I-Connect007
GOOD FOR THE INDUSTRY

ENGINEERING SOLUTIONS FOR PCB MANUFACTURING

SMT MAGAZINE

JUNE 2016

SPECIAL SECTION: IPC'S IMPACT WASHINGTON, D.C. 2016

thepcblist.com
thepcblist
THE BEST WAY TO FIND A PCB FABRICATOR, ANYWHERE.

2016 IWLPC

13th International Wafer-Level Packaging Conference

October 18-20, 2016 San Jose, California

SMTA Surface Mount Technology Association

Chip Scale Review — The Future of Semiconductor Packaging

Event Schedule

October 18-19, 2016

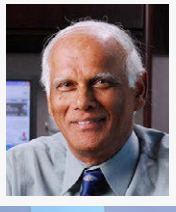
- Exhibition
- Panel Discussions
- Technical Presentations:
  - 3D
  - MEMS
  - WLP
  - Manufacturing Challenges

October 20, 2016

Four Half-Day Workshops
To Be Announced.

Keynote Presentations

Advanced Technology Platforms for Next Generation of Smart Systems
Klaus-Dieter Lang, Ph.D.
Fraunhofer IZM



The Promise and Future of Wafer Fan-out Technology
Rao R. Tummala, Ph.D.
Georgia Institute of Technology

Exhibits & Sponsorships

Reach, Generate Exposure, Share New Products & Enhance Relationships

*Early Bird Rates	Member	Non-Member
One Booth (8′ D x 10′ W)	$1,300	$1,500
One Table (6′ Table)	$900	$1,100

Sponsorships Available

- Reception
- Keynote
- Refreshment
- Lunch
- Charging Station
- Keynote
- Notebook & Pen
- Wifi

2016 Sponsors

Platinum

SUSS MicroTec

Deca Technologies

Gold

nepes

PacTech

EVG

Silver

Lam Research

Media Sponsors

MEPTEC

3DInCites.com

ROGER GRACE ASSOCIATES MARKETING COUNSEL

MEMS JOURNAL

Exhibit & Sponsorships
Kim Newman
knewman@chipscalereview.com

www.IWLPC.com

Conference Questions
Jenny Ng
jenny@smta.org

Strategically Located!

- QUICK
- RELIABLE
- EFFICIENT

PROTOTYPE

24 Hour Turns for Printed Circuit Boards with three Strategic locations in the USA

QUICK
RELIABLE
OFFSHORE PRICE
PRODUCTION

Over the past 15 years, 325,000 prototypes have been successfully delivered from overseas to over 7,500 customers

That's The Imagineering Edge!

imagineering inc

847-806-0003 www.PCBnet.com sales@pcbnet.com

ITAR-SAM Registered ● ISO 9001:2008 ● UL Approved
● Certified Woman Business Enterprise (WBE) ● Certified Woman-Owned Small Business(WOSB)
● Certification# RWOSB14859 ● RWBE14858 (13 C.F.R Part 127)

Solder Paste Printing: Challenges and Solutions

This month, *SMT Magazine* discusses the challenges, best practices, and critical factors to consider in solder paste printing amid tighter tolerances and smaller pitches, lines and spaces.

The SMT Source™

Latest News About Manncorp SMT Assembly Equipment

June 2016

STENCIL PRINTERS

FOR ANY APPLICATION OR BUDGET

FULLY AUTOMATIC INLINE

3 Models from

$59,995–$79,995

LED · QFP · 01005 · µ

AUTO-ALIGN/AUTO-PRINT

1 Model

$34,995

LED · QFP · 01005 · µ

CAMERA-ASSIST ALIGN/ AUTO-PRINT

3 Models from

$18,995–$22,995

QFP · 01005 · µ

FREE SHIPPING* Ordered by 6/30 Many in stock

MANUAL ALIGN & PRINT

4 Models from

$1,495–$6,995

LED · QFP

Immediate Contact:

Chris - East: 215.869.8374

Ed - West: 215.808.6266

*Continental U.S. Only

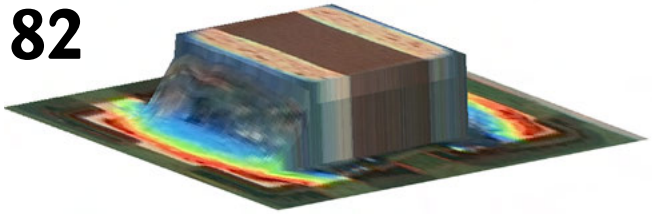
Manncorp™ 50 YEARS

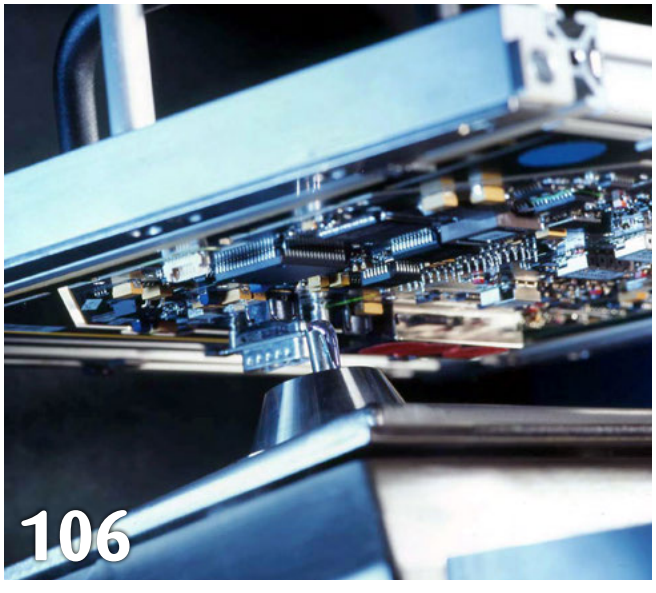
www.manncorp.com

Through June, 2016

Huntingdon Valley, PA • San Diego, CA • Mexico • Brazil • Shenzhen

© 2016 by Manncorp

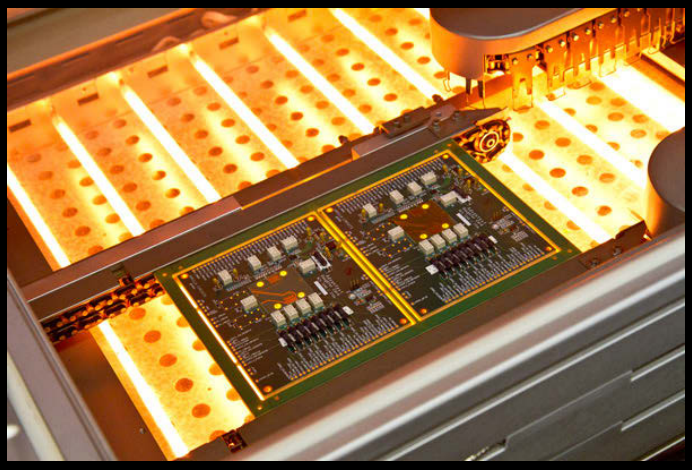




More Content:

Silicon Valley's LEADING Full Turnkey PCB Manufacturer

FABRICATE & ASSEMBLE – ALL IN HOUSE



FABRICATION

- Rigid PCBs 2-60 Layers
- Simple to Industry-leading Technology
- Quick Turn Prototypes and Production
- Proprietary Wrap Plating Process for Unsurpassed Reliability

ASSEMBLY

- Latest FUJI Equipment
- 100% SPI Standard
- 100% AOI Standard
- 100% 2D/3D X-ray Standard

GORILLA circuits

To Request a Quote through our website and enjoy NO tooling, programming, or stencil costs, Click Here!

www.gorillacircuits.com

ISO 9001:2008 – MIL PRF 55110/31032 - ITAR

Solder Paste Exploration

by Stephen Las Marias
I-CONNECT007

The trend towards miniaturization and increasing complexity of board assemblies as more functionality is being packed in smaller and smaller electronics devices no doubt continues to be among the critical issues in the electronics assembly industry today. Not to mention the ever-shrinking component sizes that are further complicating the manufacturing challenges.

To improve the electronics manufacturing process considering the challenges above, we have to drill down to the specific issues that have the greatest impact on the assembly line. And what better way to know these issues than to ask our readers?

So for this month's issue of *SMT Magazine,* we did a survey to find out the biggest factors affecting the electronics assembly process as the industry moves to tighter and tighter tolerances (finer lines and spaces). In our survey, we found out that while the most common line and space widths range from 2 mils to 5 mils, some respondents said they are doing boards with line and space widths down to 1 mil.

I remember when I was in the university, our electronics shop work included making basic and regulated power supplies, crude FM stereos, and digital clocks. Even then, our class was already having challenges in trying to fit the components in smaller boards to save on cost of copper clads (in case we confused our designs—wherein we used graphite transfer papers—and etched the copper out already) and the overall packaging of the project; and all the while making sure that the circuit would work. I can still imagine the design I did for my digital clock, and the meticulous way I laid out the lines for the circuit. We don't have computer programs to help us with those circuit designs then. But I digress.

Back to our survey: Given such tighter tolerances, respondents say PCBA testing and inspection, and soldering—in particular, solder paste printing—are their greatest challenge when it comes to electronics assembly.

This led me to what I would call my "solder paste printing exploration." From the manufacturers of solder pastes, to equipment makers, all the way to the solder paste inspection guys—and most importantly the electronics assemblers themselves who are using these products—I talked to the "supply chain," so to speak, all the way from Shanghai (during the recent NEPCON China 2016 exhibition in April) to our science and technology parks here in the Philippines, to find out what's going on in the solder paste printing process, what factors impact the process, the challenges, and the best practices to consider to be able to address those challenges and improve efficiencies and yield in the SMT line.

The users I talked to are Philippine-based EMS firms Integrated Micro-Electronics Inc. (IMI) and EMS Components Assembly Inc. (EMSCAI). At IMI, I spoke with Joemar Apolinario, Aurelio Bantigue and Rodney Bebe to get their insights on the solder paste printing process given the tighter tolerances and finer pitches in line with the continuing miniaturization trend in the electronics manufacturing industry. We discussed the process challenges, the impact of solder pastes in the printing process, and the criteria for solder paste selection and qualification. We also talked about design and process strategies to get the best solder paste printing results.

At EMSCAI, Knoll Evangelista said that more than 50% of defects in assembly of printed circuit boards are attributed to the solder paste printing process. Given that, he explained why solder joint reliability is the biggest issue when it comes to solder paste printing, and the key factors to consider to ensure the quality of solder joints.

Given the users' challenges and issues, I interviewed the solder paste suppliers to get their insights on such challenges and find out how they are helping their customers address their solder paste printing issues.

Mitch Holtzer, global director of customer technical support at Alpha Assembly Solutions and one of our expert columnists here at *SMT Magazine,* talks about the impact of solder pastes on the printing process and the best practices to consider such as good board support, a reasonably well-controlled environment, high-quality stencils, and using a wipe frequency suitable for the process set up.

Meanwhile, Watson Tseng, general manager of Shenmao America Inc., talks about the variables involved when it comes to selecting the appropriate solder paste for the printing process.

From the equipment side, I interviewed Adam Sim of Speedline ITW EAE at the recent NEPCON China show in Shanghai to discuss issues, including the challenges in solder stencil printing, how printing cycle times may be improved, and the key factors to consider when selecting a solder paste printing solution.

I also sat down with Eric Gu of Nordson Asymtek China to talk about the challenges and innovations in solder paste dispensing, and factors to consider when using solder paste dispensers.

At NEPCON China, I also spoke with Thomas Bredin of Mycronic about solder paste printing, and innovations being done in jet printers to help them get up to speed with challenges such as tighter tolerances and finer lines and pitches in board assemblies.

Finally, to know more about the solder paste inspection (SPI) side, I spoke with Vi Technology's Jean-Marc Peallat and Chong Choon Hee about the latest innovations happening in the SPI space to help customers address inspection challenges amid tighter tolerances and finer lines and pitches.

In their technical article, "3D Solder Joint Reconstruction on SMD based on 2D Images," Pedro M. A. Vitoriano and Tito Amaral of Visteon tackle the issue of solder joint inspection in surface-mount devices and present an algorithm for 3D solder joint reconstruction to help improve the inspection process.

Of course, *SMT Magazine* would not be complete without our expert columnists providing their insights on current manufacturing issues and technologies in the SMT industry. For his column this month, Alpha's Mitch Holtzer discusses which method—pad defined or mask defined—is optimal in designing electronic circuits.

Tom Borkes, of The Jefferson Project, discusses production engineering students as customers, and how the education system can really help by providing real-world experience to complement their academic achievement.

Bob Wettermann of BEST Inc. addresses the debate around the issue of baking out moisture in a PCB during rework, and the need for the board, as well as neighboring components, to be taken into account in terms of moisture protection during the reflow cycle.

And last but not least, Robert Voigt of DDM Novastar continues on his topic of choosing a selective soldering system, covering the common types of soldering technologies available as well as nitrogen inerting systems.

I hope you enjoy this month's issue of *SMT Magazine*. For comments or suggestions, or if you want to become a contributor to our magazine, feel free to send me a note.

Next month, we'll focus on one of the challenges reflected in our survey above: PCBA testing and inspection. Stay tuned! **SMT**

Stephen Las Marias is managing editor of *SMT Magazine.* He has been a technology editor for more than 12 years covering electronics, components, and industrial automation systems.

The Solder Paste Factor in Printing

by Stephen Las Marias
I-CONNECT007

Mitch Holtzer, Alpha Assembly Solutions' global director of customer technical support, and expert columnist at *SMT Magazine,* talks about the impact of solder pastes on the printing process, and the variables involved when selecting the type of solder to use for specific applications.

Stephen Las Marias: ***What are the biggest challenges that your customers face when it comes to solder paste printing?***

Mitch Holtzer: One issue that comes up frequently is managing the paste bead height. Too large of a diameter roll can cause paste to get hung up in the squeegee holding mechanism, and customers mistake this for paste rheology issues. Depending on the paste delivery process, some customers are able to manage this issue, but many struggle.

Las Marias: ***From your perspective, how do the tighter tolerances and even narrower PCB lines and spacing impact the solder paste printing process?***

Holtzer: Modern solder paste formulations and smaller particle size powder has been able to keep up with features like 0.3 mm pitch BGAs and 01005 components down to an area ratio of 0.5. Reflowing large and tiny deposits of paste can be a bigger issue than printing.

Las Marias: ***What about solder pastes? What are the variables involved when it comes to selecting the appropriate solder paste for the printing process?***

Holtzer: The combination of flux and powder are key to a solder paste recommendation. Sometimes customers ask for type 5 and type 6 powder when they don't really need the fine feature print capability afforded by these PSDs.

INTEGRATED.

EXPERIENCED.

CERTIFIED.

EQUIPPED.

GET THE WHITE PAPER:
Reliability Issues with RF Circuits Influenced by Design and Manufacturing Processes

DOWNLOAD

Electronics Contract Manufacturing & Engineering Services

INTEGRATED

- Design
- Layout
- Engineering
- NPI
- Supply Chain
- Assembly
- Testing

EXPERIENCED

- Aerospace
- Military C4ISR
- Medical
- Cyber
- RF
- Space
- Unmanned Systems

CERTIFIED

- ITAR
- IPC Class 3 Trusted Source QML
- IPC J-STD-001 w / Space
- AS9100
- ISO-9001
- ISO 13485

EQUIPPED

- Established 1998
- Complex SMT Assembly
- Five (5) Processing Lines
- X-Ray, AOI, and SPI
- Box Build and Wire Harness
- Design and Test Engineering
- Purpose-Built CM Facility

ZENTECH

COLONIAL ASSEMBLY & DESIGN
a wholly-owned subsidiary of Zentech Manufacturing, Inc.

www.zentech.com • 443-348-4500 • sales@zentech.com

Represent Zentech in Key Markets

Above industry average commission structure and existing AVL status at most military primes.
Contact John Vaughan at vaughanj@zentech.com

© 2016 Zentech Manufacturing. All rights reserved.

***Las Marias:** What other industry trends are affecting the solder paste printing process?*


Mitch Holtzer

Holtzer: The desire to eliminate wave soldering and less expensive components and laminate material has forced many end users to use lower melting lead-free solder alloys, using pin-in-paste processing.

***Las Marias:** What factors do users need to consider when it comes to solder paste printing?*

Holtzer: Best practices would include good board support, a reasonably well-controlled environment (especially temperature), high-quality stencils, and using a wipe frequency suitable for the process set up.

***Las Marias:** What about equipment?*

Holtzer: Jetting is good for high-mix, low-volume applications. It is slower, but changeover is much quicker, and you don't need a stencil. Screen printing is much faster in terms of assemblies per hour, making it well-suited for longer production runs.

***Las Marias:** What do you think is the future for solder paste printing?*

Holtzer: The trend from type 3 pastes to type 4 will continue as components and spacing continue to decline.

***Las Marias:** Is there anything we haven't talked about when it comes to the solder paste printing process that you think we should be talking about?*

Holtzer: Although printing is a very important part of the SMT process, most common defects occur during reflow, after all of the cost of components and time has been spent on the boards. Reflow associated costs are much more expensive.

***Las Marias:** Thank you very much, Mitch.*

Holtzer: Thank you. **SMT**

New Glue Instantly Hardens with Electric Current

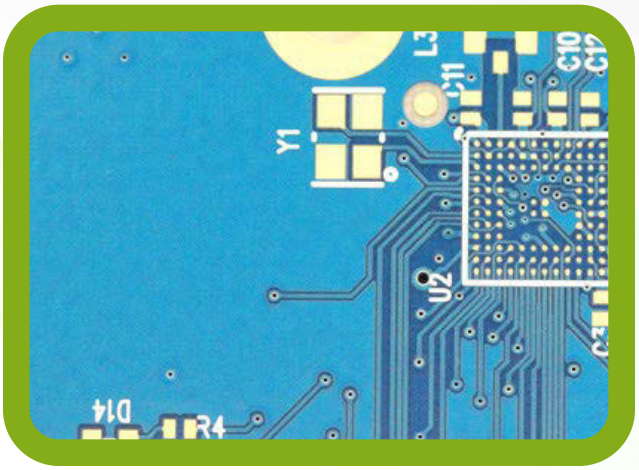
Researchers at Singapore's Nanyang Technological University have developed an adhesive that they believe may be a game-changer in manufacturing fields as diverse as biological implants and automobiles. The new adhesive is a liquid gel that "cures" to form a polymer bond when a voltage of less than 2V is passed through it. The glue stops curing as soon as the current is turned off. Users can fine-tune the bond's strength and flexibility by varying the current's voltage and duration.

The bonding agent is a light, low-viscosity flowing liquid that allows users to coat and exactly position the materials to be joined. Applying voltage to the gel cures it to a strong bond with high elasticity and shear strength.

Currently available quick-curing adhesives used in industry are activated by light, heat or chemical catalysts, each of which limits uses to particular materials and appropriate environments. Such quick-curing adhesives are used widely in the manufacture of medical devices, automobiles and other consumer goods, where they are favored over more labor-intensive, heavier mechanical fasteners such as rivets, screws or bolts, which weaken the materials to be fixed. However, there has been little innovation in the field for decades.

Potential uses for electro-cured adhesives include biological devices for which photo- or thermo-setting glues are problematic, such as bioelectronics or polymer electronics designed for attachment to living tissue. The new adhesive could also make automotive assembly lines more efficient, as photo- and thermo-setting glues require costly, high-maintenance hardware.

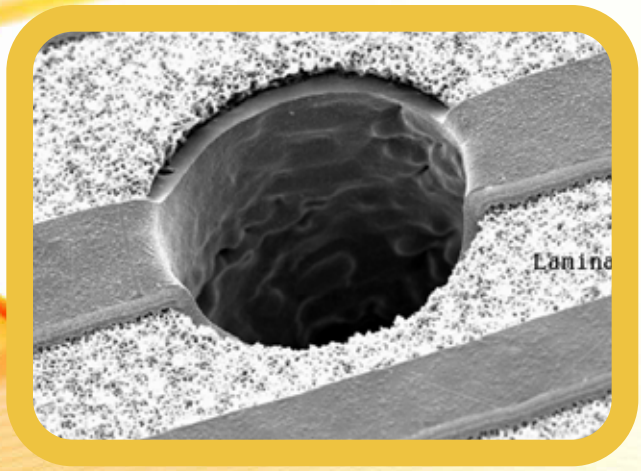
We deliver Advanced PCB Solutions!



Fine line PCB, 2 mil line /space

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

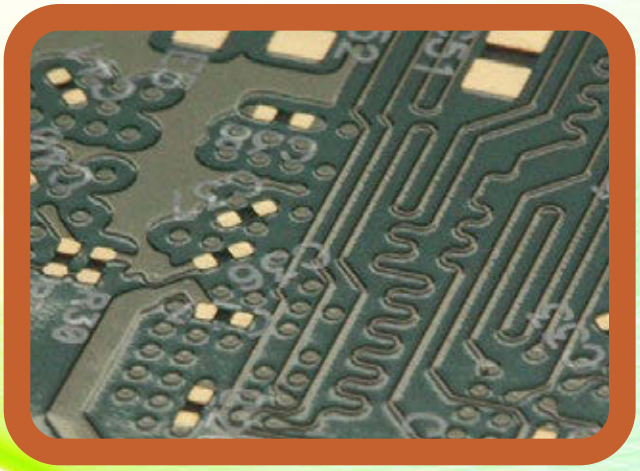
CLICK TO LEARN MORE



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



Impedance Tolerance <5%!

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

CLICK TO LEARN MORE

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

CANDOR INDUSTRIES, INC.

ADVANCED PCB SOLUTIONS

www.candorind.com | sales@candorind.com | (416) 736-6306

Supply Lines Highlights

Selective Soldering: Design, Process Challenges and Practical Solutions
A SMART Group workshop at the Bromsgrove, UK, premises of equipment and process materials distributor APP Electronics, set out to provide the answers, with a program combining technical presentations, live demonstrations and hands-on sessions, introduced and moderated by Nigel Burtt, senior electronics manufacturing engineer at Renishaw and SMART Technical Committee Chairman.

Supplier Spotlight: Transition Automation
In an interview with I-Connect007, Alden Lewis, vice president of sales at Transition Automation, discusses how they are tackling solder paste printing problems with squeegee technologies.

RTW IPC APEX EXPO: Rehm Discusses Advantages of Moving Heat Technology
Rehm Thermal Systems general manager Paul Handler speaks with I-Connect007 guest editor Steve Williams about the advantages of the moving heat chamber compared to traditional technology in the market today, and why their systems are all Industry 4.0-ready.

BTU Receives Its 25th Industry Award
BTU International Inc. received a 2016 SMT China Vision Award in the category of Software – Process Control for the new Recipe Pro.

MIRTEC Reports Record 56% Growth in Q1 2016
MIRTEC has reported a 56% sales revenue growth for its North American Sales and Service Division for the first quarter of 2016, mainly driven by the overwhelmingly successful launch of their MV-6 OMNI 3D AOI Platform.

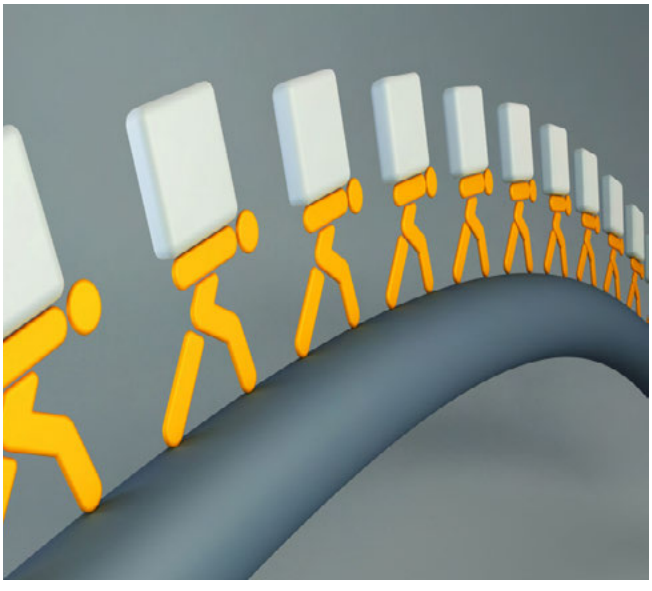
KYZEN Receives SMT China Vision Award for its AQUANOX A8820 Advanced Aqueous Stencil Cleaner
KYZEN has been awarded the 2016 SMT China Vision Award in the category of Cleaning Materials for its AQUANOX A8820 Advanced Aqueous Stencil Cleaner. The award was presented to the company during the recent NEPCON China event in Shanghai.

Mentor Graphics' Valor Production Plan Tool Achieves SAP Certification
Mentor Graphics' Valor Production Plan software has been certified by SAP as powered by the SAP NetWeaver technology platform.

Nordson MARCH's ModVIA Plasma System Easily Doubles Capacity to Accommodate Production Growth
Nordson MARCH introduces its ModVIA plasma system, a fully integrated, flexible system that doubles its capacity from four to eight cells (8-16 panels) to easily accommodate manufacturing production growth.

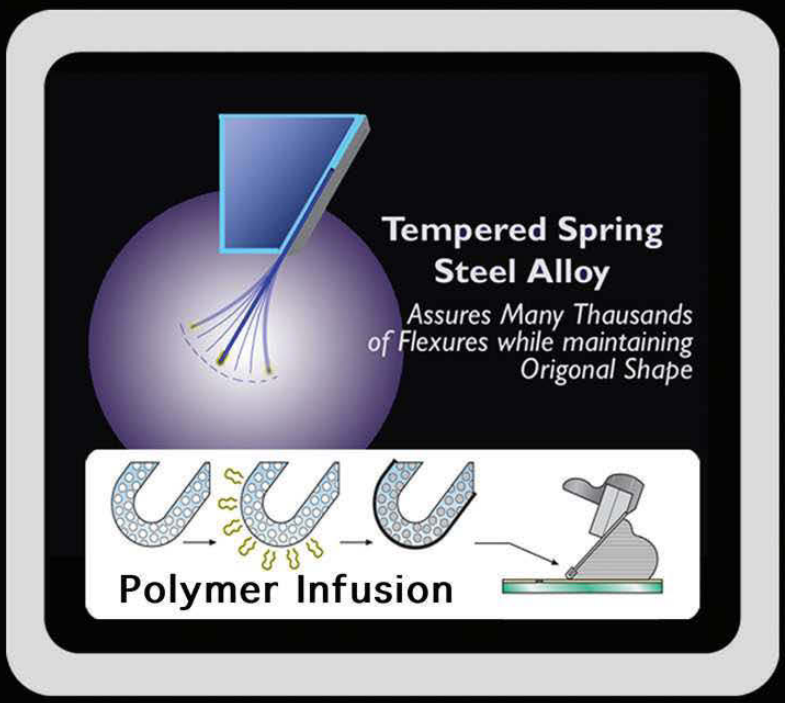
Electrolube Strengthens US Operations
Electrolube has recently assigned Bruce Ward to take on new responsibility for the East Coast of North America due to a recent increase in demand in the United States.

KIC Brings in New COO
KIC has hired Isidre Rosello to plan and direct KIC's operations, new product and services development, company objectives, initiatives, and drive the company's financial and operational goals.



Permalex®

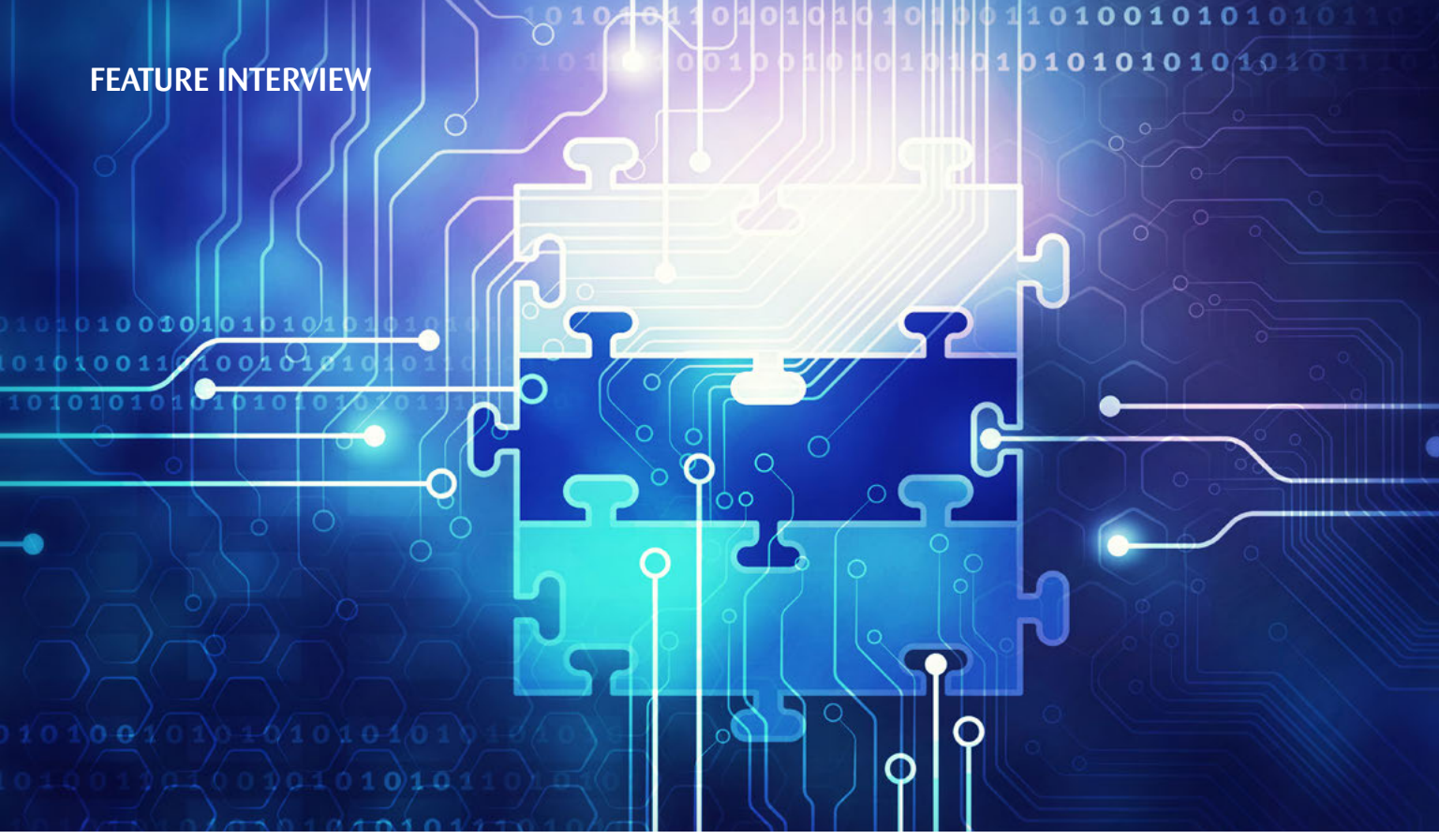
POLYMER INFUSED METAL SQUEEGEES



Technology for Improving SMT
Print Quality and Production Yield

TRANSITION AUTOMATION

www.transitionautomation.com



SOLDER PASTE PRINTING: Challenges and Best Practices

by Stephen Las Marias
I-CONNECT007

In an interview with *SMT Magazine,* Watson Tseng, General Manager of Shenmao America Inc., talks about the challenges in solder paste printing, key parameters and best practices to consider, and variables involved when it comes to selecting the appropriate solder paste for the printing process.

Stephen Las Marias: ***What are the biggest challenges that your customers face when it comes to solder paste printing?***

Watson Tseng: Many of our customers are facing greater challenges assembling smaller and smaller components onto their PCBs and flex circuits. Due to the decreased size, the stencil aperture has to be much smaller than before, for example 0.25 mm or less. It increases the challenges of solder paste printing—the area ratio (AR) becomes smaller and the transfer rate goes down. Getting a good deposition of solder paste becomes more and more difficult. Some customers increase the printing speed to get higher output rate. Some are printing at 120 or 150 mm/s or even faster. A good solder paste designed for high-speed printing is necessary for this application. Otherwise, there will be a lot of shorts or slump failures found after stencil printing.

Las Marias: ***From your perspective, how do the tighter tolerances and even narrower PCB lines and spacing impact the solder paste printing process?***

Tseng: Yes, they do. Especially when you have both large and small components on your board. It's a huge challenge to do stencil printing for this kind of build. A good stencil design, a good solder paste, a precise stencil printer is required to achieve the best result.

Las Marias: ***What about the solder pastes? What are the variables involved when it comes to selecting the appropriate solder paste for the printing process?***

BLACKFOX
Premier Training & Certification

Do you require IPC Training?

Blackfox is the Answer.

Blackfox is one of only a few that offer training in all 6 IPC certification programs.

We also provide training for instructors, operators, lead free, high temp solder and more!

CLICK FOR COURSE SCHEDULE

For more information contact us at 888.837.9959 or sharonm@blackfox.com

www.blackfox.com

Tseng: Solder paste plays an important role in stencil printing. We need a solder paste with proper physical properties to yield high and uniform transfer rate, good deposition without slump or shorts, and long stencil life even at high speed printing.

Selection of solder powder is also very important. Changing from larger powder size to smaller one—for example, from type 3 to type 4.5 or even type 5—helps to get good printing performance. But it's not always good. Fine powder solder paste tends to oxidize faster, so that the stencil life and solderability, especially in air atmosphere, are not as good as that of larger solder powder. And it costs more as well. This is a trade-off.

