I-Connect007
GOOD FOR THE INDUSTRY

OCTOBER 2023

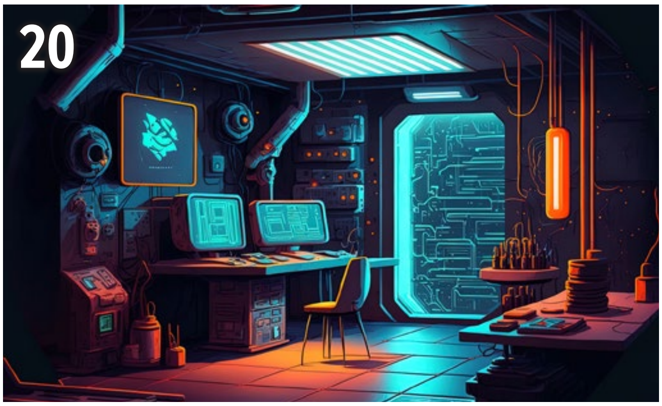
PCB007 MAGAZINE


Companionship at its Best
THE SOLUTIONS GUIDE TO...
Thermal Management
ventec INTERNATIONAL GROUP
I-Connect007
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Thermal Management with Insulated Metal Substrates
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Thermal Management with Insulated Metal Substrates
Volume 2
Didier Mauve and Robert Art
Ventec International Group
This must-read sequel to Ventec's book series on Thermal Management describes the applications, IMS products and support services to help you understand and overcome thermal management challenges.
Download Now
I-007eBooks

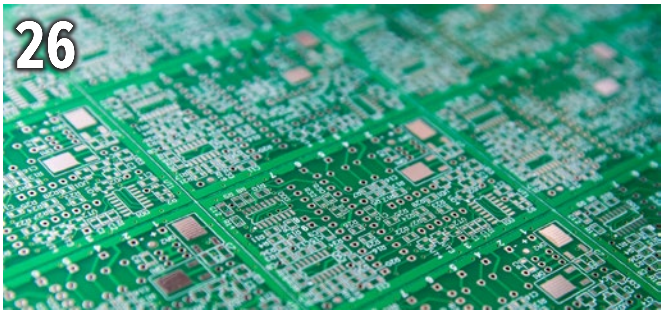
PCB007 MAGAZINE

High-reliability Fabrication

In this month's issue, our expert contributors discuss the latest in technologies, trends, complexities and resources regarding high-reliability fabrication. We also detail the tradeoffs involved in maximizing reliability while reducing your costs, and introduce you to experts and resources that can help raise your reliability levels.



PCB007 MAGAZINE



COLUMNS

ARTICLES

HIGHLIGHTS

SHORTS

DEPARTMENTS

Exploring High-reliability Fabrication

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

As you can see by our space-age cover, reliability can make the difference between life and death. While your next board may not be as mission-critical as an astronaut's lifeline, you certainly don't want your board to fail in the field, ever.

But the road to high-reliability fabrication success is often marked with potholes, and the U.S. seems to be hitting all of them. Why is Asia so far ahead of the U.S. in high reliability, UHDI, and additive processes?

High-rel fabrication presents several challenges and tradeoffs for fabricators, along with myriad market opportunities. There's never been a better time for U.S. fabricators to start manufacturing ultra HDI PCBs—if they have the know-how and resources to do it the right way.

With this issue, we set out to answer a variety of questions about high-rel fab. Where does high-rel begin—what's the actual cutoff? How much investment does it take for a

"typical" board shop to move into the high-rel realm? What are the biggest causes of high-rel failures? How do you measure reliability failure? After four years of research, why can't the North American fab industry figure out the root cause of microvia failures? Staggering microvias is just a workaround, and many times your board may require stacked vias. Why do Asian shops seem to have stacked microvia processes dialed in?

In this month's issue, our expert contributors have the answers. We discuss the latest in technologies, trends, complexities, and resources regarding high-reliability fabrication. We also detail the tradeoffs involved in maximizing reliability while reducing your costs, and introduce you to experts and resources that can help you raise your reliability levels.

We start with a conversation with Alex Stepinski, who offers an overview of high-rel fab and the bottlenecks that tend to trip up manufacturers. Columnist Preeya Kuray discusses the material science behind thermal reliability. Alex also contributed an article in which he discusses the need for fabs to pay more attention to copper grain structures. I've also included an excerpt from Reza Ghaffarian's chapter on reliability published in the *Printed Circuits Handbook.*

There's a piece from Tim Estes and Nick Meeker on failure and reliability trends, and a conversation with Marc L'Hoste, who explains why high-rel processes demand solid pre-planning, DFM, and inspection. We also have an article by Val Kapton, and columns from contributors Happy Holden, Hannah Nelson, and Travis Kelly.

See you next month! **PCB007**

Andy Shaughnessy is managing editor of *Design007 Magazine* and co-managing editor for *PCB007 Magazine.* He has been covering PCB design for 20 years. He can be reached by clicking here.

High-reliability Fabrication Overview

Feature Interview by the I-Connect007 Editorial Team

Alex Stepinski, principal and president of Stepinski Inc., and principal of Smart Process Design, has designed leading-edge PCB facilities and introduced a variety of novel systems and technologies into PCB manufacturing. We asked Alex to share his thoughts on high-reliability fabrication techniques, where manufacturers go wrong, and what fabricators need to do to step up their reliability game.

Barry Matties: Where does high reliability intersect with bare board fabrication and what should readers be thinking about?

Alex Stepinski: It's about service life. That's what the clients want. They want high reliability so it doesn't fail in the field, and when it does, it's very far away in time. You must do accelerated testing to confirm that your product will be reliable for "X amount" of time, and that it also de-risked from white rhino/black swan events and T0/latent fails.

I would break down high reliability management into two influential categories: process design and product design. During the product design, what do they control? Fabricators control KPIs to get high reliability. They should have a model, or a set of rules based upon prior history, as to what works and what doesn't work. They should also have these rules correlated to qualification methods, because one OEM might use IST, another uses a different type with different conditions, and so on.

Reliability is a subset of the test conditions, and the higher the reliability, the higher the

potential product value/willingness to pay, so fabs must make sure that they correlate their testing results to their product design attributes. But that precise regression is missing in a lot of cases, and there's a lot of diversity of test/qual methods. The big question is: What are the defects and/or parametric conditions that lead to reliability failure and can they be normalized? Today, many people are talking about HDI failures. Stacking microvias has been a common topic lately.

There are really three failure modes that occur, and everybody has a different way to deal with them. A large, influential, unnamed semiconductor company will deal with them in a completely different way than a typical fab shop. The semicon actually biases toward one failure mode by design, because they bought equipment that is designed to deal specifically with that condition, and they take advantage of it. But most regular fab shops got started on plated through-hole (PTH) technology, and the equipment is not set up for HDI. They try to do both processes with the same equipment and the trade-space is not favorable very often when it comes to HDI reliability.

On top of that, the whole process comes down to this: What chemicals and equipment are you using, and what are the steps? What's the whole process. What is the copper supposed to look like? What is the metallization that you're supposed to be using? It may not be a magazine-selling idea (laughs), because many of the challenges are often control problems and embarrassing, but that's really it. The process methods that are proven to drastically improve stacked microvia reliability are almost never discussed.

Nolan Johnson: Fabricators still struggle with these challenges, so is there a lack of progress here?

Yes. I recently got involved in reliability again for a conference in Munich. A major European aerospace organization had a problem, and I was solicited to talk about it because it was the same problem that I saw presented around 2004 in the U.S. It's now 2023, and they're presenting the same problem, with the same test methods, and almost the identical test vehicle design. The U.S. establishment went through this 20 years ago as well with almost identical stackups and structural configurations, and have actually not advanced that far past this point, but it was far enough that there was still value in a conversation.


Alex Stepinski

I offered some input on the topic, but in sitting on these reliability review meetings, you can see why nothing gets done. It's all "coopetition"—everybody is some kind of competitor. I don't understand how we can get them to do anything at all. Everybody's silent when someone asks a question. I think that's why there's no progress on the topic, and that's why people still talk about the same problem 20 years later. There needs to be well thought out incentives to manage the coopetition and improve the quality and capability of the ecosystem for tech advancement.

Matties: Are outside organizations more apt to solve problems like this?

Honestly, anybody who's in that business and running HDI fabrication facilities should have

all of their reliability equipment in-house and be testing it in-house as well. In the grand scheme of things, it doesn't cost that much when you're doing this. Outside organizations (i.e., OEMs, consortia, agencies) might develop a test vehicle or program, but it will take a year—or at least months to execute. So, while they do help, the cycles are too long to be of any benefit to everybody, and if you fail the test, you can get DQed. Also, by the time they get some data, we're already looking at or using different materials. This is something that they could run in a week if they had everything in-house, and stay ahead of the tech curve.

Andy Shaughnessy: I'm surprised that weak microvias are still a problem after four or five years of committees and groups trying to find the root cause. They've come up with workarounds such as avoiding stacking vias, but sometimes you have to stack vias.

That's right. One of the "big finds" I have heard recently in microvia reliability is that some designers were only putting in a few microvias in a large area. Of course, then it blows apart because all the stress is on one or two microvias. It's just kind of silly that this is a research topic now. We have seen this relationship just by changing the coupon pitch for years, and if you run multiple coupons at variable pitch, you can get precise values to build your own empirical model that is not just some theoretical interpolation of datasheets. This should be common sense by now for any designer or fab shop. Everyone knows, or should know, that you can't do that. The obstruction to technology advancement caused by compounding coopetition with scarce engineering resources is probably largely to blame.

> "Is your fab process consistent?"

Shaughnessy: Some companies say that material variations make it tough to build high-rel boards. What do you think of that argument?

No, materials are not the primary problem in 2023, and I have found that material suppliers have been really helpful with the data that they provide, for the most part. Sure, you can always be wildly off in material selection due to lack of due diligence, but I have found that the question with the materials suppliers is more like, "Is their data consistent with your baseline due to measurement methods?" The bigger question is, generally, "Is your fab process consistent?" Can you model with their data or not due to MSA (measurement system analysis) delta? Do you validate material properties when you qualify internally as an outcome of your process recipe? If your processes are consistent, then you may see some material variation. But I have to say, I just built some pretty advanced factories, and material variation was quite low, even though I was also biased that it may not be. The problem is more with the fabs being under-controlled, as there are major variables that most shops do not manage that are absolutely critical. I think that these are often masked because the folks doing the failure analyses are not privy to the detailed bill of process used (and neither are many of the fabs due to under-control), and if the signal-to-noise ratio on the fab side is so poor due to low due diligence on recipe development or poor controls, the materials tend to stand out as "different."

Johnson: What areas should have more focus for better reliability?

One highly neglected topic is copper grain structure. This is something that no one hardly ever talks about or addresses, and it touches many areas. Reliability and signal integrity are highly influenced by intrinsic copper structure and its effect on subsequent processing. It's very interesting. If you know the people involved, from the ones who make the copper foil to the end fab and all the chemical suppliers along the way, you realize how little control there is over this throughout the whole industry to achieve target performance.

Shaughnessy: I always thought copper grain structure wasn't something that you really worried about unless you were working at super high speeds or RF.

Sure, but when you look at the electronics market that advanced economies need to service worldwide, it's all getting pretty fast. Everything is spec'd this way more and more. One other aspect is when they have these microvia failures, nobody really looks at the copper grain structure and it's one of the most important variables that's been largely neglected (i.e., low hanging fruit). There are no industry standards for the modern requirements. There is all this information out there that nobody is aware of. How do you measure it? Does anybody measure it properly?

If you want to have high reliability, HDI, and signal integrity, you really need a way to measure copper grain structure and interface quality. It can mean an order of magnitude or more difference in bond strength.

Matties: What do you think is the greatest opportunity for fabricators today?

The greatest opportunities today are in manufacturing efficiencies. We have very low manufacturing efficiencies in the U.S.

Matties: It's not like we have to reinvent the wheel to achieve efficiency. There are a lot of roadmaps out there that could offer a place to start, and the 80/20 rule probably applies here too.

Sure. The biggest thing I see lacking is benchmarking; it's really poor. I travel a lot worldwide (46,000 miles in September alone), and I see very few folks from companies in the United States trying to learn the best new ideas, and how to do new things. It's always the same couple of companies and mostly non-technical folks (heads of purchasing or non-technical execs, mainly) looking for deals (buying more on price vs. lifecycle value). You see a lot of blind convergence and copying, and supplier reliance as well for stuff that should not be controlled by suppliers. You also see bubbles of people reinventing easily procured solutions at high cost. As Gen. Patton once said, "If everyone thinks the same way, someone is not thinking."

> "The biggest thing I see lacking is benchmarking; it's really poor."

In my opinion, this just shows us that the U.S. PCB market is a captive market supporting the Department of Defense, and they're not really motivated to go to best practices because they have no reason to. They just match expectations as the defense folks do not benchmark in a sufficiently detailed way either; it is all relative. The taxpayers are paying for it all. We should at least benchmark the detailed cost efficiencies with Asia at ground level and have a roadmap to get to some targets in order to get awards. So far, the benchmarking seems to be only at 30,000 feet or higher with many degrees of separation, and often without domain expertise.

Matties: The other side is that for the people who want great products, companies are just building captive facilities.

Right, due to frustration with the market mostly. They all tried to procure from the market first.

Matties: It's always a pleasure, Alex. Thanks so much.

Thank you all. PCB007

The Material Science of PCB Thermal Reliability

Material Insight

Feature Column by Preeya Kuray, Ph.D., AGC MULTI MATERIAL AMERICA

Printed circuit board (PCB) reliability testing is generally performed by exposing the board to various mechanical, electrical, and/or thermal stimuli delineated by IPC standards, and then evaluating any resulting failure modes. Thermal shock testing is one type of reliability test that involves repeatedly exposing the PCB test board to a 288°C pot of molten solder for a specific time (typically 10 seconds) and measuring the number of cycles it takes for a board's copper layer to separate from the organic dielectric layer. If there is no delamination, fabricators can rest assured that the board will perform within expected temperature tolerances in the real world.

From a materials perspective, what is happening to the copper and dielectric during thermal shock testing? What causes copper to separate from the organic dielectric layer in the first place?

To answer these questions, it is necessary to take a closer look at each component at the molecular level during testing. Subjection to intense thermal fluctuations causes different rates of expansion and contraction in different materials. This can generate stresses and ultimately lead to cracks or delamination between the copper and dielectric parts of the test board. The coefficient of thermal expansion (CTE) is defined as a material's dimensional change in length over a measured temperature range. Typical CTEs are listed in Table 1. Copper exhibits a Z-axis CTE of ~18 ppm/°C from 20–25°C while organic materials like epoxy

Table 1: CTE of common materials used in PCBs

Material	Category	Coefficient of Thermal Expansion (ppm/°C) from 20-25°C
Silica	Ceramic	0.5
Alumina	Ceramic	8
Copper	Metal	18
Epoxy	Polymer	55
Polystyrene	Polymer	70

MivaTek Global

2400NG - Dual Tray DI

High Throughput ● Automation Ready

Ready for your future

DART
Digitally Adaptive Rasterization Technology

NextGen

- Sub 25-micron features with traditional PCB processing
- Compliant with Miva's technology advances

Features

- NextGen Light Engine Feature Sizes: 6, 15, 30 Micron
- Quad-wave LED DLP: 360, 375, 390, and 405nm
- Worldwide DART™ DLP Patents Issued & Pending
- Panel Sizes: 24" x 30" (up to 42" x 60")
- Low Operating Costs, Flexible US Based Tech Support
- Automation compatible with third party sources
- Over 60 Machines in North America

Applications

- PCB: IL, OL, SM, Legend
- Micro-electronics: Features Down to 1 micron
- Chemical Milling
- Electroforming
- Glass Tooling

MivaTek Global, LLC, 6 Dickinson Drive, Suite 218, Chadds Ford, PA 19317 ● Website: www.MivaTek.Global
Contact: Brendan Hogan ● M (610) 620-3795 ● b.hogan@MivaTek.Global

display higher values (~55 ppm/°C). The quantitative differences in CTE between copper and organic materials can better explain why delamination may occur: Some types of materials just expand and contract more easily than others, leading to stresses and ultimately cracks during reliability testing.

But why do some materials have higher CTE values than others? Why do organic materials like epoxy and polyimide have higher CTE values compared to metals and ceramics?

The CTE is contingent on the bond strength between the atoms that make up that material. Covalently bonded materials exhibit strong shared bonds between the individual atoms, resulting in very low CTEs. For this reason, dielectric layers in PCBs are often reinforced with fillers like fused SiO_2, in which silicon is covalently bonded to oxygen (SiO_2 CTE = 0.5 ppm/°C). Other popular fillers include ceramics such as Al_2O_3 (CTE = 8 ppm/°C) or TiO_2 (CTE =10 ppm/°C). Ceramic materials exhibit ionic bonding, where the oxygen atom completely transfers its valence electrons to a metal atom, generating two oppositely charged ions. The unequal sharing of electrons yields a slightly weaker bond and contributes to a higher overall CTE (compared to materials that are purely covalently bonded).

What about metals? In the early 1900s, Paul Drude described the bonding between metallic atoms as a "sea of electrons," in which metals can be modeled as a lattice of positively charged metallic cores, all sharing a vat of communal valence electrons. The electron delocalization (or electron sharing) in metals results in an even lower bonding energy than covalent or ionic bonding, which may contribute to why pure metals like copper typically have higher CTE values than ceramic fillers.

