I-Connect007 GOOD FOR THE INDUSTRY

ENGINEERING SOLUTIONS FOR PCB MANUFACTURING

SMT® MAGAZINE

DECEMBER 2017

Solder Paste Printing

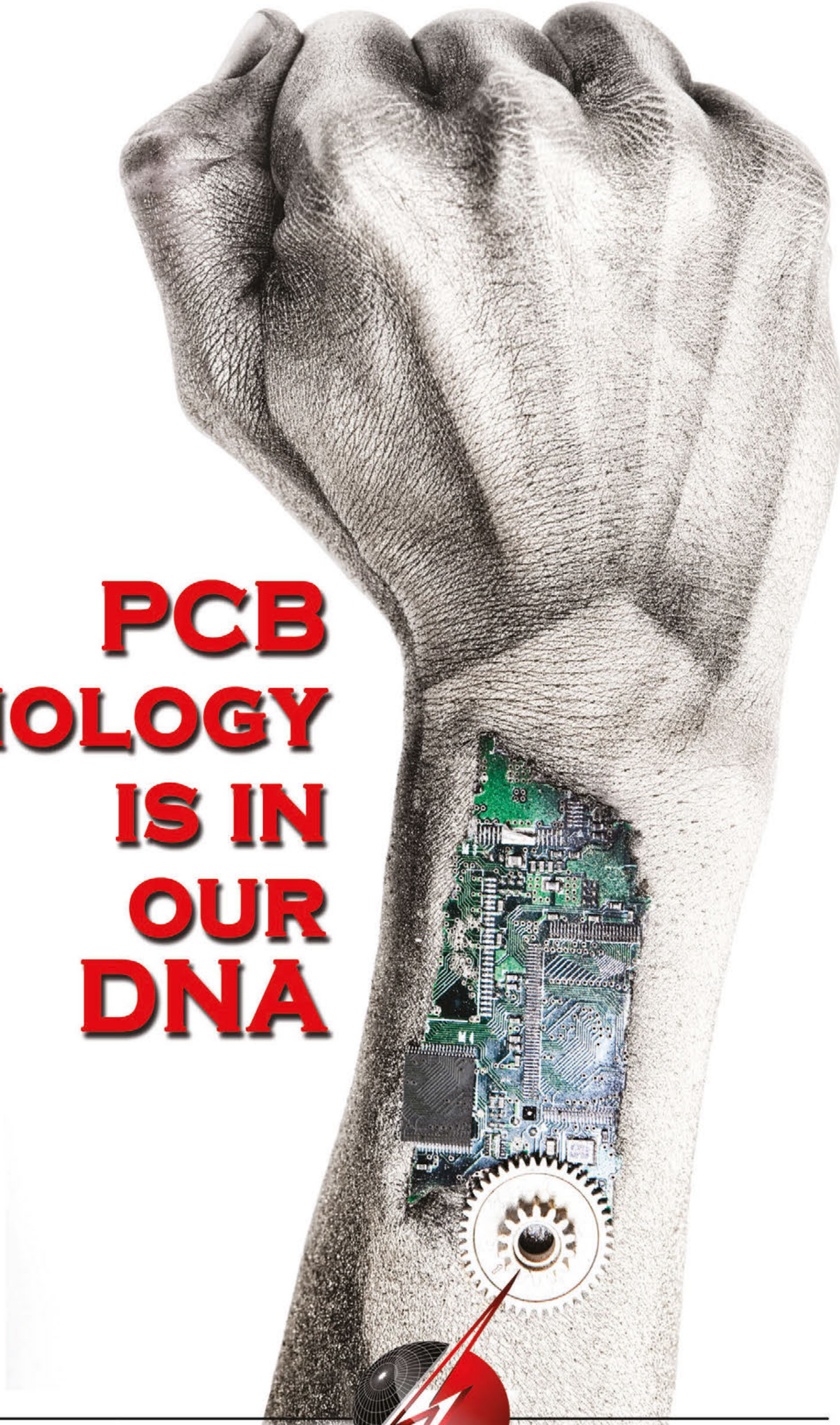
Screening Out the Defects

FREE BOOK DOWNLOAD

I-007e Books

THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO... Conformal Coatings for Harsh Environments


"A comfortable approach to the selection, implementation, and testing of protective coating processes."
Dr. Helmut Schweigart
Technology Development Head,
Zestron Europe
This FREE micro eBook, written by Phil Kinner, Electrolube's global business and technical director, will benefit anyone in the electronics industry who wants a better understanding of conformal coatings.
Free Download
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
Conformal Coatings for Harsh Environments
I-007e Books

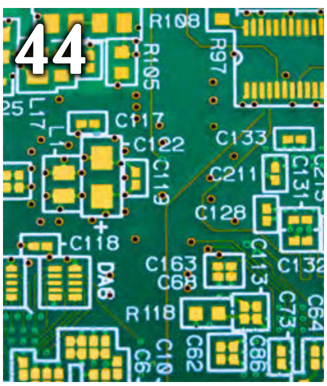


SMT MAGAZINE

December 2017 • Featured Content

Solder Paste Printing: Screening Out the Defects

The solder paste printing process accounts for almost 70% of PCB assembly defects. This month's issue of *SMT Magazine* looks into the critical factors causing these issues, and features strategies, tips and tricks to help assemblers improve the yield and quality in their solder paste printing operation.



The SMT Source™

Latest News About Manncorp SMT Assembly Equipment

Dec. 2017

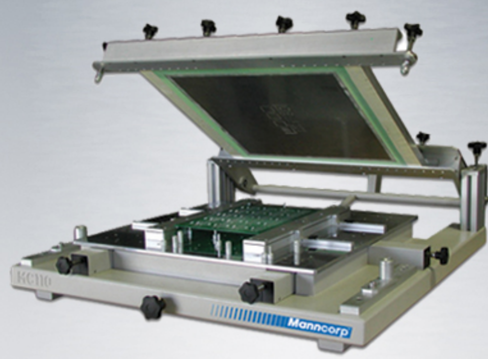
STENCIL PRINTERS

FOR ANY APPLICATION OR BUDGET

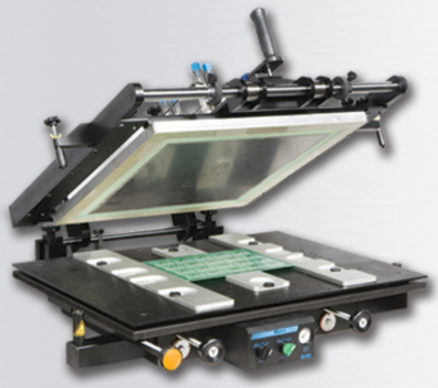
Wide variety of solder paste printers for any level of precision, automation and throughput

AUTOMATIC INLINE
$59,995

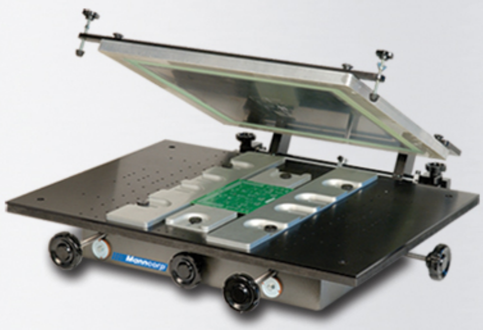
CAMERA-ASSIST ALIGN
$26,995



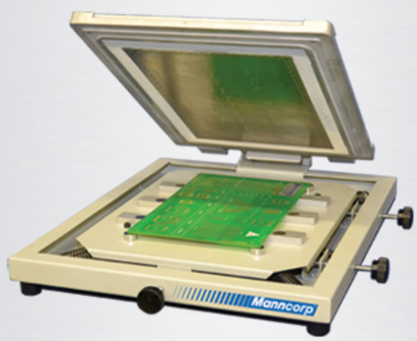
MANUAL SCREEN PRINTER
$4,995



5500 DUAL SQUEEGEE
$6,995



4500R MANUAL PRINTER
$3,595



RT2100 MANUAL PRINTER
$1,495

★ ★ ★ ★ ★ ★ ★

TAKE ADVANTAGE OF SECTION 179 TAX SAVINGS
(Equipment must be delivered by midnight December 31st.)

IN STOCK READY TO SHIP From Our East Coast & West Coast Locations

Text or Call
Chris - East: 215.869.8374
Ed - West: 215.808.6266
sales@manncorp.com

Manncorp™
www.manncorp.com

Huntingdon Valley, PA • San Diego, CA • Mexico • China

SMT MAGAZINE

December 2017 • More Content

In my recent conversation with process engineers at an EMS company here, they said one of the critical processes that determine yield in their line is the solder paste printing process. According to them, one of the key reasons for this is the incorrect printer set up, which results in issues such as insufficient solder or solder bridging. Of the three elements involved in the process—stencil, solder paste, and printer—the stencil is considered one of the major factors affecting the transfer efficiency, accuracy, and consistency, of solder pastes into the pads, especially with the continuing trend towards miniaturization.

Indeed, in our latest survey on solder paste printing, a majority of the respondents highlighted stencils as one of their key challenges. They mentioned the quality of the stencils; getting the right stencils—their stencils are done by a third party; aperture design; and stencil wear, among others, as issues around this part of the process. This is made more challenging because of the finer pitch and spacing in PCB designs. Specific problems in this regard include complete filling of apertures, paste release, and the large range of component types and sizes and the solder paste thickness requirement on the same design.

Other main issues include the accuracy and repeatability of the equipment, and the characteristics of the solder pastes being used.

Which brings me to our topic for this month's issue of *SMT Magazine*. Many studies over the years have found that up to 70% of PCB assembly defects come from the solder paste printing operation. In this issue, we look at the critical issues in the solder paste printing process, and how assemblers can address these challenges to help improve their yield and quality.

Among the features we have for this edition is a wide-ranging discussion involving AIM Solder, a solder paste provider; Manncorp, an equipment manufacturer; and Lenthor Engineering, a PCB assembler, to get a picture of what really happens in and the variables involved in ensuring the output quality of the solder paste printing operation. Tim O'Neill, Edward Stone, Matt Kan, and Dave Moody discuss the key challenges in solder paste printing, and how the industry can move forward and further improve the efficiency and quality of the process with new equipment technologies and solder chemistries.

Next, we have an article by Marco Lajoie and Alain Breton of C-MAC Microelectronics, about solder paste printing process inputs that impact the distribution of paste volume.

On the issues on stencils, we have a feature article from the Institute for Factory Automation and Production Systems (FAPS), University Erlangen-Nürnberg (FAU), and ASM Assembly Systems, discussing the effect of area shape and ratio on solder paste printing performance. Also included article from Tony Lentz of FCT Assembly, and Greg Smith and Bill Kunkle of BlueRing Stencils, about step stencil technologies and their effect on the solder paste printing process.

Then, we have a joint article from Benchmark Electronics, Shea Engineering Services, Vicor Corp., and Analogic Corp., which evaluates stencil printing technology for the miniaturization trend in electronics.

Finally, Ken Horky of Peterson Manufacturing writes about the advantages of generating your own stencil tooling.

Also in this month's issue, Dr. Jennie Hwang continues her column series on the role of bismuth in electronics. In Part 2, she outlines the bismuth effect in 63Sn37Pb solder materials.

We also have Chandran Nair, vice president for Asia Pacific at National Instruments, examining how Industry 4.0 will revolutionize electronics manufacturing.

Wow, I can't believe it's already December. Our team will be in Shenzhen, China, this month to attend the International Printed Circuit and APEX South China Fair, which is being presented by the HKPCA and IPC. That will wrap up our events this year, as we look forward to another year of interesting technology and electronics manufacturing innovations.

I hope 2017 has been a good year for all of you. On behalf of my colleagues here at I-Connect007, we wish you happy holidays, and a prosperous year ahead. As always, thank you very much for your continued support, and we look forward to bringing you outstanding content in the year ahead. **SMT**

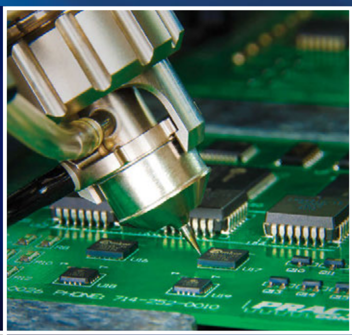
Stephen Las Marias is managing editor of *SMT Magazine*. He has been a technology editor for more than 14 years covering electronics, components, and industrial automation systems.

Check Out Our productronica 2017 Photo Gallery

Our team of editors here at I-Connect007 recently returned from Munich, Germany, to cover this year's productronica trade show. You can see our video coverage of the industry's top technologists and managers, as well as the latest technologies in the PCB design, fabrication, and assembly industries.

But we also shot a variety of still photos at the event. To see the entire productronica 2017 photo gallery, click here.

Don't Sacrifice Speed for Accuracy...



Programmable Tilt + Rotate enables dispensing into tight locations and around tall components

Dual-valve capabilities: dual-action or dual simultaneous dispensing

or Accuracy for Speed!

WATCH THE VIDEO ►

Nordson ASYMTEK's award-winning **Spectrum™ II** dispensing platform sets the standard for high speed and accuracy in precision dispensing applications for microelectronics, semiconductor, MEMS, and manufacturing applications. And its small footprint delivers maximum productivity with minimum manufacturing floor space.

Spectrum II features:

- Precision dispensing accuracy in X, Y and Z axes
- Quality assurance with patented closed-loop process controls
- Up to 6 heat stations available
- Single or dual lane conveyance
- Application and production support with a global network of dispensing experts

www.nordsonasymtek.com

Nordson ASYMTEK

The Role of Bismuth (Bi) in Electronics, Part 2

by Dr. Jennie S. Hwang
CEO, H-TECHNOLOGIES GROUP

Part 2 of the series outlines the Bi effects on 63Sn37Pb solder material, which have been substantiated by years of field performance prior to lead-free implementation. This should serve as the sound baseline for further discussion on the subject.

The incorporation of Bi in Sn-containing solders is expected to affect both physical properties and mechanical properties of the resulting solder materials. This includes melting temperature, wetting ability, strength, plastic strain and fatigue behavior. The direct addition of a sufficient amount of Bi to a eutectic alloy (e.g., 63Sn-37Pb) also alters its eutectic behavior or deviates from eutecticity. The DSC thermogram below of 63Sn37Pb plus 1 wt.% Bi indicates that 63Sn37Pb essentially maintains its eutectic property. However, at 2 wt.% Bi, the range of melting starts to appear, departing from the eutectic point.

An extensive study was carried out on the effects of a minor addition of Bi to SnPb eutectic solder. The following table summarizes the effects of addition of Bi to 63Sn37Pb up to 5 wt.% on the basic mechanical properties and the melting temperature. The dosages of 2 wt.% and 5 wt.% Bi were added to 63Sn37Pb, respectively, by separately replacing Sn or Pb. In

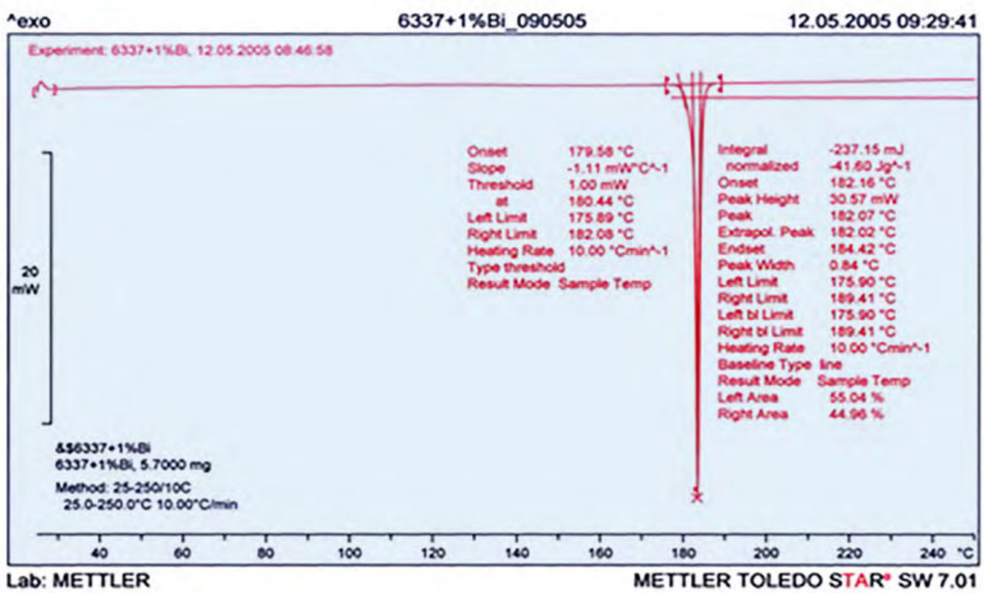


Figure 1: DSC Thermogram of 63Sn37Pb added with 1 wt.% Bi.

Alloy	Sn	Pb	Bi	Tm	σ_y^*	σ_{TS}^*	E^*	ε_p^*	ε_s^*	N_f^*
1	62	36	2	180-183	54	60	30	27		5623
2	61	37	2	180-183	51	58	26	27		5753
3	63	35	2	182-184	54	61	30	29		5012
4	60.5	34.5	5	175-180	46	62	32	12		5998
5	58	37	5	175-179	50 53	59 62	27 25	11 10		6412
6	63	32	5	177-182	48	62	33	15		6982
63/37	63	37		183	41	47	27	24		3650

Table 1: Melting temperature range and mechanical properties of 63Sn37Pb containing 2 – 5 wt.% Bi.

addition, tests were performed on alloys with Bi replacing an equal amount of both Sn and Pb. This resulted in six solder alloy compositions. The solder alloy compositions along with their melting temperature (T_m), yield strength (s_y), tensile strength (s_{TS}), Young's modulus (E), plastic strain (e_p) at fracture, and fatigue life (N_f) at a total strain of 0.2% are summarized in Table 1. All compositions are expressed in weight percent unless otherwise specified. Also included is the reference solder alloy of 63Sn37Pb.

As exhibited in Table 1, the addition of 2 wt.% Bi depressed the original melting temperature of 63Sn37Pb by 2–3°C. There was almost no distinction in the melting temperature change when 2 wt.% Bi replaced Sn or Pb or both Sn and Pb in an equal amount. At 5 wt.% Bi, both the alloy liquidus temperature and solidus temperature were lowered. The melting temperature for the solder alloys with 5 wt.% Bi in place of Sn (Alloy 5) was about 2–3°C lower than that with 5 wt.% Bi in place of Pb (Alloy 6). This indicates that Bi at 5 wt.% lowers the melting temperatures of Pb-rich phase more effectively than Sn-rich phase.

Comparing the strength of the solder alloys containing 2 wt.% Bi with that of 63Sn37Pb, the Bi addition largely increased the alloy strength and plasticity. There were no measurable differences in the tensile behavior among the solder compositions containing 2 wt.% Bi in place of Sn (Alloy 2) or Pb (Alloy 3) or equally both Sn and Pb (Alloy 1). When the content of Bi in 63Sn37Pb increased to 5 wt.%, the strength maintained, but the alloy plasticity reduced. The differences in tensile behavior among the solder compositions containing 5 wt.% Bi in place of Sn (Alloy 5) or Pb (Alloy 6) or equally both Sn and Pb (Alloy 4) were within the data-scattering range.

The content of Bi up to 2 wt.% was the most effective amount to increase both the alloy strength and plasticity. Any further increase in the Bi content from 2 wt.% to 5 wt.% exhibited little effect on the alloy strength, however significantly reduced the alloy plasticity.

The fatigue life (N_f) increased with the Bi content up to 5 wt.%, a contrast to the reduction in plasticity at 5 wt.% Bi. This is attributed to the amplitude of fatigue strain range used (De = 0.2%), which is well below the plastic strain at fracture (e_p = ~12%). Under the relatively low fatigue strain range, the high strength is a leading factor contributing to the high fatigue resistance.

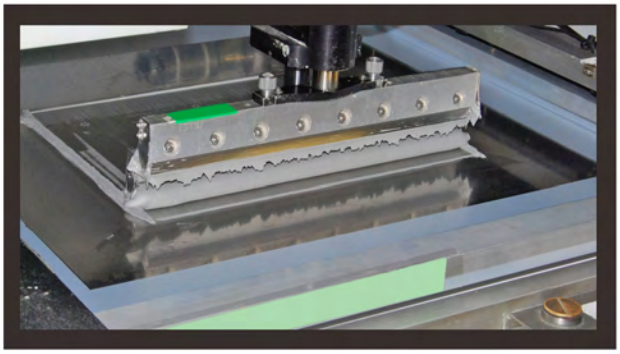
In a Bi-Pb system, a solid solubility of Bi in Pb is 23.5 wt.%, and a Sn-Bi system indicates a solid solubility of Bi in Sn is 21 wt.%. When Sn-Pb-Bi constructs a ternary system, the underlying basic physical interactions are not expected to be grossly changed. Overall, the Sn-Pb-Bi ternary solder alloys containing 2 wt.% Bi demonstrated a much higher strength, a higher fatigue life, as well as a higher plasticity than 63Sn37Pb eutectic solder. The alloy melting temperatures slightly decreased. When Bi increased to 5 wt.%, the strength and fatigue life of the Sn-Pb-Bi still remained higher than 63Sn37Pb, but their plasticity decreased significantly. SMT

Dr. Hwang, an international businesswoman, international speaker, and business and technology advisor, is a pioneer and long-standing contributor to SMT manufacturing since its inception as well as to the lead-free electronics implementation. Among her many awards and honors are induction into the International Hall of Fame-Women in Technology and the National Academy of Engineering, named R&D-Stars-to-Watch, and a recipient of YWCA Achievement Award and Distinguished Alumni Awards. Having held senior executive positions with Lockheed Martin Corp., Sherwin Williams Co., SCM Corp, IEM Corp., she is currently CEO of H-Technologies Group providing business, technology and manufacturing solutions. She serves as Chairman of Assessment Board of DoD Army Research Laboratory, Commerce Department's Export Council, National Materials and Manufacturing Board, various national panels/committees, international leadership positions, and the board of Fortune-500 NYSE companies and civic and university boards. She is the author of 500+ publications and several books, and a speaker and author on trade, business, education, and social issues. Her formal education includes four academic degrees, Harvard Business School Executive Program and Columbia University Corporate Governance Program. Further info: www.JennieHwang.com.

Researchers Develop Fully Integrated Circuits Printed Directly onto Fabric

Researchers from the University of Cambridge, working with colleagues in Italy and China, have successfully incorporated washable, stretchable and breathable electronic circuits into fabric, opening up new possibilities for smart textiles and wearable electronics. The circuits were made with cheap, safe and environmentally friendly inks, and printed using conventional inkjet printing techniques.

The researchers have demonstrated how graphene can be directly printed onto fabric to produce integrated electronic circuits which are comfortable to wear and can survive up to 20 cycles in a typical washing machine. The new textile electronic devices are based on low-cost, sustainable and scalable inkjet printing of inks based on graphene and other two-dimensional materials, and are produced by standard processing techniques. The work opens up a number of commercial opportunities for two-dimensional material inks, ranging from personal health and well-being technology, to wearable energy harvesting and storage, military garments, wearable computing, and fashion.



Equipment Matters in Solder Paste Printing

by Stephen Las Marias
I-CONNECT007

The solder paste printing process has always been considered a major contributor to yield loss. According to many studies, solder paste printing accounts for up to 70% of all PCB assembly defects.

For this month's issue of *SMT Magazine,* we interviewed experts in the solder paste printing process to learn more about the key issues leading to this huge percentage of defects, and the technology developments that are addressing these challenges and improving yields in the process. These experts, including Tim O'Neill, technical marketing manager for AIM Solder; Edward Stone, sales manager at Manncorp; and Lenthor Engineering's Dave Moody and Matt Kan, director of sales and marketing, and EMS manager, respectively, provided their insights from the perspective of a solder paste supplier, an equipment manufacturer, and a PCB assembler.

Lenthor Engineering's Kan says they mostly deal with flex and rigid-flex assemblies, so it's a whole different environment than rigid: "I can only speak from my personal experience. When I tried to deploy the same process steps that I used for rigid, I found out quickly that it didn't work for the flex and the rigid-flex world. Because with flex and rigid flex, any kinks in the board from one rigid to the next rigid, you get a misprint because you either fixture your board correctly, or you must array and keep your board as flat as possible," explains Kan. "With the flex and the rigid-flex, you have to tweak your printing parameters because everyone knows the board needs to be flat; and dealing with flex, you have different thicknesses with cover lay and pad, so your 5-6 mil stencil that you normally use for a rigid board doesn't apply in the flex circuit world. We normally go with 4 mils, and our standard is usually 3 mils. A printing problem that we encounter is when you use your normal 5 or 6 mils, you get more solder paste deposit than you would usually get on a rigid board. You have to use a thinner foil—and our standard is usually 3 to 4 mils. We use a lot more 3 mils than your average 5 to 6 mil for a rigid board. We normally use our Type 3 solder

paste, but as things get smaller and smaller, you need to get more deposit onto the smaller 0105 or 0201 components, so we would change up to use a Type 4 paste."

From a solder supplier's perspective, one of the key hurdles is the ever-shrinking solder deposit, and the challenges that it introduces into the printing process. "When we got into the business 20 years ago, the smallest component that you saw in practice was an 0603; 0402 was still theoretical. Now, we're talking about 08005s, sub-miniature and below the visual threshold. This not only causes significant challenges to the printing process, but downstream processes as well," explains AIM's O'Neill. "With regard to print defects, the area ratio or the thickness of the stencil as it relates to the size of the hole in the stencil is a critical parameter and we are confronted with apertures that are as tall as they are wide. Getting solder paste out of that aperture consistently and repeatedly is becoming more challenging because the ratios are becoming inverted. A 0.66 area ratio was considered the finest area ratio that was practical in a production environment; now, I would estimate a 0.5 area ratio is becoming more commonplace due to advancements both in solder paste technology, as well as complementary technologies, including hardware, stencil coatings, etc. The technology involved in paste printing is moving forward and giving assemblers more tools in their toolbox to address fine feature print challenges. Matt's situation with flex circuits inherently produces more variability than somebody assembling rigid circuit boards might have, so he's got an even bigger challenge in addressing consistency in his process."