Las Marias: *Aside from the miniaturization of devices, what other industry or market trends are affecting the solder paste printing process?*

Tseng: We have seen the LED industry is using stencil printing solder paste for die bonding. Solder paste is printed with thinner stencil such as 0.04 mm or 0.06 mm. The pattern is simple but the tolerance of height is tight. Again, fine powder solder paste is required in this process.

Las Marias: *Can you give a list of best practices for solder paste printing? What factors do users need to consider when it comes to solder paste printing?*

Tseng: A good stencil design, a good solder paste, and a precise stencil printer is the key to achieve best practices for solder paste printing.

Las Marias: *What about equipment—what can you say about the advantages and disadvantages of solder jet paste printing and screen printing?*

Tseng: Jet printing suits very well with NPI or low-volume high variety production. It doesn't require a stencil to get the desired solder deposition. It can easily provide adequate amount of solder to large components while the other tiny components on the same board might need only minimal solder. In stencil printing, a specially designed step stencil is usually required to do so.

However, the output rate of a solder jetting machine prevents it from replacing stencil printer. I'd like to see more application of jet paste in the near future, for example in applications of 3D printing.

Las Marias: *Are there any other technologies out there that you think could shake the solder paste printing process?*

Watson Tseng

Tseng: No, in the next 5 to 10 years, stencil printing will still remain the major process.

Las Marias: *Is there anything we haven't talked about when it comes to the solder paste printing process that you think we should be talking about?*

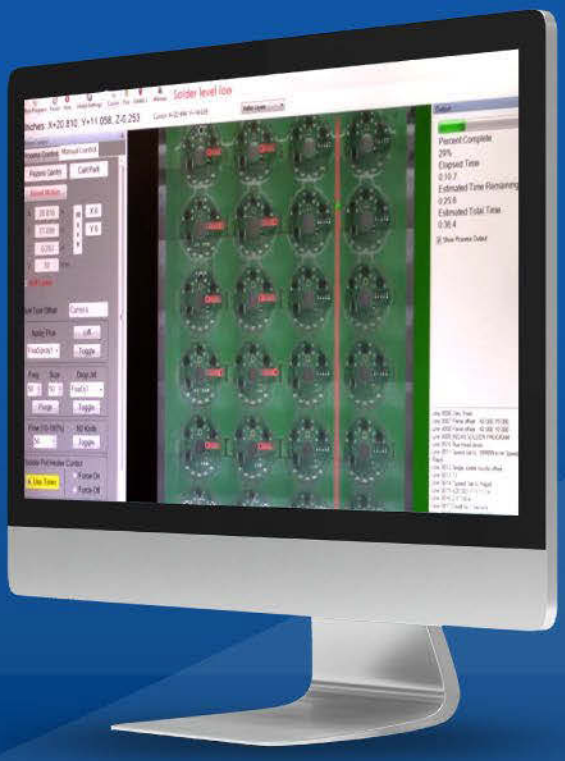
Tseng: I'd like to mention the importance the type of stencil and aperture design make. Laser cut stainless steel stencils have been the major technology used in the industry for many years. We believe it will still be the first choice for many engineers. Their use is low cost, fast and accurate. Yes, the cutting edge is not smooth enough compared with electrodeposition stencils to get the best and consistent solder paste transfer rate, but paste release is improved by nano-coating the stencils to increase the transfer rate and reduce the standard deviation of volume.

People have been studying different components for years to find the best aperture pattern so that common SMT process issues can be eliminated such as voids, shorts, solder balls, etc. However, more and more differently shaped components are designed and used nowadays. Yet, the industry lacks experience and novel approaches to find a better stencil design solution. For example, the void underneath the MOSFET component with large ground pad is still a pain for the engineers.

Las Marias: *Great. Thank you very much, Watson.*

Tseng: Thank you. **SMT**

2 new ways we are adding value while keeping soldering simple.



SWAK-OS Graphic Based Software

Enables graphical monitoring of programs throughout the selective soldering process.

LEARN MORE ABOUT SWAK-OS

Board Warp Compensation System

Algorithmically adjusts the "Z" as the nozzle travels in the "Y" direction between the rails.

LEARN MORE ABOUT BWCS

Now standard on all Selective Soldering Machines!

ACE PRODUCTION® TECHNOLOGIES

KEEP IT SIMPLE SOLDERING

www.ace-protech.com | (509) 924-4898

MilAero007 Highlights

Plexus Names Todd Kelsey to Succeed Dean Foate as CEO

Todd Kelsey, executive vice president and chief operating officer of Plexus, will succeed Dean Foate as president and CEO on October 2, 2016. Foate intends to retire at the end of fiscal year 2016.

Kitron Posts Continued Growth, Solid Order Backlog in Q1 2016

Kitron ASA has reported growth in revenue and order backlog, as well as continued improvement in underlying profitability.

IEC Celebrating 50th Anniversary

IEC Electronics Corp. will mark its 50th anniversary with several events throughout 2016, including Founder's Day recognition at the company's corporate headquarters in Newark, New York as well as celebrations later this year at each of its facilities in Newark and Rochester, New York, and Albuquerque, New Mexico.

API Technologies Announces Completion of Acquisition by Affiliate of J.F. Lehman & Company

API Technologies Corp. recently announced the successful completion of its acquisition by an affiliate of private equity firm J.F. Lehman & Co.

OCM Manufacturing Expands Prototyping Capability with New Pick and Place System

Contract manufacturer OCM Manufacturing of Ottawa has added a new pick and place system to its roster of electronics manufacturing equipment, enhancing the process of surface mounting devices onto printed circuit boards and expanding the firm's electronics prototyping capability.

Sanmina Reports Q2 Fiscal 2016 Revenue of $1.61B

Sanmina has posted revenue of $1.61 billion for the second quarter of fiscal 2016, up from $1.53 billion in the prior quarter and $1.53 billion for the same period of fiscal 2015.

Celestica's Q1 2016 Financial Results Within Guidance

Celestica Inc. has reported revenue of $1.353 billion for the first quarter of 2016, up by 4% compared to the same period last year, and well within the company's previously provided guidance range of $1.3 billion to $1.4 billion.

Vexos Dongguan Receives Business Management Award

EMS firm Vexos Dongguan was the recipient of the Dongkeng Excellent Business Affairs Management Enterprise Award for 2015.

Exception EMS' On-Demand Fast Turnaround Service Sees Major Growth

Exception EMS has invested in a third SMT line to support the growth within its dedicated On-Demand manufacturing facility in line with the 50% increase in its On-Demand service during the past 12 months.

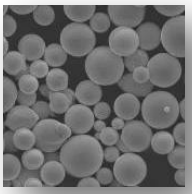
HANZA Posts 27% Sales Growth for 1Q 2016

EMS firm HANZA Holding has reported profitable growth, with significant improvement in earnings and additional new customer projects, for the first quarter of 2016.

Suffering from any of these Solder Paste Problems?

- Voids
- Head-on-Pillow
- Slumping
- Non Wetting Open
- Short Stencil Life

Eliminating These Problems Starts with Higher Quality Flux and Extremely Pure Powder, Resulting in Solder Joints of Ultimate Quality.



- At SHENMAO, we make our own powder.
- We ONLY use virgin powder.

That's why 11 of the 12 largest EMS companies in the world use our solder paste!

SHENMAO
America Inc.

Over 43 years of experience that you can count on.

www.shenmao.com | 408.943.1755

Our Solder Paste is blended in the USA

Pad Defined vs. Mask Defined: Which Method is Optimal?

by Mitch Holtzer
ALPHA ASSEMBLY SOLUTIONS

It seems that there is no universal consensus on a seemingly simple choice used in designing electronic circuits. Should the pads on the PCB be defined by the edge of the solder mask opening (mask defined), or should the solder mask opening fall short of the circuit board interconnect (pad or non-solder mask defined)?

Mask-defined BGA traces give more consistent solder paste transfer efficiency. The effect of the gasket formed between the stencil and the solder mask creates a well-defined cavity into which the solder paste deposit is tightly controlled. Figure 1 shows a print volume histogram of two sets of 15 mil (380 micron) diameter circular features on the same board. This means that the printer settings (squeegee speed, pressure, and board/stencil separation speed) were identical. The area ratio of the pad to the stencil aperture wall were the same, and of course, the solder paste was identical.

The difference in print volume and print volume repeatability favors the mask defined circles. This test used a 5 mil (125 micron) thick stencil for an area ratio of 0.75. The difference is quite significant to a person who frequently reviews these histograms.

3D photos of both mask defined and pad defined circles of the same size are shown in Figure 2. In the mask defined image, a portion of the plated pad is covered by the mask. In the pad defined example, there is considerable room between the pad and the edge of the mask. It is this area that causes the variation in the amount of paste transferred from the stencil to the board. Poor gasketing between the stencil and the solder mask is the most probable cause of this variation in transfer efficiency.

Despite the disadvantage of transfer efficiency and solder paste volume variability, pad defined boards have several process advantages. First, there is more available base metal surface area to form an intermetallic layer with solder paste/BGA alloy. Not only is the available surface area greater, there is an opportunity for the

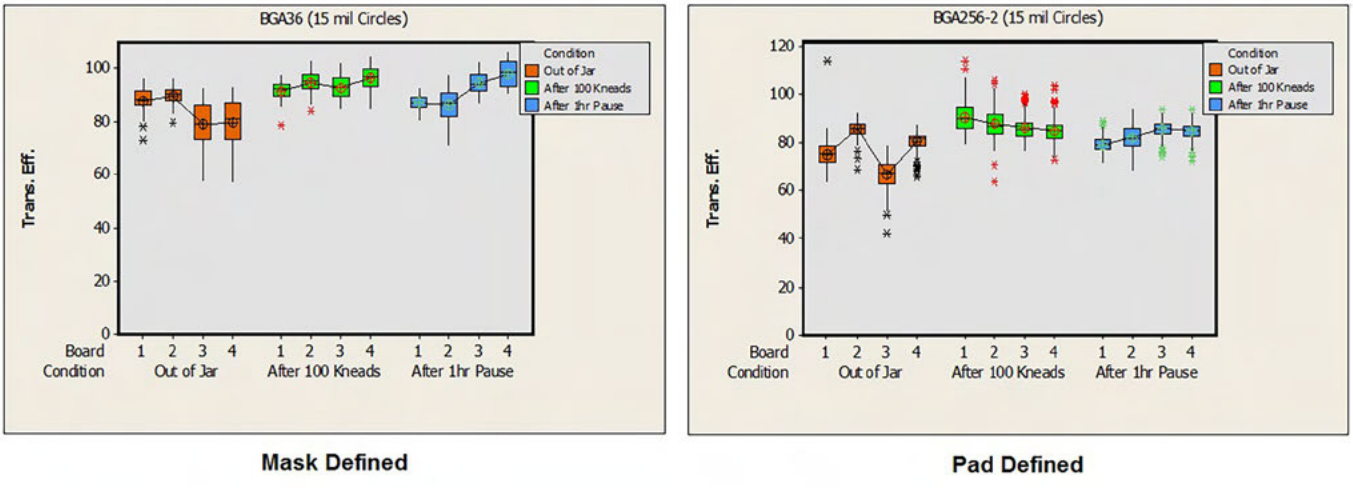
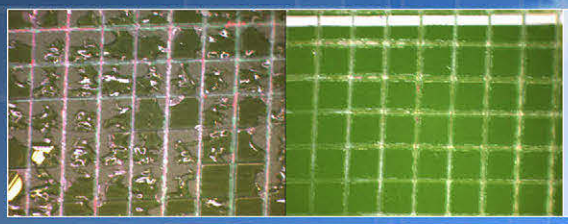
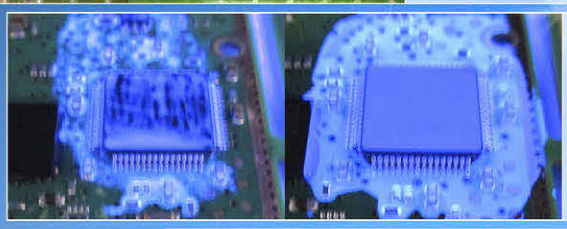


Figure 1: A print volume histogram of two sets of 15 mil (380 micron) diameter circular features on the same board.

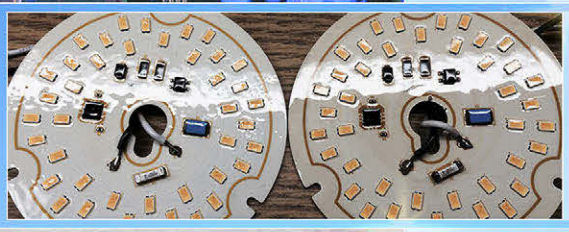
Does Plasma Prior to Conformal Coating Help?



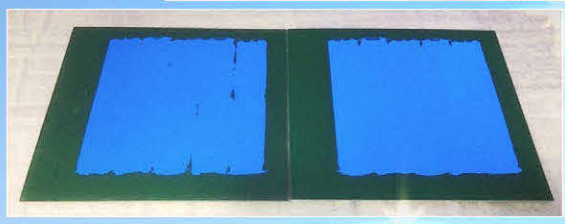
✓ Improve adhesion



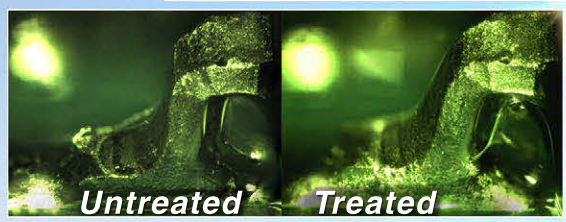
✓ Eliminate flux & mold release residue



✓ Remove organic contaminants



✓ Enhance surface wettability



✓ Enhance lead coverage

See what the Experts say!

nordsonmarch.com/PPCC | +1 (925) 827-1240

Nordson MARCH

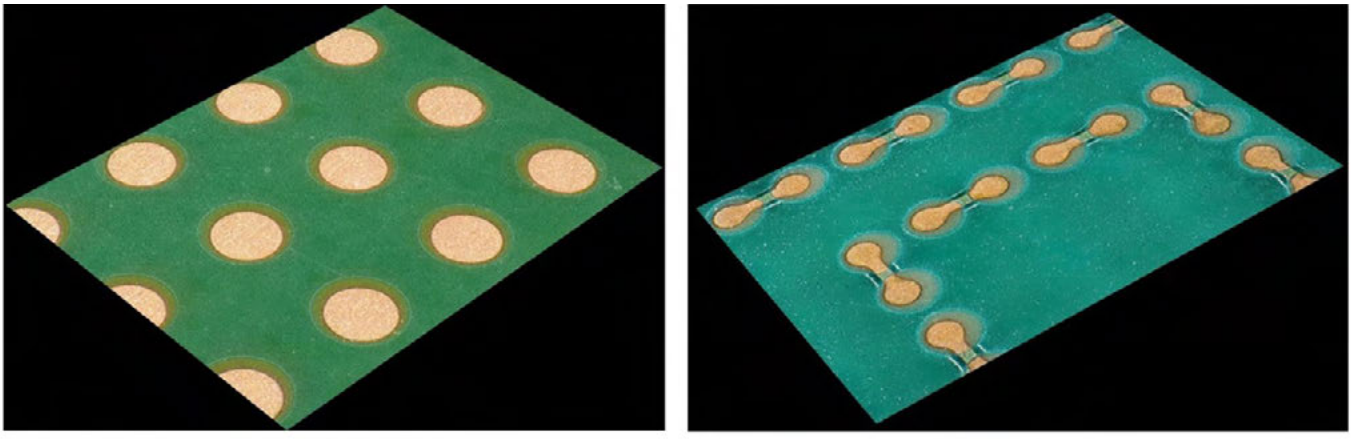


Figure 2: 3D photos of both mask defined and pad defined circles of the same size.

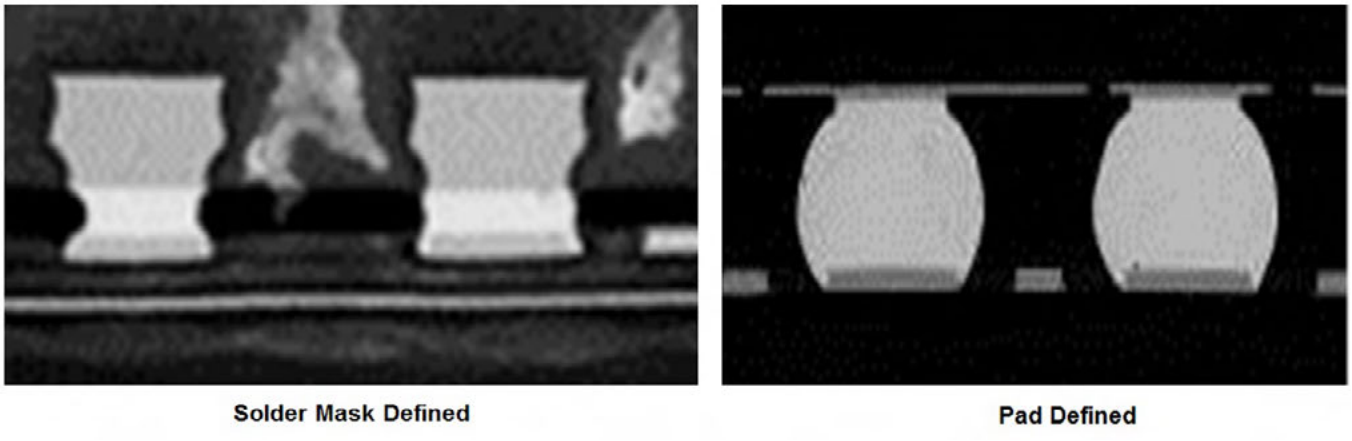


Figure 3: Transfer efficiency and solder paste volume variability in both mask defined and pad defined boards.

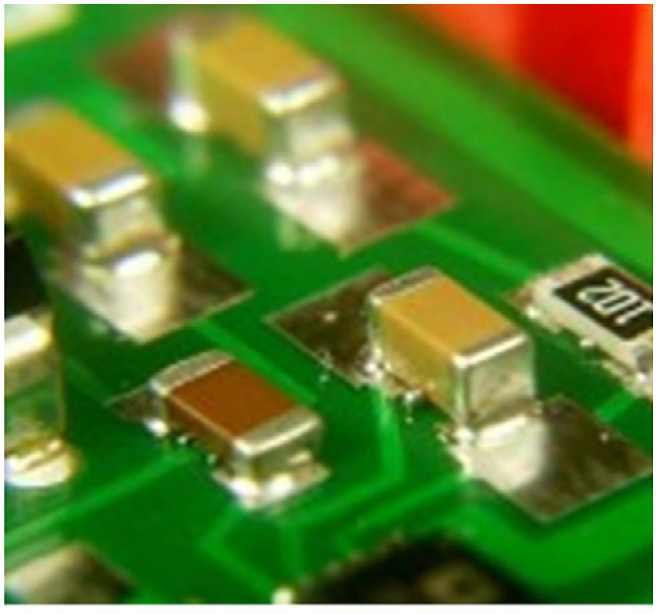
collapsed BGA/paste molten alloy to coat and react with the edges of the pad. This results in a stronger solder joint. Figure 3 illustrates this difference.

Also, the coefficient of thermal expansion (CTE) mismatch between solder mask and SAC alloy will cause stress to the solder joint during thermal cycling, which can lead to premature failure.

Another common issue with mask defined pads is the increased potential for mid-chip solder balls (MCSBs). Mid-chip solder balls are defined as large (>100 µ) beads of solder found in between the leads of a passive component.

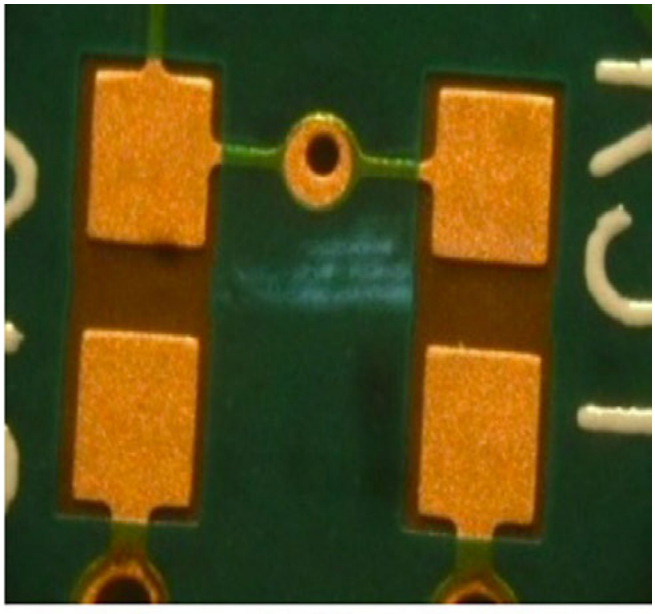
Reducing the volume of paste deposited is one method of reducing mid-chip solder balls, but eliminating the solder mask between the passive component terminations is also an effective resolution to the problem. Using both techniques is generally the best recommendation. Figure 5 shows an example of how a global contract manufacturer solved a mid-chip solder ball problem by eliminating the solder mask between passive component leads without reducing the paste deposit volume. Essentially, this was a conversion from mask defined to pad defined design rules.

Random solder balls can also be reduced with the use of pad defined joints. Slight misregistration between circuit boards and stencil apertures can commonly result in solder paste being printed slightly off the pad. If the pad is



Mid Chip Solder Balls

Figure 4: An example of mid-chip solder balls on a mask defined assembly.



Pad Defined to Eliminate MCSBs

Figure 5: An example of how a mid-chip solder ball problem was solved by eliminating the solder mask between passive component leads without reducing the paste deposit volume.

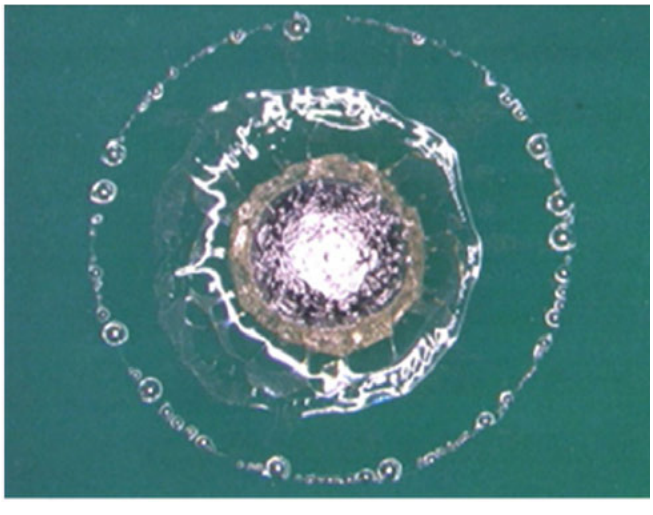


Figure 6: Random solder balls due to paste deposited on solder mask.

mask defined, solder paste may be printed directly on solder mask. Paste on mask does not form an intermetallic and frequently results in random solder balls, as shown in Figure 6.

In summary, mask defined design rules lead to better solder paste print deposit volume and volume repeatability. Because the vast majority of SMT defects are a result of printing issues, there is a strong argument for minimizing the variability in the printing process.

Producing stronger board-to-solder connections is also a highly desirable outcome. Pad defined joints help reach this objective. Reducing mid-chip and random solder balls is also a key goal for many circuit assemblers.

The answer to the question is not as simple as it might first seem. Perhaps the key is to have very good solder paste printing and pad defined design rules, as this will lead to greater electrical reliability in board assemblies. **SMT**

Mitch Holtzer is global director of customer technical service (CTS) for Alpha Assembly Solutions. To reach Holtzer, click here.

Solder Jet Printing: Keeping Up with the Challenges

By Stephen Las Marias
I-CONNECT007

At the recent NEPCON China event in Shanghai, I had the opportunity to talk to Thomas Bredin, area sales manager at Mycronic, about solder paste printing, and how the jet printers are getting up to speed with the challenges such as tighter tolerances and finer lines and pitches.

Stephen Las Marias: *What are the biggest challenges that your customers face when it comes to soldering?*

Thomas Bredin: In our experience, if you don't have a jet printer, you need to compromise on the stencil thickness, or you need to go into step stencil or 3D stencil. It becomes very challenging to have the right volume for every single solder joint. The challenge is, of course, yield; you need a lot of inspection machines to verify that you do have the right volume of the solder paste after screen printing with the different aperture sizes.

We think that when you have the freedom to use a jet printer, every solder joint will be perfect; but the machine is not always fast enough to keep up with the takt time of the line. Obviously, more and more customers are using the jet printer as a complement to screen printing to do add-on jetting. So you have a two-prong approach to achieve the perfect volume.

Las Marias: *What about the trend towards miniaturization? Nowadays, boards are getting smaller, with tighter tolerances and narrower lines and pitches. What's your comment on that?*

Bredin: In parallel to that, when you see smaller boards obviously you need to have a grid of boards actually. It's not fast enough to do one at a time. With many screen printing processes, you have a problem with the stretch of the board, and then you need it to align the stencil and so on. It can be very challenging to have the accuracy needed for those small volumes.

Again with the jet printer, you can do the same as every pick and placer. You can either locate the global producer mark on the panel or

Microtek Laboratories China

Independent Test Services for the Electronics Supply Chain

- CAF, ECM, SIR
- Thermal Shock/Cycling
- SEM/EDS, XRF, FTIR
- IC - Anion, Cation and WOA
- TMA, DSC, TGA, Thermal Conductivity
- Qualification & Conformance Testing
- Reliability Assessment
- Material Characterizations
- Failure Analysis

www.thetestlab.cn techsupport@thetestlab.cn

Thomas Bredin

go down to individual PCBs to have the perfect accuracy, even though the individual boards may be shifted or stretched.

We also see you need to shoot smaller and smaller volumes. Smaller and smaller volumes need a finer pitch, and it's an ongoing process in our R&D team to make sure that our ejector technology keeps up to pace with the customer needs. It is a challenge. To get those small volumes of solder paste is not easy.

***Las Marias:** Do you think that is the end of it? Do you think that there will be no more miniaturization after that, or will it continue?*

Bredin: It's probably going to be pushed a little further. It has a lot to do with our handheld electronics. We want to squeeze in more and more functionality in everything. I think it will go on for a while. As it looks right now, maybe screen printing has come to the end of what is doable.

That will be a great opportunity when we manage to get the ejector technology to give us even smaller droplets of solder paste. We're already down to class six solder paste. We're very dependent upon the medium and if it is jettable or not.

***Las Marias:** The type of solder matters, right? What about factors like viscosity and such?*

Bredin: Absolutely. The viscosity is one of the keys, but also we're shooting at approximately 300Hz. So, 300 times per second a piston is hitting on the solder paste. It is ejected at a very high speed. For the solder paste to stick together and not splash on the board and get all over the place, it uses a combination of the shooting of the ejector technology and the solder paste. The paste definitely needs to be adapted to withstand the treatment.

***Las Marias:** What are the key factors that assemblers should consider when selecting a solder paste dispenser or printer?*

RF/MICROWAVE CIRCUITS? AMERICAN STANDARD CIRCUITS MAKES IT EASY!

Our experts are just one of the ways ASC makes buying PCBs easy. Your expert for RF/Microwave PCBs is John Bushie.

Have a question about designing your RF PCB? Click HERE to ask John now!

Digital | RF/Microwave | Flex/Rigid-Flex | Metal Clad

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com

Bredin: It is very dependent on the type of electronics you're going to do. Screen printing is fantastic technology that's proven itself for decades, but it's come to an end where you most likely cannot see any future development areas where it can expand into. With the jet printer, even though it's existed for just a decade, we are in the beginning of the development and we're now working very close with some of the major solder paste producers. I'm sure we're going to see a lot of development in what is doable.

With the jet printer, you can do every kind of board. Takt time might be an issue, but that is something that can be solved by having multiple machines. I wouldn't be surprised if one day there will be jet dispensers with more than one ejector at a time. My advice: in order to be future-proof, consider a jet printer. Maybe not at the speed or throughput of big volume producers, but with a combination of a screen printer and jet printer, then you can definitely have both. If you don't have the luxury to invest in both, then I would say it depends on the technology and complexity of the board. If you were to say that you may have cavities coming into your production, then of course a screen printer will have great difficulties to do that.

A jet printer doing non-takt can even do 3D printing and shoot into cavities. We can also jet on top of packages, like package on package, and have a lot more freedom. If you are an EMS company, a jet printer is a great choice. For OEMs, it's more dependent on the volume you're looking at and the complexity of the board.

Las Marias: What is Mycronic 4.0 all about?

Bredin: It is our way to try and make Industry 4.0 a little bit more understandable. This is our way to present what we actually have had for many years. We've been Industry 4.0 ready and compatible for some time, and we've made it into a package we call Mycronic 4.0. The machine will sort and store automatically, choose the right program and help production to be less operator dependent. The machines communicate with each other and makes sure every machine in line is actually busy doing the same revision of the same board.

Our products in the line talk to each other. We are also capable to interface with other equipment and software that is controlling the entire factory. That is pretty much Industry 4.0. Many people are interested in Industry 4.0 and they want to be there, but they don't fully understand what it is exactly, or what implication it will have on their production. I wouldn't say that we have scaled it down, but we're trying to make it more hands-on and understandable.

To do a full Industry 4.0 factory where every single machine is talking to each other is

Figure 1: The challenge for many is to keep track of where they have their materials and to have an official material flow in the factory. Mycronic's material handling system can track the quantity and current location of each individual reel, stick or tray.

the ultimate goal. We have narrowed it down to show customers what we can do for them with our equipment. We are waiting to support the integration with any MES system or machine-to-machine communication.

There are a lot of producers in China who are really advanced and want to go that way, but it will be step by step. That's an easier way to understand what investments are needed to get to that stage. To jump from traditional production to a full Industry 4.0 is a challenge for a lot of companies.

***Las Marias:** A step toward "smarter factories."*

Bredin: One way we're trying to be smarter and less dependent on people is to have our Smart Bin in the kitting area for the pick and place. It's a complete paperless kitting instruction, where the bin that then goes into the magazine of the machine will communicate on an LCD display to the operator. When you scan a new read, you will get informed on this display where you should put it.

Also, when you are un-kitting, you will be told if another line or kitting operator is needing the same read, like if they have only one left in the factory. In the un-kitting area, you will also be told on the display where that needs to go, maybe back to the main warehouse or to your colleague doing a job next to you. That will also help make the factory smarter.

***Las Marias:** What about the future of the SMT industry?*

Bredin: I wish I knew. The complexity is growing. From our perspective, we see a huge potential with the jet dispenser and jet printer. For us, the future is really bright. At this exhibition, you can see a lot of intelligent storage capacity solutions to handle material. The challenge for many is to keep track of where they have their materials and to have an official material flow in the factory. Batch sizes seem to be smaller and smaller, so the effort needs to be put into being able to run timely production efficiently.

That's the area where I think a lot of customer requests will be. To invest in a full SMT line requires an awful lot of money. You want it to be running and producing boards. Not idle and waiting for operators or material to arrive to support it. That's where this smart factory or Industry 4.0 is important to help those running shorter and shorter batches. Mycronic grew up on speed. We've always been addressing these needs, like the way that we load and unload quickly and keep track of material. There's no changeover time in the jet printer and jet dispenser. You just select a program and start printing.

In the future, we'll never live without electronics. There's going to be more and more demand. Most likely, with the life cycle of many products getting shorter, there's a bigger demand for NPI or a lot of products at the end of the life cycle making production batches shorter in manufacturing. This is where a lot of companies are lacking at the moment: the right equipment.

***Las Marias:** I've read that sometimes, full automation is not the answer, but a mixture or a hybrid where you have a semi-automated line because you have to be flexible.*

Bredin: Absolutely. Up until recently, the only fully automated line was for consumer electronics running the same type of board day in and day out; but with the right equipment you can be extremely flexible and fully automated. All while being proactive. Give the operators and the people working the line the right information at the right time. Not when the machine stops, but before it's about to run out. For instance, our pick and place machines will get a proactive replenishment message that it communicates with the towers and will provide a need for replenishment. You set the time window. No time is wasted. Then, you can run efficiently with less people involved. That is also a reason for automation, because the labor cost impact on the total price of the product you are producing is increasing.

***Las Marias:** Exactly. Thomas, thank you very much for your insight.*

Bredin: You're welcome. **SMT**

Electronics Industry News
Market Highlights

U.S. Aerospace and Defense Industry Supported Almost 2.8 Million Jobs in 2015

The economic and employment contributions from the U.S. aerospace and defense industry are felt throughout the U.S. economy; increasing employment, economic growth and local, state and federal tax revenues, according to a new study by IHS Inc.

Smartphone Shipments Reached 292 Million Units in Q1 with iPhone Plunging 43.8%

Global market research firm TrendForce reports that the worldwide smartphone shipments in the first quarter of 2016 totaled 292 million units, down 18.6% from the previous quarter and a year-on-year decline of 1.3%.

Market for Organic Photovoltaic Panels will Reach over $355M by 2021

The market for organic photovoltaics used for building integrated photovoltaics, solar chargers and off-grid energy generation will exceed $355 million by 2021 and reach more than $580 million by 2025, according to a new report by industry analyst firm n-tech Research.

IoT to Account for 28% of Wireless Connectivity IC Market by 2021

Smart home, beacon, wearable, and other nascent IoT applications including energy management and smart cities are to propel the wireless IoT connectivity market forward, according to ABI Research.

China's Flash Memory Capacity to Reach 590,000 Wafer Pieces by 2020

Chinese semiconductor companies are in the midst of building a globally competitive NAND Flash industry and have accelerated their investments and deal-making in the related component and product chains.