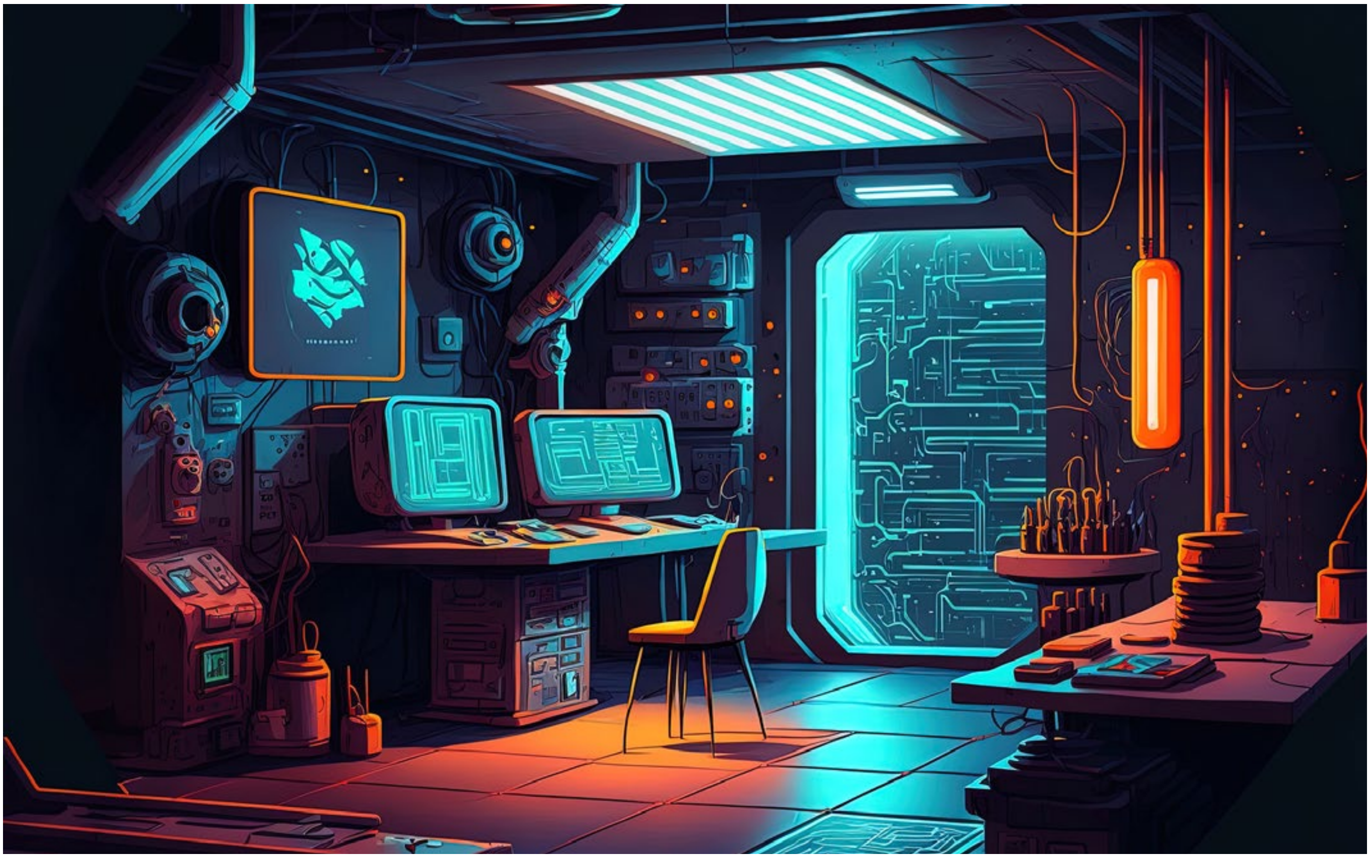
Finally, we arrive at the metaphorical glue holding together the PCB dielectric layer: the polymer. The CTE values of typical polymers can range from 20–100 ppm/°C (from 20°C–25°C), which are much higher than ceramics or metals. Polymers are defined as chemical compounds in which individual molecules are bonded together in long, repeating chains. (Imagine pearl beads bound together to make a necklace.) The low energetic barrier of polymer chains to move and undergo conformational changes can help explain their higher CTE values.

Bearing all this in mind, it is the task of formulation scientists to develop dielectric laminates, prepregs, and bond plies that will mitigate the impact of thermal expansion within a PCB and ultimately pass thermal reliability testing. By considering the intrinsic material properties of all components (and combining them in appropriate quantities), it is possible to create new materials that can meet, or even surpass, thermal reliability performance requirements. For example, fastRise™ TC (from AGC Multi Material America) is an example of a non-reinforced (pure resin, fiberglass-free) bond ply that exhibits excellent thermal reliability. By matching the CTE of copper (18 ppm/°C), this material is able to expand and contract at the same rate as the copper, mitigating stresses formed in the PCB during thermal reliability testing. In following IPC-TM-650 2.5.27, 24 panels of test coupons made from this material underwent 200 thermal cycles from 25°C to 260°C without exhibiting any failures. This is especially remarkable, considering that most dielectric materials on the market contain fairly large amounts of polymeric components.

Breaking through barriers like this illustrates the challenges and fulfillments of formulation science: pushing past intrinsic physical limitations to create something greater than the sum of its parts. It is what first drew me to become a materials scientist, and what will keep me grounded in this field for years to come. PCB007

Preeya Kuray, PhD, is a material scientist at AGC Multi Material America. To read previous columns, click here.


IPC BUILD ELECTRONICS BETTER
Community
MAGAZINE
FALL 2023
LOOK INSIDE
IPC BUILD ELECTRONICS BETTER
Contact us to become an advertiser



Financial Risks of Ignoring Copper Grain

Feature Article by Alex Stepinski
STEPINSKI GROUP

The topic of intrinsic copper structure has been largely neglected in discussions regarding the PCB fabrication quality control process. At face value, this seems especially strange considering that copper has been the primary conductor in all wiring boards and substrates since they were first invented. IPC and other standards almost exclusively address copper thickness with some mild attention being paid to surface structure for signal loss-mitigation/coarse properties.

Yet we still lack standards references as to what the actual copper grain structure should look like to optimize microvia reliability, what a target pad and capture-annulus should look like after laser drilling and post-treatment, and what copper grain structure yields which etch rate for optimizing the substrate differential etch and resolution limit. These topics generally fall into the category of individual factory know-how.

In this article, we will present examples of applications where improved measurement and control of copper grain structure and topography provide significant gains in value to the PCB fab process.

In the case of microvias, in addition to the traditional chemical analyses, white-light

microscopy inspections, weight gains/losses, and SIR readings associated with laser drill and the metallization of microvias, best practice has recently found that three different inspection steps for copper structural assessment to assure high-reliability results are also valuable to de-risking the process (Table 1).

The opportunity from these QC steps is to optimize the process recipes, and the field experience is that this can often yield large improvements in microvia reliability. The capability gains from the addition of this control scheme can then be leveraged to demand higher willingness to pay from clients, as well as improving capabilities to produce higher margin products. Multiple new greenfield fabs that we have engaged with have implemented these controls, and shown rapid ROI through both yield improvement and overall de-risking of the process.

Copper Foil and Seed Layers

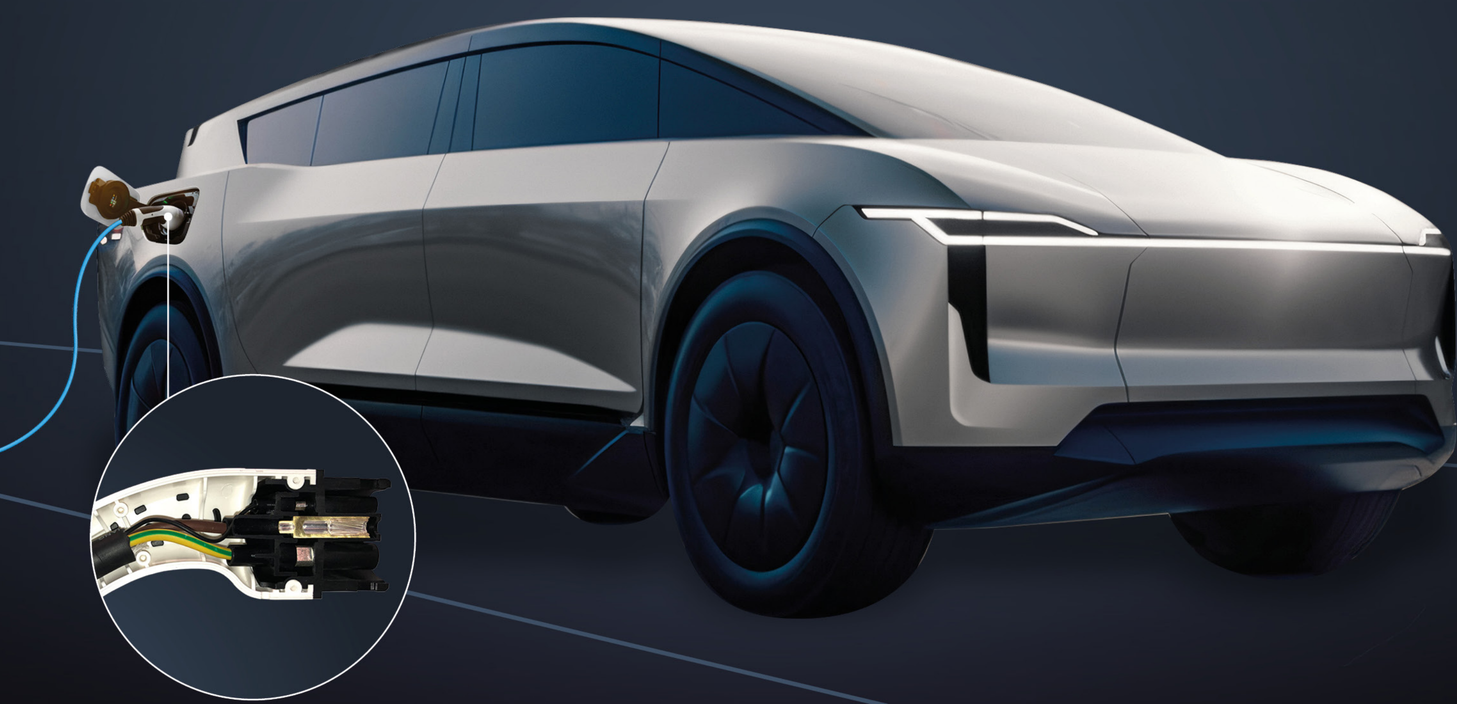
In the case of copper foil and copper-plated surface layers, the overall goal is to be able to etch through this layer as fast as possible, thereby achieving the least possible undercut/lateral-etching of the masked features and thereby achieving higher resolutions. The performance can be pushed in this direction by the utilization of low Rz/thin foils, modifying the grain structure to be more columnar for rapid etching, as well as through the utilization of 3D measurement of the average Cu height prior to etching/next-level plating for recipe set-ups.

Table 1: Microvia process: copper structure QC steps

Application	Procedure	Justification
After laser drill, for laser drill recipe set-up and qualification	Confirm that parameters are optimized to assure clean target pads, and a minimal heat-affected zone that can be cleaned by the post-treat process. (Usually done non-destructively on live product.)	Laser drill variables drift due to aging and contamination effects coupled with the fact that they are complex systems in need of mechanical calibration. It is subsequently important to have a qualified baseline that is checked at job setup.
After post laser drill desmear and microetch for product recipe confirmation	Confirm that laser drill+post-treat worked as planned (target pads are cleaned and meet mechanical roughness specs with minimal dielectric undercut. (Usually done non-destructively on live product.)	A clean, pink copper surface should be present with an even topography and a controlled Rz to assure that all post-laser processes did their job prior to metallization.
After galvanic copper recipe set-up and qualification	Confirm that the recipe is yielding the best engineered deposit for the application.	You can make microvias many times more reliable by tuning the grain structure (crystal size, orientation, and nano-twinning frequency are the biggest variables) and encouraging formation of annealed nano-twins across interfaces. the grain structure also has large influence on the etch rate.

mks
Atotech

Your taxi into the future



Silvertech® C – Silver-graphite for reduced wear off

Drive into the future with us! Our industry-proven silver-graphite process, Silvertech C, deposits a highly electrically conductive surface on power switches and connectors with high wear resistance requirements such as EV battery chargers. Silvertech C reduces wear-off by co-depositing weight 1–2 % graphite particles into the silver surface leading to significantly higher mating and unmating cycles of connectors. The self-lubricating effect of the graphite particles significantly reduces the coefficient of friction of silver surfaces thus reducing insertion forces without impairing its exceptionally low resistivity. The graphite particles are also an effective countermeasure against the well-known cold welding effect of pure silver surfaces.

Silvertech C is available in two types of carbon graphite sizes. This allows for more flexibility in silver thickness. The high-volume production-proven process is designed for rack tools depositing around 0.6 µm silver/min and deposits a satin-like greyish-silver surface.

To find out more about MKS automotive solutions, scan the QR-code to the right.

info@atotech.com

www.atotech.com

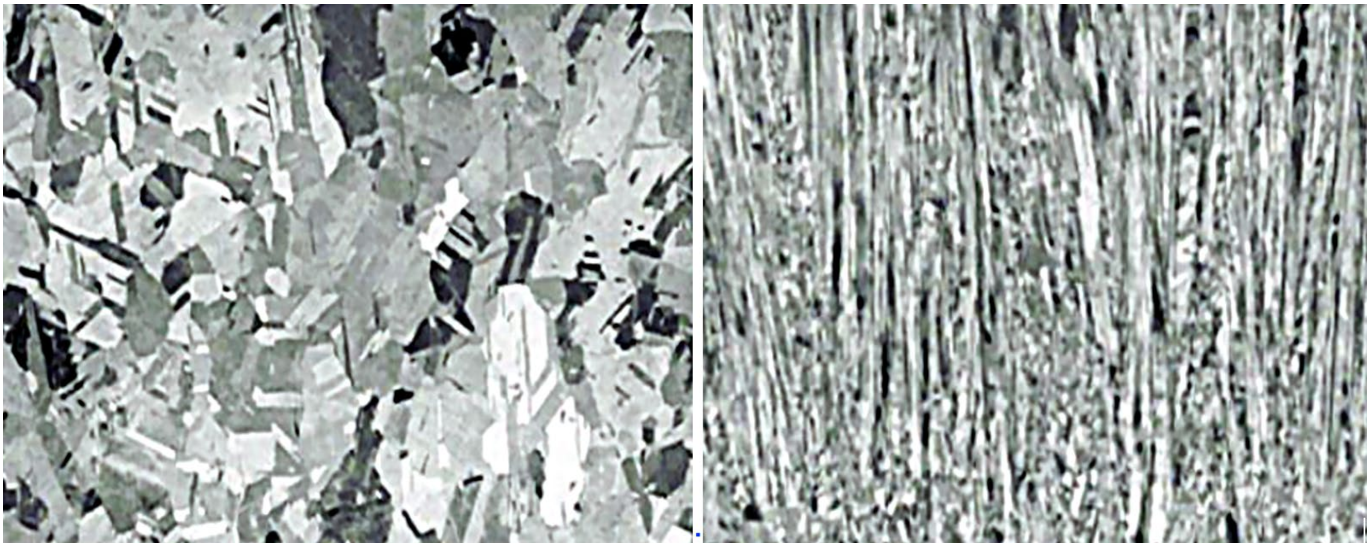


Figure 1: SEM image showing grain structure examples. Grain structure engineered for high-reliability (left), and engineered for sacrificial etching (right).

Annealing of the copper after plating with a controlled bake also changes the crystal structure, and if it is plated correctly, annealing will also make it more reliable. All these variables need to be measured and controlled to minimize process variation for producing advanced products.

The Measurement Kit

The fundamental measurement equipment used to perform all the checks in this article is a 3D microscope with a laser profilometry option (most use confocal microscopy plus lasers for profilometry). Units are available from many different suppliers on the market and range from $120,000 for simple standalone systems up to $225,000–275,000 for units mounted on automated full panel-size gantries.

The tools have been recently installed in multiple facilities in the U.S., and there is now a push to begin implementing them as part of highly automated work cells across many of the new PCB fabs. Other technologies also work for many of these applications; however, we have found that these 3D scopes with lasers provide the highest capital investment efficiency of all of the tools which we have evaluated in the NPI fab segment, due to the wide application range and ease of use to drive control and improvement.

The logic behind the 3D tools is that they do not require destructive analysis of the product, are able to provide pictures in full natural color, and require only a few hours of operator training to get going. In-process checks can be done with no prep on live product and complemented by coupon sampling for copper grain structure qualification. So, for the price of an AOI machine or less, the 3D scope/laser gives you a very precise engineering tool that can be used to perform a wide range of checks. 3D measurement of the shape of copper transmission lines post-etch, with confocal microscopy to feedback to the signal integrity model, is also a new scheme that is being implemented with these tools.

If 3D microscopy with laser profilometry is not part of your investment plan, and you are in the HDI/high speed/substrate markets, you should consider looking into it. PCB007

References

1. Online Wiki of Crystallography.

Alex Stepinski is the founder and CEO of Stepinski Group, and former vice president of GreenSource Fabrication.


A must-have for every company in the supply chain.
The IPC Checklist shows you where and which standards are used.
IPC BUILD ELECTRONICS BETTER
IPC CHECKLIST
for Producing Printed Board Assemblies
2023 Revision
Get yours here
IPC BUILD ELECTRONICS BETTER

Large Panel Processing

Happy's Tech Talk #23

by Happy Holden, I-CONNECT007

Much of our current focus is on the U.S. CHIPS Act, and for the future of PCB fabrication and assembly, this is appropriate. I started my career in manufacturing thin-film RF (sapphire) substrates, as well as conventional multilayers and HDI. I have researched and built IC substrates for the past 50 years. I know we depend on the technologies that trickle down from semiconductor and RF substrates.

Panel size is an important aspect of PCB productivity, yields, and cost. If a fabricator or assembler can process larger panels, then their productivity increases and their cost per board drops—provided that yields are not affected. Yields are an important process parameter, as they affect all costs and performance measures; they are driven by defect density, and that affects considerations of larger panel sizing. Defect density is not often talked about in PCB fabrication, unlike wafer processing, where it is a driving factor.

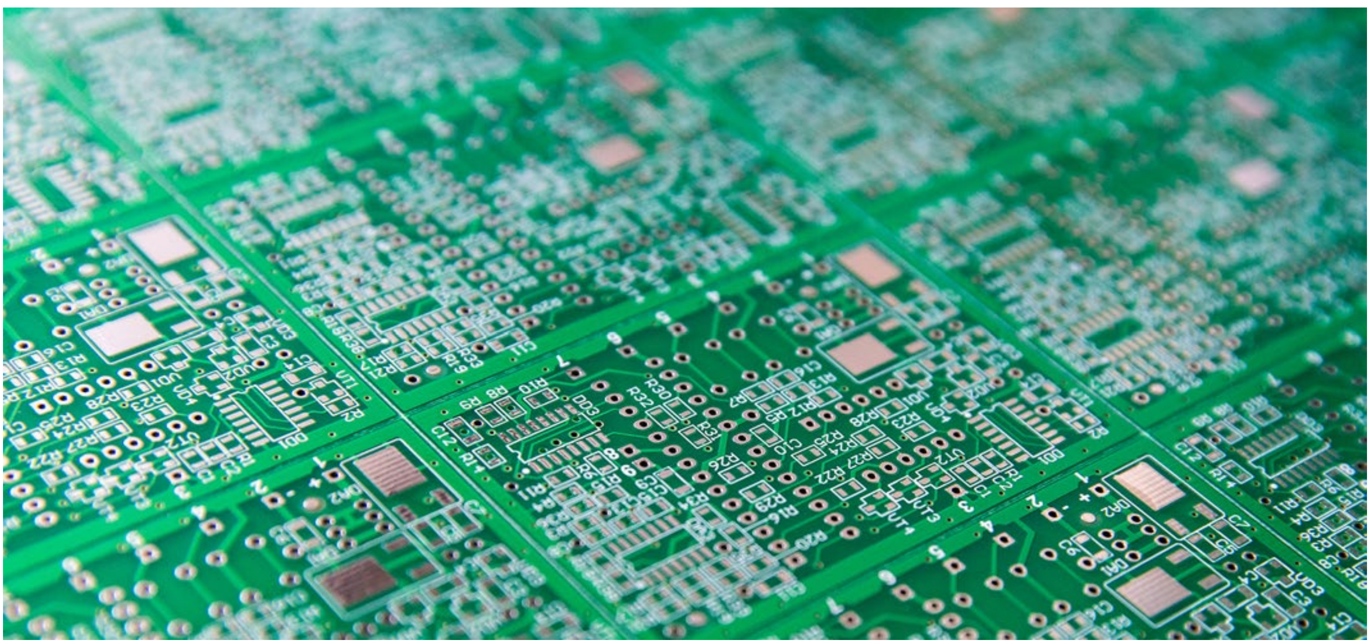
The Poisson Model is used as the defect density model to predict yields in semiconductor fabrication. I use the reciprocal of this equation to calculate a PCB's first pass yield:

FPY = 100/exp[(log CI/A)^B][1]

Where: FPY = first-pass yield (%)

CI = PCB complexity index

A, B = Fabrication capability coefficients



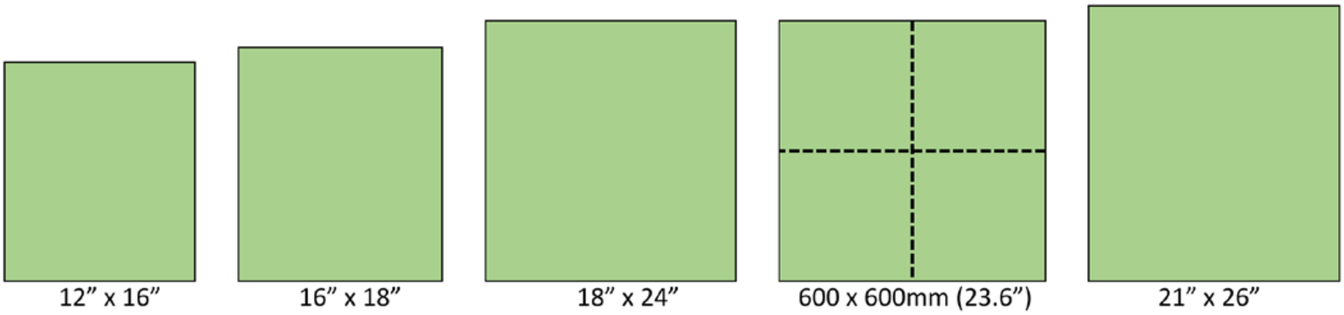


Figure 1: Organic substrate (IC and RF) panels have constantly increased in size over the years, up to 21" x 26" with 600 mm x 600 mm being the new standard for semiconductor panels.

