JANUARY 2024

I-Connect007
GOOD FOR THE INDUSTRY

PCB007 MAGAZINE

Getting to Know Your Designer

PCB007 MAGAZINE

Getting to Know Your Designer

In this issue, we examine how fabs work with their design customers, educating them on the most critical elements of fabrication needed to be successful, as well as the many tradeoffs involved.

PCB007 MAGAZINE

COLUMNS

HIGHLIGHTS

DEPARTMENTS

SHORTS

Let Your Walls Down

Nolan's Notes

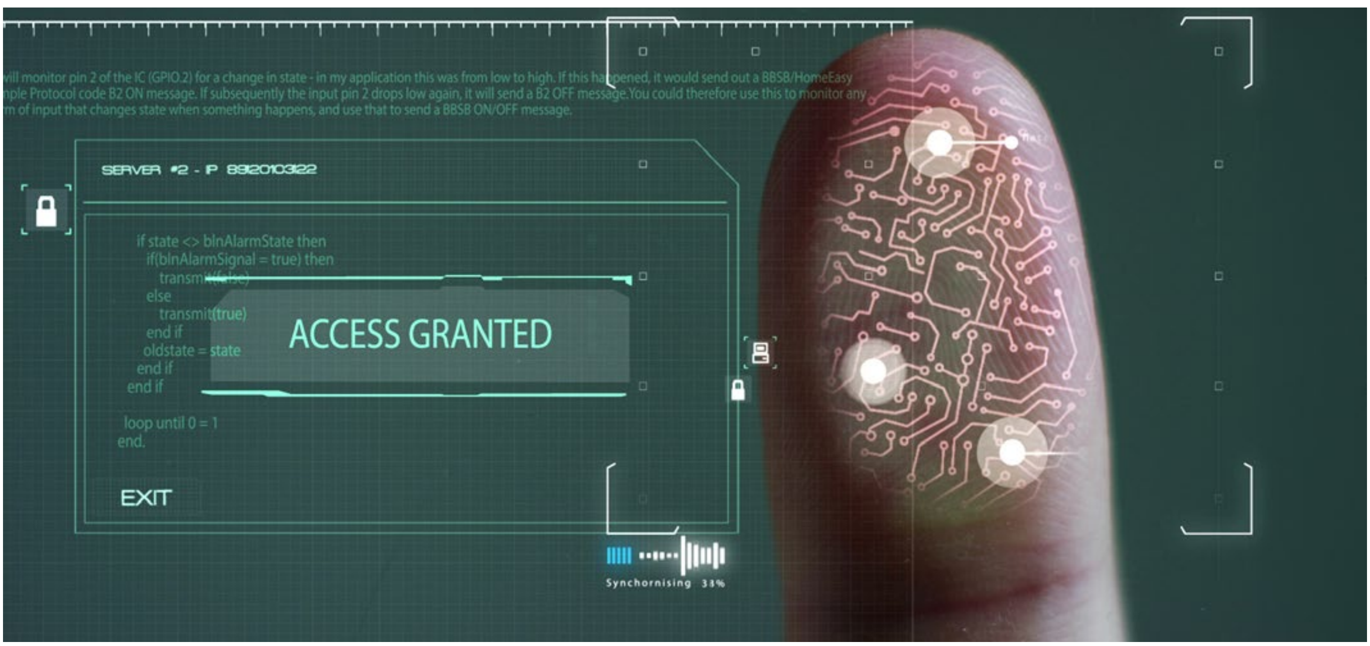
by Nolan Johnson, I-CONNECT007

I'm careful about not complaining too much in my life and in my professional writing. Complaining usually gets you nowhere. Back in college, I heard a quote that has stuck with me: "If you're going to complain about a problem, have a solution to offer along with your complaint. That's the way to move things forward." So, pardon me while I lodge a complaint and suggest a change in behavior about PCB fabricators and designers.

As long as I've been in this industry, fabricators have been downright territorial about their processes. It doesn't matter whether it's capabilities, limits, CAM processes, or even the chemicals used in the plating and etching processes, fabs don't divulge that information. Meanwhile, they want OEMs and designers to improve the manufacturability of their designs by working closely with the fabricators, and PCB customers should better understand what goes on inside the fab.

While it seems those two goals contradict each other, it's not necessarily true. There are plenty of examples of innovators giving their disruptive technology away:

- Volvo developed the seat belt, then chose not to patent it.
- Bell Labs developed and gave away TCP/IP. That's the data packet technology that makes the internet work.
- Speaking of the internet, Tim Berners-Lee gave away HTTP, allowing browsers to proliferate in our lives.



- In fact, this giving model became a philosophy in and of itself, giving birth to open-source software. Today, most high-complexity business operations software and data centers run on open-source software.

In these cases, these innovations arguably made the world a better place. Open access to the protocols enabled entire industries to be created. Holding onto processes and process data likely hampers the ability of industry to grow cooperatively. The less black magic there is in designing for fabrication, the more likely folks are to try their hand at making a board for themselves. The more fabricators are forthcoming about their processes, the more interchangeable fabricators become. That's a good thing because now you're competing on your value-add.

In the late 1980s, I was a new software engineer writing code for a fresh young startup called Mentor Graphics. In a company meeting, COO Gerry Langeler was celebrating something like 12 consecutive quarters of record sales, and stated that the reason for Mentor's success was, "Once you have them by the database, their hearts, minds, and wallets will follow." Back then, Mentor Graphics had a "walled garden" approach to the data, something it maintained for quite some time.

The challenge with a walled garden strategy is that, unless you can do 100%+ of what the customer desires, there will inevitably be a "compelling event" that will justify giving up on your products completely to shift to another vendor with that key feature. That was the way of the world in Mentor's market back then. By embracing interchangeability and inter-platform communication, customers didn't have to dump everything just to add a piece they needed. By knocking down the walls of their garden, those EDA companies grew even faster.

If prospective customers knew more, they'd be better able to determine that some of their marginal fab work at "Vendor A" would do better with you. Sure, you might lose a part number or two to another fab, but chances are good your customer will only move the work you struggle with anyway, allowing you the capacity to take on more business that's in your most profitable sweet spot.

In this issue, we investigate how to build a good communication relationship with your customers. Dana Korf, an I-Connect007 columnist, discusses how designers and manufacturers should be synchronized. NCAB's field application engineers explore how to foster loyal customer/vendor relationships. Don Ball of Chemcut uses his column space to talk about the ins and outs of getting to know your vendor, and in an interesting interview, Sean Patterson of Summit Interconnect talks about creating a culture of operational efficiencies—something we can all learn from.

In addition, this month we bring you a 2023 IPC APEX EXPO technical paper on new resin systems for board fab issues (AGC), and columnists chiming in this month include Steve Williams, Paige Fiet, Happy Holden, Henry Crandall, and Mike Carano.

Now, I'm not saying that you should open everything to the world but give some thought to what knowledge really needs to be secret, and what more you can open up to help designers understand you better. Remember, designers have a history of using new knowledge in creative and, sometimes, revolutionary ways. Let's take a listen. **PCB007**

Nolan Johnson is managing editor of *SMT007 Magazine* and co-managing editor of *PCB007 Magazine*. Nolan brings 30 years of career experience focused almost entirely on electronics design and manufacturing. To read other columns or to contact Johnson, click here.

The Designer and Manufacturer Must Be in Sync

Feature Article by Dana Korf

"Why can't we all just get along?"
—Henry Liberman, TEDx

It's no industry secret that most PCB data packages sent to fabricators from designers cannot be built as-is. The finished boards often seem to work, despite a factory estimating what the designer wanted vs. what the documentation showed, then jointly rectifying issues through lengthy technical query (TQ) cycles. In general, everyone seems to be satisfied with this process, so why do we need to improve the designer/manufacturer relationship? Why is the best solution a strong designer/manufacturer relationship, and is it even possible?

Let's perform a traditional root cause analysis and define the function of a designer vs. a manufacturer. Oxford Language definitions:

- ***Designer:*** *A person who plans the form, look, or workings of something before its being made or built, typically by drawing it in detail.*
- ***Manufacturer:*** *A person or company that makes goods for sale.*

This seems pretty straightforward. The manufacturer builds the product based on the supplied documentation. Unfortunately, the manufacturer's front-end engineering team typically completes a portion of the design after the data package is received because it can't be built as submitted. The manufacturer must update the design to maximize yield, meet cost targets, ensure reliability, and meet all performance requirements.

Typical design functions performed by the manufacturer include creating a material


UNLEASHING THE TiTAN PSR-H
Standard Solder Mask Printing in 40 seconds*
*Print speed is based on one 18 x 24 panel side
The latest in direct imaging technology from
CBTech®
Solder Mask and Dry Film Capable
EXCLUSIVELY AVAILABLE IN NORTH AMERICA FROM
TECHNICA, U.S.A.
1-800-909-8697 • www.technica.com

stackup that meets the specified mechanical thicknesses/tolerances, material selection, line width/spacing modification to generate the specified impedance/tolerance, creating the assembly array/panel design/documentation, and ensuring compliance with regulatory requirements, such as UL.

In my biannual class, "IPC Design for PCB Manufacturability," there are two primary components to a DFM review. The first is to identify and resolve design and supplied documentation errors. The second component is to perform a process compatibility check. Finally, you must resolve these issues with the identified technical contact.

Who Is the Designer?

This can vary based on the critical design requirements. It can be the person who lays out the PCB, the signal integrity engineer, the PCB technologist, electrical or safety engineer, all the above, or some of the above. This person may change for different projects. That's okay. You just need to provide a single technical point of contact as the communication interface who reports to the rest of the team. Formal documented TQs with written customer approvals should be generated for each approved change. This is important because engineers move to other companies, retire, or move on to a different project. A new engineer assigned to the part number may not agree with the previously agreed modifications and wants the manufacturer to pay for a "bad" product. This is avoided by showing who approved the change and when through documented TQs.

Single Prototype Vs. Multiple Part Number Orders

When a manufacturer receives their first order from a new customer, there generally isn't much time to spend with the designer understanding their intent. The designer's name is not provided very often because an EMS NPI engineer or procurement person will be provided as the technical contact.

It is generally a short cycle, quick turn, prototype, or qualification order. The prototype tooling must be released quickly to the factory to give them the most time to produce

the board. Generating a lot of TQs to understand the design intent is mostly frowned upon because it increases the production tooling release time, leaving less time on the manufacturing floor to meet shipping commitments.

Initial qualification orders may also have short lead times. The manufacturer qualification team (if there is one) will hand-walk the design through the process and make real-time judgment calls when issues arise. There isn't much time to synchronize with the designer. This is especially true when the designer is one level removed from the PCB fabricator, as when the board is ordered from an EMS supplier.

Multiple part numbers should be ordered from the same supplier/plant to allow the designer and manufacturer to get in sync. The front-end engineer will see common documentation errors and solutions, along with common answers to TQs that are generated. This is the start of the designer knowledge transfer to manufacturing, resulting in reduced cycle times and improved PCB quality and performance.

When the customer continues to shop for different suppliers to get the best lead time, lowest cost, etc., that makes it nearly impossible to establish a bi-directional technical relationship between the designer and PCB fabricator.

DFM Software to the Rescue?

Millions of dollars have been spent on designer DFM and fabricator engineering/CAM software, yet it hasn't fully solved the problem of unmanufacturable data packages being released for production.

Why is this? First, fabricators don't provide enough process capability and rules to customers or DFM software developers. There are multi-variable conditions and logic that can't be provided in a simple data table format. Process engineering often doesn't provide sufficient process rules to their own front-end engineering teams, resulting in conducting meetings to resolve questions. It can take a year to train a new product engineer to become proficient with the internal rules and many customer requirements.

eCAD/mCAD DFM software also doesn't have all the manufacturer's rules available. Material suppliers don't provide broadband TDR-based dielectric constants to map materials to the intended impedance values. TDR measurements are the most specified impedance test method. Fabricators must perform their own testing to generate these values. So, the analysis software both parties are using with supplied material values can't be relied on to yield a perfect stackup.

Designers generally don't have sufficient allocated time to review multiple fabricators' rules, which allows them to develop an average set of rules that can be successfully used by all the approved fabricators. Many large companies do require a nearly common set of rules to be used by all suppliers because they have technology teams that are separate from the layout engineer to develop this.

Generative AI and Digital Twin to the Rescue?

Alas, generative AI, along with all the manufacturing and processing data in the PCB digital twin will not solve the problem. The theory states that the design intent digital twin can be combined with the production lot-by-lot process and quality data using AI to yield the perfect data package for future designs. There is a minor problem with this solution: It doesn't exist.

This has the most potential to bond the designer to the front-end engineer. The multi-variable DFM requirements, along with the information garnered from prior production of the down revision or similar part number, could be answered by computers. This would reduce the human communication interchange to resolve the most complex issues. Unfortunately, PCB manufacturing is a low-margin business, with most companies maintaining very small software development teams required to develop these rules.

Conversely, allowing the front-end engineer to bounce proposed solutions against the designer's digital twin AI knowledge base without generating a TQ would provide high-speed access to the designer's knowledge and not require human-to-human contact.

Humans to the Rescue

Front-end engineer-to-designer direct interaction and relationships can generate perfect data packages. There are many examples of this. How can this be implemented and what is the business benefit?

“Front-end engineer-to-designer direct interaction and relationships can generate perfect data packages.”

I have seen many close relationships yield lower defect data transfers. I have seen as many as 25–40 TQs per design be reduced to just a random TQ every now and then simply by having the designer and tooling engineer spend time understanding each other's constraints and issues. Understanding what the primary design constraints are, such as product volume, signal integrity, cost, power density, or reliability, can guide suggested manufacturability improvements.

This is a multiphase process with substantial business benefits for the OEM and manufacturer.

First, the ordering company's procurement team must establish a good relationship with the manufacturer's sales and customer service team. The sales/customer service team needs to demonstrate that they will bend over backward to help get the product when there are issues or unexpected volume upswings. Open, honest, and realistic communication is critical for success. This works best when the same person on both sides stays engaged over time so the relationship can be established.

Second, the OEM's designer and fabricator's front-end person must establish a relationship. This allows the manufacturer and designer to understand one another's critical requirements and requests. It is common to assign a specific customer to one or two front-end engineers to help understand the unwritten requirements. Multiple people are required to cover illness and vacations or have availability on multiple shifts.

The highest impact relationships occur when both sides agree to improve and reduce the amount of TQs and quality problems, such as the front-end engineer no longer asking for modifications that the design team will never approve, or the design team adjusting their routing rules or library models to eliminate CAM edits that are repeated across multiple designs, like adding thieving patterns. The fabricator can generate a “Global QT Approval” specification. This will document the most significant, agreed-upon decisions. It can also be used in training new designers and new front-end engineers. This document is appended to the customer's internal procurement specification.

As the relationship grows, other fabricator team members become engaged to help provide recommendations to the customer design team and, conversely, to the process engineering team for improvements.

Field application engineers (FAEs) are valuable resources. They can help during the pre-layout stage, or as I call it, the “off the clock” time. The FAE will also develop a close relationship with designers to understand their requirements.

Business Value

When the designer and manufacturer are in sync, you can obtain significant reductions

in the quick-turn cycle time. TQ negotiations may add three to five days to a procured three-day cycle time, making it a five- to six-day cycle time. Reducing the number of TQ questions allows the designer to release the next design or design revision faster because they won't be interrupted as often, which results in a shorter new product introduction (NPI) cycle time. From the fabricator side, this allows the front-end engineer to spend less time analyzing the design, generating fewer questions, and releasing the production tooling faster, allowing them to release more projects per day to the production floor and increasing revenue while reducing per-project costs. Finally, the resulting PCB will be better quality and more reliable product.

Can the Designer and Manufacturer Be in Sync?

Yes, designers and manufacturers can be in sync, and it is to everyone's benefit when they are because it results in lower NPI costs, faster cycle time, higher yields, and a better-performing product. Sometimes the lowest initial price will not yield the lowest life cycle cost. There are many examples of close designer-to-manufacturer relationships that yield a lower cost or provide the manufacturer with valuable insight into how to tool up for the customer's future requirements. A strong designer-to-manufacturing engineer relationship occurs naturally when each party listens to the other and realizes they are part of the same team. **PCB007**

Dana Korf is the principal consultant at Korf Consultancy LLC, and an I-Connect007 columnist. To read past columns, click here.

PCB Market to Grow by $19.05 Billion from 2022 to 2027

The printed circuit board (PCB) market is set to grow by US$19.05 billion from 2022 to 2027. The market is estimated to be progressing at a CAGR of 5.05% during the forecast period. The report offers an up-to-date analysis regarding the current global market scenario, the latest trends and drivers, and the overall market environment.

Rising industry automation is an emerging trend shaping the market. As a result, several industries are forced to adapt to automation to sustain the competition in the market. But due to the increasing adoption, optimum power consumption can be a significant challenge. Hence, PCBs are increasingly adopted across industries to ensure efficient power consumption within the minimal size of the application. One of the main functionalities of PCBs includes withstanding peak temperatures and voltages. Hence, such factors are expected to drive market growth during the forecast period.

The market is driven by the rising adoption of smartphones. Factors such as the availability of low-cost smartphones and growing global Internet penetration are expected to fuel the adoption of smartphones globally. Furthermore, developing economies like India and China are becoming significant emerging markets for smartphones due to the rising disposable income of people and growing population.