Global Smartphone Growth Goes Flat in Q1

According to the latest preliminary release from IDC's Worldwide Quarterly Mobile Phone Tracker, vendors shipped a total of 334.9 million smartphones worldwide in the first quarter of 2016, up slightly from the 334.3 million units in 1Q15, marking the smallest year-over-year growth on record.

Global Robotics Market to Reach $53.22B by 2020

The global robotics market was $25.69 billion in 2014 and is estimated to reach $53.22 billion by 2020, growing at a CAGR of 12.91% for the period.

IoT 2016 Merger & Acquisition Activity Off to a Fast Start

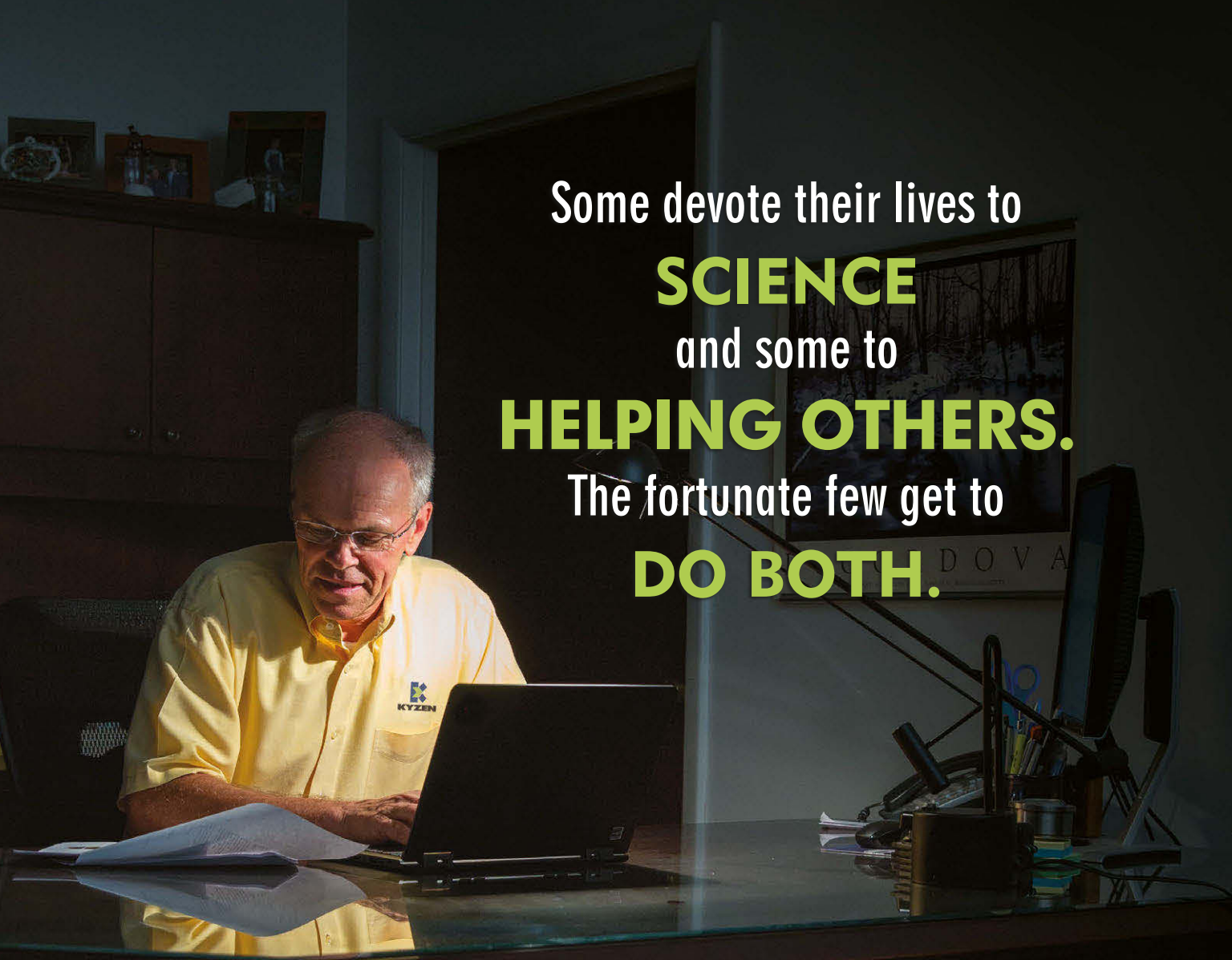
According to Strategy Analytics, there were nearly two dozen major mergers and acquisitions in the Internet of Things (IoT) and related market segments including Big Data Analytics, connectivity and wireless markets in the first four months of 2016.

2015 Semiconductor Photomask Sales Reach $3.3B

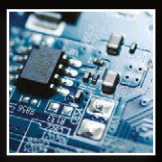
SEMI reports that the worldwide semiconductor photomask market was $3.3 billion in 2015 and is forecasted to reach $3.4 billion in 2017.

Cumulative Wi-Fi Chipset Shipments to Reach Over 20 Billion by 2021

Market analyst ABI Research anticipates more than 20 billion Wi-Fi chipsets to ship between 2016 and 2021.



Dr. Mike Bixenman, CTO

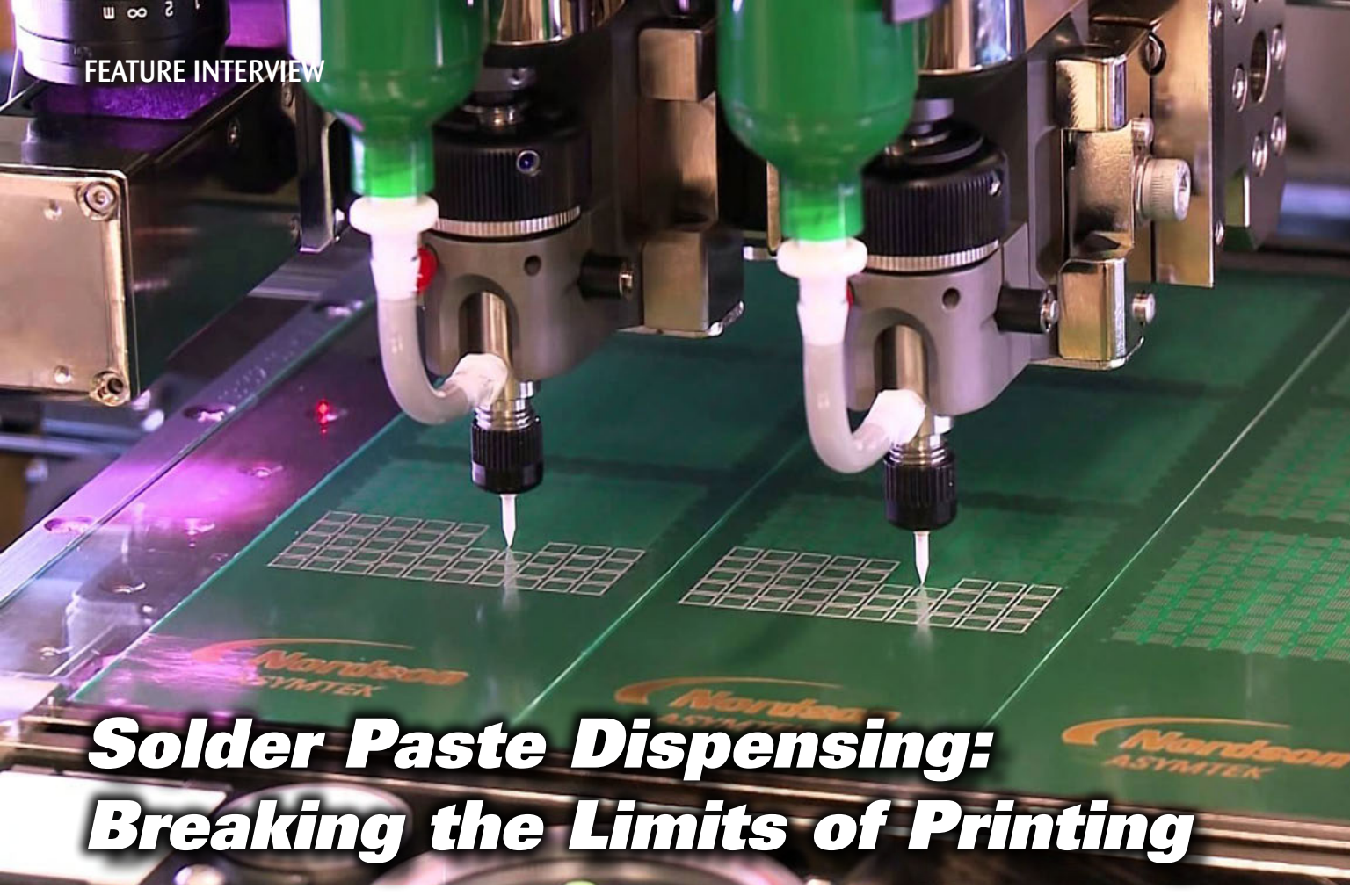


At KYZEN we go way beyond just getting the **CLEANING SCIENCE** right. We care enough to thoroughly understand your unique process and needs first, so we create the most effective cleaning technologies for your specific situation. When science and care converge, it makes all the difference.

WORLDWIDE ENVIRONMENTALLY RESPONSIBLE CLEANING TECHNOLOGIES

KYZEN
Where Science and Care Converge.

KYZEN.COM



Solder Paste Dispensing: Breaking the Limits of Printing

by Stephen Las Marias
I-CONNECT007

At the recent NEPCON China, I sat down with Eric Gu, application manager at Nordson Asymtek China, to talk about the challenges and innovations in solder paste dispensing, and factors to consider when using solder paste dispensers.

Stephen Las Marias: *Eric, what are the greatest challenges that manufacturers have when it comes to soldering?*

Eric Gu: Before, we were normally using screen printing for the solder paste process. But nowadays, products such as mobile phones, cameras or smart watches have become smaller and thinner. Therefore, the PCBs inside these products have become smaller and smaller, including the chips, the packages, and the components.

So the current printing process would be limited by this trend. Also, printing will be limited by the gap. If the components are tall, we cannot do printing. So, it will require solder paste dispensing. That's where we are. We have a needle valve for solder pasting. And also, when dispensing solder paste, you know, there are some alloys in solder paste that may cause needle clogging during dispensing. We have an extensive experience on solving these kinds of issue. We have very high level of success in solder paste dispensing.


Eric Gu

Las Marias: *Definitely the shrinking components are a big factor. But as we go smaller, the solder pastes being dispensed definitely are also becoming dot sized. What is the impact of the material in the solder on this trend?*

Gu: Yes. The component is becoming smaller. It will only require to dispense a very small dot and hold a very narrow line, which is a key challenge. Another challenge is that material inside the solder paste can cause needle clogging.

Low Maintenance and Highest Speed

Pioneering X-ray inspection combining AXI and AOI

Free Webinar

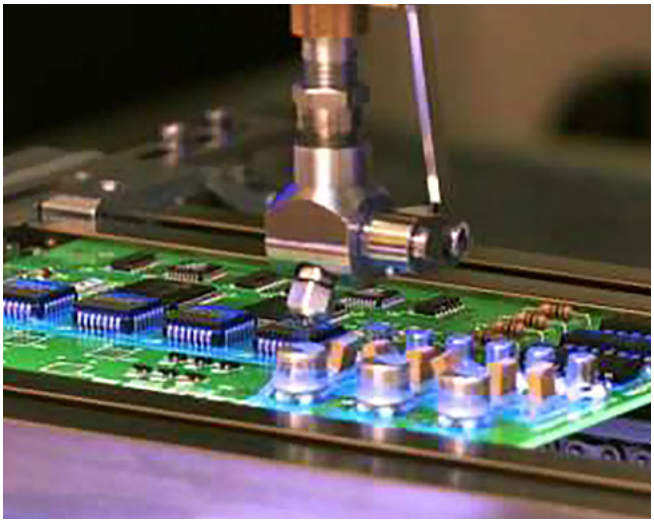
From Automotive to Medical Technology – Combined 3D X-ray (AXI) and AOI Inspection in the Series Production

June 7, 2016

www.goepel.com/axi-webinar

GOPEL electronic

Enjoy Testing



Las Marias: *So what innovations in solder paste dispensing or jetting are happening to help customers address that issue?*

Gu: Actually, we are using needle dispensing instead of jetting, but I know there are maybe one or two vendors here that are using solder jetting. Of course, solder jetting is much faster than solder-based needle dispensing. But you know, the key limitation of solder-based jetting is that we cannot use many types of solder pastes. Solder pastes have many types—type 1 through type 6. The smaller the solder particle, the more expensive the paste. For instance, a type 6 solder paste is maybe 10 or 15 times more expensive than type 5 solder paste. For solder paste jetting, maybe we can only use type 5 or type 6. But for our needle dispensing, we can use everything from type 1 to type 6—this is one advantage of needle dispensing.

Las Marias: *Why do you think most manufacturers still prefer using solder paste printing?*

Gu: Solder paste printing is much faster than solder paste jetting or dispensing. It is also much stabler because it is a one-step process. But for very complicated boards, with very small or very tight accuracy levels, you will need solder paste dispensing.

Las Marias: *Can you tell me some of the factors I should consider when selecting a solder paste dispensing system to use?*

Gu: We are using needle dispenser, so I will speak from that perspective. In needle dispensing, we use air pressure to press the solder paste into what we call a package, just like a tank. The solder paste will be soft for a little while. We use a kind of motor to push the solder paste through the needle. So the needle size selection is very important because there's always some filler inside of each solder paste, and it's different for each type. For example, type 1 will have the bigger filler size, and type 6 will have the smaller filler size. So for example, if you want use type 6 solder paste, the filler size may be 20 micrometers. Then the inner diameter of the needle must be seven times bigger than the filler size—so 140 microns. This is a rule. Otherwise, you will encounter problems.

Another thing, you need to control the solder paste flow and always keep it stable. There are many factors, like air pressure, motor speed, and dispenser head, you have to control the flow rate to make the dispense line very stable.

Las Marias: *Eric, what can you about the future of your industry?*

Gu: Technically, I think the dispense quality will become more and more strict and more complicated. Maybe it will require much smaller dispense dots, solder paste dots, or line. Right now we can dispense around 215 microns. Maybe later we will be down to 100 microns. So this is a very big challenge. Another thing is the speed, the UPH. That's why more and more vendors are developing solder paste jetting, but you have to balance the cost and the UPH. If you are using solder paste jetting, of course, the cost is maybe three or four times higher. Also, for the material, solder paste jetting can only use type 5 or type 6 pastes; so another challenge for the future is how to start using type 2, type 3, and type 4 solder pastes in jetting.

Las Marias: *Eric, thank you very much for your time.*

Gu: Thank you. **SMT**

Streamline your production with TS16949 and AS9100 certified PCBs!

Our special program for Contract Manufacturers has been designed considering the supply chain needs of **on-time delivery** of **quality PCBs** at the **best prices**.

Our PCB's help increase your product's RELIABILITY

- Complete lab analysis
- Solderability, final finish thickness report and ionic contamination report
- High-end fixtureless electrical testing
- Cross-section analysis capabilities
- Only UL-approved materials are used
- Conventional Electroless processes
- Stringent TS16949 and AS9100 norms

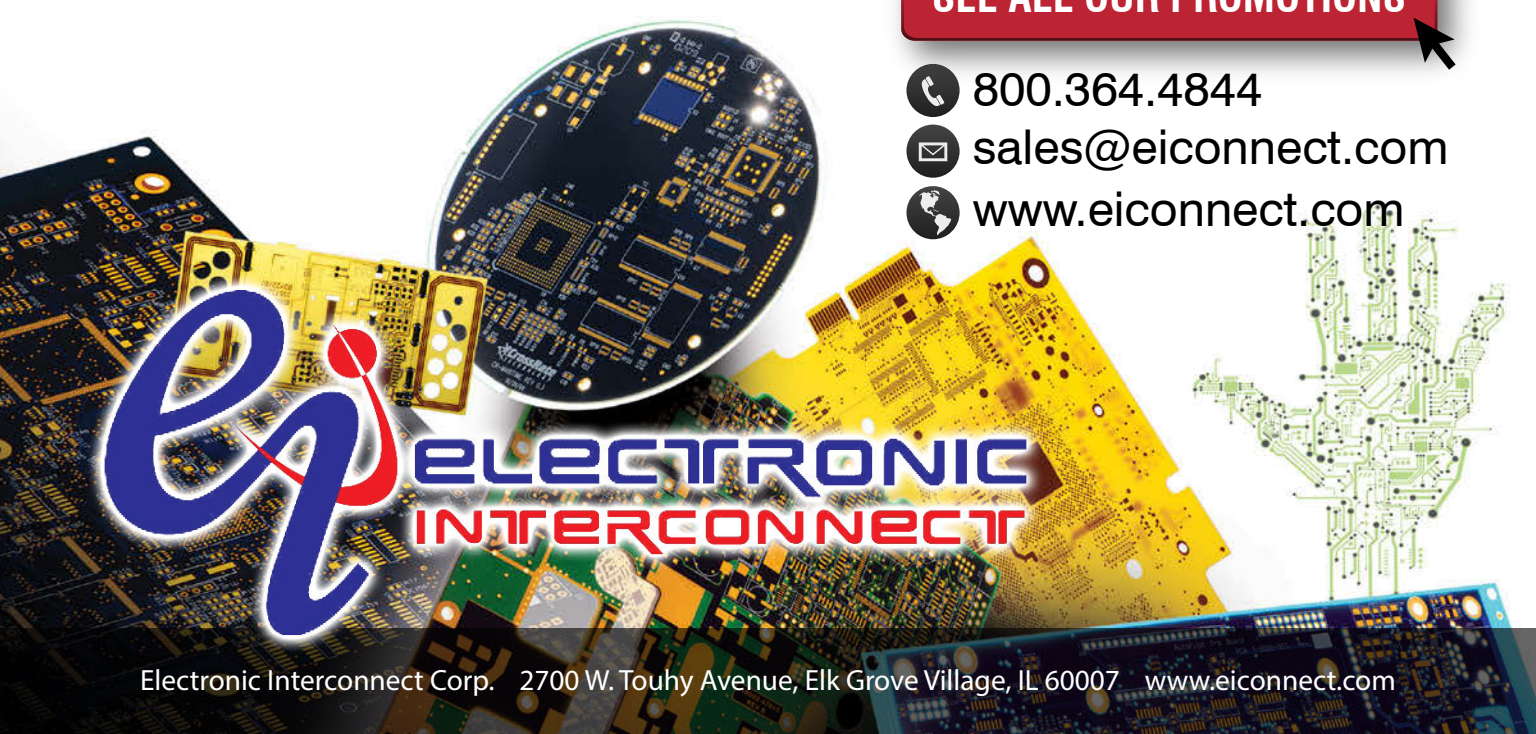
Our increased capabilities give you a ONE-STOP SOLUTION

- Single, double sided, multi-layers or Rigid-flex
- Buried, blind and micro vias
- Metal PCBs – up to 4 layers
- Material thickness from 0.005" to 0.250"
- Line width and space is 0.005"
- Carbon inks, conductive epoxies, filled holes
- UL approval on 6 oz. copper

ALONG WITH OUR NEW CUSTOMER DISCOUNTS, WE ARE OFFERING FREE TOOLING AND 5-DAY DELIVERY AT NO EXPEDITED CHARGES

SEE ALL OUR PROMOTIONS

800.364.4844
sales@eiconnect.com
www.eiconnect.com



Electronic Interconnect Corp. 2700 W. Touhy Avenue, Elk Grove Village, IL 60007 www.eiconnect.com

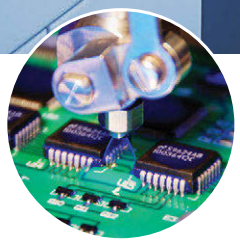
The Production Engineering Student as Customer

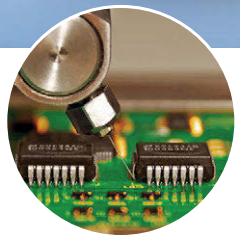
by Tom Borkes
THE JEFFERSON PROJECT

It was the winter of 1972, between autumn and spring semesters. I was finishing my senior year at university and totally immersed in my quest to graduate with my B.S. degree in engineering in June—the end of the spring semester. This took on greater significance when I was continually reminded that I had managed to squeeze a four-year program into six years! In fairness, the learning disabilities with which I suffered and those that had plagued me since I was 11 years old showed no sign of abating. I had learned that dealing with this condition required laser beam focus on my studies. Back then, science had no fancy terms for the two syndromes I struggled and wrestled with. Today, collectively, they are known as GARAR Syndrome: Girls And Rock And Roll.

It is now humorous and comforting for me to reminisce about this pivotal period in my life—ironic considering the angst that enveloped my existence at the time. The other thing that is astounding as I look back is how little practical understanding my classmates and I had concerning the relevance and value this education would have for us in our soon to be brave new real world. We would soon be attempting to blaze a path to make our mark in that real world after graduation, and we had no clue! How prepared would we be? The unwritten pact with our schools was, "If you want to be an engineer, do what we tell you, work hard, and get good grades." Maybe the beckoning, rich job market at that time with almost unlimited opportunity deadened the din of the uncertainty—life was good! We were, however, the easiest marks to hit the scene since the farmer from Iowa was drawn to the three-card Monte table while walking through Times Square! It turns out that for me, I wouldn't change my academic experience for anything—the professors in the front of the classrooms were all first rate in their areas of academic specialty—from, structural analysis

Whether large or small, we've got you covered






We've got a conformal coating solution that fits.

Whether you run a small operation or manage an industry giant, Nordson ASYMTEK has the solution that's right for you. Our conformal coating systems are built with state-of-the-art platforms, applicators, and software. They are backed by an award-winning, global network of service professionals and applications experts.

Please visit our website or contact us today.

info@nordsonasymtek.com
nordsonasymtek.com/coating

Select Coat®
SL-940

Conexis™
CX-3040

Nordson
ASYMTEK

to advanced engineering mathematics to quantum mechanics (I minored in physics).

Perhaps the incident that best illustrates this naivety occurred at that same point in time—winter, 1972. The break between the fall and spring semester, which we used to call the Christmas break, was a time of family activity. For me, it was my dad packing up the Oldsmobile on Christmas day, my mom taking an insufferable amount of time "getting ready," and then, the six of us off to church, followed by the hour drive (at least) to "Gram's" in Brooklyn, then another hour drive (at least) to the other "Gram" in New Jersey (Turnpike, Exit 11, for those of you Jerseyites).

My dad was the oldest of five children that spanned 30 years! I am the oldest of four. So, my dad's youngest sibling, my Uncle Joe, is only 10 years older than me; he was more like a brother than an uncle as I was growing up. He became comptroller of a wire and cable company in Jersey and took an interest in my engineering education since he worked with engineers. So here's the point: That Christmas, just a few months from graduation and my bursting forth into the real world, my Uncle Joe says to me, "Tom, I guess it's all about specs, right?"

"That Christmas, just a few months from graduation and my bursting forth into the real world, my Uncle Joe says to me, "Tom, I guess it's all about specs, right?"

"What?" I said. "Say that again, Uncle Joe."

"I said, I always hear the engineers in the office talking about meeting specs. I guess that's what engineering is all about, right?"

I said, "yeah," but, I didn't know what he was talking about. Spec, what's a spec? In school I had never heard that word before! So in 1972, academia met the real world for me.

This brings us to this month's topic: The Production Engineering Student as Customer.

As a customer, I know when I get a good haircut and when I get a bad haircut. The mirror I am looking into as the haircut process proceeds, either provides a feeling of reassurance or a progressively deepening sinking feeling. As educators in the manufacturing sciences, do we treat the student as the customer? The student doesn't have the ability to look in a mirror as the educational process proceeds toward graduation.

Let me start by making a bold prediction: production, industrial, process, manufacturing engineering (choose any of the above) will never be successfully taught on-line.

Other academic disciplines have geared up to create a virtual classroom for the subject matter they have traditionally taught in a space created by brick and mortar. Most of us in the electronic high-tech "building" business realize that cyberspace cannot be used as a substitute for learning in the real world of a production operation (i.e., getting up to your elbows in solder paste, or watching the metal chips being produced from a CNC milling machine or plasma cleaning a circuit board prior to wire bonding a chip scale package. In fact, I would suggest that even a physical classroom with desks in an orderly matrix configuration surrounded by white—or, as I prefer chalk—boards results in dismal failure when attempting to teach product production. I would submit as evidence the U.S. track record in globally competing in building electronic products.

The educational preparation problem runs deeper than this and is multi-faceted. At its base, however, is this: Who is doing the preparation, what real world experience do they have to complement their academic achievement, what are the preparers' motivations and objectives, and what metrics are used for judging the effectiveness of the preparation?

I like to compare an individual's educational process to moving through a pipeline. When we are about five years old we all enter the state-administered pipeline. However, we leave the pipeline at different points. The state (taxpayers) will fund the trip through the primary (elementary/middle) and secondary (high

school) segments. The reason for this is the recognition of the incredibly important role an educated population has in a representative republic (as opposed to a collective, authoritarian government). Each individual has the ultimate sovereignty and freedom that was not granted by a government, but naturally occurred when they were born. Each individual agrees to yield a portion of that freedom to a government they elected by granting the government certain limited, enumerated powers. Thomas Jefferson believed there were two crucial elements in the success of the experiment in self-government that he helped begin, one where the government served the people and every individual was equal under the law.

This disciple of democracy proposed a state-funded primary and secondary educational system for his state of Virginia. What?! The man to whom many attribute the quote, "the government that governs least governs best," (actually, the quote is properly assigned to Henry Thoreau, but certainly contains Jeffersonian sentiments as well). Thomas Jefferson proposing that the Virginia State government is granted the power and responsibility to educate the people? Can it be? Yes, because there was something more important at stake—the survival of the nascent United States of America.

Thomas Jefferson believed there were two crucial elements in the success of the "experiment" in self-government he helped begin; a government that served the people instead of the other way around. These were:

1. An educated population:

"If a nation expects to be ignorant and free, in a state of civilization, it expects what never was and never will be."

–Thomas Jefferson to Charles Yancey, 6 January 1816

2. A virtuous population—and, education helps promote virtue.

"I look to the diffusion of light and education as the resource most to be relied on for ameliorating the condition, promoting the virtue and advancing the happiness of man."

–Thomas Jefferson to Cornelius C. Blatchly, 21 October 1822

Note the following facts about our current educational pipeline:

- In the U.S., some exit the pipeline before successful completion of high school: about two out of 10.
- Among the eight out of 10 completing high school: about six of these go on to college, the post-secondary segment of the pipeline.
- Only about one-third of the six of 10 who enroll college (two), graduate with a degree from a four-year program.

– *Source: National Center for Education Statistics*.

So only two out of 10 make it through to the end of the post-secondary segment of the pipeline—that is for ALL students, ALL majors. And, we all have that sinking feeling that learning standards have dropped significantly. If the standards demanded in the decades of the '50s and '60s were maintained today, the success rate would be significantly less. This is the perception I have concerning industrial or production engineering students. It's based on discussions I have had with those who teach in academia and recent grads I have worked with in industry. However, it's not linear. There are some incredibly gifted students that form a small percentage in the total industrial engineering population, but for the most part the masses manage to slide through and into industry, relative to the standards of 50–60 years ago. This is true without even questioning the relevance of the skill set with which they graduate.

Do students today have less native intelligence? Are they incapable of the same level of academic achievement? Is the issue the additional new technologies they are forced to master? Has the physics changed? How much does the fiscal success of a post-secondary school (college) rely on keeping their classrooms filled? If the preceding part of the pipeline, the secondary segment, reduces standards for social engineering motives (everybody gets a trophy!) and sends the ill-educated on through, what does a college do? What they do is find a way to keep their classrooms filled. And now we hear some politicians say, *everyone is entitled to a free college*

education. It is their natural right! If you listen carefully you can almost hear the rumble below the ground at Monticello where Jefferson is buried under the grave monument he designed. On this stone obelisk is inscribed the three achievements for which Mr. Jefferson hoped to be remembered:

1. Author of the Declaration of American Independence
2. Author of the Statute of Virginia for Religious Freedom
3. Father of the University of Virginia

The two paragraphs after the preamble in the Declaration make what the founders felt very clear: Governments are instituted to protect the natural rights of each individual to life, liberty and the pursuit of happiness.

Note: It's the *pursuit* of happiness, not the *guarantee* of happiness.

And, ironically, this right of individual liberty (freedom) breeds inequality in individual results. So twisting Jefferson's intent to mean equality of results sets up a conflict with his true intent of ensuring individual freedom. The two cannot co-exist purely as one or the other. In the end it is left to the people to decide how much individual freedom they are willing to give up—the people get the government they deserve, as Alexis de Tocqueville, Thomas Jefferson, and others have said in one form or another since the American experiment was begun.

In a real sense then, the citizen is the *customer* of the government in the narrow sense that is defined in the U.S. Constitution. And, the citizens can fire and hire as they see appropriate.

The student *customer* normally does not have the visibility to make a judgment on the value of their production engineering education.

The real world companies a U.S. student ultimately goes to work for however indicates the skills recent grads bring to the workplace are not the ones that are needed to give high labor rate regions of our industry an edge in competing in the global product production marketplace.

There are three issues:

1. In high tech electronics design and manufacturing, as students move up through the educational pipeline the subject matter that is taught to them by academia progressively departs from the real world skills they need to be most effective. This would be okay if the abstract material (learning for learning's sake) was supplemented by practical applications that were based on the abstract material (learning for earning). This would close the gap (Figure 1).

2. The people teaching usually have little real world experience in high tech electronics design and manufacturing.

3. Other competing factors that the educational community is confronted with are in conflict with treating the student as the customer (see below).

One thing that seems intuitive is that the ability for academia to create a correspondence between what they are attempting to teach and

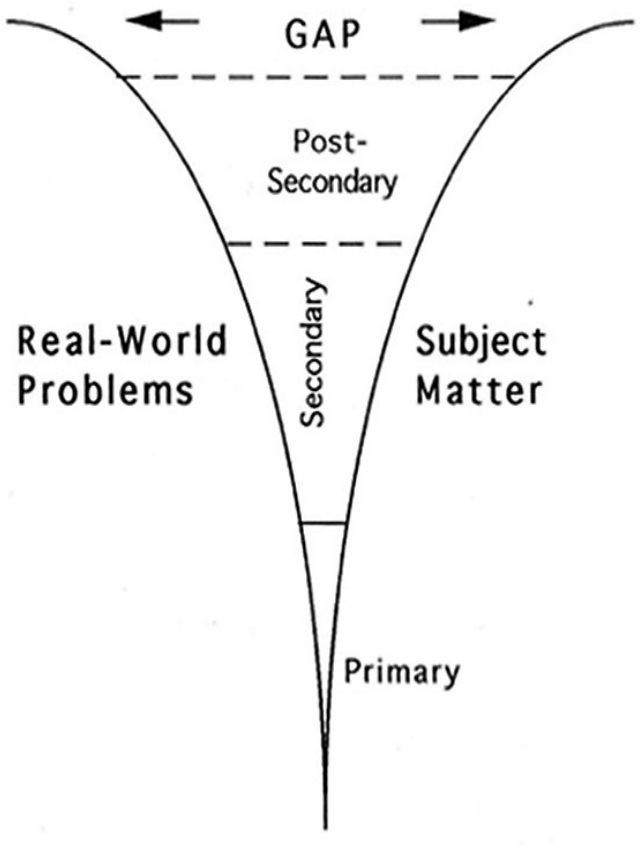


Figure 1: Gap between a subject's complexity or abstractness, and the ability to create a correspondence with real-world problems—as a function of education level.

the real-world becomes more and more difficult as we move through the pipeline.

Here are two examples:

1: 2 + 2 = 4. First, hold up two fingers, then add two more fingers; count all the fingers.

2: Take a complex, time-based waveform and decompose it into the frequency domain (a linear combination of sine and cosine waveforms in the frequency domain using a Fourier Transform). Solve the differential equations in the frequency domain and convert back to the time domain.

I think it is clear to the most casual observer that the ability to relate example 1 to the real world is much easier than example 2—that's easier, not impossible.

Colleges MUST look at the student as the customer. This means:

1. Putting the interest of the student first, not the fiscal performance of the school, or securing government grants to the school through their faculty members.

2. Understanding the industries their students will be entering when they graduate. Recognizing the skill sets (both technical and social) they need to provide their graduates with; those skills that the companies who ultimately employ the students need to help them successfully compete on the global playing field.

3. This recognition includes understanding that companies will continue to reduce labor content in product assembly by replacing labor-intensive processes with automated processes.

4. This understanding must help shape the school's curricula to provide a balance of learning for learning and learning for earning. This means a faculty that has real world experience to complement academic teaching excellence.

5. A process and mindset that encourages continually changing curricula to meet the needs of an ever-changing industry.

Skill sets for employees are getting more and more complex as the simple tasks of the past, such as taking orders as a fast food restaurant and preparing the food, will be replaced by automated systems. And it's not a matter of *if*, it's a matter of *when*.

Two final examples to support the thesis:

By now you have probably encountered the multi-drink/flavor machines available at some fast food outlets. This was a gift from above for me as I do considerable work when I am on the road in one of my Wi-Fi enabled fast food offices. But, I've noticed something lately. Many of the selections are *not available*. Why? The reason is none of the employees are capable (or, have been trained) in reloading the flavor cartridges. So, they go empty and present as *not available*. The technology even in this entry-level business is exceeding the employee skill ability.

> "The technology even in this entry-level business is exceeding the employee skill ability."

Let's stay with fast food. The demand for a $15.00 minimum wage has entered into the public discourse and debate.