When I started working in 1970, our IC and RF substrates were 100 mm in diameter (many were sapphire, as HP used silicon-on-sapphire, not pure silicon) and PCB panels were 12" x 12" (305 mm x 305 mm). Because of the popularity of our HP-35 scientific calculator, by 1972 we were making LED COB substrates on a new high-temperature laminate in 12" x 16" panels plated with nickel and silver (for thermocompression bonding). To reduce costs and improve productivity and capability, the panel size has constantly crept up to 21" x 26" (Figure 1). The new standard for IC substrate panels is 600 mm x 600 mm (Figure 1) and a production panel (Figure 2).

The semiconductor packaging industry's move to a 600 mm square panel provides the ability to segment the 600-mm panels into four 300-mm square sub-panels for use with conventional 300-mm round wafer probe test equipment. This was a driving factor for the short term (Figure 2). Extending process capa-

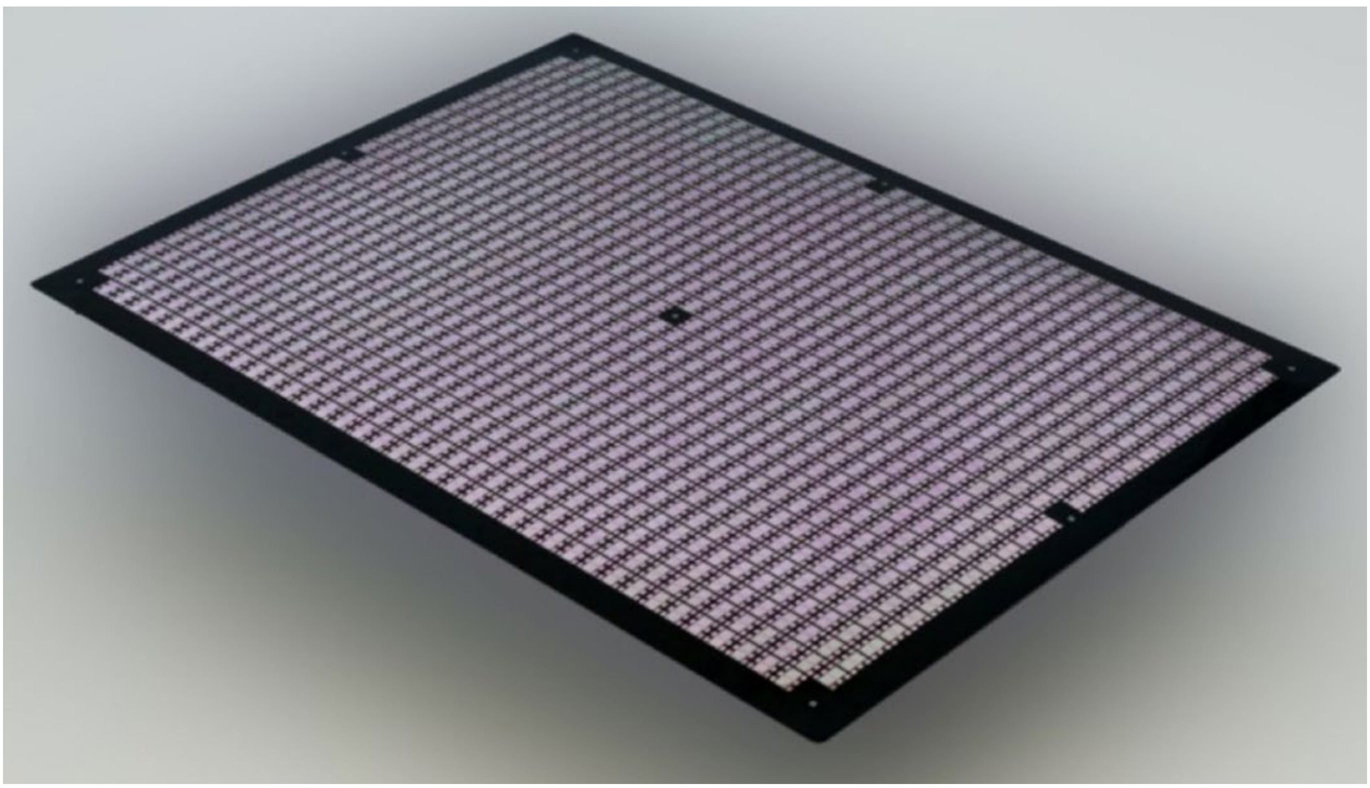


Figure 2: First results of PLC 2.0 is a fully populated panel with embedded chips. (Source: Fraunhofer IZM[2])

WORLD-CLASS TOOLS

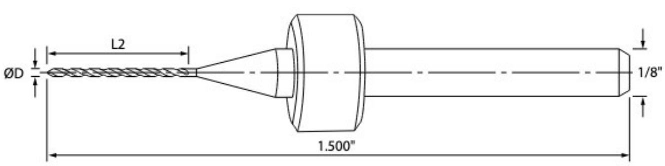
Insulectro, the largest distributor in North America of materials used in the manufacture of printed circuit boards and printed electronics, provides game-changing precision cutting tools to our customers.

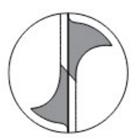
CALL 949.587.3200 FOR MORE INFORMATION

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

Drills, End Mills, Routers, Specialty Tools

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





4 Facet Point Geometry

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

bility for key lithography, metal deposition, and other processes to complete the 600-mm panel provided a challenging, yet achievable, target for equipment suppliers.

New Materials

The next generation of IC packaging is adapting to the demands of multi-die architectures. While silicon has supplied this requirement, its cost and characteristics are not ideal. Glass and improved organic materials have emerged to fill this need. This strategy is very consistent with the dramatic and emerging changes in electronic systems, such as in high-performance computing (HPC), AI, and a new era of self-driving and electric cars that potentially think and drive better than humans. This requires device, packaging, and computing architecture paradigms with an entirely different vision and strategy than transistor scaling alone. Packaging, which can be viewed broadly as system scaling, is now viewed as replacing Moore's law for enabling better devices and better systems (Figure 3).

Georgia Tech and its industry partners are developing a leading-edge glass packaging that is consistent with the needs in cost, performance, functionality, reliability, and miniaturization. In a technical article, they described the critical glass packaging technologies, and their R&D and commercialization status, as well as all the current and future applications. It compares glass packaging against other leading-edge technologies, such as Si and embedded packaging.

The requirement for the next-generation of substrate packaging must allow the shrinking geometries of interconnection and I/O pitch, lower dielectrics, and losses for the higher frequencies and suitability for the increased thermal heat dissipation required by all these devices. Glass is an ideal material, as it has

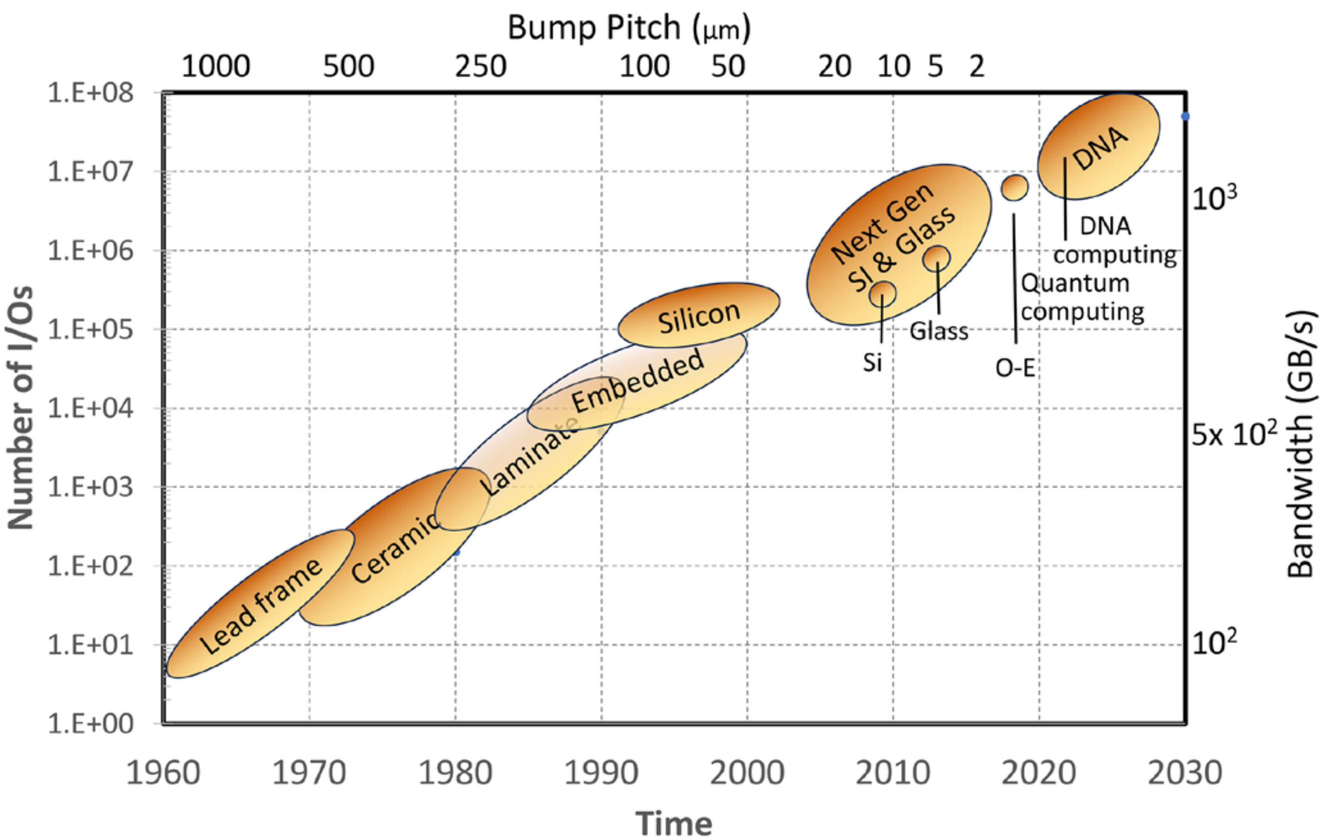


Figure 3: Moore's law for packaging I/O means exponential evolution of I/Os in the past six decades. (Source: Georgia Institute of Technology[3])

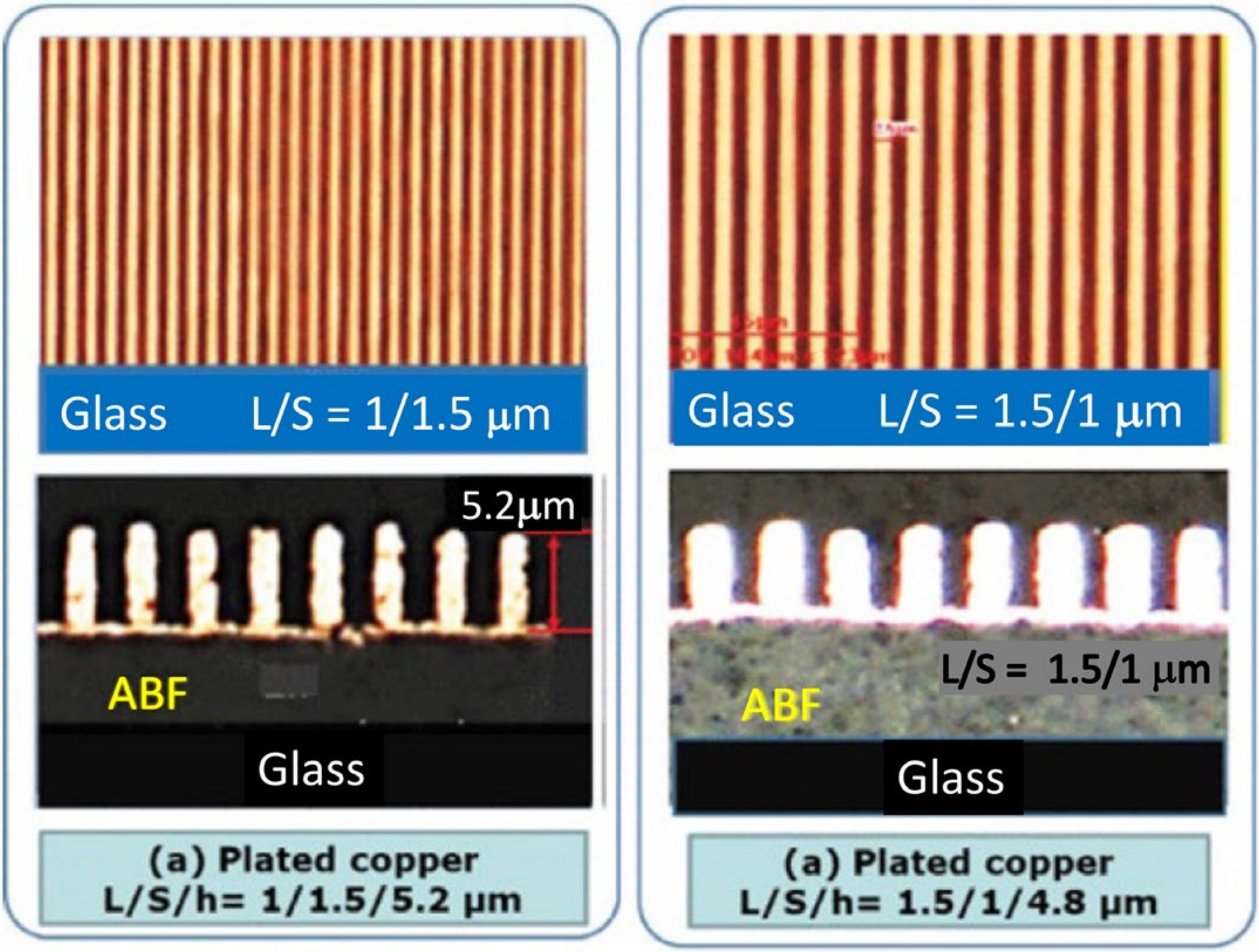


Figure 4: Next-generation high aspect ratio 1 µm RDL of organic film on glass for HPC development at Georgia Tech.

been modified over the years to perform many different requirements. Figure 4 shows a close-up of 1 µm and 1.5 µm t/s in the organic ABF film on a glass panel.

As shown in Figure 5, typical Cu pillar flip-chip bonding has a die pad pitch of 100 µm with an I/O density of 105 I/O mm^2. TSMC's integrated fan-out (InFO) has a die pad pitch of 55 µm with an I/O density of 314 IO/mm^2.[3] To further decrease interface pitch, new interconnect technologies were developed, such as Intel's EMIB (embedded multi-die interconnect bridge), which can achieve a die pad pitch of 45 µm with an I/O density of 492 IO/mm^2. The first-generation Deca M-Series, with a planarized structure above the encapsulated active die coupled with the patterning technology, achieved the same 45-µm interface pitch as compared to EMIB, without the need for complicated bridge chips embedded in substrates. With the new Gen 2 technology, this die pad pitch can be further scaled to 20 µm, thereby achieving a more than 5X increase in I/O density of 2518 IO/mm^2. Gen 2's advanced LDI and automatic optical inspection (AOI) equipment, combined with the patterning technology, provides a path for the ultra-high-density die pad pitch and RDL density required for chiplets and advanced heterogeneous integration. Through-glass-vias (TGV) and copper pillars are needed for interconnects and thermal heat spreading.

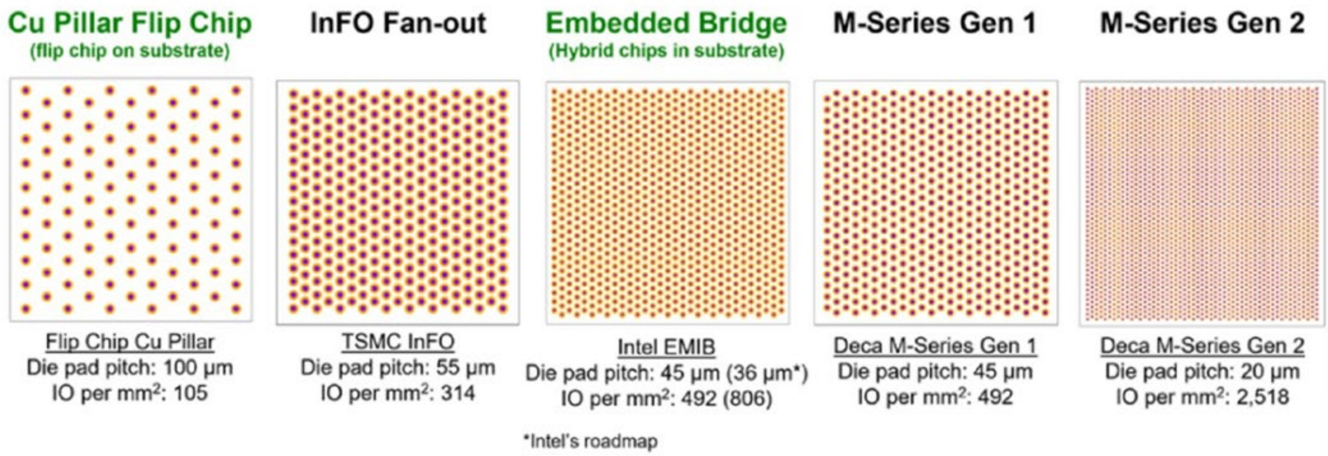


Figure 5: Comparison of interface pitch between different interconnect technologies and the M-Series from Deca. (Source: Deca Technologies, Inc.[4])

Summary

A 650 mm x 650 mm panel-level approach to fan-out (PLFO) technology is being utilized, which enables assembly of four 300-mm round or 300-mm square fan-out subpanels on a carrier panel. This technology enables the re-utilization of the reconstitution and die/package-level processing equipment, focusing the panel processing where the greatest cost benefit can be achieved in the redistribution layer process. The use of a carrier panel minimizes the warpage, permitting implementation of more RDLs without impacting processability. The flow is performed on the smaller form factor, minimizing die-shift considerations on the large panel. The same panel equipment and infrastructure can also be used for chip-last PLFO or high-density, high-quality coreless substrates. Process flow details will be shared based on a PLFO pilot line in use now. The new SEMI standard for large-panel arrays looks to lower the cost and improve performance and reliability for multi-arrays die substrates[5].