Tim O'Neill, AIM Solder.

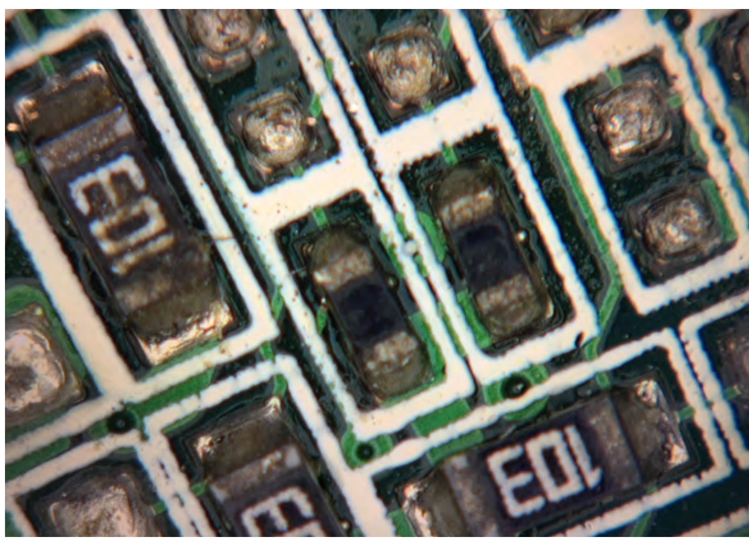
O'Neill says consistency and overall volume are the two most important variables that they are focusing on to help assemblers improve transfer efficiency. "Not only do we want to get as much solder paste through that aperture as we can, we want to get it through as repeatedly as we possibly can," he says.

In order to do that, solder paste rheology is evolving. Typically, as the viscosity gets lower, solder pastes become creamier. "It doesn't take a genius to see how a solder paste that is more fluid and less viscous is going to be easier to pass through the aperture." says O'Neill. "The problem this creates is that, when the stencil is removed, the deposit must maintain its shape or it would bridge, slump and cause other bridging and solder beading problems. Lowering viscosity alone, while it may improve transfer efficiency, won't necessarily improve consistency. Pad design, board design—there's a host of variables that need to be considered, board support being a big one—something the Lenthor guys building rigid-flex know better than anyone. Getting solder paste through an aperture is one thing, getting it through the apertures consistently is another. The advent of solder paste inspection (SPI) equipment has given the assembler the ability to measure every single deposit on that board in real time. This is especially critical now that we're reaching the visual threshold for solder deposits. SPI gives assemblers the ability to analyze, in real time, every deposit made on a circuit board. SPI closes the feedback loop so that the process engineers and supervisors can make sure the print process is well within the defined parameters."

"The SPI is a very important tool, not just for the assembler, but also for solder paste manufacturers, because it really provides the ability to measure your production output. If you're not measuring it, you don't really know what you're doing. This is especially important as you get down into area ratios at 0.66 and below. Matt brought it up: Mesh size is another way to improve transfer efficiency, however, everything in solder paste R&D is a quid pro quo. For example, the more activity introduced into a solder paste, the more reactive it becomes, which typically adversely affects shelf life, open time and transfer efficiency. The solder paste is interacting with itself as much as it's acting on the components being soldered. When I say it's a quid pro quo, if I increase the activity, you have to expect that the solder paste may not have as long a stencil life or storage time," adds O'Neill.

Reducing the mesh size has the same problem, according to O'Neill. Dropping the mesh size dramatically increases the surface area available for the same solder deposit volume. "I use this analogy when I give my talks: you've got a shoebox filled with Vaseline and marbles, and that is Type 4 solder paste. You take out all the marbles, and you fill it full of BBs. Now, you still have a shoebox full of metal, 90% by weight. The volume of metal has not changed, but the surface area that's available in that same volume has exploded when you think about the difference between a BB and a marble, or a ping pong ball and a grapefruit," explains O'Neill. "You have so much additional surface area that's interacting with the flux, and that does a number of things. As I mentioned about reactivity, the available oxide on the powder surface is interacting with the flux medium, and the flux medium is changing as a result. Add the friction involved during the printing process, the energy available in the solder paste rises, and this increases the interactions even more. And it gets more complicated when moisture from a humid environment is introduced and even more chemical reactions are initiated. All of these interactions increase when smaller mesh powder is used. Fine mesh pastes are much more sensitive to any type of input and the solder paste's characteristics can start to drift. As I mentioned at the beginning, consistency is the key to the whole process. Even if you're consistently bad, it's a process that you can control. Its inconsistency is the number one problem."

"There are a number of measures we are incorporating to improve paste print performance. We supply Type 4 rather than Type 3 as a standard mesh size, as part of addressing the challenges that ultra-fine pitch introduces. We have users insisting on Type 5, which, as you can imagine from my analogy of going from marbles to BBs from Type 3 to Type 4, I'm going from BBs to granulated sugar when I drop from Type 4 to Type 5. Our studies indicate reducing mesh size is less effective than other process adjustments that can be made and without the trade-offs I mentioned earlier. We are seeing more applications outside of traditional stencil printing and in the form of jetting solder paste where Type 6 and finer powders are used," says O'Neill. "That's the direction that we're headed, but I think there is a theoretical limit. By that I mean that we can only make the paste deposit so small until we run out of flux relative to the available surface are of the powder. All of the additional surface area that we've introduced to solve the transfer efficiency/jetting issue needs to be contended in the reflow oven. You end up with 'graping' and wetting-related defects because now, the paste deposit that's so small that very little is flux available, relative to the available powder surface area. It's forcing the solder industry to continue to innovate, both in terms of our flux chemistries, but as well as our powder manufacturing capabilities. Further complicating the development process are new defects that are a concern as the package and componentry has evolved. When I got into the business, a QFP was the biggest print challenge; then BGA came along, which introduced new challenge like voiding and HiP defects. More recently, bottom terminated devices have been adopted very quickly and they've introduced a host of new challenges for us, with ground pad voiding being the most prevalent. Ultimately, the printing process affects everything downstream; printing simply drives the rest of the assembly process."



Equipment Perspective

Manncorp often deals with OEMs and companies that are just getting into their first SMT line and developing their own in-house capability. "It's our job to suggest the appropriate equipment to be able to build the boards that the customer is looking to do," says Stone.

Because of the nature of the customers and economics—wanting to save money where they can—he feels that the stencil printer is not that important, and that they'd rather put their money into the pick-and-place machine.

"Yet, if you have a pick-and-place with capability to do 0201 or 01005 in ultra-fine pitch, and you buy the cheapest manual printer that you can get, you're not going to be able to benefit from all that capability. And that's something that you really must explain to the customer, and make sure that you're advising the correct equipment for the job at hand," says Stone. "In my opinion, a good manual printer for a start-up company is a great way to start. I mean, it's a good tool to have in your toolbox, but is it going to be able to do the most challenging devices out there? No, probably not. At the same time, it's something that you can put away for a couple years and, should you need it, it's going to work. I would say, as far challenges that have been around more recently, we're seeing a lot of activity in very large board printing, especially for the LED industry. We're seeing circuit boards that are between 1,200 mm and 1,500 mm long. That's four to five feet—exceptionally large compared to what we were used to. It used to be if you had a board that was 16" x 18", that was a really big board. Now, we're seeing boards that are really long, and you need specialized equipment to handle it. Those types of printers have to be super accurate, especially in the adjusting of the axis when it's aligning the stencil to the board. Because it is so long, your X-Y rotation is extremely critical. When it moves just a tiny little bit, when you're out there on the fringes, it's moving quite a lot. We're also seeing a lot of people getting into the 01005 printing, which has its own set of challenges. Again, you're going to need a good automatic printer, vision system, and all these programmable parameters that you'd expect in any high-end printer. That's not something that you're going to be doing on a simple tabletop device."

Edward Stone, Manncorp.

When looking at equipment for a given application, you need to make sure that all of the pieces are going to match, according to Stone. "That would be your stencil printer, your pick-and-place, and your reflow oven. It doesn't really make sense if your application is to do the smallest, most challenging and most fine-pitched components to not buy a printer that would match. When you get into the more challenging type of components, you're going to want something with fully programmable parameters, your squeegee speed, down pressure, snap-off speed and so on and so forth. Also, very important is to have something that has an undersized stencil cleaning ability to regulate the process and keep it uniform," says Stone.

Choosing the Right Solder

There are different parameters to consider when selecting the right solder paste for an application. According to O'Neill, the most important thing to consider is whether you plan to use no-clean, or you anticipate washing.

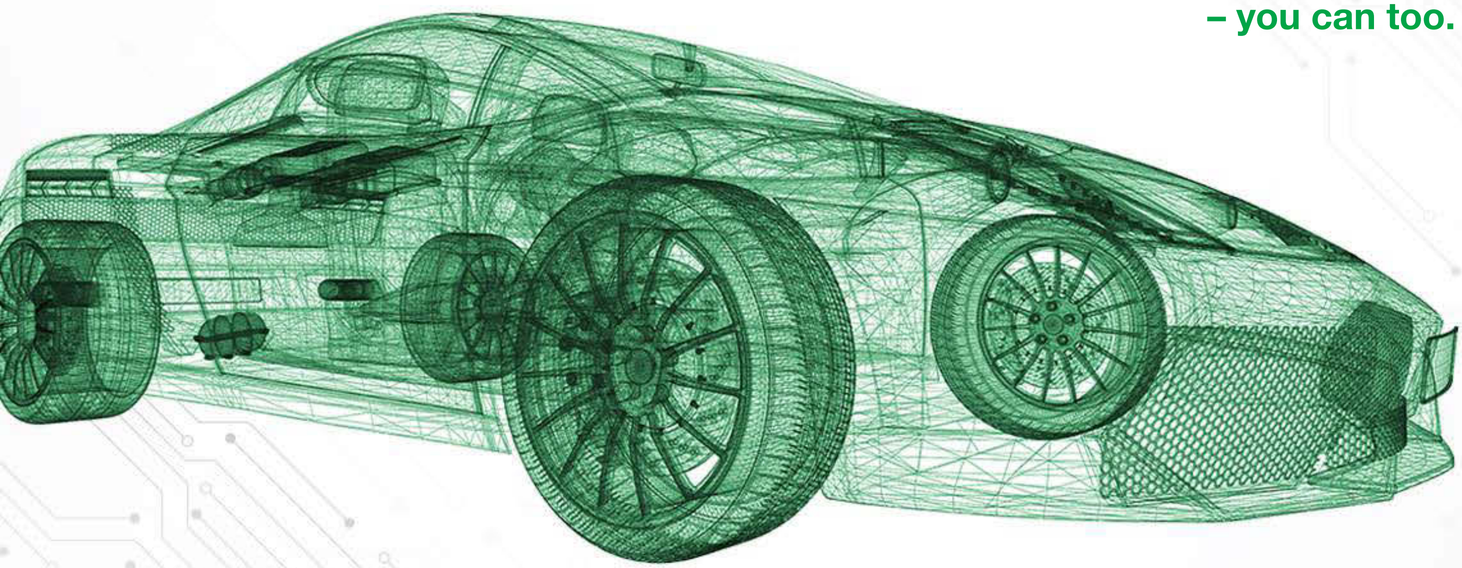
"Solder paste selection starts with that decision. Generally speaking, EMS providers prefer to wash their products because the customer's perception of quality is driven largely by visual assessment, even if it's not an appropriate metric. Contract manufacturers are being judged by the aesthetics of their final goods; therefore, washing is more common in the EMS world," says O'Neill. "On the other hand, OEMs traditionally adopt no-clean processes because their product is often sealed up in a box or an enclosure of some sort and nobody sees it. The requisite reliability testing is performed to ensure that product reliability meets their requirements, with the residues in situ. Leaving the

CML

pcbs from just around the corner

PCB Technical Expertise & Automotive Know-How

The global automotive industry trusts our PCB technical brilliance to drive their products – you can too.



CML are the leaders in printed circuit board (PCB) sourcing solutions. We care about reliable connections, placing paramount importance on convenience and reliability. We have a strong track record in the automotive market segment.

Driven by the high requirements, we transfer our accumulated know-how & technical expertise to further enhance the reliability of products and systems across different industries and multicultural markets.

This allows our customers to focus on their core business and perfect their competitive edge.
Their satisfaction is our main KPI.

CML

APEX EXPO IPC 2018

CML USA corp

Booth Number: 639, Hall E
San Diego Convention Center, California, USA
27th Feb 2018 - 1st Mar 2018
For more information: www.cml-globalsolutions.com

residues in place is a much more cost-effective model. The vast majority of consumer electronics incorporate no-clean technology and as a paste manufacturer, where we concentrate our R&D efforts. Our goal is to engineer a no-clean paste with the lowest possible residue with the best possible electrochemical properties."

"With water-soluble fluxes, we need to make sure that they can be easily removed, because cleaning under low-standoff devices presents a real challenge for manufacturers that must clean flux residue. There are applications that require cleaning, like RF signal circuitry, where the mere presence of residue will affect the signal. Another example is the application of conformal coating to improve product environmental robustness. Further refinement of a paste selection would be driven by individual application requirements. These might include voiding performance, print performance or wetting performance," says O'Neill.

"There are applications that require cleaning, like RF signal circuitry, where the mere presence of residue will affect the signal."

Designer Alloys

One of the issues highlighted when choosing solder pastes is lead-free. Interestingly, despite the industry moving into the lead-free world and processes, there are still segments that are using leaded solder, and are continually facing issues when it comes to process parameters.

One particular segment is the military market, which, according to Kan, accounts for about 40–50% of Lenthor's customer base.

"It's not their choice; the supply chain drives them into a lead-free world," explains O'Neill. "Some of them will acquiesce, reluctantly. Others will reintroduce lead into their systems. In terms of the flux chemistry, most are cleaning and most are using some sort of no-clean product or RMA product in combination with soap or solvent cleaning process. Many of these products have protective coatings applied and the coating manufacturers insist on a clean substrate to ensure the coating performs as designed. These assemblers are profoundly impacted by RoHS 2. I've assisted many customers in determining the best path forward based on their objectives. Some assemblers have chosen to reintroduce lead to their system and are reballing BGAs and using tin-lead paste. I have other clients transitioning over to lead-free as a significant percentage of military applications went to commercial off-the-shelf rather than military spec products. My understanding is that in July 2019, unless you have applied for an exemption from the EU to manufacture circuit boards with lead for products sold into the EU, you must be RoHS-compliant. I presume lead-bearing users are going to represent a very, very small portion of the European market. The challenge will be different countries having different regulations. China, it seems, is adopting the EU regulations. The U.S. doesn't have any regulations for lead-free, and you can still use lead in electronics. So, if it's a U.S. military application, they are not bound to RoHS and can continue to use lead bearing material."

O'Neill says AIM has every intention to continue to manufacture leaded solder. "But what I expect will happen is a lot like your software upgrades. You can have that old iOS that you were in love with and are afraid to migrate over to the new one, but eventually what happens is everything starts to evolve around you to a point where sticking with the old technology becomes impractical," he says. "You can't keep reballing and re-tinning forever. In one case, I had a customer who had an assembly which had 29 BGAs on it, and the BGAs were warranted by the manufacturer for three reflow cycles. That meant you could remove the lead-free balls, attach the leaded balls, reflow the first side, but you couldn't reflow the second side because that would have been four reflow cycles. Their assessment was that the four reflow cycles represented a greater risk than fully transitioning to a lead-free assembly and process. That's an example of how it sorts itself out."

"Lead-free materials now have a decade of experience behind them, and planes haven't fallen from the sky. So, at some point in time, you'd like to think the solder reliability question is less of a concern," says O'Neill. "Additionally, new alloys are being developed and the elimination of lead from the supply chain is reopening the opportunity to reintroduce other elements that were sidelined because lead was still present in the system, specifically bismuth. This may come as a surprise, but SAC305 was not the industry recommendation by iNEMI, which was like the Jedi council during the transition to lead free. Their initial recommendation was SAC387, and that was 3.8% silver, 0.7% copper. There were a great many factors to be considered in their selection process, so it was a sound recommendation. However, it was realized that 0.8% silver could be eliminated, which saved a few bucks a pound, which was quite significant when people were dumping thousand-pound solder pots. It also solved a reliability issue with higher-silver alloys. When the silver percentage exceeds 3%, a silver intermetallic is formed that becomes a brittle fracture boundary and reduces drop shock performance. As a result, SAC305 grew to become the standard and displaced the original recommendation of SAC387. SAC305 isn't written anywhere as being the standard; it's just what the industry morphed into. The other important thing was it was that SAC alloys were compatible with the lead that was still present in the supply chain. Ultimately SAC305 was the best available compromise of performance, reliability and existing process and material compatibility."

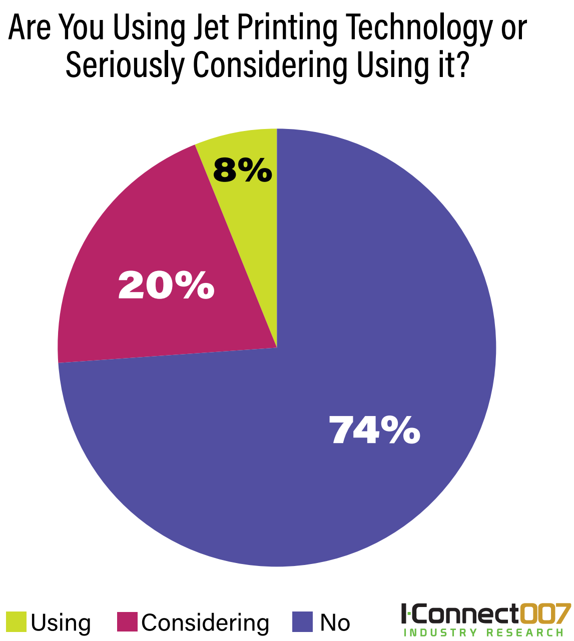
Bismuth was not considered while lead was still in the supply chain because if it were exposed to tin and lead, it would create an alloy with a melting temperature of 97°C, and it would fail very rapidly, according to O'Neill. This is the reason bismuth got sidelined.

"AIM and the solder industry as a whole are revisiting bismuth because it offers some significant benefits. Our testing indicates it can dramatically improve thermal cycling performances compared to SAC305. A little-known deficiency of SAC305 is when exposed to high temperatures for an extended period of time, the grain structure of the alloy coarsens over time and as the grain structure coarsens, the mechanical characteristics of the alloy degrade significantly. A SAC305 solder joint, if you get it hot enough for long enough, will literally disintegrate. As electronics become more powerful in less space, a byproduct is heat and we have already seen applications where SAC305 thermal cycling performance is inadequate for the application. The incorporation of the right amount of bismuth has a significant, positive influence on that," says O'Neill. "We anticipate that niche alloys, customized alloys for the application requirements, will evolve. We've already seen it happen on a widespread basis with the incorporation of an alloy like SN100C. It performed adequately, reduced costs by anywhere from 30–50%, and so was quickly adopted as a result. I think you'll continue to see that type of evolution in the alloy market space. I think it's going to continue to fracture, pun intended. SAC305 will no longer be the de facto standard. I think it will continue to be implemented, but that the other alloys will start to get introduced and solve these high-reliability concerns, specifically."

Matt Kan, Lenthor Engineering.

Jet Printing: The Next Evolution

Jet printers now have the capability to print more than one million dots per hour. Since they are mainly driven by software, jet printers no longer need stencils—thus eliminating the need to clean them—and offer faster changeover. In my recent discussion with a jet printer manufacturer, he said that while jet printers are not yet comparable to screen printers when it comes to volume, however, removing certain process steps in a jet printing environment will



somehow make the overall cycle time near that of screen printing.

"As technology evolves, it should get there. I go to the IPC APEX show every year, and the technology improves every year. It's out there. Everything's evolving, and things are getting smaller. These manufacturers are competitive from different vendors and suppliers. It'll be pretty close, but I don't know if it's going to be a one to one ratio where screen printing versus jet printing is going to be equal," says Kan. "For my personal opinion, it's good to have both in one environment. In our environment, we'll definitely take the advantages of both where, if we don't have a flat surface board, we can go ahead and use our inkjet, and if we do have a flat surface that we can tool up and do a 3D fixture and have a flat surface, we'll use the screen printer.

"My first job was an operator in 1992, and it was all manual printing, two buttons, and I stood there eight hours a day. The pick-and-place probably had two heads at the time that I was running, and as things evolved it became three heads, six heads, dual reel, front and rear, and then rotaries, and you had your Fujis and the Universals. I would strongly say that an inkjet will, maybe, start introducing two heads instead of the single one, or maybe four heads. Things are evolving. I would say 90% feel that an inkjet will be there in comparison to your regular stencil printing."

For O'Neill, jet printing represents the next evolution of solder deposition technology. "In my opinion, jetting capabilities will continue to improve and could serve several very useful roles in a surface mount line. It may not necessarily be displacing traditional stencil printing, but augmenting it. In my opinion, you'll see SPI and jetting working in combination to provide additional volume and eliminate the need for step stencils and solder preforms. A user can manipulate the volume of the solder paste on a board and make real-time corrections to solder past volume deficiencies, and that's a pretty exciting. Imagine now that you've got a jet printer on the SPI, it sees the missing solder deposit, and goes over and fixes it. That, to me, is where jet printing probably holds the most promise in the short term, because it's very difficult to imagine a point-to-point solder application that can replace the swipe of a stencil that can put down ten thousand paste deposits in a single pass. It's a throughput vs. flexibility equation that the end-user has to consider. Solder paste jetting is a viable technology that will definitely find its way into everybody's surface mount line over time."

According to Kan, jet printing works very well in the flex and rigid-flex world because the thickness of pads and solder mask is different, and you have to 3D your fixture in a way that you get a flat level printing surface. "But sometimes, you can't get away with that. In some areas, you'll have 10 mils-thick of a flex circuit, and then in some areas, you'll have 62-mils thick," he says. "It doesn't matter how you fixture your board, there's a stiffener involved that's in play that will prevent you from having a flat surface to print. Going with the jet printing can give you a lot of advantages, because jet printing is basically coming down and depositing the solder paste onto individual pads. We don't have a jet printer here, but I've had experience using one, and what we found out is that down below about 10 or 12 mils it starts to drift—meaning, you get inconsistency of printing. There are some challenges there."

For Stone, one of the problems with a jet printer is that it is cost prohibitive for a lot of


Fast and razor sharp
Flexible 2D/2.5D/3D X-ray Inspection
with StingRay Detector
www.goepel.com

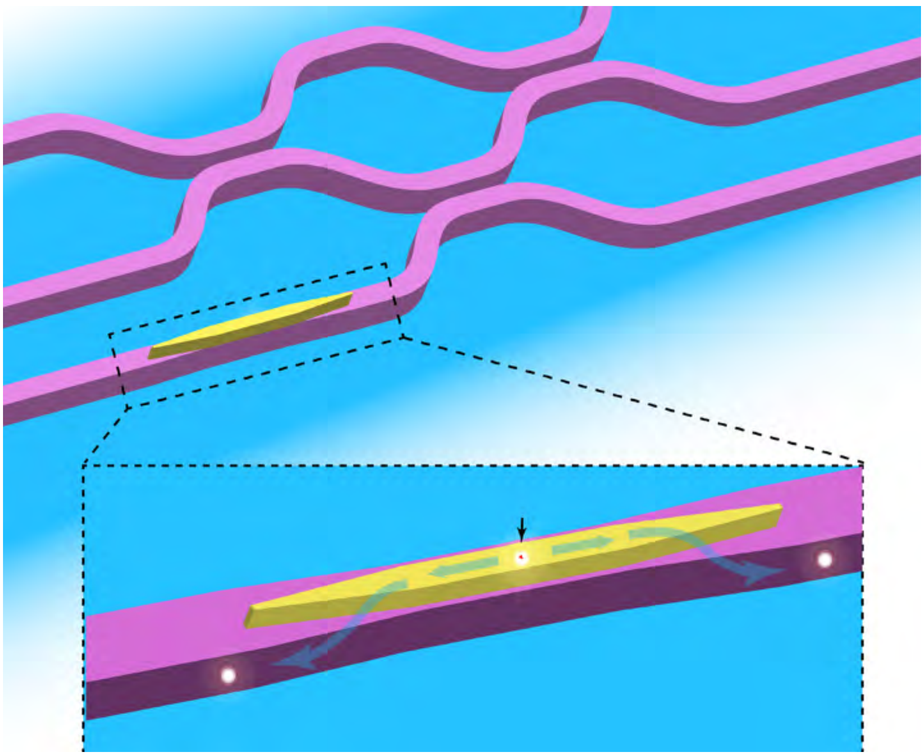
customers. "People are talking about how they want a line to do low volume. If you think about it, the jet technology is awesome. It gives you an incredible amount of flexibility, it's capable of doing the very fine pitch, the very small parts, and you don't have to order stencils," says Stone. "But at the same time, there's a cost involved in getting the equipment to start. You could buy a lot of stencils for the difference in cost from a good automatic printer to a jet-type printer. The other thing is the jet printer is going to require a lot more maintenance, and it's going to require that you use the higher cost solder paste rather than the typical Type 4 or Type 5. You're going to be up there in Type 5 or Type 6 solder pastes, so your cost of those goods is also going to go up."