(Source: PRNewswire)


IPC BUILD ELECTRONICS BETTER
Community
MAGAZINE
WINTER 2024
LOOK INSIDE
IPC
BUILD
ELECTRONICS
BETTER
Contact us to
become an advertiser

Getting to Know Your Vendor

The Chemical Connection

Feature Column by Don Ball, CHEMCUT

After working for a capital equipment supplier for almost 50 years, I've found that the most important part of getting to know your vendor is good communication among all parties. While contact between fabricators of a constantly changing product line and the designers of those products may occur daily or weekly, conversations between you and your equipment supplier may be years apart. That lengthy gap often means that previous contacts may have been promoted, retired, or moved on to other opportunities. You may have also migrated to a new supplier with whom you have little or no history. In either case, you will be interacting with someone you are unfamiliar with (as they are with you). Therefore, it is essential for both sides to communicate clearly so expectations will align.

It sounds easy enough, doesn't it? Yet many problems encountered with the equipment end of the business are due to miscommunication between the supplier and the customer. Usually, these come as assumptions made by both parties that are not addressed during the discussions leading up to the final design and purchase order.

A simple example occurred a few years ago. A potential customer contacted us about etching equipment to thin down copper sheets to a specified thickness with tight specifications. They specialized in copper gaskets for high-compression racing engines, cutting them from a copper sheet with a water jet. Their problem was that the milled copper sheet came in standard thicknesses. The mill was able to deliver custom thicknesses but at a greatly increased

price. They were buying standard-thickness copper sheets and sanding them down to the desired thickness with the automatic sander. Not only was this slow and dusty but the sanding work hardened the copper and required an annealing step before cutting. Etching it to thickness would be quicker, cleaner, and less costly. A few test runs in the lab showed that the etching process would easily meet their specifications, so a production system was proposed and accepted, and a purchase order was cut.

The communication problem became apparent when the equipment was delivered. The customer was not familiar with these types of purchases and assumed they were receiving a turnkey system where the equipment would be delivered, installed, filled with cupric chloride, tested, and ready to run. Unfortunately, they did not make this expectation clear to us during our discussions. We were not entirely without fault here as we offer installation assistance as a separate item, but since the customer did not request it, we assumed they didn't need it. As a result, when the truck arrived at the customer's location, there wasn't even a forklift available to remove the equipment from the truck, let alone move it into the building.

Fortunately, it was a small, self-contained system that only needed water and electric hookups, so it was set up quickly, the chemistry found, and process training provided in short order. Had this been a larger and more complex system, the simple communication failure would have been a major problem for both parties.

One way to avoid this type of communication gap is to set up a factory acceptance test, where the finished system is set up in the factory and run, usually with water, before delivery. The customer representative(s) review the quotes and specifications page by page with the factory engineers to determine whether everything is working as promised or expected. Any anomalies are usually found at this stage, and it is much easier to make corrections while the equipment is still at the factory rather than after it is delivered. Not doing a factory acceptance test can result in a wide range of problems, anger, and frustrations that could have been avoided.

> "Let's work together to identify and fix the problem and worry about assessing the blame later."

A few years ago, a customer wanted to replace a 20-year-old two-stage chemical cleaning system (provided by a competitor that is no longer in business) that was beginning to fall apart. It seemed easy enough. They just wanted a work-alike copy of their old machine. The first stage was an aggressive alkaline clean followed by a three-stage cascading water rinse. The second stage was a mild acid stop bath followed by another cascading water rinse and dryer. The original installation drawing was provided and copied as closely as possible. When the system was finished, they decided to forego a factory acceptance test to save some time and expense. The system was duly shipped and installed.

The initial results were terrible. The panels came out of the system discolored and with obvious wheel tracks which would not be acceptable for continuing to the next process step. The customer suspected the wheel material was different from their old system; several wheel changes were tried to no avail. We were puzzled, as we had no problems with the dozens of these types of systems in the field with the same construction materials using the same chemistries. As the days passed with no solution, anger and frustration grew. Then it was discovered that the pump taking the overflow from the second stage rinse (after the acid treatment) to the first stage rinse after the alka-

line clean had not been hooked up. The customer had not been mixing the alkaline cleaner solution properly but got away with it in the old system because the rinse after the alkaline clean became slightly acidic from being supplied by water from the acid rinse. This neutralized the alkaline chemistry on the surface of the board as it came out of the alkaline cleaner before it could mar the board's surface. The pump was hooked up correctly and production was underway quickly. A factory acceptance test would likely have picked this up and saved everyone much time and aggravation.

The moral is that even when you think you have good communication and trust between both parties, it never hurts to double-check. As former U.S. president Ronald Reagan often advised, "Trust but verify."

One final thought (that should put us in good stead through all walks of life) is that sometimes, despite the best intentions, miscommunication and mistakes occur. I know I've made the occasional error. There are generally two responses when this happens: scream and shout, point fingers and threaten to sue; or swallow your anger and frustration, then say, "Let's work together to identify and fix the problem and worry about assessing the blame later." I've been on the receiving end of both responses, and I don't think I have to tell you for which customer I am more willing to go the extra mile. **PCB007**

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

I-Connect007 Columnist George Milad Passes Away

I-Connect007 columnist George Milad passed away on Dec. 7, 2023, at the age of 81.

George was well-known in the PCB industry for his technical expertise in physical organic chemistry. He was employed as a national accounts manager at Uyemura International Corporation, was the author of the chapters on plating and surface finishing in *Printed Circuit Handbook: Seventh Edition,* and had a series of publications on electrolytic plating and metallic surface finishes.

He was the recipient of the 2009 IPC President's Award, chaired the IPC Plating Committee for many years, and was a member of the IPC Technical Activities Executive Committee.

George was known as a lifelong learner. Born in Egypt, George immigrated to the United States in 1970. According to his obituary, "His view of life was a compilation of his continuous search for truth, his own experiences, and most importantly his shared experiences with those he loved and admired. He believed that our duty on this earth is to leave a legacy of compassion and service, and was proud to say that his legacy is in good hands with his children and grandchildren."

On LinkedIn, Uyemura posted this sentiment from Tony Revier, senior strategic advisor and president emeritus: "George Milad was an industry icon. His embracing nature, commitment to excellence, and unique ability to engage others earned him great admiration and universal respect. We feel the loss of this treasured member of the UIC family."

Condolences offered on Uyemura's LinkedIn page remembered George as "my favorite competitor," "kind, soft spoken, and a true gentleman," "a loss to the industry," "a prolific author," "an intelligent friend" and "great mentor."

Teresa Rowe, senior director, standards and practices, said: "George participated in IPC standards development projects with passion and the spirit of volunteerism. Never one to shy away from helping his industry, he also volunteered in recent years to be an Emerging Engineer Mentor. We will miss him."

George's I-Connect007 column is titled "The Plating Forum," and his contributions were greatly appreciated and widely read. Search his past columns here. Both on a personal and professional level, George Milad will be missed.

Fostering Loyal Relationships With PCB Design Engineers

Fresh PCB Concepts

Feature Column by Ryan Miller and Jeffrey Beauchamp, NCAB GROUP USA

Not to diminish the work of engineers from the past, but today's PCB design engineers have much more technology to learn about than engineers of the past. To make matters worse, technological advancements are moving faster than ever, which puts more strain on design engineers to stay updated. Additionally, PCB designers have more options than ever. While this can, at times, be wonderful, it can also be overwhelming and daunting. In this landscape of new and complex challenges, nurturing a solid and enduring relationship with PCB design engineers becomes increasingly important and substantial.

There's no secret to creating a loyal relationship with a PCB designer, but it does require some teamwork.

During our time in the PCB manufacturing industry, we have had the opportunity to build relationships with PCB designers from all walks of life, who all share common needs. From this, we know that just selling PCBs that meet specifications and timely deliveries is not sufficient to create this relationship. Instead, loyal relationships with PCB design engineers can be fostered by considering the following ideas, which are often overlooked amidst the daily grind of our industry.

Maximizing Reliability Across the Board

In the PCB industry, reliability is the most sought-after goal. Throughout our careers, the PCB design engineers we have worked with not only look for a reliable product, but also

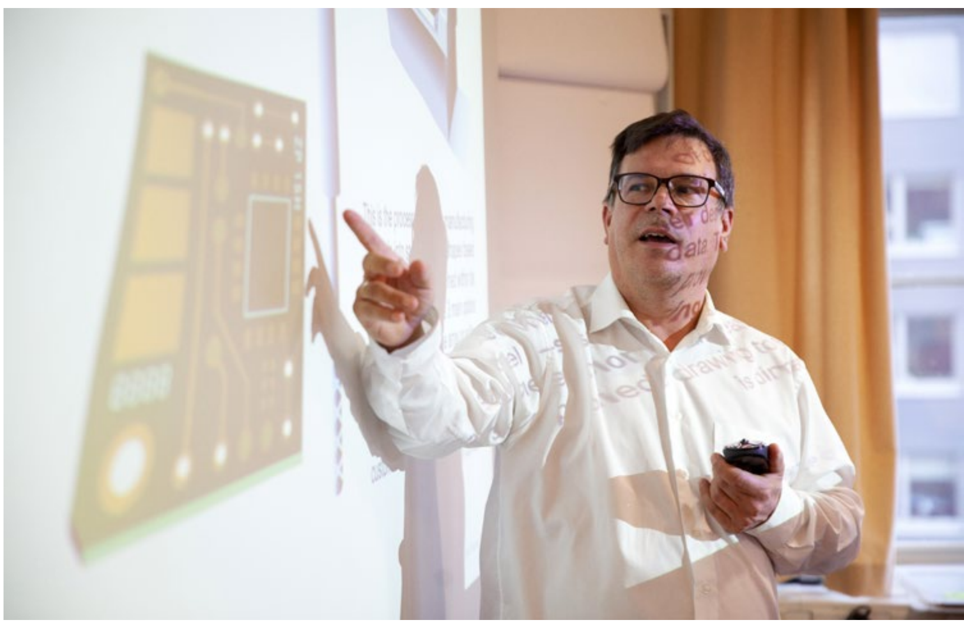
for reliable delivery. In scenarios where time-to-market is limited, even minor setbacks can trigger significant problems.

To mitigate such issues, PCB design engineers typically choose a factory that already knows their product and, therefore, can deliver the finished product on time and accurately to the specification. To consistently do this, the whole factory must work as a team. While one late or inaccurate PCB may not seem like much, it can halt the overall product delivery. From our experience, engineers prefer not to work with companies that frequently hold up their projects.

Many PCB design engineers we meet are interested in improving PCB design, as well as price and delivery time. Including additional personnel from various departments at the board fabricator can demonstrate the depth of expertise within your company. Bringing this expertise to the customer's product development cycle can not only improve reliability and manufacturability but it develops mutual trust in the working relationship. By taking on this multifaceted or cross-functional approach, you can better provide the customer with more comprehensive support and value.

Building Trust

In addition to reliable production and delivery, PCB design engineers want consistent products. When we were new in this industry, we paid little attention to the seemingly insignificant aesthetics of a batch of boards. We thought that since most PCBs are enclosed, no one would see them; we couldn't have been more mistaken. With extensive experience gained in assisting PCB designers with design troubleshooting, the value of producing consistent products has become very clear. Failures are inevitable in this line of work; when designers troubleshoot, they engage in a process of elimination. If the PCBs are not consistent, it complicates the troubleshooting process. Producing consistent circuit boards ensures our customers have a reliable partner in their professional journeys, and further builds trust between customer and supplier.



Taking on a Mentorship Role

As a design progresses, design engineers can sometimes require customer support to provide expert advice regarding their designs. The PCB design process has many moving parts and constraints; at times, designers need a quick solution to finalize the PCB design and data. When we meet a new design engineer for the first time, it's typically because they had a problem that needed to be fixed yesterday. While it's not an ideal situation to start a working relationship, it's a chance to establish a trustworthy professional relationship with the customer by providing them with a solution to their problem. During this initial meeting, the aim is always to understand the customer's needs and project requirements.

It helps to know some background and experience level for our customer contact. For example, when NCAB sends a data package to a factory, the CAM staff will reply with questions—sometimes, rightfully so, with a lot of questions. Some designers, through no fault of their own, lack the factory knowledge required to answer these questions, and yet, the factory needs this information to ensure an accurate product. This is where understanding the customer's expertise is critical to acting as an intermediary between the customer and the factory. With this knowledge, we can tailor the support based on the customer's needs.

Added Value: Showcase Your Knowledge and Competence

Another goal during the initial meeting is determining whether the new customer wants educational resources. Providing educational presentations to PCB design engineers not only fosters trust and mutual understanding, but it showcases the value you can provide. If your company does not have a staff member capable of delivering educational presen-

tations, you may consider leveraging one of your front-end engineers knowledgeable with the process at hand. Our experience has been that PCB design engineers love to learn. Basic presentations on such topics as PCB manufacturing and PCB cost drivers go a long way in building a trustworthy relationship. Other subjects—new technologies, for example—may call for more tailored presentations, allowing you to distinguish yourself from other manufacturing facilities. Help your customers see you as a PCB authority, and they will know who to call for expert advice.

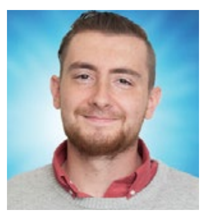
Transparent Communication

All relationships must be built on transparent communication. PCB design and manufacturing relationships are no different. When the PCB is being tooled up at the factory, many questions arise. While some are simple, others require changes the designer needs to know about. If we know the designer well, we can make certain decisions on their behalf to keep the PCB moving at the factory. However, major decisions must be made by the designer. Then, when we send questions to the customer, most have already been answered. For the more difficult questions, we may offer several solutions, or just one. The point is that we always communicate the outcome of each option to the customer. One quick way to destroy a trustworthy relationship with a customer is to modify the design without transparent communication. Though it's not supposed to occur in our industry, it still does and it's important to acknowledge this. After the PCB has been delivered, reach out for feedback and provide what's requested.

Fostering loyal relationships with PCB design engineers is no different than any other relationship. The more support we can provide them, the more value we add to the relationship. Delivering reliable and consistent circuit boards should be the most important goal. However, if the boards are consistently late, your relationship with the customer will wane and fade away. If transparent communication is not an integral part of your approach, you risk losing valuable customers. Finally, educate and mentor PCB design engineers. Going the extra mile is always appreciated by design engineers. This requires effort, but as someone wiser than us once said, "The extra effort is worth it." PCB007

Ryan Miller and **Jeffrey Beauchamp** are field applications engineers with NCAB Group.
To read past columns, click here.


Ryan Miller


Jeffrey Beauchamp

Culture Change is Key to a QMS

The Right Approach

by **Steve Williams**, THE RIGHT APPROACH CONSULTING

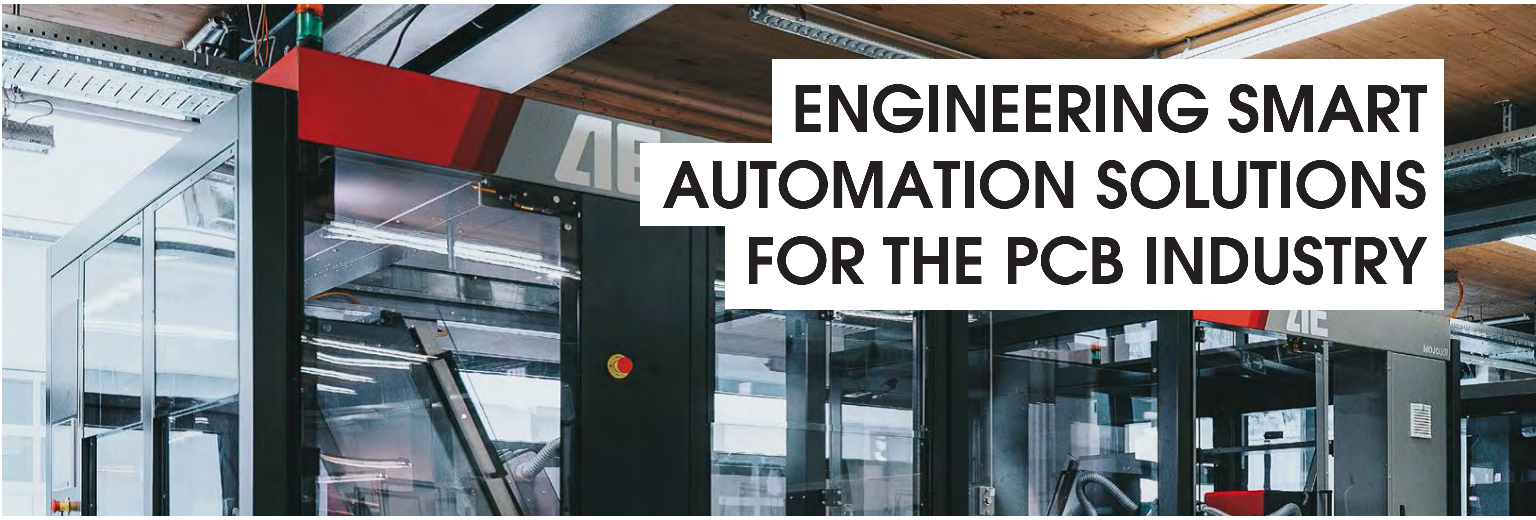
Why is organizational change needed? Because in America, people tend to accept that a certain amount of error is normal. You expect the plane to be late. You expect the mail to be undelivered. You expect consumer electronics and products to break down. This translates to our manufacturing operations as well.