"You can't live on $8.05 per hour!" (Florida's minimum wage)

"Raising the minimum wage will cost jobs!"

"But how can you run a business like this with fewer employees?"

Well, the same way high labor rate regions of the world can compete in high tech electronic product assembly: Reduce labor content through automation. And, you know what, it's here.

This YouTube video shows an ordering/burger preparing robotic system that will replace minimum wage workers.

The robot clip begins at about the 29 second mark after the ad.

So, it's all about having marketable skills, whether it's the fast food industry or the high-tech electronic product assembly business. And as history teaches us, labor-intensive skills will be ultimately displaced by machine and replaced with higher level skills, whether it is critical thinking, working in teams or developing automation. How about that, Mr./Ms. Person in the ivory tower?

Bob Dylan wrote:
You may be an ambassador to England or France
You may like to gamble, you might like to dance
You may be the heavyweight champion of the world
You may be a socialite with a long string of pearls

But you're gonna have to serve somebody, yes
Indeed you're gonna have to serve somebody
Well, it may be the devil or it may be the Lord
But you're gonna have to serve somebody

—*Gotta Serve Somebody* by Bob Dylan,
(Slow Train Coming)

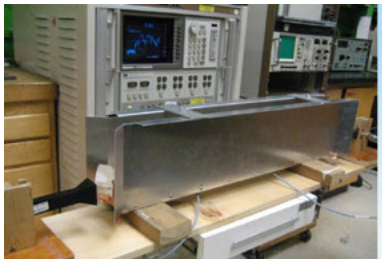
Who will our educational system serve?

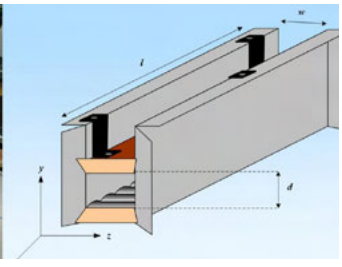
Hey, what do YOU say? I'd like to hear your thoughts. **SMT**

Next Month: The STEM Trap

Tom Borkes is the founder of The Jefferson Project and the forthcoming Jefferson Institute of Technology. To reach Borkes, click here.

Engineers Discover New Gatekeeper for Light





Imagine a device that is selectively transparent to various wavelengths of light at one moment, and opaque to them the next, following a minute adjustment. Such a gatekeeper would enable powerful and unique capabilities in a wide range of electronic, optical and other applications, including those that rely on transistors or other components that switch on and off.

Researchers in the University at Buffalo School of Engineering and Applied Sciences recently reported a discovery that brings us one step closer to this imagined future.

The finding has to do with materials that are periodic, which means that they're made up of parts or units that repeat. Scientists have known since the early 20th century that periodic materials have special qualities when it comes to light. Such materials can reflect light, as butterfly wings do, and if you understand the internal structure of a periodic material, you can use an equation called Bragg's law to determine which wavelengths will pass through the material, and which will be blocked due to reflection.

The new UB study shows that a completely periodic material structure is not needed for this kind of predictable reflection to take place.

"We have shown that Bragg's law is a special case of a more generalized phenomenon that was discovered in this study and named as a Bloch wave resonance," said Victor A. Pogrebnyak, an adjunct associate professor of electrical engineering at UB. "This discovery opens up new opportunities in photonics, nanoelectronics, optics and acoustics and many other areas of science and technology that exploit band gap wave phenomena for practical use."

"Electrons behave as waves that can also exhibit a Bloch resonance, which can be used as a powerful method to control currents in nanoelectronic circuits," said Edward Furlani, Pogrebnyak's co-author and a UB professor in the Departments of Chemical and Biological Engineering and Electrical Engineering.

A key advantage that Bloch wave resonance offers: It enables the blocking of a larger range of wavelengths simultaneously than previously known effects described by Bragg's law. Applications that could take advantage of this broader "band gap" range include white light lasers and a new type of fast-switching transistor.

DIBBLE 4 LEADERS

THE FOUR NEW AGREEMENTS FOR SUSTAINABLE SUCCESS

Profitability is more about leadership and management than any other factor.

Any business can be transformed quickly, easily and sustainably with the proven effective systematic methods we employ. For over 25 years, our clients have averaged a 300% ROI on this work. Returns of 1000% are not unusual, especially in early stages of implementation. **Contact us today** and get started on the path of doing the right things that allow growth and profitability to happen.


BOOM!

The Four New Agreements to EXPLODE PROFITABILITY

click to read

Learn more about the roadmap used to build great companies with a high level of profitabiity in this article from the March 2016 issue of **The PCB Magazine.**

For 25 years we have been doing Four New Agreements consulting and training, significantly improving businesses. This stuff really works!
—David Dibble

4newagreements4leaders.com

858-775-0722 Pacific

SPI Parameter Considerations for Tighter Tolerances

by Stephen Las Marias
I-CONNECT007

In an interview with *SMT Magazine,* Vi Technology's Jean-Marc Peallat, vice president global sales, and Chong Choon Hee, Asia sales application manager, speak about the solder paste inspection (SPI) challenges when it comes to tighter tolerances and finer lines and pitches in board assemblies, and the latest innovations happening in the SPI space to help customers address these issues.

***Stephen Las Marias:** Jean Marc, what are the challenges that your customers face when it comes to SPI as tolerances become tighter and pitches smaller?*

Jean-Marc Peallat: Your question raises two challenges. First, with miniaturization and the need for more intelligence, PCBs are becoming very dense and therefore, pads are becoming smaller with less space in between. Traditional SPI uses "region of interest" (ROI) surrounding the pad. This one defines the area of the measurement, where the paste deposit should be, but also the location of the z-references. With size and pitch becoming smaller, the ROI is getting very small and very close to the pad. It becomes more and more difficult to define stable references that guarantee good measurement. Also, it may come across the ROI border. For traditional SPI, this results in a bridge defect. With a large field of view, our new SPI is not using ROI surrounding pads, and is not limited or impacted when pads are getting small and close to each other.

The other challenge concerns the calculation and the setup of parameters. Traditional SPI uses tolerances defined by a percentage of the theoretical volume or height. It is like one size fits all, tolerances for 0402 will be defined the same way than as 01005. Unfortunately, the physics of the printing process is not as simple as this. Effective transfer, the ratio between theoretical volume and real deposit, varies widely

NT

NathanTrotter

Trusted for Over 200 Years

High Quality Solder; Commodity Pricing

- Large availability of Leaded and Lead-Free Alloys
- Flexible Pricing Options
- Full Service Solder Recycling Program

Coatsville, PA
610-524-1440

Tin Technology & Refining

www.nathantrotter.com/products/solder
For solder recycling visit www.tintech.com

with the pad size. When pads are very small, the deposit is only a fraction of the volume of the aperture. With our next generation of SPI, we are using a parameter called area aperture ratio, which is the key for small pads. We set up the tolerances with this area ratio. For example, the 01005 is not the same as the 0402. So our tolerances for very small pads are adapted to these pads. Moreover, process variability is greater for smaller pads, or with smaller area aperture ratio.

> "With our next generation of SPI, we are using a parameter called area aperture ratio, which is the key for small pads."

Chong Choon Hee: To complete this, the traditional way is that the region of interest is around the small pads with dedicated z-references. If you use an ultra-large field of view, the system is able to take more references for measurement on a very large portion of the PCB. The compilation of all these references gives the exact z-plan of the PCB, including the warpage calculation. Therefore, when it comes down to small pad measurement, the accuracy of the system is enhanced.

Las Marias: *What about the volume of the solder?*

Peallat: When the deposits are becoming smaller, volume is key, because the shape of the deposits induces high variability of the process for very small pads. With poor transfer efficiency, shape is not consistent. Volume measurement depends on z-reference. That is where most of the SPI makers are struggling. With our technology with angled cameras, global z-referencement, the measurement of the volume is a lot more accurate than others.

Chong: As the specs get smaller and smaller, the transfer efficiency is getting more difficult to control. Right now, the trend is to have an SPI to inspect the solder volume.

Las Marias: *How does the different type of the components on the board impact the solder paste inspection process?*

Chong: For example, if you have a component and you have a volume of deposit on one side and the other side has lesser volume, there might be a pulling effect on the other side more, when you go through the reflow process.

Most likely this would result in a tombstone effect. It is important to have a co-planarity check on the volume to know whether the ratio between the pad 1 and pad 2 volumes are the same, so that they have the same pulling force. Because if there is a volume imbalance, there will most likely be a tombstoning effect.

Peallat: That is the same for the BGA. For example, when you have a 256-pad BGA, it is important to compare the volume in between the pads because if will have less volume just on one pad, it may create a void. It is very difficult to find out afterwards. You have to use X-ray, which is a very expensive technology.

If you are able to take that approach at the printing process, then you reduce the cost of repair or rework. With a big field of view, you see the whole BGA at the same time. All other SPIs are cropping the image around the unique pad and you never have the whole picture. That is the key advantage of our SPI.

Chong: Earlier, we were talking about the large field of view that we have, the more region of interest that we can locate for this referencing. If the pad field of view is smaller, it can only calculate a smaller region. If there is some difference or some disturbance in the surroundings, you may have a miscalculation.

Las Marias: *What do you think are the factors to consider when selecting an SPI solution?*

Peallat: From a measurement point of view, I will say volume measurement accuracy. If you are placing very small components, it's really the measurement of the volume. I will say the

We've been delivering quality circuit boards since 1985!

Since 1985, U.S. Circuit has been a premier supplier of both commercial and military Printed Circuit Boards in the United States. We have experienced continued and steady growth in a challenging economy through consistently high performance in on-time delivery, superior quality and an emphasis on the highest level of customer service. We are proud to be the Circuit Board Manufacturer of more than 400 growing companies.

Click to Download Our Brochure

COMMERCIAL • MILITARY • MEDICAL • BAREBOARD

U.S. CIRCUIT

U.S. Circuit, Incorporated
2071 Wineridge Place
Escondido, CA 92029
+1 (760) 489-1413
www.uscircuit.com

trend in SPI is to verify the repeatability of the system. That's key. The next step is the accuracy of the system.

You can be very repeatable, and be out by 10–20% of the volume. A 10% variation on the small pad is huge. Repeatability is the foundation, but the accuracy is key for the future.

Las Marias: *Do you think we have already approached the limit as to how the tolerances can be smaller?*

Peallat: We always say that we are almost close to the minimum that we can do. What is amazing with this industry is that they always find a smaller component. Today, we are just placing 01005, but they are already looking at the next generation. I would say we are closer and closer to the limit. We will see smaller and smaller components still to come.

Las Marias: *Apart from having a larger field of view, what other innovations do you think will help your customers address their SPI issues?*

Peallat: Right, larger field of view is one. The way we set up tolerance is key as well with our area aperture ratio, which is important because you are able to select your tolerances versus the size of the pad and also the transfer efficiency. That's key. It's not the hardware or technical innovation, it's more the way we manage the process. For me, that's the second key innovation.

Chong: For us, it's how we make use of the AAR, actually doing the running of the production we collect all the measurements of everything that passed. From there, we have a historical measurement from which we can tune the upper and lower limit of the tolerance from the production batch. Then, we can optimize the printer process accordingly.

Las Marias: *What can you say about the future for SPI?*

Peallat: I would say the future for SPI is not the future for inspection. SPI is the part of the global inspection of the line. The trend, for me, is more about using the SPI with the AOI and getting the value of the overall inspection process.

At Vi Technology, we are developing solutions that bundle both inspections. SPI is not just a step in the process, but also a step in the inspection of the board itself. We are developing software solutions that will help our customers by getting more information about the boards, not only about the SPI.

Chong: At the moment, SPI can do close loop. It will be able to work with multiple machines and share this information in a bigger database. We are working with our AOI to have a complete database so that we know the quality after printing, and how we manage the data to enhance the process in the SMT line. In the future, our database will work with other AOI equipment or coalesce all this information to provide a factory a more value-added solution. Right now, everyone is talking about the smart factory solution; we also envision being a part of that smart factory solution.

Las Marias: *SPI integrated with AOI. Wouldn't that be a bit overlapping?*

Peallat: Not at all; it's just two different steps that are complementary inspections. There are defects or trends that you see on the SPI which, combined with the information from the AOI, provides you a better view. If you close this loop, then you learn more about your process. This learning curve helps you to narrow your inspection criteria upfront and also at the last stages of the assembly. By learning all of this, you optimize your workforce around the line and your process, and you minimize you cost.

Las Marias: *Great. Thank you very much for your time.*

Peallat: Thank you. **SMT**

IPC IMPACT WASHINGTON, D.C 2016

IPC's IMPACT Washington, D.C. 2016: Who, What, Where, and Why

All photos in this section courtesy of IPC

by Patty Goldman
EDITOR

CEOs, CTOs, VPs, presidents, upper management, even engineers and worker bees: I call out to you to read this message. Your very life, at least your working life, may depend on it and I am truly not being dramatic.

I had the opportunity in mid-April to attend IPC's IMPACT 2016 conference in Washington, D.C., and it was quite a learning experience—and I didn't even get to most of the meetings. I'm not big on government, politics, our Congress, or probably 99% of the things that go on in our nation's capital (but I do love the museums). However, I learned that we have to work with what we have. So in this special IMPACT Washington, D.C. section, I have included nine interviews I conducted with people in our industry who can tell you in their own words what it was like to be involved and what they think of IMPACT—and whether it's worthwhile for you to attend.

A very serious and determined group of your peers—top management from IPC member companies representing PCB, EMS, equipment and materials suppliers—listened carefully to IPC's staff experts on the immediate, most pressing concerns of our industry. This year, three hot issues were chosen to follow up on with members of Congress and their staff. (It is best to limit the agenda to just a few items so as to not dilute the message nor distract the intended audience.)

The three major issues addressed at this IMPACT were:

- TSCA—The EPA's interpretation of the Toxic Substances Control Act makes it more difficult to recycle chemicals like copper etchant than to simply treat and dispose. IPC's argument: "We want to do the right thing and recycle as much as possible. Do you really want to discourage this?"
- Dept. of Labor—New proposed regulations would significantly raise the baseline salary of those who can be considered exempt from federal overtime pay regulations, effectively making more people eligible for overtime pay. Plus a formula is being proposed that would continue to raise this baseline on a yearly basis, pushing many salaried employees to become hourly, with attendant time card requirements.
- NNMI—the National Network for Manufacturing Innovation is a public-private partnership that draws on the resources of the federal government, local governments, universities, research institutes and industry to accelerate manufacturing innovation. IPC is urging full funding and long-term planning for the network.

Attendees were also asked to extend thanks for passing the now permanent R&D tax credit.

Since this time it appears that the DOL has issued the new regs affecting overtime pay, not the best of news for business. However, IPC is part of a coalition to continue to educate mem-

bers of Congress on the impact this will have on our industry.

On the other hand, as of May 23, IPC's language on by-products (TSCA item) has been included in the compromise being worked out between the House and Senate. This will indeed be a benefit to (mainly) PCB fabricators, keeping recycling practical and sensible. A vote is expected in House this week and the Senate possibly next week. This is a big win for our industry and is a direct result of efforts at IMPACT.

And now to the "Why." I think I'm like most of you—I abhor politics, politicians and all things that smack of them, which of course includes at least half the population of Washington, D.C. However, ya gotta do what ya gotta do, as they say. And as John Hasselmann says, "You are either at the table or on the menu," meaning that if we don't speak up and let Washington know what is important and vital to our industry, then we are at the mercy of whatever regulations suit their fancy—or are on the agenda of the myriad government agencies and/or special interest groups (think EPA, OSHA, Greenpeace, etc.).

It became obvious to me through conversations with the attendees that some of the congresspersons and their staff viewed corporations as the enemy, though others were more open-minded. It's so easy to look the other way (or vote the other way…) when a corporation or business is far away and seen as a big blob full of greedy people who don't want to share their wealth (magically produced, apparently). But when actually sitting down face to face, suddenly that abstract enemy entity becomes real, the company president becomes a real person and then he mentions the 10,000 or 1,000 or even 50–60 people that work for him (duh, voters!), and perspectives change.

And so it was and is. One thing I heard time and again was the importance, the criticality of a face-to-face meeting with one's representatives in Congress and/or a member of their staff. More than one attendee mentioned visits to their facilities by their representative and the very positive impression it made on some. A bonus for the CEO was that the tours sometimes became a town meeting for their employees, which is definitely a win-win.

All of this happens and happened at IMPACT Washington, D.C. 2016. Many of the participants had been to IMPACT several times before but some were newcomers. IPC's Washington staff carefully prepared the agenda, the talking points, so to speak, and thoroughly coached participants on how to approach various representatives. In one case, specific "hot buttons" were to be carefully avoided. This was serious, important business. I can't stress that enough—as important as that next piece of equipment or facility upgrade, in fact probably more important, considering the number of things in Washington working against staying in business.

So don't sit back and wait for someone else to go. Start thinking about and planning for IMPACT 2017, next April. There will be a new administration, new members in Congress, and more educating to be done. New bills will be proposed. Will they be pro-business? Will they help or hinder your business?

In the meantime, contact IPC's John Hasselmann and ask him to help set up a visit or tour with your representatives at your company. Bookmark and regularly check IPC's Government Relations page for updates on legislation and other info that could affect your company, along with the latest issue of the Global Advocacy Report. Do be proactive and take part. It's good for you, good for your business and good for our industry.

I hope you find this special section enlightening and inspiring. And thank you. **SMT**

Veteran IMPACT Washington, D.C. Attendee Matt Turpin on the Event's Benefits

I made contact with Matt Turpin, CEO of Zentech, before the first evening's dinner. We sat down to discuss what he hoped to gain by attending the event.

Patty Goldman: *Matt, I'd like to know what your expectations are of this meeting.*

Matt Turpin: The IPC does this every year, and I've been here the last five years for IPC. It's a great opportunity for the IPC and members of the IPC to meet with their local officials on Capitol Hill as well as other people on Capitol Hill and kind of deliver the message of what's important to IPC and the IPC vendor, whether it's in terms of RoHS compliance or conflict minerals, etc. This year it's Defense Department labor regulations and things like that. It's a good way for IPC to get its point across and to influence what happens on Capitol Hill.

Goldman: *How has that worked in the past?*

Turpin: It's worked out well. Some of the issues in the past have been the R&D tax credit and it looks like that is permanent at this point. I think some of the things relative to changing the narrative with conflict minerals is going slow, but it keeps the issue alive and shows that it's not the slam dunk that Dodd-Frank thought it was going to be. There are other issues where they have had some success, like the NNMI (Na-

Zentech's CEO Matt Turpin and VP John Vaughan during a preliminary meeting.

tional Network for Manufacturing Innovation) that the White House was big on, getting that properly funded and through Congress. That was a big push and it's been a big success.

Goldman: *Do you think this is directly attributed to IPC and its members being here in Washington?*

Turpin: Absolutely, yes. The IPC are tying up and spending member dollars doing this and they've got a local lobbying group that helps them with setting up. As part of that, they're making sure that they're getting back their bucks. Every year we talk about what are we going for, and what progress to aim for. The Government Affairs Committee orders routine board calls and committee calls to find out what we're working on, what outcomes to expect and what kind of progress.

"Every year we talk about what are we going for, and what progress to aim for. The Government Affairs Committee orders routine board calls and committee calls to find out what we're working on, what outcomes to expect and what kind of progress."

Goldman: *You've seen real progress?*

Turpin: There has been real progress. Absolutely. There's always something new.

Goldman: *I assume there is always something you have to worry about and work on.*

Turpin: Congress is always trying to come up with new ways…

Goldman: *New ways to mess it up [Laughs]. What do you particularly want to get out of this session?*

Turpin: The sessions have a number of different purposes. One is that it's good to get IPC members together. They tend to bring in CEOs for this event. It's good to do the networking and to find out what other people are faced with separate from the regulatory issues. It's also good for the CEOs that come to this to understand what the regulatory climate is like and what those issues are. Because I know when I first started coming, I really didn't understand all the issues that IPC was going to bat to Congress for in terms of representing their constituents within the IPC. I enjoy that aspect of it.

For me personally this year, I'm taking a more active role in helping get across the message in terms of the new Department of Labor regulations that are being proposed—as related to exempt and non-exempt status and raising the baseline salary level of those who can be considered non-exempt.

Goldman: *Anything else you would like to say about this?*

Turpin: I would say the only other thing is that anybody who is reading this article and is aware of the IPC, or some of the events the IPC does, whether it's APEX or whether it's IMPACT or another event, if they're a CEO, it's worthwhile to get involved and to help out. It helps them personally and it helps the industry as a whole.

Goldman: *Some people would probably say it's expensive to come here, like paying for the hotel, travel and that kind of stuff. How do you feel about the money end of that?*

Turpin: Everything has a cost. You could certainly argue that not participating also has a cost. I personally think that it's worthwhile and that on the whole the cost is definitely worth the benefit to people individually and to members as a whole.

Goldman: *Thank you, it's nice to talk with you.*

Turpin: Thank you. **SMT**

IPC's Hasselmann on IMPACT Washington, D.C. 2016: Why it Matters

I spoke with IPC's VP of Government Relations, John Hasselmann, immediately following the welcome dinner at IMPACT Washington, D.C. 2016. Among the topics we discussed was the importance of industry executives coming to Washington to present a collective message to policymakers.

Patty Goldman: *John, how was the dinner discussion tonight?*

John Hasselmann: Thank you, Patty, it was good. This was an opportunity for those who have been here before, and those who haven't, to network and get acquainted. We also had two speakers, Republican senior strategist Charlie Black and Democratic senior strategist Scott Pastrick, to talk about the current state of politics, not just in Washington but nationally. When you go into a meeting with a member of Congress or a policymaker, it's important to remember there are other things happening in the headlines, and they may be distracted by that. We need to be the advocate and say, "We need to talk to you about this even though there's a lot of other stuff going on out there."

Goldman: *Right, your goal is to keep them focused on your issues, and of course they've got a million other issues that they are also focused on.*

Hasselmann: Exactly. But the collective group here is a very powerful voice. When you have these executives coming into town, representing the electronics industry, and they are here together on the same page, not necessarily as competitors, that makes an impact.

IPC's John Hasselmann(L) and John Mitchell (R) pose with Congressman Bill Johnson (R-OH) with the capitol building in the background.

INTRODUCING THE AKILA ALL-IN-ONE DOFFS UNIT

For Use with MS2®

Why sell your dross to a recycler for a fraction of its true value? Ask us about the new compact Akila off-line solder recovery unit that, along with our MS2® Molten Solder Surfactant, takes the guesswork out of the solder recovery process.

- Easily convert dross into usable solder bars
- Fully enclosed machine
- Fume-free operation
- "Plug & Play" set up
- Small footprint

PK METAL

www.pkaymetal.com

Contact sales@pkaymetal.com for more information. (323) 585-5058

Goldman: *I think a lot of people don't understand that it can make a difference. They think, 'I'm just one little person, what can I possibly accomplish? The policymakers have a million other things on their minds. How do you make a difference?*

Hasselmann: Well, your voice as an individual is important, but when there is a collective voice that is unified, it's very important. I always point out that firefighters, nurses, policemen, teachers, etc., all have representation here.

Goldman: *Tremendous representation.*

Hasselmann: Exactly, and so should manufacturers and the electronics industry. We're just using the tools in our toolbox to represent the industry and be that collective voice. You mentioned that these members of Congress and their staff have all these issues to think about, and they're all over the map meeting with tons of people. IPC represents the whole supply chain. IPC members are in every continental state, and IPC represents almost a million workers. So when you tell that story to members of Congress, they'll listen. We're their constituents in almost every district. So if we come in and we're organized, if we have a succinct message and we have credible data, we can have a real impact.

"IPC members are in every continental state, and IPC represents almost a million workers. So when you tell that story to members of Congress, they'll listen."

Goldman: *What do you expect to accomplish over the next two days? What's the mission?*

Hasselmann: This year is a little different because we're in a presidential election year. Congress has very few legislative days this year, so we have a small window of opportunity to educate these policymakers, and to lay the groundwork for the coming year with a new President and a new Congress. A lot of our goals here are focused on meeting policymakers that may be even more influential next year. So we're meeting with surrogates from the presidential campaigns tomorrow; some are current members of Congress, others are former high-ranking officials who now represent these campaigns. But we're also solidifying and still educating a lot of these policymakers about IPC and who we are and what we stand for as an industry.

Goldman: *I can imagine what happens is you get some of them educated and then they move on. You've got to keep on educating.*

Hasselmann: That's why we're here, for the industry. The 48 hours that these executives come to town for IMPACT help us do our job for the rest of the year. That's not just in the U.S.—we're focused globally—but this particular event in D.C. is very important. It's exciting and I love doing it.

Goldman: *One of the things that I hope to accomplish is to get across to a lot of people that aren't here why there's good reason to be here.*

Hasselmann: Well, thank you for that. When I started here three years ago, one of my goals was to make sure that we had a stronger communications effort. What's that cliché? If a tree falls in the woods, and no one is around to hear it, does it make a sound? I think that is what happened in the past, like there was a small segment of the industry that knew the importance of what we were doing on the advocacy front, but it wasn't enough.

Now the industry is getting more involved—through the board of directors, through our government relations steering committee, and through various other groups. This is a good thing; we need to be at the table. You know, the other cliché is: If you're not at the table, you're on the menu!

I think because our industry is so connected to government, whether it's here or in the EU or

(L-R) John Hasselmann, Nilesh Naik and Bhawnesh Mathur on the terrace during a cocktail hour.

in China, the decisions really affect our members.

***Goldman:** We all know that from RoHS, right? Everybody got blindsided by that.*

Hasselmann: Yes. RoHS, conflict minerals, and there are others. When something is coming across a policymaker or regulator's desk, or there's some sort of proposed rule or a piece of legislation, I want someone to say, "Have we called IPC?" Because that's my goal: to be at the table. There are a lot of other trade associations, but we represent the whole supply chain, we're global, we do standards with a lot of professional development and educational training around these standards, and we do trade shows. That's very unique compared to some of the traditional trade associations that are based here in Washington, D.C. who just focus on advocacy. We've got a lot of real-world experience to back up our advocacy.

***Goldman:** A lot of people probably think, and I'm just trying to be the devil's advocate here, why not let IPC do all that? Why do I need to come to town? What is the importance of having companies or company executives come? What difference does it make?*

Hasselmann: The congressional staff see us all the time, but it's much different when a CEO takes the time to call a member of Congress, or to come to Washington or Brussels.

If a CEO comes in who's on the front line every day—making decisions that affect their employees, making decisions about whether they need to have a factory here or there, who is looking at their bottom line in terms of keeping revenues up and keeping their employees and customers happy—then members of Congress and policymakers are going to understand how important it is when they make a decision, and that they're going to impact us.

If CEOs take the time to be advocates, policymakers are going to listen because they have to make those decisions, too. We're always trying to get the decision makers together in the same room to come up with a plan and make it work. And that's the goal.

***Goldman:** And if your members are not there?*

Hasselmann: Then you can't complain.

***Goldman:** That is also true; how do you complain about something if you didn't engage or take part?*

Hasselmann: That is the beauty of this. I'm a firm believer in petitioning the government, and under our Constitution, we have the ability to do that. Without their trade associations, the nurses, firefighters, engineers and CEOs aren't going to come to D.C. all the time, so we try to bring that opportunity to our members.

Let me give you another example. We instituted a program called "Meet the Policymakers," in which we bring them into the factories and facilities and let them see all the innovation that is happening and to meet the workers. In the last few years, we have done over 30 or 40 site visits with members of Congress and our member companies around the U.S., and we have done some with our members in China with Chinese officials as well.

***Goldman:** So they get to meet the voters.*

Hasselmann: And the workers are their voters, exactly. It's a two-way street. When they see the factories they're just like, "Wow."

***Goldman:** They probably can't even conceive of what goes on in there either.*

Hasselmann: They're usually very interested, and when we get the employees together, they are thrilled to be able to engage with them. We'll set up meet-and-greets or town hall meetings and just let them ask whatever they want, and that brings the advocacy opportunity to the employees, too.

***Goldman:** If any IPC members called up and said, "Hey, can you arrange something with my congressman and help me get them into our factory?" That can actually happen?*

Hasselmann: Yes, we will do our best, no question. That's why we're here. But it's not all happening here, it's really happening out there—all the work, the innovation, the R&D, and the jobs. We're just trying to make sure that policymakers are educated and that the legal, regulatory and legislative environment is conducive to continuing to grow our industry.

We want to be at the table. We want to be able to debate the pros and cons of anything that may have a negative or positive impact on the industry. We want to be a stakeholder, and we're going to be that voice. We're making a lot of strides, and this event is really helping us to do that.

> "We're making a lot of strides, and this event is really helping us to do that."

***Goldman:** How many congressmen will you guys be seeing over the next few days?*

Hasselmann: It's a two-track approach. As a group, we have about ten key meetings with members of Congress and administration officials. But then we also arrange individual meetings for our attendees with their representatives where they have facilities. So they can go in and establish those relationships and we facilitate that, and that allows us to spread the word even more.

We probably have 30 to 40 of those individual meetings, where it's an opportunity for that CEO or that executive to talk about very specific issues of concern to them and whatever they want to raise. The policymaker wants to know, "What's keeping them up at night? What can I do to be helpful? What do you need? What can I do to cut some red tape for you?" They want to do that in a heartbeat for businesses in their district.

This is the opportunity for our members to talk about those very local issues, but also we're there to talk about some of the broader issues where we can come back and work with them

because it's an opportunity to help their constituents. That's where we're trying to connect dots.

***Goldman:** Connecting dots is a good way of saying it.*

Hasselmann: I get excited about this, as you can tell. [Laughs]

***Goldman:** Well, like you said, this event is only one part of your job overall, but it's going to make your job easier.*

John Vaughan of Zentech told me he's been coming for six years, and in the first few years he didn't really notice much, and then he started seeing stuff happen, like the things that you guys were advocating becoming some real success stories. Things don't happen right away though, it takes time. These guys can't just come here and then expect next month that something's going to happen as a direct result. It doesn't happen that fast.

> "We try to help facilitate that conversation and try to offer policy solutions that help the industry."

Hasselmann: I think they get it. For the most part it is like long-term planning, like when these CEOs have to look ahead with a five-year plan. There's a new Congress here every two years, they roll out an agenda, and we have an agenda, too. Where do they mesh? We try to help facilitate that conversation and try to offer policy solutions that help the industry.

***Goldman:** I was just thinking there are some very loud voices here in Washington these days. You probably can't get louder than them, so you just have to be that voice of reason in there.*

Hasselmann: Oh yes, and I think that's what we are. Our issues are bipartisan. We have great market research on the industry, we have data, and we have information that backs up what we are advocating for. For example, when we're talking about some of these environmental issues, we have data that explains why we think this approach is better than that approach. Policymakers and their staff thrive on all that. Instead of just rhetoric that people get so wrapped up in, we try to be that voice of reason because we can bring in the data that supports our message, and that's my role: To make sure it's bipartisan and that we have the meat behind our message.

***Goldman:** And does that get through?*

Hasselmann: It does. The serious policymakers appreciate it, and they'll come back to us. With all the engineers and the technical and compliance people involved with IPC, we can go and ask them, 'How would this regulation affect you?' Then we have the execs and CEOs to weigh in and say, 'Okay, we think this will be good for the industry, let's go.' I like that we can draw on the expertise of all segments of a company.

***Goldman:** I noticed, too, that a lot of the issues on your agenda, like conflict minerals, are not just challenges in our industry. It's not like we're saying do this for me, me, me. IPC's agenda is good for business, for the industry, and for the country.*

Hasselmann: Exactly, it's even more powerful than just us. We're working to be seen as a leader among the various industry groups. It's very important to form and be part of broader coalitions of industry groups to bring more voices to the table. Coalition building is critical, and we have a lot of organizations that we work with, here and abroad, depending on the issue. We have members of IPC that are members of other organizations, and they want us to work together and achieve more synergy. They understand that the collective voice among different industries is just as important for getting something done.

***Goldman:** John, I really appreciate you taking time to speak with me today. I wish you the best in the next 48 hours.*

Hasselmann: Thank you, Patty. **SMT**

STI Electronics Participating in IPC's IMPACT Washington, D.C.

Dave Raby is president and CEO of STI Electronics and an eager participant at this year's IMPACT Washington, D.C. event. We talked early on the first day of the event, before a heavily scheduled day for the attendees.

Patty Goldman: *Dave, what are your thoughts about this year's IMPACT event, overall?*

Dave Raby: I'm excited. I came to this event for the first time last year and have been looking forward all year to coming back. It's really great to meet the other people in here. It's all senior executives from companies in our industry from all over the country. Whenever senior executives from the same industry get together, we usually find out we share many of the same concerns and that is true with this group. Washington, D.C. is also a foreign world to most of us, and it's great to get up here and see what's going on here and see what our representatives are thinking. What is even better is, through the efforts of IPC, we can actually have an influence on some things and can give them our opinions. As Americans, it is what we're supposed to do but I don't think most of us do anything unless there's a group like this supporting it and organizing it.

Dave Raby

Goldman: *Yeah, there's some reluctance.*

Raby: It's hard to come up here as a lone wolf and just say, "Hey, I want to support this bill."

Goldman: *First of all, how would you get to see anybody?*

Raby: That's a good question but also how would you even know what bill was out there? IPC does a great job of scheduling visits with key people from all over the country and also with the representatives from my states. STI has employees in four states and tomorrow I'll be visiting both senators from one of the states and our U.S. representative from two others. IPC staff does a great job of keeping us informed on the legislation that is at various levels of the process and gives us a very good idea on where the people we are meeting with stand. We can then express our views and let them know we appreciate their support and encourage more support, or let them know why we see their particular stance as a problem. My opinion may or may not change their opinion but I've been impressed with how they really will listen and consider how a certain piece of legislation will affect my company and their constituents. It was fascinating to me when I came last year just to see how the government works. We often complain about how it doesn't work, but in re-

OML

OPEN MANUFACTURING LANGUAGE

The standard for the Internet of Manufacturing (IoM) has arrived!

