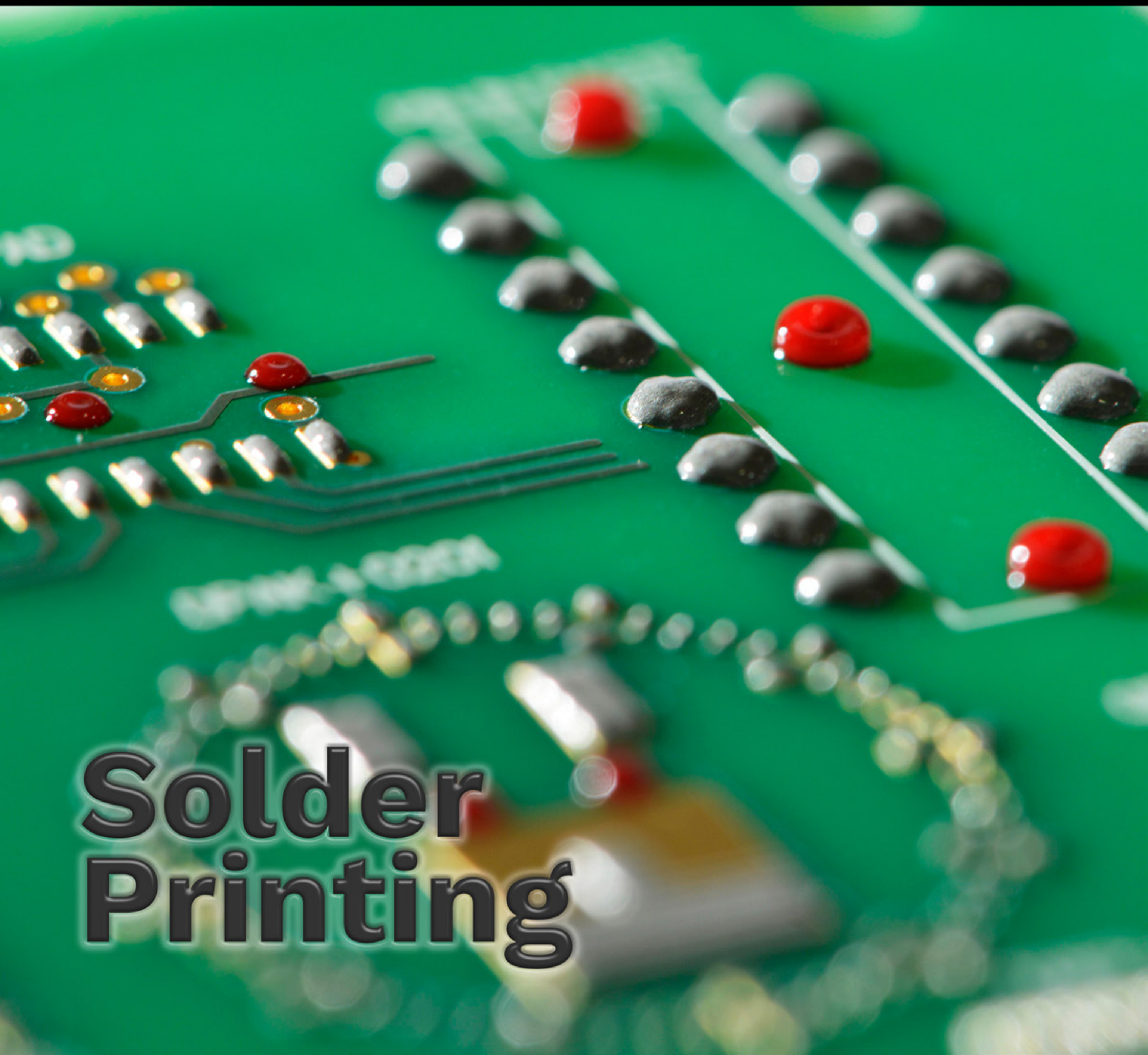
SEPTEMBER 2024

I-Connect007

SMT007 MAGAZINE

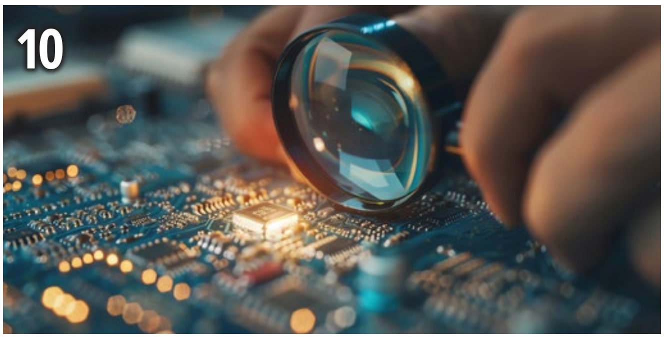


Solder Printing

SMT007 MAGAZINE

Solder Printing

In this issue, we turn a discerning eye to solder paste printing. As apertures shrink, and the requirement for multiple thicknesses of paste on the same board becomes more commonplace, consistently and accurately applying paste becomes ever more challenging. Join us as we look at the latest in solder printing equipment, printable solder pastes, and the opportunities they present.



FEATURE INTERVIEWS

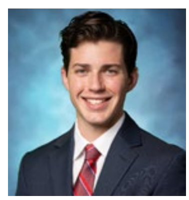
10 **Scrutinizing Solder Printing**
with Adam Murling
and Dr. Ron Lasky

28 **Mycronic's Jet Set Technology**
with Wolfgang Heinecke

44 **Auguring in on Dispensing**
with Sunny Agarwal

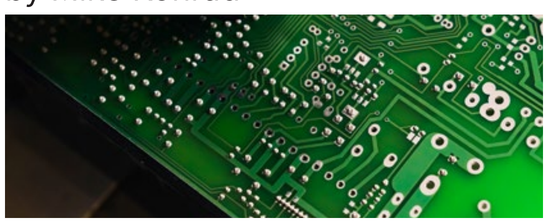
FEATURE ARTICLE

36 **Making Waves With Solder Jet Printing**
by Josh Casper

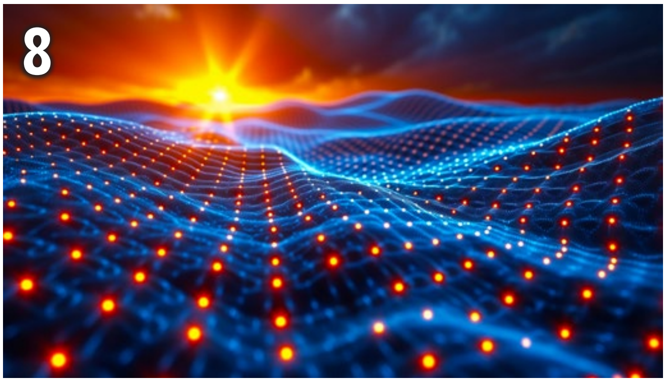


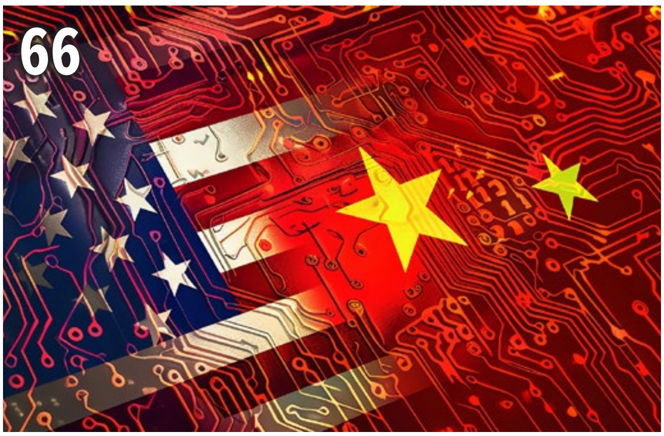
FEATURE COLUMN

16 **The Pivotal Role of Solder Paste**
by Mike Konrad



SMT007 MAGAZINE





SHORT

26 Breakthrough for Next-generation Digital Displays

COLUMNS

8 **Solder Printing: A 1:1 Ratio of Technical and Creative**
by Nolan Johnson

66 **Our Strength Comes From Working Together**
by Tom Yang

ARTICLES

52 **PCB Surface Topography and Copper Balancing Under Large Form Factor**
by Neil Hubble and Gary Brist

72 **Some Fundamentals of Tin-Bismuth Metallurgy and Electromigration**
by Prabjit Singh, Raiyo Aspandiar, Haley Fu, and L. A. Swaminathan

HIGHLIGHTS

42 **MilAero007**

86 **SMT007 Top Ten**

DEPARTMENTS

89 **Career Opportunities**

96 **Educational Resources**

97 **Advertiser Index & Masthead**

Solder Printing: A 1:1 Ratio of Technical and Creative

Nolan's Notes

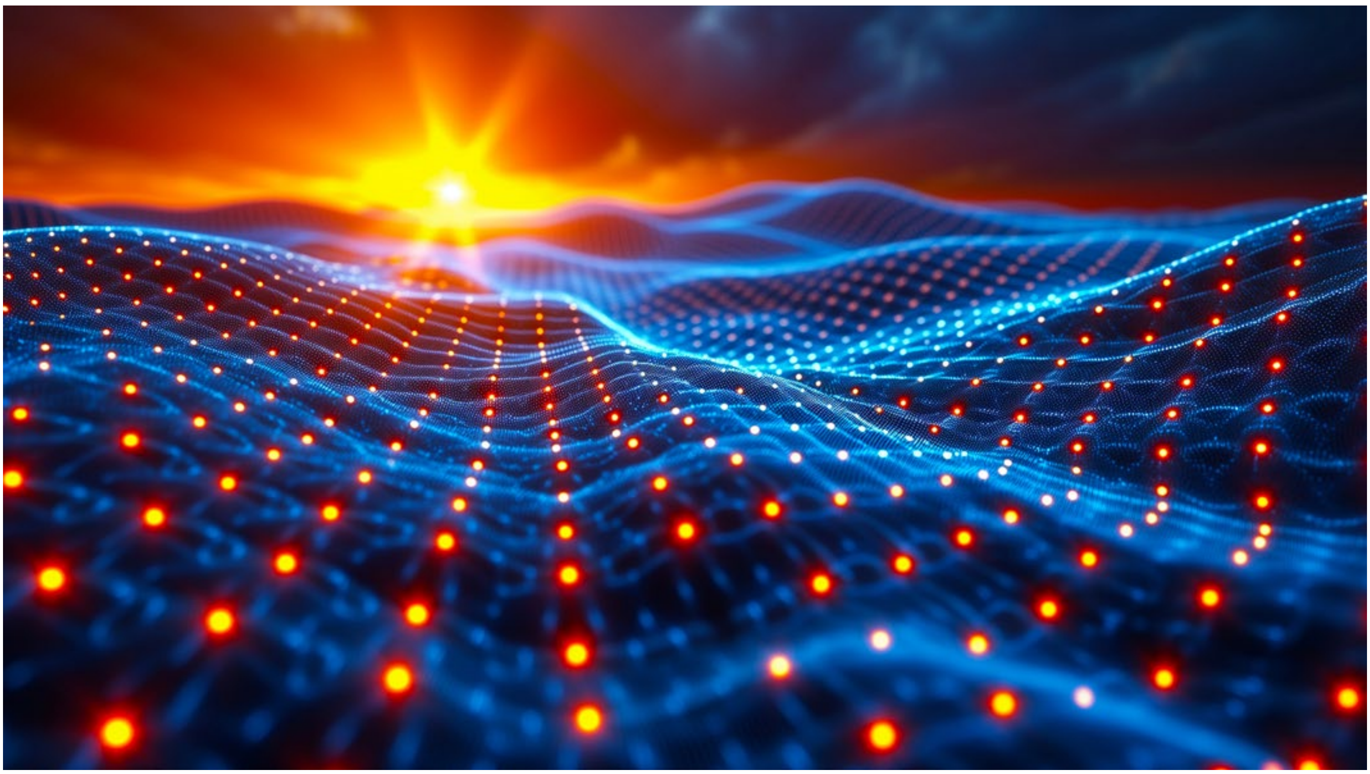
by **Nolan Johnson**, I-CONNECT007

When things get technical, I'm in my happy place. It's why I work in technology in the first place. I also happen to enjoy creative pursuits, and my real happy place is when both creative and technical happen at the same time. Over the years, it seems that most of us engaged in printed circuit design and manufacture are similarly wired. I don't think I'm alone in stating that finding a creative solution to a technical problem is one of the highlights of our workday. We get bonus points if the solution is particularly simple or uncomplicated.

Technological innovation is creative problem-solving at its very core. Straight-up-and-down engineering logic and thinking that turns everything into a flowchart collaborate with creativity's leaping, intuitive, lateral thinking. For me, that's when the magic occurs. Some call it serendipity, which can emerge from some of the most mundane parts of our everyday experiences.

One ubiquitous component in PCB assembly is solder paste. It's what makes assembly happen. We can't attach components without solder, and solder paste is a requirement for surface-mount components. It hardly seems like there's much room for innovation in something so common and basic.

But they probably thought the same thing in 1968, when a 3M researcher named Spencer Silver, who was trying to create a strong adhesive for use in aerospace, inadvertently cre-



ated an adhesive that "stuck lightly to surfaces but didn't bond tightly to them." Meanwhile, Silver's colleague, Art Fry, was frustrated that the paper scrap bookmarks he used to mark passages would often fall out. Fry put his problem-looking-for-a-solution and Silver's solution-looking-for-a-problem together, and Post-It Notes™ was the result.

Truthfully, quite a lot happens when creating and applying solder paste. As components get smaller, the grit of the solder needs to get finer. Likewise, as BGAs and the like get larger, the solder ball gets smaller to make room for more contact points, requiring smaller solder grit. Even when our components get larger, the solder still gets finer. Where it gets complex, though, is that these divergent parts increasingly require different thicknesses of solder paste to bond properly. For stencils, this can be a particular challenge.

Meanwhile, solder print technologies—primarily dispensing and jetting—are gaining acceptance. The tried-and-true method of solder stencils still works, of course. But printhead-based alternatives have made strides. If you've not looked at these pieces of equipment before, now might be the time to do so. It's not lost to me that legend printing, solder mask jet printing, and solder paste printing are all technologies simultaneously on the rise.

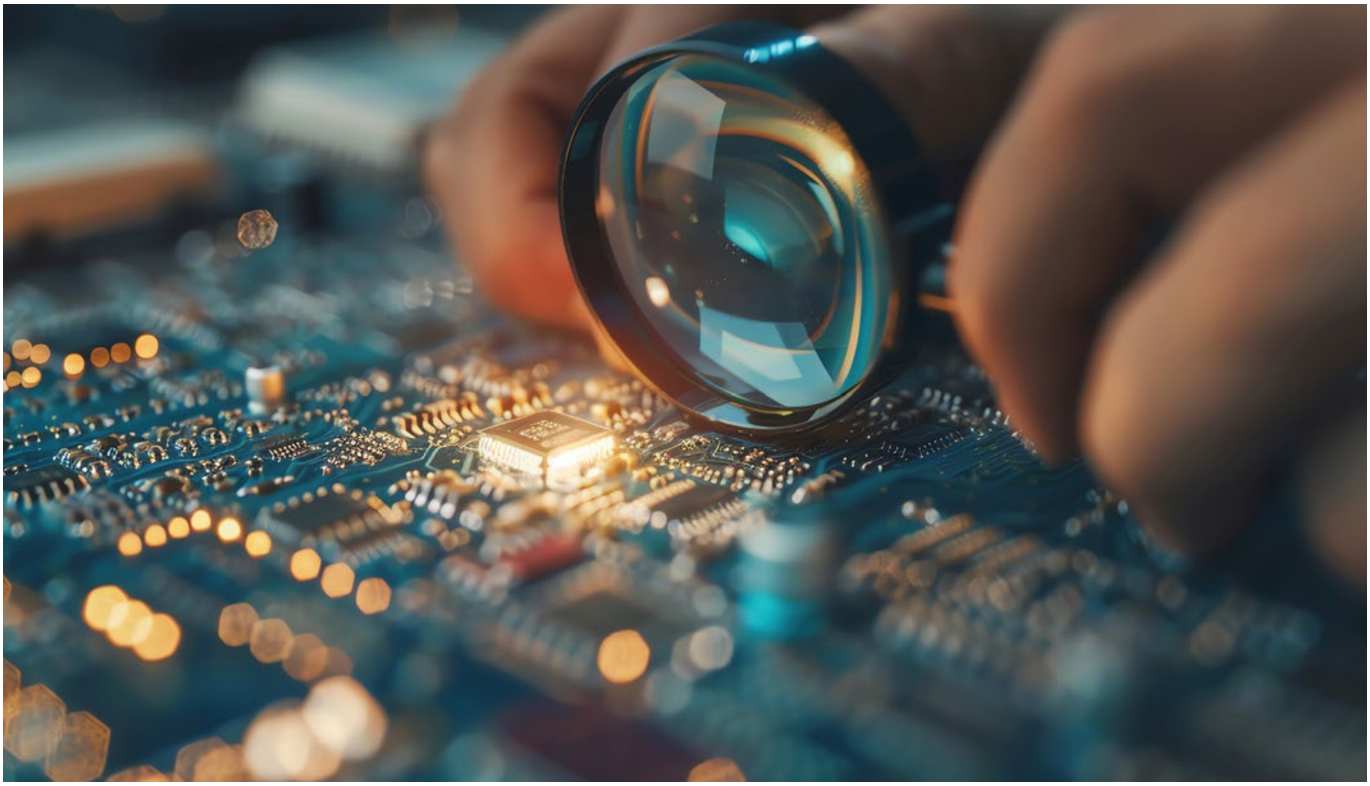
But changing solder paste, or solder paste application, can cause cascading changes down the line. Changes in the flux formulation can require changes in cleaning, reflow, and virtually everything.

So, for this issue, we get down to the nitty-gritty of solder paste. Ron Lasky and Adam Murling kick open the door with a discussion on current solder trends for printing, and Mike Konrad and Tony Lentz follow right behind with an overview of the technologies. Wolfgang Heinecke of Mycronic gets into the details of jetting, and Sunny Agarwal of Camalot Dispensers does the same for dispensing, and Josh Casper debuts in *SMT007 Magazine* with equipment insights.

Neil Hubble and Gary Brist share their most recent research on the increasing importance of surface planarity to the soldering step. Finally, to cap it off this month, we include a paper from Dr. Prabjit (PJ) Singh (IBM, iNEMI) on the electromigration of bismuth in solder joints. After all the conversation in this issue on solder granularity, Singh's work shows what happens to the bismuth over time.

Whether you're a front office executive, line supervisor, operator, engineer, customer service/sales, or an R&D expert, we've got information you can use on solder print in this month's issue. With luck, we'll have helped foster some creative responses to solder print technology for you. I know you'll enjoy it. **SMT007**

Nolan Johnson is managing editor of *SMT007 Magazine.* Nolan brings 30 years of career experience focused almost entirely on electronics design and manufacturing. To contact Johnson, click here.



Scrutinizing Solder Printing

Feature Interview by Nolan Johnson
I-CONNECT007

As members of the technical staff at Indium, Adam Murling, technical manager, and Dr. Ron Lasky, senior technologist and professor at Dartmouth University, know their way around metallurgy and solder formulation. I corralled them for a conversation on solder application techniques from the solderer's perspective and their insights did not disappoint.

Nolan Johnson: Adam, what are some threshold moments when you would use solder jetting over solder printing?

Adam Murling: It can be a tandem approach; it doesn't have to be all-or-nothing. Let's say you have a four- or five-mils-thick (100 to 123 micron) stencil, but you also have really fine-pitch BGAs on the same device. There's an argument to be made for not cutting those stencil apertures out because the area ratio will be too challenging. Essentially, you are printing everything except the fine features, which you jet or dispense.

After printing and inspection, you could have a jet printer or dispenser downstream to take care of those finer deposits. It is important to make sure the solder paste chemistry in your dispensing or jetting equipment is compatible with the paste you're using in a screen printer; compatibility is key.

But what happens when some component leads are too large? You have mostly smaller devices—the 008004, 01005, and even the 0201 passive components and 0.3-mil (75 micron) pitch BGAs, etc.—but you also have connectors that require more solder volume than a three-mil stencil can provide. In these situa-


Adam Murling

tions, you can always step stencil, which has been a practice for a long time; however, it causes some process challenges such as solder deposit uniformity and inspection accuracy.

If you had both pieces of equipment (stencil printing and jetting), you could then do additive manufacturing and essentially jet the paste on top of your three-mil height, which was deposited through the stencil, to get more solder volume before placing the connector in place.

If you factor in additive-manufacturing thinking to solder paste application to get a certain height, does that become a design concern as much as an assembly concern?

Murling: Nolan, everything's heading toward miniaturization. You're having these automotive manufacturers get into the space, and they're not able to find those larger parts that they're comfortable working with anymore. Everyone's focusing on the smaller devices. They're more readily available with the same power output, but they still need those connectors. You will need a step stencil, or you could do the additive approach.

Ron Lasky: Adam, isn't jetting a niche thing? It's not something that 20% of assemblers will do. Or am I wrong?

Murling: It's definitely a solution for specific needs. One place is in some government work, where they only need to build a few boards here and there. Throughput is a key consideration when deciding whether to go with printing or jetting.

Lasky: If you have an application where you will only build, say, five boards, you won't really need to worry that jetting is a slower process. It's very flexible and can do just about anything, but it's very slow. So, if you're only building five boards, then who cares about the time jetting might take? If you're building a million smartphones a month, however, then jetting is just too slow and you will need to use stencil printing.

Is that where jetting wants to be filling a niche or a specific need? Or are the jetting manufacturers trying to move forward into something more production-capable?

Murling: I'd probably defer to the manufacturers for that answer. But jetting is also very useful in R&D labs when they're developing boards and processes, because they're super flexible, as Ron mentioned. So, they're also using the R&D side.

Lasky: In the 1990s, a dispenser manufacturer—which dispenses and gives essentially the same result as jetting—always hoped they could make their process more mainstream to replace stencil printing applications. The great limitation, as we're discussing, is that it's slow, and the curse of manufacturing anything in high volume is a slow process. They've done a lot of work to make the process faster, but they can't compete with printing in most cases.

It is important to point out that there are some jobs that only dispensing or jetting can do, such as underfill for BGAs or CSPs, among others. We don't want to minimize the importance of either jetting or dispensing, but they won't replace common stencil printing processes.

Murling: Jetting is normally faster than dispensing, and you can normally jet smaller and more consistent solder paste deposits.

Is that actually true?

Lasky: If so, why would anybody want to dispense anymore?

Murling: Think of jetting as the next generation of dispensing. Dispensing is more widely understood point-to-point. It is important to realize that when I refer to these processes, it is in the context of dispensing or jetting solder pastes. There are other materials that require this technology, such as underfills, epoxies, adhesives, etc.

As I mentioned earlier, jetting can process a full circuit board; however, it is slower than a printing operation. If you are one of the low-volume manufacturers where a single component could cost thousands of dollars, it may make sense to jet the board as opposed to printing it. This is where the economies of scale come into play. Quality manufacturing equipment is rarely cheap. Make sure you do your research so you are getting the right equipment for your manufacturing.

What needs to be done with solder formulation to support printing vs. dispensing vs. jetting?

Murling: On the formulation side, we definitely need to consider it. On the Mycronic head, for example, there's normally some type of heat applied to lower the overall viscosity so the solder can be shot through the air and deposited appropriately.

Flux is reactive to heat, so it takes careful consideration when exposing solder-based flux to heat for a long time. Developing a flux chemistry that can handle these process challenges is an important design consideration.

Powder size is also a large consideration. The finer the needle or the final orifice for jetting, the finer the powder must be. This is to make sure that the solder is homogenous throughout the entire syringe, as well as being compatible with the other solders you're using on your board. These are all constraints we take into consideration while developing solder pastes.

If you're picking a different formulation for the different dispensing approaches, happen to use different methods when prototyping, and then move into production with a different application method, how much of a consideration is that? Does the design team need to specify it?

Murling: From a reliability perspective, flux normally defaults to IPC-J-STD-004C—SIR and ECM testing are the current industry standard tests to determine reliability of flux residues, whether that's reliability with that solder paste alone or in combination with other materials. This testing is used commonly for no-clean formulas because they are intended to be left on the PCB post-reflow. We do see SIR requests for water-soluble formulas that are intended to be cleaned post-reflow, so this testing really verifies your cleaning process. Really, how clean is clean?

From a metallurgical reliability perspective, though, the user normally dictates the mission profile, which would determine the proper alloy choice. It is important to make sure that the alloy you are using in your dispensing and

> **“Really, how clean is clean?”**

jetting processes aligns with the alloy you are using in other parts of the assembly. The last thing you want to do is create an alloy on the board that is not fully understood. The automotive and aerospace markets dictate higher reliability standards than most others, and because of that, require alloys that perform superior to traditional SAC305.

If it’s that important, does it become a fairly involved conversation between the assembly house and the design team to make sure that they’re lining things up the way they’re intended?

Murling: Yes. If you’re a contract manufacturer, and you have the option to pick, you don’t necessarily think about flux compatibility considerations. You might select a jetting solder paste that’s not compatible with the other chemistries on the same board. You want to make sure that your chemistries are compatible. The solder paste vendors can always tell you the details of their own formulations.

When choosing the right solder formulation and solder flux formulation, how critical is it on the design side?

Murling: We have quite a large technical support team here at Indium Corporation. We can help any customer with whatever information they need, whether it’s printing, dispensing, or jetting.

I want to point out that the Mycronic user interface, as an example, will scan a barcode for the job, and the machine does all the work. You tell it where to put the paste; it knows how much paste, and it knows what pressures to use. It knows the heat required. It knows all the variables inside the machine. So, you’re not tweaking and fine-tuning print speed or pressures like you would on a printer, or the auger speed or time pressure on a dispenser.

Are you talking about the operator on the floor? Somebody has to program the setup for that particular job. The machine has to be looking at the data somewhere.

Murling: That’s one of the values Mycronic brings. Actually, it’s difficult to make a paste that’s Mycronic-approved because the solder paste supplier needs to do a basic round of tests to be qualified. Mycronic then does several tests to develop the information in the barcode. After that, the barcode doesn’t change.

The machine scans the barcode and knows what to do. The only difference is the board that’s programmable. I’ve heard very good things about the ease of use for programming. All that’s needed is to specify where the paste goes. The claim is that an engineer can program a board with thousands of apertures in 10 to 20 minutes. That’s pretty quick.

Given that this application is usually for lower-volume things, that would make a lot of sense.

Murling: For R&D centers for NPI, for whom they may be testing to see if the product works, you put down paste, attach components, and verify. And then when you’re ready for high volume.

Adam, what’s your prognosis for jetting?

Murling: I do see a growth opportunity for jetting, maybe not as a standalone production solution, but as a supplemental application to printing and for the NPI side.

Gents, thank you both for your insights.

Murling: You’re very welcome.