Glass packaging is emerging as a next-generation packaging platform beyond organic and silicon packaging. It has been developed in both chip-first and chip-last 2.5D and 3D architectures. Georgia Tech and its industry partners have developed all the building block technologies necessary to manufacture. **PCB007**

References

1. *HDI HANDBOOK,* Chapter 3, by Happy Holden.

2. "Next Steps For Panel-Level Packaging," by Mark Lapedus, *Semiconductor Engineering,* Dec. 20, 2021.

3. "Glass Panel Packaging as the Most Leading-edge Packaging: Technologies and Applications," by Rao Tummala, Bartlet Deprospo, Shreya Dwarakanath, SMTA Pan Pacific Symposium, Hawaii, 2020.

4. "Large-panel fan-out perspective on cost, yield, and capability," by Clifford Sandstrom and Robin Davis, *Chip Scale Review*, November 2022.

5. "A Hybrid PLP Technology Based on a 650mm X 650mm Platform," by Eoin O'Toole, Semiconductor Engineering, July 25, 2023.

Further reading

"600MM Wafer-Level Fan Out on Panel Level Processing With 6-Sided Die Protection," by Jacinta Aman Lim and YunMook Park, Proceedings of the International Wafer-Level Packaging Conference 2020, San Diego.

Happy Holden has worked in printed circuit technology since 1970 with Hewlett-Packard, NanYa Westwood, Merix, Foxconn, and Gentex. He is currently a contributing technical editor with I-Connect007, and author of *Automation and Advanced Procedures in PCB Fabrication*, and *24 Essential Skills for Engineers*. To contact Holden or read past columns, click here.

Planning, DFM, and Inspection: Key to High-reliability Fab

Feature Interview by the I-Connect007 Editorial Team

At PCB West, Andy Shaughnessy and Kelly Dack spoke with Marc L'Hoste, VP of West and South Americas for ICAPE Group, a company that supplies and manufactures high-tech PCBs at locations around the world.

We asked Marc to share some advice regarding high-reliability fab. In this conversation, Marc is clear that planning, pre-work, and inspection are the key ingredients to high-reliability success.

Kelly Dack: Marc, you fabricate quite a few high-reliability PCBs. How do you define high reliability and what is your perspective on what high reliability means in our industry?

Marc L'Hoste: IPC class 3 stands out for high-reliability printed circuit boards, but it also depends upon the type of industry and the type of customer. When we look at our own customers, about 20% of them are in the automotive industry, which has always been very complex to navigate. Automotive suppliers have very high standards to follow, such as the IATF ISO certification. For example, the quality levels are usually measured in parts per million (PPM) of manufactured parts. Over the past 15 years, we've been able to deliver fewer than 40 PPM for the automotive industry, which means less than 40 defective PCBs per million—significantly better than we see in stan-

Marc L'Hoste

dard consumer production where we are usually talking about 250 PPM. The same applies to other critical industries, such as aerospace, with certifications like AS9100, and critical processes with sampling and DPPM (defective parts per million) controls.

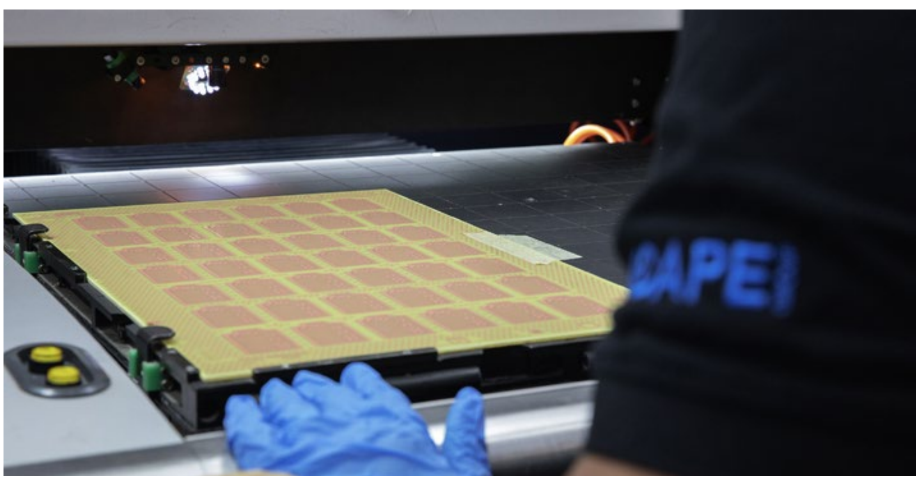
Dack: That's a pretty low defect rate. What are some of the most typical causes of failures you see in high-reliability fabrication?

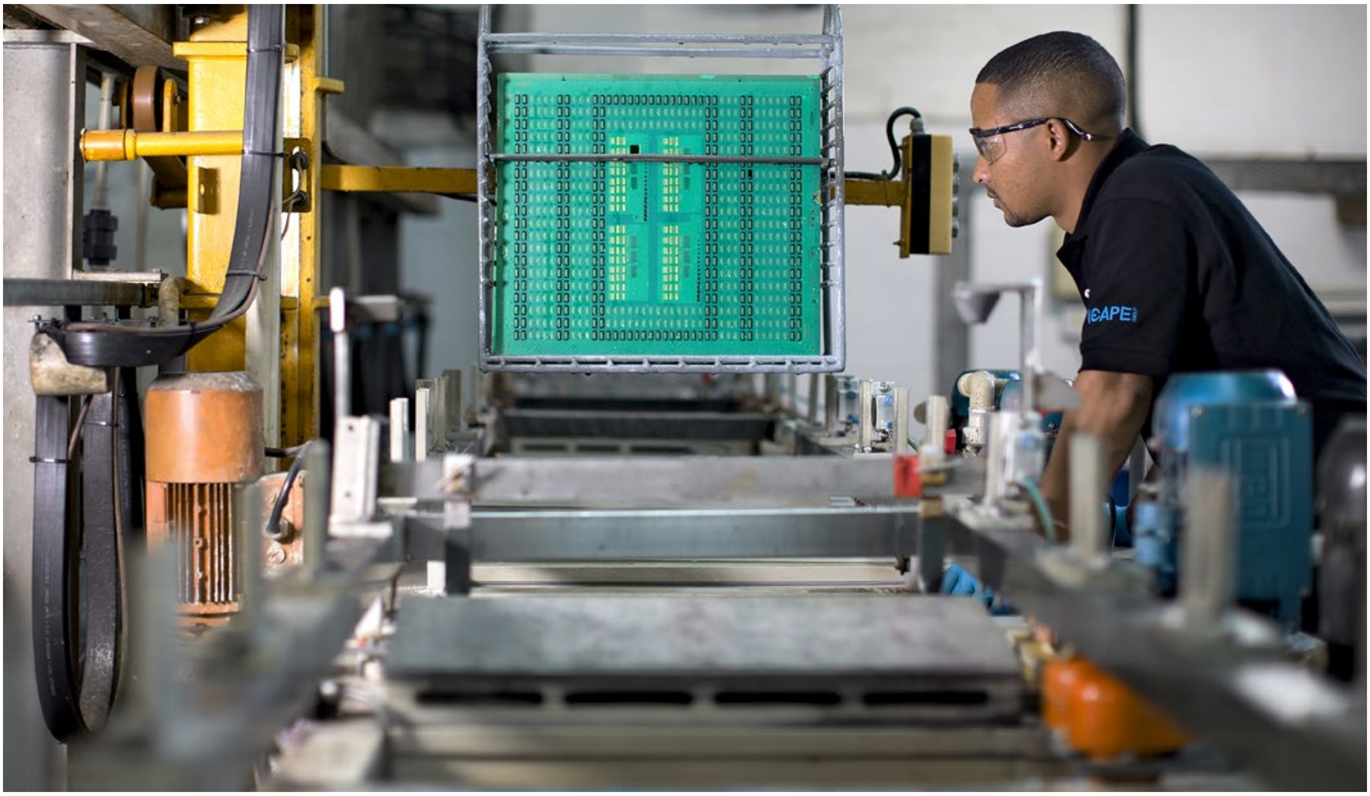
A lot of different points need to be checked. For me, it always starts with the design itself, and making sure that whoever designs the board follows the IPC guidance. We have spreadsheets and guidance sheets that we can share with our customers. Let's say Bob is tasked with selecting the right manufacturer for his boards. Bob must make sure that, if you build a flex or rigid-flex circuit board, you're not working with a factory that specializes only in rigid or metal PCBs. Selecting the right manufacturer usually means auditing them to make sure they have the right technical capabilities to build these boards. Choosing the right materials and finishes is also critical, and the finishes you select will depend upon the type of components on your boards. OSP (organic solderability preservative) is great, of course. It's very cost-effective, but if you have a BGA on your board, that might not necessarily be the right finish to choose.

Once you've nailed down all these different points, make sure that you do a good inspection on your board. A 100% test is always mandatory. Automatic visual inspection is very important, too. What we propose at ICAPE Group is a kind of double inspection. We have about 60 quality inspectors, and some of them are directly working with our engineering and manufacturing partners. Their job is to control the quality of our boards at these different locations. We have double points of inspection in the process: 100% inspection on our boards at the factory, and then inspection with our own quality people at our partners' locations before we ship.

Dack: For our designers who are designing to IPC class 3 expectations, what are some things for PCB designers to think about? Because it's possible to design a board that can't be built.

Before we even start a quotation, we do a software analysis of the Gerber files that the customers provide. In our case, we're using Ucamco software. This software makes sure





that all the basic points of the printed circuit board's dimensions, thicknesses, layers, type of solder mask, and so on, have been well defined in the Gerber files. If that's not the case—and it is often not the case—then we help our customers with guidances and give them options for different materials that they can use, depending on the application.

Dack: You offer a design review service for your customers to make sure all the technologies and materials will work together? That's a great idea.

Yes, that is the very first stage. Once we've done the quotation and receive an order from our customers, we have a secondary stage, which is technical questions—DFM. We just make sure that the board is manufacturable as it's been designed. Today, 60% of those technical questions are handled by our own engineering team of about 30–50 engineers in China, Mexico, the U.S., and Europe. Our job is to check the files, making sure we can manufacture, or preparing a list of technical questions, and always with options. We are not just telling our customer, "Oh, this doesn't work." We provide them with solutions. Then their job is simply to say, "We can accept this, but we cannot accept that." The last stage is also sending Gerbers to our customers for them to review one last time before we manufacture the PCB.

Dack: It's really good to hear that one of your first steps is a design review, to make sure that everything will line up for an EMS provider. As an EMS provider, one of our frustrations with other fabrication suppliers is that design teams send Gerber data and a panelization detail specification for how it's to be panelized. We send that for quotation and receive the quotation. But it's not until we cut the PO to build the boards that we receive the engineering queries to tell us all the ways that they can't make the boards. When that happens, it's very frustrating, so it's good to hear that you address this in a more successful way.

We've heard the same story from many of our customers. Being an ISO company, we always

strive to improve ourselves. We've used this system for at least five years. Customers are extremely happy with it because we don't only deliver reports; we also deliver a quotation. At more than 30 pages, the reports are very comprehensive.

Dack: Is your Ucamco system automated?

Yes. It's a fairly simple process. It takes a few minutes, I believe, to generate a report. Our engineering team has a look at the report before we start working on the quotation, making sure everything is noted, and eventually having a technical description for the customer if there was a big issue.

It's a system that works great for us. We have partners in many different countries, and our customers need quotations within 24 hours; with this system, we make our own quotations internally. We don't rely on petitions from different board houses. We do our own quotes. Because we're often in the same time zone as our customers, we can get inquiries in the morning and have quotations ready for customers same day.

Andy Shaughnessy: What advice would you give to fabricators who are having issues with reliability?

It's fairly simple: Take your time and do the work. Perform design reviews, DFM checks, and inspections. As I said, we have a double inspection process. Select the right partner for you and your technologies. There are thousands of board houses around the globe specializing in specific technologies.

Dack: Thanks so much, Marc. This has been great.

Thank you. I enjoyed it. **PCB007**

icape-group.com

I-007e FAST FACTS

Reliability of Printed Circuit Boards

Feature Article by Reza Ghaffarian
JPL NASA

Editor's note: This excerpt was taken from Chapter 60 of the Printed Circuits Handbook, 7th Edition, edited by Clyde F. Coombs and Happy T. Holden.

Reliability Testing Methods

Almost every reliability test program must solve the problem of determining whether an object is reliable in a time that is much shorter than the expected use period. Obviously, one cannot spend three to five years testing a personal computer that will be marketed for an even shorter time span or 20 years testing a military system. Depending on the failure mechanism, there are two approaches that may be combined: 1) accelerate the frequency of the occurrence that causes failure and test the ability of the object to survive the expected number of events; or 2) increase the severity so that fewer occurrences are needed. Drop tests that simulate shock during transportation are an example of the first approach. Since the time between drops does not affect the amount of damage caused, a lifetime of drops can be conducted in rapid succession. However, the effect of temperature and humidity on corrosion over the lifetime of the product can be tested only by increasing the temperature, the humidity,


CHEMCUT
CORPORATION
CHEMCUT
CORPORATION

the concentration of contaminants, or some combination of these factors. The difficulty is ensuring that the test reproduces and/or correlates to the failure mechanism in service.

To use this data for making true reliability predictions—that is, the probability of failures at a given time under given conditions—testing must be continued until enough parts fail that a life distribution can be estimated. Unfortunately, this process can be time consuming, so qualification tests are often substituted. Qualification test protocols specify a maximum number of failures that may be observed in a specified period in a sample of specified size. If few or no failures occur, a qualification test provides almost no information about the failure distribution; for example, the probability of failure during the next time interval is unknown. This limitation of qualification testing is minimized when the life distribution for properly manufactured samples is already known or can be estimated based on experience with similar designs.

> "For each failure mode, construct an acceleration model."

Many reliability or qualification testing approaches do not follow either of these schemes. Instead, they test the ability of the product to survive a sequence of tests under extremely severe conditions for a short time or small number of exposures. Again, this type of testing may be adequate when it is supported by long experience with both the product type and its use environment; however, it is risky because it is not based on ensuring that probable failure modes will not occur in the life of the product. When new technologies or geometries are introduced, the old tests may not always be conservative. By the same token, irrelevant failure modes that would not occur in service may be introduced by the harsh test conditions.

Design for Accelerated Reliability Testing

Accelerated reliability test design is summarized into seven key steps.

1. Identify the service environment and the acceptable failure rate over a specified service life.

2. Identify actual environment of the PCA (modified service environment). The service environment should be translated into the ambient environment actually experienced by the PCA. For example, the temperature experienced by the PCA is influenced by both power dissipation and cooling. The mechanical environment is influenced by shock-absorbing material, resonances, and so on.

3. Identify probable failure modes (e.g., solder-joint fatigue, conductive anodic filament growth). Accelerated reliability tests are based on the premise that the frequency and/or severity of the environmental exposure can be increased to accelerate the incidence of the failure that occurs in service in a known way, that is, that the data can be used to predict the life distribution for the in-service PCA environment. This assumption makes sense only if the same failure modes occur in the test as in real life. It cannot be overemphasized that the accelerated tests must be designed around the real failure modes. Probable failure modes may be identified from past service experience, the literature, or preliminary testing or analysis.

4. For each failure mode, construct an acceleration model. An acceleration model that allows test data to be interpreted in terms of the expected service environment is crucial to life distribution estimation. It is also extremely helpful in designing good tests, so ideally the acceleration model should be developed before the accelerated reliability tests are carried out. As an example, for solder-joint reliability modeling, Coffin-Manson relationship relating cycles-to-failure to strain in solder joints to rigid components can be used. The relationship predicts that increasing strain will decrease the number of cycles to failure in a specific way. Within a certain temperature range, increasing the temperature cycling range is a way of increasing the strain.

In general, the acceleration model should be based on the rate-controlling step in the failure process. In some cases, the rate will be determined by an Arrhenius type equation; for example, if diffusion is the rate-controlling process:

$$D = D_o \exp(-Ea/KT) \qquad X \propto \sqrt{Dt}$$

$$t_2 = (D_1/D_2)\, t_1 = t_1 \exp[-E_a/K\,(1/T_1 - 1/T_2)]$$

where D = diffusion rate
D_o = diffusion constant
E_a = activation energy for the process
K = Boltzmann constant

and T_1 and T_2 and t_1 and t_2 are two temperatures and corresponding equivalent diffusion times.

Note that even when temperature is an important factor, *an Arrhenius relationship may not exist;* in the preceding thermal cycling example, the failure rate is roughly proportional to $(\Delta T)^2$. Some acceleration models will be explored in the following sections.

The limits of applicability of an acceleration model are as important as the model itself. Increasing or decreasing the temperature too much may promote new failure modes that would not occur in service or invalidate the quantitative acceleration relationship. For example, if the temperature is elevated above the glass transition temperature (*Tg*) of the board, the z-axis CTE increases sharply, and the modulus decreases. This may actually lessen the strains imposed on solder joints, but it may also promote plated through-hole failures.

Finite element modeling/analysis (FEM/FEA) can be invaluable in developing and/or applying acceleration models for thermal and mechanical tests. Two-dimensional nonlinear modeling capability will usually be required in order to get meaningful results. Models can be constructed to estimate the stresses and strains in the material (e.g., the Cu in a PTH barrel or the solder in a surface-mount or through-hole joint) under operating conditions as well as under test conditions. These estimates will be far more accurate than the simple models provided in this overview because they can account for the interactions between materials in a complex structure and both elastic and plastic deformation.

5. Design tests based on the acceleration models and accepted sampling procedures. Using the acceleration model and the service environment and life, select test conditions and test times that simulate the life of the product in a much shorter period of time. The sample size must be large enough that it is possible to determine whether the reliability goal (acceptable number of failures over the service life) has been met.[15] Ideally, the life distribution in the accelerated test should be determined, even when the test period must be extended to do so.