The Open Manufacturing Language (OML) is a real-time communication standard for PCBA manufacturing that defines the interconnectivity of assembly production processes and enterprise IT systems.

For the first time, IT teams, solution providers, and equipment providers can easily integrate shop-floor data to create manufacturing execution solutions based on a single, normalized, vendor-neutral communication interface.

Take part in shaping the future!

Become a member of the OML Community where PCB Assembly industry professionals have FREE full access to the OML specification, white papers written by industry experts, and a place to share ideas in our community forum.

Visit http://www.omlcommunity.com and join the community!

ality, it has done a pretty good job over the last 240 years or so.

Most of the legislation starts for a good reason. I compare it to my shower thoughts at home. I've come up with what seemed like some of the world's greatest solution or idea as I'm getting ready to go to work in the morning. I'm so excited when I can get to the office and tell my staff about them. Most of the time, as soon as I start saying them out loud I realize that it may actually be the dumbest idea I've ever had. The legislation starts for a good reason and seems like a good idea at the time and, by the time you get through the process that they go through, there's hopefully people like us that have come in to say, "Yeah, this really is a good idea but could you add this to it?" Or, "We really don't like this. Have you thought of what this would do to business? While it seems like a good idea overall, maybe if you took out this line, then it really would do what you want." Right now, you've got the unintended consequences that are going to happen.

***Goldman:** You really have to think about the consequences.*

Raby: Exactly, and no offense to the lawmakers but they are trying to improve our lives and businesses but usually have no idea how they work and some of the unintended side effects some of their laws can have. Just making up an example here but there could be something that will save my company $10,000 but causes me to have to hire a full time person just to fill out the paperwork. That didn't save me $10,000. That put a productive person out of work. That's a big reason for being here this week.

Just as a citizen, it's frustrating to watch the democrats versus the republicans, republicans versus democrats. If I have an idea, no matter how good it is, you're not going to like it, and vice versa. It's nice to get up here and see they actually do talk to each other (or at least many of them do), and there are some things that they cooperate on.

***Goldman:** There are things behind the news that happen.*

Raby: Right. Apparently, talking to each other doesn't make the news.

***Goldman:** That's for sure; it's not as exciting. Do you think you'll come next year? Do you see this as a good event to attend?*

Raby: Yes I do. If it's anything like last year—and from the schedule we have, it looks like it will be—I'll be back next year. I also want to encourage other business leaders and owners to do the same. It costs two or three days of time plus your travel expenses and can have a direct benefit on your company's and your industry's future as well as your country's future. Plus, it is interesting, educational and we have some fun, too.

***Goldman:** There's a pretty heavy schedule this year, from what I understand.*

Raby: We're hearing from four different presidential campaigns this morning, which is something that of course we didn't have last year. It may not be a kind way of saying it, but it's a straight-from-the-horse's-mouth type of thing that you don't get sitting at home on the couch. It'll be interesting to hear how that goes and we get to question each candidate on their thoughts on keeping (or making) American manufacturing competitive with the rest of the world. Then we're also meeting with several congressmen and it will be great to get their perspective on things, and also give them our perspective.

One of the things I'm talking about is the NNMI, the National Network for Manufacturing Innovation. We are meeting today with Senator Orrin Hatch, who's the head of the Senate Committee on Finance. Senator Hatch is number three in line to be president, as far as a succession plan.

***Goldman:** That's not somebody you can just knock on his door and talk to.*

Raby: I'm from a little town in Alabama. That's not somebody I'm used to talking to. I'm a little nervous about that, but I also know from last year, IPC will get me through it.

Congressional meeting.

Goldman: *Right, they keep you well-informed of all the important stuff.*

Raby: They've educated me, but they're also going to be sitting next to me. If I start to stumble, they'll steer me in the right direction, or if the senator asks a question that I don't know the answer to, I just look over and they'll help to answer it.

Goldman: *And John Hasselmann said they have all the data ready as well. So they'll have a lot of answers.*

Raby: John and his entire team do a fantastic job.

Goldman: *Okay. It's probably about time to head in to the meeting. Well thanks so much for your time, Dave.*

Raby: Thanks Patty, I appreciate you being here, because publicizing this can do nothing but help it.

Goldman: *That's what I hope to see happen also.*

Raby: IPC does a great job on this. There's nobody else up here looking out for us in the electronics industry. **SMT**

Creation Technologies on IMPACT Washington, D.C. 2016

Meeting with congressional leadership, peers, competitors—it's all valuable, according to Bhawnesh Mathur, president and CEO of Creation Technologies. Here are his thoughts on the benefits of the event after the close of the first day of IMPACT Washington D.C.

Patty Goldman: *Bhawnesh, tell me about your day here at IMPACT. How was it for you?*

Bhawnesh Mathur: There are so many good things that come out of these annual IMPACT events. First of all, I feel like it's always important to partner with our political leaders. Their interests and our interests are the same. We want to see the economy grow. We want to find jobs. We want to move technology forward. I think when people have aligned interests, they should meet, talk and find ways of working together. It's always good to do that.

The specific issues that we talked about today will help us get better aligned. It's not an overnight thing, but for the last several years that we've been coming here we've learned to speak with each other. We've developed credibility with each other. We've actually shared accomplishments with each other that we can celebrate and so on. Every time we come here I think we move the ball forward and that's a good thing.

I also enjoy meeting all of my peers, and suppliers and competitors. I think it's pretty cool that we can be competitors in the morning and come here and work together on behalf of the industry, and that benefits everybody. Obviously we're trying to win against each other on certain days, but if our industry doesn't win, we don't win. I feel a sense of urgency to do everything we can to help the electronics and manufacturing industries grow and get some of the benefits.

Goldman: *How many years have you been coming to IMPACT?*

Mathur: I think this is my fourth or fifth year.

Goldman: *Do you recall what prompted you to attend the first time?*

Mathur: I do remember. I was in Denver, Colorado then, and I thought if I met my politicians, my senators, and my congress people, I could do more with them in Denver. I came and advocated on behalf of my company at that time. I told them, if you help me I can create jobs here, and I asked them if they could help my company get economic relief in this or that way. And we started a dialogue, and it actually helped. It didn't solve all of our problems of course, but it was enough to get me started. I also began to network with their connections and that helped too. So I definitely benefited from attending the first time.

Goldman: *So you came back the second time?*

Mathur: I came back, and now I manage a company that's in 12 different locations around the world, with nine in the U.S. and Canada. So I find benefit. Representative Paul Ryan visited our Milwaukee manufacturing facility the week before he got named to Speaker of the House, and we told him we could use some help with an OEM that we wanted to partner with. He said, "I visited your facility, I spent the whole day here and I liked it all. I'll talk to those guys." And I don't know if that played a part, but we did end up winning the business in the end.

Goldman: *Have you had congressmen come and visit in any of your other locations?*

Mathur: Yes, we like to invite them—there are lots of benefits to that. First, they get to understand who we are and a little about our industry. One of the benefits, which is really at the top of our list now, is when people like Mr. Ryan visit, is giving our people a chance to meet local leadership and ask questions at a town hall meeting. They likely would not otherwise get this chance. We may have anywhere between 200–500 people in a Creation business unit and this is a way for us to bring the community to our team. That's another benefit of coming to work with our company.

Goldman: *So actually getting a congressman to your facility came about because of coming to IMPACT?*

Mathur: Yes, when we meet these fellows, we're always trying to figure out the next step, and one of the next steps is to say, "Why don't you come visit us?" That actually became an initiative that John Hasselmann and his team run now. We measure how many visits we have, we have targets and a process. We've invited Prime Minister Justin Trudeau of Canada, we've invited President Obama, and he wrote us a letter back saying "No thank you, but Penny Pritzker, the Secretary of Commerce, will be available to come meet with you."

Goldman: *Nice!*

Mathur: Yes. We're finding that we get some recognition, our people can meet the leadership, and that's a good thing.

Goldman: *And conversely, your congressman learns a little bit about you and about the industry.*

Mathur: Absolutely, that's right. Many of them completely understand what we're doing and some of them have no idea what we do. It becomes especially clear when they take a tour, though, because it's hard to explain exactly what we do in a conference setting like this.

Goldman: *A picture is worth a thousand words, but I'm sure a tour is worth a gazillion words.*

Mathur: For sure. When you see 200–300 people making state-of-the-art medical equipment, which is an area Creation specializes in, and you see an incubator we made where a baby is brought after she is born, or a CAT scan machine that we designed, or an ultrasound machine that we worked on, that's very powerful.

Goldman: *It sounds like lots of good things happened today then.*

Mathur: Absolutely.

Goldman: *How many congressmen did you meet with?*

Mathur: Formally, three.

Goldman: *Of course everybody has their own three that they meet, I guess.*

Mathur: Everyone has their own three. I feel like because I've been coming to IMPACT for a while, I try to not be as vocal. I think others need to participate and feel like they've been a part of it. But our message is getting across, and I certainly believe the view that it's the authentic person who makes the biggest difference. You can't just hire someone to come in your place and represent whatever it is you believe in.

Goldman: *Excellent! Any final thoughts?*

Mathur: I think politics are always dynamic. One of the things I'm learning is that while we've done all of this, you can't ever stop or let your foot off of the accelerator. If anything, I think we need to do more of it in more places. Creation Technologies is a global organization with global customers, so there's Europe, Asia and other places where we can get involved and make a difference.

Goldman: *Bhawnesh, thank you so much.*

Mathur: Thank you. **SMT**

IPC is One Thing, but Constituents are Quite Another

As I was unable to attend the actual meetings with the representatives, I wanted to catch the thoughts of those who did. I've known Nilesh Naik, CEO of Eagle Circuits in Dallas, Texas, for many years. We sat down for a chat after the first day's events.

Patty Goldman: *Nilesh, tell me how things went on the first day of IMPACT.*

Nilesh Naik: It's been a great day. Unfortunately I've missed Capitol Hill Day, or IMPACT, for the last two years, and was excited and glad to make it this year. It's just a great opportunity to visit with senators and congressmen. The exciting thing is they passed the R&D tax bill last year, so it was a good chance to finally say thank you. Interestingly, we've been on the Hill for the last seven or eight years asking for it to become a permanent tax credit. It's neat to see the process come to a complete end, and actually have a permanent tax credit. So I'm excited from that point of view.

It's quite amazing; every time we come to Capitol Hill, whether it's the legislative system, the congressmen or the senators, they do genuinely want to listen and hear from you. It's always great to present your perspective and situation when you get a chance to talk to them. They say, "These are my constituents," and they do listen.

Goldman: *So it's been worthwhile for you to attend?*

Naik: Absolutely worthwhile.

Goldman: *You get to meet with your own congressman tomorrow, is that right?*

Naik: Yes, and I'm looking forward to that. We've been fortunate enough to have Congressman Johnson come to our factory, so that was just even more fun. The Congressman has actually seen the manufacturing process. He's touched the product and talked to our people.

Goldman: *They actually begin to understand…*

Naik: They absolutely understand. Surprisingly, they do know a lot of what's going on. The challenge for them is they've got numerous other issues that are also going on. They're looking at a bigger picture and still saying, "Hey, how do we solve your problem?"

Goldman: *Your congressman knows what's going on. He's been to your factory, but what about all those congressmen who represent those companies that are not here? What do they know about your business or circuit boards and our industry?*

Naik: It's a great question, and one of the things that always concerns me. It is disappointing—that we don't get more of our member companies out here. This is the one time of year where you get a chance to meet your congressman, and you get to talk about your industry. It's amazing, they're eager to listen and learn, but you've got to come tell them. If you don't, you miss out on it. You've got to build those relationships.

Goldman: *It's a great opportunity to get your voice heard.*

Naik: Absolutely. They hear you, and today there were very positive responses with all the congressmen and senators that we spoke with. They're all for it. They understand what we're talking about. Heck, they want manufacturing

back in the United States. It's not like they want to get rid of it. When you can point out certain things that are hurting U.S. manufacturers, they're going to listen and they're going to see what they can do to fix that.

***Goldman:** That's good.*

Naik: We had a great talk about that, and we had a great talk on TSCA. One very concerning topic that will affect all the PWB guys is the changes in regulations that are being proposed by the EPA. They're trying to get us to document all recycled material. The way the EPA has proposed the regulation, they are actually disincentivizing the PCB shop to recycle product. It would be cheaper for us to go send our waste streams to the landfills. This is totally counterproductive.

Obviously, recycling is the best way to do things, but the way it's being written and the way it's being proposed by the regulators right now will actually be a disadvantage. Again, this is a great conversation to share with the senators and congressman and say, "Wait a minute, that doesn't make sense. Why are we doing this?"

***Goldman:** How far along is that? How critical is it to get that message out right now?*

Naik: I believe it's something that is going to come up within the next few months. The EPA is close to the end of writing their regulations. It's something that all the board shops need to be aware of or else we will just be creating more work.

***Goldman:** They all should at least be calling their congressmen right now.*

Naik: Without a doubt. The challenge is exactly that. If we don't bring it to their attention, then they don't know how to fix it. Then all of the sudden you've got the EPA who just goes and creates all this stuff. If you want change, you have to be a part of the change.

***Goldman:** You've got to have the answers for them.*

Naik: Absolutely. From that point on, the IPC does a fantastic job, with Fern Abrams and the whole team that John Hasselmann has here. They've got a hand on all the issues affecting us as an industry, so they can give you that feedback and answers. But at the end of the day, the congressmen don't want to listen to IPC staff. They want to hear from their constituents. The constituents have to show up. Actually, they can even just make a call. Even if they call the congressman, that's a huge thing. One interesting thing we learned today was even just a hundred call-ins about a particular topic will actually change the needle and may even change the direction a congressman would go. We do have to get active if we want to have change.

***Goldman:** I imagine face-to-faces are even more persuasive. If a number of people get in touch with the representative in their district and get a face to face with them, that's got to have even more sway.*

Naik: A face to face is totally worth it. If a congressman is having a town hall meeting in your district and you go see them, it makes a huge difference. Especially if they hear the same message a couple of times, they are going to come back to D.C. and talk to the legislative assistant and say, "Okay, this is what I heard back in the district, so what are we doing about this?" Your voice is heard, but you've got to speak up.

***Goldman:** What are you expecting for tomorrow?*

Naik: I'm looking forward to seeing Congressman Johnson tomorrow and possibly Senator Cornyn. I'm looking forward to both of those meetings. Again, just see if we can further the message and continue asking for their support.

***Goldman:** Excellent. Thank you so much, Nilesh.*

Naik: You're welcome. **SMT**

The Many Reasons why People Attend IMPACT Washington D.C.

At the end of Day One, spirits were high within the IMPACT group. They had heard from the top presidential campaigns, met with key representatives on the top three issues, and were now relaxing on a private terrace with a great view of the U.S. Capitol building. Everyone was looking forward to a pleasant awards dinner. I had the opportunity to converse with Tom Edman, CEO of TTM; Ed Moll, VP of Viscom; and Tim Redfern of Redfern Associates, who was at the event representing Insulectro. Not able to convince them to give up their spots on the terrace, the four of us had a great chat on the spot.

Patty Goldman: *Tom, why don't you begin by telling me how your day was? What did you learn?*

Tom Edman: It was a good day. Number one, I thought the morning was very interesting. While we didn't get a chance to hear from the presidential candidates, we had a chance to hear from their proxies. I thought they did an excellent job of representing the candidates, and to hear it in person and hear some of the positions that they've taken. That was very interesting.

Tim Redfern: I also found that interesting. I have never experienced anything like that before, talking to proxies, and getting the experience of both the democratic and the republican campaigns was an interesting comparison. I think the importance of selecting the right proxies came through today.

Goldman: *That's good. How about your afternoon sessions?*

Redfern: In the afternoon we actually came over to Capitol Hill and had meetings with four different congressmen and senators. We got a

(L-R) Tom Edman, Tim Redfern, Rick Lies, and Ed Moll.

chance to really get up close and personal and actually have real dialogue, which I thought was a great opportunity and in some cases very timely for what's happening right now.

***Goldman:** Is this your first time to IMPACT?*

Redfern: Yes, it is my first experience here.

***Goldman:** Why did you decide to attend?*


Tim Redfern

Redfern: Mikel Williams inspired me, and he's not even here! He's been encouraging me for the last four or five years to participate and get involved in this event. I talked with him a couple weeks ago and told him I was coming and he said, "I'm not going to be there this year." He's always attended and has been very involved on the IPC Government Relations committee. But I'm really happy to be here. It's a great opportunity to see how this side of the business works.

***Goldman:** Ed, is this your first time here?*

Ed Moll: Yes, it is.

***Goldman:** And what are your impressions?*


Ed Moll

Moll: I've loved it. This has been very informative for me. I've never done anything like this and this is the first time I've had the opportunity. I'm here because my boss, Carsten Salewski, couldn't make it because of business travel, so he asked me to attend for him and I jumped at the opportunity.

***Goldman:** Even from the very start this morning there was some serious discussion about which topics to discuss and the agenda—and that was impressive in itself. Some of us don't always pay attention to this stuff, and there's a lot that goes on behind the scenes that affects everybody. That was surprising to me.*

Moll: The entire day was eye-opening for me. Meeting with Congressmen or their staff makes you feel as though you are making a contribution to our industry. I look forward to the meetings scheduled for tomorrow. It's been more than interesting being here.


Tom Edman

Edman: I was one of the spokespeople for the TSCA reform and I think the timing couldn't have been better. I've come to Capitol Hill not with IPC, but with other organizations before, and we never had this kind of timing. We have a bill that has been passed by the house and the senate but was then sent to conference. So we had a chance to directly impact some of the wording, we hope, that will go into the final bill. I think that is an unusual opportunity and definitely an opportunity on something that we feel is very important to the industry. From that standpoint I think we had a very good day.

***Goldman:** What do you plan to do tomorrow?*

Edman: Tomorrow we're off to the individual meetings with each of our local representatives where we expect to have more focused discussions about where we operate and more local issues. Today we were focused on broader industry issues that IPC had set up in advance for us to discuss.

***Goldman:** Thanks very much guys.*

Moll: See you next year. **SMT**

Making Connections at IMPACT Washington, D.C. 2016

Rick Lies is CEO of Chemcut, an equipment supplier to the PCB industry. He's a veteran of the IMPACT Washington, D.C. events and shared his experiences at this year's gathering at the close of Day One.

Patty Goldman: *Rick, we're at end of Day One here at IMPACT. I've been busy getting everybody's thoughts on the day. How did things go for you today?*

Rick Lies: It was definitely a full day. As usual, IPC has done a great job of putting together a group of speakers and meetings for us. It's always interesting and informative to come to Washington D.C. and see what our legislators and our representatives are doing and what their thought patterns are.

I've been doing this for four or five years, but this year was a little bit different because we had the surrogates from the different candidates come in. I don't think they changed my mind on any of them, but they probably reinforced some things that I already was thinking. They definitely reinforced the fact that things are not going completely smooth for either party.

With the meetings that we had, IPC did a good job with the talking points, which are important. The companies here mainly come to support IPC and the customers—the people we sell to. A lot of the things that they were going after, like the Toxic Substance Control Act

Rick Lies

Congressman Bill Johnson (R-OH) addresses the IMPACT 2016 participants after dinner.

(TSCA) don't directly relate to us, but if it affects their business it ultimately affects our business. So for them to be successful we need to modernize it. Other issues like the R&D tax credits are important to us, where they need to increase the tax credit from 14 to 20%.

Goldman: ***What would you say to somebody who is thinking about coming to IMPACT next year—or not thinking about it?***

Lies: I definitely think it's worthwhile to come, if for nothing other than to support the industry. Again, IPC does a great job of identifying and developing an action plan for the issues that are important for the industry, our survival, and our growth in the future.

Goldman: ***Have you seen results over the five years you've been attending?***

Lies: Definitely, there have been good results. The R&D tax credit was made permanent, and TSCA is seeing legislation moving forward that will modernize it.

Goldman: *Fantastic.*

Lies: What's interesting is I think people need to come up here to reinforce these things with their own representatives. It has helped me develop a relationship with our local Congressional Representatives and Senators.

Goldman: ***So you actually could invite them into your factory.***

Lies: In 2015, we had our Congressman Glenn Thompson, who represents our district, take a tour of our facility in State College. Since then he has actually called us to set up another visit.

Goldman: ***That's great. What are your expectations for Day 2?***

Lies: The first day is always sitting down and everything is very formal, you have your meetings you go to. Tomorrow is more the individuals to meet their own representatives and senators.

Goldman: ***Now will you meet anyone new tomorrow, like your senator?***

Lies: I'll meet with my representative, Congressman Thompson. We're not a large community so he knows many of the people that work for Chemcut.

Goldman: ***But still, there's nothing like having that relationship. Thanks so much, Rick.***

Lies: No problem. Thank you, Patty. **SMT**

A First-Timer's Perspective on IMPACT Washington, D.C. 2016

I met with Faisal Pandit, president of Panasonic Factory Solutions Company in Illinois, for a quick chat about his experience at IMPACT Washington D.C. 2016.

Patty Goldman: *Faisal, how was your day? I'm curious about what you got out of it and what your experience was like.*

Faisal Pandit: This is my first year at IMPACT and I've got to tell you it has been very exciting and a great opportunity for me to take in a lot of valuable information. I've been in the electronics manufacturing industry for more than 25 years and there are certain serious impediments affecting the growth of this industry in North America. It's important for us to take a very proactive stance in trying to remove those impediments if we are to ever drive any meaningful organic growth.

So an opportunity to interact with our leaders who make decisions for us is a wonderful thing. IPC put some serious issues on the table and the congressmen listened. Ultimately, when you tie the growth of manufacturing—or the importance of manufacturing—to job creation, that resonates well with politicians.

Goldman: *Somehow they just don't get that until you tell them.*

Pandit: Right. They may not necessarily link it otherwise. I think that worked out quite well in terms of communicating the message and getting that going, but as somebody said earlier, in Washington things move at an incremental pace. There are no revolutions or anything major right away.

Goldman: *Right—you are not going to see anything tomorrow.*

Pandit: It's a matter of continuing to raise your voice and having these interactions from time to time, but overall it was a great day.

Goldman: *So what made you decide to come this year?*


Faisal Pandit

Pandit: I'm on the supply side of the industry and in the past I didn't really think about attending. But this year was different because I'm personally a big advocate of reviving manufacturing in North America, and we as a company are trying to work with some private and public partnerships to help enhance the manufacturing skillset in North America, which I consider to be a major impediment to the growth here.

I know a lot of people are focusing on STEM programs and things like that. We are in the early stages of trying to put a focus on the manufacturing skillset within community colleges, within high school programs and things like that. We are trying to see what we can do as a company, and I think it would require some level of support from various levels of the political establishment. By coming this year, I wanted to get a sense of what people are talking about in terms of political issues and get an understanding of the process and how we can leverage these contacts and build up on what IPC is doing.

Goldman: *Did you meet your objectives or your expectations?*

Pandit: Absolutely. I learned a lot about what IPC is doing on the regulatory side and on the political establishment contact point of view, and I think it's very positive. It did meet my objectives. I think IPC has strengthened its focus on becoming a very powerful voice for the industry.

Goldman: *That's excellent. Would you return next year?*

Pandit: I look forward to being here again.

Goldman: *Thank you so much, Faisal.*

Pandit: Thank you. **SMT**

Shaping the Issues that Matter Most at IMPACT Washington, D.C. 2016

IMPACT Washington, D.C. 2016 encompassed two intensive days of meetings for the participants. I tried to catch their thoughts at various stages of the event. Optimum Design Associates' VP and General Manager Everett Frank was happy to provide some concluding thoughts.

Patty Goldman: *Everett, here on this final day of IMPACT Washington D.C., what are your overall impressions of your time here and how it's been for you?*

Everett Frank: It's been wonderful. I think this is my fourth year attending IMPACT.

Goldman: *It seems everybody comes back to IMPACT.*

Frank: Yes, most people do. There's a very high return rate. It's a great opportunity to connect with what's going on in the industry, particularly from the regulatory perspective. We spend a good portion of time advocating for industry issues, both with departments of the administrations and with members, and so it's a very good opportunity to impact those issues.

Goldman: *IMPACT is a good name for the whole thing.*

Frank: Exactly—you can't overuse that term.

Goldman: *How were your meetings yesterday?*

Frank: Really good. It's always interesting and fascinating to look at the different perspectives and where they're coming from. As an example, we were on the labor issue quite a bit yesterday, and trying to relate our issues differently versus republicans and democrats and how we present our business needs in a way that resonates with both sides.

Everett Frank

Goldman: Do you feel there were accomplishments yesterday?

Frank: Well, I guess the proof is in the pudding, but they were well received. We certainly consistently hear back from the members that these kinds of conversations are productive to them and that they're impacted by them. They listen, I think. You always kind of wonder how much, but I do think it moves the needle and that us being here reinforces the industry's messages.

Goldman: *How about this TSCA issue they keep talking about?*

Frank: You know, honestly, it's not a direct impact to my business.

Goldman: *Just to your customers or suppliers?*

Frank: It does impact my suppliers. In our industry, PCB manufacturers who buy the chemicals are the ones who are impacted. So my supply chain is impacted. I buy from those guys, but I don't sell to them.

Goldman: *That's important too.*

Frank: It's very foundational. I mean really everything in electronics rests on what the PCB fabrication guys do. I was joking with some of them yesterday that represent those interests that they needed to speak more highly of what their companies do when they introduce themselves. Because some of these companies in the room are just foundational to the technology in our country and in our world. The things that the chemical and PCB fabrication guys do are just so important. Like Tom's company, TTM, is the largest company in the world

of PCBs, which are literally the backbone of everything that happens. Then companies like Isola are the raw goods material suppliers to them. Isola is the most innovative company in PCB materials in the world. It's not just some company called Isola. These companies are just crucial to technology development.

Goldman: *You probably want to point to your congressman's little phone and point out they wouldn't have that without us.*

Frank: It's really true. To the congressmen and women's credit, the blur of issues and the range of issues that they face is just stunning. By and large they engage, they understand, and they know the issues we come in and bring up. It's not some obscure thing they have never heard of.

Goldman: *Anything else you'd like to add about what's been happening the last couple of days? My impression is that everybody here is very focused. This is no lark down here, this is serious business, and very important business at that.*

Frank: Yes, our government relations staff at IPC and the Prime Policy group just do an outstanding job of shaping the issues. Obviously the industry can sort of highlight the main issues, but if you were to look across the range of things, there are hundreds of things going on in the regulatory world that affect our industry, but they do a great job of shaping and funneling us to the critical things. There's an interplay there, too, between what's important to the industry and what we can actually have an impact on. For instance on TSCA, that's an element where we can actually make an impact on certain things.

Goldman: *Fight the battles that you can fight and win.*

Frank: These things are just the craziest things too, like in the past there's been language that gets inserted into various types of regulations that might take years to settle. For example, we spent a couple of years on very subtle language related to how PCBs are delineated on the U.S. Munitions List, which controls ITAR classifications. Two or three years over literally one sentence. But to our industry and to our PCB manufacturers in particular, that language was very, very important. But unless we tell Congress, how would they know?

Isola's president Jeff Waters (L) with Everett Frank, VP/general manager (R) at Optimum Design Associates.

Again, our government relations team and Prime Policy put us in the right meetings at the right times. We were in meetings over the years with not only members and committee staff, but with staffers at DoD and in the White House who were specifically controlling that language. The opportunity to do that is impressive. If you step back and think about it, it's impressive that IPC is coordinating an effort to target us at issues that can really make a difference in our industry.

Goldman: *It impacts the everyday workings of your companies.*

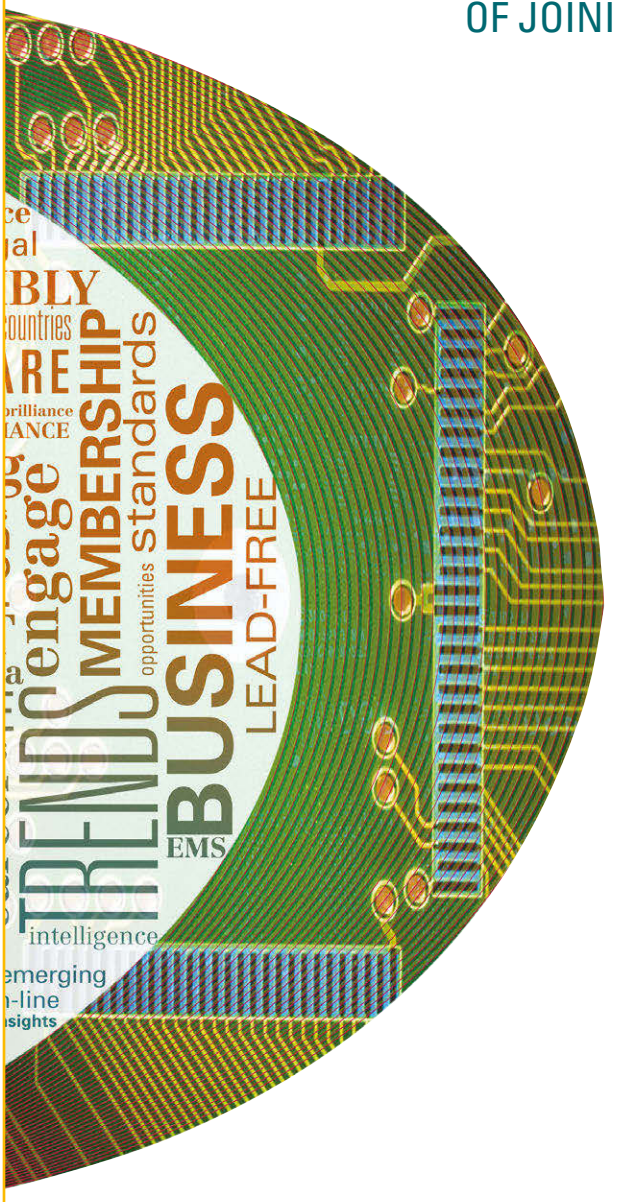
Frank: They do a great job with that. This year they have us focused on the Department of Labor and the classifications of direct versus indirect employees. What the Department of Labor is talking about doing is just crazy talk and frankly I wasn't really that aware of it until IMPACT this year. They've pointed it out to us as something that we can go and be heard on.

Goldman: *Everett, thank you so much.*

Frank: Thank you. **SMT**

Experience the BENEFITS

OF JOINING THE INDUSTRY'S PREMIER ASSOCIATION



IPC membership will expand your resources and influence in the electronics industry

- One single-user download of each new or revised IPC standard
- Significant savings on IPC standards, publications, and training materials
- Special registration rates for IPC conferences and educational events, including IPC APEX EXPO
- Access to participate in, and receive IPC market research studies
- 24/7 online access to members-only resources

www.ipc.org/membership

Improving the Solder Paste Printing Cycle Times

by Stephen Las Marias
I-CONNECT007

At the recent NEPCON China show in Shanghai, I interviewed Adam Sim, senior sales manager at Speedline ITW EAE. We discussed a variety of issues, including the challenges in solder stencil printing, how printing cycle times may be improved, and the key factors to consider when selecting a solder paste printing solution.

***Stephen Las Marias:** What are the greatest challenges when it comes to soldering?*

Adam Sim: On the solder printing process, the biggest challenge that most customers have is miniaturization. Based on our customer feedback, products are getting smaller, such as smartphones and other handheld devices, and functionalities are getting more complex. These trends result in more components being packaged in a smaller and smaller area. This, in turn, is driving the manufacture of even smaller components.

The second challenge is throughput. You have to get the highest output and yield to offset manufacturing cost.

So from a solder paste printing perspective, we have to address these two issues to help customers.

***Las Marias:** What about reliability?*

Sim: As components get smaller and smaller, the yield and reliability will drop accordingly. That's the reason why we continue to improve our printers—to address these issues.

***Las Marias:** What innovations are happening in solder paste printers to help customers address their printing challenges?*

Sim: With the miniaturization trend, for the printer to achieve higher yield, the machine has to be somehow more accurate. Accuracy is becoming a critical parameter in this process. The only possibility to address this issue is to develop more accurate machines. That's one way to do it.

***Las Marias:** The machines definitely have to be accurate. But are certain technologies more accurate than solder paste printers?*

Sim: Solder paste dispensers will have an edge on this front, but in a real production world,

AVOID THE VOID™

Increase your reliability, not your customer complaints.

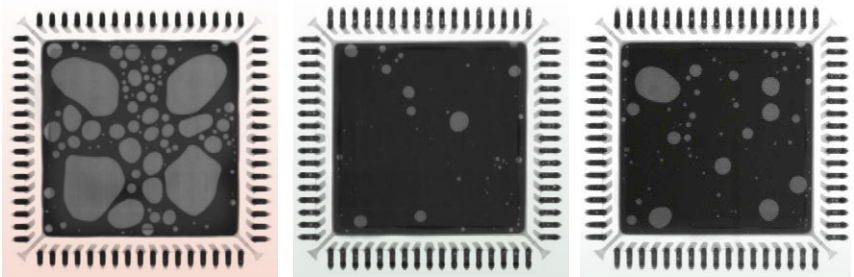
Indium10.1 Solder Paste

Lowest Voiding Performance

No-Clean Pb-Free Solder Paste

Indium8.9HF Solder Paste

Halogen-Free No-Clean Solder Paste



Typical Voiding
>40% Void Area

Indium10.1
<10% Void Area

Indium8.9HF
<15% Void Area

Contact our technical engineers today:
techsupport@indium.com

Learn more:
www.indium.com/avoidthevoid/SMTUS

From One Engineer To Another®

ASIA • CHINA • EUROPE • USA

INDIUM® CORPORATION®

©2016 Indium Corporation

Adam Sim

you will sacrifice on throughput. They are slower. Imagine, printing will take one dot (stroke) whereas dispensing will take thousands of dots. So the cycle time is different. It's a very different process. Although solder paste dispensing has a better edge over getting these finer pitches done.