Lasky: My pleasure. SMT007

The Pivotal Role of Solder Paste

The Knowledge Base

Feature Column by Mike Konrad, SMTA

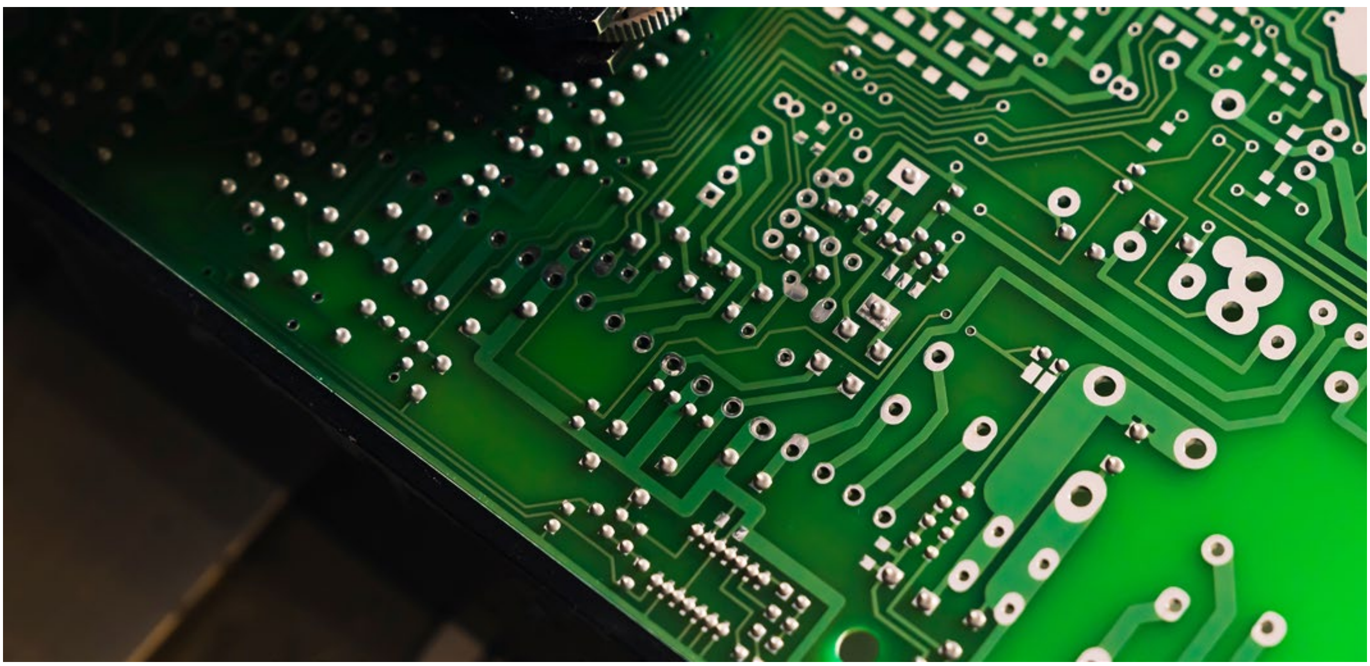
In the complex world of electronics manufacturing, the humble solder paste plays a pivotal role. Often overshadowed by more conspicuous electronics assembly equipment and materials, this blend of powdered metal and flux is the unsung hero that binds the electronic circuits together. But beyond its adhesive and conductive properties, the selection of the right solder paste is crucial for ensuring the long-term reliability of electronic devices.

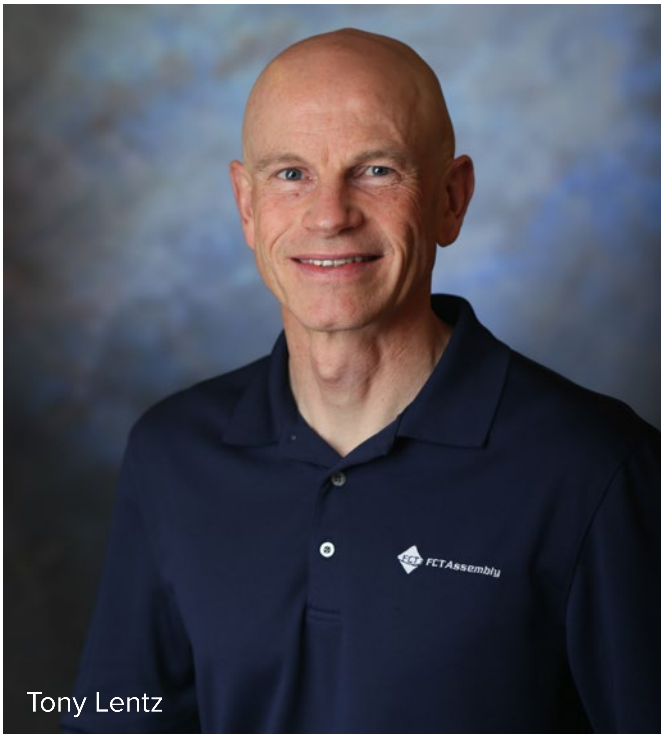
In this article, we delve into the nuances of solder paste selection with insights from Tony Lentz, field application engineer at FCT Assembly and a seasoned specialist in the field of soldering materials. Tony explains why choosing the right solder paste isn't just a technical detail but a strategic decision that can make or break the overall reliability of an electronic product. From ensuring optimal thermal performance to preventing premature failures, the right solder paste is a cornerstone of reliable, high-performance electronics.

As we explore the intricacies of solder paste formulations and their impact on reliability, you'll gain a deeper understanding of why this seemingly modest material deserves your attention—and how it can influence the future of technology.

What are the main factors to consider when selecting a solder paste for a specific application?

Tony Lentz: The first factor is the solder alloy. This is based on the requirements of the PCBA





Tony Lentz

and the intended use. The second factor is the solder powder size, based upon the application and the PCBA requirements: printing, dispensing, jetting, etc. The third factor is the type of flux desired: water-soluble or no-clean, activity level, halide/halogen content, and special properties. The first two factors influence the flux choice, as the flux is formulated to work specifically with certain solder alloys and solder powder sizes in the intended application.

How does the composition of solder paste affect its performance and reliability?

The two major components of solder paste are the flux and the solder powder. Fluxes are formulated to optimize the formation of solder joints with that solder powder. Fluxes are tuned for the application (print, dispense, jet) and reflow conditions (high- or low-temperature soldering, nitrogen atmosphere, laser soldering, etc.)

The solder alloy has a major influence on the reliability of the solder joint. The structural integrity of the solder joint depends upon the solder alloy as well as the other materials of the PCBA. The alloy also determines the reflow temperature and oxidation potential that the flux must accommodate.

What is the difference between a "standard" solder paste and a "high reliability" solder paste?

Standard solder pastes are typically made with common fluxes and alloys like SAC305 and Sn63/Pb37. They perform well in Class 1 and 2 electronics with short to moderate expected life and work in "office" type environments.

High-reliability solder pastes usually are named so due to the solder alloy. High reliability solder alloys are formulated to withstand harsh environments and challenging usage conditions. Automotive under-hood electronics, aerospace electronics, telecommunications equipment, etc., are all used in challenging conditions, and the solder alloy must stand up to these conditions.

"High reliability" may also refer to no-clean fluxes formulated to withstand challenging electrochemical and environmental conditions. Some fluxes enhance solder joint strength. Other fluxes survive high heat and humidity cycling to freezing conditions. Some fluxes are designed to provide increased electrochemical resistance for high voltage or high current applications. No-clean flux technology can be used to enhance the reliability of the PCBA for some applications.

Solder pastes are available in different "types." This generally refers to the solder sphere size. What are the different types and how are these types selected?

The size of the solder powder is chosen based on the size of the smallest stencil aperture for printing applications. The "5-Ball" rule (five of the largest solder balls should be able to fit across the width of the smallest rectangular stencil aperture) is often used as a rough guideline for solder powder size.

It is common for manufacturers to change solder powder size based upon print performance. For example, if Type 4 solder paste

Table 1: Solder powder types and sizes

Type	Mesh	Size > 80% (µm)	Size > 80% (mil)	Smallest Aperture 5-Ball Rule (mil)
2	-200/+325	45 - 75	1.8 - 3.0	15.0
3	-325/+500	25 - 45	1.0 - 1.8	9.0
4	-400/+635	20 - 38	0.8 - 1.5	7.5
5	-500/+800	15 - 25	0.6 - 1.0	5.0
6	-800	5 - 15	0.2 - 0.6	3.0
7	N/A	2 - 11	0.08 - 0.43	2.2

does not print through the smallest stencil apertures, then Type 5 solder paste might be used for that product.

Dispensing and jetting applications require smaller solder powder sizes (Types 5–7) to be consistently deposited in the small volumes required. The size is typically dictated by the provider of the solder paste coupled with the requirements of the equipment.

Can you discuss the importance of flux types in solder pastes and how they influence the soldering process?

Fluxes are the key technology of solder pastes and must work in conjunction with the solder powder, PCB, and components to create a solder joint. Solder powder could not easily create solder joints without flux, and we solder paste suppliers have our own "secret recipes."

Fluxes are classified in J-STD-004 with the following system:

- Material: Rosin (RO), resin (RE), organic (OR), inorganic (IN)
- Activity: Low (L), moderate (M), high (H)
- Halide content: 0 or 1 (≥ 0.05% by weight in the flux residue)

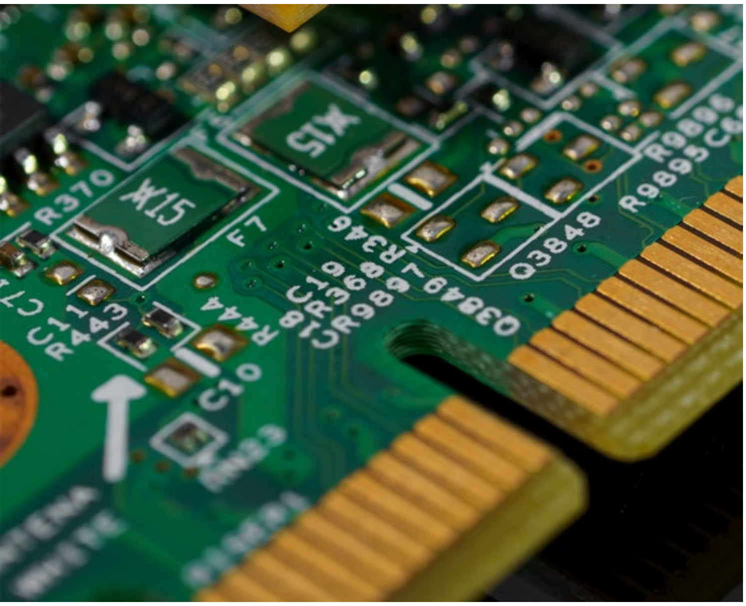
This classification system gives some basic information about the chemistry of the flux and how it may perform in the soldering process.

The flux has several expectations placed on it. Some key expectations include:

- It must be stable/non-reactive with the solder powder that it is mixed with and provide long shelf life, stencil life, open time, etc.
- It must provide rheological properties that make the solder paste suitable for printing, dispensing, jetting, etc.
- It must remove oxides from the metallic surfaces of the solder powder, PCB pad, and component leads, and protect those surfaces from further oxidation during reflow in air.
- It must aid in wetting and flow of the solder to form the solder joint.
- No-clean flux residues must be "safe" to leave on the PCBA and not cause corrosion, dendritic growth, or other electrochemical failures during the life of the PCBA. Water-soluble flux residues must be easy to wash off with water, even under low standoff components.

With so many electronic applications today being placed into harsh environments, to impart to IoT (Internet of Things), is there a difference in the solder paste selection when electronics are deployed into harsh environments?

The solder pastes used for harsh environments typically contain "high reliability" alloys which can survive the environment.



In harsh environment applications, no-clean fluxes are typically encapsulated in conformal coating or potting materials, and the flux must be compatible with those materials.

Some harsh environment applications require flux removal, and the flux must be able to be removed whether no-clean or water-soluble. Some fluxes can be challenging to remove.

When no-clean flux was first introduced, it was uncommon to clean circuit assemblies after reflow. Today, more assemblers are choosing to clean their assemblies after reflow, even when reflowed with no-clean solder pastes. Did this require a change in the flux formulation to allow it to be more easily cleaned?

No-clean fluxes are always evolving to improve performance, including cleanability. Older technology no-clean fluxes used resins and other ingredients that formed a hard shell or a sticky/gummy residue. Dissolving a hard resinous shell or a sticky/gummy residue can be difficult.

Today's fluxes are formulated with softer, more pliable flux residues in part to aid in cleanability, but also to facilitate electrical probe testing through the flux residue. Additives such as surfactants are commonly used for dissolution or suspension of the flux ingredients, but surfactants can also provide better cleanability.

There has been quite a bit of discussion about low-temperature solder pastes. What are the benefits and drawbacks of utilizing low-temperature solder pastes?

Low-temperature Pb-free solder pastes are used to minimize temperature-related defects on the PCBA. BGA components are notorious for warpage which leads to head-on-pillow and non-wet-open defects. These defects can be challenging to troubleshoot as the BGA can have intermittent electrical connections. Lower reflow temperatures minimize warpage and can reduce the rate of warpage related defects.

Low-temperature solder pastes can also reduce temperature-related damage to the PCB and component materials. Flex circuits, CEM (paper based) laminates, and some components can be damaged by standard SAC305 reflow temperatures.

Low-temperature solder pastes are typically made with bismuth/tin or indium/tin alloys. Both bismuth and indium reduce the melting point of tin significantly. High indium alloys can be costly as compared to standard SnAgCu alloys.

A drawback of using bismuth-based low temperature solder paste is the high air reactivity of bismuth. Bismuth/tin alloys oxidize readily in air and need additional protection from the flux to prevent this. The flux formulations are typically more aggressive for bismuth/tin alloys which may lead to shorter shelf and stencil life. High bismuth alloys can also be brittle which leads to solder joint fracturing when drop shock is a risk.

How does the storage and handling of solder paste affect its quality and performance on the production line?

Honestly, I get this type of question more than any other, and I could write a book on this topic. I will try to give a short answer.

Solder paste gives optimal performance a few days after manufacturing, and solder paste performance degrades with age. Storing solder paste sealed in the original packaging in a cooler reduces the reaction rate between the flux and the solder powder and extends the performance life.

Temperatures higher than refrigerated temperature, air exposure, and mixing all accelerate chemical reactions within the solder paste. These reactions decrease the performance of the solder paste to the point of becoming unusable in extreme cases.

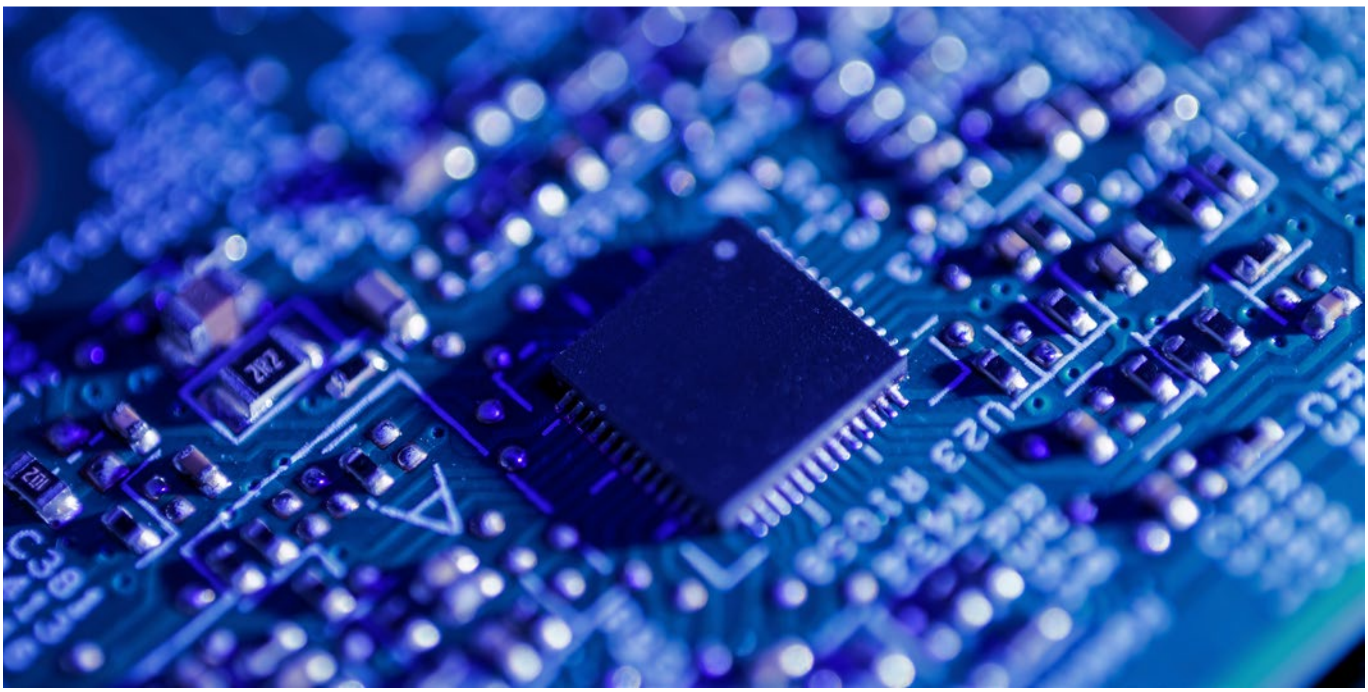
Some solder pastes are more "reactive" than others, which may lead to shorter shelf, stencil, and working life. Proper storage and handling are critical for these solder pastes to extend the performance life.

Could you share some insights on how to evaluate the compatibility of solder pastes with different soldering processes and equipment such as jetting?

Evaluations of solder paste for an application like jetting typically start by using the solder paste in the equipment on a test PCB. This is sometimes called alpha testing. This can be done internally if the solder paste manufacturer has the equipment, or externally at an equipment manufacturer or a user of the equipment. Ideally the candidate solder paste is tested and compared to an existing solder paste that is used in the application. If any incompatibility is found with the solder paste in that application, then alternate solder pastes may be evaluated, or a new solder paste formulated. The alternate or new solder pastes will go through the same initial testing (alpha) in the process/equipment.

Once a solder paste is found to be compatible in a process, then the next stage is to have the solder paste tested at a PCBA manufacturer (beta site). This normally involves using the solder paste on "real" PCBAs. QC inspection of the PCBAs may be more intense than standard production quality control to be sure the candidate solder paste is working well. Again, it is best to compare the performance of the candidate solder paste to an existing solder paste for that process.

The way to measure compatibility of a solder paste with a process/equipment is determined by the application. In the case of jetting, solder paste must function with smaller solder powder sizes like Type 6 or 7. The rheology of the solder paste is key for jetting, and




PREMIER GLOBAL SUPPLIER
to AUTOMOTIVE
FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING
Industry leading Flex Design Support
Flexible Circuits, Rigid Flex, Membrane Switches, Plastic Moldings
Specialized EMS/Assembly Services
Assemblies and Product Module Builds for Automotive
IATF 16949
EXPLORE the POSSIBILITIES!
FLEXIBLE CIRCUIT Technologies®
Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

can be tested using rheometers, viscometers, and the like. Some jetting equipment suppliers are very involved in partnering with solder paste suppliers to optimize a solder paste for use in their jetting equipment. Jetted dot volume, consistency, and repeatability over time are critical and must be tested using the jetting equipment.

What are some common mistakes manufacturers make when selecting solder paste, and how can they be avoided?

Manufacturers tend to buy alternate solder pastes from their current supplier without an evaluation simply due to the need for a different solder paste for a new PCBA or application. For example, if a new PCBA is to be built that includes smaller components than what are typically used, then Type 4 solder paste may not print adequately. Type 5 solder powder may be required, but the solder paste in use may not be optimized for Type 5 solder powder size. Unforeseen issues can occur when using a solder powder that a solder paste was not designed for.

It is also common for manufacturers to try existing solder pastes for alternate applications, which the solder paste was not designed for. For example, a manufacturer uses a solder paste in a standard print and reflow (SMT) application. Then a dispensing machine is brought in to run specialized PCBAs, e.g., prototypes. The natural next step is to use a "dispensing version" of the printable solder paste. Again, unforeseen issues can occur because the solder paste was not formulated for that application.

Another common mistake is for manufacturers to bring in an alternate solder paste with the goal of improving the process, and run it on a random PCBA that happens to be going through the line. The PCBA may not be challenging and may not show any difference in solder paste performance. This typically results in the manufacturer continuing to use the current solder paste and living with the disadvantages of it.

Using an evaluation process is the ideal way to select a solder paste. Solder pastes should be evaluated and compared to existing solder pastes in the same application. If PCBA defects are the reason for acquiring another solder paste, then the evaluation should include testing on the PCBAs that show the defect. Quantitative data should be gathered, and solder pastes ranked according to performance. In some cases, the data is weighted by category to give more value to what is most important to the user, and devalue the less important categories. For example, a solder paste is evaluated for pauses in printing and voiding performance, but pauses in printing rarely occur in the normal production environment. Then the pause in printing score may be weighted lower than voiding performance.

Solder paste evaluations can be quite detailed and creative, including supplier details, shipping time, shelf life remaining when received, stencil life, various printing categories, various reflow categories, flux residue characteristics, and many other desired properties. Evaluations that give quantitative scores help manufacturers to choose the best solder paste for their application.

How do you foresee advancements in solder paste technology affecting the electronics manufacturing industry in the coming years?

Solder paste technology is always progressing, including flux and solder alloy advancements. Flux advancements are targeted at solder paste performance improvements, and

optimization for new solder alloys. Recent flux advancements include:

- Improvements of head-on-pillow and non-wet-opens for BGA soldering
- Optimization for small solder powder sizes (Types 6 and 7) for miniaturized electronics
- Improvement in no-clean flux reliability for more rigorous surface insulation resistance (SIR) and electrochemical migration (ECM) conditions. There is also a shift to completely halide- and halogen-free solder pastes, which may improve reliability.
- Optimization of no-clean solder pastes for high-power applications like electric vehicles

Solder alloy advancements include high-reliability alloys for harsh environments, and low melting alloys. The elements used to reduce the melting point of Sn (tin) based solders are Bi (bismuth) and In (indium), and both elements can improve the "reliability" of Sn-based solders when used as alloy additives. In some cases, the "high reliability" alloys also have a lower melting point than SAC305 and may fit into both categories.

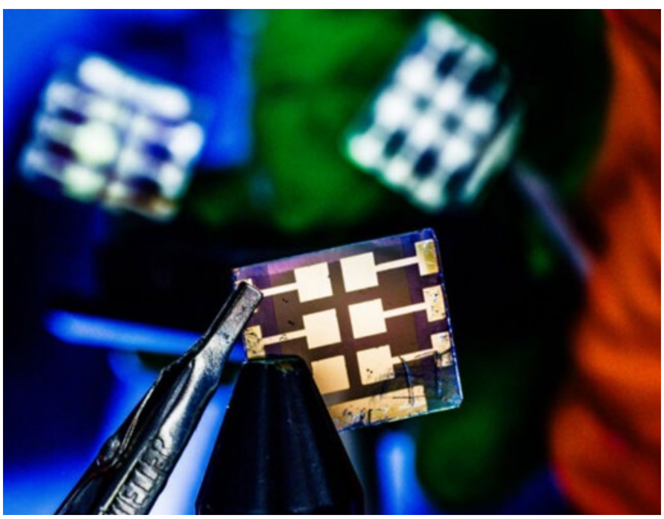
High-reliability alloy development is mainly targeted at the automotive industry as the current automotive alloys have some weaknesses, and the volume of consumption is very high. Low melting alloys are being developed to reduce defects like head-on-pillow and non-wet-opens for BGA soldering and other temperature-related defects. Using lower solder temperatures facilitates the use of less costly laminates like CEM (paper based) and reduces energy consumption and costs. **SMT007**

Mike Konrad is founder and CEO of Aqueous Technologies, and vice president of communications for SMTA. To read past columns, click here.

Breakthrough for Next-generation Digital Displays

Researchers at Linköping University, Sweden, have developed a digital display screen where the LEDs themselves react to touch, light, fingerprints and the user's pulse, among other things.

"We've now shown that our design principle works. Our results show that there is great potential for a new generation of digital displays where new advanced features can be created," says Feng Gao, professor in optoelectronics at Linköping University (LiU).



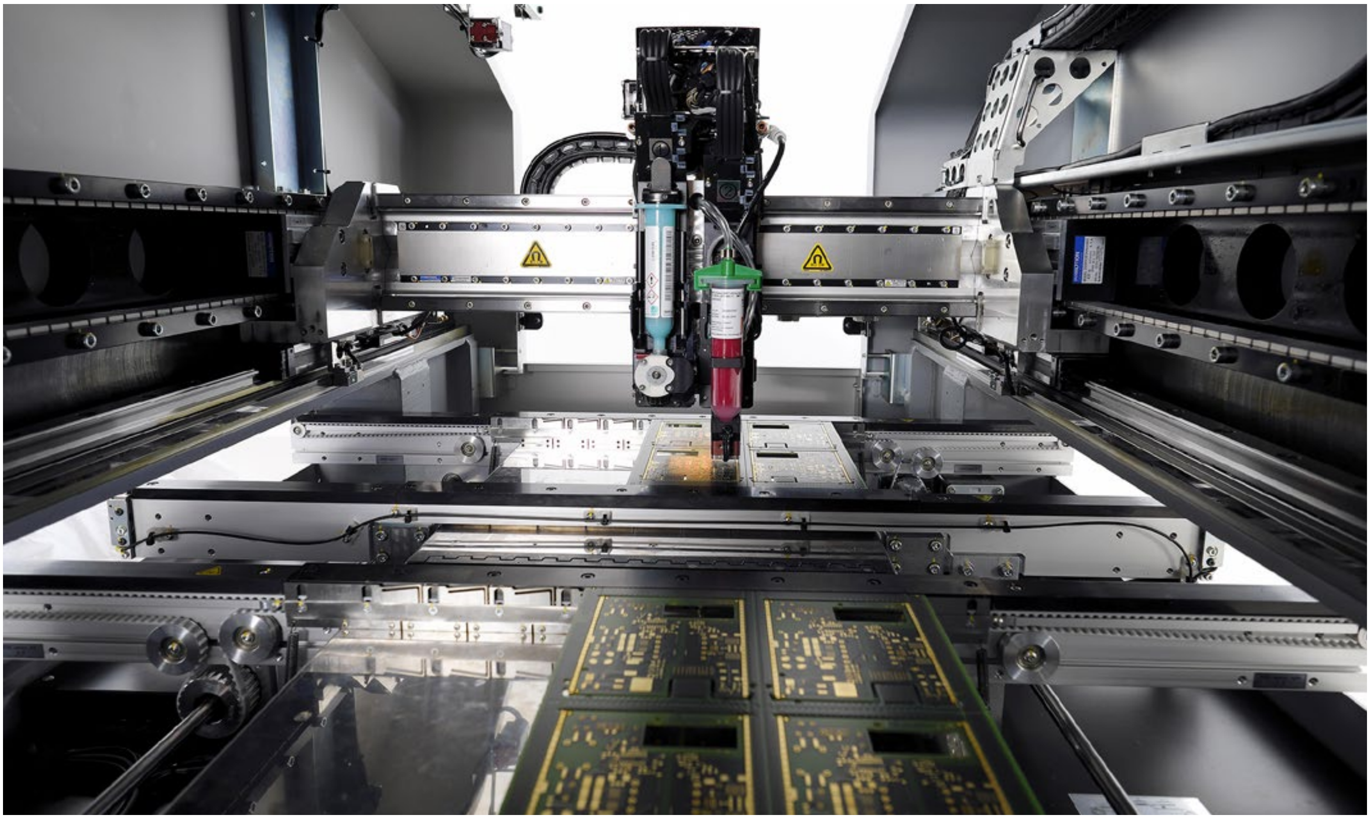
The most modern LCD and OLED screens on the market can only display information. To become a multi-function display that detects touch, fingerprints or changing lighting conditions, a variety of sensors are required that are layered on top of or around the display.

The LEDs are made of a crystalline material called perovskite. Its excellent ability of light absorption and emission is the key that enables the newly developed screen. The device can also be charged through the screen thanks to the perovskites' ability to also act as solar cells.

But many challenges remain before the screen is in everyone's pocket. Zhongcheng Yuan, a researcher at the University of Oxford, previously a postdoc at LiU, and the other lead author of the paper, believes that many of the problems will be solved within ten years.