Essential to Success

As mentioned earlier, solder paste printing is the source of the majority of defects in the SMT process. One very important factor to address the issue is selecting a very good stencil printer that's going to match the types of challenges that users are going to have in the assembly process, according to Stone. "The printer is not the part where you want to go cheap," he says. "A good printer is essential to success, as well as having a good reflow oven." SMT

Hybrid Circuit Combines Single-Photon Generator and Efficient Waveguides on One Chip

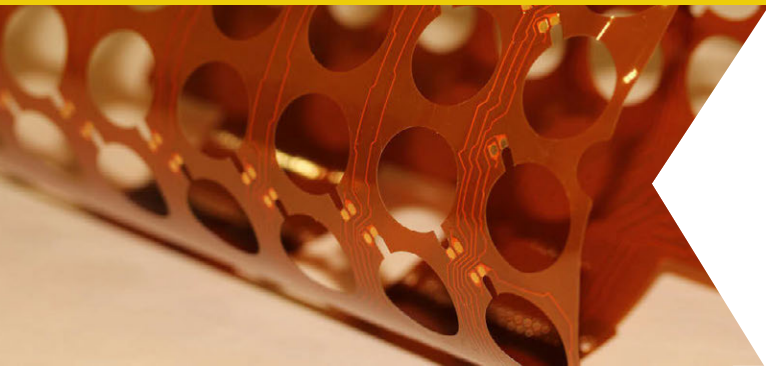
Scientists at the National Institute of Standards and Technology (NIST) have taken a new step forward in the quest to build quantum photonic circuits. The quantum circuit architecture devised by the team is among the first to combine two different types of optical devices, made from different materials, on a single chip—a semiconductor source that efficiently generates single particles of light (photons) on demand, and a network of "waveguides" that transports those photons across the circuit with low loss. Maximizing the number of photons, ideally having identical properties, is critical to enabling applications such as secure communication, precision measurement, sensing and computation, with potentially greater performance than that of existing technologies.



Developed by Marcelo Davanco and other NIST researchers along with collaborators from China and the U.K., the architecture employs a nanometer-scale semiconductor structure called a quantum dot—made from indium arsenide—to generate individual photons on the same chip as the optical waveguides—made from silicon nitride. Such hybrid circuit architectures could become building blocks for more complex systems.

We deliver Advanced PCB Solutions!

Our unique processes enable us to create and deliver Advanced PCB Solutions that meet the needs of the demanding market at competitive prices!



Impedance Tolerance <5%!

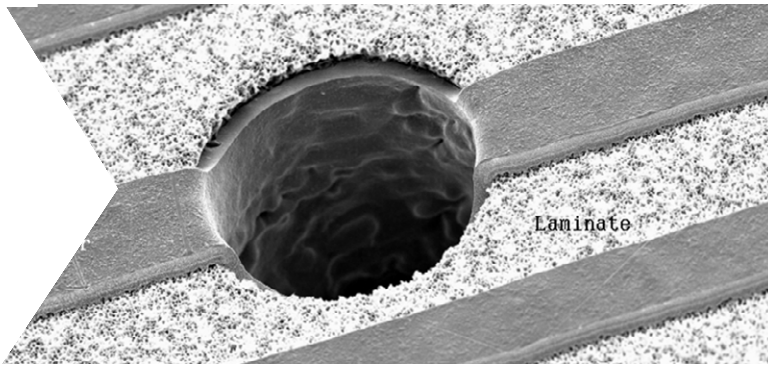
With our panel plating process combined with continuous foil lamination, our impedance tolerances are second to none!

CLICK TO LEARN MORE

Landless Via Technology

We're the only manufacturer in the world to have perfected a liquid photo-imageable resist (PiP) in electrolytic form, used for all our production.

CLICK TO LEARN MORE



Tensile Strength achieved with Candors Panel Plating

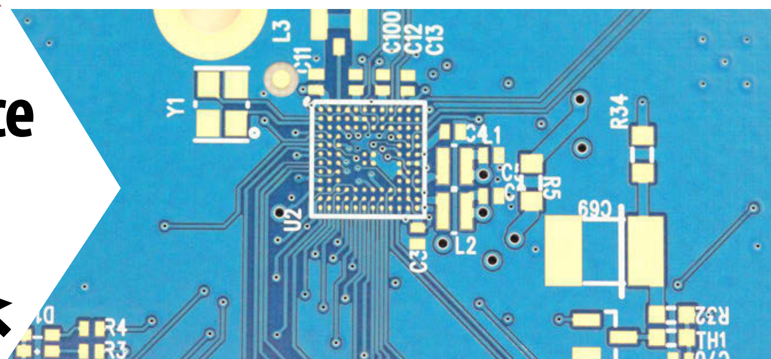
Practically perfect plating exceeds class 3A elongation, purity, and tensile strength

CLICK TO LEARN MORE

Fine line PCB, 2 mil line /space

We have developed a unique set of processes based on electro-depositable liquid photoresist.

CLICK TO LEARN MORE



CANDOR INDUSTRIES INCORPORATED

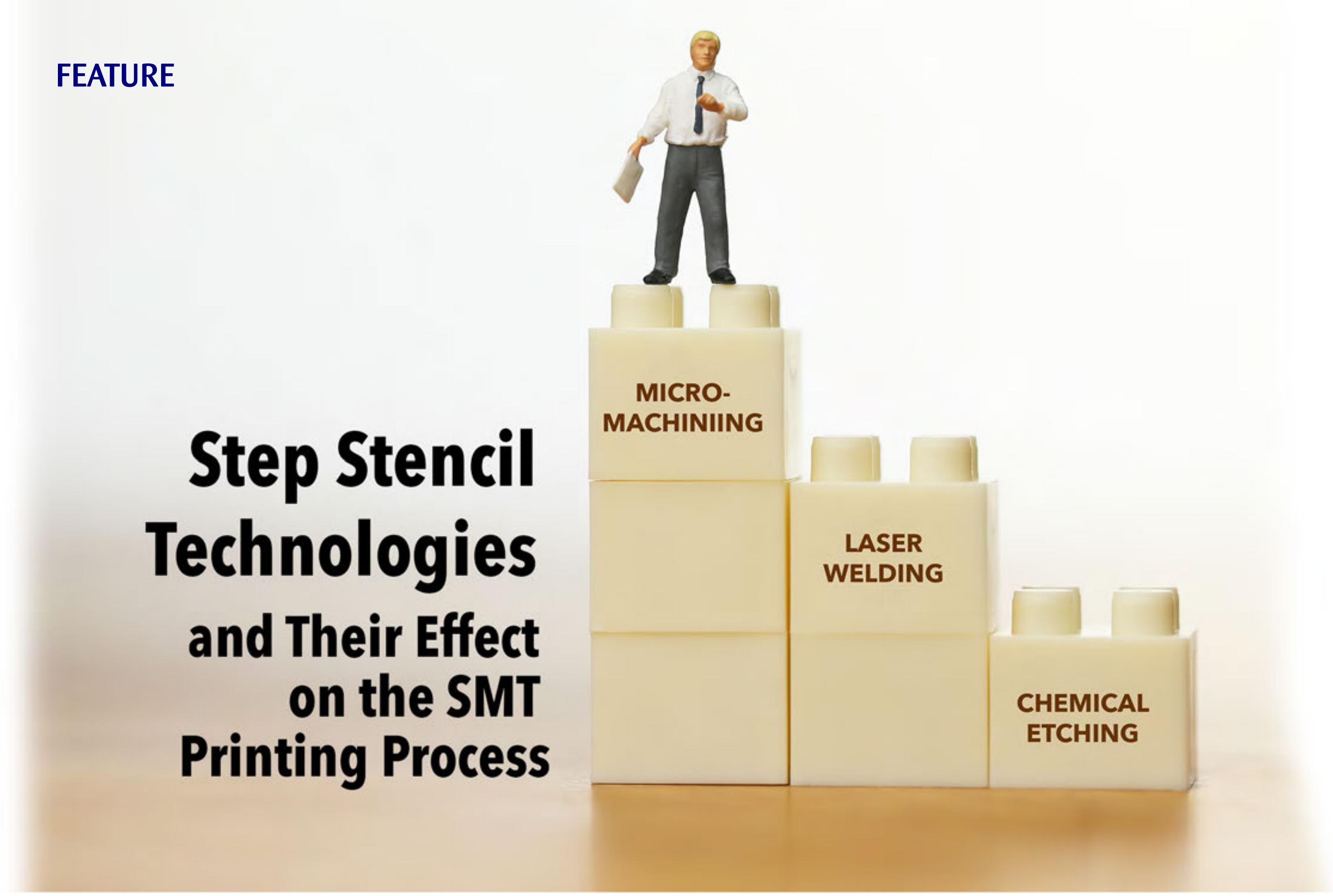
ADVANCED PCB SOLUTIONS

PHONE: (416) 736-6306

WEB: www.candorind.com

SALES: sales@candorind.com

FEATURE



Step Stencil Technologies

and Their Effect on the SMT Printing Process

by Greg Smith and Bill Kunkle,
BLUERING STENCILS,
and Tony Lentz, FCT ASSEMBLY

Components such as quad flat no lead (QFNs), land grid array (LGAs), micro ball grid array (micro BGAs), 0201s and even 01005s continue to push manufacturers to use thinner stencil foils to apply the correct volume of paste onto their boards, but larger components such as edge connectors still require larger paste volumes. Step stencils have been used to accomplish this for many years. Historically, the primary method for producing these step stencils has been using a photochemical etching process. Recently, new methods of manufacturing step stencils have emerged including both laser welding and micro-machining.

Photochemical etching is an established process and has been around for decades. It is a subtractive process and is very similar to the process used to etch PWBs. The stainless-steel stencil foil is coated with a photo-resist, imaged using a photographic process and developed leaving the resist to protect any areas that will not be reduced in thickness or etched. The foil is placed into an etching machine where chemical etchant is sprayed onto the stencil which dissolves the stainless-steel foil until the correct thickness is achieved. Once the desired stencil thickness is achieved, the photo resist is removed. The depth of the etched or stepped areas using this process is dependent on the time that the stencil is exposed to the etching chemistry. The chemical etching process is shown below (Figure 1).

The laser welding process takes stencil foils of different thicknesses and welds them together. There is no chemical etching involved, only

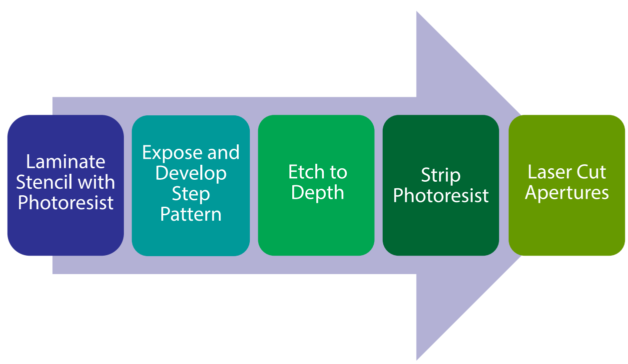


Figure 1: Chemical etching process to create a step stencil.

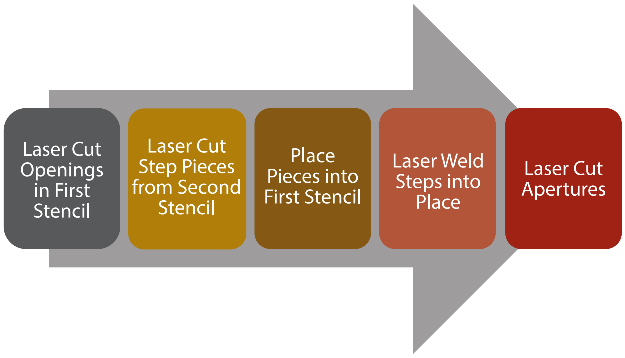


Figure 2: The laser welding process to create a step stencil.

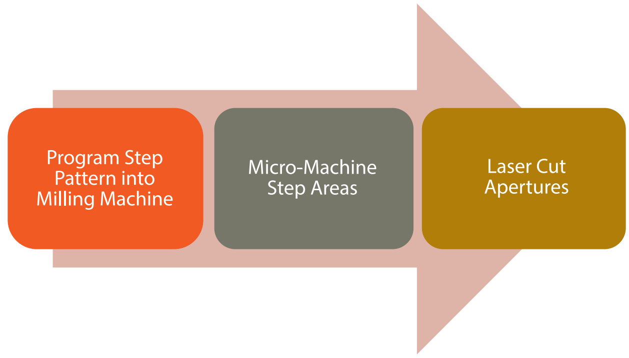


Figure 3: The micro-machining process to create a step stencil.

laser cutting and laser welding. The step openings are cut out of the first stencil. The corresponding step areas are cut out of a second stencil foil of the desired thickness. The step pieces are placed into the openings of the first stencil. Then the pieces are laser welded into place. The thickness of the step area is determined by the thickness of the steel used. The laser welding process is shown below (Figure 2).

The micro-machining process is a subtractive process similar to the etching process, but no chemicals are used. The micro-machining process uses a very specialized computer numerical controlled (CNC) milling machine to remove very small amounts of material at a time. The micro-machining process is shown below (Figure 3).

These three processes for creating step stencils result in different textures within the stepped area. The textures of the step stencils are shown below (Figure 4).

Experimental Methodology

A step stencil design was created with step-down pockets of varying thicknesses. The base stencil thickness was 4.0 mils (101.6 microns) and the step-down pockets were 3.5 mils (88.9 microns), 3.0 mils (76.2 microns), 2.5 mils (63.5 microns), and 2.0 mils (50.8 microns) thick. Each step area was 1-inch square (25.4 mm) and the step design is shown below (Figure 5).

The thicknesses of each step pocket were measured using a FARO arm device. The measurements for each step technology were compared and contrasted.

An aperture pattern was created for the following components: 03015 metric, 01005, 0.3 mm BGA, 0.4 mm BGA, and 0.5 mm pitch QFNs. Apertures for each component were cut at varying distances from the step edges; 10, 20, 30, 40, and 50 mils. The intention was to determine how close solder paste could be printed to the step edge for each step stencil technology.

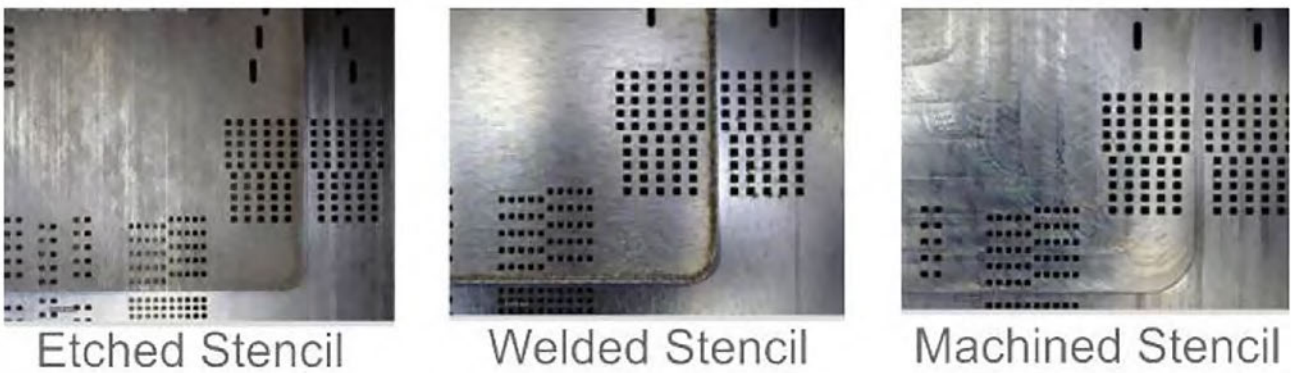


Figure 4: Textures of step areas for the three step technologies.

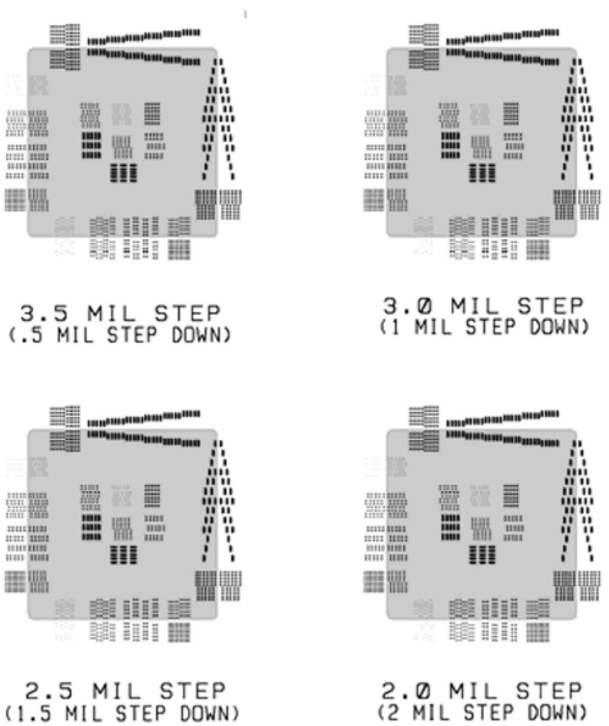


Figure 5: Step down pocket design.

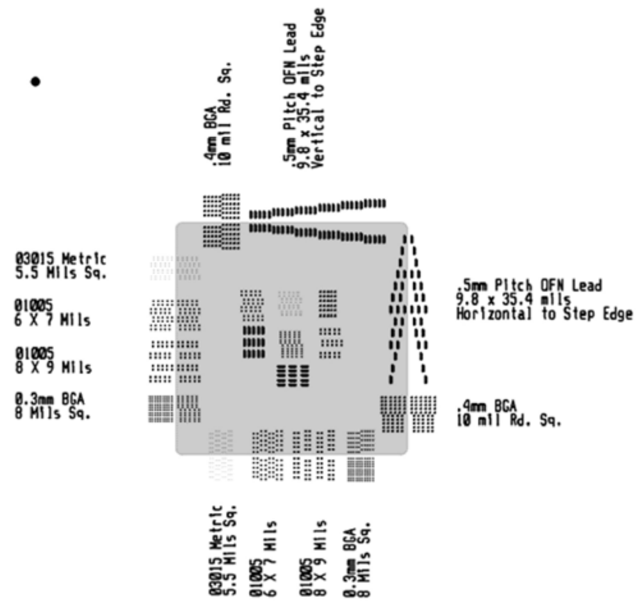


Figure 6: Step stencil aperture design.

Apertures were also cut into the center of each step area for comparison. The aperture layout is shown below (Figure 6).

Each stencil was made with two sets of steps and apertures. One set of steps and apertures were coated with a fluoro-polymer nano (FPN) coating (Figure 7).

The effects of the FPN coating were compared to the uncoated part of the stencil on printing of solder paste. A 10-print study was run on each step stencil using a popular no clean, SAC305 Type 4 solder paste. The circuit boards used were bare copper clad material 0.062" (1.57 mm) thick. The printer used was a DEK Horizon 02i. The printer parameters are shown below (Table 1).

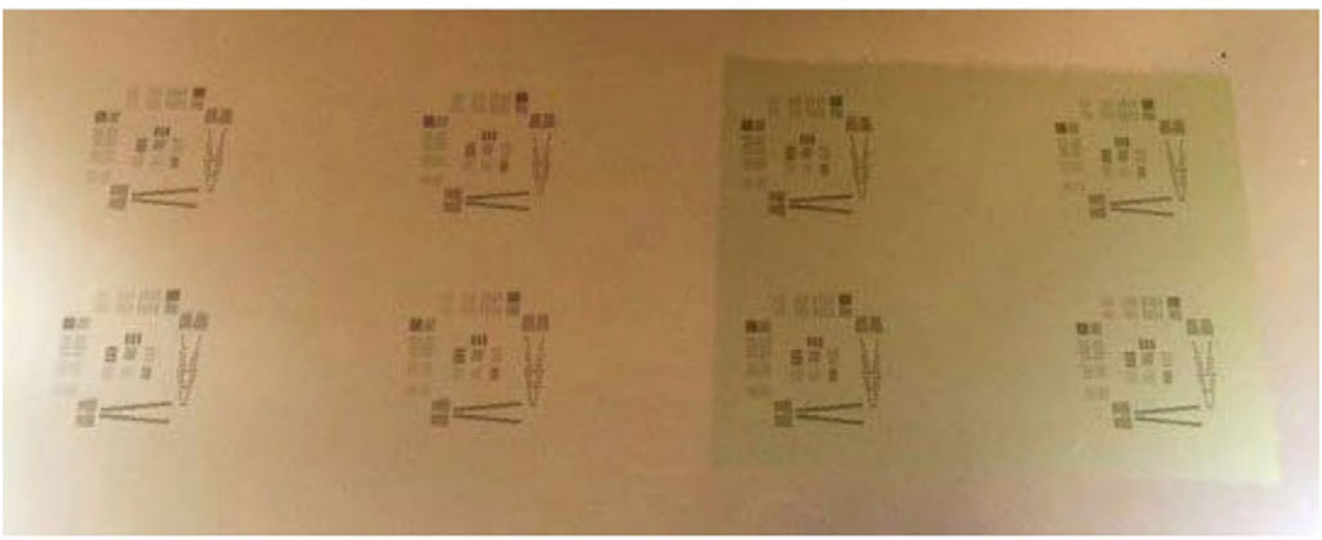


Figure 7: Step stencil contact side with FPN nano-coating.

Parameter	Value
Squeegee Length	300 mm
Squeegee Pressure	0.18 kg/cm (5.4 kg)
Squeegee Speed	30 mm/sec
Squeegee Angle	60 degrees
Separation Speed	1.0 mm/sec
Cleaning Cycle	W/V/D every print
Cleaning Solvent	Isopropanol (IPA)
Solder Paste	NC SAC305 T4

Table 1: Solder paste printer parameters.

The solder paste volumes were measured using a solder paste inspection system (SPI). The solder paste volume data was analyzed using statistical analysis software and the results are presented in this paper.

Results:

Step Stencil Thickness Measurements

Measurements were taken in each step area for each technology. The 3.5 mil step area was not included due to issues with the measurement data. Thickness measurement data is shown below (Table 2).

In general, the chemical etching process created deeper step downs than the nominal value, and the welding and machining processes created steps that are closer to the target depth. The standard deviations of step depth are an indication of flatness or roughness in the step areas. The chemical etching process produces a surface that is rougher than the original surface (Figure 4). The welding process involves fixturing a stencil blank into the step area and the blank may not sit perfectly flat as it is welded. The machining process leaves striations on the surface as the cutting tool removes material.

Step Thickness (mils)	Step Depth (mils)	Etched – Depth / STDev (mils)	Welded - Depth / STDev (mils)	Machined - Depth / STDev (mils)
2.0	2.0	2.36 / 0.11	2.08 / 0.19	2.01 / 0.17
2.5	1.5	1.69 / 0.16	1.60 / 0.18	1.25 / 0.15
3.0	1.0	1.15 / 0.19	0.97 / 0.18	1.08 / 0.17

Table 2: Step down thicknesses for each step stencil.

Overall, the standard deviations are very similar for each technology.

Solder Paste Printing Data —Etched Stencil

The solder paste volume box plots for the 3.0 mil, 2.5 mil, and 2.0 mil thick etched steps are shown below (Figures 8, 9, and 10, respectively). These are broken out by distance from step edge, aperture size and nano-coating.

In general, the larger apertures which are 9.8 x 35.4 mils in size give higher printed solder paste volumes. The smaller aperture sizes

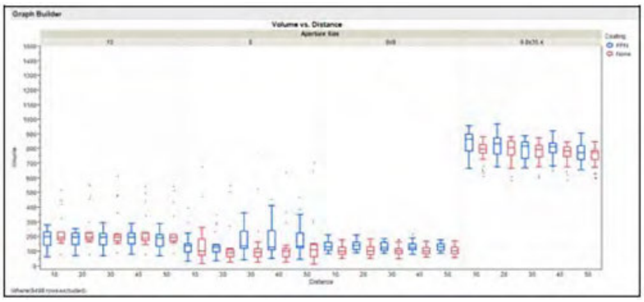


Figure 8: Solder paste volumes for the 3.0 mil etched step.

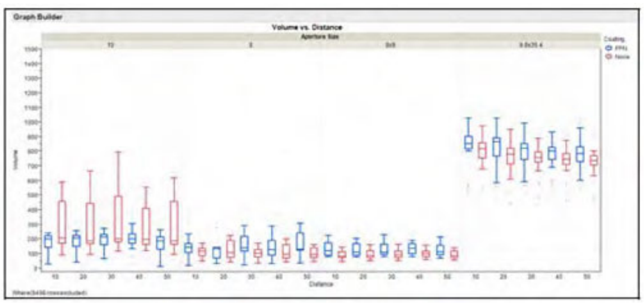


Figure 9: Solder paste volumes for the 2.5 mil etched step.

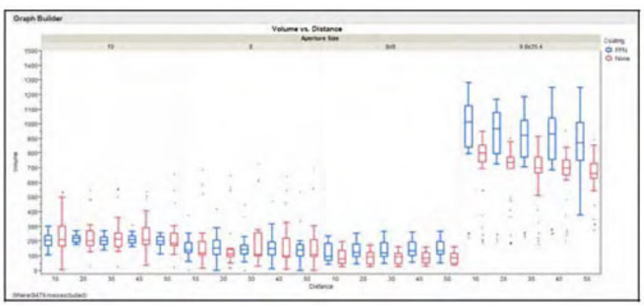
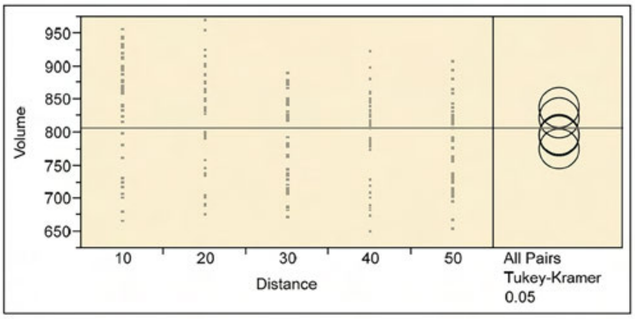


Figure 10: Solder paste volumes for the 2.0 mil etched step.