***Las Marias:** What are the key factors to consider when selecting a solder paste printing solution?*

Sim: Typically, the customer will look at the machine accuracy—that's always the first thing—because accuracy is the key, especially for tighter tolerances and finer pitches. If the machine is not accurate, you just cannot move on. The second thing is throughput.

***Las Marias:** What is the impact of solder pastes on printing?*

Sim: Different customers will use different pastes. Paste characteristic is typically defined by their viscosity and paste type. Printing a high-viscosity paste and low-viscosity paste are totally different. Different paste types, which means the solder ball size will be bigger in some, and smaller in others will also greatly impact the printing. There's a general guideline for printing—and not all pastes can be printed into fine lines. There has to be tradeoffs.

***Las Marias:** Do you think your innovations in solder paste printers address your customers' problems today?*

Sim: I would think so because our products are the next generation printers in the market. We have been in this industry for a very long time, and as such I would say that our technology is very far ahead. The key factors that will affect customers' processes are machine capabilities, in particular, the machine accuracy and throughput. More than 10 years ago, we came out with the MPM Accela, and this machine was the most accurate at that time. Accuracy is ±12.5 microns at Six Sigma with Cpk greater or equal to 2.0.

A Six Sigma process is one which expect to have not more than 3.4 defects features per million opportunities, thus it is critical to achieve it. That's the consistency and repeatability of the printer, which is why we put a very detailed specification on it. At that time, we believe that the system could print all the way down to 0201 component size. We know that devices will become small, but we are not sure how far.

Today, we are kind of hitting the edge. We can still see smaller components now, but the next generation of components, about 03015 and smaller, means that we have to develop a new technology.

Over the past two years, we have been developing a printer that will work for the next 10 years. That's where our new MPM Edison came out.

From 12.5 microns down to eight microns. It's a big change. If you think about it, we don't build printers for the next two years. We build them for the next 10 years. So whatever the component type is today, we have to think 10 years ahead and what they will be like.

Accuracy is a big thing. Customers have to

have an assurance on that. The correct method to justify and to verify the machine accuracy.

***Las Marias:** Has solder paste printing reached its limit? Where do you think are tolerances and pitches headed?*

Sim: Due to the requirement to reduce in product size and increase in functionality, customers now are starting to explore packaging all the functionalities within a chip. For instance, in a phone, the Wi-Fi, wireless, GPS, everything can be compacted into packages and this packaging process also requires printing. So even if customer packed everything inside, you will still have the challenge of accuracy for these systems in package.

Now, most customers find the 0.3 CSP a challenge, so they have been using that as a guideline for their machine selection. But we know the electronics industry is looking at smaller than that. I don't know how small it can be, but customers will continue to explore smaller and smaller components.

***Las Marias:** How's your market in China?*

Sim: This year, the market is not much different from last year. I don't think there will be a lot of growth, but even then it will be a marginal one. China is still the largest market in the world, no question about it. And the whole market is segregated into very different segments. Handheld devices carry quite a big proportion of the SMT market. We can see the demand for handheld devices is still good, but it's not growing in double digit levels.

***Las Marias:** Is there anything else you would like to add?*

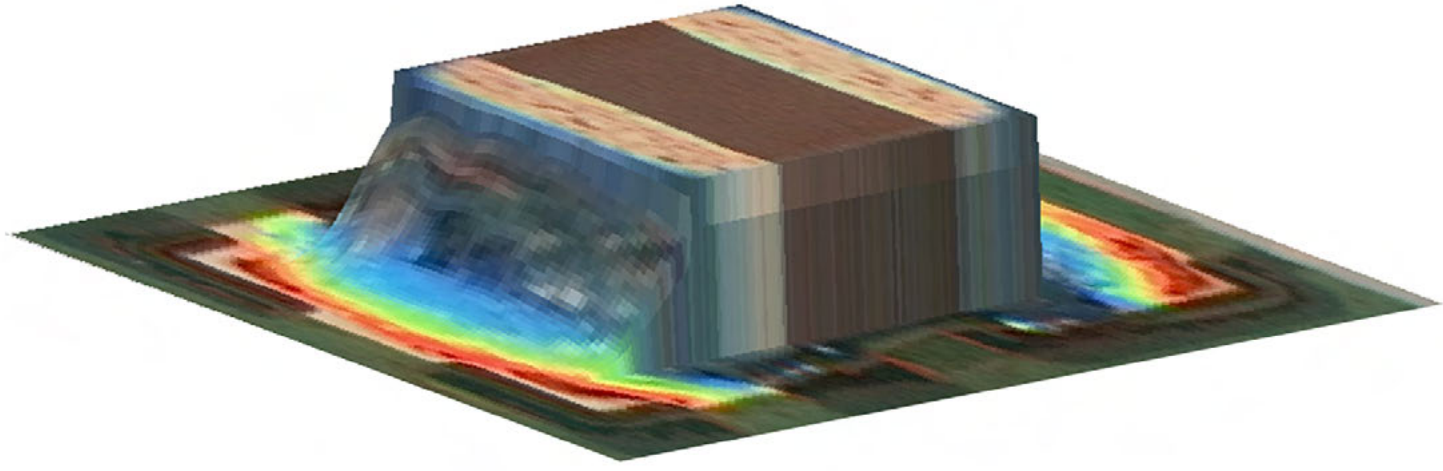
Sim: Another factor that customers should focus on is throughput. If you can have a high throughput, you can drive down cost. And all customers think of that. Although, customers are going for dual lane printers. Three years ago, we came out with a pretty decent solution where we packaged two printers into one—a back-to-back configuration. We are helping customers in deploying dual-lane SMT line, instead of the single-lane SMT line, to enable products on top and bottom, running at the same time, and in the process we saved on footprint. That's one thing we have been working on for the past years. On top of that, we came out with something very interesting. For many years, people are have measured the speed of the printer using the core cycle time. Core cycle time typically doesn't include the cleaning process. Most of the time, the printer's measurement when it comes to speed is the core, which means the PCB moves in, the machine does the alignment, the board moves up to the stencil, goes down, and exits. This is how the current printing industry measures the cycle time—but we feel this is not realistic, because customers will never run production that way. We realized it's the full cycle time, not the core cycle time, that's an important consideration.

So we came up with the parallel processing concept. In a printing process, there are many things happening. Your board moves along the line, then after alignment it moves up for printing to take action, and after printing it is followed by board separation, then exit. Then wiper cleaning process will take place to clean the stencil, because after printing on several boards, the stencil will get dirty or clog. This has been a very typical printer cleaning process. We realized that if we can make certain actions or sequences running concurrently, then we can cut down the entire cycle time tremendously. For example, if my board is moving in for the alignment, my stencil can go and do cleaning. Which means the cleaning of the stencil does not affect my overall cycle time. Imagine a very traditional way of printing: I print, I clean. Now, while printing, I am cleaning. It is a big development. We have introduced this to many customers and they are getting really excited because they can really see and feel the difference. So if we can make the cleaning process negligible in terms of cycle time, then the overall throughput will be greatly improved because they can print more.

***Las Marias:** Adam, thank you very much.*

Sim: Thank you. **SMT**

3D Solder Joint Reconstruction on SMD based on 2D Images



by Pedro M.A. Vitoriano, VISTEON,
and Tito G. Amaral, POLYTECHNIC INSTITUTE OF SETÚBAL

Abstract

Automated optical inspection (AOI) systems are commonly used in PCB manufacturing. The use of this technology has been proven as highly efficient for process improvements and quality achievements. The most challenging point in inspection of surface mounting devices (SMD) is the component solder joints, due to their specular reflects. Several studies have been made to improve this situation. This paper presents an algorithm for 3D solder joint reconstruction (3D-SJR). The criteria used in the classification of the solder joints was the IPC-A-610D (Acceptability of Electronics Assemblies).

Introduction

One of the most common technologies used in PCB manufacturing is surface mounted technology (SMT). By employing SMT, the production process speeds up, but the risk of defects also increases due to component miniaturization and denser packing of boards[1]. In these conditions, the failure detection has become critical for any SMT manufacturing process. AOI systems for PCBs have become a proven solution, replacing the traditional human-based inspection method[2].

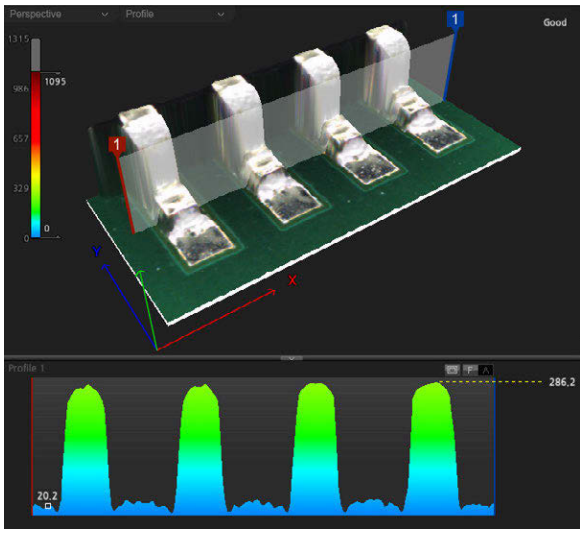
Historically, the primary place for AOI systems has been after solder reflow or post-production because post-reflow AOI systems can inspect for most types of defects (component placement, component polarity, solder shorts, missing solder, etc.) at one place in the manufacturing line with one AOI system. Hence, the faulty boards are reworked before they are sent to the next process stage.

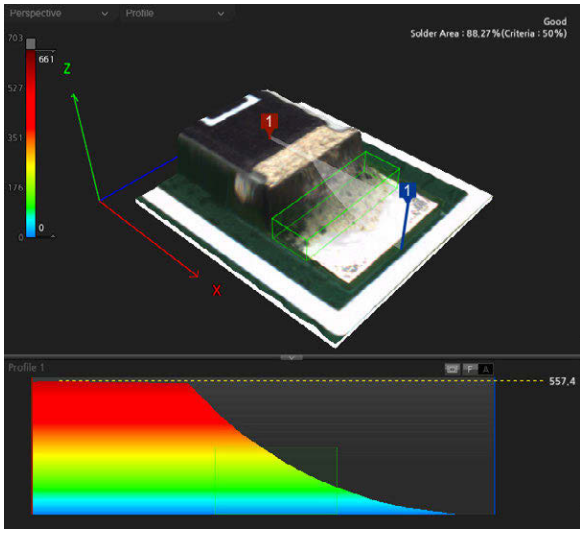
Solder joint inspection has been a critical issue for quality control, mainly due to their specular reflections[3]. AOI systems have been faced with the same difficulty for solder inspection[4]. In order to improve the solder joint inspection and classification, structured light solutions have been used[5].

With structured light solutions, a low angle illumination ring improves the solder joint classification. These light systems have proven to be a very good lighting solution for SMT inspec-

MIRTEC

Hurry!
Limited-Time Offer!





INTRODUCING THE ALL NEW MV-6 OMNI

HIGH-SPEED, HIGH-PERFORMANCE 3D AOI MACHINE

UNDER

$100K (USD)

WHILE SUPPLIES LAST!

- OMNI-VISION® 3D Inspection Technology
- 15 Mega Pixel CoaXPress Camera Technology
- 10 um Telecentric Compound Lens Design
- 10 Mega Pixel SIDE-VIEWER® Camera System
- Eight Phase Color Lighting System
- Full 3D Co-Planarity and Solder Fillet Inspection Capability

www.mirtec.com

tion and solder analysis. Some studies using structured light and advanced light reflection theories have been developing 3D reconstruction of solder joint shapes[6-8].

IPC established the A-610 document to help manufacturers achieve the highest possible SMT production quality. The standard specifies three classes of electronic devices, depending on how mission critical the final application is. For each class, IPC-A-610 defines assembly of component packages onto PCBs using measurable dimensions related to component position and solder fillet size[9].

IPC-A-610 rules specify that the solder joint should reach at least 25% of the component electrode height, however 2D AOI systems aren't capable of measuring the height of the solder joint. The measurement of such characteristics will require a 3D measurement system[10]. The proposed approach presents a 2D AOI system using the 3D-SJR method, for improving solder joint classification (Figure 1).

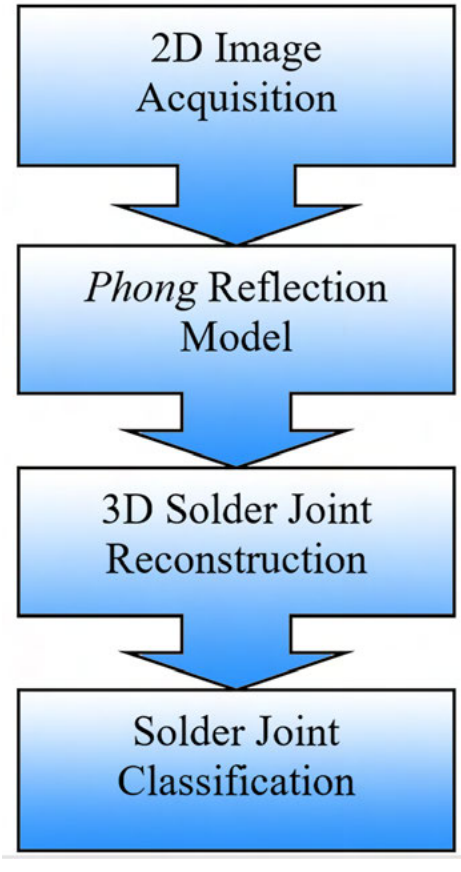


Figure 1: Block diagram of the proposed approach.

2D Image Acquisition

The image acquisition system was installed in the SMT manufacturing line. The equipment used on this study was the VT-RNS-II from OMRON[12]. Figure 2 shows an RGB image obtained in the image acquisition stage.

The illumination system used in the acquisition step consists of three rings of LED lights, each ring with a different LED color and placed at a different angle (Figure 3).

Phong Reflection Model

The proposed approach uses the Phong reflection model to analyze the solder joint area. The Phong reflection model (1) has three components: the ambient light reflection (Ia), the diffuse light reflection (Id) and the specular light reflection (Is).

$$\mathbf{Ir} = \mathbf{Ia} + \mathbf{Id} + \mathbf{Is} \qquad 1$$

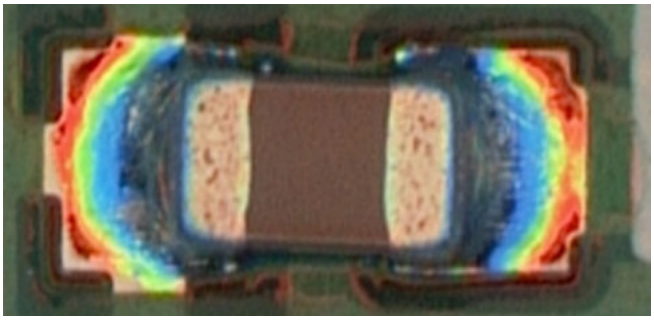


Figure 2: Initial image.

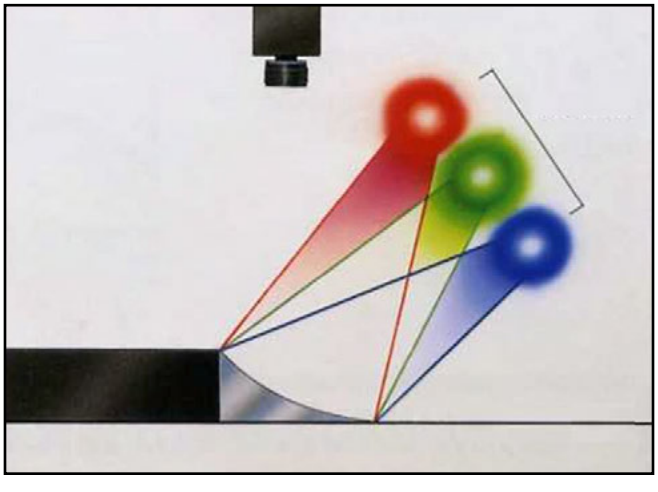


Figure 3: Light rings[12].

Expanding the Phong model, the equation (2) will allow to define the illumination reflection Ir, according to the material characteristics, nShiny, illumination angle, ⌀i, observation angle, ⌀o, and the surface angle, ⌀s. The parameter Ia, Id and Is correspond to ambient light intensity, diffuse light intensity, and specular light intensity.

$$\begin{aligned} \mathbf{Ia} &= \mathbf{ia} \\ \mathbf{Id} &= \mathbf{id}.\boldsymbol{cos}(\emptyset\mathbf{i} - \emptyset\mathbf{s}) \\ \mathbf{Is} &= \mathbf{is}.\boldsymbol{cos}(\emptyset\mathbf{i} + \emptyset\mathbf{o} - \mathbf{2}.\emptyset\mathbf{s})^{\boldsymbol{nShiny}} \end{aligned} \quad 2$$

Using the Phong model, it will be clear that the reflection light intensity will depend on several variables shown in equation (3).

$$\begin{aligned} &\mathbf{Ir}(\emptyset\mathbf{i}, \emptyset\mathbf{o}, \emptyset\mathbf{s}, \mathbf{ia}, \mathbf{id}, \mathbf{is}, \boldsymbol{nShiny}) \\ &= \mathbf{ia} + \mathbf{id} \times \boldsymbol{cos}(\emptyset\mathbf{i} - \emptyset\mathbf{s}) \\ &+ \mathbf{is} \times \boldsymbol{cos}(\emptyset\mathbf{i} + \emptyset\mathbf{o} - \mathbf{2}.\emptyset\mathbf{s})^{\boldsymbol{nShiny}} \end{aligned} \quad 3$$

Since the proposed approach works with metallic surfaces, the diffuse light reflection was neglected and the intensity of light reflection will depend only on the ambient lights and specular reflection (4).

$$\mathbf{Ir} = \mathbf{Ia} + \mathbf{Is} \quad 4$$

To analyze the solder joint surface, a square area, P(x,y), was drawn with limitations of the pad and component localization according to specification [11] (Figure 4). Since the initial image is an RGB color image, the square area has the correspondent's three plans (red, green and blue components).

$$\mathbf{P}(\mathbf{x},\mathbf{y}) = [\mathbf{P_r}(\mathbf{x},\mathbf{y}) \quad \mathbf{P_g}(\mathbf{x},\mathbf{y}) \quad \mathbf{P_b}(\mathbf{x},\mathbf{y})] \quad 5$$

The light reflection on the square area could be calculated using the Phong reflection model (6).

$$\mathbf{P}(\mathbf{x},\mathbf{y}) \cong \mathbf{Ir}(\emptyset\mathbf{i}, \emptyset\mathbf{o}, \emptyset\mathbf{s}(\mathbf{x},\mathbf{y}), \mathbf{ia}, \mathbf{id}, \mathbf{is}, \boldsymbol{nShiny}) \quad 6$$

Considering the following parameters as constants (⌀i,⌀o,ia,id,is,nShiny), it will be possible to apply the inverse function of the Phong reflection model to obtain the solder joint surface angle ⌀s(x,y), for each pixel coordinate defined in P(x,y) (7).

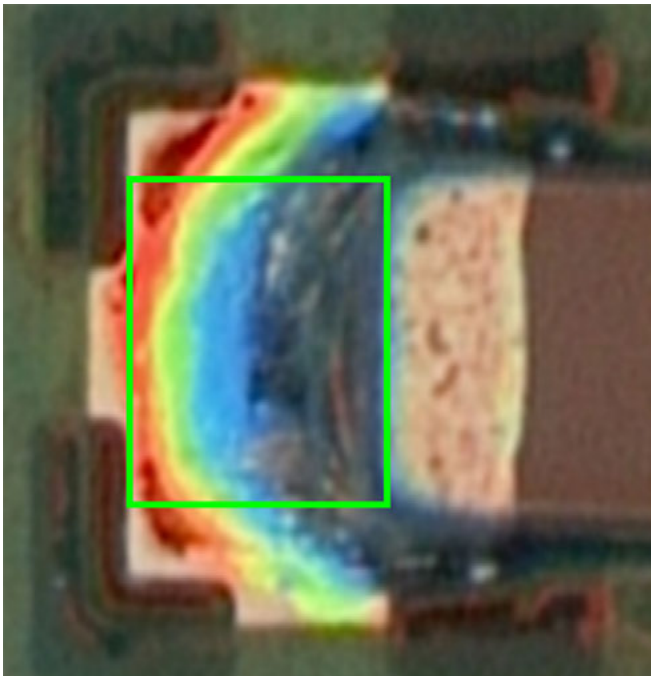


Figure 4: Solder joint area.

$$\emptyset\mathbf{s}(\mathbf{x},\mathbf{y}) = \mathbf{Ir}^{-1}\big(\mathbf{P}(\mathbf{x},\mathbf{y})\big) \quad 7$$

The 3D reconstruction of the solder joint surface can be determined by:

$$\mathbf{z}(\mathbf{x},\mathbf{y}) = \sum_{\mathbf{xi}=\mathbf{0}}^{\mathbf{x}} \boldsymbol{tan}\big(\emptyset\mathbf{s}(\mathbf{xi},\mathbf{y})\big) \quad 8$$

To convert the z(x,y) function from pixels to metric dimensions, the CR parameter was used for the camera resolution in µm/pixel.

Ambient Light Reflection

Since the ambient light reflection doesn't contain any information from the surface angle, this light component was removed from the inspection area by (9), keeping only the observed specular reflection Ps(x,y) for the following developments.

$$\mathbf{Ps}(\mathbf{x},\mathbf{y}) = \mathbf{P}(\mathbf{x},\mathbf{y}) - \mathbf{Ia}(\mathbf{x},\mathbf{y}) \quad 9$$

The ambient light reflections Ia are calculated along the area of solder joint using (10).

$$\mathbf{Ia}(\mathbf{x},\mathbf{y}) = \boldsymbol{min}\,(\mathbf{P}(\mathbf{x},\mathbf{y})) \quad 10$$

Since the solder joint area is defined by three plans, the inspection area without the ambient light reflection can now be given by (11).

$$\mathbf{Ps}(x,y) = \begin{cases} \mathbf{Ps_r}(x,y) = \mathbf{P_r}(x,y) - \mathbf{Ia}(x,y)\left(1 - \dfrac{\mathbf{P_r}(x,y)}{\mathbf{P_I}(x,y)}\right) \\ \mathbf{Ps_g}(x,y) = \mathbf{P_g}(x,y) - \mathbf{Ia}(x,y)\left(1 - \dfrac{\mathbf{P_g}(x,y)}{\mathbf{P_I}(x,y)}\right) \\ \mathbf{Ps_b}(x,y) = \mathbf{P_b}(x,y) - \mathbf{Ia}(x,y)\left(1 - \dfrac{\mathbf{P_b}(x,y)}{\mathbf{P_I}(x,y)}\right) \end{cases} \quad 11$$

where

$$\mathbf{P_I}(x,y) = \mathbf{P_r}(x,y) + \mathbf{P_g}(x,y) + \mathbf{P_b}(x,y). \quad 12$$

Figure 5 shows the obtained image after removing the ambient light reflection from the initial image.

Specular Light Reflection

The Phong model allows calculating the specular light reflection based on the knowledge of the specular light intensity, the observation and illumination angles and the characteristics of the reflector material (13).

$$\mathbf{Is}(\emptyset i, \emptyset o, \emptyset s, \mathbf{is}, nShiny) = \mathbf{is}.cos(\emptyset \mathbf{i} - \emptyset \mathbf{o} - 2.\emptyset \mathbf{s})^{nShiny} \quad 13$$

The value used in the observation angle position was 90°, and the illumination angles used in the structured light system are given in Table I. These parameters are determined initially and remain constant during the procedure of solder inspection.

As in the case of the ambient light reflection, the specular light reflection has three plans corresponding to the red, green and blue components (14).

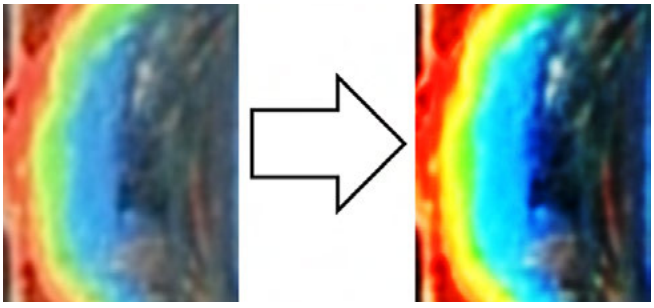


Figure 5: Inspection area without the ambient light reflection.

$$\mathbf{Is} = [\mathbf{Is_r}\ \mathbf{Is_g}\ \mathbf{Is_b}] \quad 14$$

The specular light intensity for each LED light ring is obtained by finding the maximum intensity value of each specular light reflection plan (15).

$$\begin{aligned} \mathbf{is_r} &= \boldsymbol{max}\ (\mathbf{Ps_r}) \\ \mathbf{is_g} &= \boldsymbol{max}\ (\mathbf{Ps_g}) \\ \mathbf{is_b} &= \boldsymbol{max}\ (\mathbf{Ps_b}) \end{aligned} \quad 15$$

The nShiny parameter is a constant value that depends on reflective material only. Table II shows the nShiny value for the two different solder pastes, with and without lead.

With all constants (øi,øo,ia,id,is,nShiny) calculated previously for the AOI system, the calculation of specular light reflection Is, will depend only on surface angle øs,as shown on (16).

$$\mathbf{Is}(\emptyset s) = \begin{bmatrix} \mathbf{is_r}.cos(\emptyset \mathbf{ir} - \emptyset \mathbf{o} - 2.\emptyset \mathbf{s})^{nShiny} \\ \mathbf{is_g}.cos(\emptyset \mathbf{ig} - \emptyset \mathbf{o} - 2.\emptyset \mathbf{s})^{nShiny} \\ \mathbf{is_b}.cos(\emptyset \mathbf{ib} - \emptyset \mathbf{o} - 2.\emptyset \mathbf{s})^{nShiny} \end{bmatrix} \quad 16$$

Light Ring	Position Angle
Øir	85°
Øig	55°
Øib	25°

Table I: Light Ring Parameters.

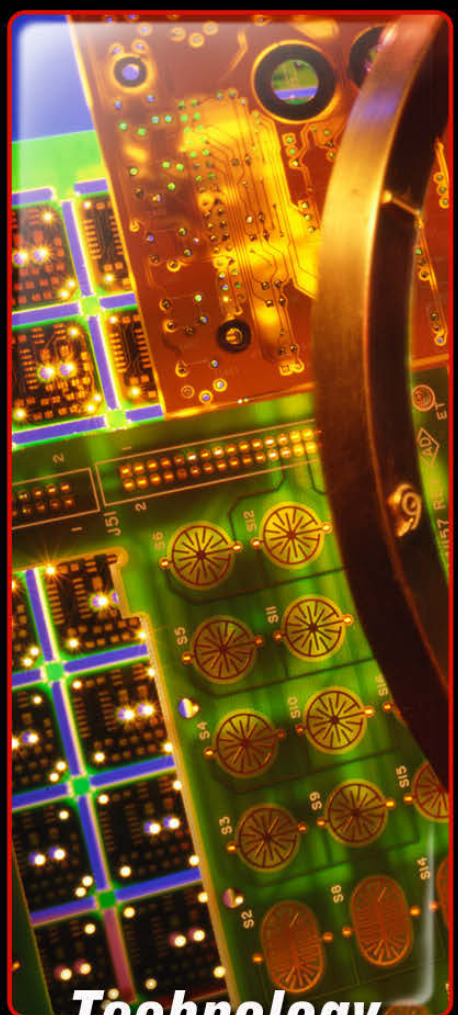
Solder Paste	nShiny Value
Lead	9
Lead Free	7

Table II: nShiny Parameter.

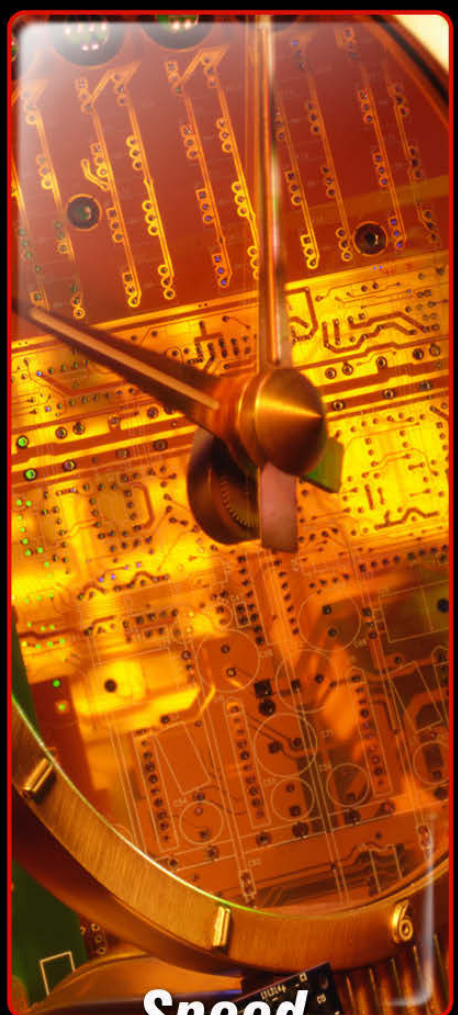
prototron.com

We Take the Time to do it Right

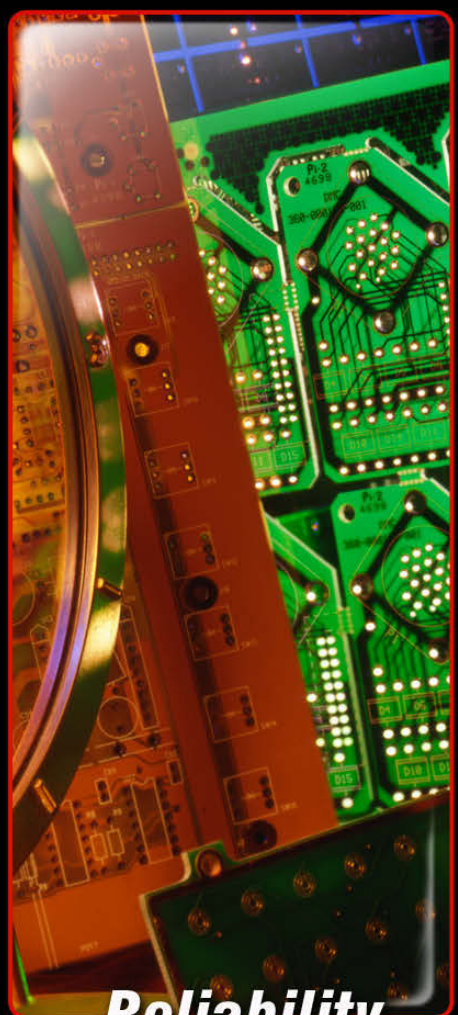
because you don't have the time to do it over.



Technology



Speed



Reliability

Prototron Circuits

High quality quick turn manufacturing, just another reason we are

America's Board Shop

Redmond Facility
15225 NE 95th Street, Redmond, WA 98052
Toll: 888.847.7686 Phone: 425.823.7000

Tucson Facility
3760 E. 43rd Place, Tucson, AZ 85713
Toll: 800.279.5572 Phone: 520.745.8515

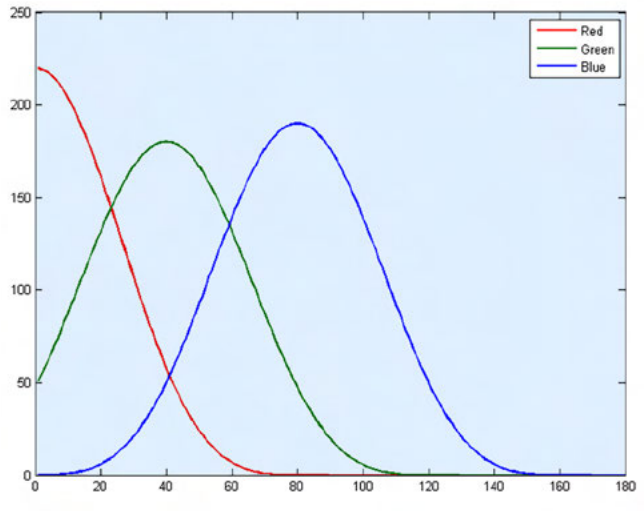


Figure 6: Phong calculated illumination reflection.

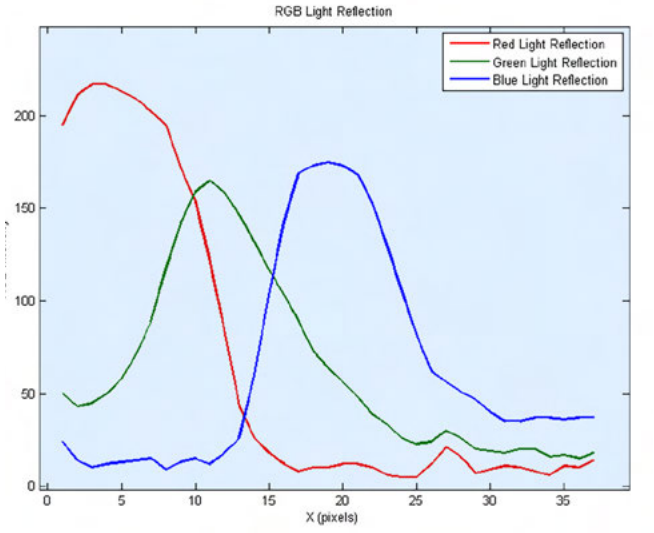


Figure 7: The observed RGB color variation along the solder joint profile.

