I-Connect007
GOOD FOR THE INDUSTRY

ENGINEERING SOLUTIONS FOR PCB MANUFACTURING

SMT® MAGAZINE

MAY 2015

Solder Paste Printing: Quality Assurance Methodology

p.28

Enclosed Media Printing as an Alternative to Metal Blades

p.50

PASTE PRINTING & COMPONENT PLACEMENT

Miniaturization with the Help of Reduced Component-to-Component Spacing

by Jonas Sjoberg, et al, page 12

MEPTEC
THE NEXT GENERATION

13th ANNUAL MEPTEC

MEMS TECHNOLOGY SYMPOSIUM

Enabling the Internet of Things:

Foundations of MEMS Process, Design, Packaging & Test

MEMS based products are key enablers in the Internet of Things (IoT) revolution. The availability of large numbers of reliable and cost-effective MEMS sensors and actuators has driven an explosion in the number of IoT applications to reach the market. The Internet of Things is pushing MEMS technology to new levels of performance and bringing forth new requirements.

This event will showcase advances in core technologies that form the foundation of the creation of MEMS-based products. Experts from the field will present the latest innovations in MEMS fabrication processes, packaging, assembly, & test. Insight will be provided as to new technologies, materials and software that will fuel the creation of new devices coupled with traditional MEMS technologies to address new markets and new requirements for the Internet of Things. We explore how the four foundations of MEMS technologies are expanding to include such exciting new techniques as 3D printing, flexible electronics and novel materials.

We hope you will join us to learn from experts about practical, commercial and significant technology advances in MEMS Test, Design, Process and Packaging, each driving key MEMS enabled products. Come hear about the technologies that form the "Sensory System" at the heart of the Internet of Things – it's not just about the network!

Wednesday, May 20, 2015
Holiday Inn San Jose Airport • San Jose, California

KEYNOTE SPEAKER

Creating a Trillion Sensors Based Future
Dr. Janusz Bryzek
Chair, TSensors Summits

Sensors are one of the eight exponential technologies enabling growth of goods and services faster than growth of demand for them. Exponential technologies enable Exponential Organizations (ExO), which demonstrate sales growth to a billion dollars in one to three years. New Exponential organizations are expected to replace 40% of Fortune 500 companies in the coming decade, in a similar mode to Kodak replacement by Instagram in 2012. Presented will an amazing showcase of available sensor based products.

DIAMOND SPONSOR
ASE GROUP

SILVER SPONSOR
Amkor Technology®

EXHIBITING COMPANIES TO DATE		
AGC	APC Advanced Packaging Center	boschman
imt	invenios bonding your future	MILESTONE TECHNOLOGY Failure Analysis Lab
NTK TECHNOLOGIES, INC.	PROMEX INDUSTRIES INC. MICROELECTRONICS ASSEMBLY TECHNOLOGIES	QUALICENT ANALYTICS
silex MICROSYSTEMS	SoftMEMS	tousimis
TYSTAR CORPORATION	ULVAC	unisem

Symposium Committee
- Gene Burk, IMT
- Sean Cahill, Maxim Integrated
- MaryAnn Maher, SoftMEMS
- Ramesh Ramadoss, Microprobe