Connecting Letters Report

Level			Mean
10	A		839.52500
20	A		823.35000
40	A	B	798.30000
30	A	B	796.65000
50		B	776.22500

Figure 11: Tukey-Kramer HSD analysis for the FPN coated, etched 3.0 mil step and the 9.8 x 35.4 aperture.

show some differences in solder paste volume. Tukey-Kramer Honest Significant Difference (HSD) testing shows that most of these variations are statistically similar. This means that there is very little difference in printed solder paste volume from 10 to 50 mils from the step edge. There is one exception to this (Figure 11).

The printed solder paste volume is higher for the 10 and 20 mil distances than the 50 mil distance. This indicates that the squeegee is not able to conform into the step pocket and squeegee the paste cleanly away from the surface. The same Tukey HSD analysis is true for the uncoated version of this step and aperture size.

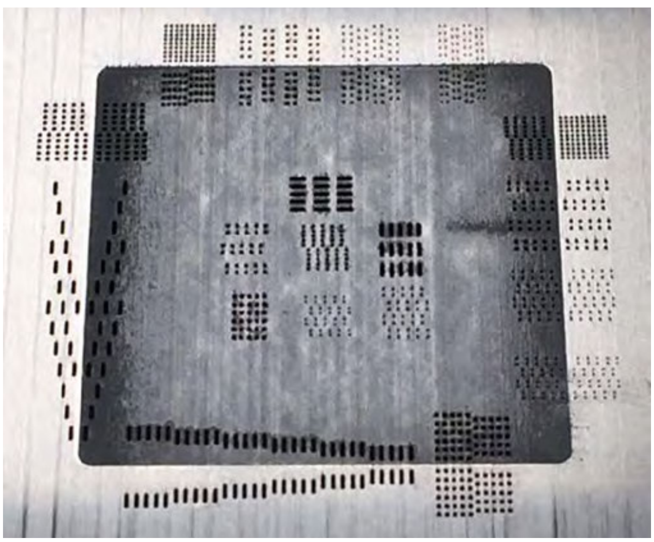


Figure 12: Etched 3.0 mil step after one print.

It is easy to see solder paste residue left near the step wall after printing. This seems to correspond to higher printed-paste volumes.

Solder Paste Printing Data—Welded Stencil

The solder paste volume box plots for the 3.0 mil, 2.5 mil, and 2.0 mil thick welded steps are shown below (Figures 13, 14, and 15). These are broken out by distance from step edge, aperture size and nano-coating.

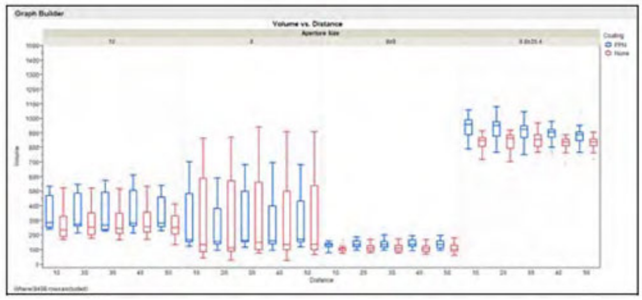


Figure 13: Solder paste volumes for the 3.0 mil welded step.

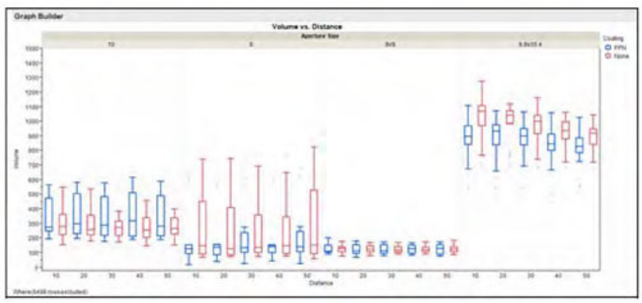


Figure 14: Solder paste volumes for the 2.5 mil welded step.

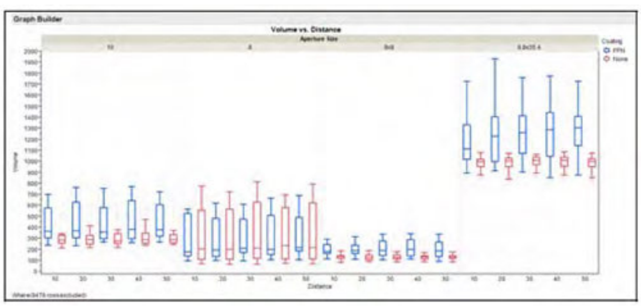
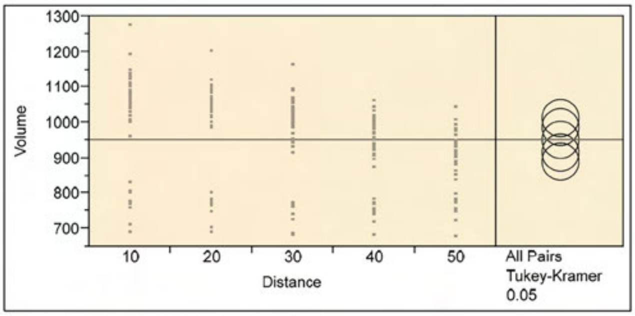


Figure 15: Solder paste volumes for the 2.0 mil welded step.



Connecting Letters Report

Level				Mean
10	A			1015.3250
20	A	B		988.8000
30	A	B	C	954.1750
40		B	C	916.4750
50			C	891.8500

Figure 16: Tukey-Kramer HSD analysis for the uncoated, welded 2.5 mil step and the 9.8 x 35.4 aperture.

The printed solder paste volume did not vary much from 10 mils to 50 mils away from the step edge regardless of welded step thickness. This is very similar to the results seen with the etched steps. Tukey-Kramer HSD testing shows some interesting results (Figure 16).

In this case, the printed solder paste volume at the 10-mil distance is significantly higher than the 40 and 50 mil distances. The printed solder paste volume is significantly higher at the 20-mil distance than then 50 mil distances. Again, this indicates that the squeegee could not conform down into the step to remove all the solder paste from the surface of the stencil during printing. This was also true for the FPN coated version of the 3.0 mil welded step and the same aperture size.

Solder Paste Printing Data—Machined Stencil

The solder paste volume box plots for the 3.0 mil, 2.5 mil, and 2.0 mil thick machined steps are shown below (Figures 17, 18, and 19). These are broken out by distance from step edge, aperture size and nano-coating.

The printed solder paste volume did not vary much from 10 mils to 50 mils away from the step edge regardless of machined step thickness. This is very similar to the results seen with the etched and welded steps. Tukey-Kramer HSD testing shows some significant differences in the results (Figure 20).

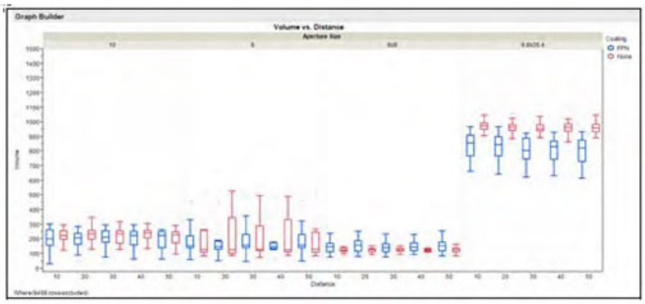


Figure 17: Solder paste volumes for the 3.0 mil machined step.

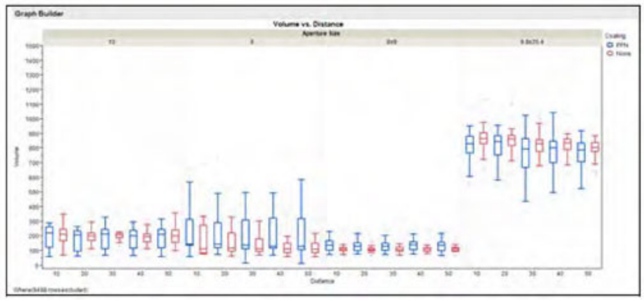


Figure 18: Solder paste volumes for the 2.5 mil machined step.

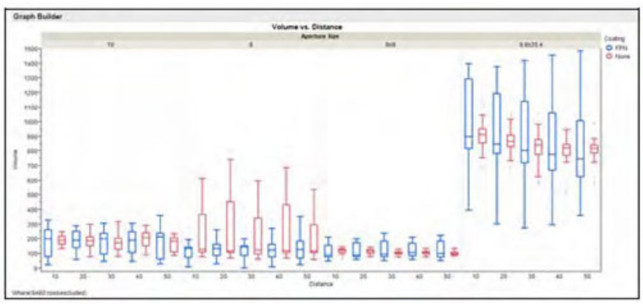
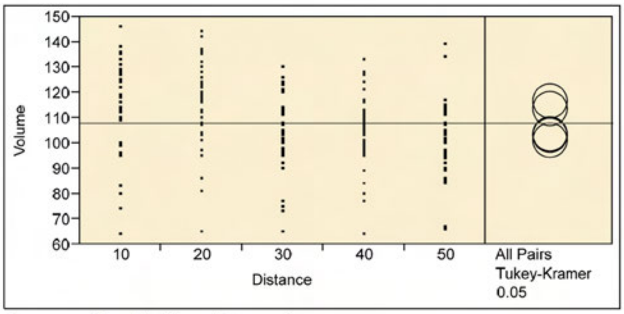


Figure 19: Solder paste volumes for the 2.0 mil machined step.

This Tukey HSD analysis shows that the printed solder paste volume is significantly higher at the 10 and 20 mil distances than the 30, 40, and 50 mil distances. This is also true for the 9.8 x 35.4 mil aperture with 2.0 mil and 2.5 mil machined step thicknesses.

Conclusions

Chemical etching, laser welding, and micro-machining are each valid methods of producing step stencils. Each process produces



Connecting Letters Report

Level			Mean
10	A		116.82500
20	A		113.92500
40		B	104.20000
30		B	103.72500
50		B	101.42500

Figure 20: Tukey-Kramer HSD analysis for the uncoated, machined 2.0 mil step and the 8 x 9 aperture.

different step surfaces. Regardless of the technology used to produce the step, solder paste volumes for the QFN apertures tend to be higher 10 to 20 mils from the step edge than 30 to 50 mils away. These increased solder paste volumes could lead to shorts with aperture designs for the tested QFN component designs. It would be possible to place apertures this size 30 mils from the step edge and obtain an acceptable solder paste volume. The smaller 8, 8 x 9, and 10 mil apertures gave statistically similar solder paste volumes from 10 to 50 mils away from the step edge. Although further investigation is needed, the data shows that small aperture components can be placed as close as 10 mils from the step edge and still obtain acceptable print volumes.

The FPN coating showed a slight increase in volume across all apertures measured. When printing these small apertures, it is recommended to apply a FPN coating. It is apparent that these step stencil technologies bear further investigation to differentiate between them.

Future Work

Testing is ongoing with these three step stencil technologies. Solder paste volumes at the center of the step area will be compared to volumes near the step edge. The solder paste volumes from a single level stencil of the same thickness as the step-down area will be compared and contrasted to the volumes in the step etched stencils. The printed solder paste volume from apertures oriented horizontal to the squeegee will be compared to apertures oriented vertical to the squeegee within step areas. Squeegee pressure and speed will be varied and the effects on solder paste volume will be studied. Finally, we plan to increase the number of boards printed to obtain a larger set of data to expand on these findings.

Acknowledgments

We appreciate the support of MET for providing the welded step stencils for this investigation. We also appreciate the support of Fine Line Stencil, who provided the etched and machined step stencils and the nano-coating. **SMT**

Editor's Note: This article was originally published in the proceedings of SMTA International.

Greg Smith is the manager of stencil technology at BlueRing Stencils.

Bill Kunkle is a technical service manager at BlueRing Stencils.

Tony Lentz is a chemist and field application engineer at FCT Assembly.

MilAero007 Highlights

NEO Tech Opens Second Agave Facility to Further Expand Production
NEO Tech has announced that its second Agave manufacturing site, located in Ciudad Juarez Mexico, is now open and fully operational.

Nortech Systems Collaborates with Panasonic to Expand Manufacturing Capabilities
Nortech Systems and Panasonic have collaborated on a new, total-line-solution installation of integrated hardware and software.

Jabil Launches Blue Sky Center in Singapore
Jabil Inc. has launched its newest Blue Sky Innovation Center in Singapore, extending Jabil's network of Innovation Centers in America and Europe to Asia.

Zentech Secures United States Navy Blanket Order Agreement
Zentech–Fredericksburg Operations, formerly Colonial Assembly and Design, has received a United States Navy Blanket Order Agreement (BOA) for rapid technology integration of Special Mission Equipment (SME) on both manned and unmanned platforms in support of Maritime Patrol and Reconnaissance Aircraft (MPRA) and Persistent Maritime Unmanned Aircraft Systems (PMUAS).

Joseph O'Neil Joins Power Design Services as CEO
Power Design Services Inc. has appointed Joseph O'Neil to the position of chief executive officer. Prior to joining Power Design Services, he was CEO and president of Hunter Technology Corp. He is also chairman of the Board of Directors of IPC and is a member of the IPC Designer Council Steering Committee.

Kimball Electronics Reports 12% Revenue Growth for Q1 FY2018
EMS firm Kimball Electronics Inc. has announced consolidated net sales of $253 million for its first quarter ended September 30, 2017, up by 12% to compared to the first quarter of fiscal year 2017.

Key Tronic Reports Q1 FY2018 Revenue of $109M
EMS provider Key Tronic Corp. has reported total revenue of $109.2 million for the first quarter ended September 30, 2017, down from $117.1 million in the same period of fiscal year 2017.

Libra Industries Now Certified to AS9100 Rev D
Libra Industries' Dallas, Texas facility has successfully completed its certification for the AS9100 Rev D SAE International Aerospace Standard.

CTS Reports Robust Q3 Sales and Earnings Growth
CTS Corporation has posted third quarter sales of $106.2 million, up by 6.6% year-over-year.

Sanmina Receives 2017 Supplier Excellence Premier Award from Raytheon
Raytheon Company has recognized Sanmina Corp. with a Premier Supplier Excellence Award for their outstanding achievement.

TIPS & TRICKS:

GENERATING STENCIL TOOLING

by Ken Horky
PETERSON MANUFACTURING

Many engineers are leaving the editing up to the stencil fabricator these days, describing the PCB array, aperture undersize/oversize, shape conversions, etc., for the fabricator to then edit. From the outside, this may appear as a time saver for all of us overworked process engineers, but considering how many stencil redos have been required and how many processes that have run "sort of OK," there's a tremendous amount of scrap and rework that could be saved from just a little more attention paid to stencil tooling. I prefer to make my own mistakes rather that receiving a stencil (with the project due tomorrow) and finding that someone else didn't do something right, or misunderstood my design intent.

It's a huge advantage for the experienced engineer to be able to produce good, first run results most, if not all, of the time. You can learn what works best for your equipment in your environment. There are several Gerber editing tools available that will let you create a library and apply changes semiautomatically or automatically to your data.

One of the most common edits used for resistors and capacitors is the "home plate" design. The left aperture below (magenta) is a typical aperture reduction. The right aperture below (cyan) is the home plate pattern. Black being the component outline and terminals, and red being the original pads.

This is a very useful aperture design for mitigating solder balling and tombstoning. It can also be a difficult shape to describe, and convey where to apply it, to a stencil fabricator. Some large connector pins may benefit from oversizing the aperture to improve mechanical interconnect and ease inspection. Large heatsinks, such as those for DPAKs (Decawatt Package), can be divided into multiple apertures to reduce squeegee scooping and voids. Bowtie patterns can reduce solder balling on large gull wing pins.

I position the print image within my foil perimeter in the data, depending on the assembly type, and indicate the location for the stencil ID. My operators like the image forward

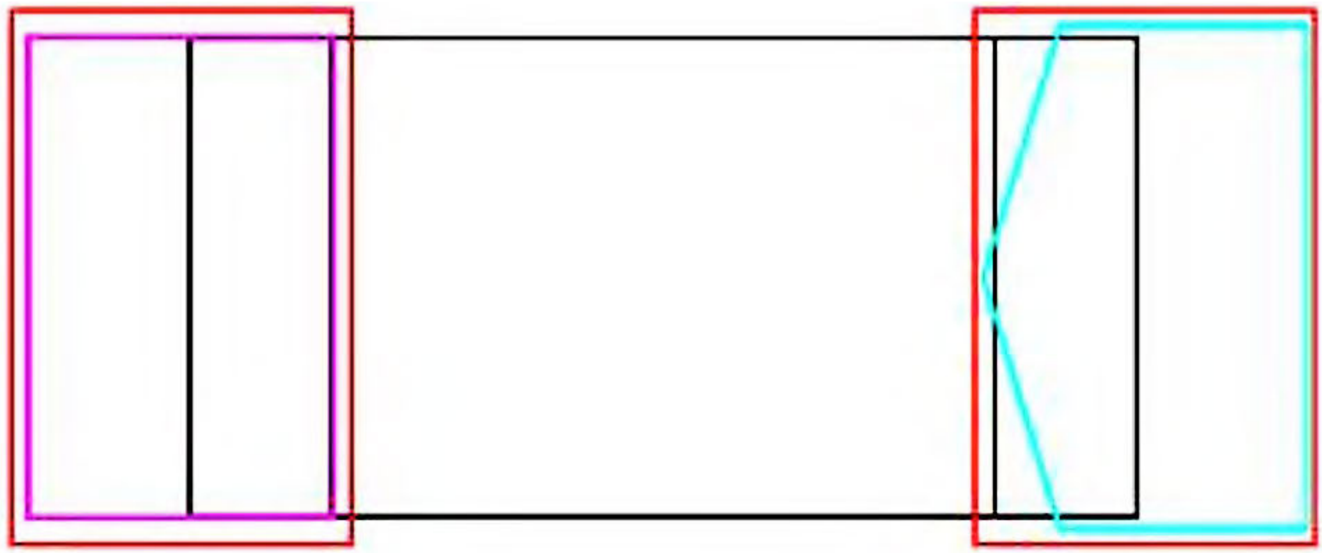


Figure 1: Home plate design.

of center for easier access, a better visual in process, and less ambiguous installation. We like to put both images of a double-sided board on the same stencil to reduce change over time. The distance between the two images needs to match your printer's blade span and over travel distance.

Editing your own data also allows you to eliminate the check plot, further reducing your design cycle time. It also reduces the fabricator's editing time, further improving their turn time.

Rather than spending time generating a document to describe the edits to your stencil fabricator and hoping they'll get it right, make the edits yourself to get what you intended and improve your process. **SMT**

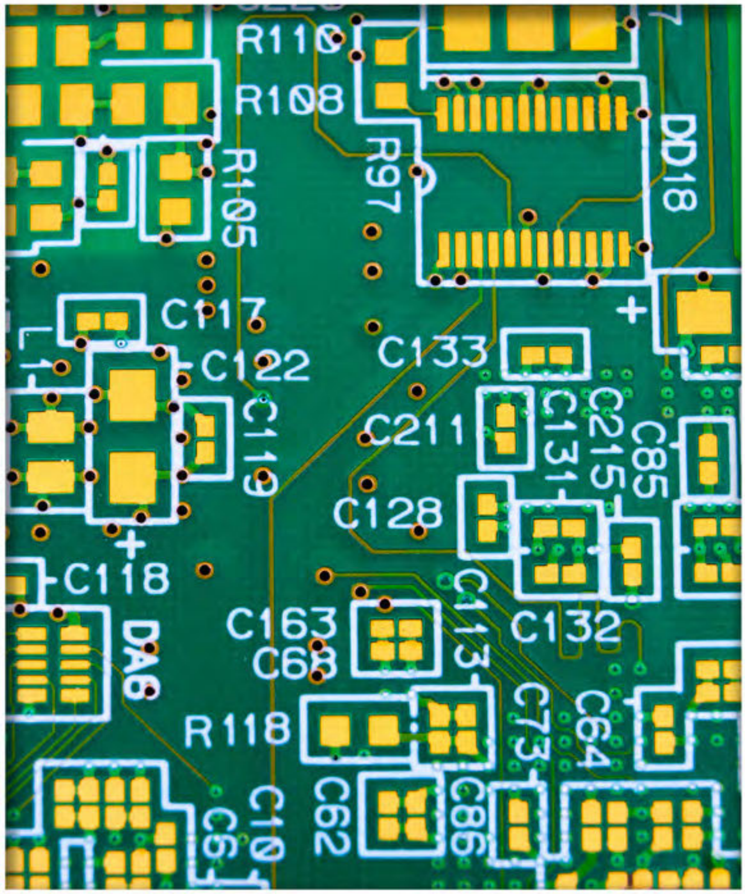
Ken Horky is a process engineer at Peterson Manufacturing. He may be reached at khorky@pmlights.com.

Graphene Enables High-Speed Electronics on Flexible Materials

A flexible detector for terahertz frequencies has been developed by Chalmers University of Technology researchers Xinxin Yang, Andrei Vorobiev, Andrey Generalov, Michael A. Andersson and Jan Stake, using graphene transistors on plastic substrates. It is the first of its kind, and can extend the use of terahertz technology to applications that will require flexible electronics, such as wireless sensor networks and wearable technology.

The detector has unique features. At room temperature, it detects signals in the frequency range 330 to 500 gigahertz. It is translucent and flexible, and opens to a variety of applications. The technique can be used for imaging in the terahertz area (THz camera), but also for identifying different substances. It may also be of potential benefit in health care, where terahertz waves can be used to detect cancer. Other areas where the detector could be used are imaging sensors for vehicles or for wireless communications. The unique electronic features of graphene, combined with its flexible nature, make it a promising material to integrate into plastic and fabric, something that will be important building blocks in a future interconnected world.

The research on the terahertz detector has been funded by the EU Graphene Flagship, the Swedish Foundation for Strategic Research (SSF), and the Knut and Alice Wallenberg Foundation (KAW).



The Effect of Area Shape and Area Ratio on Solder Paste Printing Performance

by Stefan Härter, Jens Niemann
Jörg Franke,
INSTITUTE FOR FACTORY AUTOMATION AND PRODUCTION SYSTEMS (FAPS);
Friedrich-Alexander
UNIVERSITY ERLANGEN-NÜRNBERG (FAU);
and Jeff Schake and Mark Whitmore
ASM ASSEMBLY SYSTEMS

The ongoing miniaturization trend in the SMT production induces new challenges and highly integrated systems. In passive components, the miniaturization leads to the introduction of the EIA size 01005 or smaller. Typical 01005 components are chip resistors and chip capacitors with the dimension of 0.4 mm x 0.2 mm. Despite numerous publications in this field already addressing the printing of such devices, a defined wholly optimized process remains unsolved and inspires further novel research ideas on this topic.

This paper focusses on the stencil printing process, because the highest amount of failure is assumed to be based on this process step.[1-4] Furthermore, the paper extends the preliminary work[5-7] by fundamental considerations. Thereby different values for the area ratio will be part of the investigation, which are purposely set to very low limits.

The influence of the aperture shape and orientation on the solder printing performance will be discussed. It is based on different forms of rectangles. Starting with a square the dimensions are incrementally changed, so that the square converts further into a rectangle. Furthermore, each rectangle is additionally rotated by 90° to be able to evaluate the influence of the apertures direction towards the squeegee.

In addition to the previously described stencil aperture attributes, this research also explores different stencil thicknesses, solder pastes and a variation of the squeegee speed. The evaluation of all data will be based on the two criteria of transfer efficiency and standard deviation. For both experiments the same stencil layouts and same solder paste are being used. The paper concludes with an outlook and suggestions on the modification of the current calculation by limitations of aperture dimensions.

- Prototype to Production
- Time Critical Production
- Commitment to Customer Service
- Best Quality & Delivery Performance
- Fair and Competitive Pricing/Costs
- Rapid Response to Quote Requests
- Flexibility Scheduling
- Stock/Consigned Inventory Programs

Review Our Advanced Capabilities Now

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

The Absolute Best Value in High Technology Circuit Boards

847.891.5800 • www.eagle-elec.com

Experimental Setup

The experimental setup includes three main variables, namely the layout by the PCB, the solder paste and the stencils used. A description of the experiment process is further explained.

PCB Layout

For the stencil printing experiments, a black anodized aluminum plate is used as substrate material with dimensions of 160 x 160 x 1.5 mm. This material is highly rigid and planar, representing a near perfect printing surface to minimize its influence on the print process outcome. Furthermore, the black aluminum material enables higher contrasts at the SPI, leading to more precise measurements. Figure 1 shows the general design of the printed solder paste deposits on the PCB.

Solder Paste

The stencil printing test also aimed to compare four different no-clean SAC305 solder paste formulations varying by type and by manufacturer. Solder paste of type 4 and type 5 were used. By the IPC J-STD-005 at least 80% of the alloy powder in a type 4 paste measures 20-38 µm while a type 5 paste contains the same ratio of alloy in 15-25 µm diameter particles. Due to the small dimensions of the tested apertures a difference in the printing performance attributed to particle size (i.e., type) is assumed a reasonable possibility. Two paste vendor sources were also included in this study, named A and B, which were supplied in both type 4 and type 5 products. As the distribution of the solder paste particles is comparable, A and B mainly differ in the composition of their flux systems which affects rheology and printing capability.

