝電子

Customer Service Representative, UK

We are looking to expand our UK Customer Service/Internal Sales team. As Customer Service Representative you will provide great sales and customer service support and respond to the needs of clients from industries including Aerospace, Defence, Automotive and Pharmaceutical. Duties include:

- Maintain & develop relationships with new and existing customers
- Make rapid, accurate cost calculations and provide quotations
- Accurately input customer orders through bespoke MRP System
- Liaise with colleagues at Chinese HQ and other Overseas Business Units to manage domestic and international requirements
- Assist sales team with reporting, sales analysis and other items at their request

Skills and abilities required for the role:

The ideal candidate is a proactive self-starter with a strong customer service background. Friendly, approachable, and confident, you should have a good phone mannerism and be computer literate.

- Previous experience in a Customer Service background, ideally management or supervisor role
- Experience with MRP Systems
- Good working knowledge of Microsoft Office Tools such as Outlook, Excel etc.

What's on Offer:

- Excellent salary & benefits commensurate with experience

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits.

Please forward your resume to HR@ventec-europe.com

[apply now](#)



JOHNS HOPKINS

CAM / Process Engineer

The JHU/APL PCB Fabrication team is seeking a Computer Aided Manufacturing Engineer to support front-end data processing of APL manufactured hardware. You will directly contribute to hardware fabrication in support of National Security, Military Readiness, Space Exploration, National Health, and Research related to fundamental scientific advancement. This position includes a variable mix of core CAM work scope with additional opportunities for hands-on support such as bare board electrical testing, laser drilling, and mechanical CNC drilling and routing.

Responsibilities:

1. Computer Aided Manufacturing for rigid PCB, rigid-flex, and flexible circuits
 - a) Perform design checks, panel layout, coupon generation, file generation, stackups
 - b) Support manufacturability reviews with internal APL engineers (customers)
 - c) Generate work travelers
 - d) Communicate status to supervisors and internal customers
2. Support transition of software tools (Genesis 2000 to InCAM Pro)
 - a) Edit design rules checks and generate automation scripts
 - b) Develop new ideas to further the technical progress of our product
 - c) Develop CAM area through continuous improvement initiatives
3. Interface and inform APL Engineers on PCB design for manufacturing guidelines
4. Operate bare board electrical tester
5. Backup operator for CNC drilling, routing, laser drilling (on-site training)

For more details and to apply:

<http://www.jhuapl.edu/careers> and search for CAM.

[apply now](#)

Career Opportunities



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

The CAD/CAM engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creating manufacturing data, programs, and tools required for the manufacture of PCB.

Essential Duties and Responsibilities

- Import customer data into various CAM systems.
- Perform design rule checks and edit data to comply with manufacturing guidelines.
- Create array configurations, route, and test programs, penalization and output data for production use.
- Work with process engineers to evaluate and provide strategy for advanced processing as needed.
- Itemize and correspond to design issues with customers.
- Other duties as assigned.

Organizational Relationship

Reports to the engineering manager. Coordinates activities with all departments, especially manufacturing.

Qualifications

- A college degree or 5 years' experience is required. Good communication skills and the ability to work well with people is essential.
- Printed circuit board manufacturing knowledge.
- Experience using CAM tooling software, Orbotech GenFlex®.

Physical Demands

Ability to communicate verbally with management and co-workers is crucial. Regular use of the telephone and e-mail for communication is essential. Sitting for extended periods is common. Hearing and vision within normal ranges is helpful for normal conversations, to receive ordinary information and to prepare documents.

[apply now](#)



BLACKFOX

Premier Training & Certification

IPC Instructor Longmont, CO; Phoenix, AZ; U.S.-based remote

*Independent contractor,
possible full-time employment*

Job Description

This position is responsible for delivering effective electronics manufacturing training, including IPC Certification, to students from the electronics manufacturing industry. IPC instructors primarily train and certify operators, inspectors, engineers, and other trainers to one of six IPC Certification Programs: IPC-A-600, IPC-A-610, IPC/WHMA-A-620, IPC J-STD-001, IPC 7711/7721, and IPC-6012.

IPC instructors will conduct training at one of our public training centers or will travel directly to the customer's facility. A candidate's close proximity to Longmont, CO, or Phoenix, AZ, is a plus. Several IPC Certification Courses can be taught remotely and require no travel.

Qualifications

Candidates must have a minimum of five years of electronics manufacturing experience. This experience can include printed circuit board fabrication, circuit board assembly, and/or wire and cable harness assembly. Soldering experience of through-hole and/or surface-mount components is highly preferred.

Candidate must have IPC training experience, either currently or in the past. A current and valid certified IPC trainer certificate holder is highly preferred.

Applicants must have the ability to work with little to no supervision and make appropriate and professional decisions.

Send resumes to Sharon Montana-Beard at
sharonm@blackfox.com.