6. Analyze failures to confirm failure mode predictions. Since an accelerated test is based on the assumption that a particular failure mode in the accelerated test is the same one that occurs in service, it is important to confirm by failure analysis that this assumption is valid. If the failure mode in the accelerated test is different from the one expected, several possibilities should be considered.

1) The accelerated test is introducing a new failure mode different from the one that will occur in service. Usually, this means that the acceleration of one parameter (e.g., frequency, temperature, humidity) was too severe.
2) The initial determination of the dominant failure mode was incorrect. In this case, to understand the significance of the test

results, a new acceleration model must be developed for this failure mode. The new failure mode may be promoted more or less effectively by the test conditions than the mode originally assumed.

3) There may be several failure modes. In this case, the two failure distributions should be considered separately, so that life predictions will be meaningful. The difficulty in determining which of the above scenarios holds is that for genuinely new technologies or service environments, the failure mode in service may not be known. In these situations, it is desirable to conduct a parallel test with less aggressive acceleration for comparison.

7. Determine life distribution from accelerated life distribution. The accelerated life distribution should be determined by fitting the data with the appropriate statistical distribution, such as the Weibull or log-normal distribution. The life distribution in service can be determined by transforming the time axis of the life distribution using the acceleration model. This predicted life distribution in service can then be used to estimate the number of failures in the specified service life.

The following discussion of testing for some specific failures will provide examples of this methodology.

Printed Circuit Board Reliability Tests

Thermal: PTH failures are the predominant source of PCB failures in service, and predicting them is the primary goal of PCB testing at elevated temperatures. PTH reliability testing should simulate the thermal excursions of a PTH throughout its life. Generally, the most severe thermal cycles are experienced during assembly and rework. One of the older acceptable thermal stress test is MIL-P-55110 (also found in IPC-TM-650). Following baking at 120–150°C, the specimens are immersed in an RMA flux and floated in a eutectic (or near-eutectic) Sn-Pb solder bath at 288°C for 10s. For lead-free, the bath temperature is at 260°C or higher. Following the test, the samples are cross-sectioned, and the PTHs are examined for cracks. This is a severe test that ensures that the sample will survive a single wave-soldering or solder pot rework cycle.

Most thermal cycling tests for PCBs cycle the PCB repeatedly over a wide temperature range; many are actually thermal shock tests using liquid-liquid cycling. The results of five accelerated tests with different temperature extremes, ramp rates, and dwell times have been compared by the IPC, which also provides a simplified analytical model to estimate PTH life.[11] The results of all tests suggest the same approaches for maximizing PTH reliability, but they do not all correlate well quantitatively.

Figure 1 shows a number of suitable test coupons for PCB PTH/microvia evaluation. These include:

1) A test coupon that contains 3,000 PTHs and varying annular ring sizes[13]
2) An IST (interconnect stress testing) coupon with hundreds of PTH and microvias with direct current induced thermal cycling[16]
3) A HATS (highly accelerated thermal shock) coupon with four daisy-chain nets and up to 36 coupons per chamber and air-to-air cycling[17]
4) PCQR[2] (process capability, quality, and relative reliability) test panel for standard comparison of printed circuit board manufacturing processes[18]

Mechanical: PCBs are rarely subjected to mechanical tests that could cause electrical failures; however, adhesion of both Cu and solder mask to the laminate is critical and is often tested. Loss of solder mask adhesion can provide a place for corrodents and moisture to accumulate, which can be the cause of electrical failures when the board is exposed to temperature and humidity.

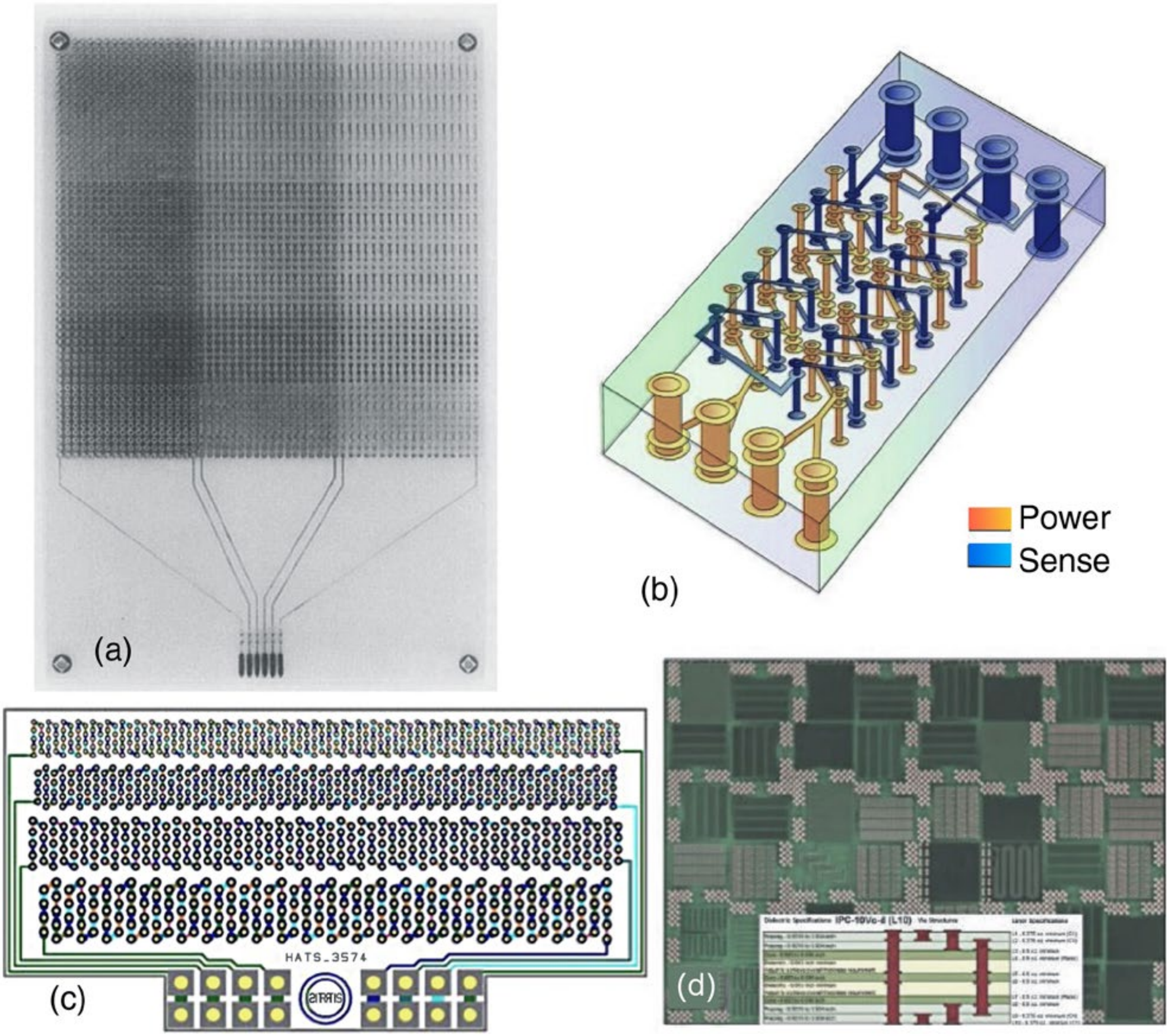


Figure 1: Representative of PTH reliability test coupons. (a) coupon contains three sets of 1000 PTHs interconnected in series, (b) interconnect stress testing (IST) coupons, (c) highly accelerated thermal shock (HATS) coupon, and (d) process capability, quality, and relative reliability (PCQR2) coupon.

Adhesion is commonly tested using the peel test described in IPC-TM-650, Method 2.4.28. The simplest version of this test is conducted by scribing the adherent and dividing it into small squares. If the Cu or solder mask pulls off with a piece of tape with strong adhesive, the adhesion is inadequate. More quantitative tests that measure the actual peel strength are performed primarily by laminate and solder mask suppliers.

Temperature, humidity, bias: These tests are designed to promote corrosion on the PCB surface and conductive anodic filament growth, either of which can cause insulation resistance failures. Surface n insulation tests utilize two interleaved Cu combs with an imposed DC bias across the combs. These combs may be designed into existing boards or a coupon such as the IPC-B-25 test board (Figure 2) may be used. The measured resistance (ohms) from the comb pattern can be converted to surface resistivity (ohms per square) by multiplying the measured resistance by the square count of the pattern. The square count is determined geometrically by measuring the total length of the parallel traces between the anode and cathode and dividing by the separation distance. Special precautions are needed

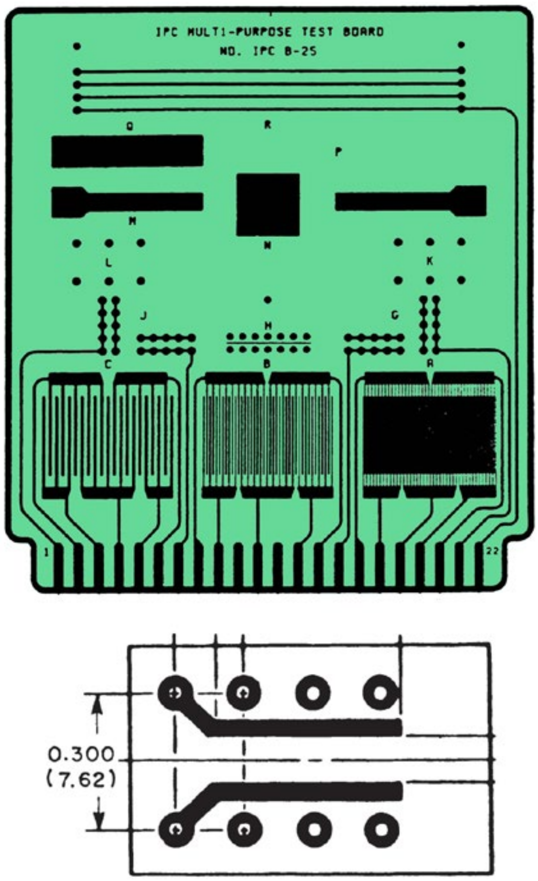


Figure 2: Test coupons used to check moisture, insulation, and metal migration resistance: (a) the IPC-B-25 test board, used to qualify the process; (b) The Y coupon, designed to be incorporated into production boards for statistical process control. (Source: IPC-SM-840.)

to make accurate measurements of insulation resistance.[19] Measurements of resistance above 10^{12} are very difficult and require careful shielding. Measurements of resistance below 10^{12} can be conducted in most laboratory environments if certain precautions are taken.

The actual tests are usually conducted at elevated temperature and humidity with an applied DC bias. A test for moisture and insulation resistance of bare printed circuit boards is included in IPC-SM-840A. The severity of the test depends on the intended use environment; for typical commercial products (Class 2), the test is conducted at 50°C, 90 percent RH, and 100 Vdc bias for seven days. The minimum insulation resistance requirement is 10^{8} Ω. The military test procedure for moisture and insulation resistance is specified in PCB military specification, MIL-P-55110.[20] The moisture resistance test should be conducted in accordance with military specification, MIL-STd-202, Method 106, with applied polarization voltage (100 Vdc) and Method 402, Test condition A.

IPC-SM-840A also includes a test for electromigration resistance. The test is conducted at 85°C/90 percent RH at a 10 Vdc bias with a limiting current of 1 mA for seven days. A significant change in current constitutes a failure. The samples are also microscopically inspected for evidence of electrolytic metal migration. A common test for dendritic growth due to flux residues is 85°C/85 percent RH/1000 h at a -20 Vdc bias. These tests are empirically based; however, several investigators have attempted to develop acceleration factors for these and similar tests.[21,22]

More recently, accelerated reliability testing become an inevitable requirement, particularly for high density interconnect (HDI) PCBs with microvias. IPC, as a leading PCB authority, has shifted its focus toward performance-based acceptance testing, recommending using test coupons to uncover latent microvia failures—a departure from acceptance of reliability solely based on conventional microsectioning evaluations. IPC issued a warning since in a number of examples in high-profile hardware, failures were not manifest itself after bare PCB fabrication, inspection, and acceptance using thermally stressed coupons with traditional microsections and light microscopes alone.

Data has been presented showing that conventional inspection techniques are no longer an effective quality assurance tool for detecting failures of microvia-to-target plating. So, it is critical to review the above acceleration test methods and tailor them for a product-specific environmental requirements, e.g., automotive and aerospace, in order to perform effective screening and life projections based on physics of failure and underlying mechanisms. For detailed reliability guidelines, IPC's 9700 series is an invaluable resource specifically tailored testing covering thermal cycling, mechanical loading (such as bending) and evaluating resistance of PCB pad to cratering. PCB007

References

11. "IPC-TR-609, Round Robin Reliability Evaluation of Small Diameter Plated Through Holes in Printed Wiring Boards," *IPC Tech. Report,* September 1988.

12. IPC.org.

13. "Reliability of Printed Circuit Assemblies," Chapter 60, by J. Glazer, *Printed Circuit Handbook,* 6th edition, 2007.

14. "Three-Step Concept (TSC) in Modeling Microelectronics Reliability (MR): Boltzmann-Arrhenius-Zhurkov (BAZ) Probabilistic Physics-of-Failure Equation Sandwiched between Two Statistical Models," by E. Suhir, *Microelectronics Reliability,* Vol. 54, 2014.

15. *Applied Reliability,* Van Nostrand Reinhold, by P. Tobias and D. Trindade, New York, 1986.

16. "IST coupon testing," PWB Interconnect Solutions Inc.

17. "HASTS testing coupons," website, Integrated Reliability Test Systems, Inc.

18. "PCQR^2," Integrated Reliability Test Systems, Inc.

19. ASTM D 260-78, Standard Test Methods for DC Resistance or Conductance of Insulating Materials.

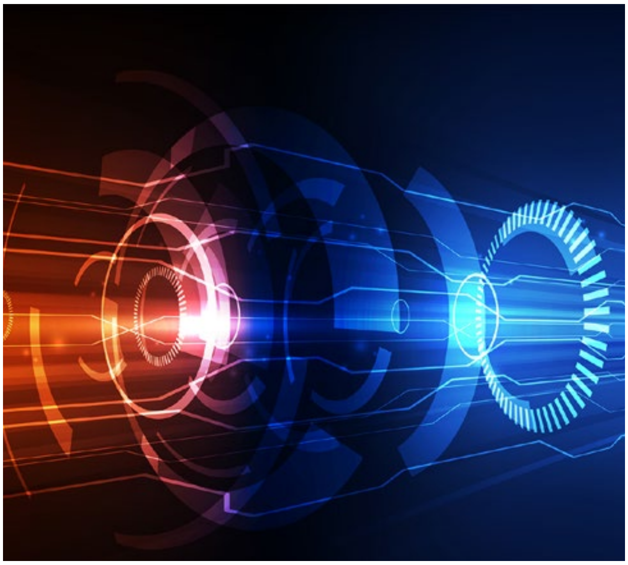
20. Military specifications, Land and Maritime, 2015.

21. "Accelerated Life Testing of Flexible Printed Circuits: Part I and II," by P.J. Boddy, R.H. Delaney, J.N. Lahti, E.F. Landry, and R.C. Restrick, 14th Annual Proceedings of the IEEE Reliability Physics Symposium, 1976.

22. "How Clean Is Clean in PCA," by K-L Wun, M. Ostrander, J. Baker, Proceedings of Surface Mount International, 1991.

Reza Ghaffarian, PhD, has spent almost 30 years leading reliability and quality assurance activities in advanced packaging at the Jet Propulsion Lab.

Microvia Failures: Time for a New Paradigm



by Tim Estes and Nick Meeker

The weak microvia interface problem continues to be a significant reliability issue for the PCB manufacturing industry. Many OEMs have mitigated the issue by utilizing staggered rather than stacked microvias and not placing microvias directly over buried mechanical vias.

While this has provided temporary relief, the need to solve the issue still exists. We currently see very little research and/or development directed towards improving microvia reliability. It is as if the industry has resigned itself to testing the quality of microvias post-fabrication rather than improving the process used to manufacture microvias.

At a higher level, we believe our manufacturing processes have become too complicated, resulting in lower yields, higher costs, and longer lead times. We need a paradigm shift in how printed circuits are manufactured. America has always led the way in the development of new technologies and it is time for our industry's creative people to be given the opportunity to rethink how we manufacture printed circuits.

(Source: Conductor Analysis Technologies Inc.)

MilAero007 Highlights

IPC Day Netherlands: A Focus on U.S. and EU Aerospace Electronics ▸

I was delighted to accept the invitation to attend the Oct. 4 IPC Day Netherlands: Aerospace Electronics, at ESTEC, the technical and scientific heart of the European Space Centre in Noordwijk. It's not the easiest place to get to from the UK, something I regretted.

USPAE, DoD Launch $10 Million Defense Business Accelerator ▸

In a major initiative to innovate how the Department of Defense (DoD) spurs commercial technology development, the U.S. Partnership for Assured Electronics (USPAE) and DoD launched a Defense Business Accelerator (DBX) to open doors for industrial base growth and stimulate private investment.

MacroFab Announces ITAR Compliance ▸

MacroFab Inc., the operator of North America's largest technology platform for electronics manufacturing, is pleased to announce its recent ITAR registration from the Directorate of Defense Trade Controls (DDTC). This registration underscores MacroFab's commitment to meeting the evolving security needs of its clients, reinforcing its position in the manufacturing industry.

CHIPS Act, One Year On ▸

Fresh off his annual meeting with the Printed Circuit Board Association of America (PCBAA), Travis Kelly, CEO at Isola Group and chairman of the PCBAA, gave us an update on government legislation in the United States that directly and indirectly affects the printed circuit board industry.

IPC's PCB Technology Trends Report Details How PCB Manufacturers Meet Current and Future Technology Demands ▸

PCB Technology Trends 2023, IPC's biennial global study, is now available. The survey-based study shows how printed circuit board (PCB) manufacturers are meeting today's technology demands and looks at the changes expected by 2028 that will affect the whole industry.

Amitron Becomes MIL-SPEC Certified ▸

Amitron, a leading U.S.-based manufacturer of printed circuit boards (PCB), brings an additional 80,000 square feet of manufacturing space into the overall MIL-SPEC ecosystem with high automation, and the latest equipment technology and processes for military and aerospace applications.

Northrop Grumman's Integrated Battle Command System Tested in Homeland Defense Scenarios ▸

Northrop Grumman Corporation's Integrated Battle Command System (IBCS) defended the National Capital Region against simulated cruise missile and compromised aircraft in a series of recent demonstrations, an evolution of technical capabilities for the system.

Airbus Helicopters Pioneers User-friendly Ways to Fly eVTOLs ▸

Airbus Helicopters' demonstrator FlightLab has successfully tested an electric flight control system in preparation of a new human machine interface (HMI) that will equip CityAirbus NextGen, Airbus' eVTOL prototype.