With the specular light intensity of each LED light ring, it is possible to draw the calculated specular light reflection for each surface angle Is(⌀s) (Figure 6).

Since the intensity of light reflection, Is, depends only on the surface angle, ⌀s, now it will be possible to determine the surface angle by applying the inverse function of Phong (7).

The inverse Phong was implemented by using the Manhattan distance between the observed specular reflection from the solder joint surface area, Ps(x,y), and each angle from the Phong illumination reflection, Is (17).

$$\emptyset s = imin\left(|Ps - Is|\right) \qquad 17$$

Since both the observed light reflection, Ps(x,y), and the calculated specular reflection, Is, consist in three plans of RGB components, the equation (17) will be developed to (18).

The solder joint surface angle is determined as the minimum index obtained for angles between 0º and 90º, representing all possible angles of the solder joint surface.

$$\emptyset s(x,y) = imin\left|Ps_r(x,y) - Is_r(\emptyset)\right| + \left|Ps_g(x,y) - Is_g(\emptyset)\right| + \left|Ps_b(x,y) - Is_b(\emptyset)\right| \qquad 18$$

Figure 8 shows the result function, ⌀s(x,y), where the variation of solder joint surface angle, ⌀s, can be seen along P(x,y).

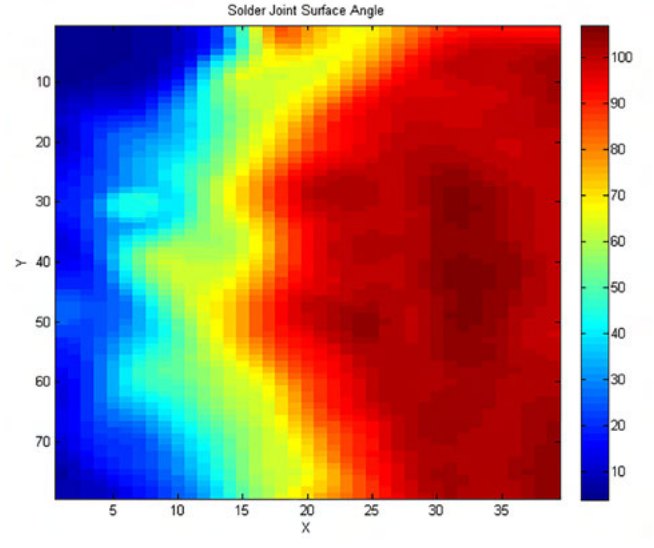


Figure 8: Surface angles along the solder joint.

3D Solder Joint Reconstruction

The reconstruction of the solder joint surface shape, z(x,y), is possible by integrating the solder joint surface angle, ⌀s(x,y), along the solder profile from the beginning of the solder pad to the component electrode (19).

$$z(x,y) = \sum_{xi=0}^{x} tan\frac{\emptyset s(xi,y)}{2} \qquad 19$$

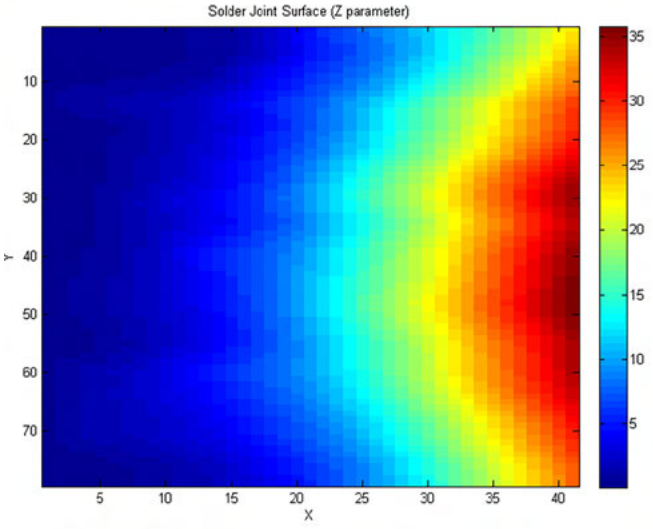


Figure 9: Solder joint height variation.

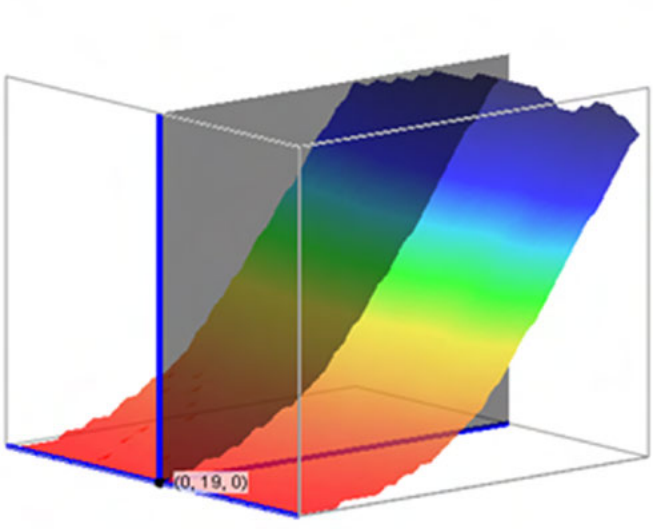


Figure 10: Solder joint reconstruction of the inspection area for a specific y coordinate.

Figure 9 shows the height variation of the solder joint from a minimum value of 25×20 µm obtained at the component electrode and a maximum value of 35×20 µm. The obtained minimum value is higher than the minimum value specified on IPC A-610 for the solder joint height at the component electrode[9].

Solder Joint Classification

The 3D-SJR allows the implementation of new classifiers to analyze the solder joint, namely length, height and volume (Table III). The solder joint height is computed for a specific y coordinate of the inspection area. The parameter cr is the camera resolution used in the acquisition system.

The IPC-A-610 standard specifies the limits for the solder joint of SMD components. According to this standard, the solder height (SJH) should be higher than 25% of the component

Solder Joint Length	$\mathbf{SJL(y)} = \sum_{xi=0}^{x} \mathbf{z(xi, y)} > \mathbf{th}$
Solder Joint Height	$\mathbf{SJH(y)} = \boldsymbol{max}\mathbf{(z(x, y))} \cdot \mathbf{cr}$
Solder Joint Volume	$\mathbf{SJV} = \sum_{xi=0}^{x} \sum_{yi=0}^{y} \mathbf{z(xi, yi)} \cdot \mathbf{cr^3}$

Table III: 3D-SJR Classifiers.

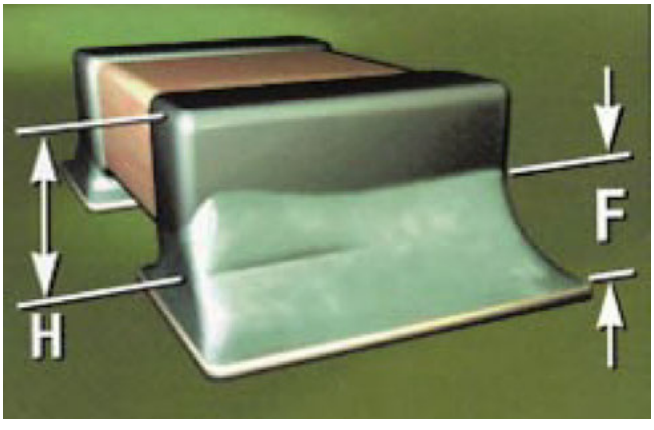


Figure 11: Solder joint height[9].

electrode height[9]. Figure 11 shows an example described in the IPC-A-610 document, where the specification for the solder height can be seen close to the component electrode. The parameter H represents the component height, and the parameter F represents the solder joint height at the component electrode. The reconstruction of the surface solder shape (3D-SJR) determines the F value to be used in the solder joint classification on the AOI system.

Taking into consideration the IPC-A-610 standard regarding the specification of the solder joint along the component electrode, as seen in Figure 12, it was implemented a new classifier (20).

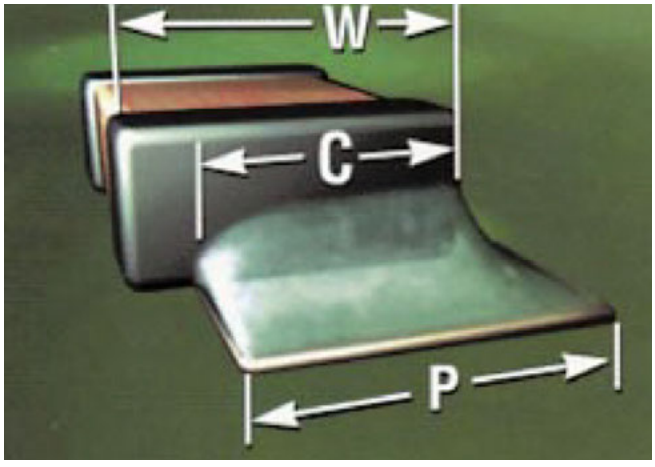


Figure 12: Solder joint length[9].

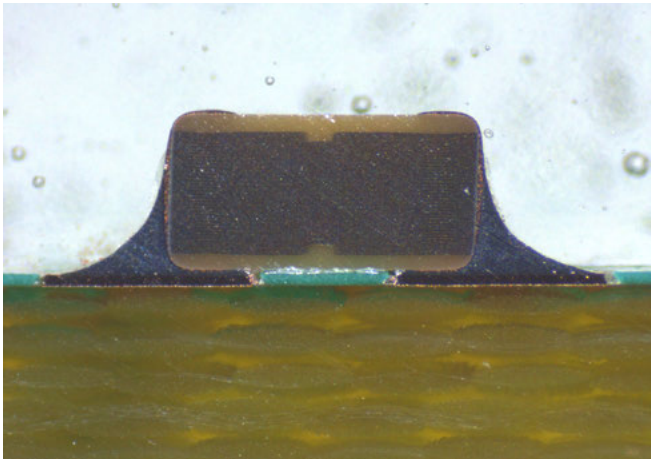


Figure 13: Solder joint profile.

This classifier analyzes the percentage of component electrode with solder joint according the IPC-A-610 standards. This percentage value should be higher than 75% for Class III products and 50% for Class I and II products. The parameter Cy corresponds to component size and parameter Py to the pad size on the Y axis.

$$SJ = \frac{\sum_{yi=0}^{y} \begin{cases} 1 \text{ if } SJH(yi) > th \\ 0 \text{ otherwize} \end{cases}}{\min(Cy; Py)} \times 100 \quad (20)$$

Experimental Results

The inspection algorithms were implemented using National Instruments' LabVIEW with the Vision Development Module, and Matlab for image processing. The images were acquired under several VT-RNS II AOIs from OMRON, installed on PCB manufacturing lines and the camera resolution, cr, was 20 um/pixel.

To validate the proposed approach for solder joint reconstruction, some cuts were made over SMD components (Figure 13). Applying the proposed 3D-SJR, the 3D model of the solder joint surface illustrated in Figure 14 confirmed the correlation between the proposed method and the solder joint surface shape. The solder joint reconstruction allows the operator to visually check the component state and confirms the failures detected by the AOI system.

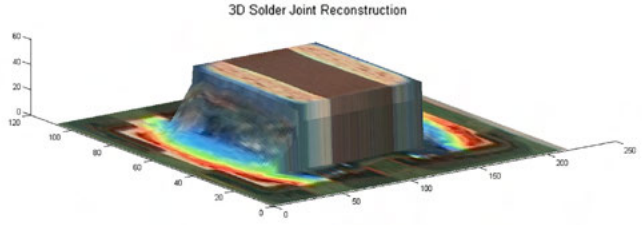


Figure 14: 3D-SJR applied over the ceramic capacitor.

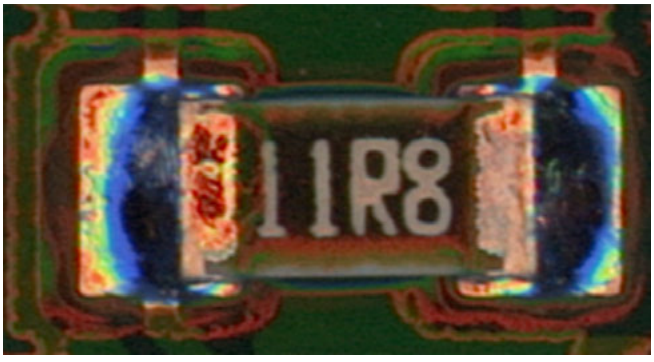


Figure 15: Good solder joint sample.

The 3D-SJR approach was applied over different component packages (0402, 0804, and 1206). Figure 15 illustrates an SMD component with a good solder joint.

According to the IPC A-610 standard, the solder height at the component electrode must be at least 25% of the electrode height, Cz (19). Considering the component size of the package illustrated in Figure 15 and the camera resolu-

PCB BUYERS - compare nearly 1900 manufacturers now at *The* PCB List.

Quick Search

Advanced Search

thepcblist

The best way to find a PCB fabricator, *anywhere*.

www.thepcblist.com

tion, the minimum allowable height of the solder is 12.5 pixels on the Z axis.

$$SJH(y) > \frac{Cz.25\%}{cr} = \frac{1000\mu.25\%}{20\mu/pixel} = 12.5\ pixels \quad 21$$

Figure 16 shows the obtained reconstruction of the solder joint surface where the contour lines of the solder joint is within the specifications of the IPC A-610 standard.

Figure 17 shows a SMD resistor where the left pad has a lower solder quantity when compared with the right pad.

Applying 3D-SJR, it is possible to verify that the solder joint is not according to the specification on the IPC A-610. In these specifications at least 75% of electrodes should have a minimum solder joint of 25% the component electrode height.

Figure 18 represents the contour lines of the solder surface obtained by applying the 3D-SJR method.

The obtained results of the solder joint (SJ) classifier for the two samples show that on the bad sample only 37% of the electrodes have solder joints higher than 25% of the electrode height. In this case, 3D-SJR allows the verification that the solder height in the component electrode is not according to IPC A-610 standard specifications.

Conclusions

Due to the specular reflection of the solder joint, the solder joint analysis is the most difficult step in AOI systems. With the structured ring light system, it was possible in the image acquisition to correctly identify the solder shape and improve solder analysis.

The proposed 3D-SJR method improves the actual AOI system solution for solder joint analysis and achieves better results in fault detection not possible with the traditional 2D AOI systems.

Nowadays because of the difficulty of 2D AOI systems to provide solutions for inspection in several cases of failure detection, companies have started buying 3D AOI systems that are in

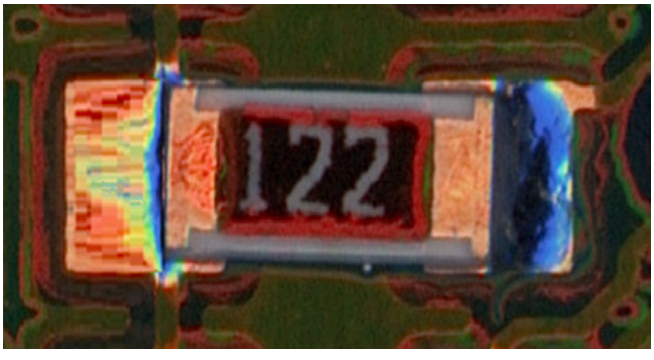


Figure 17: Bad solder joint sample.

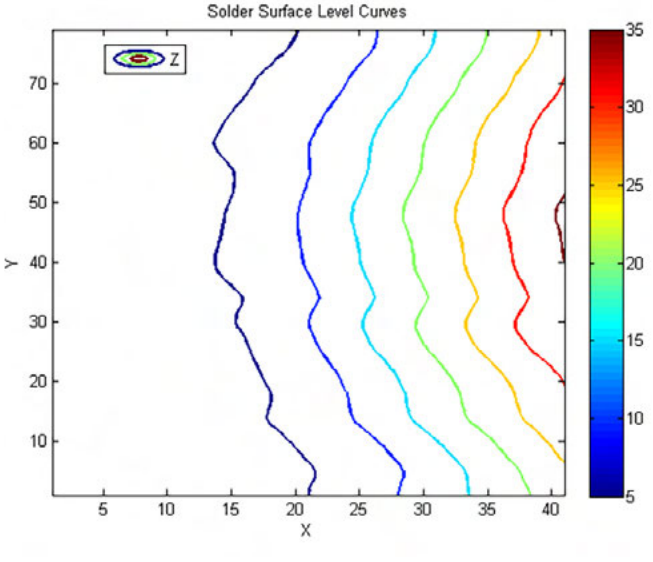


Figure 16: Solder surface contour lines.

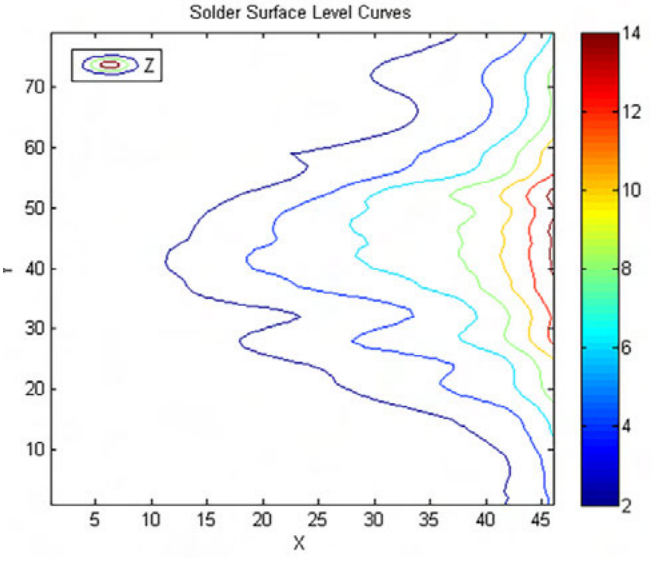


Figure 18: Solder surface level curve (bad sample).

most cases more expensive and have slower inspection cycle times.

The proposed method gives 3D analysis capability to the traditional 2D AOI equipment, practically without increasing the financial cost and with no cycle time impact, bringing 2D AOI equipment back into the game of SMT technology. SMT

References

1. G. Acciani, G. Brunetti and G. Fornarelli, " A Multiple Neural Network System to Classify Solder Joints on Integrated Circuits," International Journal of Computational Intelligence Research (2006) Vol. 2, No 4, pp 337–348.

2. Tae-Hyeon Kim, Tai-Hoon Cho, Young-Shik Moon and Sung-Han Park, "Visual Inspection System for Classification of Solder Joints," Pattern Recognition 32 (1999), pp 565–575.

3. Kuk Won Ko and Hyung Suck Cho, "Solder Joints Inspection Using Neural Network and Fuzzy Rule-Based Classification," IEEE 98, pp. 1565–1570.

4. Jiquan Ma, Peijun Ma and Xiaohong Su, "Solder Joints Detection Method Based on Surface Recovery," Computer and Information Science, Vol. 3, No. 2; May 2010.

5. Xuemei Zhang and Gareth Bradshaw, "Solid Shape Modeling—Working Toward 3D AOI," SMT Magazine Archive, November 1, 2004.

6. Kwangill Koh, "Identify Lifted Objects using Z-axis Measurement and 3D AOI," SMT Magazine Archive, July 1, 2009.

7. Tae-Hyeon Kim, Tai-Hoon Cho, Young-Shik Moon and Sung-Han Park, "An Automated Visual Inspection of solder Joints Using 2D and 3D Features," IEEE 96, pp. 110–115.

8. Y.K. Ryu and H.S. Cho, "Visual Inspection Scheme for Use in Optical Solder Joint Inspection System," IEEE April, 96, pp. 3259–3264.

9. IPC-A-610E, from Association Connecting Electronics Industries.

10. Dr. Malachy Rice, "Using AOI to verify IPC compliance," Agilent Technologies, Dublin, Ireland, Test & Measurement World, 11/1/2004.

11. Pedro M. A. Vitoriano, Tito G. Amaral, Octávio Páscoa Dias, "Automatic Optical Inspection for Surface Mounting Devices with IPC-A-610D compliance." IEEE 4/11, 978-1-4244-9843-7.

12. VT-RNS II documentation from OMRON.

Pedro M.A. Vitoriano is a product development electrical engineer at Visteon. To contact Vitoriano, click here.

Tito G. Amaral is a professor at ESTSetubal, Polytechnic Institute of Setúbal, Setubal, Portugal, and at the Institute of Systems and Robotics – Polo Coimbra, Coimbra, Portugal.

Using Solid-state Materials with Gold Nanoantennas for More Durable Solar Cells

Scientists at Japan's Hokkaido University are making headway in the fabrication of all-solid-state solar cells that are highly durable and can efficiently convert sunlight into energy. The team employed a method called "atomic layer deposition", which allows scientists to control the deposit of very thin, uniform layers of materials on top of each other. Using this method, they deposited a thin film of nickel oxide onto a single crystal of titanium dioxide. Gold nanoparticles were introduced between the two layers to act like an antenna that harvests visible light.

The team tested the properties of these fabricated devices with and without an intermediary step following the deposition of nickel oxide that involves heating it to very high temperatures and then allowing it to slowly cool – a process called "annealing".

Photocurrent generation was successfully observed on the all-solid-state photoelectric conversion device. The device was found to be highly durable and stable because it does not contain organic components, which have a tendency to degrade over time and under harsh conditions.

However, the resultant device's properties are still insufficient for practical use and its efficiency in converting light to energy needs to be improved. Further research is needed to understand the roles of each layer in conducting energy to improve the device's efficiency.

Solder Paste Printing: A User's Perspective

by Stephen Las Marias
I-CONNECT007

I recently sat down with Integrated Micro-Electronics Inc.'s Joemar Apolinario, engineering manager; Aurelio Bantigue, DFM engineer; and Rodney Bebe, process engineer, to get their insights on the solder paste printing process. They discussed the challenges and key considerations to make when dealing with tighter tolerances and finer pitches in line with the continuing miniaturization trend in the electronics manufacturing industry. They also talked about the impact of solder pastes in the printing process, and the criteria for their selection and qualification. Finally, they highlighted design and process strategies to get the best solder paste printing results.

***Stephen Las Marias:** Please talk about the solder paste printing process and the challenges you face.*

Joemar Apolinario: The solder paste printing process is the most challenging process in the SMT operation because majority of the defects—around 60 to 80%—are coming from this process. These include non-wetting, insufficient volume, and short (solder bridging). Factors that greatly impact the solder paste printing quality may vary from the stencil design, the solder paste type, the machine capability (parameters), setup and tooling, and even the engineering skills of those who handle this process.

The challenge is that the devices now are getting smaller and smaller. We also have the complex products—those boards that combine critical components such as BGAs and ICs, which require finer pitches and smallest diameters—and non-critical components. The third challenge is the reliability. We have products that require higher reliability—and those are very sensitive to solder cracks or voiding defects.

For most of the defects that we have encountered, the solder volume being deposited to the pad does not meet the requirement.

***Las Marias:** How do you address those challenges?*

Aurelio Bantigue: From a DFM standpoint, what we have done so far is we review the stencil design or the aperture design for each of the components. In our view, we consider both the PCB design and the components that will be mounted to the board. We recognize that the trend is toward smaller components, and the challenge is the appropriate aperture design for smaller pins and pads. What we can do is consider the appropriate solder volume. But the more chal-

STRIVING FOR EXCELLENCE IN EVERYTHING WE SAY AND DO.

- 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

Tour our Facilities

View our Capabilities

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

lenging issue is the complexity—you are mixing small components, and larger components that require more solder paste, in a board. So, it will be more difficult. There are several design considerations that we do. We can do electroform step stencils, or sometimes, we have to overprint the solder paste—you have the solder paste outside the pad, you will have enough volume for those larger components. We can deposit small volume for the small components and enough volume for the larger components.

***Las Marias:** How does the solder paste impact the printing process?*

Apolinario: The solder paste has a big impact on printing as it is the main ingredient of the process and it has a direct impact on reliability. The choice of the right type and composition of solder paste may help achieve good printing quality.

> "The choice of the right type and composition of solder paste may help achieve good printing quality."

But, the key here is having the right parameter to use, for instance, the design of the pad and the stencil. We need to select the appropriate type of solder to match the stencil design and pad design to have a good printing result. Another factor is the machine. We need to use the appropriate machine to get good results. We have different types of solder printing machines—we have the screen printing as our typical printing method; we are now also using jet printing process. So, that's the factor. Apart from the multi-step stencil design to accommodate those complex PCBAs, we are now trying electroform nano-coated stencil. This is something new in our process—nano-coating by immersion coating process. The design of the stencil is electroformed, then it goes through a nano-coating process. This process makes the stencil smoother and non-sticky, to have a good transfer efficiency and to reduce cleaning sequence in the process.

Rodney Bebe: The solder powder size varies from type 1 to type 8—the higher the type number, the smaller the solder powder size. Technically, the smaller the balls, the more alloy you can deposit onto a pad. Another thing is the alloy content itself. Solder alloy varies, majority of our current solder are SAC (tin, silver and copper) as this is widely used for industrial and automotive and cost effective. Another consideration is customer requirements. If the product is automotive grade, definitely you must have high reliability solder pastes. So solder pastes go through evaluation and tests to compare which is the best to use. For instance, we use a certain type of solder paste for automotive applications. For commercial or industrial PCBs, we use another type.

Currently, we are exploring the use of higher-grade solder pastes. One is Innolot alloy which is touted to have top-level reliability; it's an SAC-based alloy with small amount of antimony, bismuth, and nickel.

***Las Marias:** Is there a need to collaborate with solder suppliers?*

Apolinario: Yes, we need to collaborate, especially when there are tighter requirements, or if there are requirements we were not able to prove from our current solder. We need to work with solder paste makers to study or come up with the best formulation of alloys that can be a solution for certain problems. But it will be done through a series of evaluation, including temp-cycle process, manufacturing tests, and also functional test. We have to prove first before we proceed to use a particular solder paste.

Bebe: We have a generic formulation—for example, we are using SAC solder for automotive electronics applications, which requires higher reliability—and we have a specialized solder, which also needs to be approved by the customer.

***Las Marias:** How does the trend towards finer pitches and tighter tolerances affect the solder paste printing process?*

Apolinario: As devices become smaller, the probability of defects from a solder paste printing standpoint increases. The key is we need to do a printing process that is consistent and with high accuracy to serve those requirements. We have to consider the appropriate stencil design, the machine's accuracy, and the solder paste. Also, devices such as ICs and BGAs, which are the critical components of the PCBA and are sensitive to defects. For example, voiding in BGAs cannot be seen without using x-ray inspection.

So, if the printing process is not good, it's a waste of time. The challenge is to obtain an accurate result in the first pass.

Las Marias: What can you say about the advantages and disadvantages between solder jet printer and screen printing?

Apolinario: Both printers can be used in boards with tighter tolerances and smaller pitches. The difference, however, is that jet printing is quite slow—it's cycle time is about six minutes—while screen printing can be used in high-volume production with a printing cycle time average to less than 30 seconds. The advantage of jet printing is that you don't need a stencil. You can use jet printing in qualifications for low-volume, high-mix, or in NPI. Screen printing, meanwhile, is used in medium- to high-volume production. Accuracy-wise, they are comparable. The only difference is the speed—solder paste printing is much faster compared to jet printing.

Bantigue: Aside from speed, jet printing has limitations when it comes to smaller components because there is a minimum dispense dot size for jet print. For example, if you are dispensing 0.40 mm diameter of solder paste on jet print, you cannot use it for the 0.4 mm CSP mounting. Smaller components usually use solder paste printing instead of jet. Although jet printing will not require a stencil, you actually use a more expensive solder paste—usually type 5 or finer solder—unlike for the solder paste printing, you can use type 4 or larger solder pastes.

Las Marias: Does it make sense to have both equipment in a factory?

Apolinario: Yes. If you are running a low-volume project, it is better to use jet printing. If you have an R&D or NPI project, running one board, two boards, then it makes sense to use a jet printer. But for your medium- to high-volume operation, then you should be using a screen printer. In terms of performance, accuracy, it's just the same for both.

Las Marias: What are the best practices to consider when it comes to solder paste printing?

Apolinario: First, the product should undergo the DFM to come up with the optimum aperture design and finish of both stencil and PCB. For complex boards that combine small and big components, you have to consider the multi-step or step-design stencil, or what we call over-printing.

Next thing to consider is the solder paste selection—what type and kind of solder to use, what's best for certain products or projects.

Lastly, you need to consider what equipment to use.

Las Marias: What do you think is the future for solder paste printing?

Apolinario: We are seeing smaller pitches, such as 0.3 mm (300 microns) and below, as a challenge going forward. This is the same challenge for the equipment manufacturers—they have to be able to support this trend.

Bantigue: Maybe, combining both solder paste printing and flux (sticky flux) dip for CSP. Instead of using solder paste on the pads of CSPs, you will dip the solder spheres (of CSP) on flux before mounting on the printed circuit board.

Las Marias: Thank you very much for you time, and it's great speaking with you guys.

Apolinario: Thank you, Stephen. **SMT**

To Bake or Not to Bake (in Rework)—That is the Question

by Bob Wettermann
BEST INC.

When performing rework on printed circuit boards, the issue of the moisture having to be baked out of the PCB is often debated. Whether it is localized mini-wave rework, hand-soldering or convection rework, the board, as well as neighboring components, needs to be taken into account in terms of moisture protection during the reflow cycle. The board components' protection of moisture ingress and subsequent part "popcorning" or heat damage is governed by the IPC-J-STD-033 Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices, whereas the board moisture ingress and its potential measling or delamination is governed by the IPC-1601 PCB Handling and Storage Guidelines. Along with the components mounted to the PCB, in some cases, the boards need to be considered as MSD devices.

In the case of PCB rework, the component to be removed is typically going to be discarded, which means there is no concern with the removed part having moisture damage. If the device is to be removed and sent back for further failure analysis, then baking of the board/parts is recommended. If you look at the rework guidelines from TI, Kyocera, Intel and other major component vendors, you will find that they recommend to bake the boards and parts prior to another reflow. In addition, care must be taken to make sure that parts that are temperature-sensitive, such as batteries, capacitors, plastic connectors and under filled or glued components, may need to be removed or protected correctly.

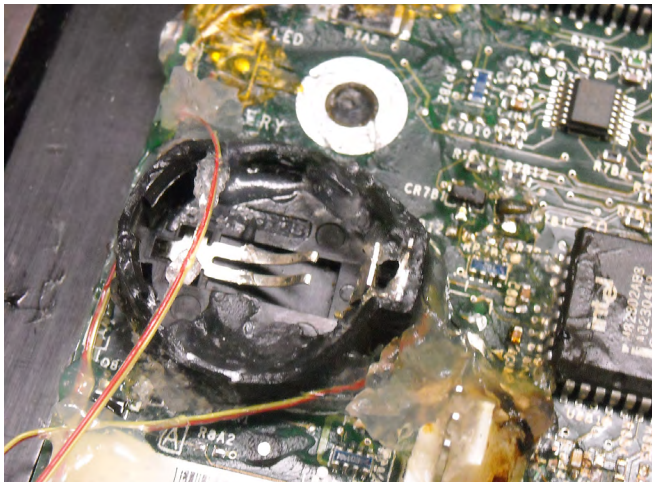


Figure 1: Shielding gel protects heat-sensitive components in areas close to rework.

As part of the process the board rework areas should be properly masked and shielded so that the components are protected from reflow temperatures. There are a variety of materials that can serve to protect neighboring devices (Figure 1) in order to make sure they remain unharmed during the rework process both from a parts removal and replacement standpoint. Shielding of the components so that they remain undamaged by localized reflow temperatures may be necessary depending upon their proximity to the reflow location. Other reflow sources or complete part removal are other strategies that will protect the components from thermal damage or MSD exposure.

After baking is complete, make sure to properly tag and mark the board or parts such that the exposure history can be documented. Place these components or boards into a dry environment such as a dry box in order to house the parts or boards prior to being exposed to the environment again. An alternative is to place them in a moisture barrier bag, and putting desiccant and a moisture indicator card on the inside prior to sealing in order to have solid MSD controls in place.

Review of the IPC/JEDEC J-STD-033C

The J-STD-033 describes the handling, packing, shipping and use and reflow of moisture-sensitive electronic components on PCB assemblies. Keep in mind the handling of the components themselves and how sensitive to heat damage they may be by referring to the IPC-J-STD-075, Classification of Non-IC Electronic Components for Assembly Processes.

MIRACO

Engineering And Providing Balanced Interconnect Solutions

Miraco, Inc. • Manchester, NH • miracoinc.com • 603-665-9449

...connecting technology

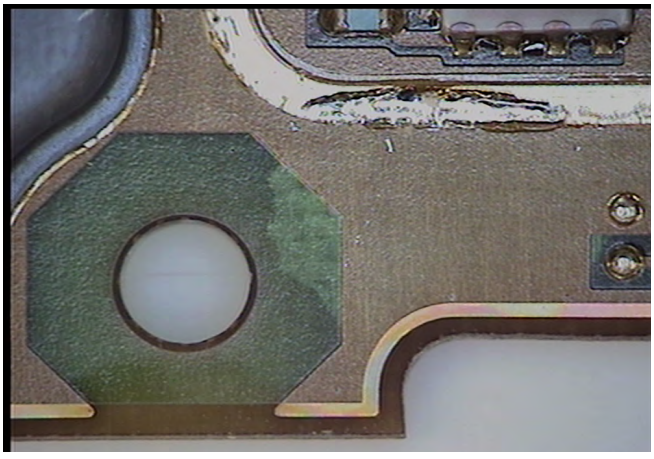


Figure 2: Resultant delamination near rework area on PCB.