Stencils

In total three stencils were used for the experiments. Firstly, the stencils differ by their thickness and secondly by the size of their apertures (compare Table 1 to Table 3). The general structure is identical. The layout (Figure 1) can be divided into rows and columns. Each row

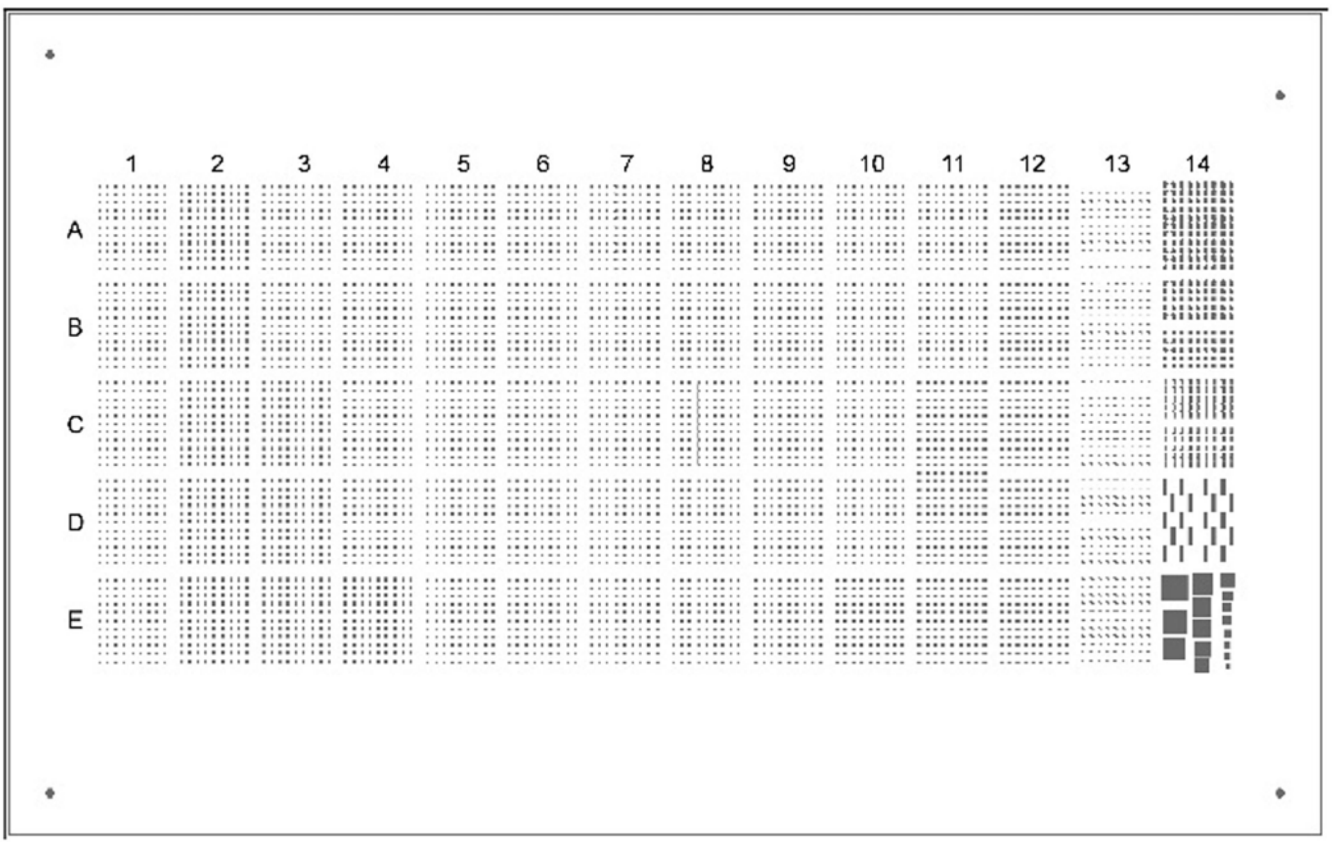


Figure 1: PCB and solder paste deposit layout.

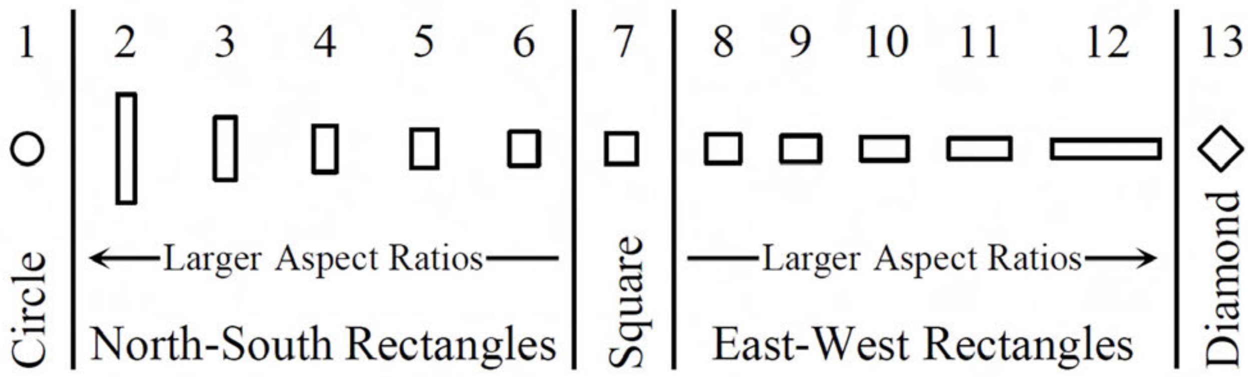


Figure 2: Column groups and aperture labels.

Area Ratio	Column	2 12	3 11	4 10	5 9	6 8	7
0.45	X [µm]	94	104	114	124	134	144
	Y [µm]	308	234	195	172	156	144
0.5	X [µm]	110	120	130	140	150	160
	Y [µm]	293	240	208	187	171	160
0.55	X [µm]	126	136	146	156	166	176
	Y [µm]	292	249	222	202	187	176
0.6	X [µm]	142	152	162	172	182	192
	Y [µm]	296	261	236	217	203	192
0.65	X [µm]	158	168	178	188	198	208
	Y [µm]	304	273	250	233	219	208

Table 1: Aperture dimensions, 80 µm stencil.

Area Ratio	Column	2 12	3 11	4 10	5 9	6 8	7
0.45	X [µm]	130	140	150	160	170	180
	Y [µm]	293	252	225	206	191	180
0.5	X [µm]	150	160	170	180	190	200
	Y [µm]	300	267	243	225	211	200
0.55	X [µm]	170	180	190	200	210	220
	Y [µm]	312	283	261	244	231	220
0.6	X [µm]	190	200	210	220	230	240
	Y [µm]	326	300	280	264	251	240
0.65	X [µm]	210	220	230	240	250	260
	Y [µm]	341	318	299	284	271	260

Table 2: Aperture dimensions, 100 µm stencil.

represents an area ratio, starting in Row A with AR of 0.45 and ending with AR of 0.65 in Row E. Each column in these tables represents different aperture shapes. Column 1 always has the shape of a circle, Column 7 is a square and Column 13 is the form of a diamond. Column 14 contains special types of structures that will not be discussed in this paper. Columns 2-6 and 8-12 are paired with the same dimensions and they only differ in their orientation towards the squeegee. Columns 2-6 face with the small side the squeegee (i.e., north-south orientation), whereby columns 8-12 face the squeegee with the long side of the rectangle (east-west orientation). Figure 2 illustrates the aperture design pattern across a row to show differences between the respective column groups of apertures.

The stencils used were made of stainless steel foil with laser cut apertures and produced for attachment in a Vector Guard (VG260) master frame system. A stencil nano-coating was not applied to any of these stencils.

Area Ratio	Column	2 12	3 11	4 10	5 9	6 8	7
0.45	X [µm]	116	136	156	176	196	216
	Y [µm]	1566	525	351	280	241	216
0.5	X [µm]	140	160	180	200	220	240
	Y [µm]	840	480	360	300	264	240
0.55	X [µm]	164	184	204	224	244	264
	Y [µm]	677	467	374	321	288	264
0.6	X [µm]	188	208	228	248	268	288
	Y [µm]	615	468	391	343	311	288
0.65	X [µm]	212	232	252	272	292	312
	Y [µm]	591	476	410	366	335	312

Table 3: Aperture dimensions, 120 µm stencil.

Print Test Procedure

Before the main experiments, a pretest was conducted to select the parameters for the squeegee speed, squeegee pressure and the separation speed using solder paste A. Parameters were determined by the DOE Optimization function within the Minitab software program and established per paste type. For each experiment 125 grams of solder paste material was used in combination with 200 mm long 60° SS squeegees. Table 4 identifies the equipment and print test parameters used in this study.

The test routine (Figure 3) consisted of starting with four knead printing cycles, followed by an automatic under stencil wipe and then proceeding to the formal printing test schedule whereby SPI data collection commenced. Ten boards were printed at fast squeegee speed followed by ten more boards printed at slow speed. This test print sequence was repeated twice each using both paste brands and types for all three stencil thicknesses. The under-sten-

Printing Machine Details	
Model	ASM DEK Horizon 03iX
Board Clamp	Foiled Overtop Type
Tooling	Grid-Lok Gold
Under Stencil Wipe	Wet/Vacuum/Dry ea. 5th Print
Squeegee	200 mm, SS, 60°, 15 mm O/H
Type 4 Paste Print Process	
Fast Print Speed	V = 80 mm/s, P = 5 kg, S = 6 mm/s
Slow Print Speed	V = 30 mm/s, P = 5 kg, S = 6 mm/s
Type 5 Paste Print Process	
Fast Print Speed	V = 55 mm/s, P = 6 kg, S = 2 mm/s
Slow Print Speed	V = 30 mm/s, P = 6 kg, S = 2 mm/s
SPI Machine Details	
Model	KohYoung 3020T
Camera	10 µm Resolution
Bare Board Teach	Not required due to the used substrate
Threshold	20 µm

Table 4. Experimental tools and setting.

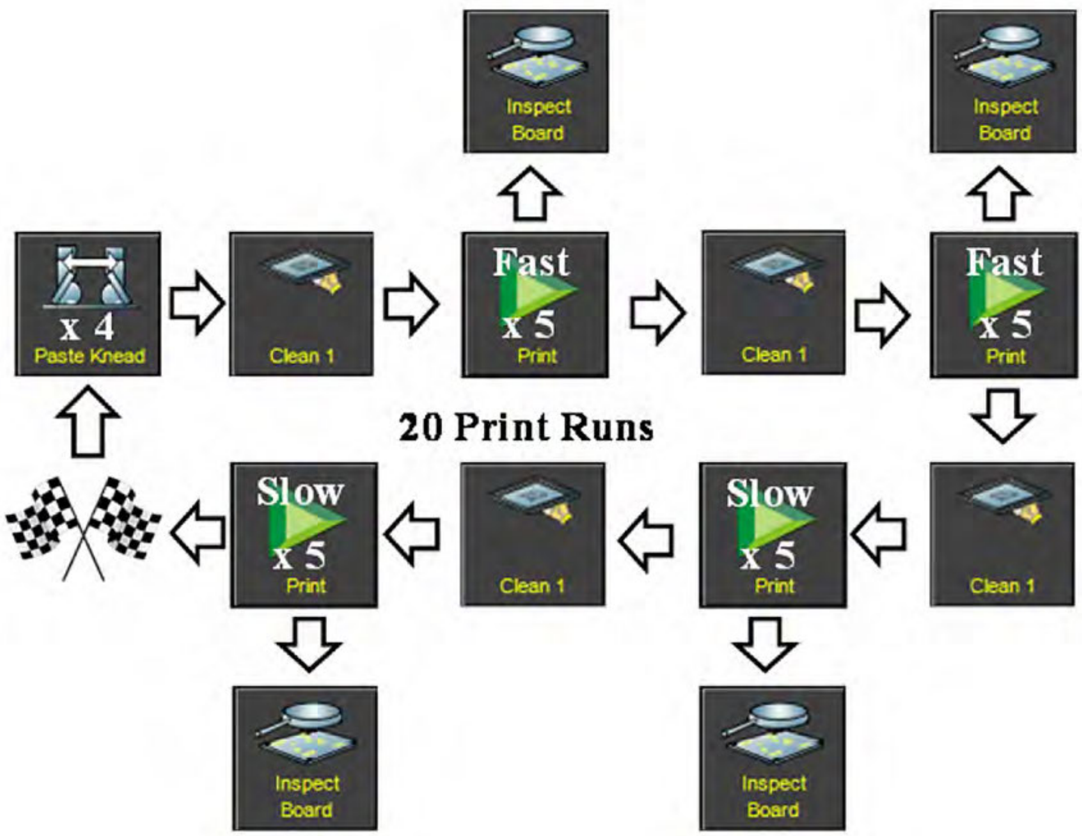


Figure 3: Print test sequence with 480 total prints.

cil cleaner was programmed to perform a wet/vac/dry cycle after every fifth consecutive print. The complete DOE comprised 480 solder paste inspected boards.

Effects of the Aperture Shape

The main object in this experiment is the investigation whether the aperture form or shape has an influence on the printing results. In the first part, the general result will be taken into consideration with the analysis of all measured data of the transfer efficiency. In further steps the result will be filtered by the criteria of the solder paste type and the stencil thickness. The form comparison is based on adjacent apertures 2 vs. 3 vs. 4 etc. vs. 12. The shape can be analyzed using columns 1 vs. 7 vs. 13. Based on square apertures in columns 7, the orientation is observed comparing 2 vs. 12, 3 vs. 11, etc. up to 6 vs. 8 with the slightest varying aperture form.

Printing Results

With about 8% higher transfer efficiencies, square apertures perform better than the circular structures. Further analyses of the basic forms indicate no significant difference between square and diamond. In the following, the variations of the rectangular apertures will be discussed.

Figure 4 shows the overall view on the transfer efficiency depending on the area ratio and the form. It includes the results of all printed boards. The form is visualized by the column number (Figure 2). In general, the figure proves the plausibility and the correctness of the data, because the transfer efficiency rises with increasing area ratios. Furthermore, the measured values show a clear tendency that the aperture form and orientation have an influence. If the aperture form is taken into investigation the high aspect ratio east-west oriented rectangles exhibit highest transfer efficiency (column 12). Beginning from the square (column 7) both orientations of rectangles perform better than the square, which seems to be nearly the minimum of every graph. In addition, the aspect ratio (the difference between the wide and small side of a rectangle), which is rising towards the ends of the graph, shows possible correlation with the transfer efficiency.

Besides the aperture aspect ratio, the aperture orientation shows evidence to influence the process. The apertures representing columns 2-6 have north-south orientation (i.e., the small side of the rectangle faces the squeegee) while the columns 8-12 are east-west oriented (i.e., the wide side faces the squeegee). Especially the area ratios 0.45 and 0.5 show the clear tendency that the aperture orientation has an influence. The transfer efficiency of the 0.45 area ratios raises about 3.5% for north-south orientation and about 8.8% for east-west oriented apertures compared to squares. This trend weakens with increasing area ratios but remains recognizable throughout all graphs. If the standard deviation is investigated the insights show that the east-west orientation also leads to a slightly higher standard deviation at low area ratios. For this parameter the squares show better results. The standard deviation for the area ratios 0.55 to 0.65 shows no significant differences and no trends can be observed.

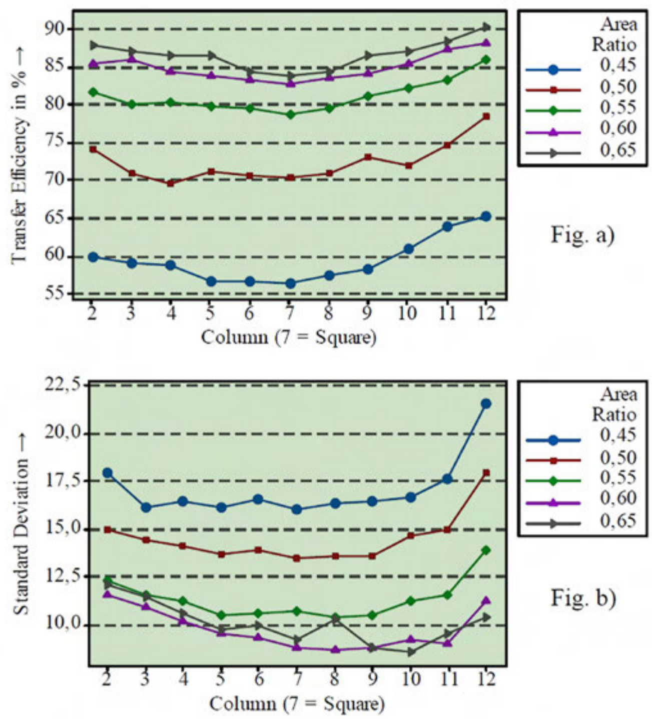


Figure 4: Influence of aperture shape and orientation based on transfer efficiency (a) and variation (b).

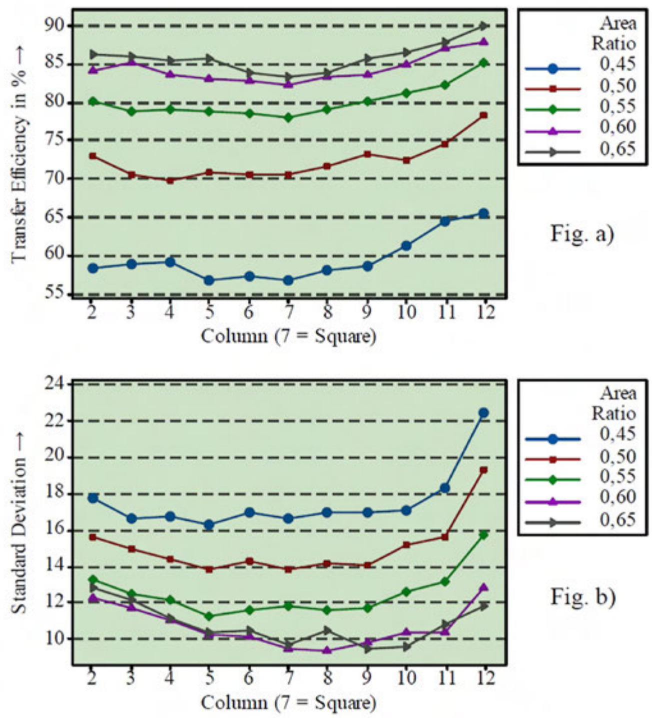


Figure 5: Influence of the aperture shape and the orientation for solder paste type 4.

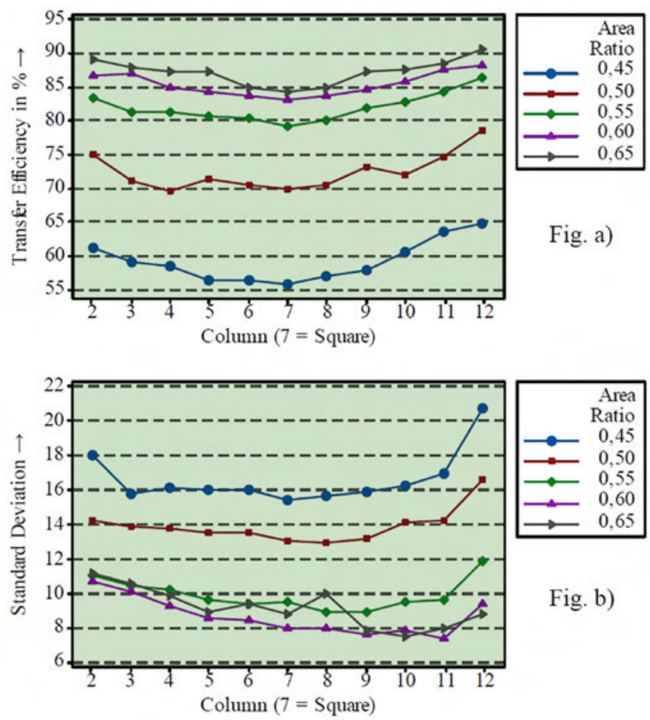


Figure 6: Influence of the aperture shape and the orientation for solder paste type 5.

Solder Paste Type

Due to the bigger assumed influence using a solder paste with coarser particles the first analysis will be focused on solder paste type 4. The general appearance of the trends in Figure 4 (combined test results) seems to be consistent with Figure 5 (type 4 paste results). The graphs show that the rectangle shape and the orientation have an influence. The effect in general seems not so obvious compared to Figure 4 but can still be recognized. The solder paste type 4 shows that the transfer efficiency of the 0.45 area ratios raises about 1.6% for north-south orientation and about 8.7% for east-west orientated apertures compared to squares.

The investigation of the standard deviation shows that the values for the two orientations are comparable for the variations 5-9. The biggest aspect ratios for the north-south orientation lead to a slightly increasing standard deviation for columns 2-4. The east-west orientation in column 10-12 implies the highest standard deviation for all area ratios. Based on further examinations slightly better printing results are provided using solder paste of type 5, as illustrated in Figure 6. Especially at area ratios of 0.55 and above a higher transfer efficiency is observed. The effect at area ratios of 0.5 and 0.45 is still measureable, but less marked.

The standard deviation is more evenly distributed using solder paste of type 5 and shows less variance compared to a solder paste of type 4. The solder paste type 5 shows that the transfer efficiency of the 0.45 area ratios raises about 5.4% for north-south orientation and about 9.1% for east-west oriented apertures compared to squares.

Stencil Thickness

Following industry accepted area ratio principle, thinner stencils are considered to provide better release behavior for miniaturized structures. While use of reduced stencil thickness raises the area ratio for identical apertures, this stencil design strategy also subtracts volume capacity from the aperture. Within our investigations three stencil thicknesses are examined.

The effect of the 80 µm stencil thickness is illustrated in Figure 7. The graphs in general show a consistent behavior of the transfer efficiency. The measured values rise along with higher area ratios. Apertures with east-west orientation again show higher transfer efficiencies. Against the overall view, the effect seems to have a maximum for columns 5 and 9 at area ratios of 0.5 and above. AR of 0.45 shows a continuous trend to higher transfer efficiencies with a decrease in the outer shapes. The analysis of the standard deviation for the 80 µm stencil thickness shows the clear trend for more unwavering distributions at higher area ratios. Again, the rectangular aperture shows a reduced standard deviation but against the overall view no significant variance in the outer columns occurs.

The analysis of the measured data for the 100 µm stencil thickness shows significant differences in the transfer efficiencies, as shown in Figure 8. Area ratios above 0.55 consistently print above 75% transfer, while a strong decrease is observed for the area ratios 0.5 and especially 0.45. Compared to the 80 µm thickness no significant decrease of the transfer efficiency can be observed. Excluding the area ratio 0.5 data, all others have the common trend where the highest aspect ratio rectangles track lower in transfer efficiency relative to neighboring data points.

The standard deviation of the 100 µm stencil follows the overall trend with higher deviations for the area ratios 0.45 and 0.5. Area ratios above 0.5 show the minimum for the variance in the area of the rectangular shape. A strong influence is detected for north-south orientated apertures resulting in higher values for the standard deviation towards column 2.

The average values of the 120 µm stencil show the highest transfer efficiency of all stencils and it seems to be the stencil with the highest ability to perform the prints. The increase of the transfer efficiency for the east-west aperture orientation is clearly recognizable as illustrated in Figure 9.

Similar to the trends shown in Figure 4, the two smallest area ratios have marked transfer efficiency differences between the two aperture orientations, particularly at high aperture aspect ratio. Conversely, the transfer efficiencies produced by the north-south aperture orientation for the two smallest area ratios are less sen-

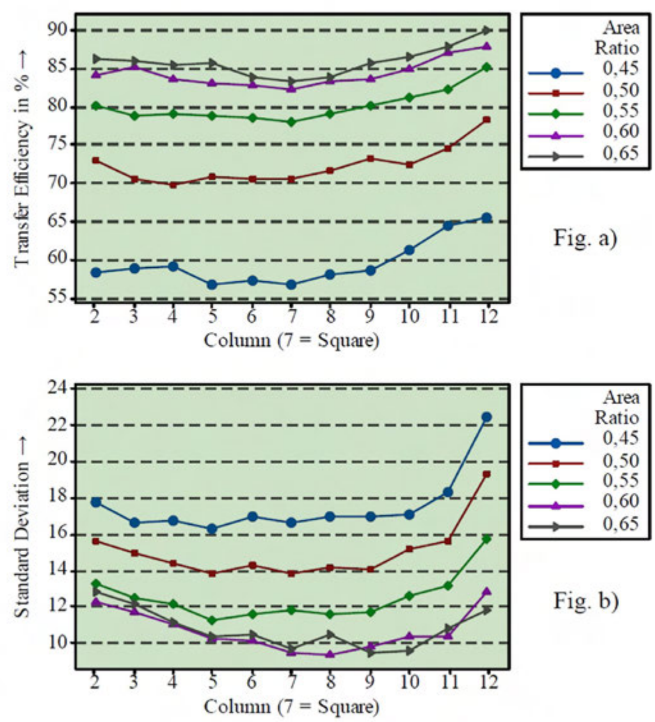


Figure 7: Influence of the aperture shape and the orientation for the 80 µm stencil.

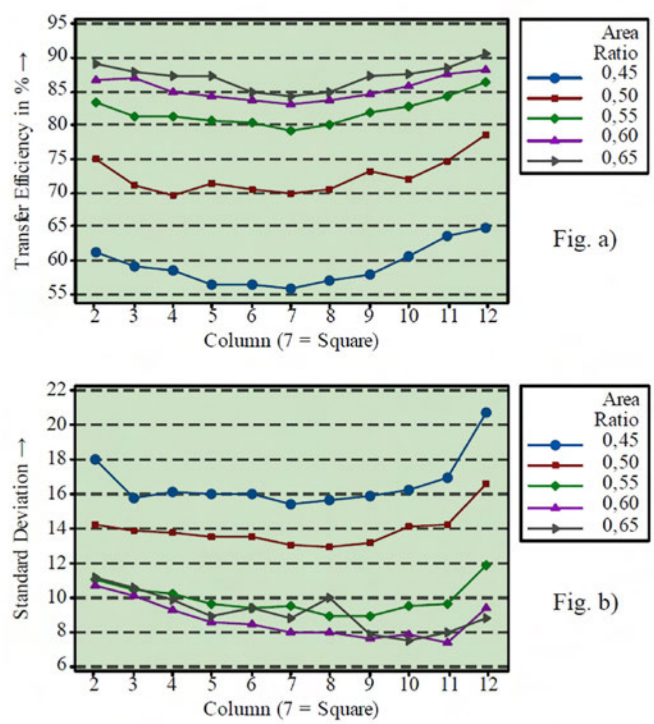


Figure 8: Influence of the aperture shape and the orientation for the 100 µm stencil.