Culture Shock

We see this premise every day in our businesses. We expect a certain amount of our process output to be defective, and we plan for it. This mindset extends further into our quality systems, where often the focus is on detection and not prevention. Quantum operational improvement requires a cultural shift that not just expects but demands exceptional quality to be the norm and perfection the goal. As the great Vince Lombardi once said, “Perfection is not attainable, but if we chase perfection, we can catch excellence.”

I was enjoying a fine cigar with an old friend several years ago when the discussion turned to organizational change. I greatly respected his opinion, so I asked, “How do I begin to steer a company in a new direction that’s so transformational it’s sure to meet a heavy dose of resistance, skepticism, and attitude?” He said, “You have to change the culture, and that is no small task. Steve, did I ever tell you my definition of

AE ADVANCED ENGINEERING



ENGINEERING SMART AUTOMATION SOLUTIONS FOR THE PCB INDUSTRY

/ PMA 101-CPB

Automatic Mylar Sheet Remover

The most successful automated mylar sheet remover in the world with its revolutionary peeling method.

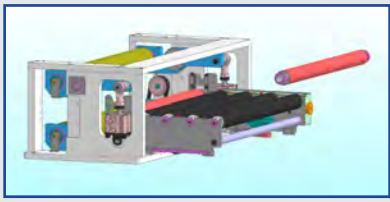
/ SRV 652 SERVANT

Robot Loader / Unloader

Universal loading and unloading robot with a flexible handling concept suitable for a large spectrum of applications.

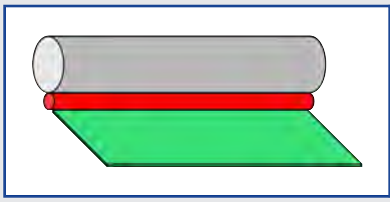
Hakuto

630NP/6630NP
DRY FILM LAMINATOR

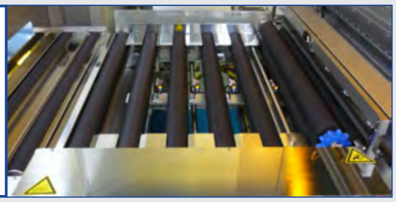


Change Rolls in Seconds

Cassette Loading of Film



EVEN Pressure and Heat



Ultra Thin Core Transport

North American Partner for Sales • Installation • Training • Service

MATRIX
CIRCUIT BOARD MATERIALS

1-800-668-5447 • www.matrixusa.us

TORONTO • SANTA ANA • SANTA CLARA • CHICAGO insanity? Insanity is doing the same thing over and over again and expecting different results." As he has a particular talent for doing so, my good friend had distilled a very complex issue down to a single, critical point. As the evening wound down and we began to go our separate ways, I said, "Good talk Bert. By the way, how's that relativity thing you've been working on?"

Current Culture

Changing an organization's culture is a tough, challenging task. In most cases, the current culture has formed over many years of interaction between employees at all organizational levels. Organizational cultures form for a reason and changing the accepted norm can sometimes feel like swimming upstream. In small, privately owned businesses, the current culture often matches the style and values of the company founder. Organizational culture grows over time and frequently mirrors the prevailing management style. Since managers tend to hire people "just like me," the established culture is carried on through new employees for generations. People are creatures of habit, and basic human nature dictates that they will typically be uncomfortable with change. A key to overcoming this is to acknowledge this human nature predisposition and actively manage the change process through the tools and techniques discussed here.

Sense of Urgency

For real change to happen, leadership must demonstrate the importance of the effort; in other words, they must "walk the walk." Dr. Joseph M. Juran recognized this truth over 40 years ago when he said, "Every successful quality revolution has included the participation of upper management. We know of no exceptions." This means creating a sense of urgency around the need for change that goes far beyond lip service. You must establish an open and honest dialog about the importance of the change and how it will benefit all by improving organizational performance. Urgency is contagious; getting people to start talking about the proposed change will spread like wildfire and feed on itself.

Unfreeze, Change, Refreeze

There are many models for change management, but I prefer the three-stage model developed by physicist and social scientist Kurt Lewin in the 1940s known as Unfreeze, Change, Refreeze. This model explains organizational change by using the analogy of changing the shape of a block of ice. Visualizing change as a process with distinct stages will allow organizational leaders to prepare for what is coming and develop a plan to manage the transition. One of the major reasons change fails is that organizations rush into change blindly and without a plan to manage change effectively.

Stage 1: Unfreeze

The critical task during the first stage is to establish an environment that challenges the current norms and sets the stage for change to occur. The goal is to move people from the current frozen state to a change-ready or unfrozen state. The unfreezing stage is important to break down barriers to change and move away from the "we have always done it this way" mentality.

Stage 2: Change

During this transitional stage, people are aware that the old ways are being challenged,


UcamX Pro+
The only CAM seat
with a missile launcher
• YELO Automatic DfM Adjuster
• Tombstone and Solder Escape Prevention
• Secure Etch Compensation with Horns
• Parallel Processing, 64 bit

but at this point, there may be no clear understanding of the new ways that will replace them. Care must be taken to keep fear of the unknown from paralyzing the effort, and strong leadership during this phase will be the difference between success and failure. The goal of this stage is to get and keep people in an unfrozen state while establishing new ways and norms. Convincing people that it is okay to break away from traditional norms and adopt new ways of doing things is crucial before moving into the refreezing stage.

Stage 3: Refreeze

The goal of this stage is all about reconnecting people to their new comfort zones and establishing this state as a safe, familiar environment. Refreezing seeks to reinforce the new behaviors and enable permanent change through rewards, celebrating each success, and continuous communication.

Communicate, Communicate, Communicate

When it comes to changing the culture of a company, there is no such thing as over-communication. Involving the entire workforce from day one will be a key to successfully changing the current culture. Secrecy is the enemy of success. This process calls for full disclosure, employee participation, and individual empowerment. Effective communication will allow the creation of a type of controlled mayhem; in other words, it's an environment that can build a strong motivation to seek out a new equilibrium. Without this motivation, it will be difficult to get the employee buy-in that will be required to facilitate any meaningful and permanent change. This journey will be an emotional one that will challenge the status quo, often evoking strong reactions and resistance. Remember the human nature thing? Paradigm is overused, but I don't have a better word for describing "the way things are done." This will be a paradigm shift that can only be overcome with effective, thorough, and constant communication.

Create an Environment of Excitement

I firmly believe in making work as fun as possible, and this is the perfect situation for that approach. A quality management system is a topic not often associated with an interesting, much less fun transition. A cultural change as significant as this presents the opportunity to smooth the transition by creating an environment of excitement around the change through communication, publicization, and engaging the workforce in the implementation decision-making process.

Change is Constant

The Greek philosopher Heraclitus had it right 2,500 years ago: The only thing constant with change is change itself. As I mentioned earlier, people are creatures of habit and will be uncomfortable with moving out of their comfort zone. Change is difficult, and some people will not adapt; this is normal. As with any major cultural change (ownership, management, quality system), the only certain thing is that not everyone will drink the Kool-Aid. While this can be a trying experience, stay the course. **PCB007**

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

Get to know your PCB designer

Average age
44

Total in U.S.
10,437

Gender
85% Male

Average salary
$74,841

Entry level salary:
$48,000

Essential job skills

- RF **10.44%**
- DFM **6.55%**
- PCB Layout **6.47%**
- Schematic Capture **5.85%**
- IPC Standards **4.46%**
- Other skills **58.7%**

Educational level

- Associate **45%**
- Bachelor's **36%**
- Diploma **8%**
- HS Diploma **6%**
- Other degrees **5%**

Source: Zippia.com

Easing the Learning Curve for Young Professionals

The New Chapter

by **Paige Fiet**, TTM-LOGAN

My first semester of college included a course on engineering fundamentals that focused on teamwork, problem-solving, ethics, and, of course, coding. I had no experience in coding. In fact, downloading the program to my laptop alone almost required visiting the IT department. This class was my second course on my first day of school. Shortly after the introductory speeches, we were asked to write a "simple" code that output the phrase, "Hello World." Instant panic sets in as my other three team members started typing away. Was I supposed to have learned this in high school?

Throughout the rest of the semester, I stumbled my way through coding. It took multiple one-on-one sessions with the teaching assistant before I really understood why a semicolon was needed at the end of a command, let alone how to make a "for" loop. Now that I'm out of college, I understand this experience was not unique but something that many engineering students unfortunately experience. While some might say it built character, it was discouraging to feel behind from my very first day.

This story is not exclusive to the world of coding. It can be easily translated to the experience many of us have when first joining the electronics industry. The language used to describe features or defects is unique to the industry and can be just as intimidating as writing an "if" statement for a new coder. The knowledge required to be successful in the PCB world is not currently taught in high schools or most colleges. There is little outreach from companies to local schools to provide skills-based training and mentorship. Students are often left to teach themselves, assuming they can find their way into electronics before landing a full-time position. Even if students are taught electronics, these classes mainly focus on components

and circuit analysis, not on choosing the right dielectric materials or surface finishes based on the end-use application.

To ease the learning curve for young professionals, we should move the on-ramp forward by a few years. Companies must focus more on high school and college students instead of waiting for recent college graduates to stumble into the electronics industry. Recruitment this early might look like after-school mentorship programs, funding electronics courses/labs, and hosting internships. These activities can give students a taste of what it's like to work with electronics, who will then focus their coursework on a curriculum that benefits their future careers. These opportunities provide a competitive advantage when it's time to apply for full-time positions.

Next, proper training programs need to be put in place. Companies with strong training programs create a strong support system for new hires. I believe a robust training program is the foundation for career success. These programs should include a point of contact for employees to ask questions, a trainer to work alongside them, and an onboarding track to teach general terms and definitions. Even new hires with some PCB experience won't know it all. Employers must help fill these education gaps.

Along with training, there needs to be more engagement with the industry, not just within a company. Involving today's youth in trade shows, for example, is a great way to engage them. These shows are breeding grounds for networking and innovation. Employees are able to meet like-minded individuals, both within their own company and others, who understand the hardships they face. The saying about taking a village to raise a child is the same for growing a professional career. Companies can also engage youth through clubs or events designed for their employees' experience levels.

Finally, we will not gently ease young professionals into electronics without strong mentorship programs. The professional world may not come with teaching assistants, but it can come with mentors. IPC's Emerging Engineer Program and other company-specific programs provide needed backing to individuals with little to no knowledge of electronics. A good mentor can be the trainer, but it is better when the mentor is not directly involved with the employee's day-to-day responsibilities. Mentors provide answers to "silly" questions as well as guidance to all that the industry entails. I would argue that mentorships are the most critical agent a company can use.

> "I would argue that mentorships are the most critical agent a company can use."

Most students entering their careers in electronics feel just as underwater in their first weeks and months as I did that first semester learning how to code. Without formal learning opportunities or mentorship programs, these budding professionals will be left to flounder and may not stick around to figure it out. Attracting youth to the industry is the first step, but maintaining and growing that pipeline is another. I've already seen some great examples from companies that are successfully attracting and maintaining talent. More companies must follow suit, or we may lose more generations of really qualified candidates. Our future depends on it. **PCB007**

Paige Fiet is a process engineer at TTM-Logan, and in the IPC Emerging Engineer Program. To read past columns, click here.

Driving Operational Efficiencies at Summit Interconnect

Interview by Barry Matties
I-CONNECT007

As the largest privately-held printed circuit board manufacturer in North America, Summit Interconnect is headquartered in Irvine, California, and has eight facilities, including one assembly shop. Summit COO Sean Patterson came back into the industry after a career move that took him to Amazon and now reflects on issues including cultural alignment, the new workforce, and PCB capacity in the United States.

Barry Matties: As chief operating officer for Summit Interconnect, what's a typical day like for you?

Sean Patterson: My day starts with checking in on the day's production, working with sales on upcoming quotes from a customer perspective, and then getting into larger projects for the rest of the day with an eye toward our long-term vision. Overall, I'm looking at our operations from the day before as well as trends from the prior weeks for all eight facilities. Seven are fabrication facilities and one is a quick-turn assembly shop.

You have made some acquisitions, the most recent being Royal Circuits in Hollister, California. Is that right?

Yes, we acquired Royal Circuits in Hollister, Advanced Assembly in Denver, and South Coast Circuits in Santa Ana, California.

You're operating in California, perhaps one of the most expensive states to do business. What's your strategy for offsetting costs?

We focus on efficiency and verifying our processes up front before we start production. This maximizes our workforce to keep labor costs down. We also focus on employee retention by providing long-term career planning with opportunities to move into higher-level engineering positions.

How do you help your employees take advantage of paid training?

At IPC this year, of the 20 people who went through the Emerging Engineering program, eight were from Summit, so we had the largest team there. It's healthy that we all cooperate to bring more people into the industry and not just move them from site to site. That's not helping our industry. We want to continue that IPC training, developing a career path throughout our company and within the industry.

How do you manage operational effectiveness across multiple locations?

You focus on alignment, but you can't do everything all at once. For example, what is the best process for yielding a circuit board? In my facilities, I have seven design of experiments (DOEs) happening at once and we are finding the best process that is also the least expensive. Then we implement that cooperatively throughout all facilities. Each site is making continuous improvement unto itself, so while we can implement certain "best in class" processes across the board, each site also deals with its own issues. We have a roadmap for achieving it but getting there is challenging. This is "change management."

When you acquire a company, you acquire their culture, which is another aspect of change management.

Summit acquired eight new facilities within seven years, so we're working to bring the cultures together and implementing operational improvement programs where there are varying mindsets. Culture is a long-term plan.

Sean Patterson

I really like the culture at Amazon, where I used to work; they have one of the best "at scale" cultures in the world because it has meaning to them. It's not just lip service. For Summit, it starts with sites talking to other sites, suppressing geographical barriers as much as possible, and having our general managers and engineering teams go from site to site to observe and implement best practices. We are getting to know each other. But we have to move away from ad hoc requests on individual processes at a specific site to systematic process improvements across all sites through a culture of helping each other. That is the culture we are working on.

Culture is built through the operating process.

Yes, and it's really important, especially on the quality management system side, to have regimented processes that everyone follows across the corporation. We are also intentional about making space for employees who have ideas about how to do something better. We are careful not to shut down any idea because there's no monopoly on good ideas. We shouldn't have hubris about that. That builds our culture as well. We haven't written down our principles and such just yet. Right now, we are determining what those are.

Culture is not written policy; it's how you act.

We have two cultures: the internal culture and the customer-facing culture. The end customer's needs can be lost on those in the plant when they are dealing with a problem right in front of them. So, communication becomes our most important tool. Are you putting your best foot forward to service that customer, and remembering that every customer has a customer? It's a little easier at companies like ours that do a large portion of our work in aerospace and defense, but we have to remember our internal customers as well. At the plant level it comes down to a level of customer service that we need to continually work on.

> "We have two cultures: the internal culture and the customer-facing culture."

Sean, market conditions have been up and down. If you're in the defense industry, they may be on the way up again. Is that what you are seeing?

While defense has maintained expectations and even seen growth, commercial projects are of some concern. Other executives tell me they're seeing this same trend. It's important to maintain a balance and we like our portfolio right now. The market segments help balance one another.

One of the great challenges for companies right now is hiring—finding people who will actually show up. How are you offsetting or managing that?

About a year ago, it was difficult to hire. We just didn't get in applications. But it's been better recently. Now we get candidates, and we don't see as much competition when it comes to compensation. You need to have a hiring culture too, where you engage people as they're coming in the door. There's a lot of opportunity for our industry to do things differently.

As far as training, we really like IPC's Emerging Engineer program. It's cross-functional and promotes cooperation among normally competitive manufacturers to improve the entire industry. I can have my engineers mentored by someone in another company. Europe provides a good example of this "coopera-tition," where they have apprenticeship programs in the trade skills. They learn how to do everything with their hands.

Broader education must happen within the industry, at our facilities, and at the operator level. We think about it at the engineering level, but we need to do a better job for the operators by giving them full visibility of PCB plant operations. You can understand everything end to end if you put the time in. That's something I like about PCB manufacturing. It's a multi-engineering sort of discipline—a butterfly effect. If an operator sees something happen but doesn't understand where it came from to even know who to call, they just pass on the defect. We can also do a better job of educating beyond what a specific operator does.

That's what I really like about I-Connect007, and I'm always pointing people to your content because outside of that, there's not much literature. The textbooks are confusing. Look at metallization, for example. It should start with hands-on training, and then go back to the textbook to read about every possible way to metallize something. Start with hands-on, then move to a more traditional academic environment.

Sean, where is technology headed? In what areas are you looking to grow?

We are watching the organic growth in the United States. It's certainly the first time in a while that we see investments being made—TTM in New York, Calumet in Michigan, and SEL's new plant. Northridge and SpaceX also made announcements of new facilities. It's probably the most capital investment toward PCBs the United States has seen for decades, and there's a lot of conviction around that, which needs to be maintained throughout the supply chain, including our customers. We need to truly understand the market environments in which we are working, like Mil/Aero. The complexity we see now in engineered products is not what it used to be, so we must get out of the commoditized pricing world. If we are forced to play down in that commoditized pricing world, it's simply not enough to feed the industry in a meaningful way. Through all that, I believe, we can get the flywheel going and get the continued investment that's needed.

But not all PCB fabricators can afford a large capital investment.