(Source: Linköping University)


SMTA International
INNOVATION UNVEILED
ELECTRONICS MANUFACTURING CONFERENCE AND EXPOSITION
OCTOBER 20 - 24, 2024
ROSEMONT, ILLINOIS
smtai.org



Mycronic's Jet Printer Technology

Feature Interview by Nolan Johnson
I-CONNECT007

In this interview, Wolfgang Heinecke, head of global product management at Mycronic, discusses advancements and applications of jet printing technology, which offers solutions to the challenges faced by traditional stencil printing. He highlights the key benefits of jet printing, and explains the qualification process for solder paste compatibility as well as the software-driven nature of jet printing, which allows for quick program creation and real-time adjustments.

Nolan Johnson: Wolfgang, let's start with your background.

Wolfgang Heinecke: I started at Siemens in 2000, working in product management and marketing before I moved to SIPLACE sales in France. I joined ASYS Group in 2013; I spent five years in Singapore, in charge of sales and service for ASYS in the rest of Asia, before joining Mycronic in 2018 in Germany as sales director for DACH. About 18 months ago, I joined Mycronic headquarters to lead product management. I have a team of five people, each one in charge of different product lines, including jet printing.

Jet printing is a completely different approach. It avoids all the challenges of stencil printing. Nothing touches the PCB so there's no need for stencils. You don't need alignment between the stencil and PCB. You have the freedom to operate and to bring in the amount of solder paste you need in each place.

The printing challenges are increasing. If you look, for example, into heavy EV boards, you'll find small chip components close to tall con-


Wolfgang Heinecke

nectors. This puts stencil printers through a lot of challenges because you need either step stencils or you need to find a solder paste compromise for the complete PCB. However, with a jet printer, we have the possibility to solve this.

The jet printer itself is built on a high-performing gantry system and a printhead capable of depositing ultra-small droplets: one to two nanoliters, up to 300 times per second (300 Hertz). This means one million dots per hour or even more. This combination makes jet printing technology perfect for random dot dispensing.

In 2005, with the MY500, the goal was to replace stencil printing. We're not there yet fully. Stencil printing has its place when it comes to printing speed—like in a placement machine, the jet printer goes from pad to pad, shooting solder paste deposits on the fly.

Nevertheless, market demand for jet printing is on the rise due to the increasingly challenging characteristics of PCBs. Some large automotive electronics manufacturers are looking into this because their EV boards have bigger component variety. Now they are combining stencil print and jet print to bring all the advantages together.

The need seems to be moving toward surgically applying the solder paste where you need it, to the thicknesses that are needed. Clearly, component trends are driving this need. Components are simultaneously becoming very small, and on the other end of the spectrum, BGAs are starting to move toward 100 millimeters a side with thousands of solder balls underneath.

In the beginning, jet printing was mainly done by small shops and prototypers. If I have a lot of changeovers from one product to another, I always need a different stencil. Now, I'm in a stencil-free environment, so this has a positive effect on the time to market. Many big accounts have a jet printer in their prototyping department, and now we are bringing jet printing to the line to answer the challenges they have in volume production.

How are companies moving from prototyping to production for jetting of solder paste? What has jet printing R&D been doing to reach the throughput numbers that high-volume production would want?

Typically, when discussing an add-on scenario, I have a stencil printer in front to do the easy printing. Then, I can add a jet printer to add paste where needed or to print more complicated structures. That works well.

Meanwhile, Mycronic also has its own stencil printer, MYPro S, and we are working on our own solution. We have placed jet printing at some big customers, working with whatever stencil printer they already had. We are trying to improve the application. For example, some big automotive customers approached us and said, "I have this big power socket, which definitely needs much more paste. But next to it I have 0402 or 0201 components, and this is just not possible, even with the step stencils."

Are you suggesting you could do jetting on top of what you just stencil-printed to create the thicknesses or structures in the solder paste you need?

Correct. We are doing this today. You cannot mix different paste suppliers, but we are in touch with many of our paste suppliers. They even have it in their datasheet where they say, "You can mix this paste together with this paste on one pad, no problem. If you stay within the same paste manufacturer, there's no issue. If you mix suppliers, that's probably not a good idea." We did as well our own test together with the Fraunhofer Institute, showing that this is working perfectly fine.

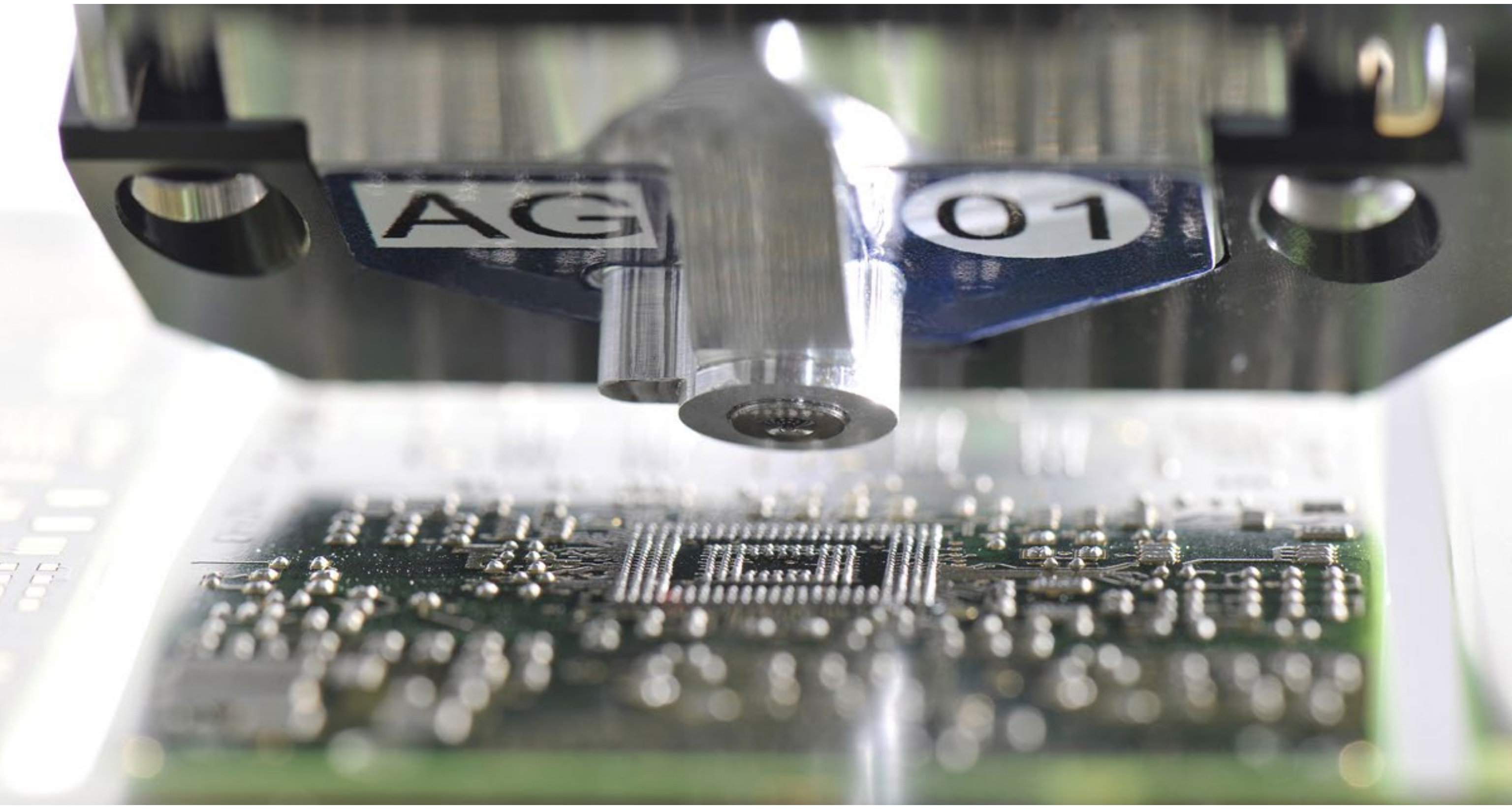
At this point, is the jetting technology—the printheads—stable, or is there still development that needs to be done?

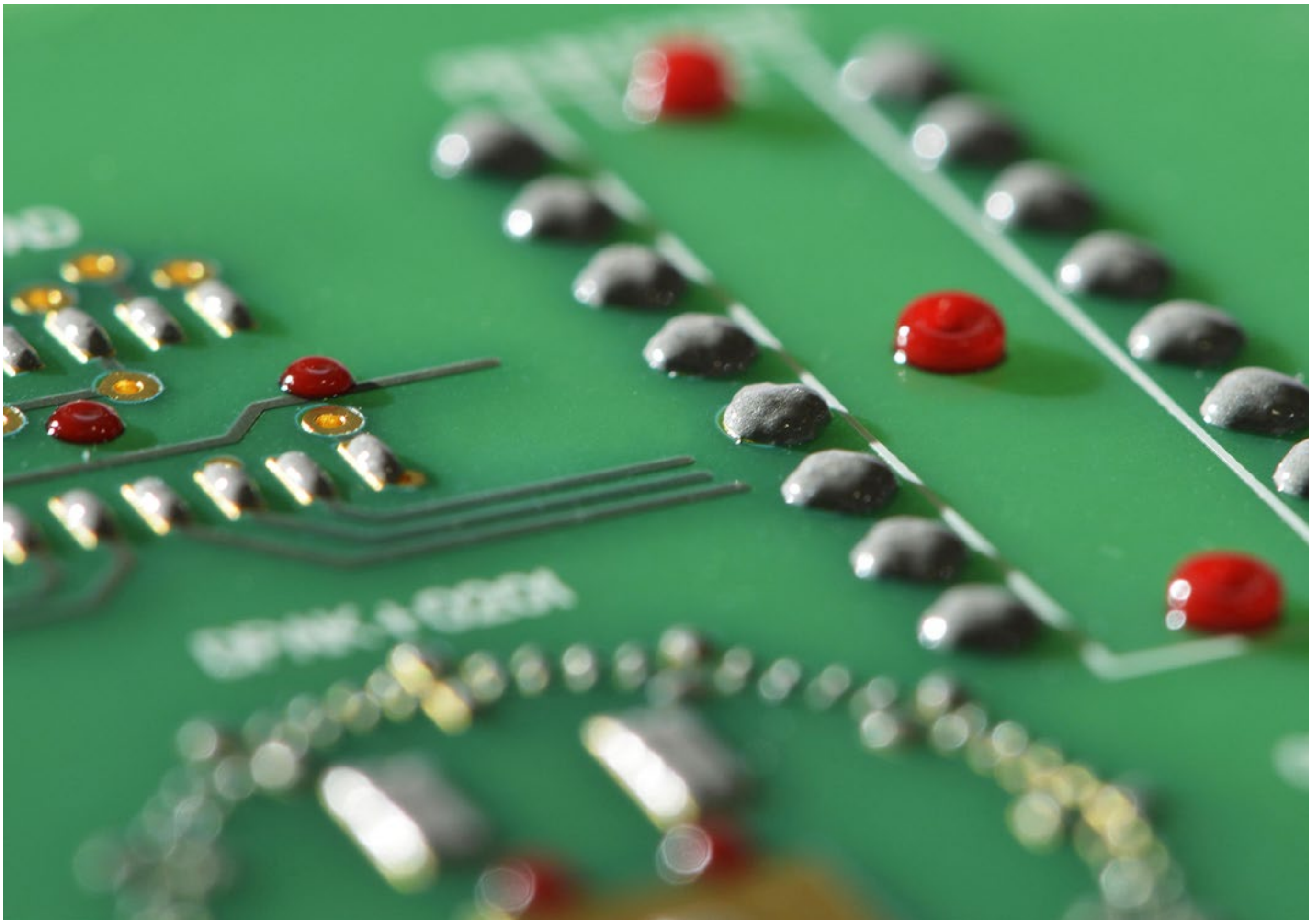
The technology is stable. We have a lot of development still, and if I look back, we have developed new capabilities, including pin-and-paste, printing into cavities, printing on warped boards, package-on-package, low-temperature and water-soluble paste, and fine-pitch components. We can compensate for stretched panels, and we have improved the speed. But we always listen to our customers' demands for smaller dots and higher speed. Faster inspection is a regular request and, recently, we have significantly accelerated the speed of our integrated 2D solder paste inspection feature.

Jet printing is a very software-driven technology. As you can imagine, there's a lot of innovation. It is a stable process in that it's working perfectly fine for many customers around the globe, but there's always development and improvement.

Some of the features you mentioned relate to the printhead, but many would be handled on the software or servo side: Stretch correction and alignments involve coordination between the vision systems and the linear motors.

Production is getting more complex. You have this wider range of components, a mix of large and small, plus what I mentioned about EV automotive. The pads are getting closer to each other. I was talking to some customers who told me they would love to continue to build on 0603 components or 0805, but it's so much more expensive. Their purchasing department is asking them to move to 0201 or smaller components, pushing everyone to miniaturization.





I assume Mycronic is coordinating with the solder paste manufacturers to ensure the paste and machinery are compatible. Tell me about that process.

We have more than 25 qualified solder pastes from different suppliers. You'll find, for example, unleaded, leaded, water-soluble paste on the list, and we are constantly qualifying additional pastes with our solder paste manufacturing partners based on customer request.

To ensure the quality of a solder paste deposit, we create cassette models for all qualified solder pastes. These models can then be used by the customer to achieve the best result. We guarantee that they have a certain range in size and volume of the dots, which the jet printing can be fully and repeatably achieve.

As jet printing becomes more popular, more customers continue requesting additional pastes for their applications that they would like to have qualified as well.

Can you walk me through a typical qualification process?

Imagine you are a customer with a paste that is not yet qualified. You talk to your local contact at Mycronic, who brings this up to the Mycronic team in HQ, where we will get in touch with the paste supplier and work together on the qualification process. This includes the stability over a minimum of three different solder paste manufacturing batches and a broad variety of testing jobs to ensure the complete jetting range in size and volume. A cassette model providing best parameters for the now qualified paste will become part of the jet printer software.

As an alternative, customers can go for quicker "jettable" or "application specific" qualification. For these approaches, we do the qualification just for the specific customer application or a single manufacturing batch of solder paste.

Can you give a rough estimate of how long it takes to qualify?

It depends on the business case, of course. If there are millions of jet printers and tons of solder paste to be sold once you qualify this paste, it goes typically faster. That said, it depends on the availability of three manufacturing batches. Typically, it takes three to 12 months to qualify a new paste. To bring a new paste on the "application specific" or jettable" list, it can be done within 10 days.

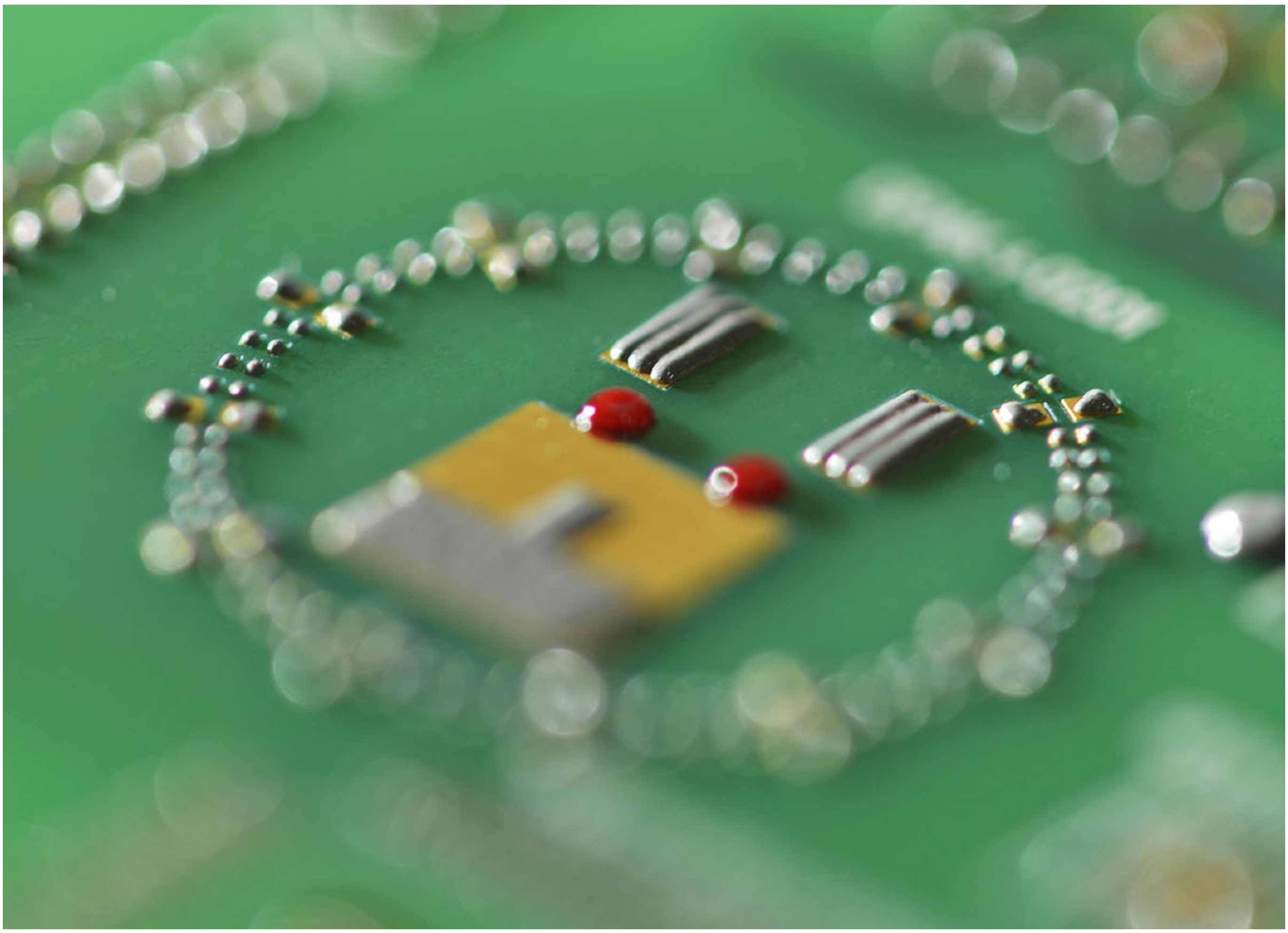
There is a lot going on here. How do you make that complexity more approachable from the user perspective?

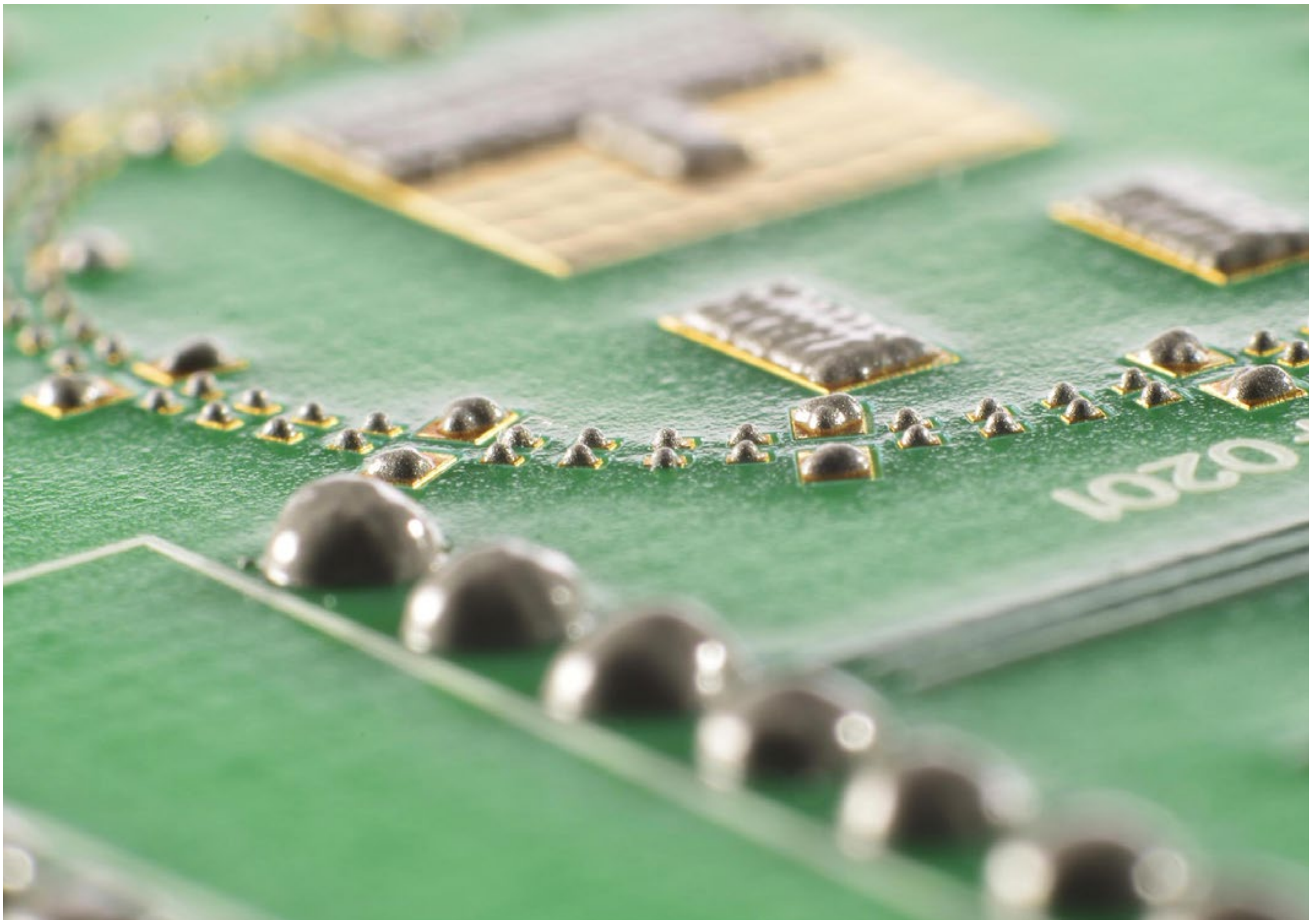
With jet printing technology, you can easily compensate for the limitations of stencil printing technology—even when using step stencils. Jet printing can create deposits with a small volume of solder paste for fine pitch or small components and close to pads for very large components. This is because each solder pad is created independently. I can customize how much paste I want to put at each location. This is one of the sweet spots for our jet printer; I can decide where I want the paste and how much paste I want: any mix, any volume. This is the big advantage to have with a jet printer in that we are not limited by any stencil constraints.

It is really a very accurate machine with all the flexibility you can imagine in the printing process.

To do this, you need to be able to program each board and be very specific about what you put where. Can you tell me about Mycronics' software development for those applications?

The program creation for the jet printer is very swift. It takes about five minutes to complete an ordinary PCB. First, you import the CAD





data, and then the pads and component shapes are automatically generated by the software. If necessary, you can fine-tune how the solder paste print pattern is carried out. Then the program is published. It's basically a digital stencil. To start printing a new batch of circuit boards, you select the print program and the number of boards to produce. Just press start, and that's it.

This process can be further automated when you have CFX and Hermes capabilities, which allow you to get information about transport width, etc. We have been talking for some time about Industry 4.0 capability. The jet printer is the only application that can fulfill Industry 4.0 because otherwise, there is always some hardware to be changed. For example, you need to change the stencil, and there are ways to automate this. A pure software changeover—which Industry 4.0 is aiming for—can only happen with a jet printer.

With the recent release of MYPro Create, our first version of multi-machine programming, we make programming of the machines even easier. Using one program for all Mycronic machines is what we aim for.

It would be easy to imagine a jet printer being connected to the entire line and getting feedback through CFX from inspection, for example, allowing the printer to make slight adjustments to a particular pad or region of the board to resolve inspection data failures.

You can also do board repair and inspection, adding paste only where it's needed. This is another way to use the jet printer to improve the quality of your print.

Thank you, Wolfgang. It's been great to meet you.

Same to you, Nolan. **SMT007**

Making Waves With Solder Paste Jetting

Feature Article by Josh Casper
HORIZON SALES

As electronics shrink and PCB density grows, traditional solder deposition methods such as stencil printing face significant challenges. One solution making substantial waves is solder paste jetting. Solder paste jetting is no secret. In fact, the development and commercialization of this technology has been around since the early 2000s. So, why now? We've arrived at a time in which the maturity of this technology has intersected with new demands of electronics manufacturing. This has opened doors to new frontiers in the world of electronics manufacturing. Here, we'll explore the rise of paste jetting technology: enhancements seen in recent years, unique use cases for this technology, and the business segments in which we are seeing the growth of this technology.

Enhancements in Solder Paste Jetting

When looking at what has changed over recent years, several factors have contributed to the enhancements of solder paste jetting:

Improved valve and nozzle technology

Modern solder paste jetting systems have refined their nozzle and piezoelectric technologies, allowing for finer control over the deposition process. Enhanced materials and refined actuator design allow for more precise control, leading to finer and more accurate paste deposition. Systems are now capable of consistently achieving deposition sizes down to 200 mm.

Enhanced software

Advances in software have given the user more sophisticated control over jetting parameters. These enhancements optimize the jetting process in real time, adjusting for factors like viscosity and environmental conditions to ensure consistent paste placement. Height mapping features can account for PCB warpage or even jetting on raised features or down into cavities.

Increased speed and throughput

While many will point out that solder paste jetting is not as fast as automated stencil printing, paste jetting systems have made significant leaps in recent years through both hardware and software enhancements to drastically enhance throughput. Today's jetting systems can achieve speeds of 700K to 1 million dots per hour (DPH).

Improved and varied paste formulations

Advancements in solder paste formulations have also been a major factor in paving the way for new opportunities in the solder jetting realm. Modern solder paste formulations are designed with optimized viscosity to ensure they flow smoothly through jetting nozzles. This helps achieve consistent deposition without clogging or inconsistent flow. In addition, the use of finer solder particle sizes enables the production of smaller and more precise solder deposits.

Josh Casper

Unique Use Cases

Many of you already understand that solder paste jetting excels in high-mix, low-volume environments. Unlike traditional stencil printing, which requires the creation of a custom stencil for each design iteration, solder paste jetting allows the user complete flexibility to adjust deposition parameters with the touch of a button. While I believe this is an important point to note, I've instead chosen to focus on some of solder paste jetting's lesser-known use cases. In this section, we'll explore unique applications and challenges well suited for solder paste jetting technology.

PCB rework

In repair scenarios, precision is paramount. As electronics continue to shrink and PCBs become ever denser, the popularity of solder paste jetting systems used in PCB rework has grown in recent years. Solder paste jetting allows for accurate deposition of solder exactly where it is needed, streamlining delicate rework on high-density PCBs without the risk of disturbing adjacent components. This level of accuracy is particularly handy when reworking expensive integrated circuits (ICs), in which rework failures are not an option.

Supplemental soldering

Another area quickly gaining traction is supplemental soldering. This is important particularly in cases where varying volumes of solder are needed on particular parts of the board. Traditionally, this was handled by "step stencils," a method in which specific stencil apertures are designed with layers of varying thicknesses to deliver the desired volume of paste. Designing and manufacturing these stencils can be complex and costly and have no guarantee of performing as intended on the first iteration. Solder paste jetting provides a targeted approach, allowing the user to easily add or reduce solder volume via SW adjustments. Supplemental soldering is also used in areas in which pads for a critical device on the board require a much higher level of precision.

Typically, when utilizing solder paste jetting for supplemental purposes, the system is only focused on a handful of areas on the board. This means solder paste jetting is used in conjunction with automated stencil printing, rather than as a full-scale replacement.