FAILURE HERE
IS NOT AN OPTION
Our ‘Can't-Fail Scenario’ Attitude is built on 40+ years of industry-wide reliability and experience.
We specialize in HDI Rigidflex for Mil/Aero and Medical applications, are certified to NADCAP and AS9100 as well as meeting all applicable ITAR Compliancy requirements.
Let us help extend the capabilities, reliability and technology of your next design!
LEARN MORE ABOUT OUR HDI SOLUTIONS
eltek
a Nistec company
www.nisteceltek.com

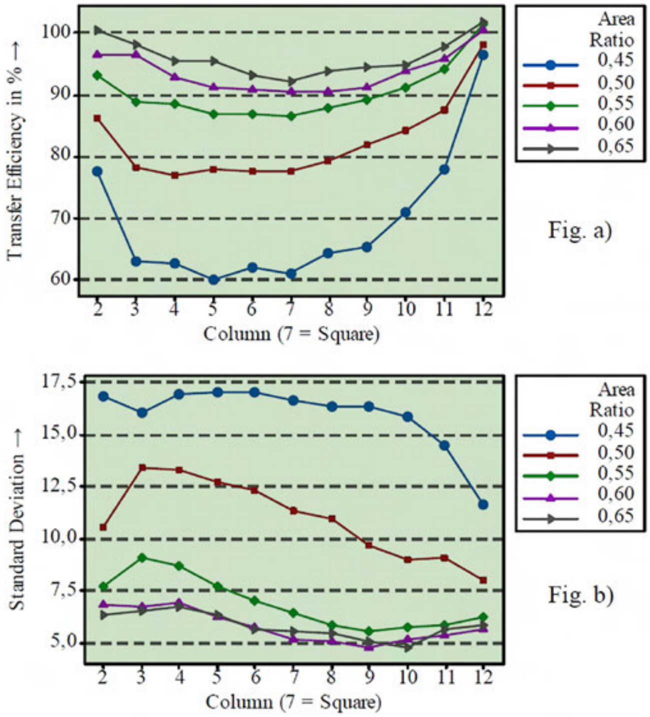


Figure 9: Influence of the aperture shape and the orientation for the 120 μm stencil.

sitive to changes in aspect ratio. In general, the rectangles with the east-west orientation typically achieve the highest transfer efficiency, especially at low area ratios.

In comparison to the other stencil thicknesses the course of the graphs show some deviation especially concerning the effect of the orientation. One reason might be the difference on the absolute size of the aperture and the different shapes of the aperture depending on the stencil thickness. Thereby the 120 μm stencils occupies the biggest apertures with distinctive differences in their shapes.

The course of the standard deviation has all the same remarkable points in common. The trends in Figure 9 show a decrease of the standard deviation starting at the square and following the east-west orientation before they undergo a rise in columns 11 and 12 for area ratios of 0.55 and above.

Area ratios show an ongoing decrease also for apertures with lowest values of the standard deviation in column 11 and 12. In contrast the vertical apertures in north-south orientation have a higher standard deviation than a square. Overall the course of the graphs for the 120 μm stencil thickness is not consistent and no general trend can be observed.

Discussion

The results of the experiments comparing the aperture form show the influence on the stencil printing process especially for small area ratios below 0.6. The upcoming discussion introduces some ideas for the integration of the results in present standard rules.

The first insight of the experiments shows that a rectangle shape leads to better results than the square. Combined with the analysis of the influence of the stencil thickness it is known that the effect depends on the size of the aperture. All stencils have the same area ratio, which also means that the absolute size of each aperture is not identical. Combining the aspect ratio with the transfer efficiency, a higher aspect ratio might lead to higher transfer efficiency. Transferring this point to the orientation of the aperture evidence of printing improvement under test conditions can be found where the wide side of the aperture faces the squeegee.

Continuing with the aperture orientation analysis it was found out that there are differences between the performance of the east-west and the north-south directions. This leads to the idea to extend the common calculation of the area ratio. The analyzed data has shown the orientation and form of the aperture influences the printing results, particularly at small area ratios where unexpectedly high average paste transfer occurred. To include this thesis in the area ratio calculation a levelling factor (short: NF) will be introduced.

Under the present condition it raises the real area ratio to a higher level by including the form and orientation of the aperture. Thereby a new area ratio is calculated, which could serve as the new decisive area ratio. To develop the levelling factor the slope of the average improvement of the transfer efficiency was calculated. For this experiment a nearly linear coherence can be determined. The idea is to use the determined slope, which was calculated on a transfer efficiency base and transfer it to calculate the area ratio. The slope will be multiplied with the difference of the length of the smaller

side of the square and the aperture, to include the form of the aperture. Based on the results, the NF-factor can be calculated as:

$$NF = 1 + \frac{0.15\left(B_{square} - B_{aperture}\right)}{100}$$

To calculate the new area ratio the NF-factor will be added to the common area ratio calculation:

$$AR = \frac{area\ of\ aperture\ opening}{area\ of\ aperture\ walls} * NF$$
$$= \frac{A * B}{2 * T * (A + B)} * \left(1 + \frac{0.15\left(B_{square} - B_{aperture}\right)}{100}\right)$$

This extended formula applied to conventionally low area ratio east-west oriented apertures is considered to improve accuracy of area ratio specification for such aperture designs. While the apertures with north-south direction still show better transfer efficiency results compared to squares, the correction by the NF implementation reduces such bias. This exercise serves as a first idea to rethink common rules and consider new requirements provoked by miniaturization to find new ways of achieving more process adapted stencil designs.

Conclusion and Outlook

This paper discussed the effect of the aperture shape and orientation on solder paste printing performance. For the experiments three different stencil thicknesses (80 µm, 100 µm and 120 µm), two different solder paste types (Type 4 and 5) and two manufacturers were used.

For apertures with identical area ratios a strong influence is induced by a varying aspect ratio and the orientation of these structures to the printing direction. The results focusing on the transfer efficiency and the standard deviation show, that the shape of a rectangle and an east-west orientation achieve the best results, as illustrated in Figure 10.

Furthermore, the solder paste type can have an influence, especially when relatively narrow apertures are printed. Based on the boundary conditions of the PCB layout within the printing tests the thicker stencils have the higher aspect ratios and achieve better printing results.

In general, the printing results show that also area ratios lower than 0.66 can be quite regularly printed above 75% transfer efficiency, with dependencies on the shape of the apertures. This leads to the idea to extend the current calculation of the area ratio by introduction of the leveling factor NF. Using NF for the calculations, the present assumed design limita-

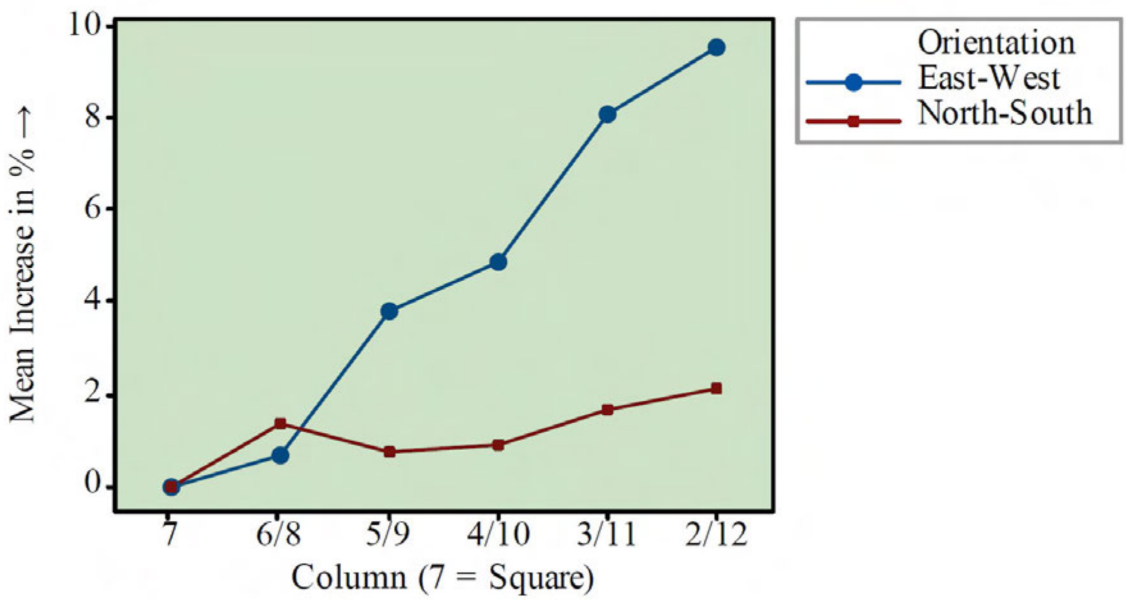


Figure 10: Mean increase of the transfer efficiency in relation to the aperture shape and orientation.

tions are extended to regard the orientation and shape of the stencil apertures.

Additional research has to be done to find proof for the ideas given in this paper. Different types of solder pastes and stencil thicknesses extend the data basis and help to refine the first introduced modeling. This could also lead to nonlinear correlations that have to be included for more precise calculations. New materials as finer grained solder paste types, stencil technologies and stencil coatings shift the limits for obtaining a well-controlled stencil printing process. Additionally, new calculations based on fundamental printing tests contribute to understand printing behavior of miniaturized structures to achieve a robust solder paste printing performance. **SMT**

References

1. K. Vijay, "Miniaturization—Solder Paste Attributes for Maximizing the Print & Reflow Manufacturing Process Window," IMAPS Europe (Hrsg.): 18th European Microelectronics and Packaging Conference (EMPC), Brighton, pp. 26–29, 2011.

2. C. Combet and M.-M. Chang, "01005 Assembly, the AOI route to optimizing yield," Vi TECHNOLOGY, Jul. 2009.

3. R. S. Clouthier, "The Complete Solder Paste Printing Processes," (en), SMT Magazine, vol. 13, pp. 6–8, 1999.

4. C. H. Mangin, "Where quality is lost on SMT boards," Circuit Assembly, no. 2, pp. 63–64, 1991.

5. S. Härter, "Evaluation of the Stencil Printing for Highly Miniaturized SMT Components with 03015 mm in Size," in Proceedings of Surface Mount Technology Association International Conference (SMTAI), Rosemont, Illinois, 2015, pp. 756–764.

6. S. Härter, C. Läntzsch, and J. Franke, "Fundamental study on a secure printing process using NanoWork stencils for 01005 components," in Proceedings of Surface Mount Technology Association International Conference (SMTAI), Rosemont, Illinois, 2014, pp. 878–884.

7. M. Whitmore, J. D. Schake, and C. Ashmore, "The impact of stencil aperture design for next generation ultra-fine pitch printing," in Proceedings of the 35th International Electronics Manufacturing Technology Conference, 2012, pp. 1–7.

Editor's Note: This article was originally published in the proceedings of SMTA International.

Jeff Schake is a senior engineer, Advanced Print Technologies Printing Solutions, of ASM Assembly Systems LLC.

Mark Whitmore is the senior manager, Advanced Print Technologies Printing Solutions, of ASM Assembly Systems LLC.

Stefan Härter (chief engineer)*, **Jens Niemann** (scientific assistant)*, and **Jörg Franke** (head of chair) are from the Institute for Factory Automation and Production Systems (FAPS), Friedrich-Alexander-University Erlangen-Nürnberg (FAU).
** At the time of writing.*

Cleaning Trends: The Challenges of Miniaturization and Proximity

At the recent SMTA International conference in Rosemont, Illinois, I-Connect007 Managing Editor Patty Goldman caught up with Ram Wissel, VP of Global Technology at KYZEN, to talk about the latest cleaning challenges, and bringing Industry 4.0 into the world of cleaning in PCB assembly. They also discussed the misconception about "no-clean", the challenges it brings, and how the company is helping customers to address their issues toward no-clean solders and fluxes.

Read the full interview here.

Supply Lines
Highlights

Solder Preforms 101: Ask the Expert
At the recent SMTA International 2017 event, Jerry Sidone, product manager for the engineered materials at Alpha Assembly Solutions, speaks with I-Connect007 Managing Editor Patty Goldman about the voiding challenge, especially with bottom termination components, and how solder technologies from Alpha are helping PCB assemblers address this issue.

Cleaning Trends: The Challenges of Miniaturization and Proximity
At the recent SMTA International conference, I-Connect007 Managing Editor Patty Goldman caught up with Ram Wissel, VP of Global Technology at KYZEN, to talk about the latest cleaning challenges, and bringing Industry 4.0 into the world of cleaning in PCB assembly.

Mycronic Acquires Vi TECHNOLOGY
Mycronic AB acquires 100% of the shares in VIT S.A.S (Vi TECHNOLOGY), situated in Saint Egrève in France. The purchase price amounts to €8 million (approximately SEK 77 million) on a debt free basis and will be financed through own funds.

Koh Young and ASM Assembly Systems Working with KSMART Partners Toward Zero-Defect Line
Koh Young Technology is collaborating with its KSMART partners like ASM Assembly Systems to realize machine connectivity and create a zero-defect production line.

Panasonic Starts Accepting Orders for Manufacturing Operations Optimizer MFO
Panasonic Corporation has started accepting orders for a new software solution, "Manufacturing Operations Optimizer MFO", a system that improves productivity by utilizing a simulation model that accurately emulates the mounting process of electronic components on printed circuit boards.

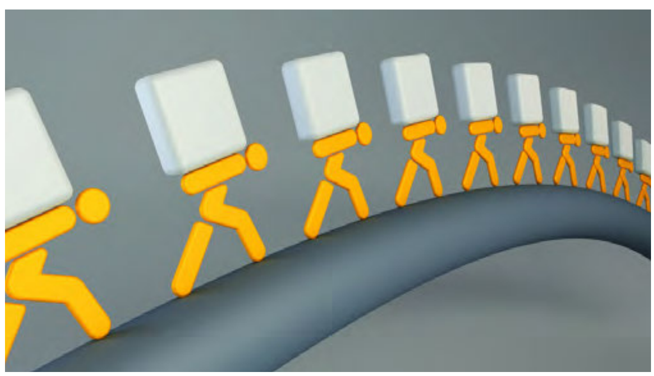
Cogiscan Presents Accelonix France with Business Development Achievement Award
Cogiscan Inc. has awarded Accelonix France with its Business Development Achievement Award to express their appreciation of the team's contribution to their business development.

Picosun Technology Improves Lifetime and Reliability of PCBAs
Picosun Oy has invented an ALD-based method with which the operational lifetime and reliability of printed circuit board assemblies (PCBAs) can be efficiently improved.

Indium Announces New Corporate Leadership
Indium Corp. has promoted Greg Evans to CEO, and Ross Berntson to president and COO. Former CEO and company owner William Macartney III will continue serving as the chairman of the board.

AIM Promotes Rodrigo Cacho to Business Development Manager
AIM Solder has promoted Rodrigo Cacho to the position of business development manager.

Electrolube Awarded IATF 16949 Automotive Standard
Demonstrating its continuing commitment to achieving excellence in manufacturing and service provision, Electrolube has been awarded the International Automotive Task Force's new global quality management standard, IATF 16949.



Solder Printing Process Inputs Impacting Distribution of Paste Volume

by Marco Lajoie and Alain Breton
C-MAC MICROELECTRONICS

The volume of solder deposition, like any process, has variations that may be characterized by a statistical distribution curve, whether normal (from causes inherent to the process and of predictable probability) or non-normal (impacted by special causes, intermittent or inherently unpredictable). For assemblies of moderate complexity, density, cost, and reliability requirements, one may tolerate fairly large variations in volume and still obtain acceptable yields and quality of solder joints, assuming that a solder paste inspection (SPI) will effectively segregate gross defects like bridges and misalignment. Surely, this eases the task since a few hours of engineering suffice to launch a new product using standard recipes, and manufacturing may flow with minimal interruptions from SPI calls or process inputs sliding off tight windows of operation.

Why Does it Matter?

As complexity, density, cost and reliability requirements increase, there may be value in narrowing the distribution curve. It is common sense that less variation serves the interest of quality of the more complex and dense circuit boards. It can also affect the cost as it should increase the yield and reduce rework and scrap. It does this by reducing the normal variations but also by revealing the outliers, driven by identifiable special causes which, otherwise, would have remained hidden in larger acceptance windows.

Reliability is typically assessed by tests performed on relatively small sample size of products from a limited number of lots, sometimes repeated at relatively long intervals. In other words, how much of the actual distribution was represented in the samples tested and how different are the parts produced between test intervals? Narrower distribution should help ensure reproducibility of reliability performance across test samples and production units alike.

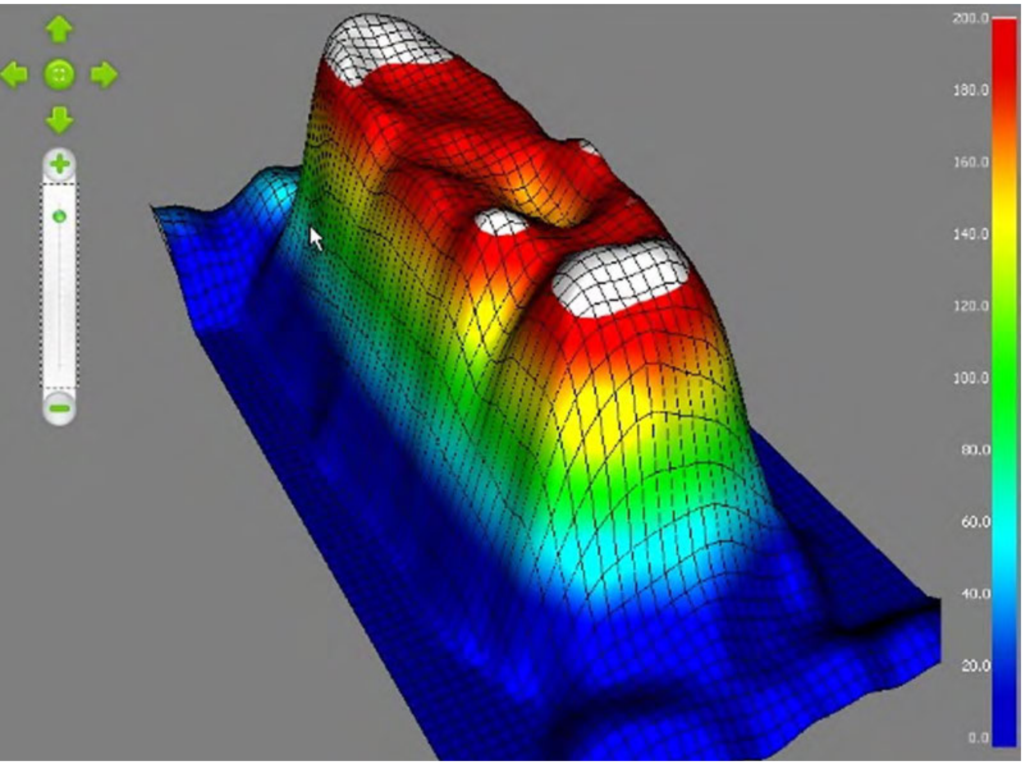


Figure 1: The volume of solder deposition has variations that may be characterized by a statistical distribution curve, whether normal or non-normal.

What Target?

Common sense dictates that narrower is better. But how narrow is enough? A typical industry target for solder joint quality is IPC-610, with subclasses 1 to 3. Performing a simple DOE, one would find that most IPC criteria can be met across a fairly large distribution of solder volumes. However, mere compliance to IPC may not meet expectations when confronted with the subtle causes of defect modes like tombstones, voiding in thermal planes, or open corner balls of BGA. Besides, IPC criteria defining ranges of acceptable solder volume from “wetting is evident” to “solder shall not touch a package body” hardly sounds like a valid predictor of product reliability under harsh conditions.

In other words, you are left with the task of identifying the weak spots: your highest pareto causes in yields, your first fail modes to occur in highly accelerated life testing (HALT) or other test-to-fail methodology. Then, identify best nominal targets and narrow distribution for each, most of the time iteratively until you consider having reached stability. This is lots of work, far from the ease of using standard recipes. With experience and methodology, patterns and targets get more predictable, and the investments start paying off.

What Process Inputs?

In search of better performance criteria (e.g., yield, reliability…) once you suspect that distribution of solder volume is a significant contributor, what process inputs may impact your distribution? Consider the 6M classification: measurement, machine, material, mother nature, method, manpower.

Measurement

“Know thy SPI” might say the Greek philosopher Thales, if assigned to the task. Measurement starts with proper measurement system analysis (MSA), which includes calibration and Gage repeatability and reproducibility (R&R). However, wet solder paste samples may not remain sufficiently stable through the course of the analysis. Options include, for calibration: a 3D reference of a known volume, traceable to international standards; for Gage R&R: dried (long stabilized) paste, with care to present samples across the entire range of measurements.

Additionally, SPI measurements may include various artifacts originating from: incorrect reference plane (referencing to solder resist instead of copper), PCB warping, shadowing from adjacent solder depositions, etc. The most recent generations of SPI include specific features to address these. Chances are that these are minor and only impact the few measurements that are so critical in your application. The point is to understand the limitations of your measurements, so their variations are not confused with other process inputs, or lead you to wrong conclusions.

Make measurement part of a formal control plan (i.e., who, what, when, how is the process measured, under what control limits, and what reaction is expected if out of range). Include technical management as part of reaction plans to first-time quality (FTQ) or SPC signals, so they can intervene in close time and proximity while the trail is hot, and continuously improve the process.

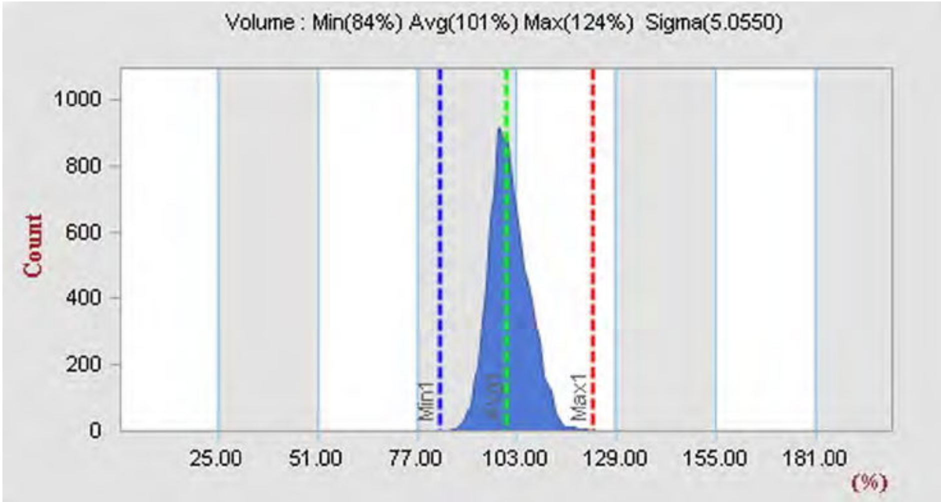


Figure 2: How narrow is enough?

Machine

Machine includes process equipment and tooling. Assuming programming and maintenance are executed thoroughly, printers are usually stable for the purpose unless anomalies went undetected, like: planarity of stencil mounting, measurement from the load cell (pressure), squeegee mounting, etc.

As for tooling, ask your supplier for the tolerances of your stencil. On a given unit, variations in thickness and apertures alone may cause significant variations between circuits across the same panel. Similarly, ordering a new stencil from the same image may not yield the same results. Also, the limits and trends of the tolerances may differ between fabrication methods (laser cut, electroformed, nanocoated, etc.). Finally, consider its maintenance plan: How efficient is cleaning after use? At what frequency and under what criteria is it evaluated for reuse?

Squeegees are also a key variable for solder deposition. Include blade material (metal or polyurethane) and maintenance protocol (wear, cleanliness) in your potential variables.

Finally, board support matters, with a solid block yielding the most repeatable results. Obviously, printing on second side forces to trade support for component clearance and strategies differ: blocks with custom cavities, grid-patterned automated posts, or manually placed posts. Some PCB designs and types (thin, odd-shape, flex…) may require process carriers whose design (planarity, clamping) are of prime importance.

Material

PCB design and construction must be factored in. Solder mask defined pads or proximity of solder resist influence gasketing of the stencil; the impact is worsened by the typically loose tolerances on thickness (>0.5 mil range). The variability may be reduced by under-etching the stencil in solder resist areas close to the target pad. Also, surface finish may play a role, especially with HASL (relatively thick and uneven).

One must also pay attention to panelization, leaving enough distance for the squeegee to land and pull-up away from the clamping mechanism (i.e., in the area uniformly stabilized with board support).

Paste is at the heart of the matter. Minimize variations in its thixotropic properties by managing homogeneity (automated mixing is preferred), stabilization at room temperature, and elapsed time from stencil to reflow. Avoid chemical or particulate contamination by only using stencil solvent and clean rolls designed for the purpose.

Mother Nature

Control of the process environment is critical. Ambient temperature and humidity, combined with time, will impact thixotropy of paste. Reduce particulate contamination with tight management of cleaning procedures (equipment, shelves, tooling, ceilings…) using window covered conveyors, and strategically locating ionized air blowers along the path from bare panel to reflow.

Method

Printing parameters such as pressure, speed, separation, etc., are programmable and standard recipes are called up and loaded at job set-up. Ensure tight management of parameters and recipes, so they remain constant from batch to batch, and are not left to random trials in reaction to process outputs.