Yes, that's true. There are too many that require large cap recapitalization and are on the edge of going out of business. I'm concerned about capacity in the United States as a result. While we now see companies investing in that capacity, much is captive and does not support difficult or complex boards. We need the capacity for complexity, and that takes a whole other level of investment.

Some believe the U.S. has a lack of capability as well as capacity. What are your thoughts on that?

It's a chicken and egg problem. You need the conviction so you can get beyond the purchase order/transactional relationships to more genuine relationships with your customers. The customer then understands the challenges of the industry, particularly so that the tens of millions of dollars in investments that are needed can happen. This needs to happen with the legacy companies, in particular. We must continue to work on this with our customers at the right levels, which is not on the supply chain buying side.

It is the design conception side.

Right. As an industry, we are hurting ourselves with the mentality of just getting a purchase order in the door. Anyone can build anything once. This discussion needs to start in the engineering halls with the same people who go out to the supply chain.

What drives that? Oftentimes, it seems like the fabricators want that, but the customer is not receptive.

You need to be the squeaky wheel. For example, you can lose a job from a customer, but they go to the next company, and encounter the same problem you already told the customer about. That's an unfortunate process. Occasionally, you can get the right customer's attention, but it's also a recognition that we have a capacity crunch in the United States. At the engineer level, there's a technology gap, and that's well understood in the procurement world. Summit takes a partnership approach with customers, often working with them during the design process to improve manufacturability and help alleviate previous issues. This benefits both sides, resulting in faster time to market and improved repeatability.

We've talked a lot about people, but what about automation?

Automation is the right thing to do in the long term, and the U.S. has different problems than Europe with access to people and the cost of that labor. Machines are good at repeating quality. But let's look at inner layers, for example. Most well-run factories are at a 98% yield without automation. So, should I install an entire system that costs seven figures for a 1% increase in yield?

A big focus for me is making sure we bring more people into the industry and teach them, as opposed to only relying on people with industry experience, which drives up our labor costs, worsens our age gap, and does not support the future of the industry.

Sean, is there anything we haven't talked about that you'd like to cover?

We've covered a lot, but let's talk for a moment about the industry from a cultural perspective. We need to be builders again. I wasn't in the industry in the 1970s, '80s, and '90s—the roaring PCB times. But from my view, there were some important dynamics.

First, most of the technology was founded inside the OEMs, and then spun off because some of the people working for the OEMs who thought they could do it better on their own started their own PCB shops. Those outsourced, independent shops then became the ones developing the processes. Engineers were excited to make the next new thing, so in those early times, PCB engineers were the builders of an industry. They had a greater, more altruistic motivation and there was excitement around that. This kept people in the industry. However, much of that excitement shut down through the dotcom bust, offshoring, and simple aging.

Now, you can take an established process, put a process engineer on it, and maintain the line. But it's a maintainer mindset. It's hard to get builders to do maintainer things. They'll do it, but they're gone within two or three years. If we are so good that something becomes just standard and is not exciting to do, then we should figure out how to fully automate the process controls. But with that, our industry has not done a good job of planning for machine-to-machine connectivity—a challenge when many of our

Is it always about yield, though? Are there really many factories able to achieve 98% inner layer yields?

That's the quality argument. And yes, I think that everyone can expect 98% inner layer yields through a focused engineering effort with the board designs we see in the United States. They can get it done, even with an old etcher or whatever non-automated equipment we are talking about. It just takes the right focus, unless you are making all PTFE or all flex.

It depends on what capability you're building.

Correct. As line widths go down, you can do some automation, but that plays into the need for very expensive automated equipment like etchers and developers. It's the right thing to do when it makes sense for the investment, so you strike a balance. There is an ROI, but it's not one or two years. It's more like four to six years. When we have a capacity constraint inside the United States, as we do right now, and with current cost of capital, my choice for investment goes with capacity over automation. You also have the people argument side. You still have to train people to see and understand how these things are done, which happens better on non-automated lines.

So, you're looking at smart choices, which does reduce workforce and cost.

machine suppliers are family-owned companies in Europe.

Still, there are opportunities. With the right culture and the right story, we'll bring people on a new journey of building again in the United States. Maybe it will be building the connectivity between things or building some of these automation systems. Maybe it will be trying to establish the Six Sigma process again, because, in the United States, we've been pretty dismissive of ever achieving that. We've become product-focused instead of process-focused.

With the right story and motivation, engineers can create an ecosystem that feeds off each other. How do we get that going? One way is through "coopera-tition," because one company in and of itself is not enough to feed that. We have to create the environment. I am really looking forward to seeing something along those lines.

"Yes, every engineer should know how to code."

Operational effectiveness is where you have to start. It is not your business strategy; it's your foundation. Part of that is the digital factory and benchmarking process to become a digital factory so you can have digital twin. How far along are you on that?

We've put effort toward that, but it's a process. It is extremely important, because of the complexity of manufacturing, to be able to tell our customers when they will receive their products and actually hit those commitments.

As I look at benchmarking and digital goods, and as you're talking about sensors, does having a digital twin of your entire manufacturing process give you predictive engineering to eliminate the problems before they occur?

Yes, it does. That's the balance, just like with automation. Where's the right investment for the right kinds of returns? You could put a temperature and process pressure sensor on everything and say, "I've got everything online and it looks great." But you just spent $5 million and what did you get for it? There's a lack of education.

And a lack of skill, honestly, that can help implement these changes.

But getting the builders of the world to do it now? You know that those kinds of skill sets are more ubiquitous. You don't have to hire a PCB engineer to do it, because you can use an Arduino to do Internet of Things. You can set it up.

It's almost like hiring a coder.

Yes, every engineer should know how to code. It's like knowing a foreign language. I heard one AI thought leader propose that, in the future, every position title should be added onto, including the words "and Automation Manager." Everyone in every position should be charged as part of their job responsibilities to automate out their jobs. Those are the people that will be useful in the future, and automation doesn't have to mean robots. There are many free tools to take advantage of to drive your cost efficiency and communicate with your customers better.

Everyone has the ability to be creative. It's a muscle that you have to exercise and be encouraged to use. I tell my team that every time I push them hard on process, it's okay to fail. If you're not failing, you're not advancing or you're not advancing fast enough. There's too much risk aversion today. We have to make sure we don't make a bad board, but on the process side of things, you have to play, which really engages that creativity muscle part of your brain.

I think that's a great place to end this. Thank you, Sean, for spending your time with us.

Thank you, Barry. PCB007

MilAero007 Highlights

Launch Vehicle Telemetry Expected to Become Commercially Available Faster as InRange Moves to Market ▶

InRange uses Viasat's global L-band satellite fleet—which Viasat now operates following the acquisition of Inmarsat in May—to provide real-time telemetry data for launch missions. By using space-based communications, InRange will allow launch mission controllers to monitor the performance of missions beyond the line of sight, without the need for ground communications infrastructure.

Rheinmetall Modernizes Romania's Air Defense System ▶

The Romanian ministry of defence has contracted with Rheinmetall to thoroughly modernize the country's Oerlikon GDF 103 air defence artillery systems. For the Düsseldorf-based technology enterprise, this is the first ever major order from Romania, a NATO and EU member state. Worth around €328 million, the contract encompasses the delivery of four systems as well as training, spare parts and other services.

Boeing Delivers First Orca Extra Large Uncrewed Undersea Vehicle to U.S. Navy ▶

Boeing has delivered the first Orca Extra Large Uncrewed Undersea Vehicle (XLUUV) to the U.S Navy following acceptance testing completion this month. The XLUUV, designated by the Navy as "Orca," is a new class of autonomous submarine that can perform long duration critical missions to achieve undersea maritime dominance in changing environments and contested waters.

Second Successful Live-fire for RTX's Raytheon Lower Tier Air and Missile Defense Sensor ▶

Raytheon completed another successful live-fire demonstration of the advanced, 360-degree Lower Tier Air and Missile Defense Sensor, known as LTAMDS. This tactical ballistic missile live fire represents the latest in a series of test event successes, closely following the cruise missile live-fire test and the recently completed contractor verification testing.

Boeing-built X-37B Orbital Test Vehicle Embarks on Seventh Mission ▶

The Boeing-built X-37B autonomous spaceplane launched yesterday aboard a SpaceX Falcon Heavy rocket, marking the beginning of its seventh mission. On this seventh flight, the X-37B will test future space domain awareness technology experiments that are integral in ensuring safe, stable and secure operations in space for all users of the domain.

A Progress Report: Investing in U.S. PCB Fabricators ▶

In the geopolitical arena, the supply chain lessons learned during the pandemic continue to be addressed with long-range plans as well as short-term stopgaps. In this conversation, David Schild, executive director of the Printed Circuit Board Association of America (PCBAA), provides a progress update on the U.S. CHIPS Act, and some of the fan-out dynamics already playing out. As David explains, there is new investment in PCB fabrication that has nothing to do with the CHIPS Act.

Lab-on-PCBs for Medical Diagnosis

Happy's Tech Talk #24

by Happy Holden, I-CONNECT007

COVID-19 created a rush for simple and quick medical diagnostic tests for the public. Lab-on-PCBs (Figure 1), started in 2000 by Agilent, have evolved to other materials and applications[1].

Lab-on-chip (LoC) and lab-on-PCB (LoPCB) are devices that integrate one or several laboratory functions on a single integrated circuit or board. LoC devices are microelectromechanical systems (MEMS) devices (Figure 1) that function as micro total analysis systems (micro-TAS), generally using microfluidics principles to manipulate minute amounts of fluids. In practical terms, microfluidics is about doing chemistry on a tiny scale and trying to emulate nature. Biomedical MEMS (bioMEMS) have emerged as a subset of MEMS devices for applications in biomedical research and medical microdevices, with an emphasis on mechanical parts and microfabrication technologies. Applications include disease detection, chemical monitoring, and drug delivery. There has been rapid market growth for bioMEMS technologies, and many bioMEMS devices are already commercially available; a familiar example is the blood-glucose sensor. There is great potential for large-scale commercialization of microfluidic-based LoC and LoPCB technologies.

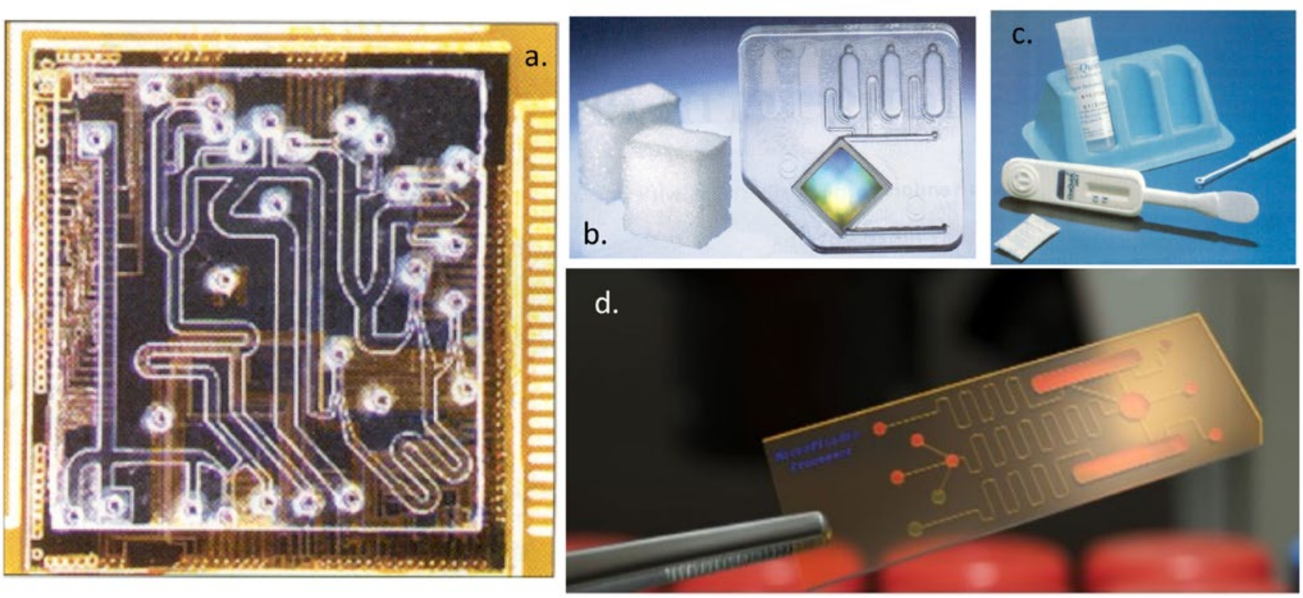


Figure 1: Medical diagnostics have evolved rapidly because of COVID-19. There are various lab-on-chips (1a and 1b) and lab-on-PCBs (1c and 1d) evolving from silicon chips to glass/paper and then PCBs. They can even be produced by additively printing on PCBs (1c and 1d)[1]. (Source: Abbott, Nano Dimension)

Elements of the Lab-on-PCB

The component devices that make up a typical LoC and LoPCB are:

- Electrophoresis fluid movement
- Microfluidics: Channels, valves, pumps, and mixers
- Separation columns
- Heating: Cooling and mixing
- Reaction chambers
- Reagent additions
- Chem-bio detectors and sensors
- UV-VIS/colorimeter sensors
- Microfluidic chips

These units combine into a functioning laboratory (Figure 2) schematic from Agilent.

Materials

Over the years, several materials have been developed for use with LoC and LoPCB. It started in the late 1990s with silicon, as the microelectronics industry developed various methods of MEMS for accelerometers for air bag sensors. From silicon wafers, the materials branched out to glass and then polymers. The most recent interest has been in PCBs and the use of various paper materials.

Silicon and glass have several advantages for fabricating LoC, but they are the most expensive. Polymers and especially PCBs are newer choices because of various available materials and the integration of electronics and types of printing technologies. While some research

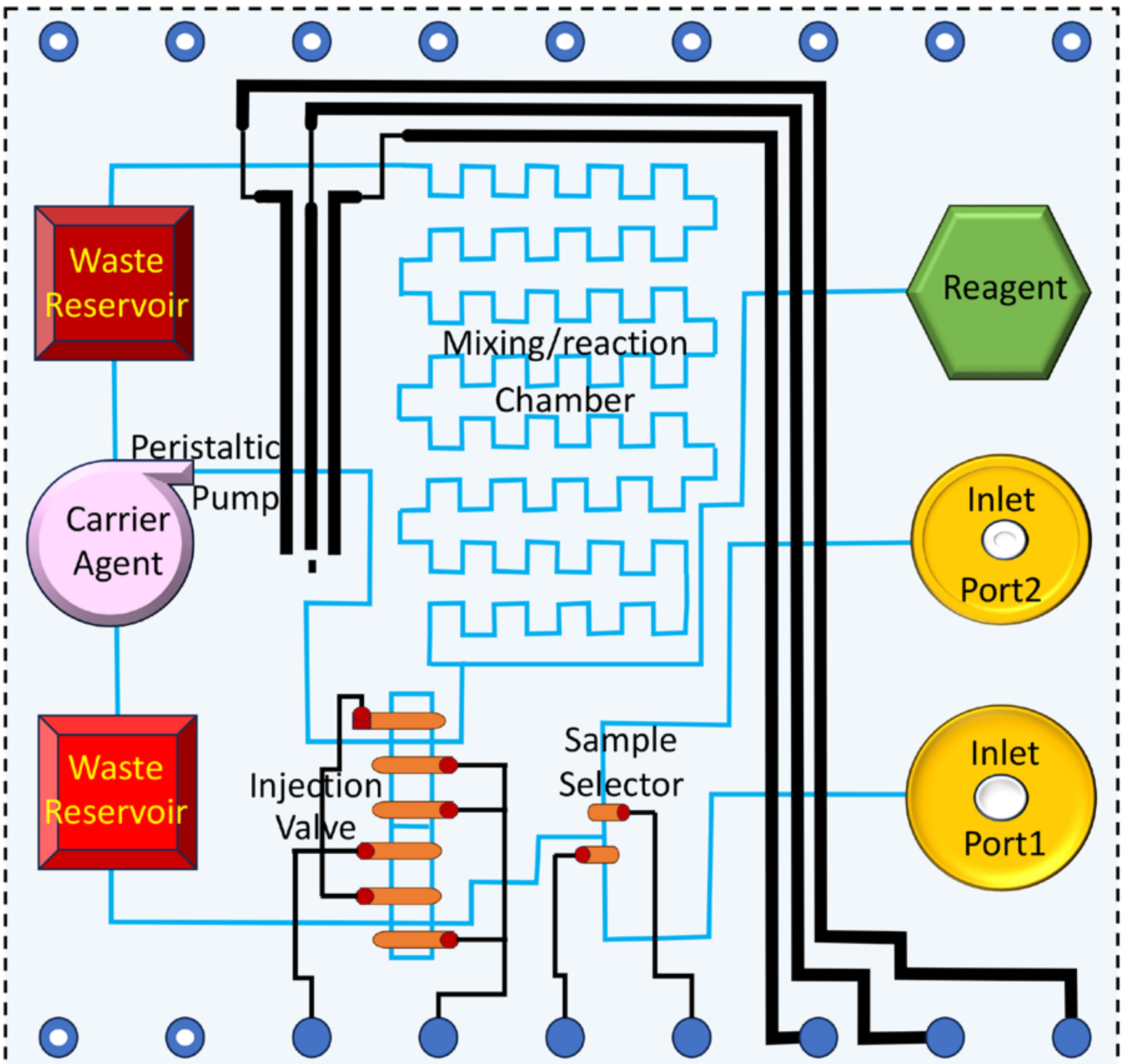


Figure 2: Schematic of a typical micro-TAS on a chip or board. (Source: Agilent Technologies)