Flexible and rigid-flex circuits

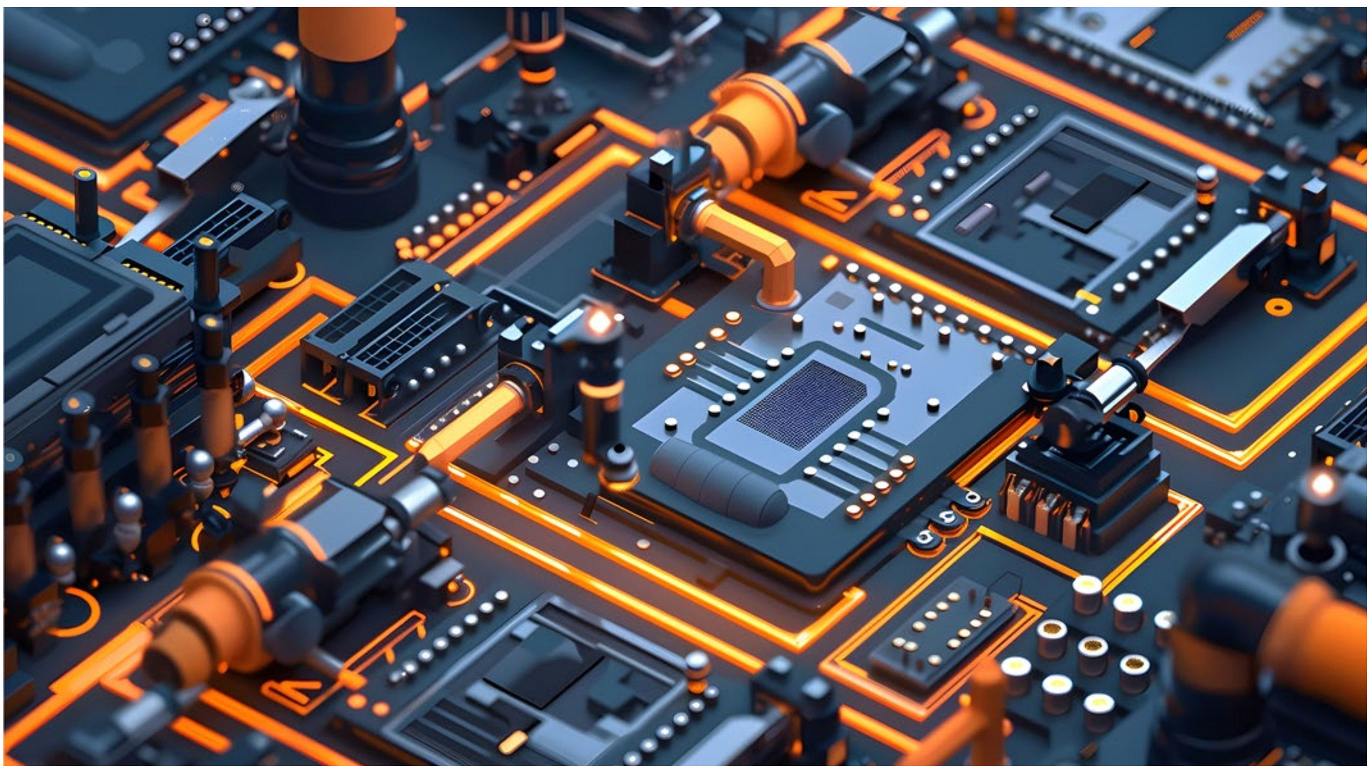
Substrates used in flex and rigid-flex circuits are often delicate and are more prone to damage utilizing traditional soldering methods. Solder paste jetting offers a non-contact application, which reduces the risk of physical damage. In addition, flex material can often be presented in irregular shapes and surfaces; solder paste jetting allows for precise deposition in these scenarios.

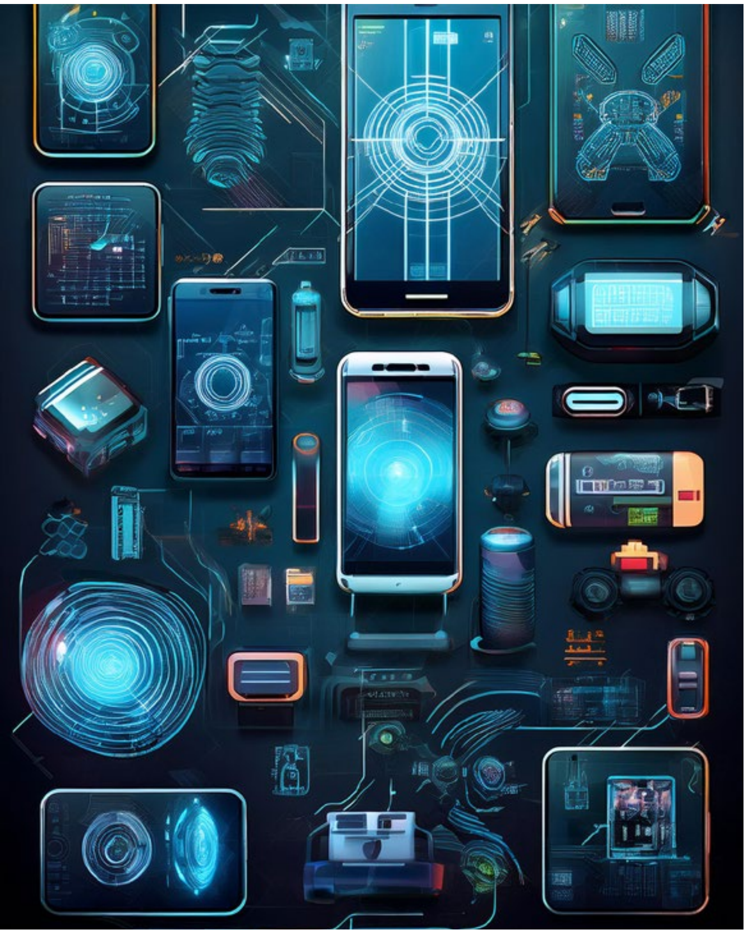
Identifying Growing Markets

As solder paste jetting technology continues to grow in popularity, it's important to identify the areas experiencing growth. In this section, we identify the key sectors within electronics manufacturing that have adopted this technology and their reasons for doing so.

Prototype development and low-volume production

Prototyping and small-batch production, which are perhaps the strongest market for solder paste jetting technology, both rely heavily





on the ability of the manufacturer to be nimble and respond quickly to customer or internal demands. Solder paste jetting allows for a significant reduction in setup time and costs. Engineers can test multiple design variations rapidly without the turnaround time and overhead of producing new stencils for each revision. This not only accelerates the development cycle but also allows for greater experimentation, including solder paste volume and dot size with different component placements and board layouts. The ability to make adjustments on the fly ensures that the final design is optimized before moving into mass production, thus reducing the risk of costly errors.

Medical devices

In medical electronics, where reliability and precision are critical, solder paste jetting provides the necessary accuracy and consistency, especially for complex and high-density boards used in diagnostic and therapeutic equipment. This has been especially true with miniature wearable and implantable devices.

Aerospace and defense

Solder paste jetting technology benefits these industries because it can handle high-density interconnects and demanding reliability standards. Eliminating the need for stencils adds an additional layer of IP protection, as no design files have to leave the building.

Advanced packaging

As electronic packaging becomes more sophisticated, with technologies like 3D ICs and fine-pitch components, solder paste jetting offers the precision needed for effective soldering and bonding of these complex packages.

Consumer electronics

High-end consumer electronics, such as smartphones and wearables, often use supplemental solder paste jetting to manage the high density of components and achieve the accuracy necessary to reliably deposit solder on fine-pitch requirements of modern designs.

Conclusion

Solder paste jetting has advanced enough that it can now be used in a wide variety of unique ways. This technology offers another tool in the assembly process that can regularly solve both unique and routine process challenges. Solder paste jetting will not overtake the raw speed of stencil printing, but its ability to handle the challenges of the finest pitch components, and the vast array of additional applications that it can be applied to, means that this technology will grow in both implementation and acceptance. It is conceivable that most electronics assembly will involve some contribution from solder paste jetting as the industry and technology evolve. **SMT007**

Josh Casper is regional sales manager, west region, Horizon Sales.

MilAero007 Highlights

Northrop Grumman's NG-21 Resupply Mission Successfully Launches to the International Space Station ▶

Northrop Grumman Corporation's Cygnus spacecraft was successfully launched to the International Space Station by SpaceX's Falcon 9 rocket from Space Launch Complex 40 at Cape Canaveral Space Force Station in Florida in support of the company's 21st resupply mission (NG-21) under NASA's Commercial Resupply Service-2 (CRS-2) contract.

US Army Special Operations Orders Two More MH-47G Block II Chinooks from Boeing ▶

In June, U.S. Army Special Operations Aviation Command (USASOAC) awarded Boeing a $115 million contract to produce two more MH-47G Block II Chinook aircraft and begin advanced procurement on future helicopters. Including the new order, the Army has contracted for a total of 46 MH-47G Block II aircraft.

Lockheed Martin to Acquire Terran Orbital ▶

Lockheed Martin announced the signing of a definitive agreement to acquire Terran Orbital, a global leader of satellite-based solutions primarily supporting the aerospace and defense industries.

UHDI Fundamentals: UHDI Applications for Aerospace ▶

Ultra high density interconnect (UHDI) technology refers to advanced manufacturing processes used to create extremely compact and highly efficient electronic circuits at the sub-1-mil line and space level. In aerospace applications, UHDI is crucial due to the stringent requirements for weight, reliability, and performance in a challenging environment. Here are some specific aerospace applications of UHDI.

BAE Systems Completes Testing, Ships Primary Instrument for Roman Space Telescope ▶

BAE Systems has successfully shipped the Nancy Grace Roman Space Telescope's Wide Field Instrument (WFI) to NASA's Goddard Space Flight Center in Greenbelt, Maryland, signaling the completion of integration and testing of the state-of-the-art instrument.

Archer Delivers First Midnight Aircraft to The United States Air Force ▶

Archer Aviation Inc. announced it has delivered its first Midnight aircraft to the USAF to evaluate as part of its AFWERX Agility Prime contract valued at up to $142 million.

Menlo Microsystems Awarded Defense Innovation Unit Contract to Develop Advanced Circuit Breakers for Defense Systems ▶

Menlo Microsystems Inc. (Menlo Micro), the company responsible for bringing its Ideal Switch® technology to market, has been awarded a contract by the Department of Defense's (DoD) Defense Innovation Unit (DIU). Menlo Micro will develop a first of its kind, high-current advanced circuit breaker for power distribution for the Naval Nuclear Propulsion Program's Advanced Technology Innovation Pipeline (ATIP) Program.

DESIGN TIPS #124:

ETCH COMPENSATION

What is minimum space and trace? The answer depends on the starting copper weight.

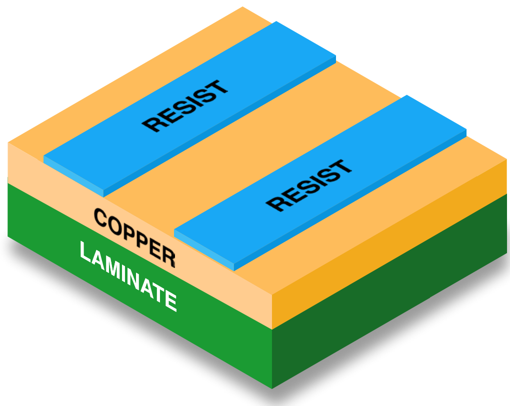
This is because we must do an etch comp on the traces in CAM to compensate for known etch loss. The space between traces after compensation will play a role in whether a board can be manufactured.

The lower the spacing width, the higher the cost. Designers don't always account for the proper starting copper weight after edge compensation.

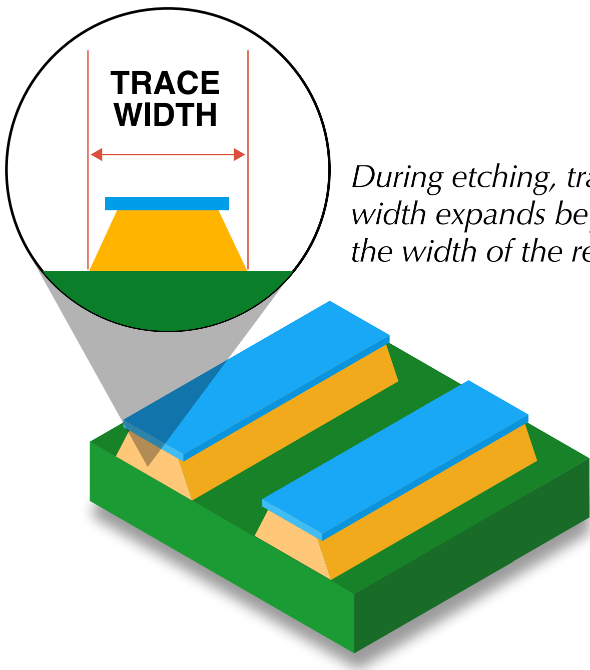
Design tips:

- For accurate starting copper weight, **add a half mil (.0005") to all copper features**.
- **Start with 3/8 or 1/4 oz. foil**, reducing etch comp and less likely to cause a spacing issue.
- **Boards that call for full body electrolytic gold are not comped** to avoid gold slivers occurring during the etching process.

Before etching

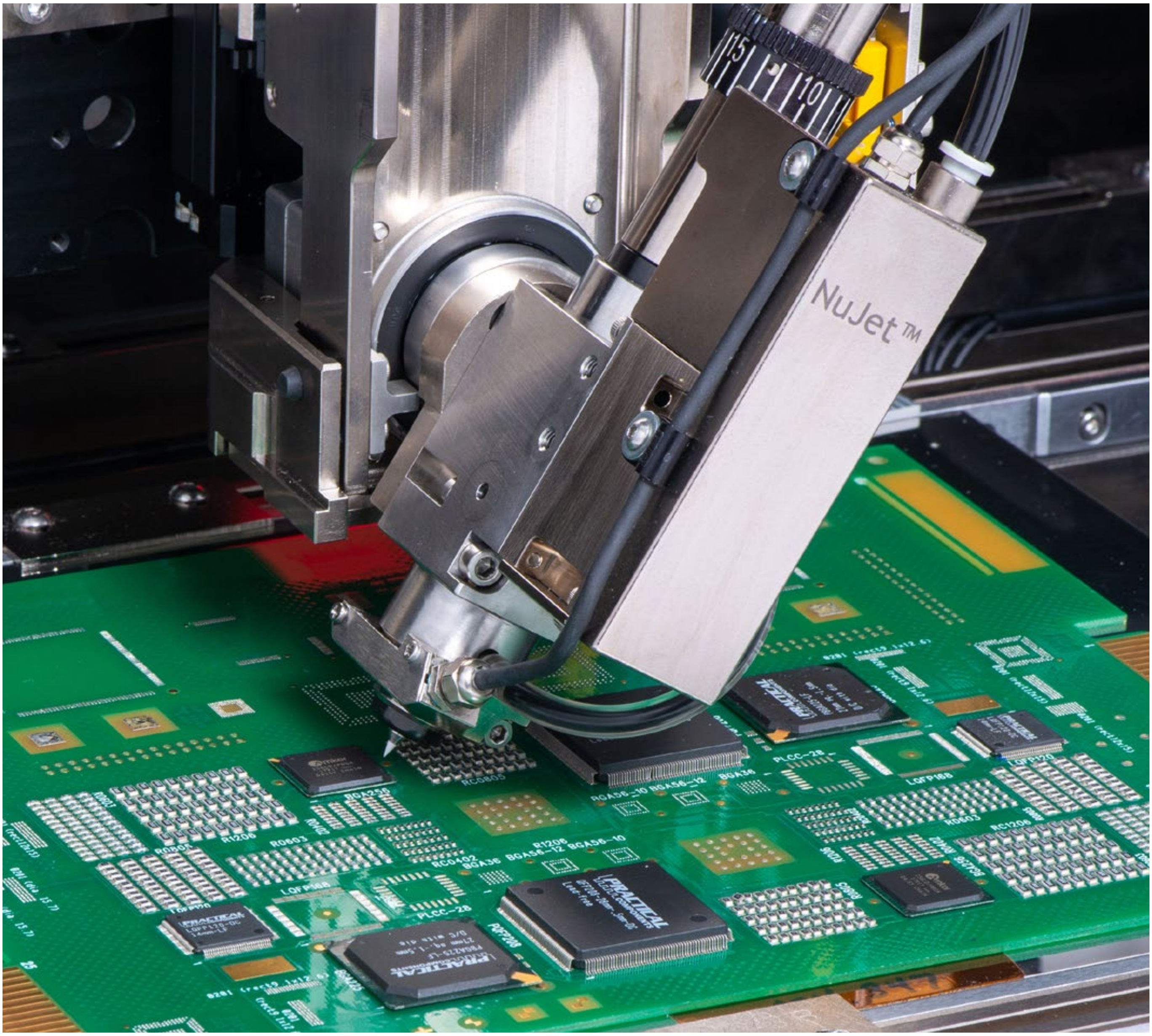


After etching



During etching, trace width expands beyond the width of the resist.

Prototron Circuits
America's Board Source
prototron.com (520) 745-8515



Augering in on Dispensing

Feature Interview by Nolan Johnson
I-CONNECT007

Dispensing technology has come a long way since the 1970s, especially with the advent of jet dispensing in the mid-1990s. Traditional methods involved contact dispensing, where a needle touched the substrate to deposit material. Jet dispensing, however, allows dispensing from a distance, improving precision and speed.

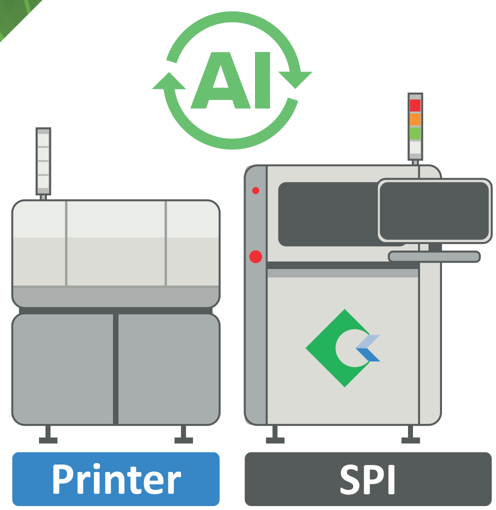
In this interview with Sunny Agarwal, a senior applications and process engineer at Camalot Dispensers R&D, we talk about the choice between traditional methods and jet dispensing. Sunny says it depends on material properties, especially viscosity. Dispensing is very specific to application requirements, Koh Young Technology

SOLDER PASTE INSPECTION

THE WORLD'S LEADING MEASUREMENT ACCURACY

KPO Printer uses a proprietary A.I. engine developed in the U.S. to help machines solve print process challenges. The software exercises complex algorithms to diagnose issues, recommend parameters, and improve print quality.

KPO Printer



ADVISE • DIAGNOSE • OPTIMIZE

Using real-time PCB warp compensation and foreign material inspection across the entire PCB, the 8030 Series, aSPIre3, and Meister-S deliver optimal performance across a variety of printed circuit board and semiconductor applications. These SPI solutions leverage our AI Platform for process optimization with the award-winning KPO for real-time print process optimization. Together, the Koh Young SPI solutions deliver higher standards in metrology-level True 3D™ SPI.

kohyoungamerica.com | america@kohyoung.com |

and excels in scenarios involving tall components, where precise droplet placement is crucial.

Nolan Johnson: Sunny, let's start with a description of Camalot and your role there.

Sunny Agarwal: The company was founded in 1987 and made its initial foray into electronics assembly through the design of an automated liquid dispensing system for an aerospace application. We created a legacy of innovation and a solid reputation of producing high-quality, durable products, and have progressed with a series of high-speed dispensing systems. Innovation has continued around the dispensing technology.

We work on the CBI model—customer-based innovation—so we understand the customer's pain points and build a product to address those pain points. I joined Camalot in 2013 as a research assistant, and now I'm a senior applications engineer. I'm highly involved in the product development of a high-speed fluid dispensing system for PCB assembly applications. I also guide regional process engineering teams located in Asia, Europe, and Mexico on full scale project implementation from in-house process development and testing for new hardware capability assessment to onsite new product evaluation, followed by post-sale operations support.

We provide application support to existing and new customers to resolve their application needs. If they have a new process requirement, they come to us. We provide them with the correct configuration for the machine they need to do all that setup in a PCB manufacturing environment.

Thank you. Now, what is dispensing?

Dispensing covers a very broad spectrum as it is material driven. It has been around since the 1970s and has progressed a lot since then, especially from the mid-1990s, when jet dispensing came into existence.

Before jet dispensing, everything was manual on a benchtop system. It was a traditional fluid dispensing method. It could be like a time pressure dispensing, a rotary auger, or a positive displacement dispensing. But in the mid-1990s, when jet dispensing came into existence, everything changed; all these traditional fluid dispensing methods were contact dispensing. That means the dispense valve is coming down very close to the substrate with a needle and depositing the material. You're within microns of the substrate.

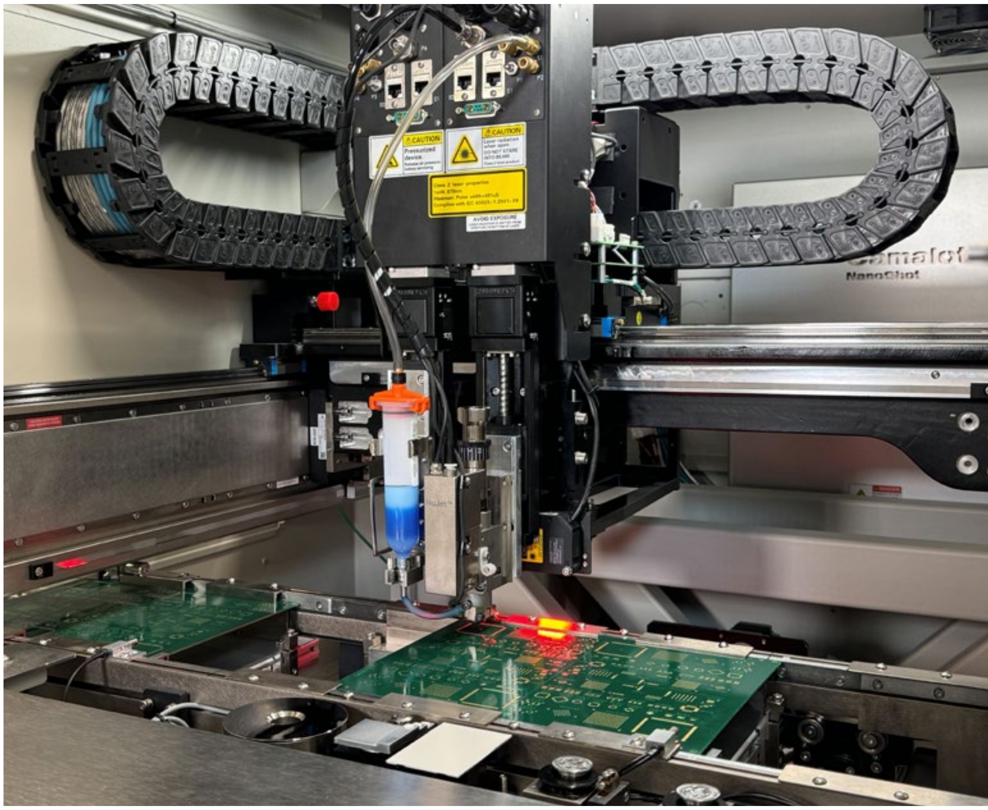
Jet dispensing dispenses more of an individual droplet, meaning the dispense valve can stay off the substrate at a certain height—two millimeters, say, depending on the requirements—and stay at that height. There is no Z-axis motion, so it was faster and more precise.

Some people still prefer traditional fluid dispensing methods. Everything is material dependent. Dispensing can range from something that's water-like to something sand-like. If you compare it to water where viscosity is around 1 centipoise at 25°C, an equivalent material would be something like a non-filled underfill, which has no filler particles in it. It has a viscosity of 300 centipoise.

It's not water, but it's very low viscosity. The other end of the spectrum could be something like thermal interface material (TIM) that is 500,000 centipoise. You cannot jet a droplet of TIM from a jetting dispense valve.

Some like the traditional methods of rotary auger pumps, or a time pressure to dispense that kind of material. One example is a thermal interface application where the material goes over the heat sink to dissipate all the heat out of the product. This is why dispensing continues.

Plus, things have improved on both the machine side (which has the gantry driven by the motion controller) and on the dispensing side, thanks to a jet or an auger valve. Both have progressed to bind up the technology and make sure we provide a dispensing solution.



Camalot Prodigy machine inside.

As you move the heads around, are you using linear motors?

Yes, and of course, there are chunks of technology to accompany the linear motor technology. It's not just a linear motor; it's a motion controller, a gantry drive, and a vision system. Everything must be considered and tied together to build a very high-precision system. There's still a lot of work to do regarding the electromechanical part because, for example, the shorter you can make the settle time, the better your throughput.

So, where does dispensing shine?

Dispensing shines in applications with tall components. That's where the jetting comes into play because you can dispense a droplet from a certain height, making sure you're not hitting or damaging the components.

Inkjet technology has not been used much because it is very slow and can only handle fluids up to a certain viscosity. It can mostly handle conductive inks and dielectric materials that are very low viscosity. At the other end of the spectrum, a stencil printer is best suited for high-viscosity materials like a solder paste type 4 or 5. The surface mount adhesive glue is another high-viscosity material. There is a huge void left between the inkjet printing and the stencil printing that has been filled by dispensing because dispensing can handle both spectrums to effectively fill the void.

How does it handle multiple thicknesses if you have different requirements across the same board?

Each jet valve has a particular flow rate, which allows it to dispense certain milligrams of material per second. You can change the flow rate of the valve by tweaking some software-driven parameters, such as the piston lift, which is the stroke of the piston or shaft moving up and down. The higher you lift it, the more material you dispense and the bigger the droplet size.

That's how you shift the output of your pump. Users can also dispense multiple droplets at the same location for thicker bead requirements.

Does dispensing happen across the entire board at one time or is this more of a step or motor function?

You can dispense the same board both ways. You can dispense in one go or in multiple passes. If there is an underfill application that requires six passes to dispense across the product, and you need to use flow timers before dispensing another layer to control how it flows on the substrate underneath the chip, you can do that. All those multiple passes can be handled by dispensing. It's very process- and material-driven.

Tell me about using dispensing in production.

Dispensing in production uses inline systems with a conveyor system. There might be requirements to preheat the board before the dispense zone. You preheat the board, making sure it gets to a certain temperature before you dispense. An underfill application is a good example. The board goes to the dispense zone, where the board will be sensed for the correct temperature, then the material will dispense along the sides of the BGA. To ensure that the material has wicked in, a capillary reaction takes place to ensure that material has flown underneath the BGA to the other side for a good fillet formation. The board moves downstream to a different machine for whatever is next in the process. All this has been handled by the machine itself. There could be factory communication happening, such as with an MES, which is controlling the machine and sending the commands to the dispenser. This is dispensing in the production environment. It runs almost 24/7 so it has to withstand that kind of product life cycle. Many of our dispensers in the field have been running for 20 years, and people are happy with them.

What are some of the new developments in dispensing?