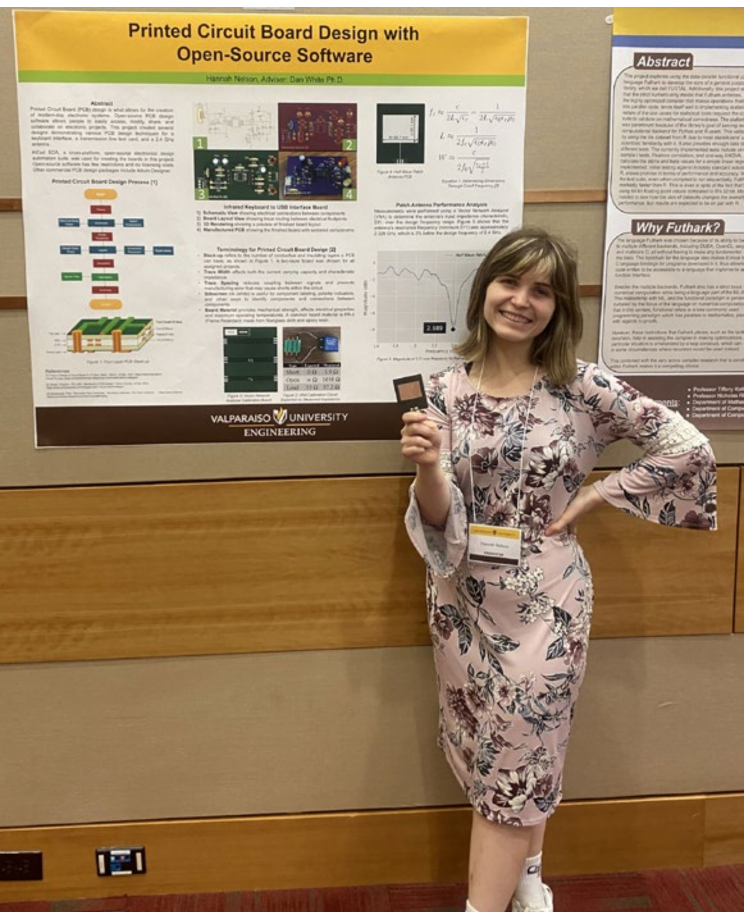
In a hurry?
We're the fastest in the business!
• ISO 9001:2015 Certified
• AS 9100 Rev D Certified
• MIL-PRF-31032-C Qualified
• MIL-PRF-55110 Qualified
• ITAR Registered
U.S. AIR FORCE
Prototron Circuits
America's Board Source
Serving the industry for over 35 years
prototron.com
(520) 745-8515

Reimagining PCB Design in Education

The New Chapter

by Hannah Nelson, TEXAS INSTRUMENTS

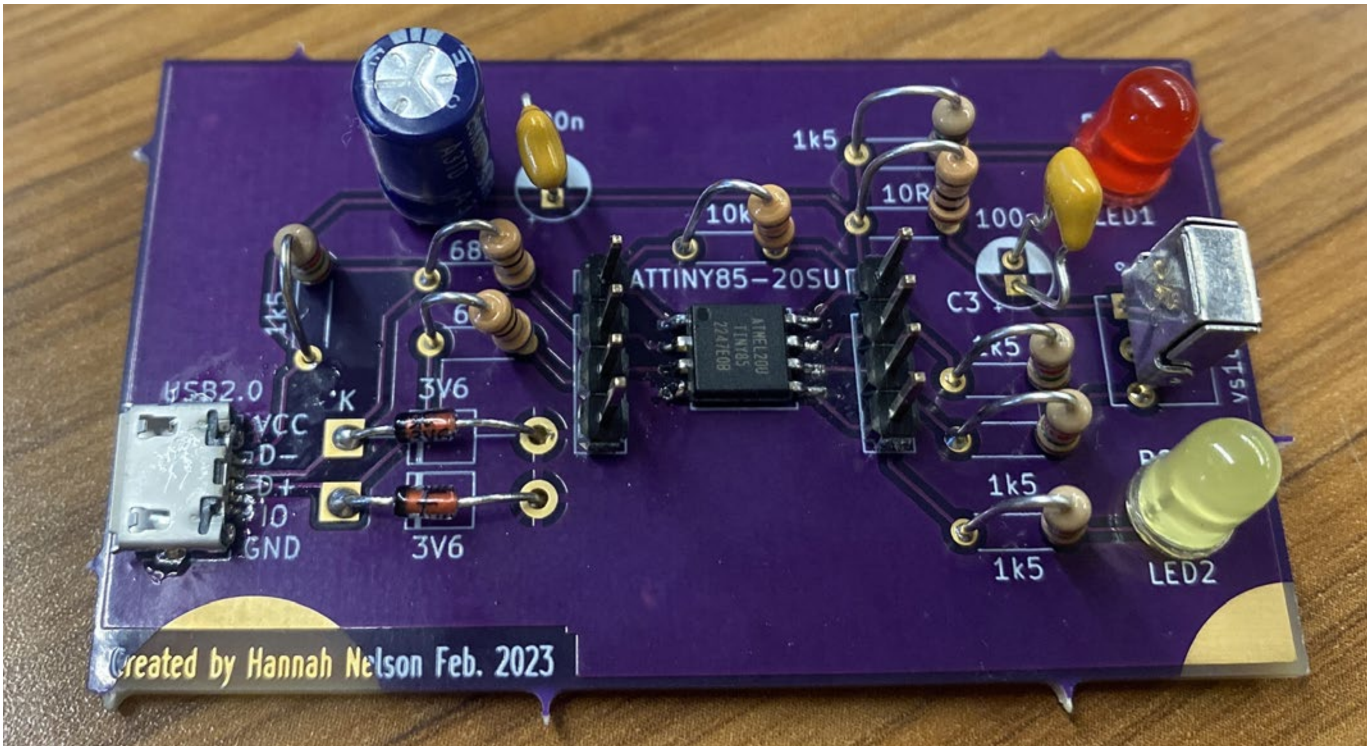
I've been reminiscing about my college classes, especially the ones proving to be the most valuable in my early career. One class that has given me the upper edge in my new job was an independent study on advanced PCB layout, where I learned about the PCB design and manufacturing processes. In that class, I designed several boards, ranging from antennas to fun kits for students, and I learned more about fundamental hardware electrical principles and the mechanics behind a PCB.



Before taking this class, it felt like there were key concepts missing in my educational experience because I could not see the theory working in real life. Once I created my first PCB, I finally understood how concepts like trace width can cause an impedance change and how the placement of a component might affect the signal integrity of the circuit. Sparks went off in my brain as I began linking theory to a reality that felt previously out of reach. Learning PCB design in the classroom may give the next generation of students an opportunity to link theoretical understanding to reality and see how the foundation of electronics can come to life.

PCB design plays an important role in high-reliability fabrication, and there is an expectation for designs to adhere to safety protocols and standards. The PCB industry comprises about 30% of business/retail/computer industries and about 28% of the communications/telecom industries[1]. Designs range from small toys to giant aircraft, so it's important that the designer works closely with the fabricator, even during the beginning stages, to ensure that standards and safety protocols are upheld during fabrication. Designers must design for reliability, while ensuring that the design meets the customer's specifications throughout the product's lifecycle. In fact, "for any new product, around 70-80% of production costs come from decisions made during the design phase."[2]

Designers need the skills to work with fabricators, adhere to standards and safety proto-



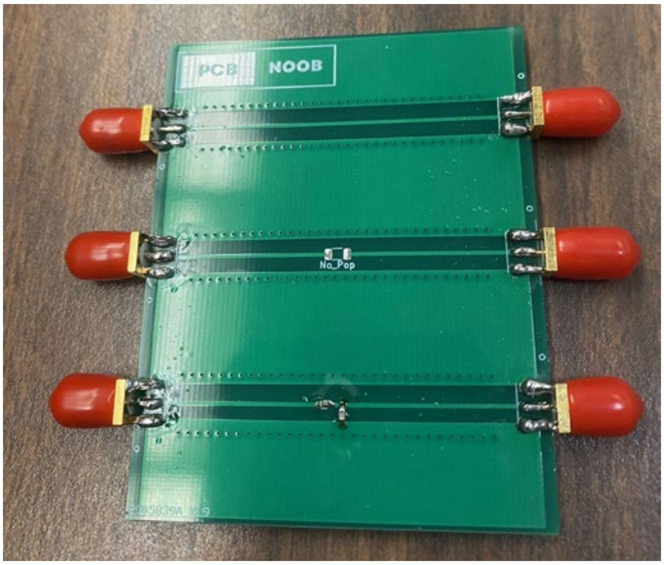
cols, and keep production costs low. What better way to gain those skills than learning them in the early stages of college and your career.

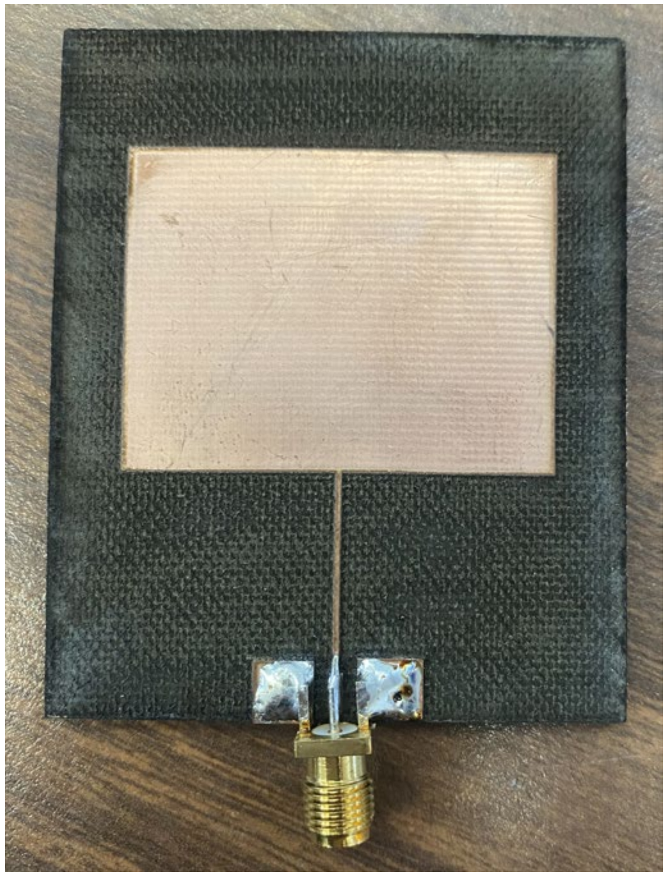
If I had not taken my PCB layout course, I would have needed to learn these things on my own. During the senior design process, projects often need PCB design or soldering—skills typically not offered in an electrical engineer's curriculum. This leaves many students to learn soldering and PCB design either before or during the senior design process. Why is it that something so undeniably necessary for senior engineering students to know is left to learn on their own? In these situations, students miss out on the opportunity to learn important practices and industry standards. Sadly, they miss out on tips and tricks used in the industry for complex projects. Think how much further ahead they would be, and how much more they could contribute to sophisticated projects, if they came into a job already having some of these design and soldering skills.

Many graduating students entering a hardware-based workforce really need to possess a basic PCB skill set, but with PCB design and manufacturing so frequently left out of the curriculum, how will they build that strong foundation?

Many universities are already making headway toward integrating PCB design and assembly within their programs, including Carnegie Mellon, Rochester Institute of Technology, and Massachusetts Institute of Technology. Even at Valparaiso University—my alma mater—they integrated a one-credit introductory course this year. Students can choose and design PCBs, select and solder available parts for their boards, and work with manufacturers to ensure the design is up to industry standards.

George Mason University, for example, introduced students to a PCB design course that enabled them "to design at least a two-layer PCB from start to finish."[3] Students went through the entire PCB design process, from "getting acquainted with the general features" to "generating a bill of materials" and "ordering parts for the project." Training on soldering and a tour of a PCB manufacturing lab helped students learn and understand the different steps within the PCB fabrication process, such as using a hot reflow oven and the proper uti-





lization of solder paste. Interestingly, students also learned the importance of time management when designing a board, to “spend sufficient time building the circuit” that would ensure the board could be manufactured during the specified project time because simulation needs adequate time to ensure the circuit works properly. The course taught them the importance of starting the design process early enough to allow for design optimization.

Overall, students completed the class “more motivated and better prepared to tackle more sophisticated design projects.” To me, this shows that when someone can obtain a basic understanding and skill set, they’re ready to take on more rigorous design challenges. Students learn to manage their time while optimizing designs.

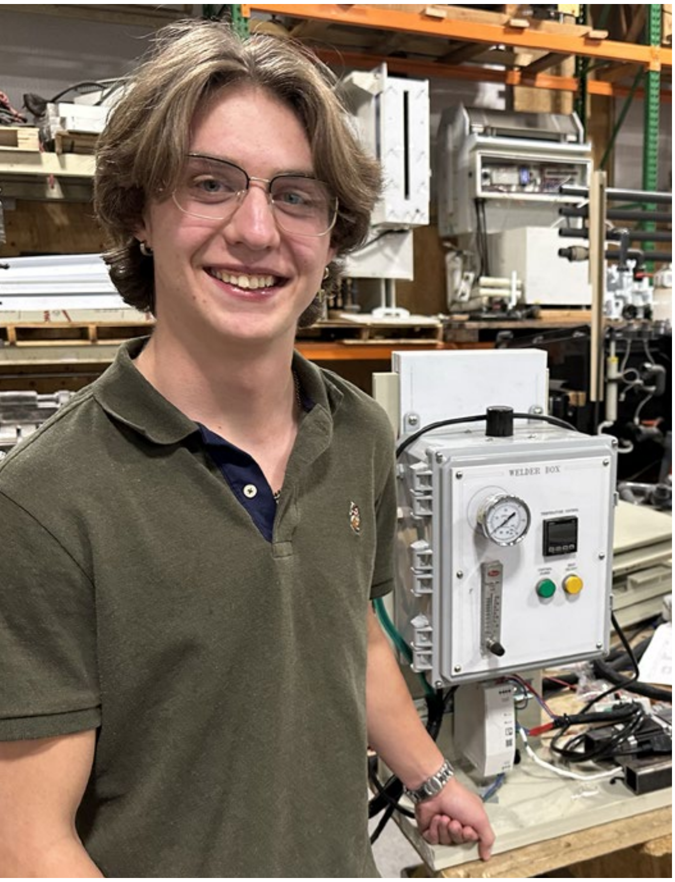
Even if colleges cannot fit in a semester-long course, I would love to see these topics taught in a few general engineering lab sessions to at least familiarize students in a hands-on setting. If neither option is available, I implore the greater electronics manufacturing industry to support their local IPC chapter. Together, we can pull industry forward.

When looking back on my college experience, our IPC chapter provided opportunities for us to learn skills like soldering and PCB design. The IPC Education Foundation even offered free courses to learn more about the PCB manufacturing process. The things I learned there I am now using whenever I am tasked with designing a PCB. With PCB design experience under my belt, I am better equipped to handle the unpredictable challenges that may come my way. **PCB007**

References

1. “The Global Printed Circuit Board Market: Key Insights, Forecasts, and Growth Outlook,” MKTPCB.com.
2. “6 PCB Design Stats to Make You Look Smart Around the Water Cooler,” by David Marrakchi, Altium, Feb. 21, 2017.
3. “Development of a Printed Circuit Board Design Laboratory Course,” by P. Kurtay, 2020 ASEE Virtual Annual Conference Content Access.

Hannah Nelson is a validation engineer at Texas Instruments and in her third year of IPC’s Emerging Engineer Program. She is a former IPC Student Board Member. To read past columns, click here.



Meet the New Workforce at IPS

Article by Barry Matties
I-CONNECT007

As the workforce continues shifting to the next generation—with a noticeable lack of appeal toward manufacturing careers—it's even more important to share the voices of those who have chosen to work in manufacturing. We believe that raising awareness will inspire others to consider manufacturing as the strong and viable career path that it is.

I recently visited IPS in Cedar City, Utah, which has been manufacturing wet process equipment for printed circuit boards fabrication for more than 30 years. While there, I conducted candid one-on-one interviews with several of the team members. They shared their views on manufacturing, their roles and challenges, their burgeoning passion for what they do, and valuable advice for others looking at manufacturing as a career.

Something I found very interesting was IPS's long-term partnerships with the local high schools, trade schools, and colleges. Mike Brask, president of IPS, says this is not only good for the community, but part of his company's core values. He introduced me to several young employees, part of the next generation of manufacturing professionals poised to make their mark.

Meet Caleb Aagard

Caleb Aagard is a local high school student and the youngest intern at IPS. Caleb found this opportunity at IPS through his high school's

IPS intern Caleb Aargard with his mentor Ben Herring, and IPS president Mike Brask.

CTE intern program and has taken full advantage of the experience, while sharing some concerns and advice for other high school students.

At just 16 years old, Caleb already has goals and a clear direction for his career. He was so well-spoken about his passion for learning about manufacturing, and his journey through it. Mike recognizes this and has given Caleb responsibility for important tasks. Caleb also has a direct mentor, but he quickly emphasizes that the entire IPS team has been helping him learn, including an allowance for mistakes. You'll see how, with the guidance of his mentors, he is learning from those mistakes.

His mentor, Ben Herring, told me that the mentorship program at IPS is not just important for high school students to gain real-world experience, but for everyone who works at IPS. Ben talks about the challenges of training and strategies to be successful.

It was easy to see how a career in manufacturing appeals to Caleb, especially when he started talking about the manufacturing tools he uses. He gave me a quick overview of a very cool project he's working on, and it was so encouraging to see his passion. I know you'll sense that right away. This is one video definitely worth watching.