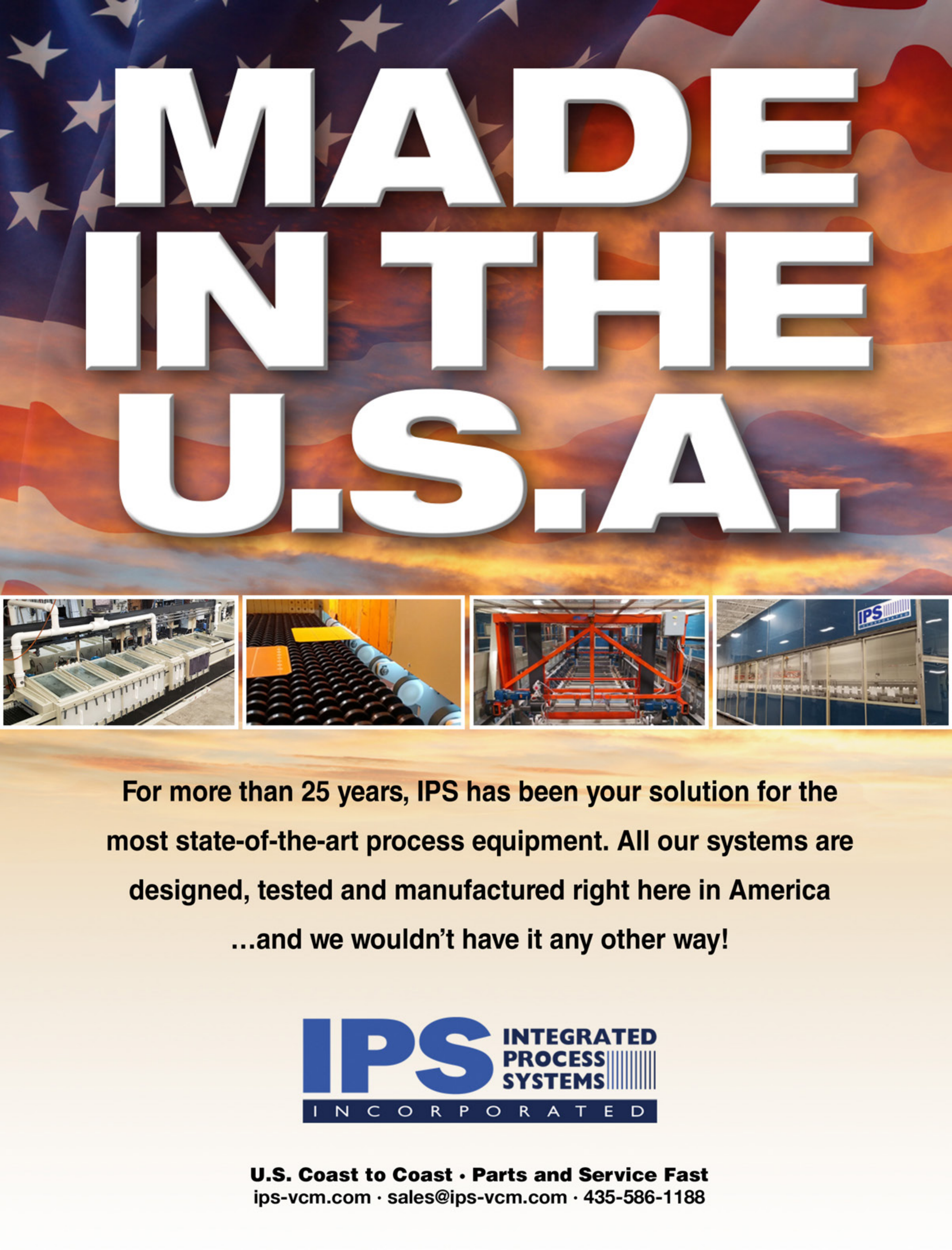

MADE IN THE U.S.A.
IPS
For more than 25 years, IPS has been your solution for the most state-of-the-art process equipment. All our systems are designed, tested and manufactured right here in America
…and we wouldn't have it any other way!
IPS INTEGRATED PROCESS SYSTEMS
INCORPORATED
U.S. Coast to Coast · Parts and Service Fast
ips-vcm.com · sales@ips-vcm.com · 435-586-1188

Table 1: Base materials for LoC and LoPCB formations[1]

SILICON	GLASS/CERAMIC	POLYMERS	PCBs	PAPER
• Semiconductor fabrication techniques • Sophisticated microelectronic devices/circuits • Commercialized components	• Semiconductor fabrication techniques • Transparent, biocompatible • Commercialized components	• Cost effective • Easy polymer processing • Elastic, flexible, transparent, biocompatible, versatile	• Cost effective • Sophisticated microelectronic devices/circuits • Commercialized components	• Cost effective • Printable • 3D, passive microsystems
• World-to-chip interfacing • Cleanroom facility required • Footprint limitation • Hermetic sealing	• Electronic/biosensor integration • Expensive material and processing • Hermetic sealing	• Electronic/biosensor integration • Process standardization • Difficult metallization /hermeticity	• World-to-chip interfacing • Electronic/biosensor integration • Standard processes/materials • Semi-hermetic sealing	• Electronics integration • Detection sensitivity • Minimum feature size defined by printing • Non-hermetic sealing • Moisture sensitive
High-spec applications	Optofluidics	Microfluidic chips	Qualitative applications	Qualitative analysis

now focuses on paper, its use is only just beginning. Table 1 lists several characteristics of each of these materials.

Operating Sequences

It is quite remarkable what can be accomplished on a miniaturized scale. Once the technology of MEMS was perfected on silicon wafers using the photolithography, etching, metallization, and lamination processes, it became possible to miniaturize chemical analysis. But it was discovered that certain activities could be achieved in the micro and fluidic arena that did not have an equivalent in the larger real-world, such as electrophoresis (fluid movement by surface tension and applied voltages).

Once a sample has been deposited on the inlet port to a LoC or LoPCB, electro-osmotic fluid movement takes over and various actions are accomplished; the sample can be heated, passed over a stationary bed of reagents, and additional fluids can be added by the external device and mixed. The sample can be reacted in reaction chambers, and then passed to various sensors for analysis.

One of the unique variations is in the Agilent DNA and RNA Analyzer that, after the LoC cell (Figure 3e) has prepared the sample, it is fed into a nano-inkjet cartridge that deposits thousands of nano-drops on a prepared optical slide with thousands of micro-spots to react to the prepared sample. After incubation, a laser scanner records the results and prints out a final report (Figure 3f).

Figure 3 shows several of these sequences and sensors that today offer medical diagnosis in a few minutes. This used to take clinical laboratories weeks to perform, and new techniques like DNA and RNA analysis were not even dreamt about a few years ago.

Examples

In Figure 4, University of Bath researchers started with LoC experiments then progressed to using PCBs. Now LoPCB components are available, like DNA sensors and heater-on-a-PCB (Figures 4a–c). The process is fabricating fluidic channels and sensors/activators and bonding biosensor chip before final sealing (Figure 4d). The performance and illustration of lactose and glucose PCB plated sensors is shown in Figure 4e. Figure 4f shows a polyimide (PI) flexible PCB with copper- (Cu) and gold- (Au) plated sensors, coated with a gel of EDOT on a graphene carrier of AuNPs and GOx (WE).

Figure 5 shows the microfluidic elements of which fluidics/osmatic fluid movement is paramount. The use of photosensitive dry film solder mask is used for forming fluidic networks, as seen in Figure 5a. Figure 5b shows CAD design


ventec
INTERNATIONAL GROUP
騰輝電子
autolam: Base-Material Solutions for Automotive Electronics
autolam
Automotive electronics technologies are evolving at an increasing rate. Paying attention to the properties of materials at the substrate level is the first step towards achieving the most stringent performance targets of today's automotive manufacturers. autolam offers the solutions demanded by the diverse and unique requirements of automotive applications today and in the future.
IATF 16949:2016
CERTIFICATION
venteclaminates.com

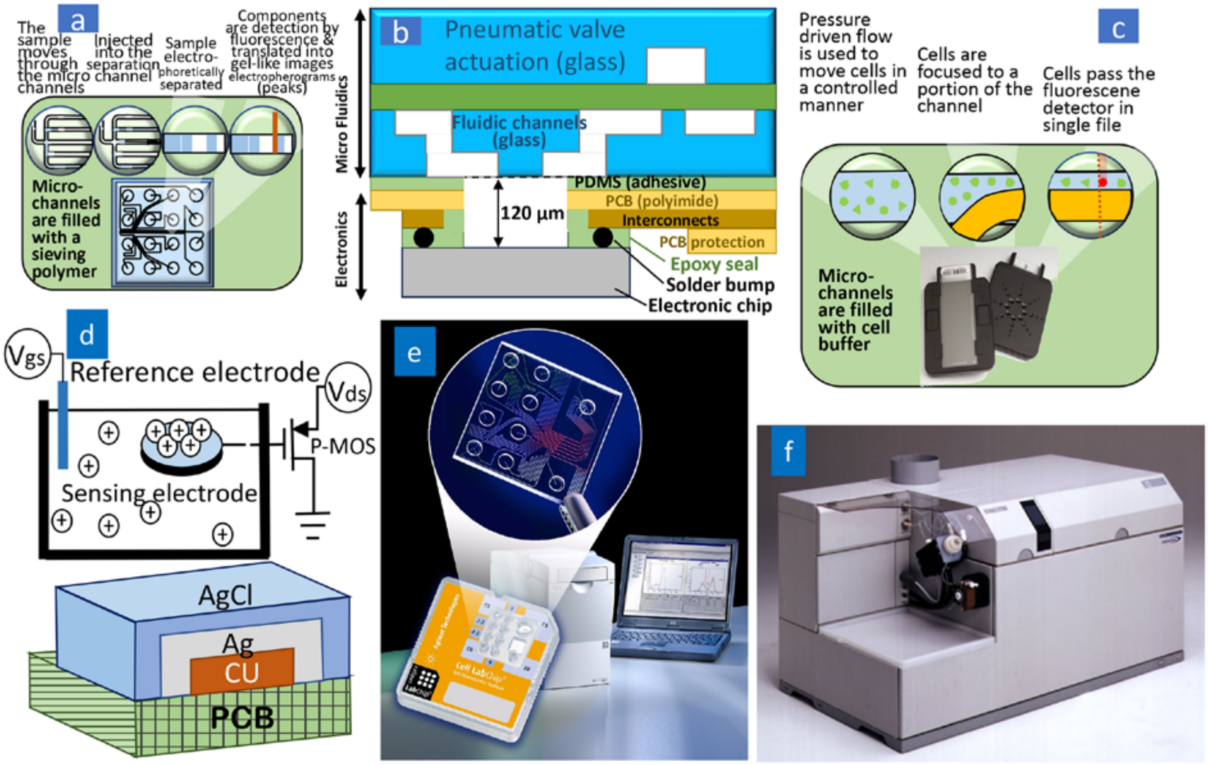


Figure 3: Analytical sequences and elements of LoC & LoPCB: a) Sample analytical sequence using electrophoresis; b) Flip-chip bonded to a flexible PCB with glass structure using photo-defined solder mask (PDMS); c) Sequence using micro-channels for micro-LC/MS of medical samples; d) Reference electrode (P-MOS), Bio-FET, construction of plated Ag/AgCl on copper; e) Agilent Nano LabChip used for medical analysis; f) Agilent Technologies has been involved in life sciences since 2001 after spin-off from HP[1].

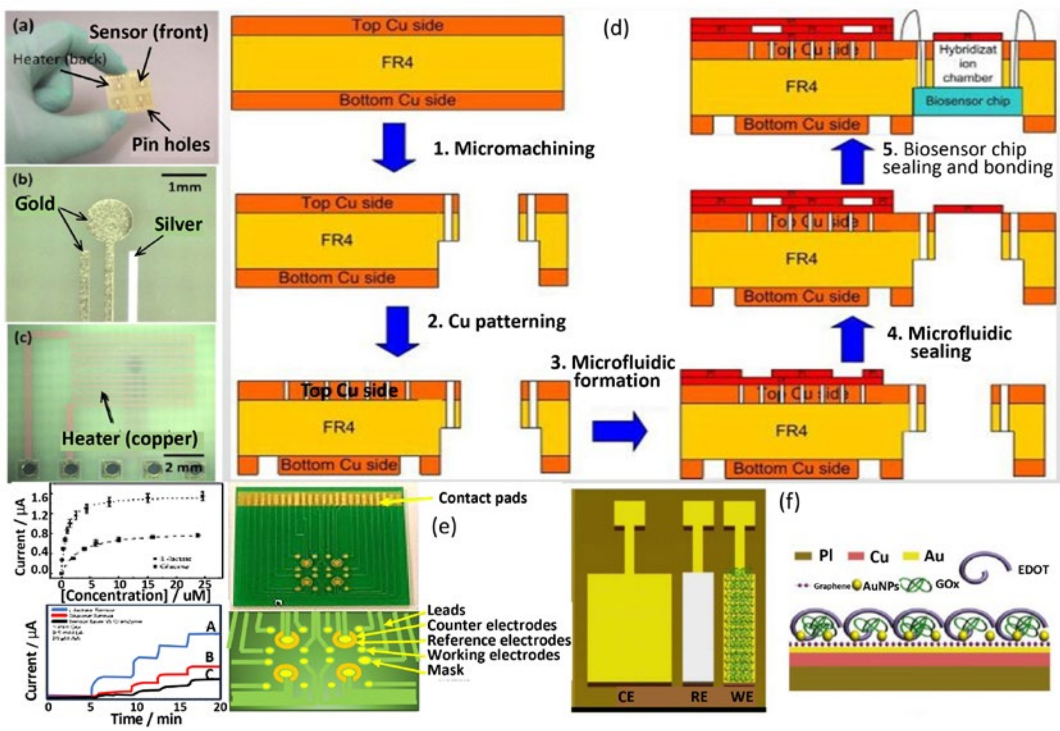


Figure 4: LoPCB components: a–c) DNA sensors and heater on a PCB; d) Fabrication of fluidic channels and sensors/activators; e) Performance and illustration of lactose and glucose PCB plated sensors; f) Flexible PCB sensors[2].

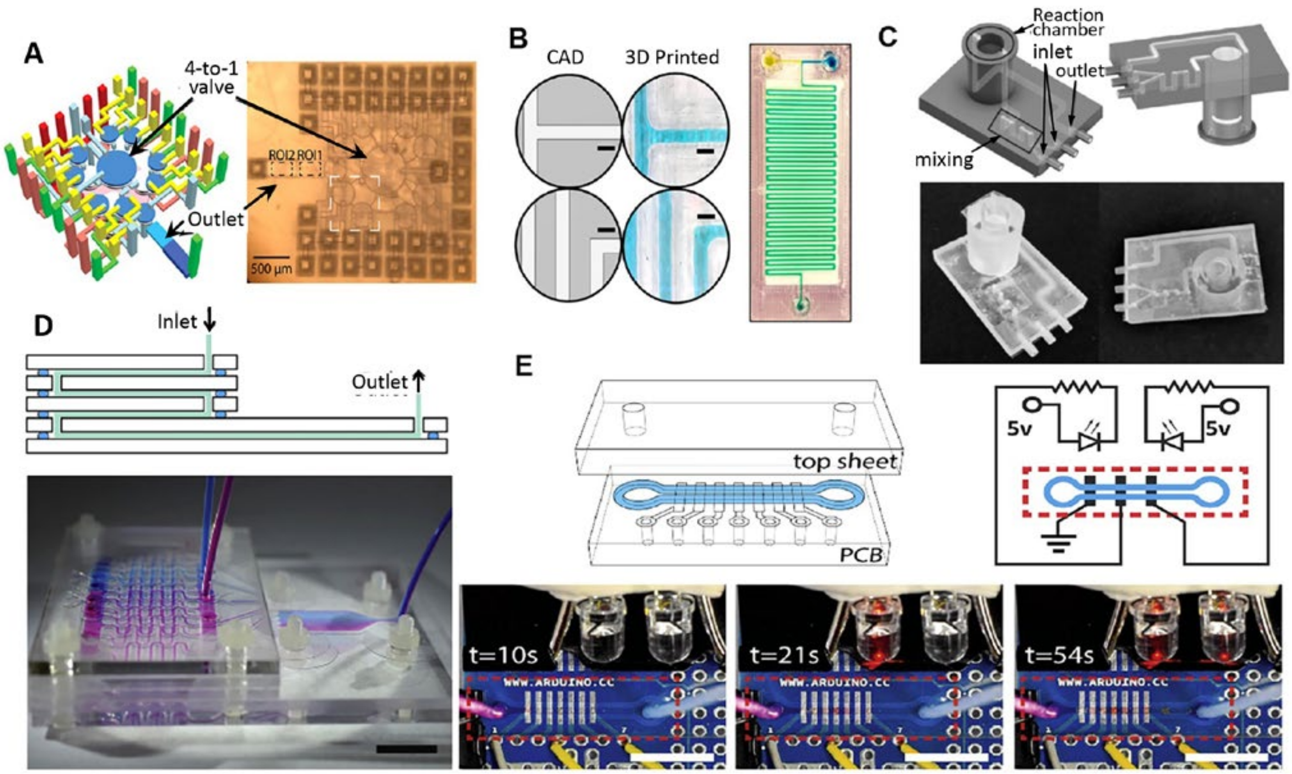


Figure 5: Microfluidic elements: a) 3D printed; b) 3D printed transparent devices; c) 3D-printed mixing stage and reaction chamber; d) 3D-printed multilayer microchannels and 3D printed fluidic channels on an Arduino PCB[3].