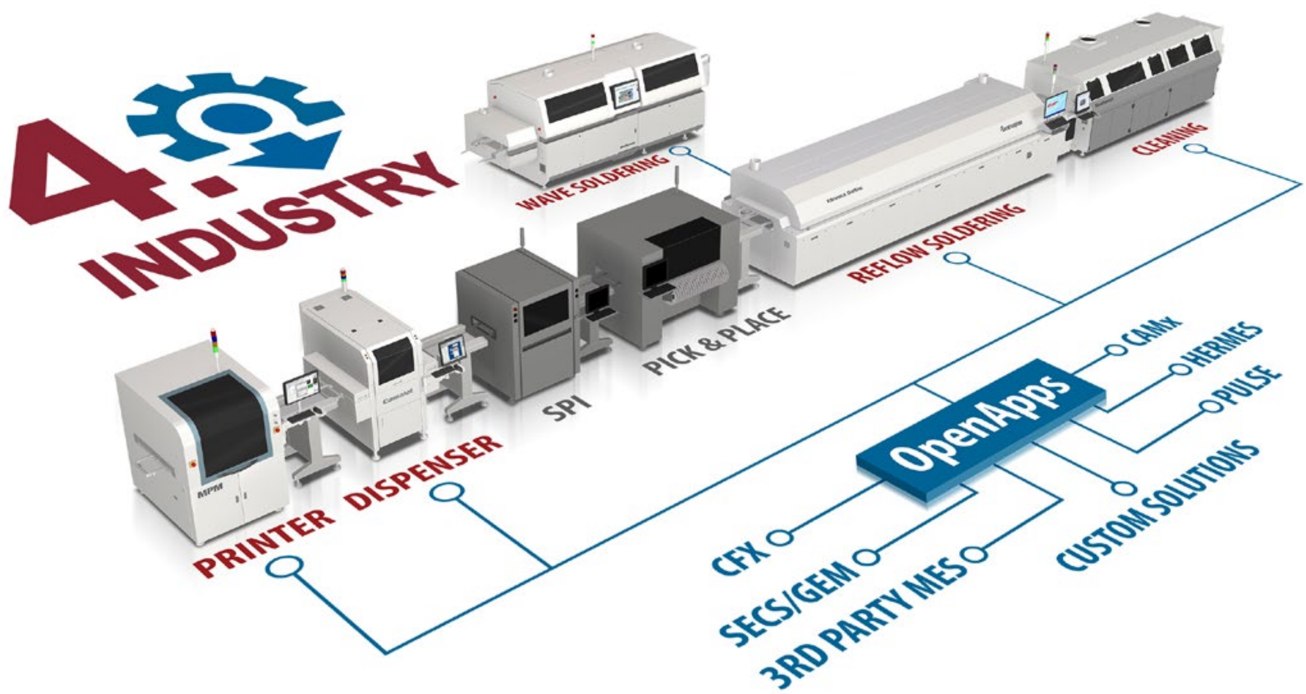
We have a customer-based innovation model to understand the pain points. We are always keen on improving our dispensing capabilities. For example, right now, we dispense everything from conductive inks to thermal interface materials, and we have been challenged with different materials.

Today's hot topic is AI and server applications which are high-performance computing applications involving new materials, like liquid metal and liquid metal paste. We've been making sure we are ready for these new, emerging technologies.

We've been talking about using liquid metal in PCB fabrication in processes like additive and semi-additive printing. Is this the same material?

It could be a different formulation. At IPC APEX EXPO, I attended a presentation on a similar kind of mate-

Camalot Prodigy manufacturing line.



rial being used in the additive process. We have been working with a material—and also with the liquid metal paste—which is more like a thermal interface material to make sure the heat is dissipated out of the die.

AI plays a part in that, but the other hot topic is sustainability. We are working to reduce our carbon footprint. We are developing a technology that improves our heating techniques: moving toward less air and energy consumption. Along with achieving faster preheating times, we can increase the throughput.

The other initiative is responding to the market, driving toward increasingly smaller feature sizes. We are trying to get more smaller deposits and smaller keep-out zones—200 to 300 microns for the semiconductor markets—because that is where the market is trending.

Finally, we have a tilt-and-rotate function where you can tilt the jet or the auger valve to dispense material on the sidewall to enable reduced keep-out-zone (KOZ) areas and improved capillary flow for underfill and edge bonding materials.

All this development has been in the works and we're making sure we have the necessary software functionality to support it. Open Apps 4.0, for example, is more like a Factory of the Future initiative which includes MES, CFX, and Secs/Gems standards. Open Apps 4.0 will be a new offering with shared architecture across all the EAE platforms.

Further out, we are moving to concentrate on self-optimization to minimize operator intervention at the machine level because we know that 95% of the downtime in production can be traced to the operators. To reduce or eliminate that downtime, we see more features like automatic machine calibrations. Once the operator changes the cartridge, it will automatically calibrate on the cartridge side, and provide automatic jet valve monitoring. All these features eliminate operator intervention.

Let's talk about temperature issues. Some jobs require heat but it's important to use energy efficiently. Yet, we keep moving toward very small parts, or very small balls on very large BGAs. Both trends require more precise dispensing. Do these trends track together?

Yes. We see the trend with both components moving next to each other as we try to get smaller droplets. It's driven by the technology roadmap because the market is moving to smaller, more precise droplets. Even the material suppliers are responding with materials that require heat on the product. If the product is not heated, it is difficult for small quantities of material to get that capillary action underneath the flip-chip, for example. So, both are related to each other in that case.

How does Camalot convert these trends into specific feature sets for customers?

There are plenty of options that we offer in dispensing. It could be from a productivity standpoint, with smaller footprint on the assembly line should you want more productivity out of a certain footprint. To handle those productivity requirements, we've developed a dynamic dual-head technology that is a fast and accurate solution for dual-head simultaneous dispensing. You can dispense on two products at the same time, doubling the productivity of the machine.

This dynamic tool head is a mini X-Y drive system, which goes on the second Z-axis, allowing for synchronous dispensing regardless of part-to-part rotation or scaling. The parts can be rotated with respect to each other in the tray or panel.

When dispensing underfill, for example, you may want to maintain the PCB at a certain temperature. If the requirement is 90°C, you want to make sure that the PCB is within the temperature range of 85–95°C to dispense underfill material. The infrared sensing is a closed loop monitoring of the PCB temperature, ensuring dispenser process stability and yield improvements for underfill applications. This is important because if the temperature of the PCB is too low, it could reduce the capillary flow reaction of the underfill. It could have less moisture absorption, which could lead to voiding, or it could also have flux defects because the flux residue didn't melt. If the temperature of the PCB is too high, then that underfill can gel before the flow is complete and that could lead to defects. People are looking for board temperature monitoring to make sure they are dispensing at the correct temperature.

Camalot Prodigy manufacturing.

The other option is a syringe cooler. This helps maintain the syringe at a controlled temperature within the heated machine environment. If your cartridge or the syringe with the material is sitting inside the machine at 50–60°C, it could activate the polymer reaction between the resin particles within the material inside the cartridge, altering its viscosity and leading to dispensing defects. To maintain that cartridge at a certain temperature near ambient 25°C, you need to ensure the syringe is not being exposed to the outside machine temperature. To protect that, you need to maintain that syringe to ensure that the pot life and the working of the material are maintained.

Another feature is the tilt-and-rotate, allowing you to gain access for dispensing on sidewalls or dispense around tall components. The tilt-and-rotate has rotary actuators, which are servo-driven to provide fast, high precision and zero backlash. With the pneumatic implementation, the backlash problem exists, which means you might specify 30°C, but it might go to 30.2°C. Precision is important when positioning the tip of the jet valve needle. This tilt-and-rotate feature has been on the market for two years, and it's getting good traction with our customers.

Sunny, thank you very much. This has been enlightening.

You're welcome. **SMT007**

PCB Surface Topography and Copper Balancing Under Large Form Factor BGAs

Article by Neil Hubble, AKROMETRIX
and Gary A. Brist, INTEL CORPORATION

Editor's Note: This paper was originally published in the Proceedings of IPC APEX EXPO 2024.

Background

As CPU and GPU packages grow larger and contain higher pin/ball counts, the importance of managing the printed circuit board (PCB) surface coplanarity for package assembly increases. The PCB surface coplanarity under a package is a product of both the global bow/twist of the PCB and the local surface topography under the package. In general, the surface topography is dependent the choice of material and layer stackup and the interaction between the innerlayer copper patterns and prepreg resin flow.

Advances in chiplet design and heterogeneous integration solutions in electronic packaging are enabling complex packages with increasing total die areas, resulting in the need for larger CPU and GPU packages[1]. Based on trends and advances in package integration, it is expected that future packages exceeding 100–120 mm on a package edge will become more common. This increases the challenge of the second-level interconnect (SLI) assembly processes when attaching the package to the PCB due to the combined coplanarity and topography variations of the PCB and package. These combined influences between the PCB and package are the key drivers of SLI defects such as solder bridging or solder joint opens during PCB assembly.[2,3] Figure 1 is a graphical depiction of how the global PCB warpage or curvature under the package must be smaller for larger packages to achieve the same PCB coplanarity under the package.

The characterization of PCB coplanarity under the package footprint has been studied historically, including influences of assembly temperatures on dynamic PCB coplanarity as the PCB and package move together through the assembly reflow temperature profile.[4,5,6]

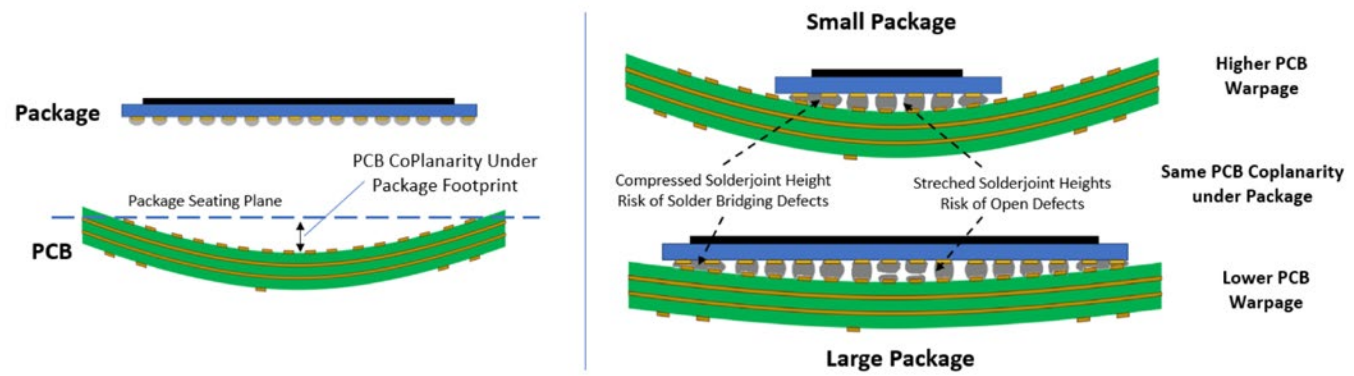
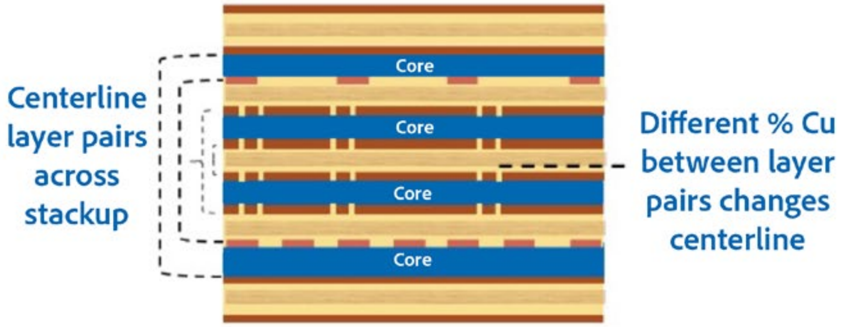


Figure 1: PCB coplanarity under package.



ΔCu	Tcu	ΔH
10%	1 oz (32 μm)	3.2 μm
20%	1 oz	6.4 μm
30%	1 oz	9.6 μm
30%	H oz (16 μm)	4.8 μm

Figure 2: Layer pairs and thickness delta vs. delta retained copper.

Other works have shown how the choice of PCB materials, fabrication process conditions, and design each impact global PCB bow/twist and warpage[7].

Modeling and observation have shown PCB stackup symmetry is essential for both the thermo-mechanical properties and the thicknesses selection of materials, with respect to the neutral axis, to manage and minimize PCB bow/twist and global warpage.[7,8,9] Care is usually taken to balance the retained copper between centerline layer pairs across the centerline of a PCB, as shown in Figure 2, to prevent shifting of the PCB neutral axis that could initiate general warpage.

While controlling global PCB bow/twist and warpage is important, it is the local PCB topography or coplanarity of the solder pads directly under an individual package that is critical in determining the SLI solder joint. Prior work highlighted that localized variations of percent copper within an individual PCB layer can result in localized changes in prepreg thickness leading to core deformation.[10] Regardless of how well percent copper is balanced across centerline layer pairs, the percent copper directly under the package is usually much different than the average percent copper for an individual layer due to antipad arrays in plane layers or pad/trace density on signal layers, as shown in Figure 3. The impact of these variations increases with higher-layer PCB designs and thicker copper layers for power delivery.

As the local total copper under a package is often different than outside the package footprint, it is possible to obtain local thickness and surface topography variations across the PCB directly under the package driven by resin shrinkage during the lamination process.[11,12,13] Under packages with large edge dimensions or when using a resin with low flow characteris-

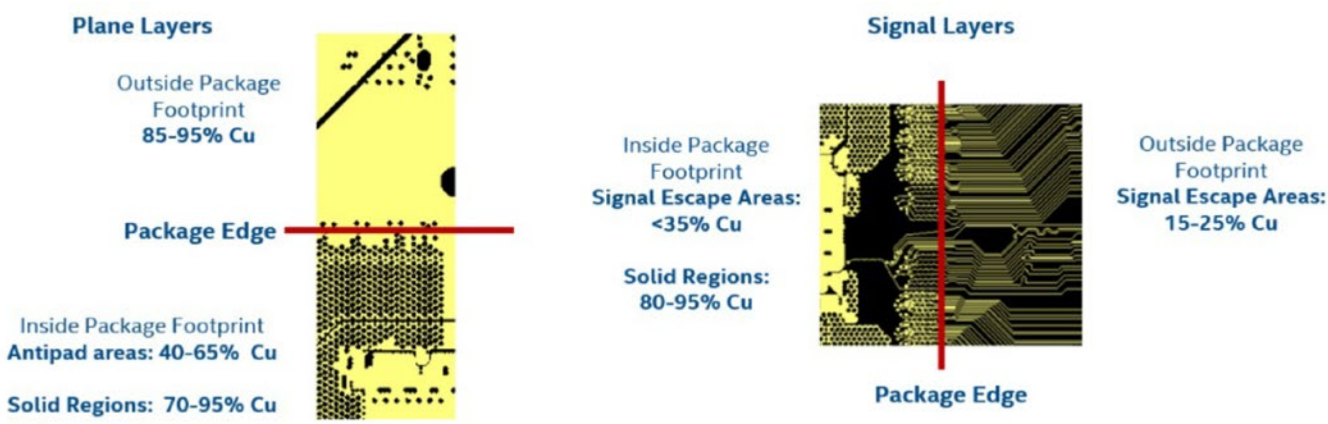


Figure 3: Representative retained copper outside and inside a package footprint area.

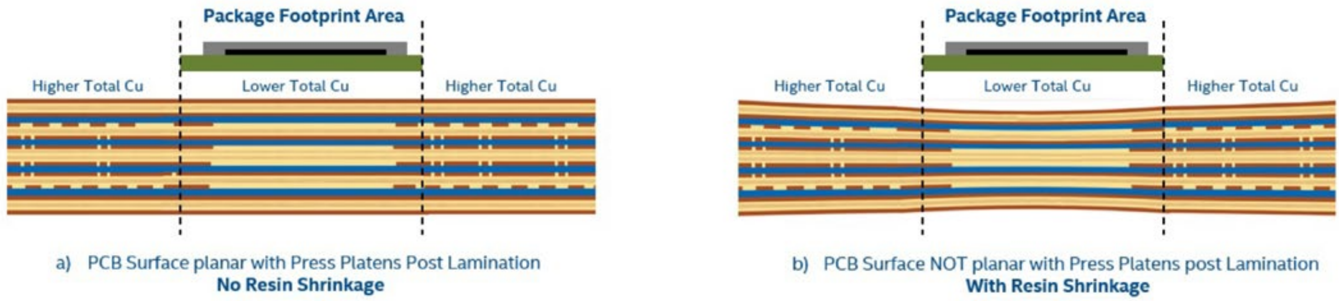


Figure 4: PCB thickness under package with and without resin shrinkage.

tics there can be insufficient resin flow during the lamination cycle that can impact local PCB thicknesses and surface topography. Figure 4a shows the classical assumption where the PCB surfaces after the lamination process are planar with the press platen plates. Figure 4b shows the potential PCB cross-sectional deformation resulting from resin shrinkage. When the resin volume is uniform across the PCB, the impact of resin shrinkage is not noticeable, but when the resin volume is not uniform across the PCB and not supported by PCB copper shapes, the impact of resin shrinkage will be proportional to the change in resin volume.

In some cases, under large package footprints, designers may stack power or shielding shapes which result in islands of higher total copper within a local area under the package. In these cases, the mechanism of resin shrinkage could generate non-standard profiles such as "W" shapes.

Experimental Methodology

A test vehicle was designed to evaluate a range of copper balancing distributions under various package footprint sizes ranging from 28 x 28 mm to 89 x 89 mm. Figure 5 shows the test vehicle stackup and basic design floorplan. To drive a high range of total copper and emulate high-end server and network systems, a symmetrical 22-layer construction with target 2.50 mm final thickness was selected which consisted of six layers of 2-ounce copper located at the center and six layers of 1-ounce copper at the top and bottom of the stackup. The basic floorplan form-factor was 205 x 285 mm with a set of components and connector placements. Component footprints included a large 89 x 89 mm footprint at 1 mm pitch located at the center of the floorplan, two 37 x 37 mm footprints at 0.8 mm pitch, and two 28 x 218 mm footprints at 0.65 mm. A power channel from the large package footprint was also included to

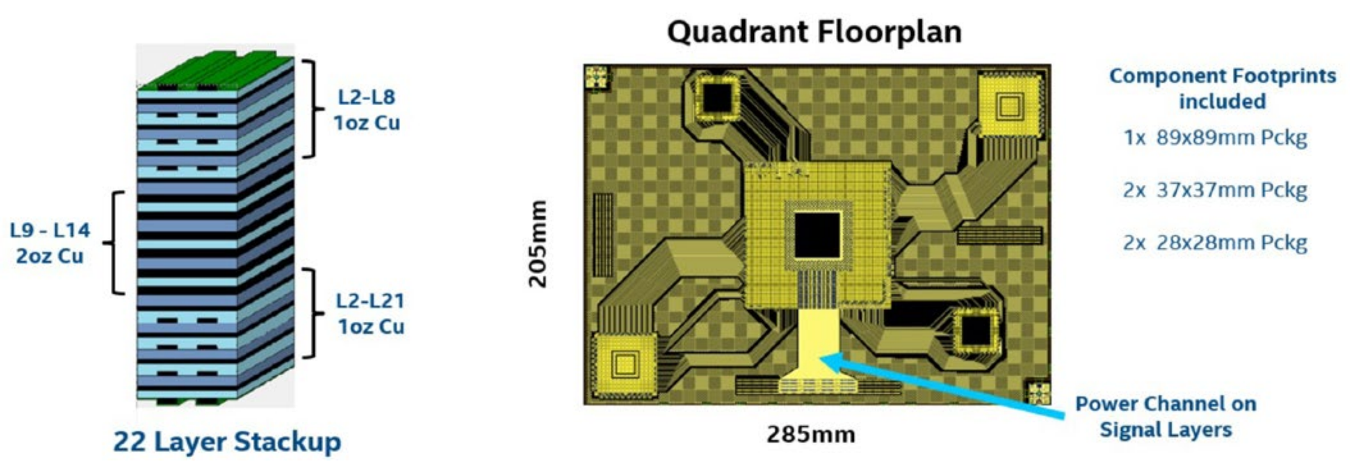


Figure 5: Test vehicle stackup and floorplan.

replicate a common power delivery strategy of added copper fill regions on signal layers.

Blocking for Manufacturing Factors, Fabrication, and Material Scope

The test vehicle design consisted of replicating the quadrant floorplan in a two-by-two grid to cover a full manufacturing panel as shown in Figure 6. Each quadrant was designed with a different copper balancing strategy under the component footprints. The copper patterns for an individual component footprint were identical for components replicated within a single quadrant. The purpose of different copper balance strategies within the same manufacturing panel was to block out manufacturing process and material lot factors. This approach allowed evaluation of design variations while maintaining the same press book location, same lamination press load, same thermal profiles, and same physical core and prepreg. Eight materials were included in the test, fabricated across eight supplier sites. Each fabricator selected the preferred core and prepreg combination to meet the target stackup. Copper distribution profile, materials chosen, and fabricators are shown in Figure 7.

Analysis Methodology

The warpage and coplanarity measurements were taken with shadow-moire technique at 1.7 µm resolution using 150 line per inch grating. The tool model was an Akrometrix TTSM-J with the loading tray design adjusted for PCB

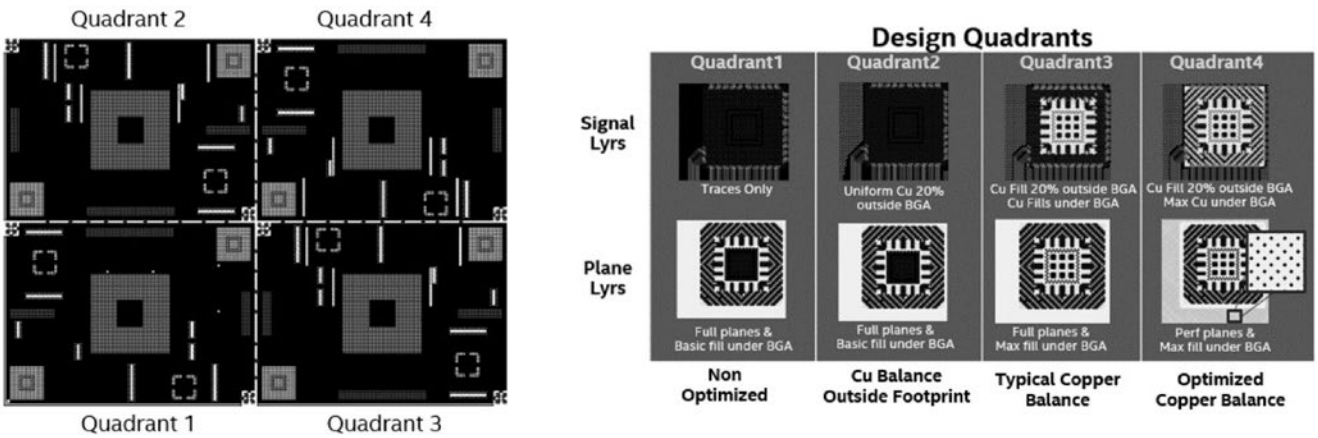
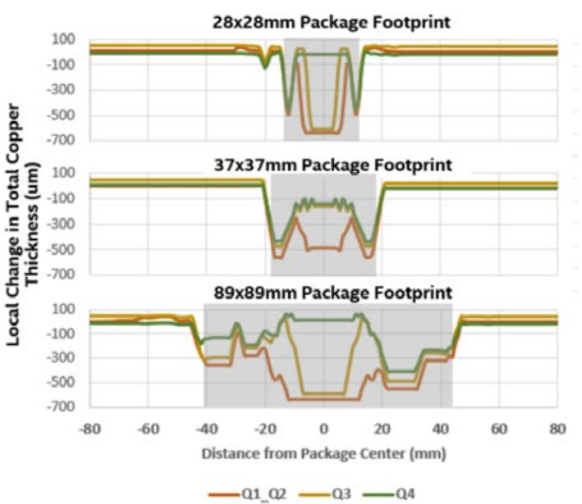


Figure 6: Full test vehicle panel and copper balance strategies by quadrant.



Materials Evaluated
- N4000-13
- IT-988GSE
- Metorwave 8000
- EM-528
- IT-170GT
- TUC-883
- EM-890K
- TUC-933+

PCB Fabricators
- WUS
- Victory Giant
- TTM-DMC
- TTM-FG
- AKM-Meadville
- ACCL
- SCC
- Founder

Fabrication Build Matrix

Material ID	M1	M2	M3	M4	M5	M6	M7	M8
Supplier Code	A, C, E	A, E, C	B, D, H	B, H	D, G	F	F	G

Figure 7: Build matrix: copper distribution profiles, materials, and fabricator build lots.

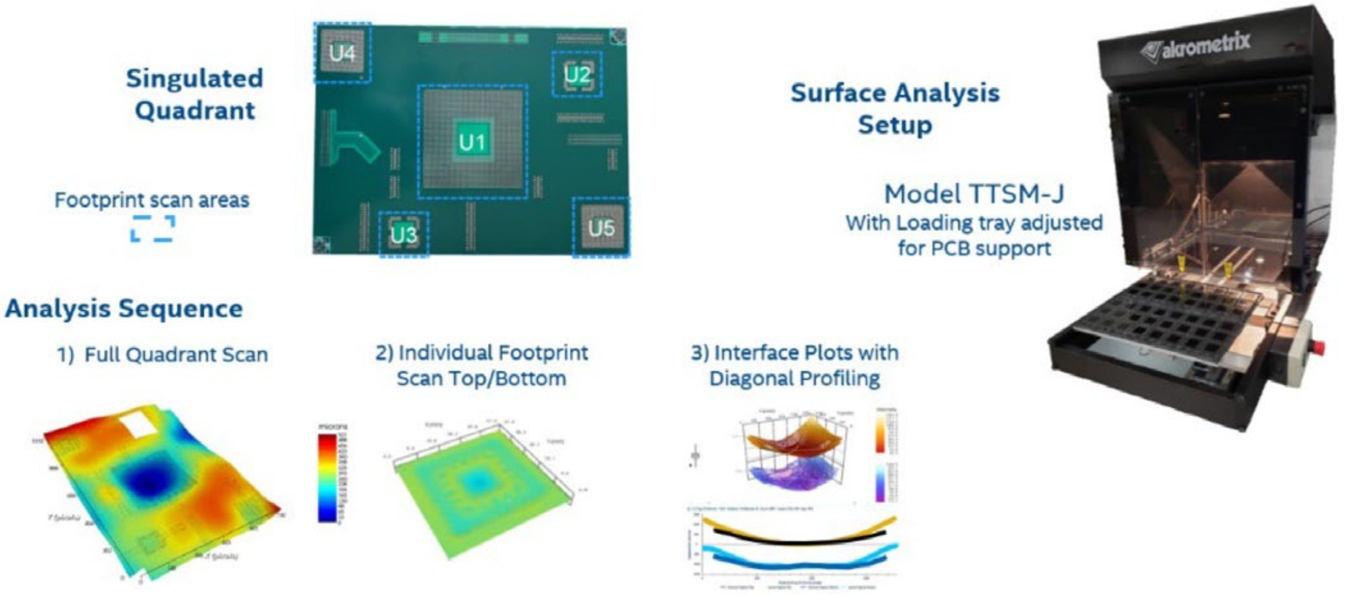


Figure 8: Surface analysis and metrics.

support. For each build lot, four manufacturing panels were selected, serialized, and singulated into the individual quadrants. The quadrant level and component footprints were measured from front and back (Figure 8). The area of measurement for each component was 12.5 mm beyond the edge of the component perimeter. Analysis metrics of warpage and coplanarity were then obtained for front and back of each component footprint. Interface plots were also obtained to evaluate local PCB thickness profiles along the diagonals of each component site in conjunction with the general coplanarity of the site.

Physical microsections were also taken after surface analysis by shadow moire. These microsections were done on a sampling basis across different build lots, quadrants, and component sites to confirm PCB thickness variations obtained from the interface plots. Measurements were taken every 1 mm along the microsection to generate profiles of PCB surfaces relative to the PCB centerline (Figure 9).

Results and Discussion

The differences in copper balance strategies had a significant impact on the coplanarity under a component. Figure 10 shows the measured results for the smaller 28 x 28 mm package footprint across the different materials and PCB fabricator build lots. Here, the materials were ordered generically from highest to lowest resin flow going left to right. The selection of material had a very significant impact when the design was not optimized for copper balance. Whereas, when the copper balancing was optimized, the choice of material had little impact on average coplanarity with only a noted increase in variability for materials considered to be lower flow. This observation matched general expectations based on the delta copper profiles for the 28 x 28 mm footprint as shown in Figure 7.