Generally, the recommended bake conditions for SMT packages are 8–24 hours at 105–125°C, but consult with IPC J-STD-033 for greater details and specificity. Don't forget that a partial bake is sometimes worse than no bake at all because initially the ingressed moisture is being pushed towards the critical interface at the center of the package.

Review of IPC-1601

The IPC-1601 standard describes the MSD protection and moisture ingress mitigation of the PCB itself. It prescribes exposure times and bake cycles. Even though it's localized, if there is moisture globally in the board, you could damage it at or near the site where you are applying the solder reflow temperatures. Typically, boards will show measling or delamination (Figure 2) locally near the rework heated area if there has been moisture absorption. Internal defects such as partial cracks and delaminations cannot be detected with standard test and inspection tools, but they will reduce the MTBF and increase the risk of early life failures.

The key concern in baking PCBs will be impact on the solderability of its surface finish. Some surface finishes are much more robust than others. For example, organic solderability preservative (OSP) and immersion silver (ImAg) or tin (ImmSn) surface finishes may not do as well as electroless nickel immersion gold (ENIG) or hot air solder leveling (HASL) surface finishes. There are various IPC documents that provide guidelines for baking PCBs of different surface finishes. As a default, use a 150°C 24 hour bake-out cycle with the above caveats.

Controls

There are various controls that should be in place as part of PCB rework having to do with MSD controls. Make sure the trays, reels or boxes of parts clearly identify the parts and boards, the floor life, MSD level and make sure that they are time-stamped. Also make sure to maintain the association between the log sheet and the individual tray, reels and parts. Keep track of cumulative exposure time associated of both replacement components and boards. Be aware that the floor life clock is not reset by reflow. Make sure that the remaining floor life of MSDs is tracked for assembled on boards for double-side reflow and rework. When the factory ambient conditions exceed 30°C, 60% RH, the floor life indicated on the MS label is no longer applicable. In this case, the floor life must be de-rated. These are some of the basic elements that need to be in place for an effective verifiable MSD program.

Conclusion

Depending on the construction of the PCBs and whether or not the parts on the PCB contain MSD devices, boards may need to be baked prior to rework. Care must be taken to make sure the elevated temperatures can be taken by both the PCB and the components on the board fall within the temperature withstand guidelines of both. If the board or the parts are baked, the subsequent handling and protection of MSD devices needs to be assured. The simplest is to consider the board another MSD device in which the investigation of the moisture sensitivity level and appropriate protection outlined in IPC-1601 will tell you how to handle the assembly in rework. If there is no danger to the board or parts in the bake-out process, let the axiom "when in doubt bake it out (at the lowest temperature)" apply for modern SMT assemblies. **SMT**

Bob Wettermann is the principal of BEST Inc., a contract rework and repair facility in Chicago.

thepcblist

THE best way to find a PCB fabricator, anywhere.

PRINTED CIRCUITS INC.

Minneapolis, Minnesota, U.S.A.

Website | Contact Me | RFQ

Printed Circuits Inc

THE FUTURE OF RIGID FLEX CIRCUITRY

Overview | Contact | Specs | About | Videos | Photos | Brochures | News

Printed Circuits Inc. specializes in manufacturing high reliability rigid-flex and multilayer flex boards for medical, military and other critical electronics packaging applications.

We have the expertise to bring these components together, and to help companies bring their advanced designs to market. Both our customer base and the markets that we serve have expanded. Rigid flex is becoming a popular solution for difficult packaging solutions, due to size, weight and reliability constraints.

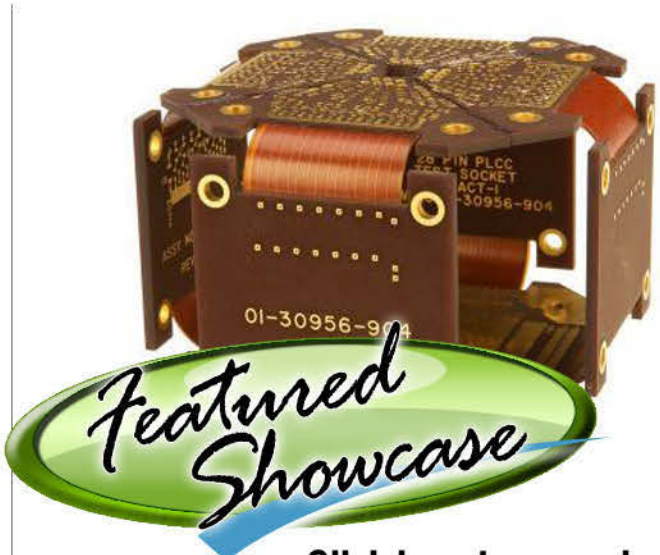
Markets: Communication, Computers, Industrial, Medical, Military/Aerospace

Board Types: Flex, Rigid-Flex

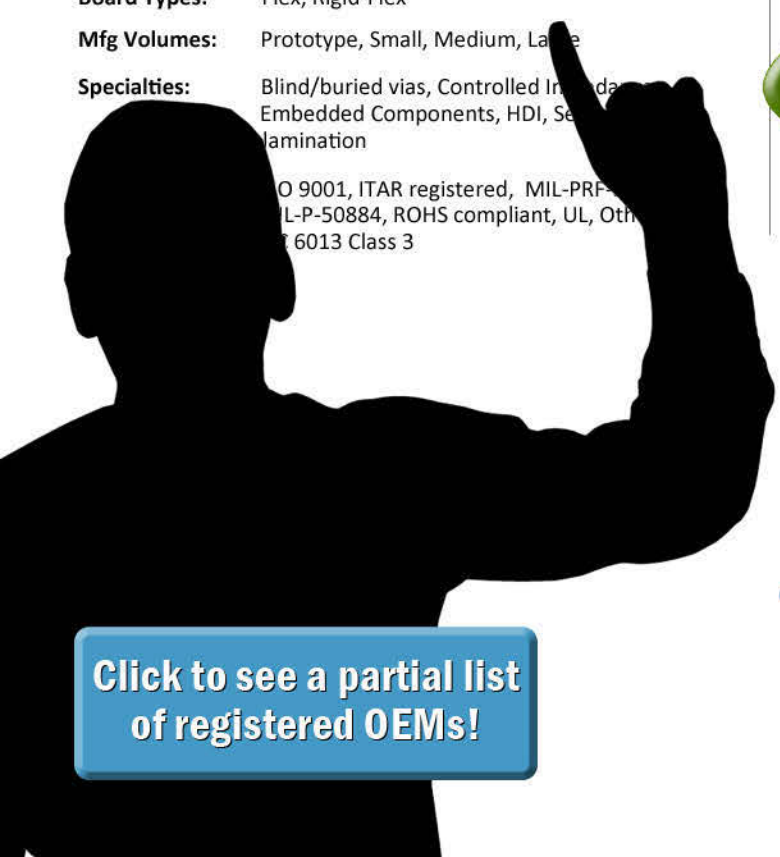
Mfg Volumes: Prototype, Small, Medium, La[illegible]e

Specialties: Blind/buried vias, Controlled Im[illegible]da[illegible] Embedded Components, HDI, Se[illegible] lamination

[illegible]O 9001, ITAR registered, MIL-PRF-[illegible] [illegible]L-P-50884, ROHS compliant, UL, Oth[illegible] [illegible] 6013 Class 3



Click here to see a demo



Click to see a partial list of registered OEMs!

Why YOU should Showcase:

- Capabilities listing for advanced search functionality
- Specialties and certifications listing
- Ability to upload brochures and videos
- Quick and easy “Contact” and “RFQ” buttons
- News, web and contact links

www.thepcblist.com

The Reliability Factor in Solder Paste Printing

by Stephen Las Marias
I-CONNECT007

Knoll Evangelista is the director of the EMS Components and Group Operations of Laguna, Philippines-based electronics manufacturing services firm EMS Components Assembly Inc. With more than 25 years in the electronics assembly industry, Evangelista has seen the evolution of the soldering process—from adhesive component mounting with the solder wave machines to screen printing with reflow ovens. He had also set up over 35 SMT lines for high volume production of products ranging from storage devices, consumer electronics, automotive electronics, and medical devices.

In an interview with *SMT Magazine,* Evangelista speaks about the solder paste printing challenge, what factors impact the process, and best practices to consider.

***Stephen Las Marias:** From your perspective, what are the biggest challenges when it comes to solder paste printing?*

Knoll Evangelista: Reliability of solder joints in electronic products is the biggest challenge, and it depends on the following factors: stencil printing and quality of reflow soldering.

***Las Marias:** How does the solder paste material impact the process?*

Evangelista: The size of the solder paste granules has an impact on the process as it depends on components to be mounted. The pitch distance of ICs, the ball array size, and the distance of BGAs have to be considered in selecting the solder paste material.

***Las Marias:** How do the tighter tolerances and even narrower PCB lines and spacing affect the solder paste printing process?*

Evangelista: More than 50% of defects in assembly of printed circuit boards are attributed to the solder paste printing process. The volume of paste deposit during screen printing is a big factor impacting the quality of solder joints. It requires design of experiments (DOE) to get the right parameters of screen printing machine. It also involves optical inspection to measure the volume of paste deposit to assure consistency of paste application. The surface mount components such as chip scale packages (CSP), fine pitch BGAs are the challenges in stencil printing and reflow profiling capabilities. Due to tighter tolerances, 3D X-ray inspection is required, which offers capabilities beyond those of microscopes and naked eye.

How can you find the best PCB Partner? We can tell you. We wrote the book on it.

Since 1985, U.S. Circuit has been a premier supplier of both commercial and military Printed Circuit Boards in the United States. We know what it takes to be a good PCB partner. And we want to share what we've learned with you.

Choosing a PCB partner isn't hard—if you know what to look for. We've broken it down into "The 5 Commandments" and you can download it for free.

We are proud to be the Circuit Board Manufacturer of more than 400 growing companies, and even prouder to be a good partner for all our customers!

U.S. CIRCUIT 30th ANNIVERSARY 1985 ★ 2015

The Five Commandments: Finding the best PCB Partner

Click to Download

U.S. CIRCUIT

U.S. Circuit, Incorporated
2071 Wineridge Place
Escondido, CA 92029
+1 (760) 489-1413
www.uscircuit.com

ISO 9001:2008 REGISTERED · DEPARTMENT OF DEFENSE · ITAR Registered

Las Marias: *Aside from miniaturization, what other industry or market trends are affecting the solder paste printing process?*

Evangelista: The automotive electronics industry has grown significantly. Most of the controls and sensors in cars nowadays are dependent on electronics assembly. The medical electronics market is also forecasted to grow. The common challenge for these industries is reliability of solder joints.

Las Marias: *What can you say about the advantages and disadvantages of solder jet paste printing and screen printing?*

Evangelista: Screen printing has the advantage in terms of speed especially for high-volume products, while solder jet paste printing deals with accuracy in solder paste deposit. Another advantage of solder jet paste printing is for prototyping that don't require lead time in stencil fabrication. Screen printing best fit high-volume, low-mix (HVLM) and high-volume high mix products (HVHM), while solder jet paste is suitable for low-volume, high-mix (LVHM) and low-volume, low-mix (LVLM) products.

Las Marias: *Please give a list of best practices for solder paste printing. What factors do users need to consider when it comes to solder paste printing?*

Evangelista: Design for manufacturability (DFM) is a compilation of best practices that has been developed through design of experiment (DOE) and industry practices. These are the factors to be considered:

- Stencil aperture design
- Stencil aperture opening (Ratio of paste volume and component solder area)
- Stencil thickness
- Solder paste selection (Alloy composition/ granule size)
- Printer parameter setting (Pressure/ squeegee speed/cleaning capability)
- Squeegee material
- Reflow oven selection (8 zones/10 zones)
- Reflow profiling (speed/temperature/ nitrogen)

Las Marias: *What do you think is the future for solder paste printing?*

Evangelista: Scalability is always the challenge to bring down cost. Solder paste printing will still be the future, and nothing can compete when it comes to high volume production.

Las Marias: *Thank you very much, Knoll.*

Evangelista: Thank you. **SMT**

Discovery Could Energize Development of Longer-Lasting Batteries

Dr. Kyeongjae Cho, professor of materials science and engineering in the Erik Jonsson School of Engineering and Computer Science at UT Dallas, has discovered new catalyst materials for lithium-air batteries that jumpstart efforts at expanding battery capacity. The research was published in Nature Energy.

There's huge promise in lithium-air batteries, but despite the aggressive research being done, those promises are not being delivered in real life, according to Cho. His team, including UT Dallas graduate student Yongping Zheng, have demonstrated that this problem can be solved.

Cho and Zheng's new research focuses on the electrolyte catalysts inside the battery, which, when combined with oxygen, create chemical reactions that create battery capacity. They said soluble-type catalysts possess significant advantages over conventional solid catalysts, generally exhibiting much higher efficiency.

Based on that background, Cho and Zheng have collaborated with researchers at Seoul National University to create a new catalyst for the lithium-air battery called dimethylphenazine, which possesses higher stability and increased voltage efficiency.

The catalyst research should open the door to additional advances in technology. But Cho said it could take five to 10 years before the research translates into new batteries that can be used in consumer devices and electric vehicles.

Co-authors on the study included researchers led by Dr. Kisuk Kang at Seoul National University.

Symposium on Counterfeit Parts & Materials

Marriott Inn & Conference Center
College Park, MD

June 28-30, 2016

Do you know if you have legitimate components?

Symposium Topics

Standards

New Technologies

Testing

Technology Solutions

Detection Programs

Sign up online today!

www.smta.org/counterfeit/

Organized By:

calce

SMTA
Surface Mount Technology Association

Media Sponsors:

SMT007

CIRCUITS ASSEMBLY

Selecting a Selective Soldering System, Part 3

by Robert Voigt
DDM NOVASTAR

In the previous two chapters on selective soldering, we covered the different applications well-suited to this technology, and the various types of fluxing methods available. In this column, we'll cover the common types of soldering technologies available, plus nitrogen inerting systems.

Soldering Technologies

It's important to remember that selective soldering does not necessarily replace a wave machine, because wave soldering is still the most efficient method of processing boards with only through-hole components. But selective is essential for a mixed technology board, and depending on the nozzle type used, can replicate the wave technique in a compact way.

Figure 1: Example of typical jet nozzle.

Nozzle Types

Jet (or wave) soldering is similar to wave in that it's uni-directional and provides the same benefits as wave. Its smallest solder diameter is 4 mm, it requires minimal maintenance, and it is fairly low cost. The nozzle type you use will depend on the make-up of the board and the location of SMDs on that board.

For instance, a fairly large row of connectors/leads not in close proximity to SMD components could use a wide nozzle to swipe (or wave) the entire row at once. A small area closed situated to an SMD would require a very small nozzle to avoid disturbing the surface mount device.

Jet nozzles attack the board at about the same angle (7°) as a wave machine and can deliver a high volume of solder using a tapered tip which guides solder roll-off in one direction returning unused solder back to the pot from its trailing edge. In this scenario, the direction of travel (board or nozzle) is very important.

Jet nozzles are long-lasting and usually cost only a few hundred dollars. Some boards can benefit from a custom nozzle to solder an area as quickly as possible in a dip or drag process, or even a min-wave.

- ***Pros:*** Many nozzle types, including custom configurations, can be used for most efficient speed and high quality, if the board population permits it
- ***Cons:*** Boards (or solder pot) can only move in one direction making programming a little more complex

Wettable nozzles create a uniform bubble by virtue of the surface tension of the solder. Excess solder rolls off around the entire surface of the nozzle, rather than in a single direction, thus producing less turbulence from solder roll-

2016 Programs

www.ipc.org/2016-events

June 6

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements — Conference

Boston, MA USA

June 8

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements — Conference

Chicago, IL USA

June 10

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements — Conference

Silicon Valley, CA USA

June 29

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

July 19–20

IPC Technical Education — Workshop

Chicago, IL USA

July 27

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

August

IPC Education Online: Summer School — Webinar

August 31

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

September 21

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

September 21–23

IPC India 2016 — Conference & Exhibition

Co-located at electronica India and productronica India 2016

September 24–30

IPC Fall Committee Meetings — Meeting

Held in conjunction with SMTA International

Rosemont, IL USA

September 26

EMS Management Meeting — Meeting

Rosemont, IL USA

October 19

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

October 25–27

IPC-SMTA Cleaning and Conformal Coating Conference — Conference

Chicago, IL USA

November

IMPACT Europe 2016 — Meeting

Brussels, Belgium

November 2

PCB Carolina: Regional Trade Show — Conference and Exhibition

Presented by the RTP Chapter of the IPC Designers Council

Raleigh, NC USA

November 2–3

IPC Technical Education — Workshop

Held in conjunction with PCB Carolina

Raleigh, NC USA

November 7–11

IPC EMS Program Management Training and Certification — Certification

Chicago, IL USA

November 16

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
A Vision for the Industry

December

IPC Education Online: Winter Semester — Webinar

December 7–9

HKPCA International Printed Circuit & IPC APEX South China Fair — Conference and Exhibition

Shenzhen, China

December 14

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
Be a Resource for Your Customers, Suppliers, and Team Members: making the Most of IPC Services

February 14–16, 2017

IPC APEX EXPO 2017 — Conference and Exhibition

San Diego, CA USA

Questions? Contact IPC registration staff at registration@ipc.org or +1 847-597-2861

off and helping to preserve solder quality. For this reason, a wettable solder can move in every direction, 360°, or in either the x or y direction, and it is easy to control the bubble height. This combination provides maximum flexibility for a large variety of boards.

Wettable nozzles provide finer accuracy than jet type which makes it better suited to connections in close proximity to SMDs. It also produces less oxidation because there's less contact with air. For this reason, wettable nozzles are also best suited for lead-free solder which tends to be more vulnerable to oxidation.

- ***Pros:*** Provides pinpoint accuracy for highest connection integrity on densely populated boards
- ***Cons:*** Costs more than jet wave and requires daily maintenance to prevent clogging

Custom nozzle capabilities can be made to cover very large areas of a circuit board in a single pass while masking SMD components to speed production.

Contract manufacturers who assemble a wide variety of board configurations should make sure the machine they're investigating can accommodate both types of nozzles and/or custom/hybrid configurations if they think they are ever going to need them.

Which is better: moving the board or the nozzle?

Jet wave nozzles can only attack the board in a single direction, much like wave machines. The design of wettable nozzles permits a flat angle of attack, so the board or the solder pot can move in any direction. Whether you choose a machine that moves the board or the nozzle depends entirely on your process application and the most efficient means to achieve it while retaining as much flexibility for other applications as possible. Most machine manufacturers can recommend the best situation for your

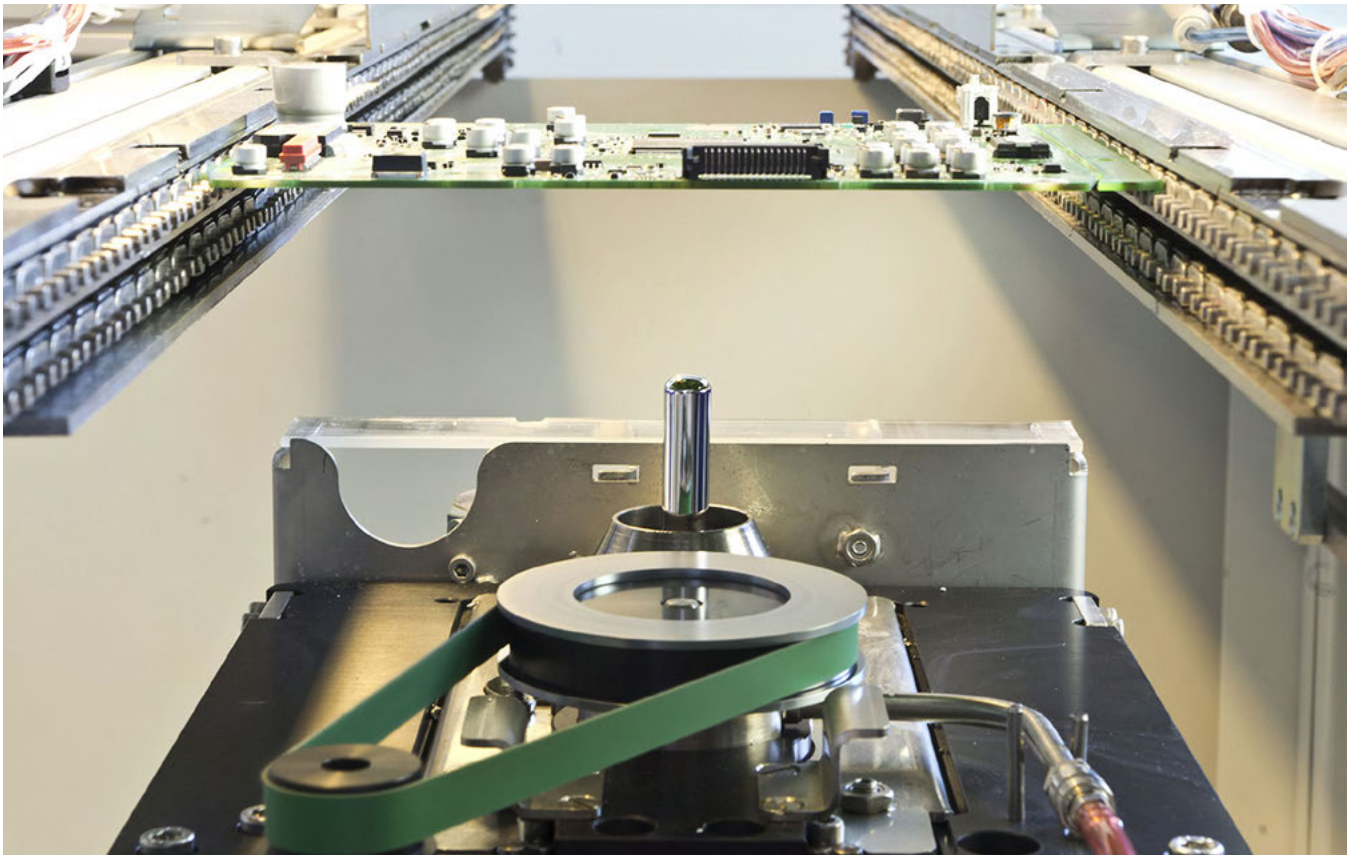


Figure 2: Example of a wettable nozzle.

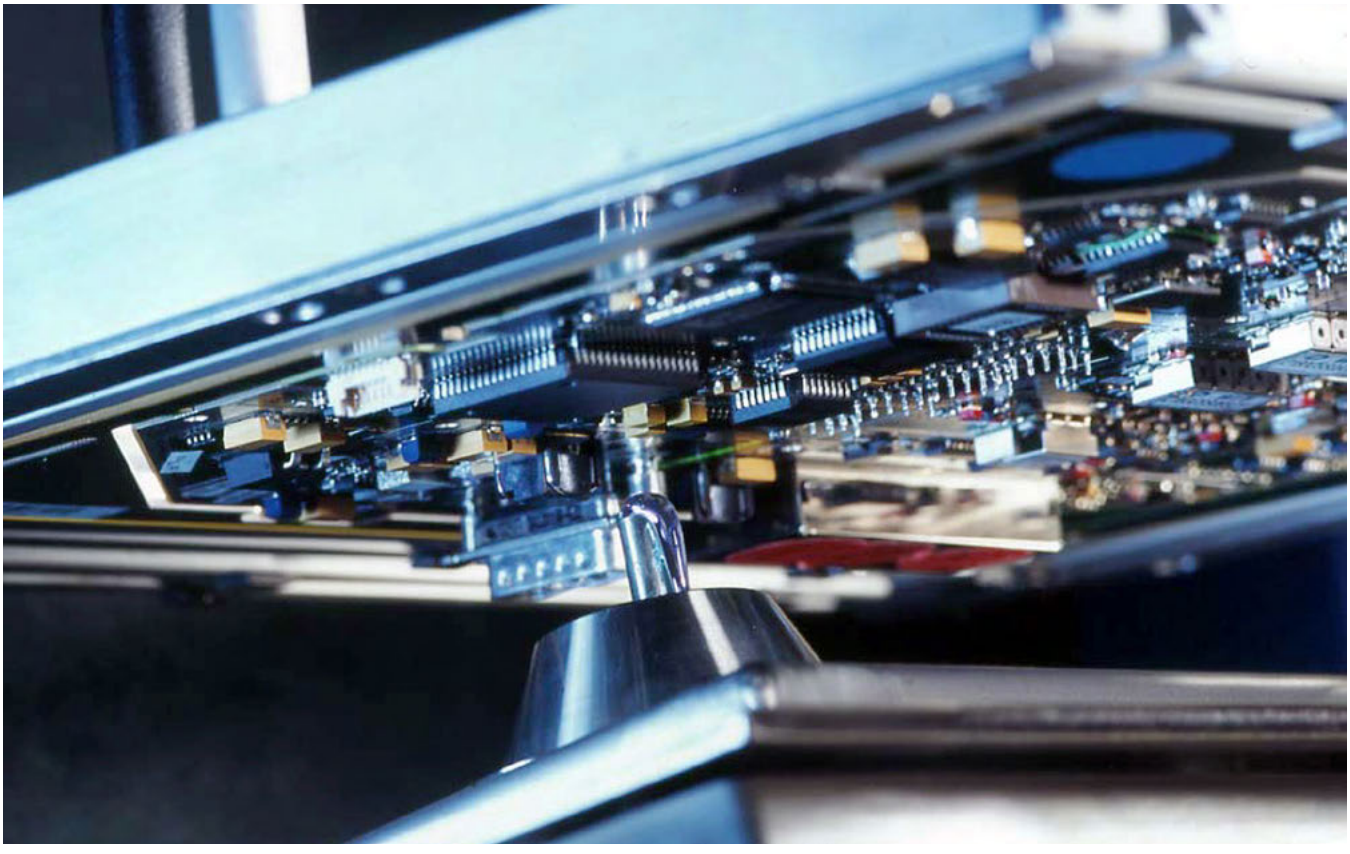


Figure 3: Nozzle position with respect to board.

needs, and can also fabricate custom nozzles for special applications.

Nitrogen inerting is an advisable and low-cost option on a selective soldering machine. Two systems are available: one that uses nitrogen bottles which require replacement and recharging, and those that use a commercially available nitrogen generator. Some nitrogen systems incorporate a pre-heat stage which can also be used to activate the flux, performing two functions at once and eliminating the necessity of a separate flux pre-heater.

Nitrogen improves soldering performance by assisting thermal capability and improving surface tension of the solder. Leaded solder is considerable more forgiving than lead-free in terms of oxidation, degradation and connection quality, so nitrogen is not always needed in those cases. However, it is absolutely necessary for any application using lead-free solder. To ensure a good solder joint using lead-free, the nitrogen bubble protects the solder integrity during the process with no voids in the final connection.

Check References

Remember to consult a variety of machine providers, talk to the manufacturers themselves if possible, and get references to contact before making a purchase. An important consideration for a complex machine such as a selective soldering system and associated options is factory support, specifically training, software, upgrades and spare parts. **SMT**

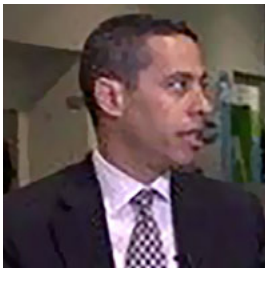
Next column: Programming for selective soldering

Robert Voigt is VP of global sales at DDM Novastar Inc. To reach Voigt, click here.

TOP TEN

Recent Highlights from SMT007

1 RTW IPC APEX EXPO: Mentor Graphics Takes Manufacturing Operations to the Next Level



Dan Hoz, general manager of Mentor Graphics Valor Division, discusses with I-Connect007's Andy Shaughnessy how they are helping electronics manufacturers take their operations to the next level through their Valor IoT Manufacturing solutions.

2 RTW IPC APEX EXPO: Introbotix Highlights HF Tech for Nondestructive Testing

Brian Butler, president and CEO of Introbotix, speaks with I-Connect007 guest editor Bob Neves about using high-frequency test technology for insertion loss testing.

3 Electronics Industry Leaders Urge Action on Pro-manufacturing Policy Agenda in Washington, D.C.

Top executives from leading electronics companies across the United States are calling on Congress to support policies to promote advanced manufacturing.

4 IPC Issues Call for Participation for IPC APEX EXPO 2017

IPC — Association Connecting Electronics Industries invites engineers, researchers, academics, technical experts and industry leaders to submit abstracts for IPC APEX EXPO 2017 to be held at the San Diego Convention Center. Professional development courses will take place February 12, 13, and 16, 2017 and the technical conference will take place February 14–16, 2017.

5 Saline Lectronics Implements Cogiscan's New Factory Intelligence Software

In line with its commitment to Industry 4.0 and a smart, connected factory, Saline Lectronics Inc. has implemented Cogiscan's Factory Intelligence software to enhance its ability to better understand the performance and needs of its SMT production lines.

6 Microsoft and Jabil Collaborate to Create Quality Assurance Platform

Jabil's new predictive analytics platform, built on Microsoft Azure Machine Learning, predicts errors or failures on the assembly floor before they occur, saving its customers time and money while delivering superior quality and shortened product lead times throughout the entire supply chain.

7 Kimball Electronics Reports Q3 FY2016 Results

Kimbal Electronics Inc. has reported a 4% year-on-year increase in net sales for its third quarter of fiscal year 2016, mainly driven by the sequential growth in the automotive sector.

8 EMS Industry M&A Activity Up in Q1 2016

According to Lincoln International's DealReader EMS report for the first quarter of 2016, there were nine completed transactions in Q1 2016, representing an increase in recent M&A activity compared to the seven transactions recorded in the previous quarter.

9 SMTA China Announces Winners of the 2016 Annual Awards

SMTA China held its annual awards presentation at the SMTA China Annual Award Ceremony, which took place April 26, 2016 at the Shanghai World Expo Exhibition & Convention Center.

10 Stadium Forms Wireless Engineering Team to Head New Kista Design Center

Stadium names newly appointed team of wireless design specialists based at new Regional Design Centre (RDC) in Kista Science City, Stockholm, which opened on May 3rd 2016.

SMT007.com for the latest SMT news and information—anywhere, anytime.

Events

For IPC's Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For the iNEMI Calendar, click here.

For a complete listing, check out SMT Magazine's full events calendar here.

IPC Conference on Emerging Critical Environmental Product Requirements
June 8, 2016
Arlington Heights, Illinois, USA

iNEMI Europe Roadmap Workshop/ Webinar
June 9, 2016
Germany

NEPCON West China 2016
June 21–23, 2016
Chengdu, China

iNEMI Asia Roadmap Workshop/Webinar
June 23, 2016
China

Symposium on Counterfeit Parts and Materials 2016
June 28–30, 2016
College Park, Maryland, USA

IPC India 2016
September 21–23, 2016
Bengaluru, India

SMTA International 2016
September 27–29, 2016
Rosemont, Illinois USA

electronicAsia
October 13–16, 2016
Hong Kong

IPC-SMTA Cleaning and Conformal Coating Conference
October 25–27, 2016
Chicago, Illinois, USA

electronica 2016
November 8–11, 2016
Munich, Germany

International Printed Circuit & Apex South China Fair (HKPCA)
December 7–9, 2016
Shenzhen, China

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

EDITORIAL ADVISORY BOARD:
BILL COLEMAN, PH.D., Photo Stencil
HAPPY HOLDEN, IPC and SMTA
CRAIG HUNTER, Vishay
DR. JENNIE S. HWANG, H-Technology Group
MICHAEL KONRAD, Aqueous Technologies
GRAHAM NAISBITT, Gen3 Systems & SMART Group
RAY PRASAD, Ray Prasad Consultancy Group
STEVE PUDLES, IPC
S. MANIAN RAMKUMAR, PH.D., Rochester Institute of Technology
ROBERT ROWLAND, Axiom Electronics
DONGKAI SHANGGUAN, PH.D., STATS ChipPAC Ltd
VERN SOLBERG, Independent Technical Consultant
GARY TANEL, QCG Inc.
TODD SCHEERER, ZESTRON America

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **SHELLY STEIN** shelly@iconnect007.com
MAGAZINE LAYOUT: **RON MEOGROSSI**
AD DESIGN: **SHELLY STEIN, MIKE RADOGNA**
INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**
COVER DESIGN: **SHELLY STEIN**

SMT® (Surface Mount Technology) is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138

©2016 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

June 2016, Volume 31, Number 6 • SMT© (Surface Mount Technology©) is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *SMT Magazine:*

JULY:
Test and Inspection
Test and inspection technologies that will help identify PCB assembly defects for the improvement of electronics assembly processes

AUGUST:
Voices from the Industry
The pulse of the PCB assembly industry: what's new, tips and tricks, market landscape, technology trends, and other things that are good for the industry

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

Subscribe

subscribe.iconnect007.com

007 Media Kit

mediakit.iconnect007.com

EDITORIAL CONTACT

Stephen Las Marias

stephen@iconnect007.com

+63 906-479-5392 GMT+8

SALES CONTACT

Barb Hockaday

barb@iconnect007.com

+1 916 365-1727 GMT-7

pcb design 007 | pcb 007 | SMT 007 | mil aero 007 | flex 007 | ein 007 | RTW

www.iconnect007.com