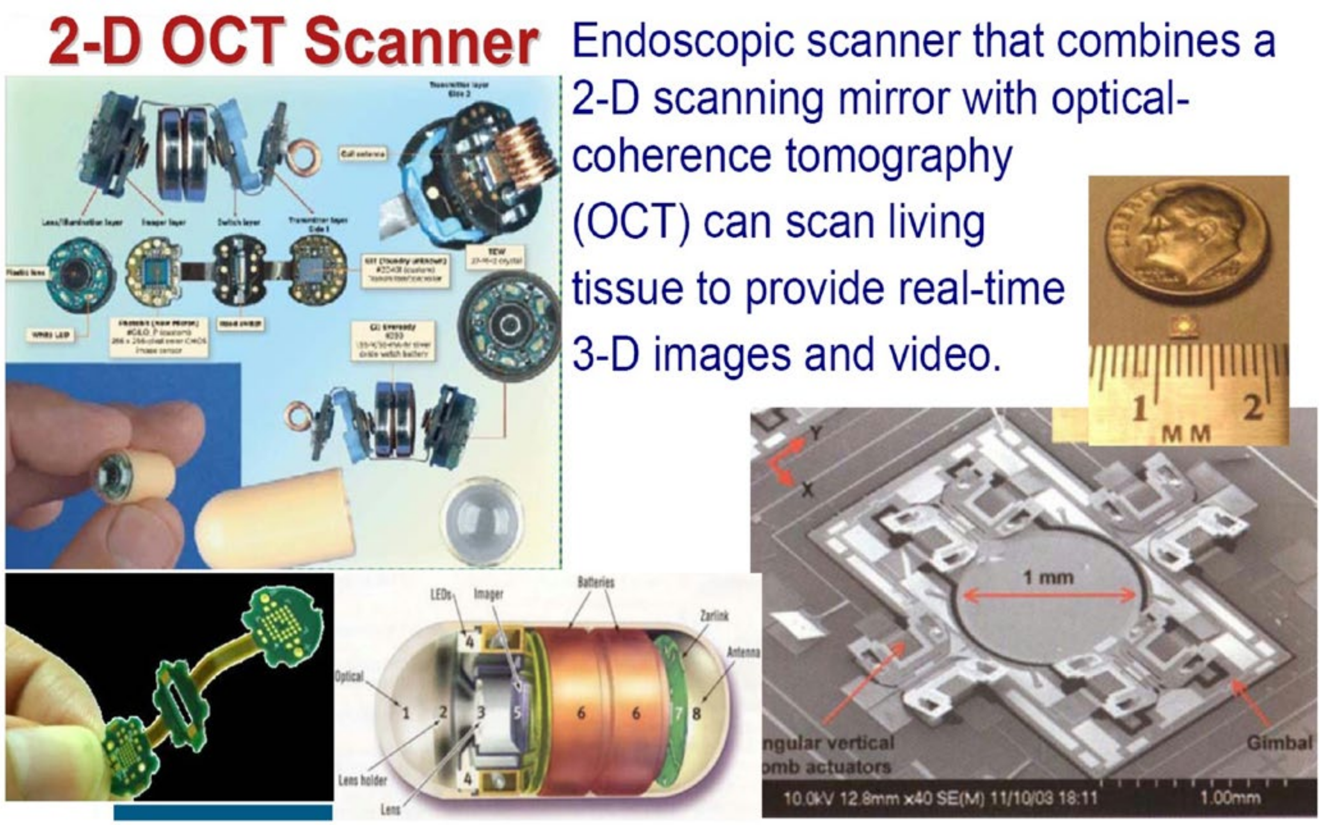


Figure 6: Some PCB medical devices are so small they can be swallowed and directed through your intestines to perform 2D optical coherence tomography. (Source: Portelligent)

and actual transparent 3D-printed serpentine mixer. In Figure 5c is a 3D-printed microdevice of a chip featuring a mixing stage and a reaction chamber where a photomultiplier can be integrated for bioluminescence detection. In Figure 5d are 3D-printed multilayer microchannels as small as 32 mm, utilizing a flexible silicone resin and then 3D-printed directly on an unmodified Arduino PCB, demonstrating a fully integrated microfluidic-microelectronic interface (Figure 5e).

Some PCB medical devices are so small they can be swallowed and directed through your intestines to perform 2D optical-coherence tomography (Figure 6) from a tear-down by Portelligent. The endoscopic scanner uses a 2D scanning MEMS mirror only 1 mm in diameter to provide real-time 3D images and video for the physician; the device is only 3 mm square. Future devices have propulsion and steering.

Summary

Advancements in µHDI PCBs and 3D printing for additive manufacturing have led to the new products of smart microfluidic packaging for medical diagnostics. The potential link between microfluidics and multifunctional biosensing can therefore be found in the realization of 3D microfluidic manifolds, propelling lab-on-PCB technology to enable low cost and rapid micro-Total Analysis Systems—micro-TAS (µTAS). **PCB007**

References

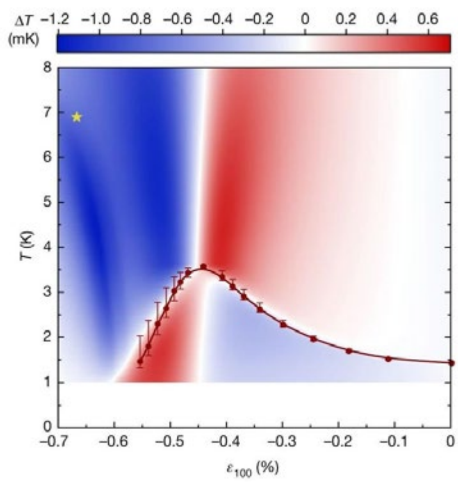
1. "Advances in Medical Diagnostics Using Lab-on-Chip and Lab-on-PCB Technologies," by Happy Holden, *PCB007 Magazine,* April 2020.
2. "LoPCB Technology," by Despina Moschou, AltiumLive, Munich 2019.
3. "LoPCB: One step away from the accomplishment of µTAS," *Biomicrofluidics,* May 2022.

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 the author of *Automation and Advanced Procedures in PCB Fabrication,* and *24 Essential Skills for Engineers*. To read past columns, click here.

Quantum Materials: Superconductor Performs Best Under Pressure

Researchers of Karlsruhe Institute of Technology (KIT) and Max Planck Institute for Chemical Physics of Solids (MPI CPfS), Dresden, have now found that mechanical pressure enhances superconductivity and, at the same time, facilitates deformation of strontium ruthenate. They attribute this to quantum mechanics excitations of the electrons.

Superconductors do not have any electrical resistance when the temperature is below the so-called transition temperature. Scientists have not yet understood why strontium ruthenate (Sr2RuO4) is superconductive. "Conventional theory cannot be applied to strontium ruthenate. But quantum mechanics helps, as it cannot only be used to describe the properties of single atoms and molecules. It also explains the collective properties of multi-particle systems," says Professor KIT's Jörg Schmalian.



Schmalian is one of the main authors [reporting] that mechanical pressure in a certain direction considerably increased the transition temperature of strontium ruthenate and, as a result, changed the excitation behavior of electrons. This pressure increases superconductivity. The researchers attributed this to quantum mechanics resonance of the electron oscillations.

(Source: Karlsruhe Institute of Technology)


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

The Flexibility We Need in Standards-driven Manufacturing

Interview by Barry Matties
I-CONNECT007

Ventec International COO Mark Goodwin and technology ambassador Alun Morgan had quite a bit to say at productronica 2023 about slash sheets, IPC standards, and how to bring the PCB designer closer to the supplier. As Mark says, what matters to designers, manufacturers, and even consumers, is that the products meet compliance standards, such as REACH. They don't need to know how something is built. They just want performance and availability.

Barry Matties: Since our last conversation regarding your recommendations with slash sheets, where are you now?

Mark Goodwin: At IPC APEX EXPO 2023, we learned that the whole industry has been misunderstanding the descriptive information at the top of the IPC slash sheet; it actually has no relevance. It is not part of the material specification.

Alun Morgan: The slash sheet structure was good 30 years ago when you could define properties by family groups, with only three or four resin types available. Today, we no longer have resin systems that are only one resin type; we have fillers and mixtures and all kinds of performance enhancements. The future moved on from that framework; it doesn't fit anymore.

Alun Morgan

That was why we joined in and worked with UL to create a comprehensive list of resin chemistry definitions for designers. If you're a PCB designer, you ask, "What do I need to know to make my decision?" Now, they can look it up and know exactly what it means when you say "primary resin," "flame retardant," or "filler." We have a basis of understanding and some common language.

Goodwin: We worked on those definitions and proposed that they be included in the preamble of the IPC-4101 specs.

Morgan: I'm proud of our work with UL in adding definitions for resin chemistry terms in UL 746E. We all worked hard to ensure we had a base level (of terms and definitions) in place. It's the first step. The next work to be done—which is a lot—is to move into the performance area, which hasn't been mapped out yet.

Goodwin: Those performance criteria should include physical, mechanical, and electrical properties.

Morgan: We've often argued that we should try and curate these within our industry sectors. That makes sense. If we've learned anything, it's that the design process is far from materials. Designers don't understand detail about the material side. Why would they? Even for PCB manufacturing, the designers don't have a comprehensive understanding. Now, the designer can say to the engineer, "I'm a designer for automotive cabins," and the engineer says, "Here's a set of standards that will be appropriate for your kind of designs," or "I design ECUs for under the hood." Now, here are the products that would fit there. It's not the end of the story, but a starting place for these guys. If you can give them a starting place, how much easier the job is.

Goodwin: This happened naturally with thermal management. It was a new thing that came along with thermal modeling so the designers have a much deeper connection to the materials in that sector than they do in any other.

Morgan: I suppose this started with LED lighting as well. Initially, before IMS was used in everything, you had a very well-defined usage area. If you were designing for LED, you were designing the same way, more or less. On the other side, we've gone the other way entirely. I see it clearly and IPC does as well.

There's more of a need than ever to understand the landscape of materials.

Goodwin: Yes, but the risk is generally in over-specifying materials that damage their project economically and all down the supply chain.

Supply chain has changed hugely in the past five or six years, and you can't build a reactive, fast, flexible supply chain for every product. It's impossible. What we have inside Ventec in the overseas business, outside of Asia, are people coming to us who may have specified another supplier on their signal integrity materials, but they can't get them. They are being required to buy large quantities of laminate they do not need, or wait several weeks for delivery.

Mark Goodwin

So, we've done fairly simple things. We've streamed our product line, taken mid low-loss material and a high-performance material, which happen to be our tec-speed 3.0 and tec-speed 6.0, and we've packaged those in bite-sized quantities, cut to size, prepackaged into quantities of 10 or 20. You can get these products in your plant very quickly from point of order. If you're close to one of our distribution centers, they can be ordered today and delivered tomorrow. If you're a little further away, order today, and it will be delivered in one to two days. So, you can have products in your plant loaded and running within one to three days.

Morgan: I am thinking back to IPC and slash sheets. As Mark said, it provides the opportunity that if you do not have all the suppliers but you know it's "slash sheet X or Y," you can go to the suppliers and look at the datasheets. You'll see who meets that requirement. We just need to refresh and repackage it a bit, so when you say "slash sheet X or Y," you know that's a group of performance characteristics, not a group of chemistries.

Do you think the next generation of designers is looking for simplification? I've heard there are about a half-million designers worldwide.

Morgan: That sounds about right but electronics is so big now. The design community is massive. I have talked to designers across the whole gambit—a designer from Tesla, a guy designing Hyperloop, and others designing consumer products. You don't have one design community. You have hundreds, if not thousands, of communities, but they all have the same issue. They do electrical and electronic design. The material side of it is far away from them. That's where we can help.

What would be nice is if we could package this into the IPC-4101 series of standards to give the designers more help, and I think we will. I may not see it in my working lifetime, but we will get there. It's an evolution, not a revolution.

Goodwin: The only thing that matters to the design and manufacturing communities, and even the consumers, is that the products we make meet all the performance and compliance standards. They don't need to know how we build them. We're polymer chemists; let us worry about that. All they need to care about is performance and availability.

Morgan: At IPC APEX EXPO in San Diego, we had a great group, and I felt far more consensus than I've seen before. We were sitting with our competitors, PCB fabs and OEMs, and everybody had a good connection. It was a super meeting, and we were hugely impressed.

What was it about the meeting that made such a difference?

Morgan: First, the structure was different. IPC worked a lot on how the meetings are run. That was important because a roomful of engineers is great, but you must also get things done. IPC certainly has tried, and we saw the benefits of that.

Second, the industry has the same goal now. We understand that this needs to be done together. We have all experienced hugely problematic supply chain issues. We hear the discussion about localizing manufacturing rather than globalizing manufacturing. We're all thinking this way now. Rather than trying to find competitive advantages, we saw that we need to stick together.

Goodwin: Now, we all understand that there must be some thought to supply chain, not just technology because you can have the best technology in the world, but if you can't deliver it at the right time, at the right price, in the right quantity and quality, it's meaningless.

Morgan: Something to think about as well is that this is a young industry still maturing. It began in the 1950s—that's my lifetime.

As our industry matures, what about advanced packaging and the demand on materials there?

Morgan: Packaging is the biggest growth area, and we see a lot coming together in packaging—electrical performance, thermal requirements, and more.

> It's not just about the materials anymore. It is about the supply chain as well.

Goodwin: Our technology direction is really driven by our IP, which is resin chemistry and coating technology; we see less glass fabric in the future. We will be coating films, copper, foils, and very thin products for buildup technologies. Our expertise comes from working on our non-reinforced thermal dielectrics, but we are now applying this to low-loss materials.

Morgan: We had a big discussion recently on this exact topic. We have a range of products and are now working on how we present them to the market. We want to make sure we target this application, that we have the right narrative, and can clearly explain the benefits of using those. We will come back to join the supply chain discussion because you can't have all the products in all the colors and all the sizes, so to speak. There are many products in this space.

We know that packaging is mostly made in Southeast Asia. We must move away from thinking this is just a Southeast Asian story. It isn't anymore. We've yet to see the impact of the CHIPS Act, but we need to be ready.

We have been arguing strongly for the need to run this through the supply chain, and IPC has been working on this as well. They plugged it into the EU just a few months ago to address our concerns to government. IPC's Alison James expertly led the group. We're talking a lot, raising the temperature for the politicians, and the money will flow.

They are finally paying attention.

Morgan: This is for both the U.S. and Europe. The money will flow eventually, so we have to make sure we use that money wisely and build into these advanced packaging technologies.

What advice would you give a young designer? What questions should they ask you?

Morgan: I would say to spend a bit of time with the material supplier, even a half-day early in your career, and ask, "How do I specify material for my design? What more should I consider? What's important? What's available?"

Goodwin: Again, it's not just about the materials anymore. It is about the supply chain as well. How can I get it? Where and when can I get it?

Morgan: The most important question is probably to go to any material supplier and say, "What are your best-selling products in this space?" The brochure is full of products and probably two-thirds can be eliminated once you know the story.

Goodwin: Some of those products in the gaps are there to allow for cost engineering as things go to volume later. But that's not where you should start.

This concept I had with the small packaging/fast delivery came when I was at a rugby match 18 months ago with some UK customers. We were having dinner afterward, and I asked them what they needed. They said, "We need products in small quantities, and we need them faster. That's basically it." I said, "What can you compromise on? The copper weight?" They said, "Make everything a half-ounce; we can plate up to one ounce, we can etch back to 12 microns. But then deliver everything quickly." We asked them if they needed every product, and they responded, "We need something in this and this range." So, Ventec went ahead and did it, and didn't ask for a PO; we just built inventory availability. Now they are ordering from us quite regularly, and other customers are starting to order because we're out there telling them it's available and they don't have to wait. This has little to do with technology, and more to do with paying attention to need.

"Mark and I go back 30 years, promoting standard builds for multilayers. Standards are still very important."

Morgan: Mark and I go back 30 years, promoting standard builds for multilayers. Standards are still very important. Many things have changed on the glass side. We all use thinner fabrics now. There are different standards, but you can still go for a standard design. If a designer can have that kind of standard build, it gives them a starting place.

Goodwin: I'd forgotten that, but it's how Ventec built its business in four, six, and eight multilayers—three cores and two prepregs. It's more complicated now, but we need this kind of creative thinking.

It's the same concept.

Goodwin: Yes, it's the same concept. You need to think a little differently and put those thoughts into action.

Morgan: You need to understand what's available and what will work in your designs. We can help with that.

Goodwin: I asked a couple of customers to send me the stackups for the last 20–30 jobs they've had so I could look at them. I was just looking for patterns. Those patterns are there and can be used to understand and build solutions, and when those patterns are extrapolated over many customers, the solutions improve.