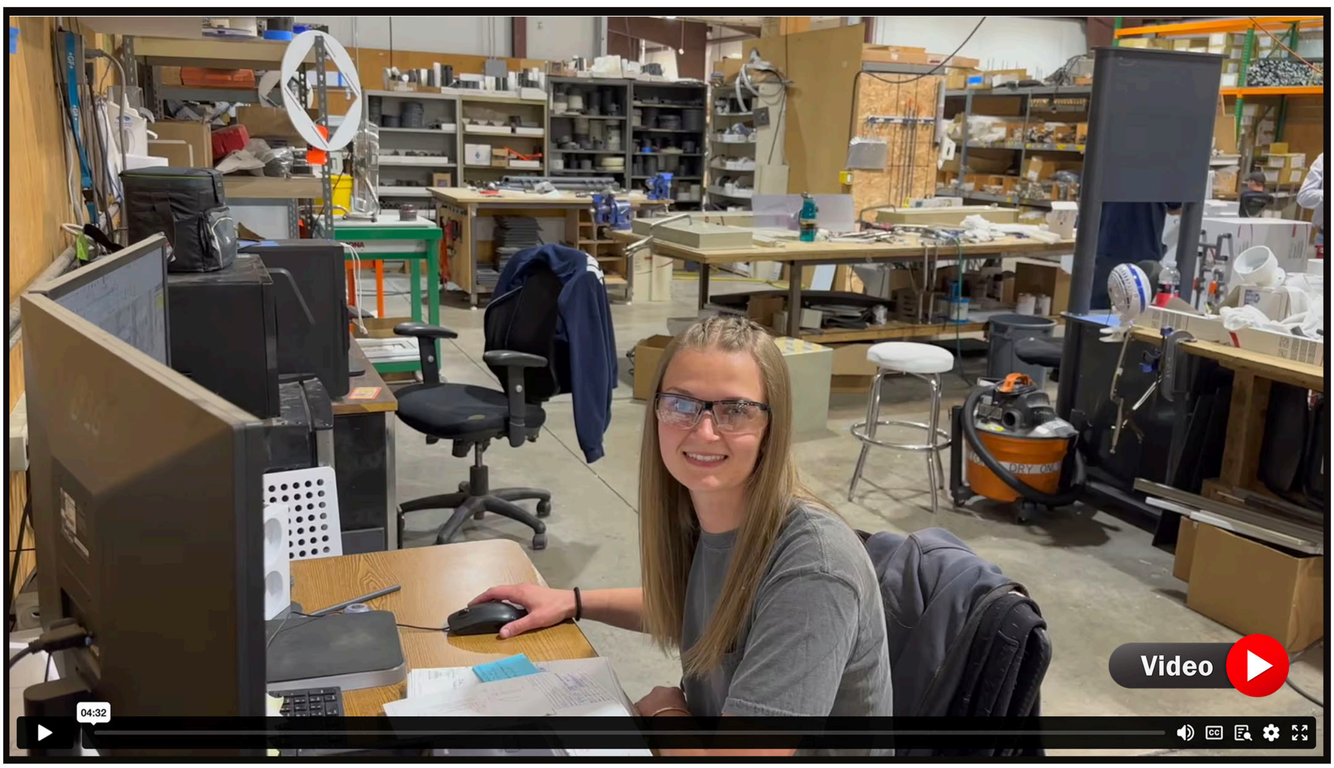
Related Video: Meet Caleb Aargard and Ben Herring (runtime 08:05)

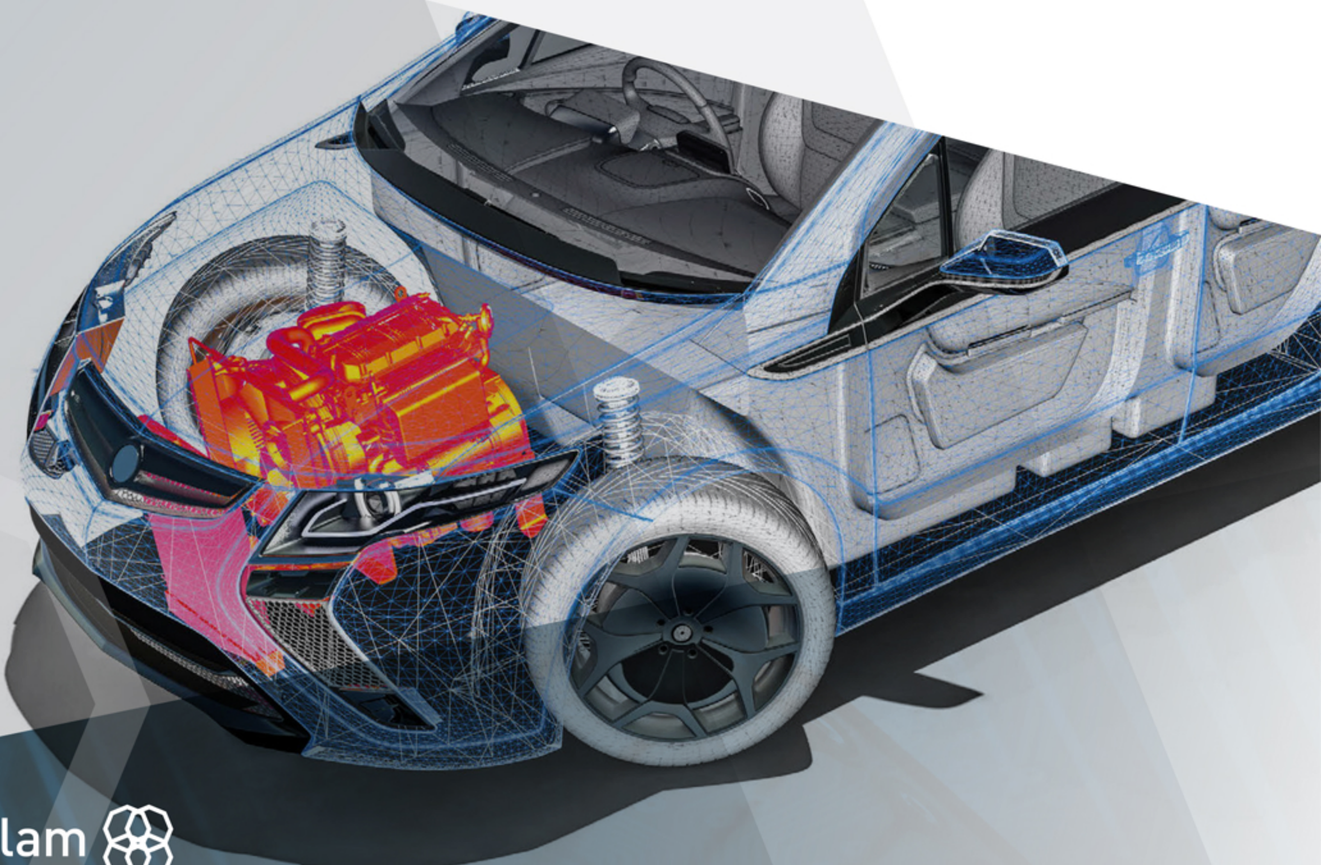
Meet Emory Ward and Alvaro Serna

Speaking of mentorships, I also spoke with Emory Ward and her mentor, Alvaro Serna. Alvaro has been with IPS for nearly 26 years while Emory, who recently joined as a design engineer, has just completed her first design for a piece of equipment. Emory graduated from Southern Utah University with a bachelor's degree in mechanical engineering and a minor in math.

Emory, who is part of this next generation, was very open about why she chose manufac-

Related Video: Meet Emory Ward and Alvaro Serna (runtime 04:32)





turing as a career, and what she loves about it. Mike Brask is a big believer in fostering growth and empowerment of the IPS team, and Emory shares a perfect example of this when she talks about—of all things—the location of her desk. She also talks about why having a mentor has been crucial to her success this early in her career.

Alvaro, her mentor, handles the special and custom projects at IPS. He's also trained and mentored many of his co-workers during his decades of service. Mike "accuses" Alvaro of being a bit too humble about his talent: "Alvaro is instrumental in the engineering and fabrication of custom products and has extreme talent in being able to see assemblies come together. He is a very strong mentor."

"Mike "accuses" Alvaro of being a bit too humble about his talent."

I was so impressed with Emory's thoughts about why she chose manufacturing as a career, what she learned in school—and the ways it didn't prepare her for the real-world environment. After 26 years, Alvaro is so attuned to not only what his company needs, but the ways he can contribute to both manufacturing and mentorship.

In our extensive conversations with Gen Zers, it is quite clear they consider having a mentor very important to their success, especially when onboarding with a new company. It's critical, they say, to have that connection with a mentor who provides both appreciation and constructive criticism.

To attract new talent to your company, your recruitment messages must promote this commitment to training and mentoring. During your interviews, share with the potential candidate a culture of training and support they can expect to be part of. Let them know about your "Alvaro." **PCB007**

What We Learned at PCB West

American Made Advocacy

by Travis Kelly, PCBAA

For the first time since we launched in 2021, the Printed Circuit Board Association of America participated in the PCB West trade show in Santa Clara, California. We were proud to join thousands of our colleagues to discuss the innovations driving our business and the very real challenges facing the American microelectronics industry.

After three days in the valley, speaking with our existing members, and building important new relationships, several things became clear:

1. Our industry knows we must act.

For too long, foreign subsidies have led to artificially low prices on PCBs produced abroad. Domestic manufacturing of PCBs and IC substrates has contracted to an alarming extent. Today we have fewer than 150 companies manufacturing PCBs in America, down from more than 2,000 at the turn of the century. The remaining companies were represented at PCB West. They survived a massive shift and they want to see this destructive trend reversed.

2. The PCB Industry is a place of innovation.

Every booth at PCB West told a story. From new chemical processes and advances in automation to workforce training programs and applications for the most challenging environments, everything on display pointed to a technology sector that is designing and building for the future, not living in the past.

3. The CHIPS Act is a catalyst for change.

Last year, Congress passed the landmark bipartisan CHIPS and Science Act and laid the groundwork for America to reclaim ground lost in the semiconductor industry over the past 20 years. While a tremendous start, the CHIPS program does not address everything chips need to function. Without investing in the whole ecosystem, we remain dangerously dependent on other nations—some less friendly than others—for the advanced packaging, substrates, and printed circuit boards that every chip needs to function. The lesson of the CHIPS Act must be that our industry can't wait for a lifeline or lawmaker benevolence. We must raise our voices and advocate for the legislation and public policy solutions we need.

4. Our "Educate, Advocate, and Legislate" mission has a long way to go.

America's continued reliance on overseas manufacturing for PCBs has arisen because we failed to appreciate the economic and national security reasons for maintaining a domestic manufacturing base. The recent pandemic made clear the risks of foreign dependence, but not as obvious was how deep the problem runs. Our team will keep meeting with lawmakers until everyone in government understands that "chips don't float."

As we head into a critical election year, there are reasons to be optimistic. We launched our coalition with only five founding members. This month we approach having 40 companies and individuals representing all aspects of our industry. Our public education campaigns and advocacy efforts in the halls of Congress continue to pay off. For example, the FY2024 National Defense Authorization Act contains language that enhances America's national security by promoting the domestic production of critical microelectronics.

I am proud of all that we have accomplished, while acknowledging that much remains to be done. As I often tell policymakers in Washington: Because offshoring and consolidation took nearly three decades to hollow out America's PCB industry, we will not restore it to full capacity overnight. We measure our progress with every lawmaker we educate, every new co-sponsor, every journalist we speak with, and every new company that joins our team. Together we will rebuild American microelectronics manufacturing.

The PCBAA fights every day for a level playing field on which U.S. PCB and substrate manufacturers can compete and win. If you're interested in joining our effort, please visit us online or contact me directly. PCB007

Travis Kelly is CEO is Isola-Group and current chairman of the Printed Circuit Board Association of America. To read past columns, click here.

AI Revolution Reshaping the Future of PCB Manufacturing

Article by Val Kaplan
CIMS

In the rapidly evolving landscape of PCB manufacturing, the integration of machine learning (ML) in the context of smart factories is ushering in a new era of efficiency, adaptability, and competitiveness within the industry. This article explores how ML is revolutionizing various facets of smart factories, including predictive maintenance, quality control, production, and inventory optimization. By harnessing the power of data-driven insights and automation, ML is enabling factories to proactively address issues, optimize processes, and enhance product quality. This transformative technology not only minimizes downtime but also paves the way for agile, cost-effective, and sustainable manufacturing operations. The article sheds light on real-world applications, challenges, and the promising future of ML in the Smart factory ecosystem.

What are Smart Factories?

A Smart factory is typically defined as an advanced manufacturing facility that utilizes cutting-edge technologies, automation, and data-driven processes to optimize its operations, improve efficiency, enhance productivity, and enable more agile and flexible production.

In a Smart factory, various components, including machines, equipment, and systems,

Val Kaplan

are interconnected and communicate with each other, as well as with a central control system, through connectivity solutions such as the Internet of Things (IoT) and other methods. This interconnectedness allows for real-time data collection, analysis, and decision-making, leading to proactive maintenance, rapid adaptation to changing demands, and improved product quality.

Most advanced Smart factories are characterized by their ability to leverage artificial intelligence, machine learning, robotics, and other advanced technologies to streamline operations and drive innovation in manufacturing processes.

The integration of machine learning (ML) and artificial intelligence (AI) in Smart factories has the potential to transform traditional manufacturing processes into agile, data-driven, and highly efficient operations. These technologies enable factories to respond to changing demands, reduce costs, enhance product quality, and improve overall competitiveness in the global market.

In recent years, PCB factories have been gradually adapting various Smart factory concepts although this process has been relatively slow and limited to the most advanced and forward-looking companies. Also, Smart factories in the PCB industry tend to mainly focus on the equipment interconnectivity and process data gathering and consolidation while data analysis is still mainly done by human expert operators rather than AI.

On the other hand, the amount and dimensionality of data that can potentially be gathered from multiple PCB processes is simply too extensive for human brains to process. This is where machine learning and artificial intelligence hold the most promise and could outperform even the best human experts.

The following are the primary areas in which ML and AI technologies can provide the most significant benefits for Smart PCB factories.

Machine Learning for Predictive Maintenance

Once the information from multiple processes is continuously fed into a central data repository, AI models can analyze data from different sensors and equipment to predict when machinery might fail. For example, such models can be trained to detect inconsistencies or gradual subtle changes from sensor readings, as well as juxtapose these data with the data from downstream processes, to find patterns.

Such multi-dimensional analyses involving huge numbers of interrelated variables is a natural fit for AI technology while being an impossible task for humans, even with the help of the most advanced analytical (non-AI) software. Such tasks are particularly suitable for unsupervised AI, specifically through techniques like clustering and dimensionality reduction. These AI models are used for finding patterns in data without explicit labels or guidance.

In short, AI technology has the potential to deliver more effective proactive maintenance, reducing downtime and saving costs.

Machine Learning for Troubleshooting of Equipment and Processes

Machine learning can be a valuable tool for troubleshooting and data-driven problem identification in PCB manufacturing. For example, ML algorithms, such as clustering or statistical models, are used to identify anomalies or outliers in the data. These anomalies can signal potential issues.

ML techniques can also be applied to determine the root causes of anomalies or problems. This may involve correlation analysis, causal inference, or rule-based systems. Such systems would identify which factors or variables are most strongly associated with the detected anomalies, and direct maintenance staff to the right equipment and even suggest a solution.

"ML techniques can also be applied to determine the root causes of anomalies or problems."

In order to enable such tracing capabilities, each panel must carry a unique identifier, typically a 2D code (data matrix code, or DMC, is the most common type). This code will then be scanned at every production stage enabling the system to trace it back to every process equipment it went through during production process.

Machine learning and AI technology is also being gradually adopted within AOI and AFI/AVI processes. The ultimate goal is to reduce human operators' involvement in defects verification to the minimum, if not eliminate it altogether, thereby streamlining the AOI process by making it faster and less prone to human errors.

In this application, the AI image recognition model is trained on an extensive set of real defects aiming at teaching it how to tell them apart from false calls such as non-critical contamination or acceptable process variations. In general, deep learning neural network AI is the most suitable technology for image recognition in general and optical inspection in particular.

Despite the promising potential, implementing ML for the AOI verification cycle is not without significant technological and conceptual challenges. The seemingly simple question, "What is a defect?", may not have the same answers between market segments, among different PCB manufacturers, or even individual jobs within the same company.

Furthermore, defect specifications frequently change and evolve over time as a result of new processes, changing end user requirements, and PCB makers' other various considerations. This means that one AI model may not be sufficient to cover all scenarios, and such models must be retrained on a regular basis with new data that better reflects the current situation. For example, an AI system would be unable to accurately identify defects that it had never been trained on or effectively filter out a previously unknown type of false calls.

Another challenge is determining the best tradeoff between two competing metrics: the filter rate (the number of true false calls removed by AI) and escapes, also known as underkill or false negatives (real defects that were incorrectly classified and filtered out as false calls by the AI model). The higher the filter rate, the more defects will escape, and vice versa. The correct approach to resolving this problem is to take a purely economic approach, weighing the benefit of having less verification equipment and reduced manpower against the cost of lower yield due to higher escape rate.

Finally, there is a type of agency dilemma where there is a conflict of interest between the PCB manufacturer's motivation to cut costs and the end user's interest for high quality. It should come as no surprise that the end user may reject their suppliers' cost-cutting efforts through the use of new technology if they have the potential to negatively impact product quality, which is their top priority. This is why they may be reluctant to accept any ML implementation by their suppliers that might jeopardize that goal.

Harnessing Machine Learning and AI for Smart Factories

PCB manufacturers, on the other hand, are in the business of making money, and cost savings with such technology directly translate to higher profit margins, though they may face resistance from their customers, particularly those with significant bargaining power.

Finally, there are practical and technical challenges associated with AI development and implementation, the most significant of which is the collection and processing of massive amounts of high-quality pre-classified (labeled) data required to develop a viable model. The data provided by PCB manufacturers can be messy, particularly in terms of classification accuracy. As a result, AI developers would have to expend significant effort to clean and reclassify the data in order to make it suitable for AI model training purposes. This is a time-consuming process that frequently results in the discarding of some amounts of low-quality data.

In conclusion, the success or failure of AI technology adoption for the AOI process is primarily determined by the optimal balance between the rate of escapes and the overall filter rate, resolving issues around accurate defect specs while periodically retraining AI models with new and relevant data, and navigating the agency dilemma between PCB manufacturers and their customers.

Machine Learning for Optimizing Production Processes

ML can significantly enhance the optimization of PCB production processes, including production planning and scheduling, as well as inventory management. For example, ML models can analyze historical sales data, market trends, and other relevant factors to forecast future demand more accurately. This information helps in setting production targets.

ML algorithms can optimize resource allocation by considering factors like machine availability, labor capacity, and job specifics (such as complexity and required process flow, as well as raw material availability) to create efficient production process routes and schedules.

For example, the AI models that we have developed allow users to identify the optimal route through production processes down to a specific equipment, which maximizes efficiency and yield while minimizing production time and the costs. Such a model learns from the vast database of historical data on the performance of each individual equipment with different types of jobs and their effect on the overall defect types and rates.

One of the biggest practical challenges in Smart factory adoption and implementation is the difficulty in collecting reliable and con-

sistent data from processes and machinery. There are two main reasons for it:

1. Lack of connectivity of some equipment, particularly the older machinery that makes data collection difficult if not impossible,
2. Lack of common data exchange protocols and formats within the PCB industry.

The lack of connectivity can sometimes be addressed by installing dedicated sensors that transmit the relevant data wirelessly. In other cases, a special software can be used to grab information directly from the user interface and send it to the central database by Ethernet or Bluetooth.

The lack of common formats and protocols basically means that data from different processes must first be converted into some type of common format in order to be useable and then “stitched” together along relevant features and data dimensions.

Also, some equipment manufacturers deliberately prevent external users from accessing and sharing their data, attempting to lock-in their customers within the ecosystem of that company’s products and solutions. In such cases, the connection can be refused, or data can be encrypted to prevent third parties from reading it.

We chose the opposite approach, ensuring safe and secure access to all data generated by our systems as well as data kept in our databases, using SDK (software development kit). This enables other users, such as third-party developers or customers’ IT engineers, to take full advantage of the wide range of actionable information available from AOI or AVI/AFI systems.