Yet, many job set-up elements rely on manual interventions: mounting stencil, squeegee, and board support, dispensing paste (unless auto-dispense is used), cleaning stencils, squeegee, process carriers, and equipment, etc. Thoroughly document procedures for each step and only allow formally trained people to operate the process.

Perhaps the most important, yet most complex "manual" operation is the response to SPI alarms: are calls allowed to be interpreted as "false" and overrun? Are there different categories of "true" calls and different panel disposition for each? What are the criteria to stop and fix (call technical support)?

Manpower

Sound methods, procedures, and training remain theory, subject to mishaps in each execution. Minimize deviation by error-proofing the methods: visual indicators, templates, sensors, alarms, etc. Schedule short and frequent audits by peers and management. Include serialization and traceable records so causes of deviations can be investigated. Communicate methodically with your operators, providing feedback on problems, their cause, their fix, and means of prevention, so lessons learned are shared and engage your entire organization in a constant learning process.

Conclusion

Narrowing the distribution of solder paste volume benefits product quality, reliability, and cost. Several process inputs impact the distribution and are reviewed following the 6M classification. Process, equipment, and human factors are included, with guidelines for best practices. **SMT**

Alain Breton is the director of operations at C-MAC Microcrocircuits ULC.

Marco Lajoie is a business development manager at C-MAC Microcrocircuits ULC.

Electronics Industry News
Market Highlights

3Q17 DRAM Revenue Hits a New High with a 16.2% Sequential Increase

Revenue of the entire global DRAM industry climbed to a new historic high for the third quarter of 2017, driven by the 5% average increase in contract prices of various DRAM products.

Pure-play PLC Smart Meter Shipments in Europe will Peak in 2017

Annual shipments of smart electricity meters in Europe will grow by 71% in 2017 to reach 23.1 million units, according to a new research report from the M2M/IoT analyst firm Berg Insight.

Flexible AMOLED Panel Supply Capacity to Exceed Demand by 44% in 2018

With flexible active-matrix organic light-emitting diode (AMOLED) panel fabs building at a quicker pace than global demand, supply capacity of flexible AMOLED panels is forecast to be 44% higher than global demand in 2018, according to IHS Markit.

Global Wearable Medical Devices Market Analysis & Trends 2017

The global wearable medical devices market is poised to grow at a CAGR of around 17.4% over the next decade to reach approximately $23.8 billion by 2025.

Gartner Identifies Top 10 Digital Disruptors in Asia/Pacific

The top digital companies in the Asia/Pacific region represent both a threat and an opportunity to global enterprises, and CIOs ignore them at their own risk, according to Gartner Inc.

Smart TV Market Size to Reach $292.55B by 2025

The global smart TV market is expected to reach $292.55 billion by 2025, according to a new report by Grand View Research Inc.

North American Semiconductor Equipment Industry Posts September 2017 Billings

North America-based manufacturers of semiconductor equipment posted $2.03 billion in billings worldwide in September 2017 (three-month average basis), according to the September Equipment Market Data Subscription (EMDS) Billings Report published by SEMI.

Semiconductor Demand in Medical Applications

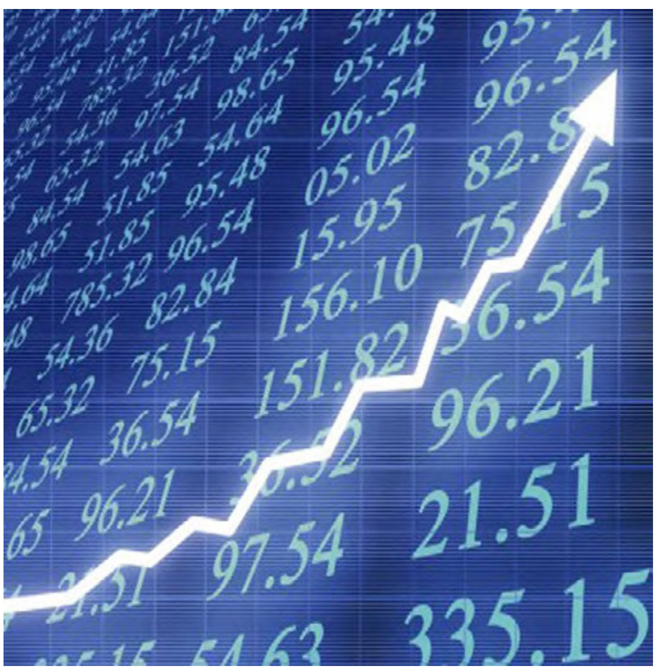
The global opportunity in the medical electronics market continues to motivate semiconductor suppliers to bring to market products for imaging, diagnosis, therapy, and home remote monitoring.

Automated Test Equipment Market Size Worth $4.6B by 2025

Increasing adoption of system-on-chip and high demand for consumer electronics are the key drivers for the automated test equipment market growth.

Domestic Supply Chain Shuffle: Increasing the Value of Local Content in Semiconductor Equipment

Over 20% of all semiconductor equipment will be shipped to China in 2017, yet Chinese OEMs account for only 2% of worldwide fab equipment. This percentage is even lower for Chinese subsystems and components suppliers.




STRONG
FOUNDATION

Evaluation of Stencil Technology for Miniaturization

by Neeta Agarwal, Robert Farrell, and Joe Crudele, BENCHMARK ELECTRONICS INC.;
Chrys Shea, SHEA ENGINEERING SERVICES;
Ray Whittier, VICOR CORP.;
and Chris Tibbetts, ANALOGIC CORP.

SMT stencil printing technology continually evolves to keep pace with device miniaturization technologies. Printed circuit board assemblers have numerous new technology options to choose from, and need to determine the most effective ones to produce the highest quality and most reliable solder interconnections.

The objective of these tests was to identify the best stencil technology for high-volume production of miniaturized SMT components. The solder paste used for this assessment was SAC305 Pb-free no-clean, Type 4 mesh. The specified stencil thickness for all stencils was 4 mils (100µm).

EXPERIMENTAL SETUP:

Test Vehicle

The test vehicle shown in Figure 1 was designed in-house for a multitude of PCB assembly tests, including new packages, pad designs, solder paste print performance and process evaluation tests. The devices selected for analysis in these tests included 0.3- and 0.4 mm-pitch BGAs and 0201s. Their area ratios ranged from 0.46 to 0.70. Locations and names of the specific devices used in the stencil analysis are shown in Figure 1.

The recommended stencil thickness for the 0.3mm microBGA location is 3 mils (75 µm) versus the 4 mil (100 µm) thickness selected for this study. Consistent release for these apertures was not anticipated but relative comparisons of

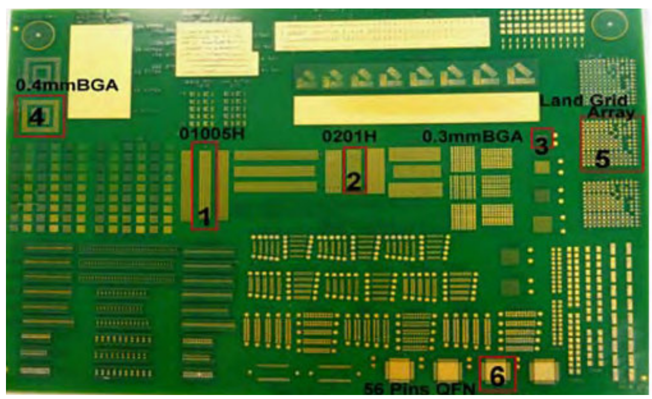


Figure 1: Test vehicle and features used in stencil analysis.

release would provide an indication of stencil performance.

Test Design

The stencil analysis included:

- 6 different stencil suppliers
- 9 different foil materials
- 5 different manufacturing processes
- 7 different nanocoatings

The experimental design was not a full factorial. Each supplier provided stencils using technologies that were either their top performers (high end), developmental technologies that they wanted to learn more about (supplier choices #1 and #2), or lower cost stencils commonly ordered (AR>0.70). One to four stencils were submitted by each supplier. A total of 18 stencils were print tested. All were created using the same Gerber file, and all were specified at 0.0040" thick with identical apertures depending on the feature. The stencil test matrix is shown in Table 1.

Supplier Letter	Stencil No.	Type
A	1	Supplier choice #1
	2	High End
	3	AR>0.70
	4	Supplier choice #2
B	5	AR>0.70
	6	Supplier choice #2
	7	Supplier choice #1
	8	High End
C	9	Supplier choice #2
	10	AR>0.70
	11	High End
	12	Supplier choice #1
D	13	High End
	14	Supplier choice #1
	15	Supplier choice #2
	16	AR>0.70
E	17	AR>0.70
F	18	High End

Table 1: Test stencil submissions.

The Benchmark test vehicles were printed at the Benchmark Electronic Inc., Nashua, New Hampshire facility on a DEK 265 screen printer utilizing 18 stencils over a duration of three days. The solder paste was Pb-free, SAC305 no clean, Type 4 mesh. For each stencil, a total of six boards with a 10-minute delay between prints were printed by the same operator, using identical and common machine print parameters. The cards were printed in the same order and the underside of stencil was dry wiped after each print. All boards were measured with a Koh Young solder paste inspection (SPI) platform. The 1st and 6th boards were photographed to provide a visual assessment of print repeatability; these images appear in Appendix I. Each stencil was photographed after completion of the dry wipe following the 6th print. These images also appear in Appendix I and provide a relative comparison of paste release. Specifically, stencil apertures with the less amounts of paste indicate better release.

Stencil Assessment Criteria

This study incorporated multiple parameters to assess stencil performance:

- Dimensional accuracy of the aperture opening or size accuracy
- Visual assessment of the print and stencil after the final print to determine the amount of paste remaining in the apertures
- Topography of the aperture walls
- Analysis of SPI data for transfer efficiency and repeatability

Size Accuracy of the Apertures

Miniature components require apertures that are within specification. Apertures that are too small increase the risk of insufficient solder defects due to poor paste release and apertures that are too large increase the risk of bridging and solder balls. Inconsistent apertures on 01005 or 0201 components increase the risk of tombstoning. The stencil providers were asked to include four small coupons outside the print area that could be removed and measured for accuracy and topography assessment. The latter is a destructive test. A stencil coupon is shown in Figure 2.

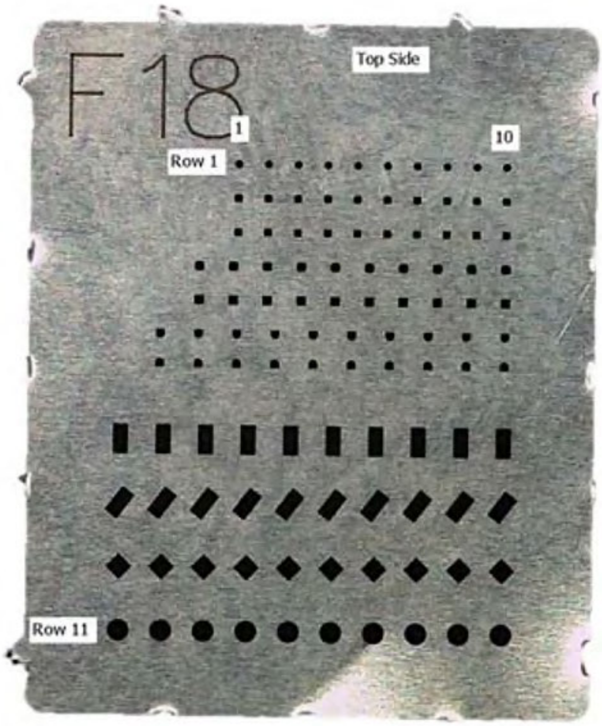


Figure 2: Stencil coupon for F18.

The coupon has 11 rows with various aperture openings in terms of size and shape, representing various components. Each row has 10 identical apertures to provide an indication of repeatability. The first row is a circular aperture with a specified diameter of 7.3 mils and the 2nd row is a square aperture with a specified length and width of 7.3 mils. These two rows are nearest the typical aperture for a 01005 component. The top side and bottom sides of the coupon were measured at Vicor using a Keyence digital microscope. The location of the aperture relative to the fiducial, or location accuracy, was not measured in this study. Each measurement was ranked as target condition (green), acceptable (yellow), or unacceptable (red). A green aperture measured within ±.3 mils to the specification, a yellow was within ±.3 to .5 mils, and red was outside of ±.5 mils. The number of green, yellow and red ratings for each stencil was tabulated and the stencils ranked based on this criteria. The most accurate stencils did not have red ratings, and were ranked based on the highest number of green and fewest number of yellow ratings. The stencils with red ratings were ranked in descending order based on the number of red ratings. If two stencils had the same amount of red ratings, the number of green and yellow ratings was used to delineate the ranking. The summary appears in Table 2.

Visual Assessment of Print and Stencil

The visual assessments of paste deposits after the first and sixth print were performed at two selected fine pitch component pads 0.3 mm microBGA and 0.4 mm microBGA locations. Each stencil was also inspected after the final (sixth) print to have perspective on the amount of paste remaining in the apertures. The photos of uncleaned stencil apertures/windows at the same two component locations were also recorded after last or 6th print. Based solely on visual inspection, the stencils were grouped into the three categories shown in Table 3.

The photos of typical square shape apertures both regular and with radiuses corners are shown in Figure 3.

Stencil	Number of Green Ratings (+/-.3 mils)	Number of Yellow Ratings (+/-.3 to .5 mils)	Number of Red Ratings (+/-.5 to .9 mils	Rank
B6	13	1	0	1
D16	13	1	0	1
C9	11	3	0	2
D14	11	3	0	2
B7	9	5	0	3
C10	9	5	0	3
E17	13	0	1	4
C11	12	1	1	5
C12	12	1	1	5
A2	9	4	1	6
B5	9	4	1	6
B8	9	4	1	6
A3	7	5	2	7
A4	7	5	2	7
D13	10	1	3	8
A1	5	4	5	9
F18	4	4	6	10
D15	6	0	8	11

Table 2: Stencils ranking by aperture size accuracy.

Supplier Ranking	A	B	C	D	E	F
Above Average	A1	B5# B6 B8	C11 *	D16# D14		
Average	A2 A4	B7	C9# C10	D15	E17	
Below Average	A3		C12	D13		F18
* Stencil was noticed stained\rusted after final print and clean (photos in Appendix I)						
# Relatively clean stencil having small residual film after 6th print.						

Table 3: Visual assessment of stencil apertures after 6th print. Representative photos are shown in Appendix I.

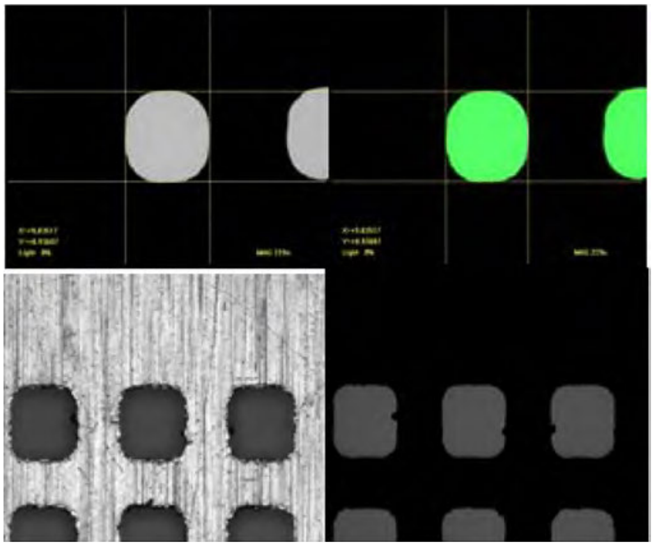


Figure 3: Square apertures with radiused corners.

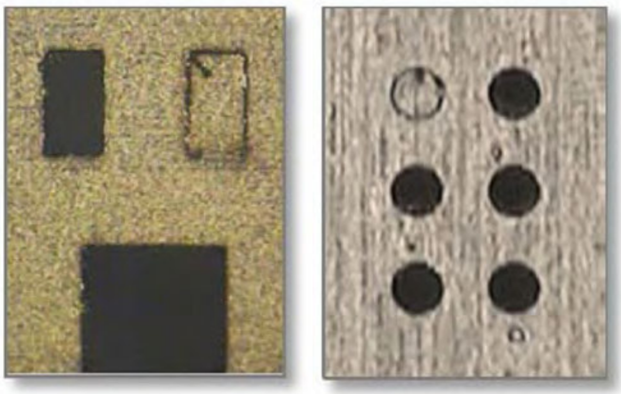


Figure 4: Typical cutting path of a laser beam.

The laser uses more energy to plunge into the middle of the aperture and less energy to provide a clean cut around the perimeter shown in Figure 4 from previous studies[1-3].

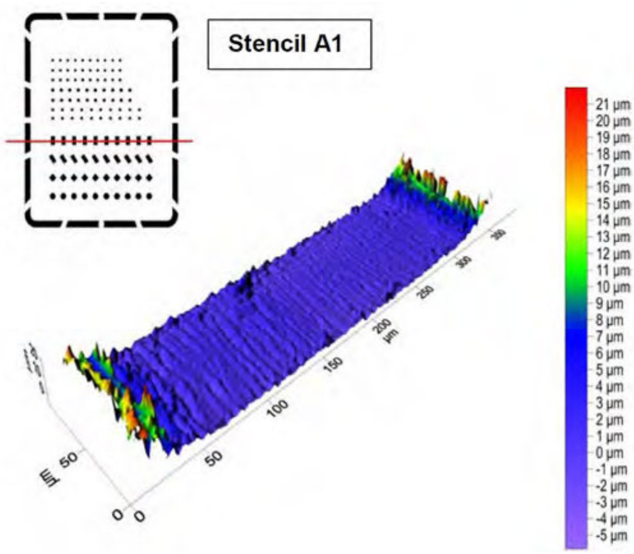


Figure 5a: Aperture topography of coupon A1.

Topography of Aperture Walls

The roughness measurement was performed on the aperture walls of a test coupon for each stencil. The coupons were cut in half where indicated in Figures 5a and 5b, and four apertures were measured for each coupon. The topography comparison of A1 and C11 single aperture show a dramatic difference in roughness.

Additionally, stencil C11 showed significant bottom side slag shown in Figures 6a and b, measuring over 30 microns high. This would raise the stencil off the board surface and likely would lead to excessive paste deposit volume caused by poor gasketing and allowing paste

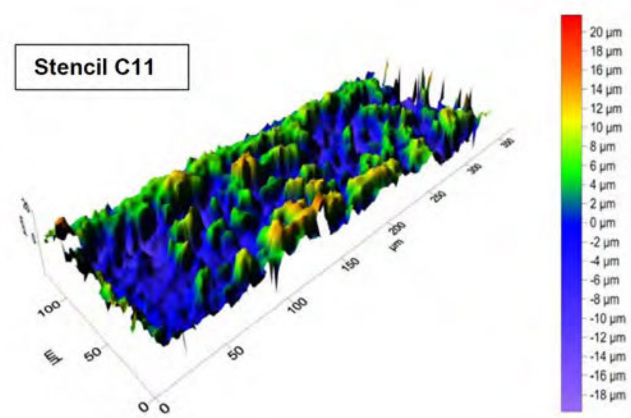


Figure 5b: Aperture topography of coupon C11.

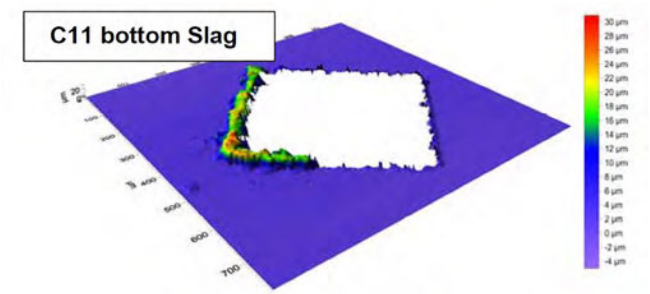


Figure 6a: Bottom side slag on C11.

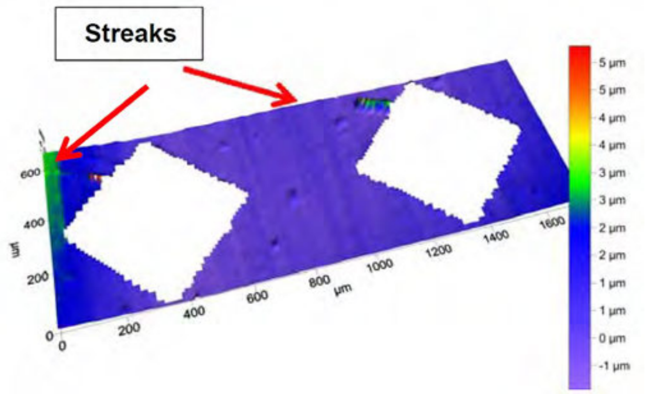


Figure 6b: Bottom side streaks on stencil C11.

Coupon #	Roughness Sa (mic)	Bottom Inspection
C9	0.69	OK
E17	0.73	light bottom slag/streak
B5	0.75	Ok
D14	0.77	light bottom slag/streak
B7	0.79	sanding swirls on bottom
A1	0.8	Ok
B6	0.86	Ok
D15	0.87	Ok
A3	0.9	light bottom side streaks
A4	0.91	Ok
B8	0.91	Ok
D16	0.99	light bottom side streaks
D13	1.08	light bottom side streaks
A2	1.15	Ok
C12	1.35	light bottom side slag
F18	1.71	Ok
C10	3.29	Ok
C11	5.12	a lot of slag

Table 4: Stencil coupon roughness and observations.

to squeeze out. Several other stencils exhibited light slag, referred to as "streaks," which is believed to be related to the relative movement of the laser and table during cutting. The average roughness and bottom slag observation for all are listed in Table 4.

The small streaks of metal measured less than five microns high, and would not be expected to have a major impact on volume.

Analysis of Solder Paste Inspection (SPI) Data

Transfer efficiency (TE) and coefficient of variation (CV%) are excellent indicative metrics in assessment of overall stencil print performance. In the present study these calculations were performed using inline SPI, mainly at miniature component sizes of interest: 0.3 mm BGA, 0.4 mm BGA and 0201H locations on all six cards printed by each stencil. The average print volume and standard deviation calculations of SPI data from six prints on each device type provide statistical indices for the comparison of each stencil's release characteristics, such as paste deposit and print repeatability.

The aperture size at 0.3 mm BGA, 0.4 mm BGA and 0201H locations were measured directly on each stencil. The theoretical volume for each aperture was calculated. The specification defined area ratios for 0.3 mm BGA-square, 0.4 mm BGA-square and 0201s—circular shape features for 4 mil stencil thickness are 0.45, 0.55 and 0.70, respectively. The actual area ratios for these features not calculated in present work are slightly different on many stencils due to measured variation in actually cut aperture sizes. A similar study showed aperture size variation was generally within 2% on any given stencil, but as much as 22% different from stencil to stencil, which can have a considerable impact on the SMT process.[3]

Transfer efficiency (TE) is calculated as the ratio of average deposit volume to the aperture volume to obtain a measure of stencil's print transferability. The coefficient of variation (CV %) is calculated as the ratio of standard deviation of paste volume to average paste volume and it serves as a measure of paste deposit repeatability from print to print. A generally accepted benchmark for acceptable paste transfer is 80% TE, with CV of 10% or less. Both TE and CV were determined and plotted for devices 0.3 mm BGA, 0.4 mm BGA and 0201s in Figures 7a and 7b.

Results and Discussion

For miniaturized components, many different stencil construction technologies were tested. A wide array of results were observed. The data indicates that the best performing stencils for the miniature components appear to be B5 and B6, and D14 and D16. Interestingly, B5 and D16 had no nanocoating on them. They were cut from name brand stainless steel on new, state-of-the-art cutters. B6 and D14 were also cut on new, state-of-the-art cutters and nanocoated with thermally cured fluoropolymer nanocoating.

The electroformed stencil demonstrated the poorest size accuracy of the 18 stencils tested, which is in agreement with three previous studies since 2011.[1-3] Stencils that were laser cut with nickel overplate did not appear to perform as well as laser cut SS without overplate, with or without nanocoating. New investigative technologies that were tested show much promise for delivering quality prints at better price points, thereby representing better values to SMT assemblers.

Continuing Work

One of the top-performing stencils is currently the process of record on Vicor's high-volume production line. Because a large amount of production data already exists for this stencil configuration, a new one will not be ordered to run on the line. Instead, two other top performers and two stencils employing promising new technology will also be run for a full week in high-volume production. The stencil with largest amount of slag and roughness will also be selected for volume runs to determine if these parameters impact end-of-line yields. The metrics for the production runs are, first and foremost, SPI yields. Secondary metrics include TE and Cpk for process control, and end-of-line yields.

Prior to the stencils entering production, they will be validated using SPI to measure two prints each on back and front strokes, and

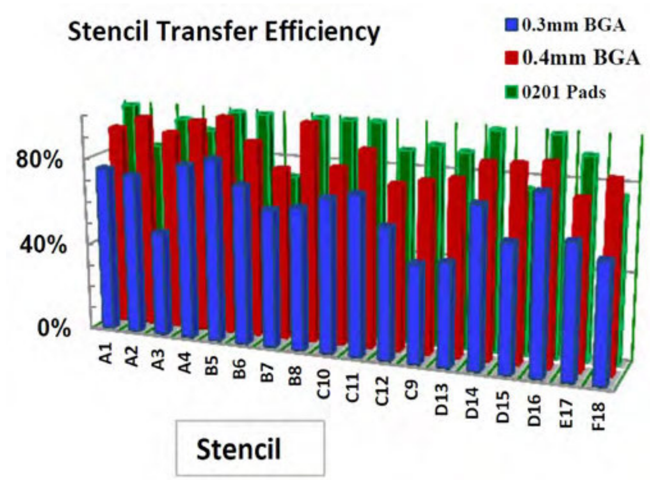


Figure 7a: Stencil transfer efficiency.