Morgan: If all customers do it the same way, it makes life easier for everybody. It makes supply chain cheaper and more efficient. We should be looking at that for industry efficiency as well. We strive to differentiate from the competition, but it helps to have standardized offerings. OEMs and designers can then go to specific standards or look at the slash sheets specifically for a suite of performance requirements. That's how it has worked in the past, and how it will work again.

As always, gentlemen, it has been a pleasure talking to you both. Thank you for your time and your contributions to the industry.

Goodwin: Thank you. **PCB007**

What Electricity Tells Us About Our Health

The Doctor's In

by Henry Crandall, UNIVERSITY OF UTAH/IPC STUDENT BOARD MEMBER

One of the earliest lessons parents impart to their children is the importance of staying away from electrical outlets. In fact, it's currently at the top of my babyproofing checklist with my first child, and a lesson grounded in our natural inclination to assume that our bodies and electricity are incompatible.

However, the truth is quite the opposite. Our bodies produce electricity. Electrical impulses form the foundation of our nervous system, and electric potential literally keeps our hearts beating. So, contrary to popular opinion, our bodies and electricity are not estranged; they coexist in remarkable harmony. This understanding has pioneered entire fields dedicated to monitoring the electrical activity of our bodies. Here, I'll delve into how electricity is ingeniously harnessed to measure our health.

The Pulse of Life: Electrocardiogram (ECG)

Our hearts are the ultimate electrical powerhouses. One of those fields springing up from the intersection of electricity and human physiology is electrocardiology. Doctors now specialize in understanding the electrical rhythm of our hearts. Their main tool is the electrocardiogram, or ECG. With electrodes connected to an operational amplifier, this simple yet

incredibly powerful tool records the electrical activity of our hearts. By placing electrodes on our skin, this device traces the periodic dance of charged ions in our heart muscles. This dance, known as the cardiac cycle, provides valuable information about our heart's health. Irregularities in this rhythm can hint at a wide range of cardiac conditions, from arrhythmias to heart attacks. With cardiovascular disease as the leading cause of death globally, the ECG is one of the most important tools in all of healthcare.

Impedance Cardiogram (ICG)

If the ECG gives us the rhythm, then an impedance cardiogram (ICG) adds depth to our understanding of heart health. It measures changes in electrical impedance within the chest to determine the heart's stroke volume, cardiac output, and even fluid levels in the body. This non-invasive technique can be a game-changer for diagnosing conditions like heart failure, offering valuable insights into how well the heart functions and the body's fluid balance.

Unmasking Cancer: Electrical Impedance Tomography (EIT)

When it comes to cancer diagnoses, electrical impedance tomography (EIT) is a silent superhero. EIT uses a harmless electrical current to create cross-sectional images of the body's tissues. It can identify abnormalities like tumors by measuring how electrical currents pass through different types of tissues. This technology offers a radiation-free and less invasive alternative to traditional imaging methods like X-rays and CT scans, making it particularly promising for pediatric and breast cancer diagnoses.

Beyond BMI: Body Composition Analysis

Our weight may be a significant indicator of our health, but it's only part of the story. Body composition analysis (BCA) takes the concept of "knowing thyself" to a whole new level. By passing a low-level electrical current through the body, BCA distinguishes between fat, muscle, and other tissues. This allows for a more accurate assessment of health risks associated with body composition, such as obesity-related conditions. It's a valuable tool for those aiming to achieve a healthier, more balanced physique.

Neural Engineering: Monitoring Brain Health

The power of electricity doesn't stop with the heart and body composition. It extends to the command center of our bodies: the brain. Neuroelectrics, a field that combines neuroscience and electrical engineering, is advancing our understanding of brain health and disorders. Electroencephalograms (EEGs) record electrical activity in the brain, helping to diagnose conditions like epilepsy and to monitor brain function during surgeries. Transcranial direct current stimulation (tDCS) uses low currents to modulate brain activity and has shown potential in treating conditions like depression and chronic pain.

In conclusion, the marriage of electricity and medical technology has ushered in a new era of healthcare. From understanding our heart's rhythms to diagnosing cancer and assessing body composition, these electrifying advancements are revolutionizing how we monitor and maintain our health. So, the next time you plug in your device, remember that electricity is not just powering your gadgets; it's also shedding light on your well-being, one electric pulse at a time. Embrace this electrifying era of healthcare and take charge of your health, quite literally. PCB007

Henry Crandall is the IPC Student Board Member. He is a graduate of University of Utah and currently pursuing a PhD in electrical engineering as the Advancing Research in College Scientists Graduate Fellow.

To read past columns, click here.

New Resin Systems Used to Solve Circuit Board Fabrication Issues

Article by Steve Schow, Bob Gosliak, Thomas McCarthy
AGC MULTI MATERIAL AMERICA

Editor's note: This paper was originally presented at IPC APEX EXPO 2023. This excerpt will include the sections on hole fill, planarity, and heavy copper resin considerations. All table and figure numbers from the original paper are preserved in this excerpt.

2. Via Fill

Buried via

Figure 10 shows a typical buried via design and the associated defects caused by CTE material mismatch.

These structures are becoming increasingly common with today's more complex structures. The hole-fill process is also becoming more common as BGA packages become tighter. As package sizes increase, requiring more routing layers, these structures adopt combinations of both blind and buried vias. Due to the increased layers and reduced PTH (plated through-hole) vias, designers are forced to use smaller drill sizes to keep the aspect ratio within fabrication capability. This drives the need for thinner cores. The resulting problem is thin subs are more prone to stretching during planarization.

The initial trials investigated a simple four-layer design containing a PTFE core requiring hole-fill. The fabricator struggled with this

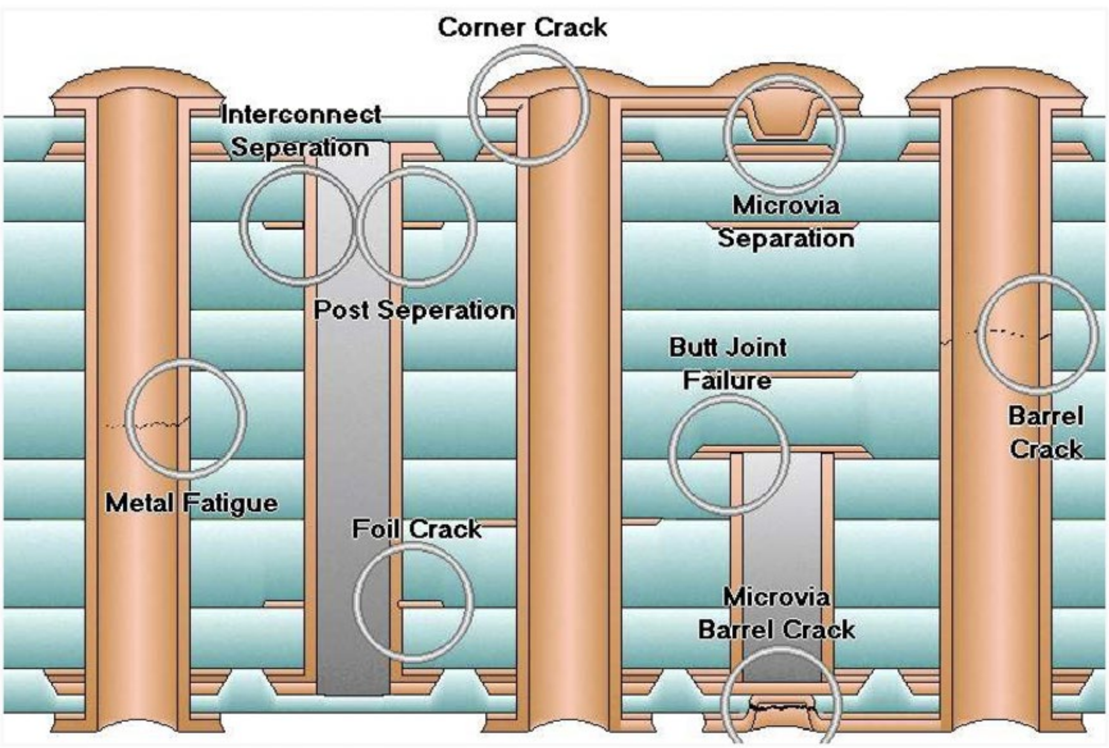


Figure 10: Buried and blind via structures and associated defects. (Courtesy: PWB Interconnect)

design due to the stretching of the subassembly after planarization. The construction was a 10-mil subassembly bonded to a 30-mil subassembly with 5000 x 14 mil vias. The trial was to evaluate simultaneously bonding the two subassemblies and filling all the vias during the lamination process.

Figure 11 shows the flow and fill of the subassemblies using FR-4 as a low-cost test vehicle (no plated through-hole plating). The initial test with FR-4 dummies was to optimize the amount of prepreg needed between the subassemblies to simultaneously bond the subassemblies together and via fill. The typical process of bonding two subassemblies together involves building the subassemblies and filling them with a via fill material prior to lamination. Epoxy via filling leaves small bumps over the surface of the subassembly. These bumps need to be planarized to yield a flat planar copper surface with the via fill only being present in the through-holes. Planarization is an abrasive mechanical process that involves sanding off the surface. Soft or non-reinforced copper-clad subassemblies are prone to mechanically induced irregular stretching. Thin 5- and 10-mil subassemblies would be far more prone to mechanical distortion than a 20- or 30-mil subassembly. The issue is layer-to-layer registration of many subassemblies bonded together where any individual subassembly with unwanted dimensional movement could cause a misregistration problem. It is preferred to planarize a much thicker subassembly that is more resistant to any mechanical distortion. By simultaneously bonding the subassemblies together that are not planarized and via filling simultaneously, the fabricator has the advantage that the newly-built subassembly, based on two individual subassemblies, is a lot thicker and less prone to distortion during the planarization process.

In this example, instead of planarizing the 10- and 30-mil subassemblies individually, the resulting thicker 40-mil subassembly can be planarized with lower risk. Cost, time, and process steps are saved. In Figure 11, the thin prepreg layer consisted of two plys of prepreg which was sufficient to fill all the volume of the mode suppression via holes and still leave roughly a 4-mil dielectric spacing between the subassemblies. Figure 12 shows the same design with the exception that the FR-4 based subassembly was replaced with the desired high frequency material, a ceramic filled PTFE based fiberglass copper-clad laminate. The trial was successful and achieved 100% hole-fill for all 5,000 vias. Figure 13 shows a similar example where the bonding ply material can be used both as a bondply and via hole-fill material.

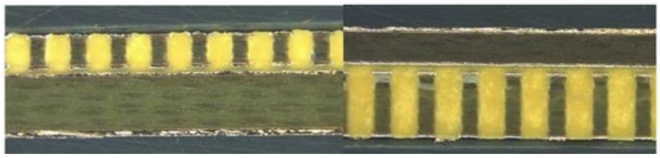


Figure 11: The top 10-mil subassembly was bonded together to the bottom 30-mil subassembly while simultaneously via filling the thousands of through-holes in the individual subassemblies.

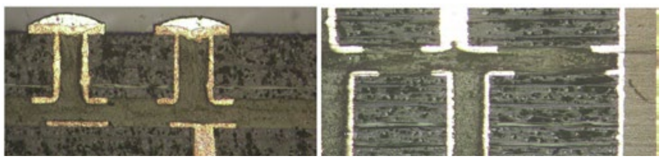


Figure 12: The same design as described in Figure 11 with the exception that fiberglass-reinforced, ceramic filled, PTFE copper-clad laminates were used to create the subassemblies.

Figure 13: A fully buried via with 100% via fill using the bond ply materials to via fill and bond the assembly together (via size 0.0045", via drilled 0.008", 0.0028" deep hole, 13K holes 6"x11" panel).

3. Planar Printed Wiring Boards

As designers are forced to put more functionality into their PWBs, this leads to variation in layer thickness (impedance) copper weights between the power, ground, and signal layers. The addition of multiple functions on a single layer, such as split planes, can cause thickness variation in the printed circuit board. Dielectric thickness variations can cause a number of issues including inconsistent laser via quality, inconsistent back-drilling depth, and inconsistent surface topography/flatness. Surface mount components are increasing in size. Increasing surface mount sizes/larger packages challenge the assembly process. Leading contract manufacturers are increasingly demanding flatter surfaces to ensure these components can be assembled and all the balls in a ball grid array package are planar to the surface of the PWB.

Figure 14 shows that the subassembly layer and the first microvia layer have some amount of thickness variation which is quite typical for a thick subassembly and can measure 3–4 mils. The non-reinforced bonding materials act as leveling agents, but the inner-most microvia layer reflects the dielectric thickness inconsistency present in the subassembly.

Figure 14 shows how the first layer of non-reinforced resin acts like a leveling agent and each subsequent lamination gives a very planar surface. Testing was completed up to 13 laminations producing an extremely flat surface. Figures 15, 16, and 17 show the flatness of the printed wiring board surface that can be achieved with a non-reinforced bonding material that acts as a leveling agent.

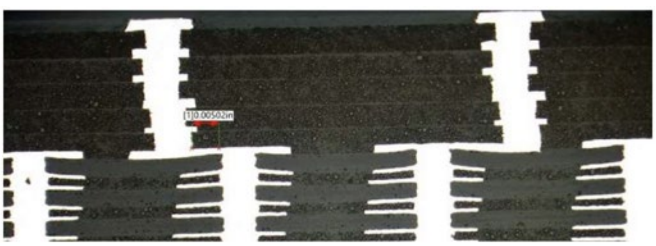


Figure 14: Cross-sectional image showing the thickness variation of the sub and the first microvia layer.

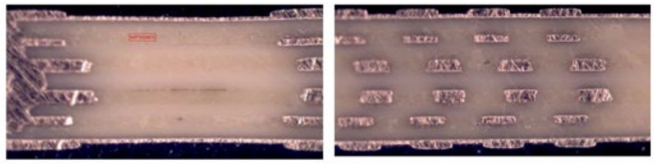


Figure 15: Cross-sectional image showing the flatness of a printed wiring board manufactured using only a non-reinforced bonding material.

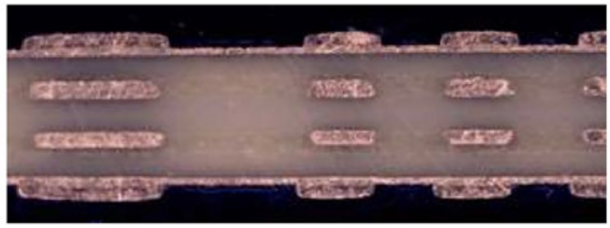
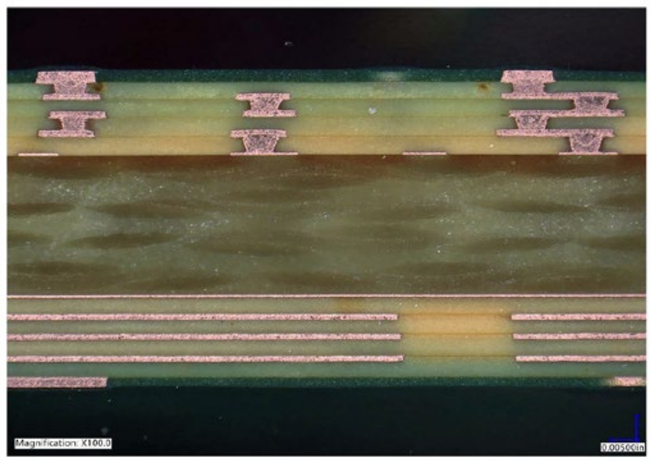


Figure 16: Cross-sectional image showing the flatness of a four-copper layer printed wiring board manufactured using only a non-reinforced bonding material.



Figures 17: Cross-sectional image showing the flatness of a four-copper layer printed wiring board manufactured using only a non-reinforced bonding material.

4. Heavy Copper Resin Flow and Fill

Another challenge facing fabricators is the lack of resin flow in some designs. This can be caused by some of the new ceramic-filled polyphenylene oxide resin systems having a very low flow rate of 10–15%, the high viscosities used to maintain consistency of the ceramic in the resin system, and the use of spread/flattened fiberglass to mitigate electric skew caused by inconsistency in the fiberglass weave. Figure 18 shows various spread weave fiberglass styles. The disadvantage of the spread weave fiberglass is that today's high viscosity ceramic-filled PPO resins do not flow, or have at best limited flow, from one side of the spread weave to the other. The spread weave behaves more like a flat plane of glass. Whatever voids/spaces there may be within the fiberglass X-Y plane do not allow the ceramic or flame-retardant particles to pass through the glass. The high viscosity of the resin system used to maintain a uniform distribution of the ceramic and flame retardant is one more inhibitor of flow from one side of the flattened weave through to the other side. The result is that flow and fill of difficult circuitry can only be achieved by resin on one side of the fiberglass. These materials are challenged to completely flow and fill plated-up HDI or two-ounce copper layers. Multiple plies of prepregs can be used but the trend is toward thinner dielectrics, not thicker layers of prepreg.

Second, because it is well known that the flattened weave-based materials are worse for flow and fill, the tendency is to use prepregs with higher (75–80%) resin contents. While these high resin content prepregs are better suited for heavy copper and plated-up layers (the high resin content can result in higher CTE values) they are more likely to crack because the resin-rich regions lack reinforcement, and will yield worse reliability in HDI stackups.

As designers require thinner dielectric materials, the choice for the fabricator becomes limited and the fabricator is forced to use fiber glass styles like 1067 or 1086 as shown in Figure 18. These glass styles are very tightly woven materials and limit the amount of resin that can flow from one side to the other. This causes issues when heavier copper like two-ounce is used and the fabricator observes voiding issues in low pressure areas. This is especially a problem in very high layer count boards (40+ layers).