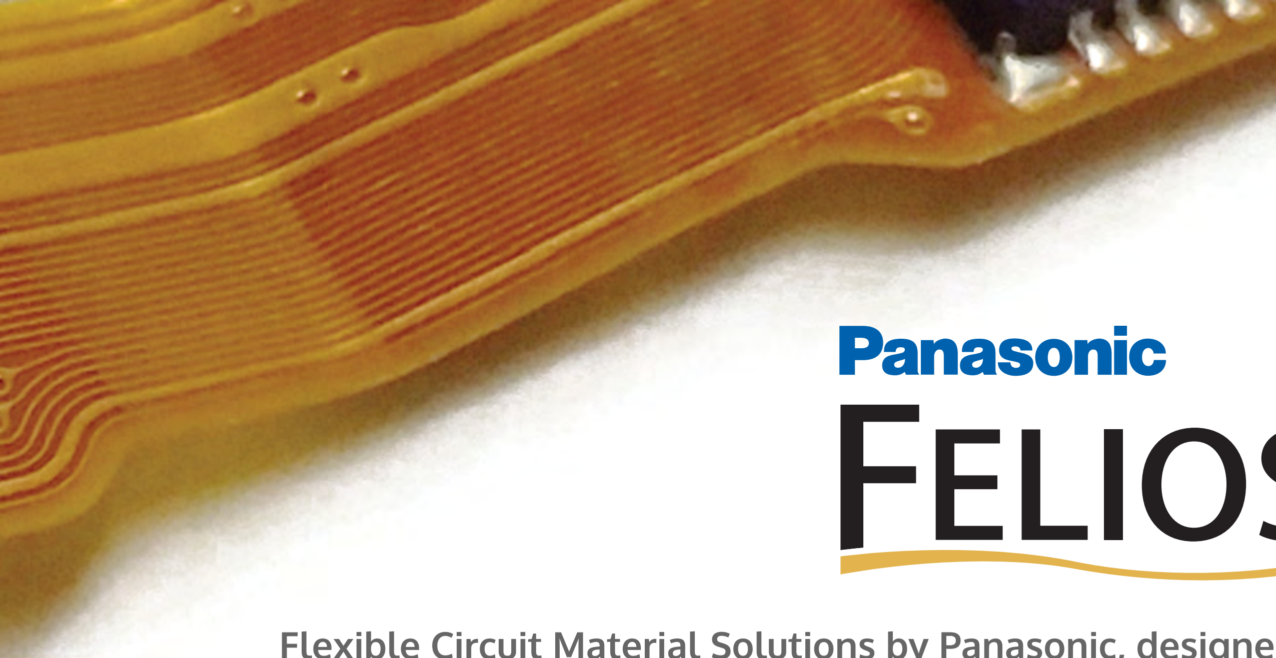
The Future of ML and AI Technologies in PCB Manufacturing

There is little doubt that, for better or worse, AI will transform almost every aspect of our lives; this process has already begun. Only a few years ago, technologies like ChatGPT were considered science fiction, yet they are now available to everyone. Every industry, including ours, has the potential to be transformed by ML and AI technologies.

What can we expect in the future once the above-mentioned challenges are resolved? One possibility is that AI systems may take over more of the day-to-day decision-making on how to manage PCB manufacturing operations, even down to optimizing each individual piece of equipment. When combined with material handling automation, such a system has the potential to automate PCB manufacturing to unprecedented levels.

Intelligent manufacturing solutions based on AI will be able to resolve quality issues on their own by automatically tweaking various parts of the process, reroute production flow to maximize yield, and optimize equipment and material usage, relying on human operators only in a few exceptional cases. Once AI can control and optimize all elements of PCB fabrication, from quoting and CAD to manufacturing and quality inspection, Smart factories will truly become intelligent, ushering in a new era of innovation and technological progress. PCB007

Val Kaplan serves as the vice president of marketing at CIMS, a globally renowned industry leader in the development of smart factory solutions including automated optical inspection, automation, advanced software, and cutting-edge AI solutions.



TOP TEN

EDITOR'S PICKS

MKS' Atotech: More Horizontal Panel Plating in the U.S.

Recently, Schweitzer Engineering Laboratories (SEL) opened its new $100 million captive printed circuit board manufacturing facility in Moscow, Idaho. The new facility features state-of-the-art PCB equipment from MKS' Atotech, one of the leading providers of advanced PCB and IC substrate horizontal manufacturing equipment for the electronics industry.

ICAPE Group Announces Strong Improvement in 1H 2023 Results

ICAPE Group, a global technology distributor of printed circuit boards (PCB), announced its half-year results for the first half of 2023, ended on June 30, 2023, and approved by the Board of Directors on September 26, 2023.

Trouble in Your Tank: Processes to Support IC Substrates and Advanced Packaging, Part 4

In a previous column, the critical process of desmear and its necessity to ensure a clean copper surface connection was presented. Now, my discussion will focus on obtaining a void-free and tightly adherent copper plating deposit on these surfaces.

Culture by Design: Building a Winning Team from the Start

"If you build it, they will come." This line is from one of my favorite movies, "Field of Dreams." It's a fictional story about a character named Ray Kinsella, an Iowa farmer, who mysteriously receives instruction to build a baseball diamond in his cornfield.

Chemcut: Blending Capabilities and Culture

Mike Soble, who works in technical sales for Chemcut, explains that despite the long distance between Idaho and Pennsylvania, working with Schweitzer Engineering Laboratories on its new facilities was truly a perfect match.

Technica USA to Distribute Symtek Automation Asia Co., Ltd's Automation Systems and Technology

Technica USA announced it has reached a Master Distribution Agreement with SAA to promote and support automation systems & technology offered by SAA.

APCT Appoints Peter Austin as President and CEO, Effective September 1


Peter Austin

APCT, a leading name in the PCB and electronics industry, is delighted to announce the appointment of Peter Austin as its President and CEO, effective September 1. Peter takes the reins from former CEO Steve Robinson, who has retired after years of dedicated service to the company.

Sigma Mecer: Turning Copper 'Green'

Andreas Littorin is CEO at Sigma Engineering AB. The company has been manufacturing equipment for the PCB industry for 45 years, mainly a mature product for copper recycling of alkaline and acidic etchants in the PCB manufacturing process.

North American PCB Industry Sales Down 26% in August

IPC announced the August 2023 findings from its North American Printed Circuit Board (PCB) Statistical Program. The book-to-bill ratio stands at 1.00. Total North American PCB shipments in August 2023 were up 26.4% compared to the same month last year. Compared to the preceding month, August shipments were down 35.7%.

NCAB Group Earns Gold EcoVadis Medal for Sustainability Performance

NCAB scored overall 73 out of 100 which places us amongst the top 5 percent of the 85,000 companies globally that have been rated by EcoVadis, and top 3 percent in the Manufacture of electronic components and boards industry.

Career Opportunities

GEN3™
Precision as Standard

Technical Sales Manager

Gen3, based in Farnborough, UK, who designs, manufactures and distributes test equipment to minimize risk of failure in the field, has an exciting opportunity for a Technical Sales Manager to join its team to drive growth in the southern half of the UK.

Responsibilities & Experience

- Promote Gen3's and its principles' equipment.
- Identify opportunities in existing and new customers.
- Report all commercial developments related to the activity of Gen3's customers, actively seeking the specification of Gen3's products, into new projects.
- Be fully familiar with all Gen3's products, technology, USPs, features, benefits and international standards.
- Follow up all enquiries for products and services; convert them into contracts/orders.
- Provide technical support – remotely and onsite.
- Be widely recognised and acknowledged as an "Industry Expert."
- Technical Sales and Account Management skills from an electronics background is desirable.
- Excellent sales, customer service, communication, presentation and negotiation skills.
- Recognised qualification in Electronics Engineering or related field.
- Knowledge of the electronics/SMT assembly process.
- Excellent written and verbal communication skills in English.
- Competent user of Microsoft Office applications.
- Ideally living in the Southern half of the UK.
- Willing and able to travel within and outside UK.
- A full, clean UK driving license is essential.

To apply, please contact John Barraclough at john.barraclough@gen3systems.com or by using the link below.

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Senior Sales Representative

Ventec Central Europe

Location: Kirchheimbolanden, Germany/Remote

We are looking for a self-motivated Senior Sales Representative—Ventec Central Europe, ideally with experience in the PCB industry. This position requires significant selling experience (15+ years) in the electronics and PCB industries. Candidates must possess a proven & consistent history of proactive sales growth with OEM customers. Most notably, they must be able to connect with OEM contacts that have decision-making capabilities.

Key Responsibilities

- Promote, sell, and close business for all Ventec product lines with focus on key OEM and PCB manufacturing customers.
- Track projects and submit monthly updates to management.
- Coordinate cross-functional resources when applicable.
- Assist in coordination and set-up of relevant trade show events.
- Assist in strategic planning initiatives.
- Assist in market and customer intelligence gathering.
- Recommend pricing strategies.

Job Requirements

- Entrepreneurial spirit, positive, high energy, and desire to win.
- Proactive and self-motivated work strategy to develop and win business for all business units.
- Excellent written and oral communication skills in German and English
- Excellent computer skills (Microsoft Office, especially Excel).
- Proven track record securing new business at OEM accounts.

Please apply in the strictest confidence, enclosing your CV, to: accountingde@ventec-europe.com

apply now

Career Opportunities

Technical Support Engineer

USA Region

ViTrox aims to be the world's most trusted technology company in providing innovative, advanced, and cost-effective automated Machine Vision Inspection Solutions for the semiconductor and electronics packaging industries. Located in Hayward, California, ViTrox Americas Inc. is actively looking for talent to join our expanding team.

Key Responsibilities:

- Delivering excellent and creative problem-solving skills for servicing, maintaining, machine buy-off, and troubleshooting advanced vision inspection machines at customer sites. Providing remote customer support to minimize machine downtime.
- Cultivating strong customer relationships and ensuring comprehensive customer service to drive repeat orders and support business development in machine evaluation.
- Proactively understanding customer needs and feedback to drive continuous improvement in existing technologies and new product development.

Qualifications & Requirements:

- A recognized diploma/advanced diploma/ degree in Science and Engineering, preferably in Electrical & Electronics/Computer Science/ Computer Studies or equivalent.
- 3+ years of relevant experience in servicing automated inspection equipment (SPI, AOI, and AXI).
- Strong communication and troubleshooting skills.
- Willingness to travel extensively across the USA.
- Positive attitude and flexibility to accommodate conference calls with headquarters.
- Applicants from the USA and Canada are welcome to apply.
- Training will be provided at our headquarters in Penang, Malaysia.

apply now

The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

Career Opportunities

TAIYO AMERICA

Europe Technical Sales Engineer

Taiyo is the world leader in solder mask products and inkjet technology, offering specialty dielectric inks and via filling inks for use with microvia and build-up technologies, as well as thermal-cure and UV-cure solder masks and inkjet and packaging inks.

PRIMARY FUNCTION:

1. To promote, demonstrate, sell, and service Taiyo's products
2. Assist colleagues with quotes for new customers from a technical perspective
3. Serve as primary technical point of contact to customers providing both pre- and post-sales advice
4. Interact regularly with other Taiyo team members, such as: Product design, development, production, purchasing, quality, and senior company managers from Taiyo group of companies

ESSENTIAL DUTIES:

1. Maintain existing business and pursue new business to meet the sales goals
2. Build strong relationships with existing and new customers
3. Troubleshoot customer problems
4. Provide consultative sales solutions to customers technical issues
5. Write monthly reports
6. Conduct technical audits
7. Conduct product evaluations

QUALIFICATIONS / SKILLS:

1. College degree preferred, with solid knowledge of chemistry
2. Five years' technical sales experience, preferably in the PCB industry
3. Computer knowledge
4. Sales skills
5. Good interpersonal relationship skills
6. Bilingual (German/English) preferred

To apply, email: BobW@Taiyo-america.com with a subject line of "Application for Technical Sales Engineer".

apply now

BLACKFOX®
Premier Training & Certification

IPC Instructor
Longmont, CO

This position is responsible for delivering effective electronics manufacturing training, including IPC certification, to adult 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 primarily conduct training at our public training center in Longmont, Colo., or will travel directly to the customer's facility. It is highly preferred that the candidate be willing to travel 25–50% of the time. Several IPC certification courses can be taught remotely and require no travel or in-person training.

Required: A minimum of 5 years' experience in electronics manufacturing and familiarity with IPC standards. Candidate with current IPC CIS or CIT Trainer Specialist certifications are highly preferred.

Salary: Starting at $30 per hour depending on experience

Benefits:

- 401k and 401k matching
- Dental and Vision Insurance
- Employee Assistance Program
- Flexible Spending Account
- Health Insurance
- Health Savings Account
- Life Insurance
- Paid Time Off

Schedule: Monday thru Friday, 8–5

Experience: Electronics Manufacturing: 5+ years (Required)

License/Certification: IPC Certification–Preferred, Not Required

Willingness to travel: 25% (Required)

apply now

Career Opportunities

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the Southeastern U.S. territory. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:

- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer focused team

Interested? Please contact Russ Adams at (206) 351-0281 or russa@prototron.com.

apply now

KYZEN
Where Science and Care Converge.

Regional Manager

Southwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the Southwest Region. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deploy-ment and management of sales reps. Identifies objectives, strategies and action plans to improve short- and long-term sales and earnings for all product lines.

DETAILS OF FUNCTION:

- Develops and maintains strategic partner relationships
- Manages and develops sales reps:
  - Reviews progress of sales performance
  - Provides quarterly results assessments of sales reps' performance
  - Works with sales reps to identify and contact decision-makers
  - Setting growth targets for sales reps
  - Educates sales reps by conducting programs/ seminars in the needed areas of knowledge
- Collects customer feedback and market research (products and competitors)
- Coordinates with other company departments to provide superior customer service

QUALIFICATIONS:

- 5-7+ years of related experience in the manufacturing sector or equivalent combination of formal education and experience
- Excellent oral and written communication skills
- Business-to-business sales experience a plus
- Good working knowledge of Microsoft Office Suite and common smart phone apps
- Valid driver's license
- 75-80% regional travel required

To apply, please submit a COVER LETTER and RESUME to: Fernando Rueda, Americas Manager

fernando_rueda@kyzen.com

apply now

Career Opportunities

EMA Design Automation®

Technical Marketing Engineer

EMA Design Automation, a leader in product development solutions, is in search of a detail-oriented individual who can apply their knowledge of electrical design and CAD software to assist marketing in the creation of videos, training materials, blog posts, and more. This Technical Marketing Engineer role is ideal for analytical problem-solvers who enjoy educating and teaching others.

Requirements:

- Bachelor's degree in electrical engineering or related field with a basic understanding of engineering theories and terminology required
- Basic knowledge of schematic design, PCB design, and simulation with experience in OrCAD or Allegro preferred
- Candidates must possess excellent writing skills with an understanding of sentence structure and grammar
- Basic knowledge of video editing and experience using Camtasia or Adobe Premiere Pro is preferred but not required
- Must be able to collaborate well with others and have excellent written and verbal communication skills for this remote position

EMA Design Automation is a small, family-owned company that fosters a flexible, collaborative environment and promotes professional growth.

Send Resumes to: resumes@ema-eda.com

apply now

pluritec®

MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and X-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport for travel.

Must be able to travel extensively.

apply now

Career Opportunities

KOH YOUNG AMERICA

Technical Service & Applications Engineer

Full-Time — Flexible Location

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement-based inspection technology for electronics manufacturing. Located in Duluth, GA, Koh Young America has been serving its partners since 2010 and is expanding the team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience preferred
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, and specifications

Benefits

- Health/Dental/Vision/Life Insurance with no employee premium (including dependent coverage)
- 401K retirement plan
- Generous PTO and paid holidays

apply now

ARLON ELECTRONIC MATERIALS

Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- Engineering
- Quality
- Various Manufacturing

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

Arlon is a major manufacturer of specialty high-performance laminate and prepreg materials for use in a wide variety of printed circuit board applications. Arlon specializes in thermoset resin technology, including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems. These resin systems are available on a variety of substrates, including woven glass and non-woven aramid. Typical applications for these materials include advanced commercial and military electronics such as avionics, semiconductor testing, heat sink bonding, High Density Interconnect (HDI) and microvia PCBs (i.e., in mobile communication products).

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity, allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information, please visit our website at www.arlonemd.com

apply now

Career Opportunities

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 to build better boards faster. Our nationwide network provides many opportunities for a rewarding career within our company.

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

apply now

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

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

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

The CAD/CAM Engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creation of 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 Orbotech/Genflex CAM tooling software.

PHYSICAL DEMANDS

Ability to communicate orally with management and other co-workers is crucial. Regular use of the phone 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

Career Opportunities

APCT
Passion | Commitment | Trust

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

YOUR JOB AD HERE

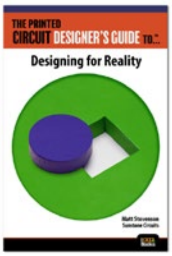
For information, please contact:
BARB HOCKADAY
barb@iconnect007.com
+1 916.365.1727 (PACIFIC)

I-Connect007
GOOD FOR THE INDUSTRY

The Printed Circuit Designers Guide to... Manufacturing Driven Design

This book introduces a new process workflow for optimizing your design called Manufacturing Driven Design (MDD). This is a distinct evolution from DFM. Readers will learn how to utilize data-driven concepts to improve design capabilities. Visit I-007ebooks.com to get your copy today.

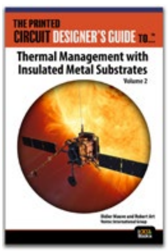
I-007eBooks The Printed Circuit Designer's Guide to...



Designing for Reality

by Matt Stevenson, Sunstone Circuits

Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process. Read it now!



Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples and focusing on specific solutions and enhanced properties of IMS. Add this essential book to your library.

High Performance Materials

by Michael Gay, Isola

This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. Get your copy now!

Stackups: The Design within the Design

by Bill Hargin, Z-zero

Finally, a book about stackups! From material selection and understanding laminate datasheets, to impedance planning, glass weave skew and rigid-flex materials, topic expert Bill Hargin has written a unique book on PCB stackups. Get your copy today!

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

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

MANAGING EDITOR: **ANDY SHAUGHNESSY**
(770) 315-9901; andy@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) 365-1727; barb@iconnect007.com

BUSINESS DEVELOPMENT: **TAMARA SITES**
tamara@iconnect007.com

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

ART DIRECTOR/PRODUCTION MGR.: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

CREATIVE TECHNOLOGIST: **BRYSON MATTIES**

COVER DESIGN: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © DOTTEDYETI**

PCB007 MAGAZINE

PCB007 MAGAZINE®
is published by IPC Publishing Group, Inc.
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

October 2023, Volume 13, Number 10
PCB007 MAGAZINE is published monthly
by IPC Publishing Group., Inc., dba I-Connect007

ADVERTISER INDEX


Problems solved!
PCB007 MAGAZINE
AUGUST 2023
Developing High-flying Strategies
PCB007 MAGAZINE
SEPTEMBER 2023
FINDING ULTRA
SEL SPECIAL SECTION
DESIGN007 MAGAZINE
AUGUST 2023
DESIGN007 MAGAZINE
SEPTEMBER 2023
RIGID-FLEX DESIGN
DESIGN007 MAGAZINE
OCTOBER 2023
SMT007 MAGAZINE
SEPTEMBER 2023
PAGE 68: PRESHOW SECTION
SMT007 MAGAZINE
OCTOBER 2023
SUBSCRIBE NOW