Figure 11 shows summary results of coplanarity for a selected high-flow, mid-flow, and low-flow material by package size and copper balance strategy, including data across multi-

Figure 9: Full microsection across the component footprint.

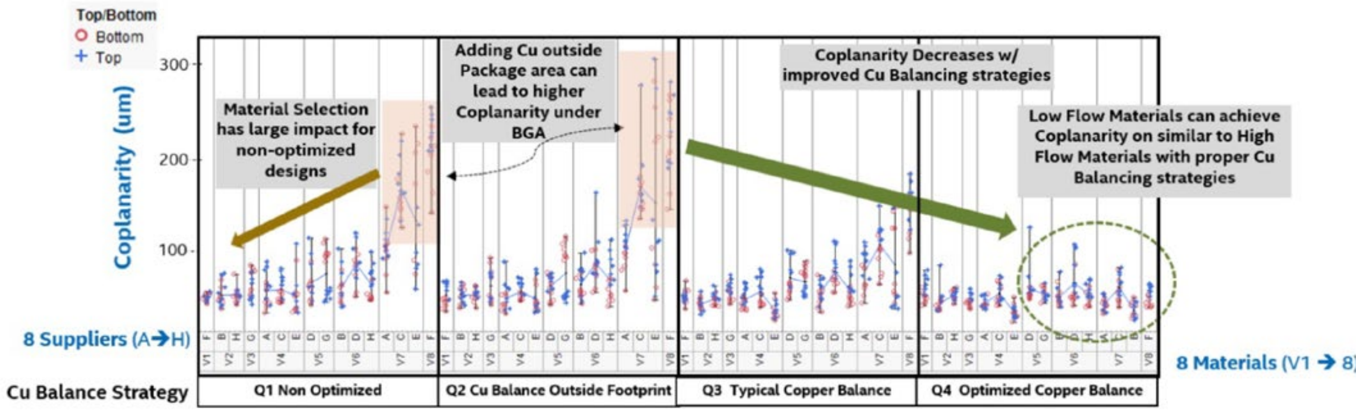


Figure 10: Coplanarity for 28 x 28 mm package by copper balance strategy, material, and fabricator.

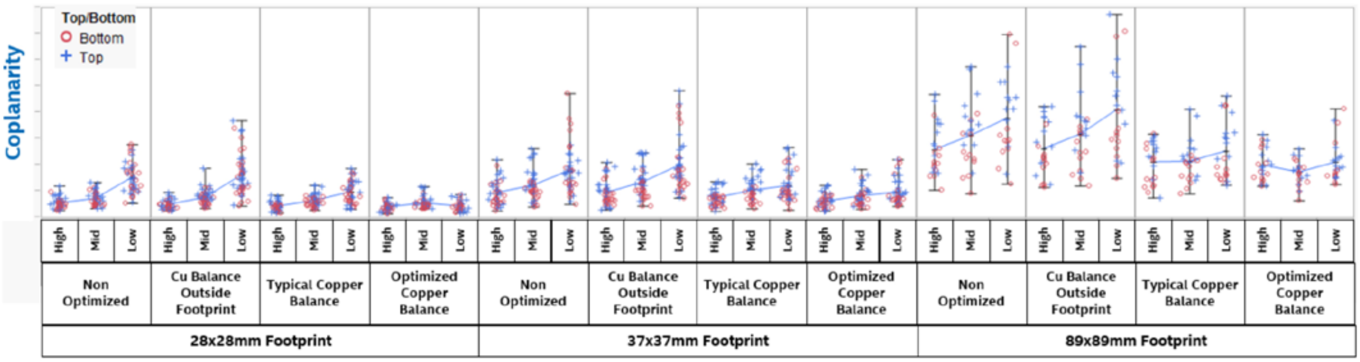


Figure 11: Coplanarity difference between higher and lower flow laminates across package size.

ple fabricators. In general, for the same design strategy and similar delta copper profiles under the component, the larger the package footprint, the higher the average coplanarity and the higher variation that was measured. This highlights the need to optimize copper balancing for coplanarity as package size increases. With optimized copper balancing, the coplanarity increased, but the difference between high-flow, mid-flow, and low-flow materials was much lower. This demonstrates that sufficient copper balancing can enable the usage of lower flow materials—even with large package footprints.

Figure 12 shows the full data set of measured coplanarity across copper balance strat-

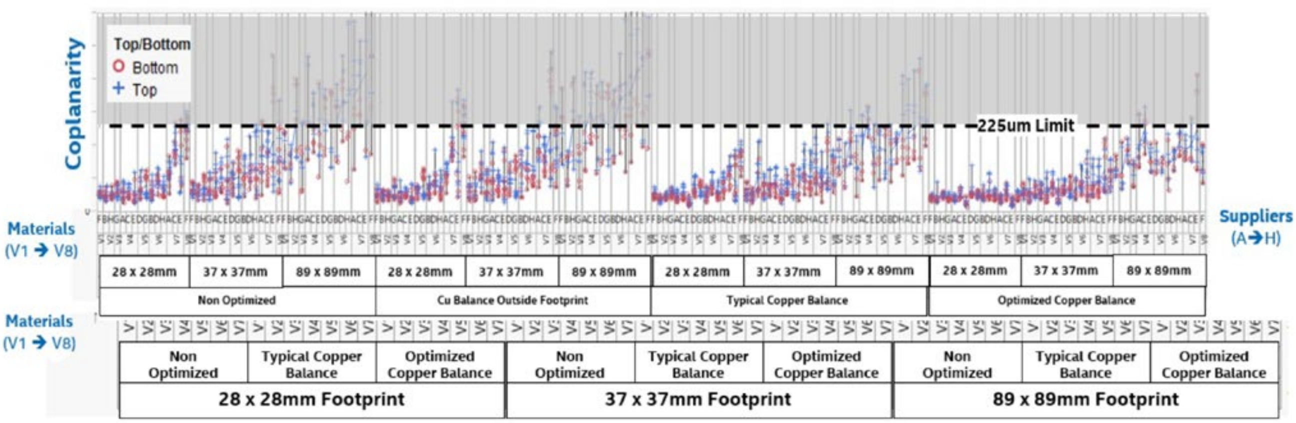


Figure 12: Measured coplanarity by copper balance strategy and package footprint size.

egy, package footprint size, material, and PCB fabricator with respect to a 225 µm coplanarity acceptance limit. It shows that, with few exceptions, small- and medium-sized package footprints can achieve low coplanarity requirements without much consideration for copper balancing. With larger package footprints, the need to balance copper becomes extremely critical.

Fabricators did not use the exact same prepreg glass styles or resin content which resulted in each fabricator having different levels of available resin volume for filling copper void areas under the component at each prepreg opening, but nonetheless, coplanarity trends were consistent across fabricators. Based on microsection analysis, this difference in stackup construction likely accounted for most differences seen between PCB fabricators rather than any differences in lamination profiles.

Coplanarity and Local PCB Thickness Variations

The data obtained from the Interface Analysis feature in Akrometrix analytic software was used to separate the contribution of local PCB thickness variations and the contribution of bow/twist and warpage of the PCB on coplanarity of the component footprint. The interface plots also allow analysis of how the superposition of the two contributions resulted in coplanarity differences between front/back and between panels. The methodology of including different copper balance strategies within the same manufacturing panel made it possible to evaluate the progression of total coplanarity and local thickness variations as the profile of local copper changed. Figure 13 shows an example progression for different changes in local copper as well as the coplanarity and local PCB thickness for two different materials.

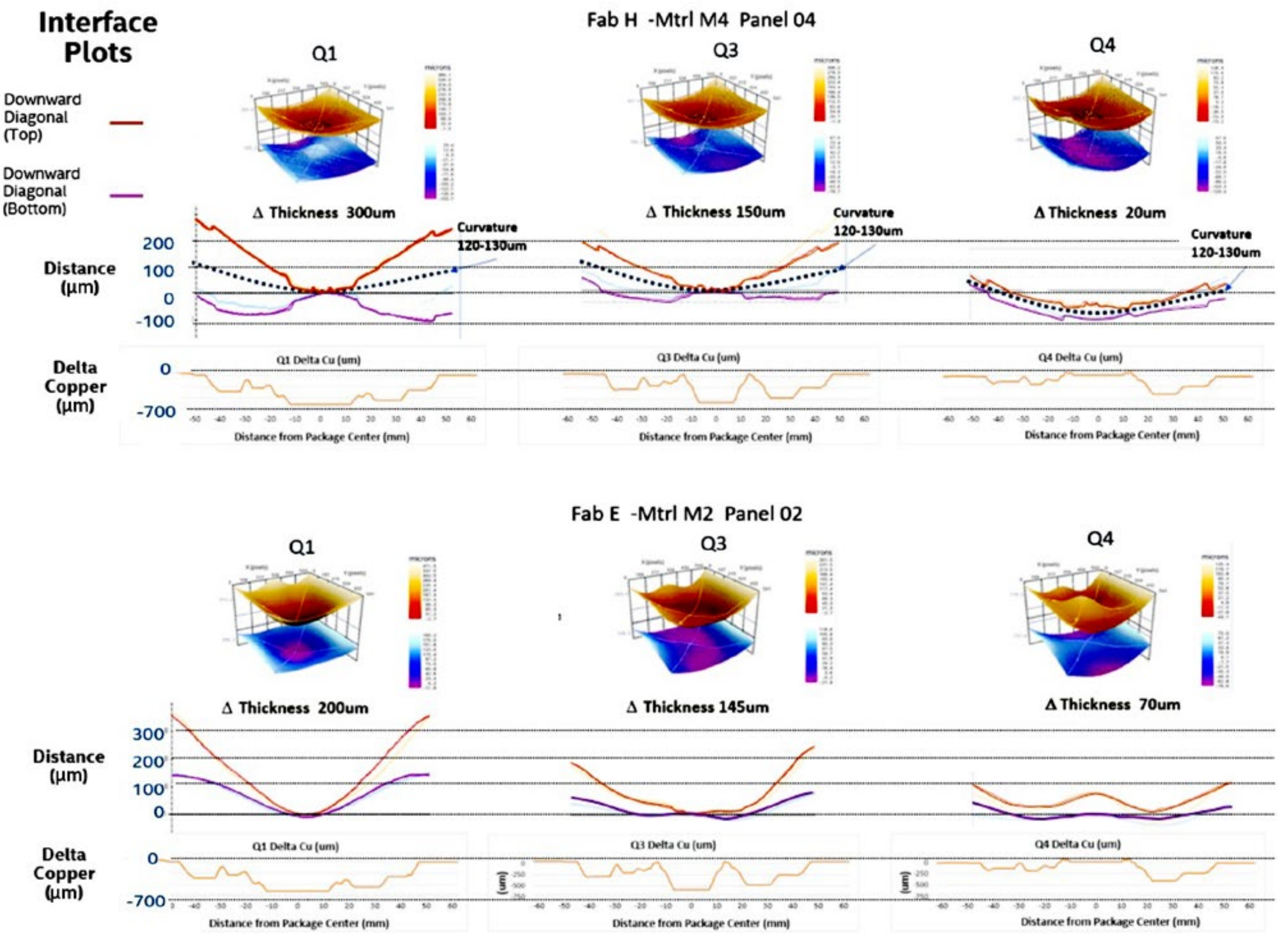


Figure 13: Superposition of local PCB thickness variations with general PCB bow/twist.

Figure 14 shows multiple instances of the superposition of local PCB thickness profiles and various global PCB curvatures for a few selected samples evaluated in this study. For example, Figure 14 A1-A4 show instances of "frown" global PCB curvatures and the resulting different differences due to local thickness variation.

In general, there was no trend for general bow/twist curvatures by material, fabricator, or quadrant/design. There were observed panel-to-panel variations between "smiley" or "frown" curvatures within most fabrication build lots. For a given manufacturing panel, each of the quadrants behaved similar and had similar warpage signatures.

The primary factors driving the local thickness variation at a given component footprint were the delta copper profile of each quadrant and size of the component footprint. The "hourglass" thickness shape always followed when the delta copper profile contained a high level of removed copper from the center. The "bowtie" thickness shape always followed when local areas within the interior of the component footprint had a region of maximum copper that approached or exceeded the copper outside of the footprint.

Figure 15 shows the relationship between local PCB thickness differences by the component footprints measured, different copper balance strategies, and material selection. These measurements were obtained via Akrometrix interface analysis and confirmed with physical microsections on a random sampling. The change in PCB thickness locally under each component had the same trends as Figures 11 and 12. The magnitude of the local PCB thickness increased with component footprint size and was dependent on the material selection. In addition, there was a very strong correlation by design, with the non-optimized designs having the highest delta thickness to the optimized copper balance designs having the lowest.

Conclusions

It was shown in this study that design and material/stackup selection can be the primary

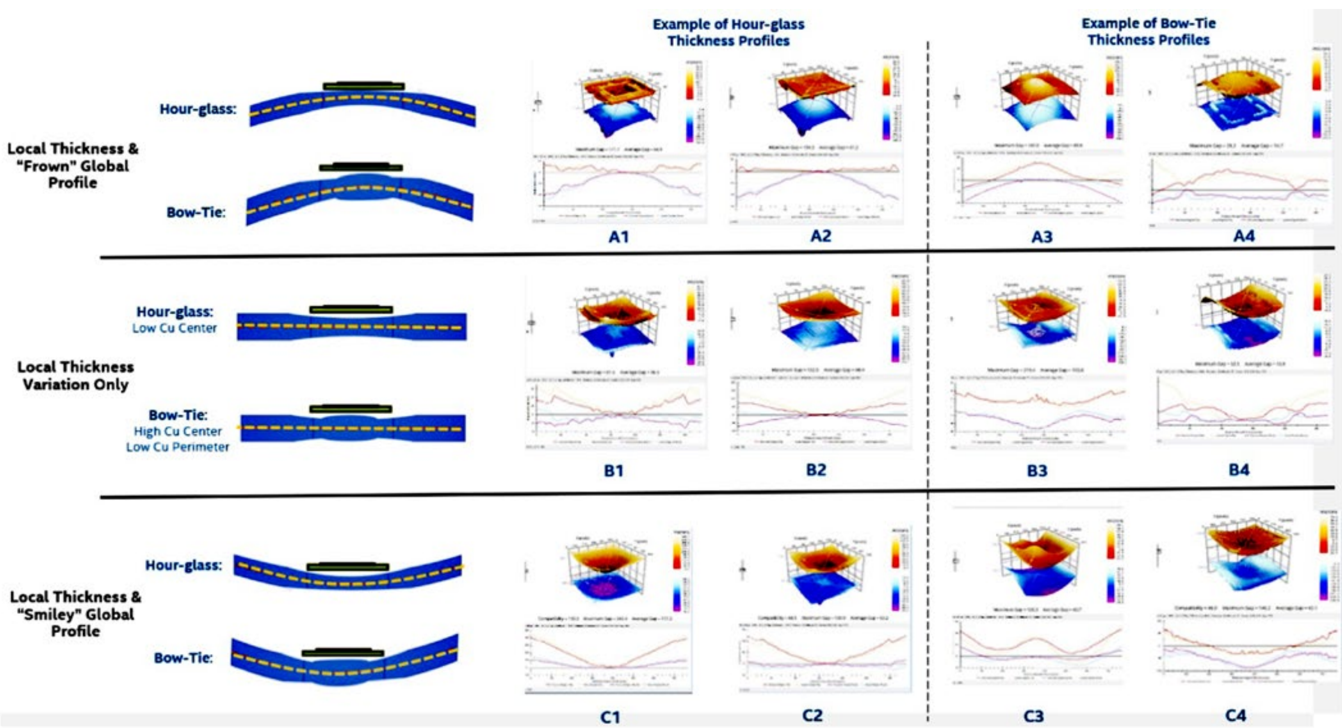


Figure 14: Local coplanarity examples of local thickness variations superposition with general PCB bow/twist.

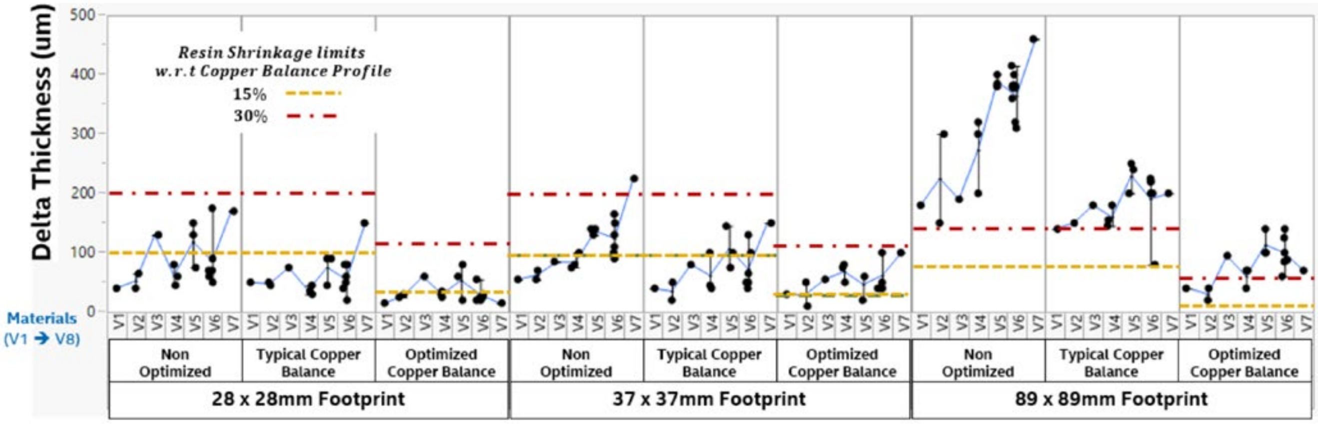


Figure 15: Delta thickness under component footprint.

factors influencing coplanarity variation under a component. It was also demonstrated that for the same copper balance strategy, coplanarity increases as the component footprint dimensions increase.

The profile of retained copper under a component has a direct relationship with the surface topography and coplanarity variation of the PCB footprint for the component. It was shown that as the total removed copper increases, the coplanarity increases. As silicon substrate packages grow in size, it is increasingly important to balance the percent of retained copper under the package footprint and match it to the percentage of copper outside the package on each layer, and balance the total across layers.

Large reductions in total copper under a component footprint leads to an hourglass PCB thickness profile and can be a major driver of high localized PCB coplanarity variations under components. Generally, this is the result of having a high concentration of antipads, or incomplete copper floods on power layers under a component resulting in a change in percent retained copper. The impact grows proportionally higher with more and/or thicker copper layers. This can be optimized by selectively adding non-functional pads for power and ground nets and fill any open areas on both signal and plane layers. Care should be taken when using mixed signal/planes on an individual PCB stackup layer as stacking localized areas of high copper under the component footprint can result in complex “W” or “bowtie” surface profiles.

The selection of materials in a PCB stackup can impact the variability and average coplanarity within a given design, especially as component dimensions increase. The impact of material selection is much less when the design is optimized for copper balancing.

The usage of interface analysis obtained by scanning top and bottom surfaces of the PCB, such as the Akrometrix method, can be used to quantify and differentiate the contribution of local PCB thickness variations and the global PCB bow/twist or warpage on the coplanarity at a component footprint. The high correlation of coplanarity to local PCB thickness variations across materials and copper balance strategies provides an opportunity for designers and PCB fabricators to gauge the “goodness” of a design and material selection by evaluating the local PCB thickness profile. By combining the technique with analysis of the total retained copper profile outside the component footprint and under the component footprint, it is possible to determine where and how copper balancing improvements should be implemented. This can also be used to prioritize between design changes or material choices vs. process changes when addressing coplanarity issues.

Acknowledgements

The authors would like to acknowledge Larry Marcanti and the rest of the HDPUG staff for the guidance and coordination of the project through its many phases. Special recognition to the those that provided laminate materials for the project and technical support through the project: Taiwan Union Technology Corporation (TUC), Elite Materials Corporation (EMC), AGC Multi Materials (AGC), and ITEQ. And an extended thanks to the PCB fabricators who supported the individual lot builds: WUS, Victory Giant (VGT), TTM-DMC, TTM-Forest Grove, AKM-Meadville, ACCL, SCC, Founder. SMT007

References

1. "Recent Advances and Trends in Advanced Packaging," by J.H. Lau, IEEE Transaction on Components, Packaging, and Manufacturing Tehcnology Vol.12, No.2, pp. 228-251, 2022.

2. "Impact of PCB Design, Materials, and Manufacturing Process On PCB BGA Land Pattern Warpage and SMT Yield," by Walwadkar S., et al, SMTAI Conference 2022.

3. "Study on the Effect of the Warpage of Electronic Assemblies on Their Reliability," by O. Albrecht, K. Meier, H. Wohlrabe, IEEE 8th Electronics System-Integration Technology Conference (ESTC), Tønsberg, Norway, pp. 1-7, 2020.

4. "An Investigation into the Predictability of PCB Coplanarity for Romm vs. Lead-free Assembly Temperatures," by J.J. Davignon, IPC APEX EXPO Technical Conference 2012.

5. "Warpage Studies of Printed Circuit Boards with Shadow Moire and Simulations," by S.J. Oon, K.S. Tan, T.Y. Tou, S.S. Yap, C.S. Lau, Y.T. Chin, 38th International Electronics Manufacturing Technology Conference 2018, Melaka, Malaysia.

6. "New Package/Board Materials Technology for Next-Generation Convergent Microsystems," by N. Kumbhat, Electronics Packaging Technology Conference 2003, pp. 331-335.

7. "Controlling Bow and Twist," by B. Willis, Electronics Engineer, November 1997.

8. "Modeling Printed Circuit Board Curvature in Relation to Manufacturing Process Steps," by G.A. Schuerink, M. Slomp, W.W. Wits, R. Legtenberg, E. Kappel, 2nd CIRP Global Wed Conference 2013, pp. 55-60.

9. "Effect of Resin Content on PCB Warpage and Anti-Shock Performance of BGA Solder Joint," by M. Lili, 17th International Conference on Electronic Packaging Technology 2016, pp. 683-685.

10. "PCB Trace Impedance: Impact of Localized PCB Copper Density," by G. Brist, J. Krieger, D. Willis, IPC APEX EXPO Technical Conference 2012.

11. "Development and Application of Novel Technique for Characterising the Cure Shrinkage of Epoxy Resins," by R.F. Minty, J.L. Thomason, L. Yang, W. Stanley, A. Roy, Polymer Testing, Vol. 73, pp. 316-326.

12. "Characterization of Prepreg Shrinkage and Investigation of its Influence on Warpage Simulation," by J. Zundel, M. Sagerer, M. Frewein, T. Krivec, 22nd International Conference on Thermal, Mechanical, and Multi-Physics Simulation and Experiments in Microelectronics and Microsystems (EuroSimE) 2021, St. Julian, Malta.

13. "Characterization and Modeling Cure- and Pressure-Dependent Thermo-Mechanical and Shrinkage Behavior of Fast Curing Epoxy Resins," by N. Lorenz, M. Muller-Pabel, J. Gerritzen, J. Muller, B. Groger, Polymer Testing Vol. 108, 2022.

Neil Hubble is president of Akrometrix.

Gary A. Brist is senior principal engineer of PCB technology and applications at Intel Corporation.


Your survival guide for
the harshest environments.
Learn how encapsulation resins are used to ruggedize electronics.
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
Encapsulating Sustainability for Electronics
Beth Turner
MacDermid Alpha Electronics Solutions
I-007e Books
Look inside
I-007e Books

Our Strength Comes From Working Together

Global Citizenship

by Tom Yang, CEE PCB

Last time, we discussed the similarities between Chinese and American companies, and I am excited to say that we found many. It made me think about ways companies from both countries and cultures can successfully work together when we put aside our differences and combine our resources. So, I did some research and found these examples.

Joint Ventures and Strategic Alliances

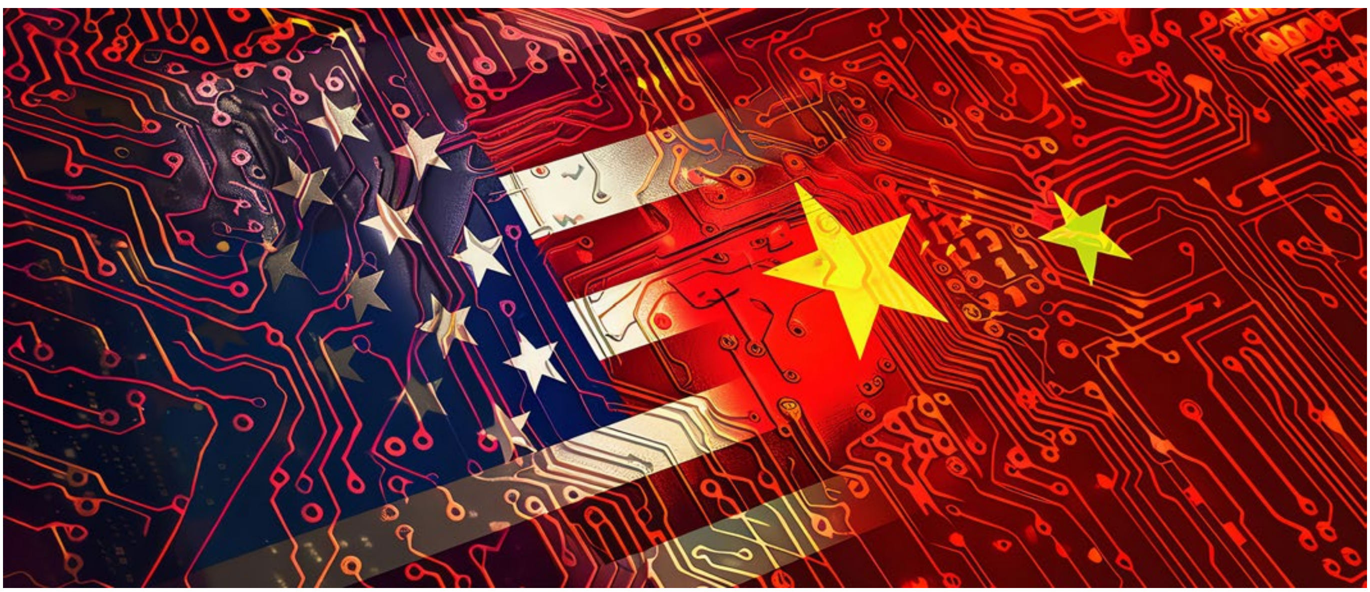
Joint ventures and strategic alliances are powerful ways for companies to leverage each other's strengths while sharing risks. A joint venture involves two or more parties creating a new business entity, characterized by shared ownership, returns, risks, and governance. By entering joint ventures, American and Chinese companies can combine their technological expertise, market knowledge, and resources to develop new products or expand into new markets. American companies can benefit from China's manufacturing capabilities and extensive market, while Chinese companies can access the U.S.'s advanced technology and global networks.

Example: General Motors and SAIC

The partnership between General Motors (GM) and SAIC Motor Corp. has allowed GM to tap into the Chinese market while enabling SAIC to benefit from GM's technological advancements and expertise in automobile manufacturing.

Research and Development (R&D) Collaborations

Investing in joint R&D initiatives can drive innovation and foster the development of cutting-edge technologies. Companies can pool their research resources to address common



challenges, from environmental sustainability to artificial intelligence. Collaborations in R&D not only reduce costs but accelerate innovation by combining diverse expertise and perspectives.

Example: IBM and Suning.com

This partnership using AI and cloud computing exemplifies how collaborative R&D can lead to significant technological advancements. This cooperation has enabled both companies to develop new solutions that enhance their competitiveness in the global market.

Supply Chain Integration

Integrating supply chains can lead to increased efficiency and cost savings. Working together, companies can optimize logistics, reduce production costs, and improve supply chain resilience. American companies can benefit from China's manufacturing infrastructure, while Chinese companies can leverage American logistics and distribution networks.