One solution is to use non-reinforced prepregs with flow rates greater than 65% and with no barriers to hinder the flow.

The test vehicle used was a planar magnetic, four-ounce, 20-layer design, which provides the worst-case scenario for high and low pressure areas due to the stacking of copper in the coil areas of the design. Due to the very high flow rates, the initial testing containing 70+ sheets of prepreg failed because of the extensive flow and lamination pressure.

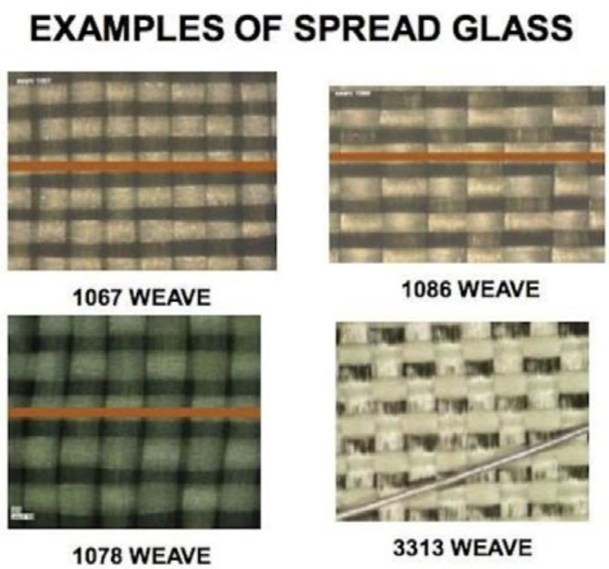


Figure 18: Images of various spread weave fiberglass styles.

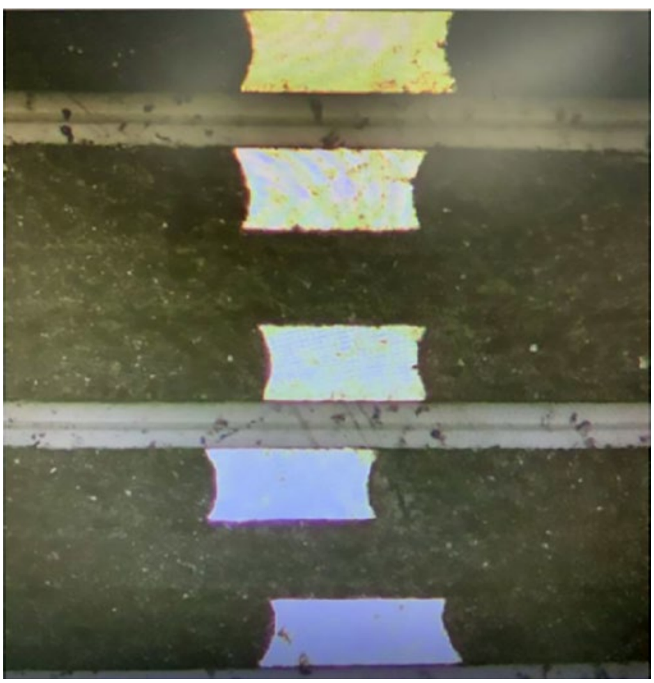


Figure 19: Cross-sectional image showing the pre-filled and final lamination layers.

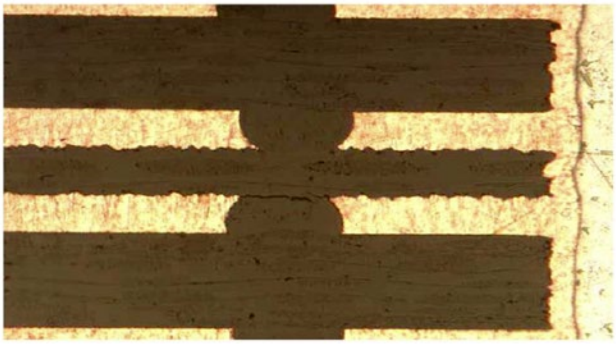


Figure 20: Cross-sectional image showing typical cracking in polyimide heavy copper designs.

In subsequent trials the heavy copper layers were filled using two plys of prepreg on each side of an etched core in a pre-fill lamination step, after which all the pre-filled cores were bonded together with an additional two plies of prepreg.

The secondary reason for selecting this build was that it was well known to have voiding defects in the low-pressure areas and cracking in the resin rich areas due to the CTE mismatch between the polyimide resin, copper, and glass reinforcement.

The Figure 19 photo was taken after six 10x solder shocks at 260°C. There was no evidence of cracking due to the lower modulus of the prepreg and the material having a CTE of 20-22ppm/°C, which closely matches to the CTE of copper (18ppm/°C).

Figure 20 is an image of the design using a full polyimide build of materials:

Conclusion

This testing shows that a low modulus, non-reinforced resin system, with a CTE closely matching copper, has many applications with today's more challenging designs.

Further work is planned, including lower Dk versions and completion of the work to verify pad cratering can perhaps be eliminated.

Summary

All the initial trials have been successful and achieved better than expected results. The non-reinforced material(s) have been shown to successfully function as a thermally reliable build-up material, to fill and flow into mode suppression vias and simultaneously act as both a bond ply and via fill, can be used as a leveling agent to afford very flat surfaces for the attachment of large ball grid array packages, and can be used to flow and fill extremely thick etched copper layers up to four ounces. **PCB007**

Acknowledgements

Robisan Laboratories carried out all simulated reflow testing.

Resources

1. IPC-TM-650 2.6.26 Method A, Test Methods Manual.
2. IPC TM-650-2.6.26B, TM-650-2.6.27A and TM-650 2.6.7.2 Test Methods Manual.
3. "The Keys to 100% Effective Reliability Testing and Failure Analysis of HDI/Microvias," by Kevin Knadle, IPC APEX EXPO 2020.
4. Avishtech, Gauss 2D, Gauss Stack, 5491 Optical Court, Suite 215, San Jose California.
5. "The Next Generation high-Tg products for Telecom and Broadband Applications," by S. Mirshafiei, November 2004.
6. "Highly Accelerated Thermal Shock (HATS) Reliability Testing," by Bob Neves, et al, Proceedings, IPC Printed Circuit Expo, March 2003.
7. "Understanding IST Technology," by Gerard Gavrel, PWB Corp.
8. "IPC Technology Solutions White Paper on Performance-Based Printed Board OEM Acceptance. Via Chain Continuity Reflow Test: The Hidden Reliability Threat—Weak Microvia Interface," by J. Magera, et al, IPC-WP-023, May 2018.
9. "Copper Filled Microvias—The New Hidden Threat," by J. Magera, and J.R. Strickland, IPC APEX EXPO 2019.
10. "Nanovoid Formation at Cu/Cu/Cu Interconnections of Blind Microvias: A Field Study," by T. Bernhard, L. Gregoriades, S. Branagan, L. Stamp, E. Steinhäuser, R. Schulz, F. Brüning, International Symposium on Microelectronics Proceedings 2019.
11. "The formation of nano voids in electroless Cu layers," by T. Bernhard, S. Branagan, MRS Spring conference, 2019, submitted to MRS Advances 2019.
12. "The Effect of Cu Target Pad Roughness on the Growth Mode and Void Formation in Electroless Cu Films," by S. Zarwel, T. Bernhard, E. Steinhäuser, S. Kempa, F. Brüning F, IWLPC Proceedings, 2019.

Electrodeposition of Copper, Part 6

Trouble in Your Tank

by Michael Carano, IPC CONSULTANT

Electrodeposition comes down to fundamentals. In the early days of plating, many users considered the nuances of metallization as black magic. Those days are long gone. Having a thorough understanding of the critical parameters that influence electrodeposition will determine success.

Key Process Considerations

Everything is a process, and the electrodeposition of copper is no exception. Critical to the success of any process is the control of key parameters that influence the plating process. The second consideration is the fundamentals. This month's column will look at some of these fundamentals—and for good reason because much has changed over the past several years. These changes include the technology of the circuit board—smaller vias and thicker boards. Whenever you make a via smaller, and you add more layers (and make the boards thicker), there is an increase in the ohmic resistance through the via. The greater the resistance, the more difficult it is to get an even distribution of plated copper through the vias.

With Ohm's Law, V = IR, you apply a voltage to a plating tank. If you have a 10:1 aspect ratio board and a 20:1 aspect ratio board, which one will be more difficult to plate? It's very simple: the 20:1 board. This is because the resistance increases significantly down through the hole. From a technology standpoint, you must make several adjustments, which the industry has done. We've seen adjustments in tank design, plating rack handle design, process control, and the formulation of new additives that enhance throwing power and overall plating distribution.

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

It's always funny when someone says, "Why can't we just do this?" Well, you're violating Faraday's Law. There are only so many factors you can manipulate, and others you simply can't. There are certain laws, like Faraday's, that you cannot violate. You don't mess with it. Therefore, you must manipulate the amperage, voltage, and other things that make a difference in getting chemistry into the hole.

In addition, cathode current density does matter. To improve plating distribution and surface-to-hole ratio (measured plated thickness in the center of the hole divided by the plated thickness on the surface), reduce the cathodic current density.

There are several tips from a mechanical/electrical standpoint that influence plating distribution and throwing power. For example, I visited a printed circuit board manufacturing facility several years ago. They were struggling with plating distribution, particularly with board designs considered higher density. A good troubleshooter must get to the root cause of the problem. I checked the chemistry. Were the key chemical additives within the process specification? Yes.

So, how to account for the plating distribution issue? Remembering Faraday's Law, we weighed a panel without circuity. The test panel was then plated for 60 minutes at 20 ASF (amps per square foot). After plating and allowing the panel to dry, the test board was weighed again to determine how much actual copper was deposited. The result was that the actual amount of copper deposited was only 78% of the predicted value from Faraday's Law.

What might account for this lack of efficiency in the plating process? We decided to look at the mechanical/electrical aspects of the plating process, and upon inspection of the cables connecting the power supply (rectifier) to the buss bars on the plating cell, we made two observations. First, parts of the cables looked worn and even exhibited some cracks in the cable covering. Second, when we touched the cables, they felt extremely warm, even hot. The concern was that the actual amperage output from the rectifier was not reaching the circuit board at near 100% efficiency. The current was being lost between the power supply and the circuit board. It was suggested that the cables be replaced with coaxial cables that had sufficient cross-sectional area to carry the current with minimal loss.

> "There are certain laws, like Faraday's, that you cannot violate. You don't mess with it."

Once the cables were replaced, efficiency improved significantly. The lesson learned was not to overlook what some consider to be less obvious.

Another key parameter is the ability of the plating rack to sufficiently conduct electricity. There's a similar principle here: If the plating racks cannot carry the current due to improper size or poor rack-to-board connections, current to the board will also be lost. Hence, the expected plated copper thickness will be less than planned.

Finally, I am often asked whether one should use constant voltage or constant current in the electroplating operation. I prefer constant current density which automatically adjusts the output to match the size of the load in the plating cell. PCB007

Michael Carano brings over 40 years of electronics industry experience with special expertise in manufacturing, performance chemicals, metals, semiconductors, medical devices, and advanced packaging. To read past columns, click here.

TOP TEN EDITOR'S PICKS

Paul Bailey, R2-D2 Expert and Former Tech Director for Disney Imagineering, to Keynote IPC APEX EXPO 2024

IPC APEX EXPO 2024 will feature Paul Bailey, mixed-media artist, technologist, vice president and general manager of AOA West, a design, production and project management company in Burbank, Calif, and former principal concept technical director for Walt Disney Imagineering.

DOD Awards $46.2 Million to Revitalize the U.S. Defense Industrial Base On-Shoring of Advanced Packaging and Assembly

The Department of Defense announced an award of $46.2 million to GreenSource Fabrication LLC (GreenSource) via the Defense Production Act Investment (DPAI) Program. The award will enhance existing production capabilities at a manufacturing facility of state-of-the-art integrated circuits (IC) substrate, high-density interconnect (HDI) and ultra-high-density interconnect (UHDI), and advanced packaging.

IPC Chief Economist's Industry Forecast for 2024

To better understand the current economic situation for electronics manufacturing, we brought in Shawn DuBravac, IPC chief economist, to provide an update with a high level global economic outlook. As you might expect, the seas have been a bit turbulent in the aftermath of the pandemic. Shawn breaks down the headwinds and the tailwinds of an economy in flux and what it means for you.

The Big Picture: What Two Hot Wars Could Mean for the Electronics Supply Chain

As we all recall, COVID-19 nearly crippled the electronics supply chain. Many critical chips had multi-year lead times. When we finally thought things were getting back to normal, now we have two hot wars: one in Eastern Europe and now one in the Middle East.


Mehul Davé

The New Chapter: My Review of Happy Holden's 24 Essential Skills for Engineers

Happy Holden has been an inspiration to me since I first met him at IPC APEX EXPO a few years ago. He is the father of the modern-day HDI PCB and former CTO at Foxconn. In his book, 24 Essentials Skills for Engineers, Happy highlights the most important and viable skills for an engineer's success.

Flexible Printed Circuits: A Design Primer

Understanding a package's electrical requirements and not over-designing permits means taking full advantage of a flexible circuit's potential compared to conventional wiring.

Elevating Fabrication: Investing in High-Tech Equipment, Processes, and Labor

Aidan Salvi is the chief transformation officer for Amitron. As such, he's visited fabricators in North America, Europe, and Asia in the past few years, and he's had a ringside seat to the challenges and opportunities facing fabricators around the globe. The I-Connect007 editorial team asked Aidan to share his economic outlook for PCB manufacturing as we head into 2024 and beyond.

Recent Practices in PCB Fab Registration System Architecture

The registration of copper features, both to interconnects and to each other, is a topic about which documented cases in the public space are less than abundant, other than individual supplier marketing with minimal back-up data. Traditionally, this area is known to individual fabs as it involves multiple processes with a lot of variation in equipment makes/models, and even base materials between different shops.

Materials Costs Continue to Improve but Labor Costs Remain a Pain Point for Electronics Manufacturers

Electronics industry sentiment improved during November with demand sentiment also taking a solid step up over the last 30 days per IPC's November 2023 Global Sentiment of the Electronics Supply Chain Report.

Insulectro Announces Passing of Company Sales Legend Neil Colgrove

"The Insulectro family lost one of its founding salesmen this year. Neil Colgrove passed away recently. For those who did not have the pleasure of meeting him, Neil was key to the success of the company as it stands today. He made a large impact on all of us, including Robert Straccia, former vice president of sales, another essential founding partner of Insulectro," stated Don Redfern, Insulectro Chairman.

For the latest news and information, visit PCB007.com

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 l ve of teaching and enthusiasm to help others learn
- Bac ground in electronics manufacturing
- Soldering and/or elect onics/cable assembly experience
- IPC certification a plus, but will certify th right candidate

Benefits

- Ability o operate from home. No required in-office schedule
- Fl xible schedule. Control your own schedule
- IRA etirement matching contributions after one year of service
- raining and certifications provided and maintained by EPTAC

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

Career Opportunities

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

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

Career Opportunities

pluritec®

MACHINES FOR PRINTED CIRCUIT BOARDS

Sales Manager, Remote

Location: North America

Experience: Minimum of 4 years in the PCB industry

Job Description: We are looking for a highly motivated and experienced sales manager to join our team. The ideal candidate will have a minimum of 4 years of experience in the PCB industry and a proven track record of success in sales. The successful candidate will be responsible for developing new business and sales network, maintaining existing accounts, and achieving sales targets. The candidate must be able to work independently, have excellent communication and interpersonal skills, and be willing to travel.

Qualifications:

- Minimum of 4 years of experience in the PCB industry
- Proven track record of success in sales
- Excellent communication and interpersonal skills
- Strong technical process background
- Ability to work independently.
- Willingness to travel

Education: Technical or related field preferred

Compensation: Competitive salary and benefits package

Pluritec develops high end equipment for the printed circuit board (PCB & PCBA) manufacturing industry. We offer a wide range of equipment including drilling and routing, wet processing, spray coating and more. We are a global supplier with more than 3,000 systems installed worldwide.

Contact Nicola Doria
nicola.doria@pluritec.org to apply.

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

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

YOUR JOB AD HERE

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

I-Connect007
GOOD FOR THE INDUSTRY

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

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

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

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

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

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

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

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

SALES MANAGER: **BARB HOCKADAY**
(916) 365-1727; barb@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 © COMEBACK IMAGES**

PCB007 MAGAZINE

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

January 2024, Volume 14, Number 1
PCB007 MAGAZINE is published monthly
by IPC Publishing Group., Inc., dba I-Connect007

ADVERTISER INDEX


Problems solved!
PCB007 MAGAZINE
NOVEMBER 2023
Registration Sweet Spot
PCB007 MAGAZINE
DECEMBER 2023
ECONOMIC HEADWINDS
PCB007 MAGAZINE
Getting to Know Your Designer
DESIGN007 MAGAZINE
NOVEMBER 2023
DESIGN007 MAGAZINE
DECEMBER 2023
ODD GEOMETRIES
DESIGN007 MAGAZINE
What You Need to Know About Manufacturing
NOVEMBER 2023
SMT007 MAGAZINE
DECEMBER 2023
ECONOMIC HEADWINDS
SMT007 MAGAZINE
JANUARY 2024
SUBSCRIBE NOW