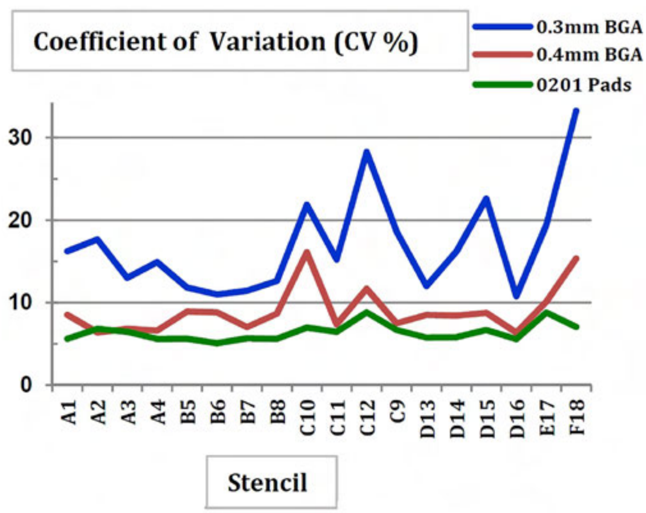


Figure 7b: Assessment of stencil print repeatability.

greater than 90% yield in the first hour of production. If print yields drop below 90% and the stencil is suspected as the root cause, it will be removed from the production line and replaced with the stencil used in the process of record to investigate the suspicion. Results from the longer-term production study will be published at a later date. SMT

Acknowledgements

The authors would like to sincerely acknowledge and thank:

- Nathan Taylor and Rey Molina of Benchmark Electronics, NH for their technical support in execution of print tests and SPI programming

- Stan Camin of Vicor, MA for the measurements of stencil coupons
- Stencil providers for stencils and related technical support

References

1. Whittier, R. and Shea, C., "Evaluation of Stencil Materials, Suppliers and Coatings," Proceedings of SMTA International Conference, September 2011.

2. Whittier, R. and Shea, C., "Fine Tuning the Stencil, Manufacturing Process and Other Stencil Printing Experiments" Proceedings of SMTA International Conference, September 2013.

3. Farrell, R. and Shea, C., "Stencil and Solder Paste Inspection Evaluation for Miniaturized Components," Proceedings of SMTA International Conference, September 2014.

Editor's Note: This article was originally published in the proceedings of SMTA International.

Neeta Agarwal is a senior staff engineer in product assurance at Benchmark Electronics Inc.

Joe Crudele is a principal process engineer in design engineering at Benchmark Electronics Inc.

Chrys Shea is the president of Shea Engineering Services.

Ray Whittier is a Sr. Principal Process engineer (CSMTPE) at BAE systems.

Chris Tibbetts is a senior advanced development manufacturing engineer at Analogic Corporation.

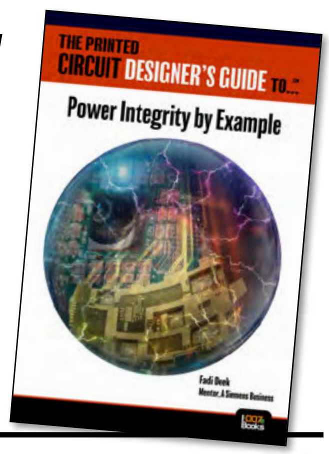
IN MEMORIAM 1960-2017

Robert Farrell was a brilliant engineer, admired and respected by all at Benchmark Electronics, as well as by customers worldwide. He was held in high esteem by academia with whom he interfaced and collaborated frequently. Bob was an active member of SMTA and served as Vice-President and President of SMTA Massachusetts (Boston) chapter. He published and presented his technical studies around the country. Bob was recognized as a true leader in the electronics manufacturing community, and his contributions to the advancement of surface mount technology will be missed by many.

I-Connect007 Launches The Printed Circuit Designer's Guide to...Power Integrity by Example Micro eBook

I-Connect007 has released the latest title in our micro eBook series: *The Printed Designer's Guide to... Power Integrity by Example.*

Authored by Fadi Deek of Mentor, A Siemens Business, *The Printed Circuit Designer's Guide to... Power Integrity by Example* is a follow-up to Deek's previous book on signal integrity, which is also available for download in our library.



This new book on power integrity provides a thorough investigation of power distribution network performance. Deek addresses problematic issues within electronic transmissions, and presents a variety of simulations and analyses in every chapter.

Download this free book today at I-007eBooks/PIE!

UPCOMING Programs

December 6–8, 2017

HKPCA International Printed Circuit and Exhibition & APEX South China Fair

Conference

Shenzhen, China

December 13, 2017

Wisdom Wednesday

Strategies for Reducing Product Warranty and Liability Risk in Manufacturing Services Agreements

IPC Members Only

Webinar

February 24–March 1, 2018

IPC APEX EXPO 2018

Conference & Exhibition

San Diego, CA, USA

March 1–2, 2018

The Pb-Free Electronics Risk Management (PERM) Council Meeting No. 35
held in conjunction with IPCAPEXEXPO

IPC Members Only

Meeting

San Diego, CA, USA

IPC TECHNICAL WEBINAR

December 12, 2017 | Cleaning Before Conformal Coating

Do you conformally coat your electronic assemblies? If so, cleaning prior to conformal coating is a must to ensure the long-term reliability of your products. This webinar provides an overview of potential failure mechanisms resulting from coating over residues. Topics to be discussed are:

- Cleanliness requirements for electronic assemblies before coating
- Overview of standards
- Cleanliness monitoring
- Methods for reliability testing and quality assurance

Join us for this educational opportunity on **12 December at 14:30pm CET**.

Cost:

IPC Members: €55

Non Members: €69

www.ipc.org/cleaning-coating-webinar

For more information, **visit www.IPC.org/events**

Industry 4.0 and the Platform-Based Approach to Testing

How National Instruments is Addressing Today's Engineering Challenges

by Stephen Las Marias
I-CONNECT007

Chandran Nair is the vice president for Asia Pacific at National Instruments, a provider of automated test systems and virtual instrumentation software. In an interview during their recent National Instruments Technical Symposium, Nair explains how the Industry 4.0 vision will change electronics manufacturing. He explains key technology enablers that would drive the evolution of manufacturing processes, and how a platform-based approach can help improve test and measurement.

Las Marias: How will Industry 4.0 take electronics manufacturing processes to the next level?

Chandran Nair: Industry 4.0 is a massively used term, and people have different views for that. But one of the common themes in Industry 4.0 as we move forward is that at the very least, it will help asset management because people will be able to build machines that can communicate with each other in the factory floor. It will help in things like utilization and predictive maintenance, that will in turn reduce downtime, increase efficiency, and reduced wastage.

At the next level, it will be the connectivity to the enterprise that will enable things like data analytics, higher level integration with the IT systems—the convergence of operational technology and information technology—and people will be able to get real-time information on things like productivity. You can even think as far as being able to make customizable products based on customer needs.

Las Marias: What are the key technology enablers for this Industry 4.0 vision?

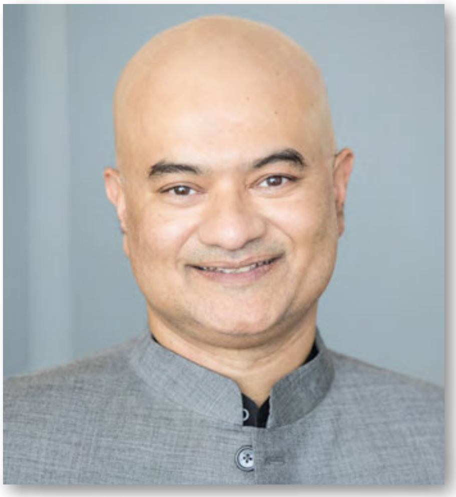
Nair: Each level will be really having smart machines—machines that can communicate with each other and communicate with the enterprise. It is powered by the increase in processing power, in the ability to use heterogenous processing systems, increase in the capability of system level software to be able to talk to multiple processing systems that are requiring data, and then finally, the ability to do analysis and the ability to provide insights based on all these data that are being collected.

Las Marias: Do you see manufacturers now adopting these technologies? If not yet, what remains to be the challenge?

Nair: The places where I see quick adoption are in high-value systems, like aerospace systems. For example, people that make high-precision turbines—they are already a step ahead with regards to integrating their manufacturing to the IT level to do predictive maintenance. That seems to be carrying on as economies of scales take place, and people understand exactly how to use these smart machines, smart devices, the ability to connect to the cloud and to do analytics, and to have artificial intelligence (AI) algorithms that can provide insights. As that happens more and more, I can see the adoption go downstream also.

Las Marias: Industry 4.0 involves automation, connectivity, and data.

Nair: Yes, and connectivity to sensors, not only connectivity among processing units. There is one more thing that I would like to add: it's also the ability to do data reduction at the edge, close to where the sensors are.



Chandran Nair

Las Marias: Where does National Instruments come in to help manufacturers in their Industry 4.0 journey?

Nair: National Instruments come in to the test side, providing the ability to do data management and integration of these data management to the enterprise level. We also come in very heavily in terms of predictive maintenance. We are also helping companies in the verification and test, because all the data collected can then be fed into artificial intelligence boxes, if you like, where there will be analytics that do these AI and give people a better understanding of how to do predictive maintenance, better utilization, and so on. So, where National Instruments comes in is sharing our platform expertise and tapping on the expertise of our customers and partners to be able to help companies get insights from the large amount of data they collect.

Las Marias: Nowadays, equipment manufacturers are developing some sort of AI technology or machine learning into their systems. Why do you think that is so?

Nair: Instead of just basing computer programming on a set of codes that the computer will perform—and it is limited to the instructions the computer is given—what happens is data is fed and models are created based on that data, so that decisions can then be made by the computer based on the wide varieties of data and scenario that are fed in. It's used in things like understanding how autonomous vehicles would react in different conditions. Basically, by getting these large amounts of inputs and creating scenarios around them, the computer learns how to create decisions based on real-time information.

When it comes to the manufacturing side, it could be where people understand, based on inputs, of what kind of test requirements are needed to increase, for example, the yield. On the production side, it could be determining the yield conditions as products are being manufactured, and how one can improve the yield. These could all be use-cases for machine learning and AI. Also, robots used in the manufacturing line will increasingly learn how to work along with humans to be able to continually optimize. Because, really, the future is not just purely robotic lines, but how robots work along with humans in to increase efficiencies—because there are some things that only humans can do.

Las Marias: Many manufacturers have legacy systems in their factory lines. How can they transition to a smart manufacturing model


PCBs WILL
CHANGE THE FUTURE!

while still being able to utilize their existing systems?

Nair: Transition is the right word. It's evolution, not revolution. The reason I say that is you are not going to throw away millions of dollars' worth of equipment that you have already invested in especially if it's working well. So, the existing machines will be made much smarter. How? You can add some sensors, so that these machines can have increased amount of data input during the manufacturing process. That information can then be passed up the stream so that you have intelligent manufacturing, for example, and you could scale up in parts. As you go to new product lines, you may bring in new equipment. The existing product lines can be made smarter based on your needs, through adding features, which can be added through people like National Instruments who are working with multivendor systems.

> "As you go to new product lines, you may bring in new equipment."

Las Marias: What does the future look like for the test and measurement industry?

Nair: For the test and measurement market, of course, the ability to use FPGAs for increasingly complex testing is going to continue. The ability to do machine learning so that test times can be reduced, and tests can be optimized—that will continue; and, the ability to integrate with the cloud so people can do these analytics on the cloud and get them, especially for large companies with multiple manufacturing sites. One of the major things is, as test systems and control systems become more and more complex and increasingly controlled and monitored through software, the need for configuration and remote management of the assets will increase so that manufacturers will be able to do updates from central stations.

Las Marias: What about the cybersecurity?

Nair: That's one of the largest areas of concern and research, and that will continue—security for factory, security for cars, security for anything that is connected to the network, which basically is almost any device all the way from your watch to a Tesla. I think that will be a continuing threat, and then we'll continue to see solutions. I don't think there will be a time when we can say there's no more threat, neither can we say that all these solutions are completely adequate.

Las Marias: What is your outlook for the electronics manufacturing industry in the next year or two?

Nair: Consumerism is still a strong trend. People want new devices, faster, cheaper, better, whether it's better phones, better cars—cars now are also consumer devices with multiple electronics—better computers, better everything. And they want it cheaper and faster. That is driving the market tremendously.

Las Marias: How is National Instruments positioned amid all the evolution happening in the global electronics manufacturing industry?

Nair: We are positioned in a great space. We've been talking about platforms for more than 30 years, and it's been proven time over time that when you do a platform-based approach to test and measurement systems, then you can scale up. With the technology changing so quickly and convergence of multiple technologies, the modular hardware with software centricity, I think we are probably among the best placed vendors to be able to tap into that fast-changing market.

Las Marias: Thank you very much, Chandran.

Nair: Thank you. **SMT**

Recent Highlights from SMT007

1 It's Only Common Sense: Marketing Your Contract Manufacturing Company

We often focus on North America's shrinking PCB fabricator landscape, which is are now down to less than 200 shops. But there are well over 1,000 contract electronics manufacturers in existence, and they come in all shapes and sizes.

2 Joining Forces: SMTA and the SMART Group in Europe

Tanya Martin, executive director at SMTA, and Keith Bryant, global director of sales for YXLON International and chairman of the SMART Group, speak with I-Connect007 Managing Editor Patty Goldman about the latest initiatives within their associations, and discuss some interesting news about an upcoming merger.

3 5 Simple Elements to Guarantee Outsourcing Success

Rather than viewing outsourcing as a way to offload unwanted tasks, OEMs should spend time carefully selecting and then maintaining a properly managed relationship with their chosen EMS partner.

4 Surprising European EMS Market Numbers

If you think market analysis is always correct and predict the future exactly, you are mistaken. The latest annual reports from EMS companies with manufacturing sites in Europe have changed the picture on the European EMS industry quite a bit and caught us by surprise. Yes, you read that correctly: SURPRISE.

5 IMI 9M2017 Revenues Up 29%

EMS firm Integrated Micro-Electronics Inc. (IMI) posted $795.2 million in revenues for the first nine months of 2017, an increase of 29% year-on-year due to positive effects of recent acquisitions and a rising demand in the automotive and industrial segments.

6 NPL and SMART Group to Hold Design, Process & Reliability Seminar

The National Physical Laboratory and SMART Group will hold a seminar November 9, 2017, that will showcase the latest research and results from NPL projects looking at solder joint and contamination failure, coating thickness measurement, solder joint reliability, and high temperature reliability for alternative solders and substrates materials.

7 IPC APEX EXPO 2018 Educational Programs Highlight Tech's Accelerating Speed of Change

Changing technologies that are driving the electronics industry will take center stage throughout the IPC APEX EXPO 2018 technical conference and professional development sessions, which will take place February 24 - March 1 at the San Diego Convention Center in San Diego, California.

8 New Software Toolkit, CFX Messaging Library Expedites Connected Factory Exchange Implementation

IPC is proud to announce that through today's donation and legal transfer of a technology from Aegis Software, the industry will have an open-source engineering toolkit that simplifies and expedites both CFX (connected factory exchange) data message creation, and message communication from machine-to-machine and/or machine-to-broker in a CFX network.

9 The Value and Importance of Training and Certifying Employees

Training, certification and education are central pillars at IPC. John Mitchell, IPC president and CEO, discusses the value of training, certification and education and how IPC's programs can help grow your business.

10 Libra Industries Strengthens Post-Reflow AOI Process

Libra Industries has completed the installation of a new Omron VT-S730 3D Post-Reflow AOI system at its Dallas, Texas facility. The company now can certify its SMT production with full IPC standard compliance using Omron's new 3D-SJI technology.

SMT007 has the latest news and information. Subscribe to our SMT Week newsletter when you register at: my I-Connect007.

Saki
The Future in Focus

Position:
Field Application Engineer

Saki America Inc., headquartered in Fremont, CA, a leader in automated inspection equipment, seeks two full-time Field Application Engineers (FAE), one in the Fremont headquarters and the other for the Eastern and Southern United States.

The FAE will support the VP of Sales and Service for North America in equipment installation, training, maintenance, and other services at field locations. The FAE will provide technical/customer support and maintain positive relationships with existing and future customers.

Strong analytic abilities and problem-solving skills are a must in order to understand customer applications and troubleshoot issues. The FAE will perform demos and presentations for customers and agents as well as assisting in trade show activities. Candidate must have a minimum of a two-year technical degree, experience in AOI, SPI, and X-ray inspection, and strong verbal and written communication skills. The position requires the ability to travel about three weeks per month. Must be a US citizen and be able to lift up to 40 lbs.

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Ventec Seeking U.S. Product Manager for tec-speed

Want to work for a globally successful and growing company and help drive that success? As a U.S.-based member of the product and sales team, your focus will be on Ventec's signal integrity materials, tec-speed, one of the most comprehensive range of products in high-speed/low-loss PCB material technology for high reliability and high-speed computing and storage applications. Combining your strong technical PCB manufacturing and design knowledge with commercial acumen, you will offer North American customers (OEMs, buyers, designers, reliability engineers and the people that liaise directly with the PCB manufacturers) advice and solutions for optimum performance, quality and cost.

Skills and abilities required:

- Technical background in PCB manufacturing/design
- Solid understanding of signal integrity solutions
- Direct sales knowledge and skills
- Excellent oral and written communication skills in English
- Experience in making compelling presentations to small and large audiences
- Proven relationship building skills with partners and virtual teams

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

Please forward your resume to
jpattie@ventec-usa.com and mention
"U.S. Sales Manager—tec-speed"
in the subject line.

www.venteclaminates.com

apply now

ACE ACCURATE CIRCUIT ENGINEERING

PCB Assembly Supervisor—full time
Accurate Circuit Engineering—Santa Ana, CA

Position Summary: Responsible for all assembly processes to ensure continued growth as directed by management.

Essential Job Functions:
- Create, implement, and supervise in-house manufacturing facility
- Recruit, hire, train, and supervise assembly floor personnel
- Extensive hands on experience with all aspects of PCB assembly
- Understanding of IPC-A-610 standards
- Research and acquire additional assembly resources
- Gather data on product shortages, lead times, price changes, etc.
- Coordinate the assembly activities with sales to ensure 100% on-time delivery
- Create, implement, and supervise daily quality processes to ensure 100% accuracy
- Document, monitor and review progress of the business unit
- Respond to internal and external customers in a timely manner
- Coordinate walk-through, site audits, etc.

Qualifications:
- Minimum 3 years as operations supervisor of electronics assembly house
- 5+ years' experience in the electronics industry
- Previous experience as a quality or operations supervisor preferred
- Ability to solve practical problems using pre-established guidelines
- Strong facility in Microsoft Office applications
- Excellent verbal and written communication skills
- Ability to work with people of diverse backgrounds
- Highly organized/excellent time management skills
- Ability to perform at the highest level in a fast-paced environment
- Valid California driver's license.

apply now

ACE ACCURATE CIRCUIT ENGINEERING

PCB Process Planner

Accurate Circuit Engineering (ACE) is an ISO 9001:2000 certified manufacturer of high-quality PCB prototypes and low-volume production for companies who demand the highest quality in the shortest time possible. ACE is seeking a skilled individual to join our team as a PCB process planner.

Responsibilities will include:
- Planning job travelers based on job release, customer purchasing order, drawings and data files and file upon completion
- Contacting customer for any discrepancies found in data during planning and CAM stage
- Consulting with director of engineering regarding technical difficulties raised by particular jobs
- Informing production manager of special material requirements and quick-turn scheduling
- Generating job material requirement slip and verify with shear clerk materials availability
- Maintaining and updating customer revisions of specifications, drawings, etc.
- Acting as point of contact for customer technical inquiries

Candidate should have knowledge of PCB specifications and fabrication techniques. They should also possess good communication and interpersonal skills for interfacing with customers. Math and technical skills are a must as well as the ability to use office equipment including computers, printers, scanners, etc.

This position requires 3 years of experience in PCB planning and a high school level or higher education.

apply now

INDIUM

Technical Content Specialist

Indium Corporation is seeking a technical content specialist to guide the development of data-rich, high-level content for the company's semiconductor and advanced assembly materials (SAAM) sales and technical literature. The technical content specialist will work with multiple departments to ensure that all externally-facing technical and sales collateral and internal training materials are consistent in format and of superior quality.

The technical content specialist will:

- Assist in the development of key content and ensure consistency of message and format across platforms
- Develop a technically-detailed understanding of Indium Corporation materials and offerings to the SAAM industry
- Curate a library of technical conference papers and associated materials, including content related to Indium Corporation materials and their performance
- Assist in the development of, and ensure consistency for SAAM promotional materials, such as product datasheets (PDS), images, brochures, whitepapers and presentations (technical and sales)
- Attend at least one technical conference and its paper session per year

Requirements:

- Technical undergraduate degree (BS in Chemistry/Physics/Metallurgy/Materials Science or Engineering discipline)
- 5 years of work experience in semiconductor assembly or advanced electronics assembly
- Excellent written and spoken English language skills; fluency in Chinese desirable
- Proven ability to work independently with verbal or written instructions

apply now

Altium

Application Engineer

The application engineer is the first contact for our customers who have technical questions or issues with our product. We value our customers and wish to provide them with highest quality of technical support.

Key Responsibilities:

- Support customer base through a variety of mediums
- Log, troubleshoot, and provide overall escalation management and technical solutions
- Create various types of topic based content, such as online help, online user guides, video tutorials, knowledge base articles, quick start guides and more
- Distill complex technical information into actionable knowledge that users can understand and apply
- Continually develop and maintain product knowledge

Requirements:

- Understanding of EDA electronic design software, schematic capture and PCB layout software
- Bachelor's degree in electronics engineering or equivalent experience
- Sales engineering and/or support engineering experience
- Circuit simulation and/or signal integrity experience
- Understanding of ECAD/ MCAD market segments
- Understanding of micro controllers, SoC architecture and embedded systems market
- Database experience preferred (i.e., MySQL, PostgreSQL, Microsoft Access, SQL, Server, FileMaker, Oracle, Sybase, dBASE, Clipper, FoxPro) etc.
- Experience with PLM/PDM/MRP/ERP software (Program Lifecycle Management) preferred
- Salesforce experience a plus

Salary based upon experience. Comprehensive benefits package and 401k plan. Openings in USA, UK, and Germany.

For more information, contact Altium.

apply now

BLACKFOX
Premier Training & Certification

IPC Master Instructor

This position is responsible for IPC and skill-based instruction and certification at the training center as well as training events as assigned by company's sales/operations VP. This position may be part-time, full-time, and/or an independent contractor, depending upon the demand and the individual's situation. Must have the ability to work with little or no supervision and make appropriate and professional decisions. Candidate must have the ability to collaborate with the client managers to continually enhance the training program. Position is responsible for validating the program value and its overall success. Candidate will be trained/certified and recognized by IPC as a Master Instructor. Position requires the input and management of the training records. Will require some travel to client's facilities and other training centers.

For more information, click below.

apply now

Prototron Circuits

Experienced PCB Sales Professional

With more than 30 years of experience, Prototron Circuits is an industry leader in the fabrication of high-technology, quick-turn printed circuits boards. Prototron of Redmond, Washington, and Tucson, Arizona are looking for an experienced sales professional to handle their upper Midwest Region. This is a direct position replacing the current salesperson who is retiring after spending ten years with the company establishing this territory.

The right person will be responsible for all sales efforts in this territory including prospecting, lead generation, acquiring new customers, retention, and growth of current customers.

This is an excellent opportunity for the right candidate. Very competitive compensation and benefits package available.

For more information, please contact Russ Adams at 425-823-7000, or email your resume.

apply now

Process Engineer (Redmond, Washington)

With more than 30 years of experience, Prototron Circuits is an industry leader in the fabrication of high-technology, quick-turn printed circuits boards. We are looking for an experienced PCB process engineer to join the team in our Redmond, Washington facility. Our current customer base is made up of forward-thinking companies that are making products that will change the world, and we need the right person to help us make a difference and bring these products to life. If you are passionate about technology and the future and believe you have the skills to fulfill this position, please contact Kirk Williams at 425-823-7000 or email your resume.

apply now

Events

IPC Calendar of Events, click here	SMTA Calendar of Events, click here	iNEMI Calendar of Events, click here	SMT007 Calendar of Events, click here

ICT Evening Seminar
December 5, 2017
Harrogate, UK

HKPCA/IPC International Printed Circuit & South China Fair
December 6–8, 2017
Shenzhen, China

47th NEPCON JAPAN
January 17–19, 2018
Tokyo Big Sight, Japan

DesignCon 2018
January 30–February 1, 2018
Santa Clara, California, USA

EIPC 2018 Winter Conference
February 1–2, 2018
Lyon, France

IPC APEX EXPO 2018 Conference and Exhibition
February 27–March 1, 2018
San Diego, California, USA

China International PCB & Assembly Show (CPCA Show 2018)
March 20–22, 2018
Shanghai, China

MicroTech 2018
April 9–10, 2018
Egham, UK

PCB EXPO Thailand
May 10–12, 2018
Bangkok, Thailand

Medical Electronics Symposium 2018
May 16–18, 2018
Dallas, Texas, USA

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

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

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

EDITORIAL:
MANAGING EDITOR: **STEPHEN LAS MARIAS**
+63 906 479 5392; stephen@iconnect007.com

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **© MIKALAI BACHKOU, ADOBE STOCK**

SMT® (Surface Mount Technology) is published by BR Publishing, Inc., 1908, Rohnert Park, CA 94927

December 2017, Volume 32, Number 12 • SMT© (Surface Mount Technology©) is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *SMT Magazine:*

JANUARY:
New Equipment and Technologies
A look into the latest developments and technologies impacting your processes and systems.

FEBRUARY:
Who's Your Customer?
How well do you know your customers, and how do you ensure the best possible service?