Example: IBM and Foxconn

Apple's partnership with Foxconn (Hon Hai Precision Industry Co.) illustrates effective supply chain integration. Foxconn manufactures a significant portion of Apple's products in China, enabling Apple to benefit from lower production costs and Foxconn's manufacturing expertise, and providing a profitable venture for Foxconn.

Market Expansion and Access

American companies can help Chinese firms navigate the complexities of Western markets, including regulatory compliance and consumer preferences. In return, Chinese companies can help American firms penetrate the Chinese market, which often has different business practices and consumer behaviors.

Example: Starbucks and Alibaba

Starbucks' expansion in China through its partnership with Alibaba showcases how collaboration can facilitate market entry and growth. By integrating with Alibaba's digital ecosystem, Starbucks has enhanced its online presence and delivery capabilities in China.

Cross-border Investments

Cross-border investments can strengthen financial ties and create growth opportunities. Businesses can access new technologies, markets, and resources. These investments can take the form of equity stakes, mergers, or acquisitions.

Example: Tencent, Tesla, and Snap

Chinese tech giant Tencent's investment in American companies like Tesla and Snap has given Tencent access to cutting-edge technologies and innovative business models, while American companies have gained financial support and entry into the Chinese market.

Cultural Exchange and Mutual Understanding

Building strong relationships based on mutual respect and understanding is crucial for successful collaboration. Cultural exchange programs, joint conferences, and regular communication can help bridge cultural gaps and foster trust. Understanding each other's business etiquette, communication styles, and decision-making processes can significantly enhance collaboration.

Example: China-U.S. Business Leaders' Summit

The annual China-U.S. Business Leaders' Summit provides a platform for executives to discuss opportunities, challenges, and best practices. It promotes cultural exchange and fosters relationships that lead to successful collaborations.

COLLABORATION

Leveraging Technology and Digital Platforms

Using digital platforms and technologies can facilitate collaboration across geographical boundaries. Virtual meetings, cloud-based project management tools, and digital communication channels can streamline coordination and enhance efficiency. Technology can also enable real-time data sharing, collaborative product development, and remote working arrangements.

Example: Zoom and Microsoft

Platforms such as Zoom and Microsoft Teams have become essential for cross-border collaboration. They were critical during the COVID-19 pandemic, allowing companies to maintain communication and continue joint projects despite travel restrictions.

Compliance with International Standards and Regulations

Adhering to international standards, such as those from IPC, and regulations is essential for seamless collaboration. Companies must ensure that their products, services, and operations comply with relevant laws and standards in both countries. Doing so reduces the risk of legal issues and enhances the credibility of collaborative ventures.

Example: Boeing and COMAC

The partnership between Boeing and COMAC (Commercial Aircraft Corporation of China) in developing aviation technologies underscores the importance of compliance with international aviation standards. These companies have worked together to ensure they meet the necessary stringent safety and regulatory requirements.

Corporate Social Responsibility (CSR) Initiatives

Collaborating on CSR initiatives can strengthen the social and environmental impact of businesses. Joint CSR projects can address global challenges, such as climate change, poverty, and access to education. Working together, companies can leverage their resources and expertise to create meaningful and sustainable solutions.

Example: Coca-Cola and the China Women's Development Foundation

The collaboration between Coca-Cola and the China Women's Development Foundation on water stewardship programs has improved water access and management in rural Chinese communities while enhancing Coca-Cola's corporate reputation.

The potential for companies from China and the United States to work together is vast, with significant benefits for both. By adopting these strategies, businesses can harness their collective strengths, drive innovation, and create sustainable growth. As global economic interdependence deepens, the success of these collaborations will not only enhance the prosperity of the companies involved but contribute to global economic stability and development.

I was pleasantly surprised that my research uncovered so many examples of excellent collaboration between American and Chinese companies. So, what do you say? Isn't it time for a collaborative partnership between a Chinese and an American PCB company? That would be a great step toward good global citizenship. **SMT007**

Tom Yang is CEO of CEE PCB. To read past columns, click here.

Some Fundamentals of Tin-Bismuth Metallurgy and Electromigration

Article by Prabjit Singh, Raiyo Aspandiar, Haley Fu, and L. A. Swaminathan

Abstract

The high homologous temperature of tin-bismuth alloys and the high electromigration effective valence of bismuth makes tin-bismuth alloys' microstructures forever changing in their applications, even at temperatures as low as room temperature. While this behavior is a serious concern to engineers who must ensure that this forever changing behavior does not have too much of a negative and unacceptable impact on reliability, it makes for interesting research. This paper describes some of the fundamentals an iNEMI team working on Sn-Bi electromigration unearthed during their study and some innovative techniques they developed along the way to better study electromigration in these alloys.

Introduction

Tin-bismuth has received a lot of attention as being the solder alloy of choice for electronic assemblies requiring low-temperature soldering for a number of reasons, including hierarchical soldering, reducing warpage of assemblies, reducing damage to components that may be less resistant to high temperatures and simply to take advantage of the energy savings resulting from low-temperature operations[1]. Incorporating tin-bismuth solder joints into assemblies has its own unique challenges aris-


The best
dispensing solution
for the best future!
axxon
MYCRONIC
www.axxonauto.com
Follow axxon
About axxon

ing from two factors. One is the high homologous temperature of tin-bismuth solders that keeps the microstructures continuously changing even at temperatures as low as room temperature. The other is the high electromigration effective valence of Bi atoms which results in the Bi-rich phase accumulating and segregating on the anode side of the solder joint. Not only does this electromigration phenomenon increase the electrical resistance of the solder joints, but it is also suspected of making the joint prone to cracking in shock and drop tests. Over the past two years, an iNEMI team has been busy studying Sn-Bi electromigration to gain a greater understanding of the phenomenon to help design experiments that can better study the effect of electromigration on the susceptibility of the solder joints to shock and vibration. This paper describes some of the fundamentals of metallurgy and electromigration that the iNEMI team learned and discovered over the two-year study. We hope these fundamentals will be useful to those working to make Sn-Bi solders reliable and useful in first- and second-level electronic packaging.

Tin-Bismuth Phase Diagram: Why the Bi-rich Phase Volume Fraction Increases With Time When Aged at Room Temperature

Binary Sn-Bi alloy has a eutectic phase diagram shown in Figure 1 with a eutectic temperature of 138.5°C and the eutectic point at 44 atomic % Bi[2]. When cooled from the liquid state to just below the eutectic temperature, two solid phases form: a Sn-rich phase with 13 at %Bi and a Bi-rich phase that is almost pure bismuth. When slowly cooled to room temperature under equilibrium conditions, the composition of the Sn-rich phase follows the a-b solvus line shown in the phase diagram. Under non-equilibrium, air cooling, the Sn-rich phase may follow the a-c line. With time at room temperature, two phenomena will occur:

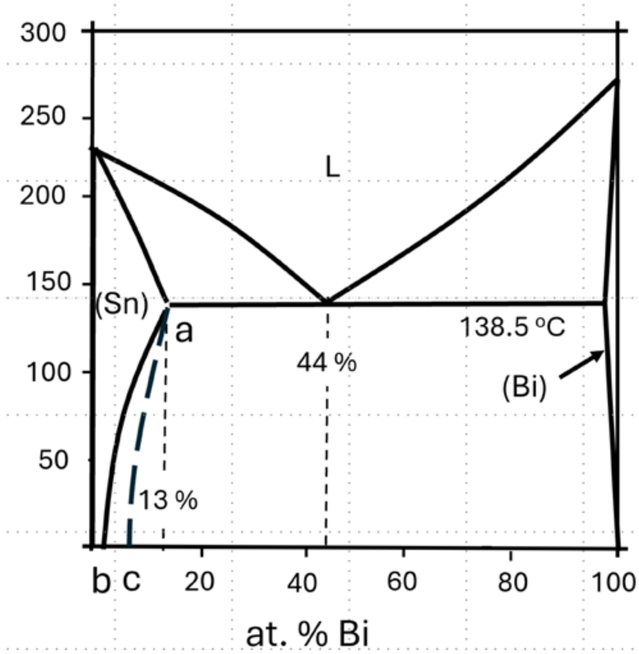


Figure 1: Su-Bi equilibrium phase diagram.

1. The composition of the Sn-rich phase will move from point c to the thermodynamic equilibrium composition point b.
2. The microstructure will coarsen with the larger Bi-rich particles growing at the expense of the smaller Bi-rich particles.

The coarsening of the microstructure, also known as Ostwald ripening, is explained in Figure 2[3]. The Bi concentration in the Sn-rich

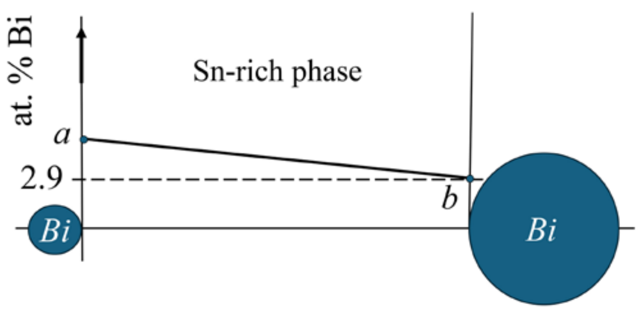


Figure 2: Illustration of Ostwald ripening in Sn-Bi alloy. The a-b solid line represents the Bi concentration profile in the Sn-rich phase between the Bi-rich particles. The gradient of the Bi composition represented by the a-b line drives the diffusion of Bi atoms from the smaller to the larger Bi-rich particles. The dotted line at 2.9 at.% represents the composition of the Sn-rich phase when, with aging, the Bi-rich particles coarsen to larger sizes.

phase adjacent to the smaller Bi-rich particles is higher that adjacent to the larger Bi-rich particles, where it is estimated to be 2.9 at%. This Bi concentration gradient in the Sn-rich phase drives the Bi atoms to diffuse from the smaller Bi-rich particles to the larger Bi-rich particles. In addition, since the Bi content of the Sn-rich phase adjacent to the larger Bi-rich particles is lower, the microstructural coarsening will result in a lowering of the equilibrium Bi content of the Sn-rich phase. So, the two phenomena, one being the Sn-Bi alloy composition moving towards thermodynamic equilibrium from point c to point b on the phase diagram and the other being the microstructural coarsening, both result in the lowering of the Bi content of the Sn-rich phase and therefore, as per the lever rule, the overall increase in the volume fraction of the Bi-rich phase in the alloy. Room temperature aging thus increases the volume fraction of the Bi-rich phase. A recent paper reported a decrease in the volume fraction of the Bi-rich phase when a Sn-Bi alloy was aged at room temperature[4]. The explanation of this error can be found in the next section.

Temporal Decrease of the Area Fraction of the Bi-rich Phase at Free Surfaces

The Bi-rich phase area fraction at free surfaces has been experimentally observed to decrease with time even at room temperature. This decrease is not due to the Bi-rich phase particles dissolving into the Sn-rich matrix but due to the Bi-rich phase particles moving away from the free surfaces into the bulk of the alloy. This can only happen if the surface tension between the Bi-rich phase and air is higher than that between the Sn-rich phase and air. Figure 3a shows the surface of a planar solder specimen of eutectic Sn-Bi alloy flanked by copper on both sides having been subjected to high electric currents at high temperatures.

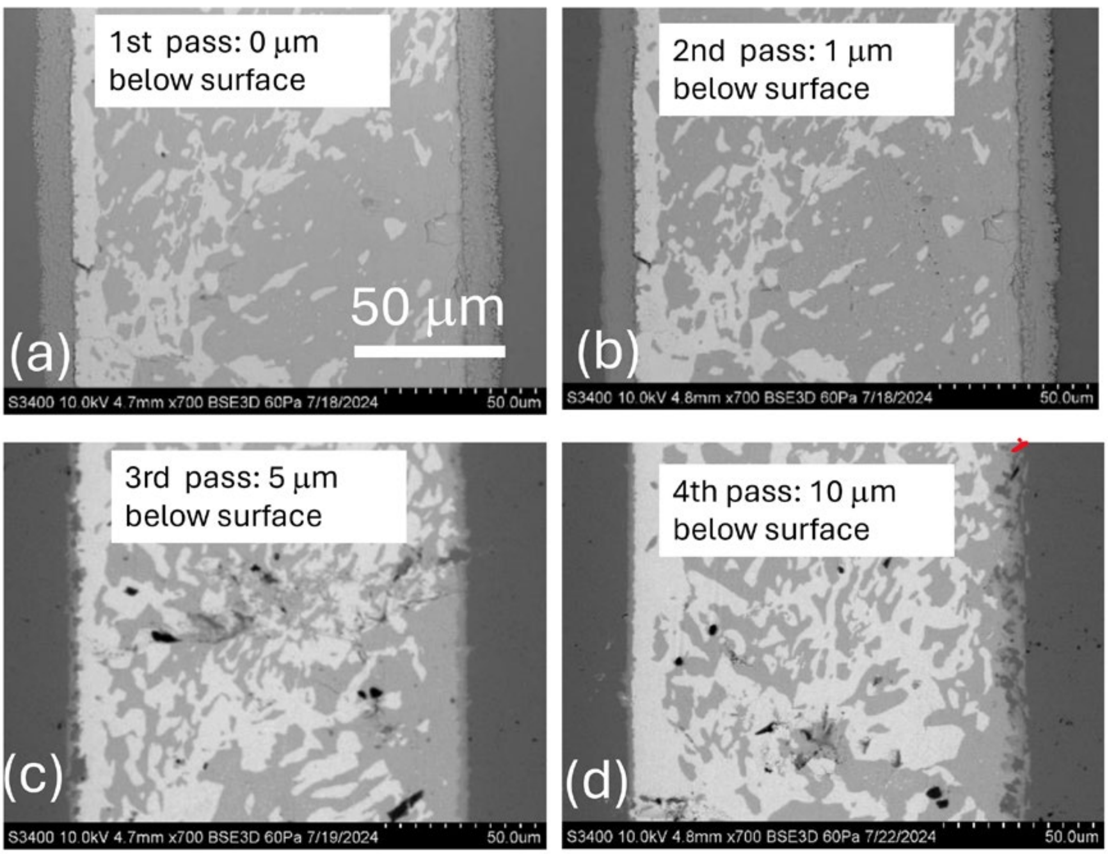


Figure 3: a) Scanning electron micrograph of a planar solder specimen that was subjected to 3 Amp current at 100, 90, 80, 70 and 60°C over a total of 52 days; b) Surface 1 µm below the surface shown in (a); c) Surface 5 µm below the surface shown in (a); d) Surface 10 µm below the surface shown in (a).

The other micrographs in Figure 3 show what is beneath the top surface of the specimen. Figures 3b, 3c, and 3d are micrographs of the surfaces revealed when 1.5 and 10 µm of the solder was ground and polished away from the surface. The temporal decrease of the area fraction of the Bi-rich phase at the surface of eutectic alloy specimens occurs even though, as explained in the previous section, the overall volume fraction of the Bi-rich phase can only increase with time as the alloy tends towards thermodynamic equilibrium.

This observation of the area fraction of the Bi-rich phase decreasing at the free surface of Sn-Bi solder points to the care one must take in drawing conclusions from observing the microstructure of Sn-Bi alloy at the polished surface. The combination of high homologous temperature of Sn-Bi alloys and the supposedly high surface tension of the interface between Bi-rich phase and air compared to that of Sn-rich phase and air causes the Bi-rich phase particles to move away from the free surface of the alloy. Over time, even at temperature as low as the room temperature, the surfaces of Sn-Bi alloys become somewhat depleted of Bi-rich phase particles. But it should be noted and emphasized that thermodynamics does not allow the decrease of the Bi-rich phase volume fraction due to aging at room temperature. The Bi-rich phase volume fraction can only increase when a Sn-Bi alloy is aged at room temperature.

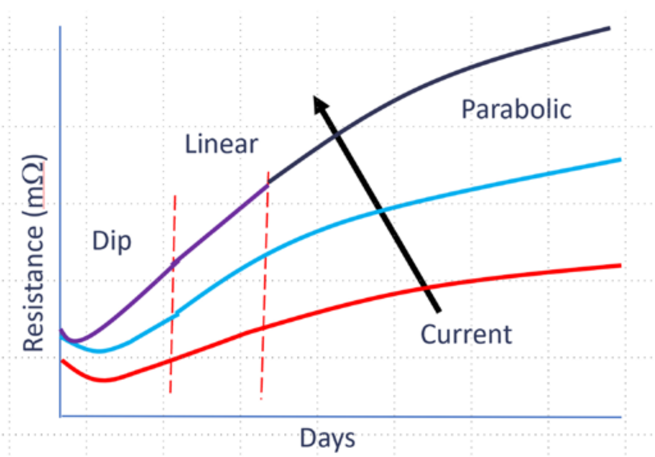


Figure 4: Stages of Sn-Bi electromigration.

Stages of Sn-Bi Alloy Electromigration

The stages of Sn-Bi electromigration are illustrated in Figure 4[5]. During the first stage, the alloy electrical resistance takes a dip because of two phenomena, both of which were explained earlier. One is the microstructural evolution towards equilibrium from point b on the phase diagram to point c (Figure 1), decreasing the Bi content of the Sn-rich phase matrix and therefore making it more conductive. The other is the microstructural coarsening, shown in Figure 5, with the larger Bi-rich phase particles growing at the expense of the finer Bi-rich phase particles, making the alloy electrically more conductive by providing straighter paths for the electron to traverse from cathode side to the anode.

The second stage—the linear stage—of Sn-Bi electromigration occurs when the Bi-rich phase particles' growth mechanism subsides, and Bi begins to form a near continuous layer at the anode end of the solder joint structure, dominating the electrical resistance of the solder joint. During this stage there is a near-original microstructure feeding the Bi atoms accumulating at the anode end. The steady supply of Bi atoms results in a near-linear rise in the alloy resistance with time. The third stage, the parabolic stage, occurs when the cathode region feeding Bi atoms to the anode side gets

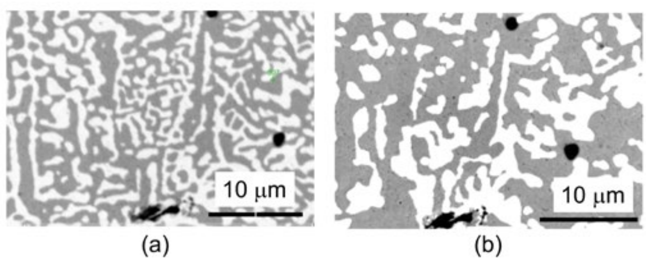


Figure 5: During the first stage of electromigration, the microstructure coarsens, from that shown in (a) to that shown in (b). In the coarsened microstructure, the electron current finds a straighter path in the more conductive Sn-rich phase matrix to flow from the cathode to the anode. The solder joint resistance, thus, takes a dip in the first stage of electromigration.

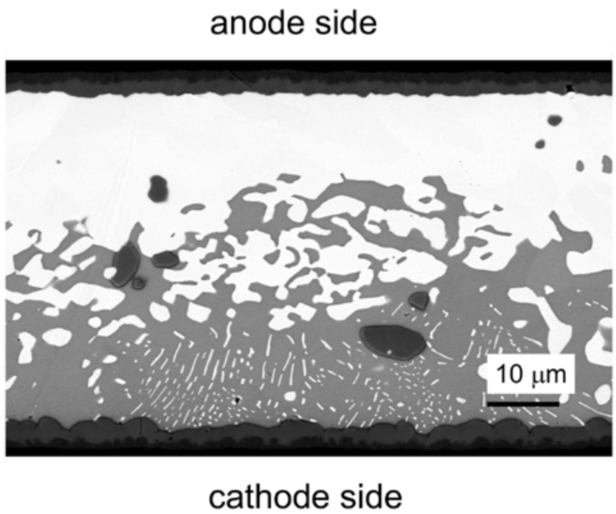


Figure 6: The third stage is when a continuous layer of Bi-rich phase has formed on the anode side and has grown with time to such an extent that the region near the cathode feeding Bi to the anode has been quite depleted of the Bi-rich phase. The lighter phase in the scanning electron micrograph is the Bi-rich phase that has accumulated at the anode side of the solder.

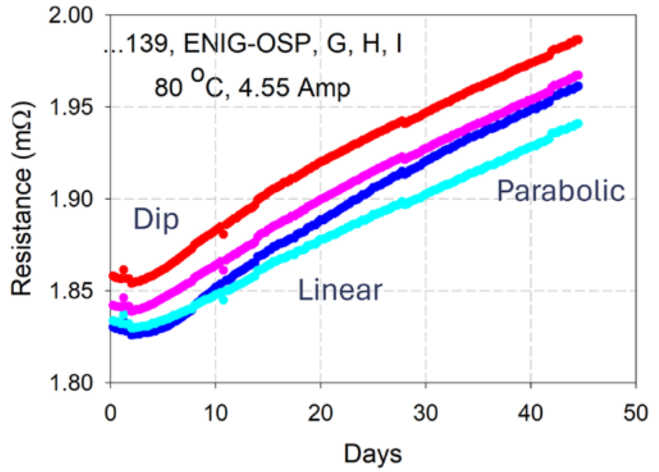


Figure 7: Resistance change of various specimens at a constant current density and constant temperature showing the three stages of electromigration.

quite depleted of the Bi-rich phase particles as shown in Figure 6. The three stages of electromigration are shown graphically at constant current density and temperature for some similar solder joint specimens in Figure 7.

Transient Behavior of Solder Joint Resistance During Step Change in Temperature and/or Current

Figure 8a shows a typical behavior of a solder joint resistance during step changes in temperature and/or current. As an example, at the one-hour mark, the current through a solder joint in an oven at 60°C is suddenly raised from 0.1 to 1.47 A. There is a steep rise in the sol-

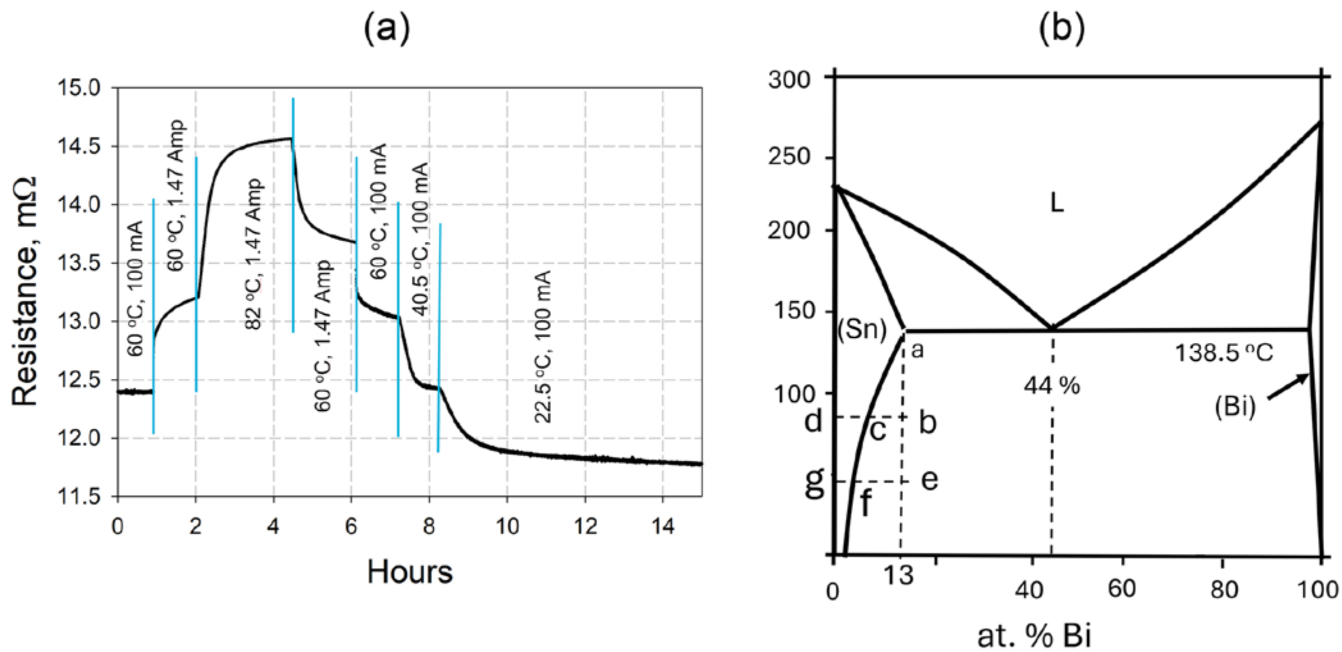


Figure 8: a) Transient behavior of eutectic Sn-Bi solder resistance due to the shape of the solvus line; b) Sn-Bi equilibrium phase diagram.

der joint resistance due to increased joule heating raising the solder temperature followed by a gradual parabolic increase in resistance that can be explained by the shape of the solvus line on the Sn-rich side of the phase diagram. Let us consider the room temperature to correspond to the g-f-e line and a higher temperature to the d-c-b line on the phase diagram of Figure 8b. Notice that the Bi content of the Sn-rich phase increases from point f to point c on the phase diagram because of the temperature rise, thereby correspondingly increasing the solder joint resistance. The increase in the Bi content of the Sn-rich phase is diffusion controlled and therefore time delayed, that is the increase of the Bi content of the Sn-rich phase is not instantaneous but gradual as the Bi-rich phase particles dispersed in the Sn-rich phase dissolve with time into the Sn-rich phase. The Bi atoms given off by the Bi-rich phase diffuse with time in the Sn-rich phase matrix, raising the resistance of the Sn-rich phase matrix in which the electric current prefers to flow. The resistance of the solder thus rises with time in a parabolic fashion when there is step increase in current flowing through the solder joint. The same resistance increase phenomenon will occur if there is a step rise in the solder temperature due to the raising of the oven temperature as shown in Figure 8a at the two-hour mark.

When the temperature of a solder joint is decreased in a step manner as shown at the 4.5-, six-, seven- and eight-hour marks, due to either the decrease in joule heating or the lowering of the oven temperature, the solder joint resistance has a sharp drop due to a decrease in solder joint temperature followed by a gradual parabolic decrease in resistance due to the decrease of the Bi content of the Sn-rich phase. If the higher temperature corresponds to the line d-c-b and the lower temperature to the line g-f-e in the phase diagram of Figure 8b, the lowering of temperature lowers the equilibrium solubility of Bi in the Sn-rich phase from point c to point f. Bismuth atoms in the Sn-rich phase diffuse to and deposit on the Bi-rich phase particles dispersed in the Sn-rich phase, thus gradually lowering the Bi content of the Sn-rich phase and the overall resistance of the solder joint.

The above-discussed gradual change in solder joint resistance due to step change in current and/or solder temperature was reported in 2024 by a team from Purdue University that stated that the cause for such a change was yet to be determined[6].

Black's Equation vs. the Physics Approach

A common engineering approach to determining the electromigration life of solder joints is to determine the electromigration lives of a statistically significant number of similar solder joints at a contact temperature and electric current density and plot the times to failure on a Weibull plot. The process can be repeated at other temperatures and current densities and the mean-time-to-failure (MTTF) results used to calculate the constants in the following Black's equation:

$$\mathrm{MTTF}^{-1} = Bj^n \exp\left[-\frac{Q}{kT}\right]$$

where *B* is a constant, j is the electric current density, *Q* is the activation energy for electromigration, *k* is the Boltzmann constant and T is the temperature in kelvin. Once the constants in the equation are known, Black's equation can be used to predict the electromigration lifetime under any current density and temperature condition.