[apply now](#)

Career Opportunities



eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

Opportunities are available in Canada, New England, California, and Chicago. If you love teaching people, choosing the classes and times you want to work, and basically being your own boss, this may be the career for you. EPTAC Corporation is the leading provider of electronics training and IPC certification and we are looking for instructors that have a passion for working with people to develop their skills and knowledge. If you have a background in electronics manufacturing and enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Ability to travel required. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and skills

- A love of teaching and enthusiasm to help others learn
- Background in electronics manufacturing
- Soldering and/or electronics/cable assembly experience
- IPC certification a plus, but will certify the right candidate

Benefits

- Ability to operate from home. No required in-office schedule
- Flexible schedule. Control your own schedule
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Pre-CAM Engineer

Illinois-based PCB fabricator Eagle Electronics is seeking a pre-CAM engineer specific to the printed circuit board manufacturing industry. The pre-CAM Engineer will facilitate creation of the job shop travelers used in the manufacturing process. Candidate will have a minimum of two years of pre-CAM experience and have a minimum education level of an associate degree. This is a first-shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to

HR@eagle-elec.com indicating

'Pre-CAM Engineer' in the subject line.

apply now

Process Engineer

We are also seeking a process engineer with experience specific to the printed circuit board manufacturing industry. The process engineer will be assigned to specific processes within the manufacturing plant and be given ownership of those processes. The expectation is to make improvements, track and quantify process data, and add new capabilities where applicable. The right candidate will have a minimum of two years of process engineering experience, and a minimum education of bachelor's degree in an engineering field (chemical engineering preferred but not required). This is a first shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to

HR@eagle-elec.com indicating

'Process Engineer' in the subject line.

apply now

Career Opportunities



APCT, Printed Circuit Board Solutions: Opportunities Await

APCT, a leading manufacturer of printed circuit boards, has experienced rapid growth over the past year and has multiple opportunities for highly skilled individuals looking to join a progressive and growing company. APCT is always eager to speak with professionals who understand the value of hard work, quality craftsmanship, and being part of a culture that not only serves the customer but one another.

APCT currently has opportunities in Santa Clara, CA; Orange County, CA; Anaheim, CA; Wallingford, CT; and Austin, TX. Positions available range from manufacturing to quality control, sales, and finance.

We invite you to read about APCT at APCT.com and encourage you to understand our core values of passion, commitment, and trust. If you can embrace these principles and what they entail, then you may be a great match to join our team! Peruse the opportunities by clicking the link below.

Thank you, and we look forward to hearing from you soon.

[apply now](#)



U.S. CIRCUIT

Sales Representatives (Specific Territories)

Escondido-based printed circuit fabricator U.S. Circuit is looking to hire sales representatives in the following territories:

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

[apply now](#)



Webinar: For more information about the iamcam concept, please request the iamcam webinar video via presales@ucamco.com.

REALTIMEwith... IConnect007

ROUNDTABLE DISCUSSION FEATURING:



Chris Hanson
Ventec International Group



Rax Ribadia
Excello Circuits



Denis McCarthy
Ventec International Group



Pete Starkey
I-Connect007

Video An IMS Thermal Materials Discussion

Roundtable: "Use of IMS Thermal Materials in Multilayer Stackups for Power Applications" with Ventec International Group and Excello Circuits.

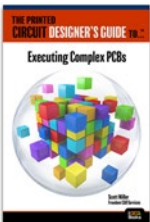
I-007eBooks The Printed Circuit Designer's Guide to...



Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



Executing Complex PCBs

by Scott Miller, Freedom CAD Services

Readers will learn how to design complex boards correctly the first time, on time. This book is a must-read for anyone designing high-speed, sophisticated printed circuit boards.



Thermal Management with Insulated Metal Substrates

by Didier Mauve and Ian Mayoh, Ventec International Group

Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.



Fundamentals of RF/Microwave PCBs

by John Bushie and Anaya Vardya, American Standard Circuits

Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs. This micro eBook provides information needed to understand the unique challenges of RF PCBs.



Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

Flexible circuits are rapidly becoming a preferred interconnection technology for electronic products. By their intrinsic nature, FPCBs require a good deal more understanding and planning than their rigid PCB counterparts to be assured of first-pass success.

Our library is open 24/7/365. Visit us at: I-007eBooks.com

PUBLISHER: **BARRY MATTIES**
barry@iconnect007.com

MANAGING EDITOR: **NOLAN JOHNSON**
(503) 597-8037; nolan@iconnect007.com

EDITOR | COLUMNIST COORDINATOR: **MICHELLE TE**
michelle@iconnect007.com

CONTRIBUTING EDITOR: **PATRICIA GOLDMAN**
(724) 299-8633; patty@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **DAN FEINBERG**
baer@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

SALES: **ANGELA ALEXANDER**
(408) 489-8389; angela@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

PRODUCTION MANAGER: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **SHELLY STEIN, MIKE RADOGNA,**
TOBEY MARSICOVETERE

CREATIVE TECHNOLOGIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © ANDREY ZYK**

PCB007
MAGAZINE

PCB007 MAGAZINE®

is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

© 2021 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

June 2021, Volume 11, Number 6
PCB007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

atg Luther & Maelzer GmbH.....	17
Atotech.....	71
Burkle North America.....	49
Chemcut.....	19
DB Management.....	59
Denkai America.....	15
Electra Polymers.....	57
Entelechy Global.....	41
Excellon.....	23
Gardien.....	45
I-007 eBooks.....	3
I-007e Flex Webinar.....	2
I-007e Roundtables.....	53
IPC.....	31, 73
Insulectro.....	7, 25, 67
MacDermid Alpha Electronic Solutions.....	63
MicroCraft.....	21
MivaTek Global.....	65
MKS ESI.....	11
Pluritec.....	33
Prototron Circuits.....	61
Summit Interconnect.....	13
Suss.....	5
Taiyo America.....	69
Technica USA.....	39
Ucamco.....	27
Ventec International Group.....	29

Problems solved!



SUBSCRIBE NOW