Another approach to determining the electromigration life of solder joints is to take a purely physics approach based on the Nernst-Einstein equation:

$$v = \frac{DF}{kT}$$

where *D* is the diffusion coefficient of the electromigrating ions, and *F* is the electrostatic force on the ions. The equation can be transformed into a more useful form for the purposes of electromigration studies as follows:

$$v = \frac{DF}{kT} = \frac{DZ^{*}e\rho j}{kT} = \frac{D_0 Z^{*}e\rho j}{kT} e^{-\frac{Q}{kT}}$$

$$\frac{vT}{j} = \frac{D_0 Z^{*}e\rho}{k} e^{-\frac{Q}{kT}} = C_1 e^{-\frac{Q}{kT}}$$

$$v \propto \frac{dR}{dt}$$

$$\frac{\frac{dR}{dt}T}{j} = C_2 e^{-\frac{Q}{kT}}$$

In the above derivation, Z^* is the electromigration effective valence, e is the electron charge, r the electrical conductivity, v is the ion drift velocity, D_o is the diffusion pre-exponential constant, R is the solder joint resistance, t is duration of electromigration current stressing, and the other terms have their usual meaning. One advantage of this approach involving the Nernst-Einstein equation is that one solder joint specimen can be stressed at various temperatures and current densities to obtain the value of constants C_2 and Q. Using these constants in the last of the above equations, the electromigration lifetime can be predicted under any current density and temperature condition. An example of the data needed for this approach is shown in Figure 7, in which electromigration data (solder joint resistance versus time) were collected at a constant current at various solder joint temperatures obtained by changing the oven temperature. The test would need to be repeated at various other currents to obtain an Arrhenius plot, an example of which is shown in Figure 8.

Joule Heating During Electromigration Testing

In electromigration testing, it is important to pay attention to joule heating, especially when the electric currents are high. Given the small size of the solder joints and the lack of line-of-sight optical access, there is no direct way of accurately measuring the solder joint temperature such as by using infrared means or thermocouples. To get accurate temperature measurements, one must resort to indirect means such as using the temperature coefficient of resistivity of the solder joint. The solder joint resistance is measured at a very low current over a range of temperature extending from room temperature to that of the oven during the electromigration test and the resistance versus temperature curve plotted. The resistance measurement current should be low enough to cause negligible joule heating. The resistance of the solder joint during the electromigration test is then measured under the electromigration test conditions of high temperature and current and graphically obtaining the solder joint temperature as illustrated in Figure 9. During the electromigration testing, the rise in the solder joint resistance over time will cause some increase in joule heating.

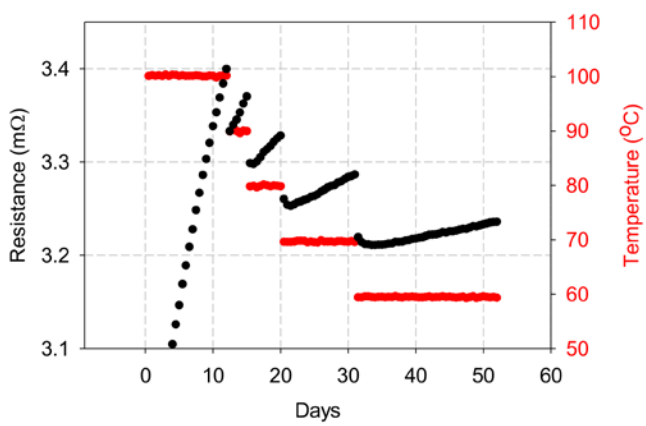


Figure 9: Resistance change on the same specimen at a constant current density at various temperatures.

Planar Solder Test Specimens

Ball-grid array (BGA), bottom-terminated component (BTC), C4, and other solder joints in actual applications are not convenient for electromigration research though at some point in the design cycle they do need to be

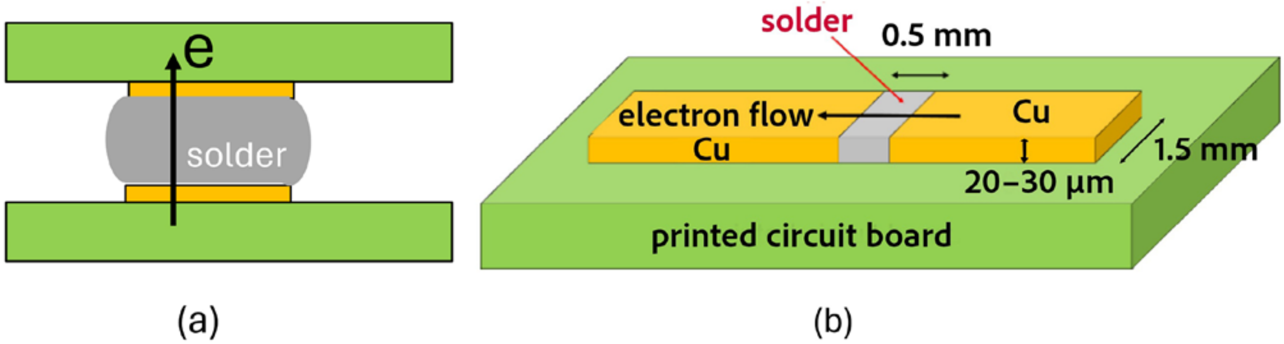


Figure 10: a) BGA solder joint between a PCB and a component; b) planar solder joint on a PCB.

qualified for use in actual applications. Electromigration research can be better conducted on planar-solder joints[9]. Planar solder joints are easy to fabricate in a typically equipped metallurgical laboratory in a matter of a day or two and electromigration can be observed on a real time basis if the test specimen is mounted on a hot stage in an electron microscope. Figure 10 compares the geometry of a BGA to a planar solder joint and Figure 11 is a top-down view of a planar test specimen with five solder joints of various lengths in the range 0.18–0.41 mm. The details of the planar solder joint fabrication have been presented earlier[10–13]. A detailed study comparing the eutectic Sn-Bi electromigration in the planar and the BTC geometry concluded that the electromigration behavior was similar in the two cases as shown in the Arrhenius plots of Figures 12 and 14. Therefore, planar solder joints can be conveniently used to study the electromigration propensity of solder metallurgies to obtain a short list of alloys for further testing in actual applications.

Conclusions

Phase diagrams are very useful in predicting solder alloy behavior once one recognizes that they are for equilibrium condition and under non-equilibrium cooling conditions, the microstructures are not quite what the phase diagram predicts. The high homologous temperatures of Sn-Bi alloys is the reason they tend toward thermodynamically equilibrium microstructures quite quickly even at temperatures as low as room temperature. The volume fraction of the Bi-rich phase in a cast Sn-Bi alloy will always increase when aged at room temperature, though observations of the surface of

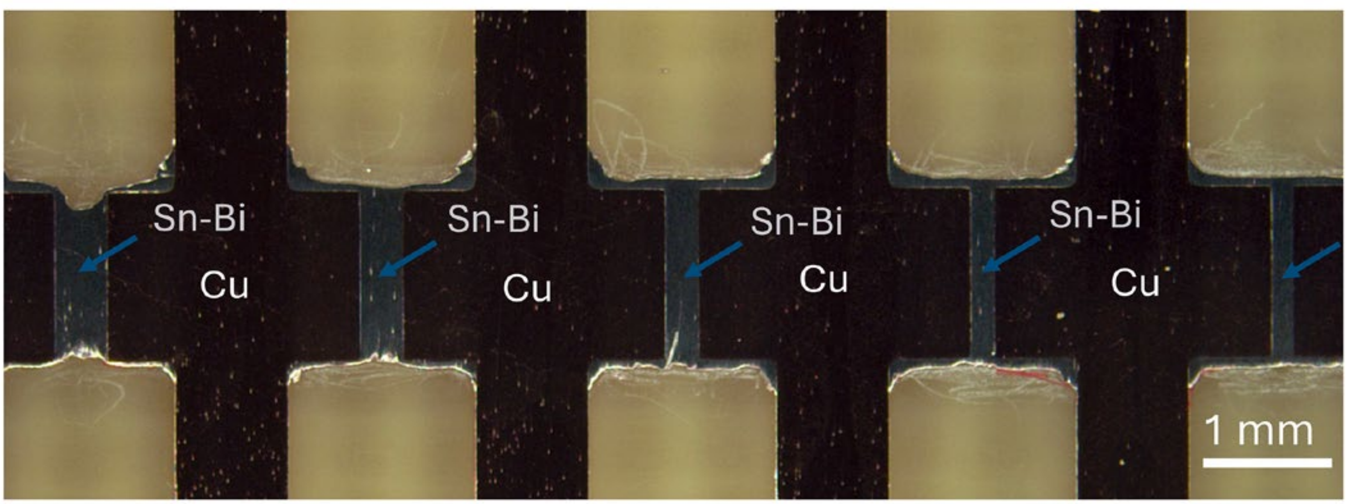


Figure 11: Five planar solder joints of various length in series.
Source: IBM Corporation, *Presented at SMTA International 2023*.

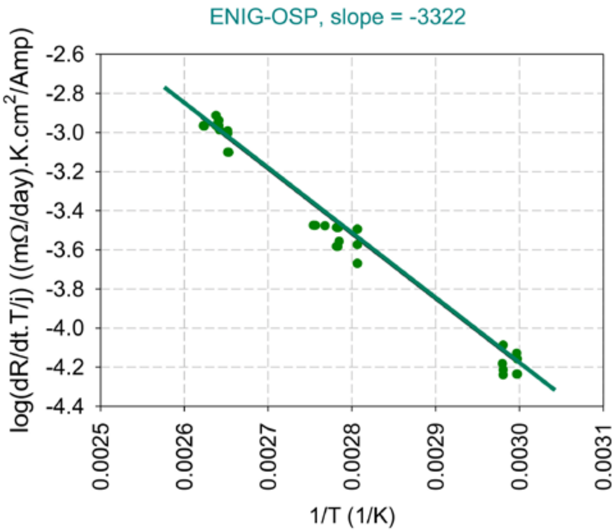


Figure 12: Arrhenius plot of a solder alloy that can be used to predict the electromigration lifetime of a solder joint using that alloy under any temperature and current density condition.

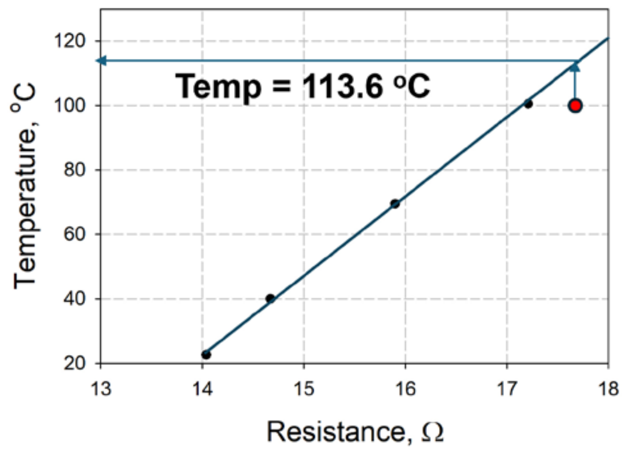


Figure 13: The linear curve was obtained by measuring the joint resistance at various chamber temperatures at very low currents at which the joule heating was known to be negligible. From the resistance measured when the oven was set at the temperature of interest for the electromigration study (100°C in this example) and the current raised to the current of interest of the electromigration study, the joule heating of the solder joint could be determined as illustrated in this figure.

a polished specimen will show decreasing area fraction of the Bi-rich phase as the alloy ages. This seeming anomaly is because the Bi-rich phase with time tends to sink into the bulk, away from the polished surface.

The transient resistance change of Sn-Bi solder as a result of a step change of current and/or temperature is due to the Bi concentration in the Sn-rich phase adjusting to the thermodynamic equilibrium value for the new temperature of the solder as per the solvus line of the Sn-Bi phase diagram.

Electromigration experiments can be designed to collect data suitable for determining the constants in the Black's equation or for determining the Arrhenius plot using the Nernst-Einstein equation. The former takes the Weibull approach requiring many specimens to obtain the mean time to failure under many different conditions to calculate the constants in the Black's equation; whereas, for the latter, one specimen is enough to obtain an Arrhenius plot that can predict the electromigration life of a solder alloy under any application condition, though more specimens would improve the lifetime prediction.

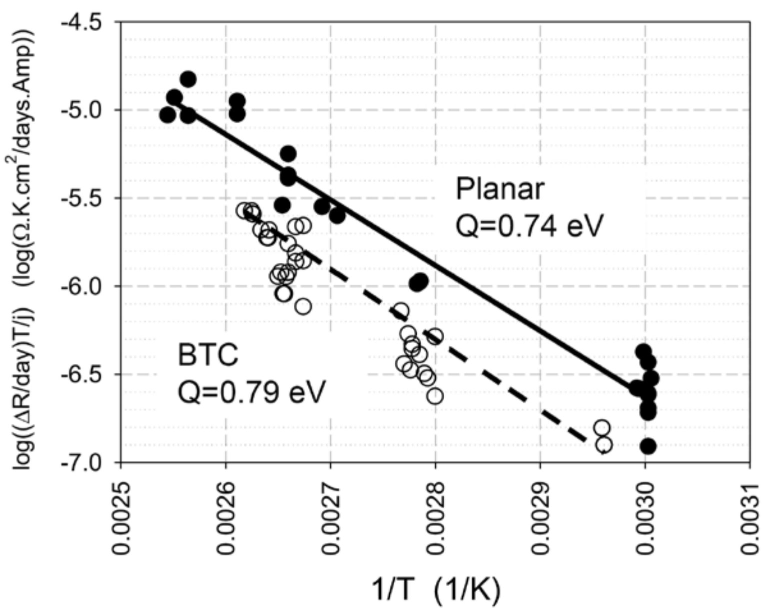


Figure 14: Arrhenius plot of Bi electromigration in eutectic Sn-Bi solder using the planar and the BTC approach.

The solder joint temperature rise due to joule heating, that occurs under high electric current condition, must be determined for the electromigration rates to be useful in predicting the electromigration life of a solder alloy under application conditions. The coefficient of resistance approach is a convenient way of accurately determining the temperature rise of a solder joint due to joule heating.

Electromigration research on solder joints such as BGA, BTC, C4, and others requires expensive fabrication resources and the microstructural changes cannot be monitored during the electromigration test; whereas, the planar solder joints can be quite readily fabricated in a typical metallurgical laboratory, and the microstructural changes can be monitored on a real time basis under a microscope. The results of the two approaches have been found to be similar. **SMT007**

References

1. "Low temperature interconnects in 1st level packaging and its challenges," by C. Arvin et al., 2022 International Conference on Electronics Packaging (ICEP), Sapporo, Japan, 2022.

2. "Binary alloy phase diagrams," by T Massalski, ASM, International, Materials Park, Ohio, 1986.

3. "The growth of dispersed precipitates in solutions," by G.W. Grenwood, Acta Metallurgica, Volume 4, Issue 3, May 1956, pp. 243-248.

4. "Microstructural coarsening and mechanical properties of eutectic Sn-58Bi solder joint during aging," by F. Wang., A. Luktuke, N. Chawla, Journal of Electronic Materials, (2021).

5. "Electromigration in Eutectic Tin-Bismuth Bottom Terminated Components Solder Joints," by P. Singh, et al., The 2024 IEEE 74th Electronic Components and Technology Conference (ECTC), May 2024.

6. "In-line test structure and non-destructive characterization of electromigration-driven phase evolution in microscale solder joints," by C. Jois, P-E Chou and G. Subbarayan, Journal of Electronic Materials, 2024.

7. "Reliability and Failure of Electronic Materials and Devices," by M. Ohring, Academic Press, 1998, page 191.

8. "Reliability and Failure of Electronic Materials and Devices," by M. Ohring, Academic Press, 1998, page 276.

9. "Tracer Study of Diffusion and Electromigration in Thin Tin Films," by P. Singh, Journal of Applied Physics, 1984.

10. "A novel setup for studing electromigration," by P. Singh, et al, Pan Pacific Microelectronics Symposium, Oahu, Hawaii, 2022.

11. "In Situ Study of Microstructureal Evolution During Electromigration in Eutectic Tin Bismuth Solder," by P. Singh et al, SMTA Pan Pacific Strategic Electronics Symposium, 2024.

12. "Electromigration in tin-bismuth planar solderjoints," by P. Singh et al., 2023 International Conference on Electronics Packaging (ICEP), Kumamoto, Japan, 2023.

13. "Comparison of electromigration in tin-bismuth planar and bottom terminated component solder joints," P. Singh, et al., SMTA International, Minneapolis, Minnesota, October 2023.

Prabjit Singh works for IBM Corporation in Poughkeepsie, New York.

Raiyo Aspandiar works for Intel Corporation in Hillsboro, Oregon.

Haley Fu works for iNEMI in Shanghai, China.

Lavanya Ashok Swaminathan works for Intel Corporation in Hillsboro, Oregon (no image available).

TOP TEN

EDITOR'S PICKS

From Chips to Systems: Building a Resilient Electronics Manufacturing Base in the EU

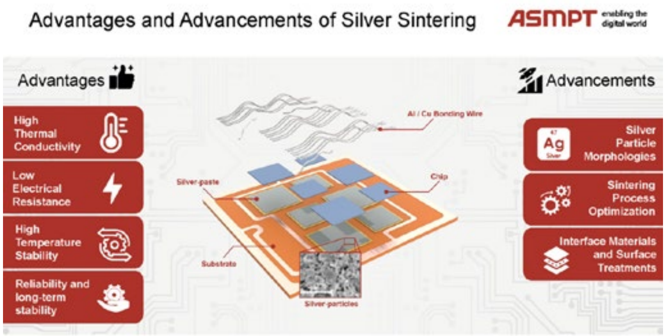
As industries increasingly depend on electronics for innovation, the role of electronics manufacturers as key facilitators of Europe's digital and environmental transitions increases significantly. The EU's Chips Act signifies a strategic commitment to bolstering a key segment of the electronics sector.

IPC Releases July 2024 Global Sentiment of the Electronics Supply Chain Report

Sentiment among electronics manufacturers slipped in July, dropping to the lowest level in a year. Despite the decline, sentiment remains above its long-term average according to IPC's July Sentiment of the Global Electronics Manufacturing Supply Chain Report.

ASMPT Makes High-strength and High-conductivity Joints Possible with Silver Sintering

ASMPT presents an innovative die and module bonding platform that can be used in a variety of ways for producing power modules.



The Knowledge Base: The Value of Industry Certifications

Industry certifications provide several key advantages for individuals. First, they enhance employability. In a competitive job market, having certifications can set candidates apart from others, demonstrating a commitment to the profession and a verified level of competence. Employers often look for certified professionals because these credentials assure a certain standard of knowledge and skill.

Global Sourcing Spotlight: More Than Just Saving Money

Global sourcing is primarily driven by cost reduction. Different countries have varying cost structures because of differences in labor costs, raw material availability, and economic conditions. By sourcing from countries where production costs are lower, companies can achieve significant savings.

SMTA Elects New Board Members

Martin Anselm

The SMTA is pleased to announce its election results for the Global Board of Directors for the term beginning Oct. 20, 2024. The SMTA will bid a fond farewell to departing President, Martin Anselm, Ph.D., Rochester Institute of Technology.

Real Time with... THECA 2024: Workforce Development Critical to Southeast Asia PCB Industry

Sydney Xiao

In this interview, IPC Asia President Sydney Xiao emphasizes the importance of workforce training and keeping ahead of the staffing needs in Thailand and the rest of Southeast Asia.

SMTA International Technical Conference Program Announced

The SMTA announced that the technical program of their annual conference, SMTA International, is finalized and registration is now open. The event will be held Oct. 20–24, 2024 at the Donald E. Stephens Convention Center in Rosemont, IL, USA.

My Drop Shock Testing Experiments to Determine Reliability in PCBs

Dropping an electronic device can cause impact fatigue in solder materials, which could lead to joint failures. Even if the solder joint survives the impact, other failures, such as pad cratering and copper trace cracks can occur. While dropping a device once may not cause any failures, repeated drops can cause cumulative damage and eventually rupture solder joints or other assembly materials. That's why drop shock testing is a useful experimental technique for designing PCBs for reliability.

BTU International, Repstronics to Enhance Presence in Mexico and Central America

BTU announces a new partnership with Repstronics. This strategic alliance aims to strengthen BTU International's presence in Mexico and central America, leveraging Repstronics' extensive footprint and regional relationships.

For the latest news and information, visit **SMT007.com**

Career Opportunities

PCB Manufacturing Technician

Join the Team at Accurate Circuit Engineering!

Located in Santa Ana, California, Accurate Circuit Engineering (ACE) delivers high-quality PCB solutions with a focus on innovation and precision.

Role: Accurate Circuit Engineering is looking for detail-oriented PCB Manufacturing Technicians for all areas in PCB manufacturing. Responsibilities include operating manufacturing equipment, performing quality checks, and documenting production data.

Qualifications:
- High school diploma or equivalent; technical training preferred
- Experience in PCB manufacturing or electronics assembly a plus
- Strong attention to detail and ability to follow instructions
- Familiarity with PCB manufacturing and testing tools is advantageous

What ACE Offers:
- Competitive wages and benefits
- Career growth opportunities
- Supportive work environment with comprehensive training

To apply, send your resume and cover letter to sales@ace-pcb.com with the subject “PCB Manufacturing Technician Application.”

Accurate Circuit Engineering is an equal-opportunity employer and values diversity in the workplace.

apply now

Capital Equipment Sales Go-getter

all4-PCB is a well-established distributor that represents a wide range of manufacturing equipment for the printed circuit board manufacturing and chemical milling industries, as well as other high-tech markets.

Looking for a self-driven, dynamic, high energy, results-focused, well-organized sales personality—a closer. Ideal candidate is a fast learner, can retain a lot of information, and understands that selling machines is about selling a process solution to a customer.

A sales-driven personality with a high level of interest in and aptitude for learning about technology can be a star performer. The target market wants consultative selling addressing customer needs.

The sales process includes all means of customer prospecting with up to 50% travel expected. Generating quotations, clarifying commercial and technical details/expectations between customers and the international equipment supply partners, negotiations, sales closing, and the required follow-up work are all part of this role. The sales territory is the U.S. & Canada.

This technical sales representative position can be located in either all4-PCB’s Glendale, Calif., office or remotely in an area with a higher concentration of potential accounts.

This role is ideal for applicants interested in a position that provides extensive international exposure, a generous commission system, and freedom to get things done in a small, yet professional, business environment.

apply now

Career Opportunities

schmoll america

Service Engineer: Chicago

Join the Schmoll America Team as a Service Engineer—Where Innovation Meets Customer Excellence! Are you a technical mastermind with a passion for solving complex problems and delivering exceptional customer experiences? Look no further than Schmoll America!

We're looking for engineers to work in the Chicagoland area.

As a Service Engineer, you'll be the driving force behind our customers' success, providing top-notch technical support and maintenance services for our PCB industry-leading equipment.

What you'll do:

- Install, commission, and maintain Schmoll equipment at customer sites
- Troubleshoot and repair equipment with ease and precision
- Provide technical training and tailored applications solutions to customers

What we offer:

- A dynamic and supportive work environment where your voice matters
- Opportunities for professional growth and development in a cutting-edge industry
- A competitive salary and benefits package
- The satisfaction of knowing you're making a real difference in our customers' lives

What we're looking for:

- Engineering degree preferred
- 3+ years of experience in an engineering role
- Strong technical knowledge of electrical and mechanical systems
- Excellent problem-solving and analytical skills
- Willingness to travel (up to 75%) to customer sites and HQ in Germany

If you're a motivated professional looking for a challenging and rewarding role, we want to hear from you! Please submit your resume and cover letter to HR@SchmollAmerica.com.

apply now

TAIYO AMERICA

Europe Technical Sales Engineer

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

PRIMARY FUNCTION:

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

ESSENTIAL DUTIES:

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

QUALIFICATIONS / SKILLS:

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

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

apply now

Career Opportunities

The Test Connection, Inc.

Sr. Test Engineer (STE-MD)

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

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

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

apply now

The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

Career Opportunities

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 Utah/Colorado, and Northern California territories. 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

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

Career Opportunities

Career Opportunities

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

CAD/CAM Engineer

Summary of Functions

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

Essential Duties and Responsibilities

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

Organizational Relationship

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

Qualifications

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

Physical Demands

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

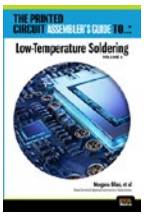
apply now

Educational Resources

PODCAST! Designing for Reality with ASC Sunstone

VP/manager and published author Matt Stevenson talks about the specifics that can affect your circuit board during the manufacturing process. Part tutorial, part tips and tricks, Stevenson details the interrelationships between design, fabrication, yields and cost optimization.

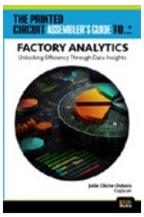
I-007eBooks



Low Temperature Soldering, Volume 2

by Morgana Ribas, Ph.D., et al., MacDermid Alpha Electronics Solutions

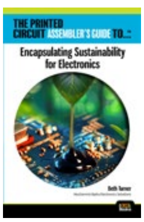
Since the first volume of The Printed Circuit Assembler's Guide to...™ Low-temperature Soldering was published over five years ago, considerable changes have occurred in the low-temperature soldering landscape. Here, the authors review the evolution of solder alloys from traditional eutectic SnBi solder to the fourth-generation HRL3 low-temperature solders. Read about innovations and challenges for achieving optimal processing with low-temp solder pastes, as well as what's next in LTS.



Factory Analytics: Unlocking Efficiency Through Data Insights

by Julie Cliche-Dubois, Cogiscan

Using and understanding factory analytics is the future for electronics manufacturers. Those who strategically prioritize analytics and properly leverage the insights generated throughout their entire operation systematically will stand the test of time. In this fast-changing, deeply competitive, and margin-tight industry, factory analytics can be the key to unlocking untapped improvements to guarantee a thriving business.



Encapsulating Sustainability for Electronics

by Beth Turner, MacDermid Alpha Electronics Solutions

This is a guide to encapsulation resins and their use in ruggedizing electronics. Learn about aspects such as their chemistry, application, and relevant test methods in different industries. The book also discusses the growing demand for sustainable solutions in the market and highlights examples of bio-based resins and the demand from emerging technologies.

Process Control

by Chris Hunt and Graham K. Naisbitt, GEN3

In this book, the authors examine the role of SEC test and how it is used in maintaining process control and support for objective evidence (OE.) Issues, including solution choices, solution sensitivities, and test duration are explored.

Manufacturing Driven Design

by Max Clark, Siemens

This book introduces a new process workflow for optimizing your design called Manufacturing Driven Design (MDD) and is a distinct evolution from DFM. Manufacturing certainly plays a critical role in this process change, and manufacturers do certainly benefit from the improved process, but it is design teams that ultimately own their overall product workflow; they are the ones who need to drive this shift. Get empowered now!

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

MANAGING EDITOR: **NOLAN JOHNSON**
(971) 202-0110; nolan@iconnect007.com

ASSOCIATE EDITOR: **MICHELLE TE**
michelle@iconnect007.com

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

TECHNICAL EDITOR: **PATTY GOLDMAN**

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

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

BUSINESS DEVELOPMENT MANAGER: **BARB HOCKADAY**
(916) 365-1727; barb@iconnect007.com

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

DIGITAL MARKETING MANAGER: **BRITTANY MARTIN**
brittany@iconnect007.com

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

CREATIVE TECHNOLOGIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **COURTESY OF MYCRONIC**

SMT007 MAGAZINE

SMT007 MAGAZINE®
is published by IPC Publishing Group, Inc.,
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

September 2024, Volume 39, Number 9
SMT007 MAGAZINE is published monthly,
by IPC Publishing Group, Inc., dba I-Connect007.

ADVERTISER INDEX

AEMS 2024 41
ASC Sunstone 17
Akrometrix 67
Alpha Circuit 29
AXXON | Mycronic 73
Blackfox Training Institute 57
CEE PCB 7
EPTAC 11
Flexible Circuit Technologies 23
GEN3 37
I-007 eBooks 3, 25, 65
IPC 49, 85
IPC Community 77
Koh Young 45
KYZEN 69, 71
Linkage Technologies 53
On the Line with 2
P Kay Metal 33
Prototron Circuits 43
ROCKA 21
SMTA 27
SPEA 15
Technica USA 5
The Test Connection 81
US Circuit 61