ASSOCIATION SPONSORS
GSA · MEMS Industry Group® · semi™

SUPPORTING SPONSOR
AMERICAN ELEMENTS
THE MATERIALS SCIENCE COMPANY®

REGISTER ONLINE TODAY AT MEPTEC.ORG

May 2015

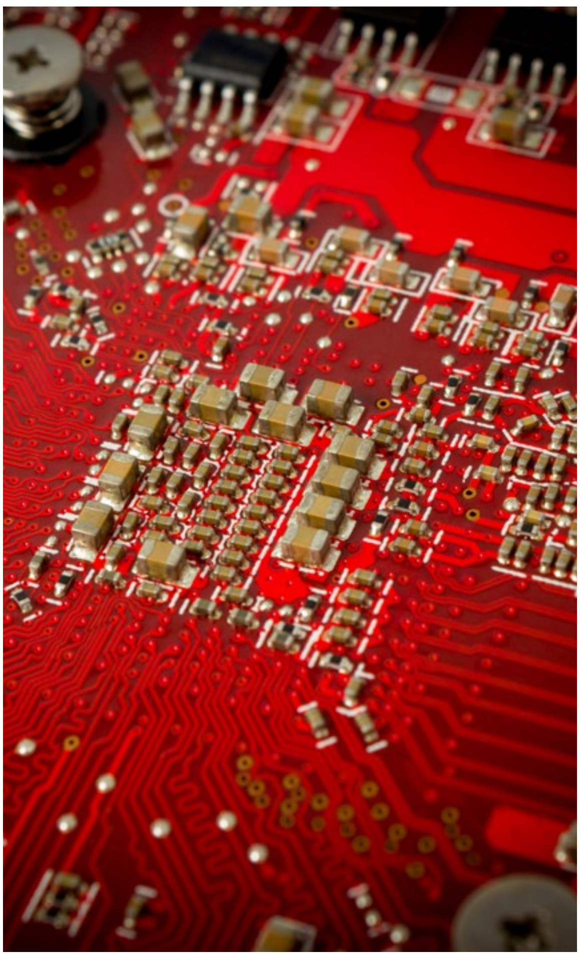
SMT MAGAZINE

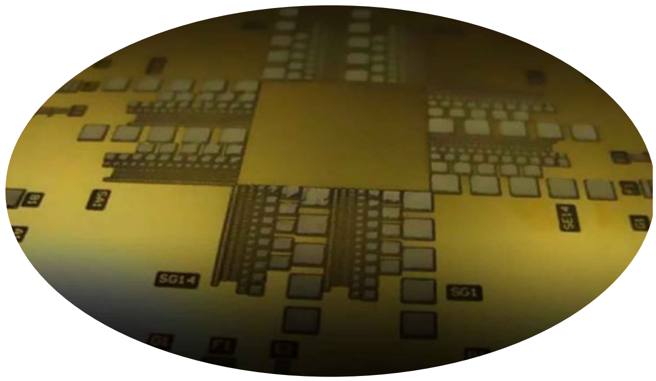
I-Connect007
GOOD FOR THE INDUSTRY

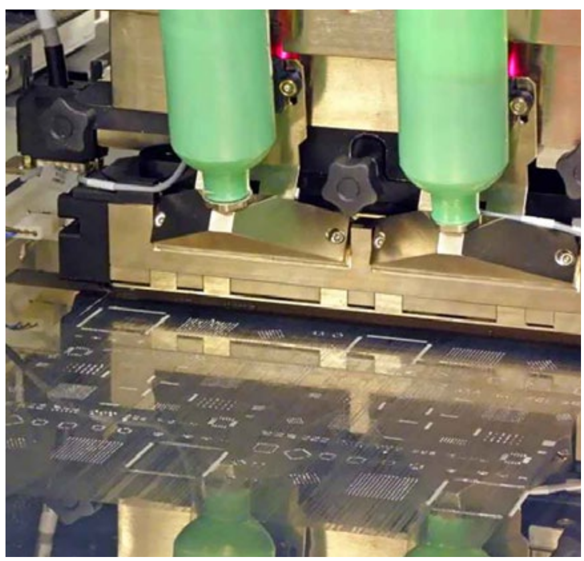
Featured Content

PASTE PRINTING & COMPONENT PLACEMENT

This month, experts from Flextronics, Ericsson, MTEK Consulting, and Speedline Technologies contribute to our features focusing on paste printing and component placement.







The SMT Source™

Latest News About Manncorp SMT Assembly Equipment

May 2015

01005 Capable Print & Place Combos

INLINE SPEED & PRECISION

- Conveyorized 10,500 CPH
- Ball screw-driven print/place
- Clog-free auto-stencil wiping
- Quick setup and changeover

MC-389 Pick & Place **$89,995**
More Info

AP430 Inline Printer **$59,995**
More Info

HIGH ACCURACY WORKCELL

- Most affordable 01005 solution
- 12 mil pitch µBGAs and QFPs
- High-mix, batch production
- 5,000/8,000 CPH (1-head/2-head)

PB2300 Fine-Pitch Printer **$19,995**
More Info

MC-385 Pick & Place **$54,995**
More Info

FREE Installation and Training

Orders placed through May 31, 2015 - US & Canada

Manncorp™

www.manncorp.com

East Coast (Huntingdon Valley, PA) 800.PIK.MANN (745.6266) • West Coast (San Diego, CA) 888.PIK.MANN (745.6266)
Mexico: 52 1 656 217 8215

MAY 2015
VOLUME 30
NUMBER 5
www.smtmagazine.com

SMT® MAGAZINE

ENGINEERING SOLUTIONS FOR PCB MANUFACTURING

CONTENTS


We bring the full range of Electronic Contract Manufacturing services to your fingertip!
ASSEMBLY
KEYPADS
FABRICATION
ENCLOSURES
This is the only place where you would put all your eggs in one basket to get fastest time to market. From concept design to prototype to full turnkey production on all your electronic products.
30th ANNIVERSARY 1985 - 2015
Imagineering
Winner Family Entrepreneurship Award 2014
QUINLAN SCHOOL of BUSINESS
imagineering inc. www.PCBnet.com
847-806-0003 sales@PCBnet.com
Certified Woman-Owned Small Business

COLUMN

SMT PROSPECTS & PERSPECTIVES

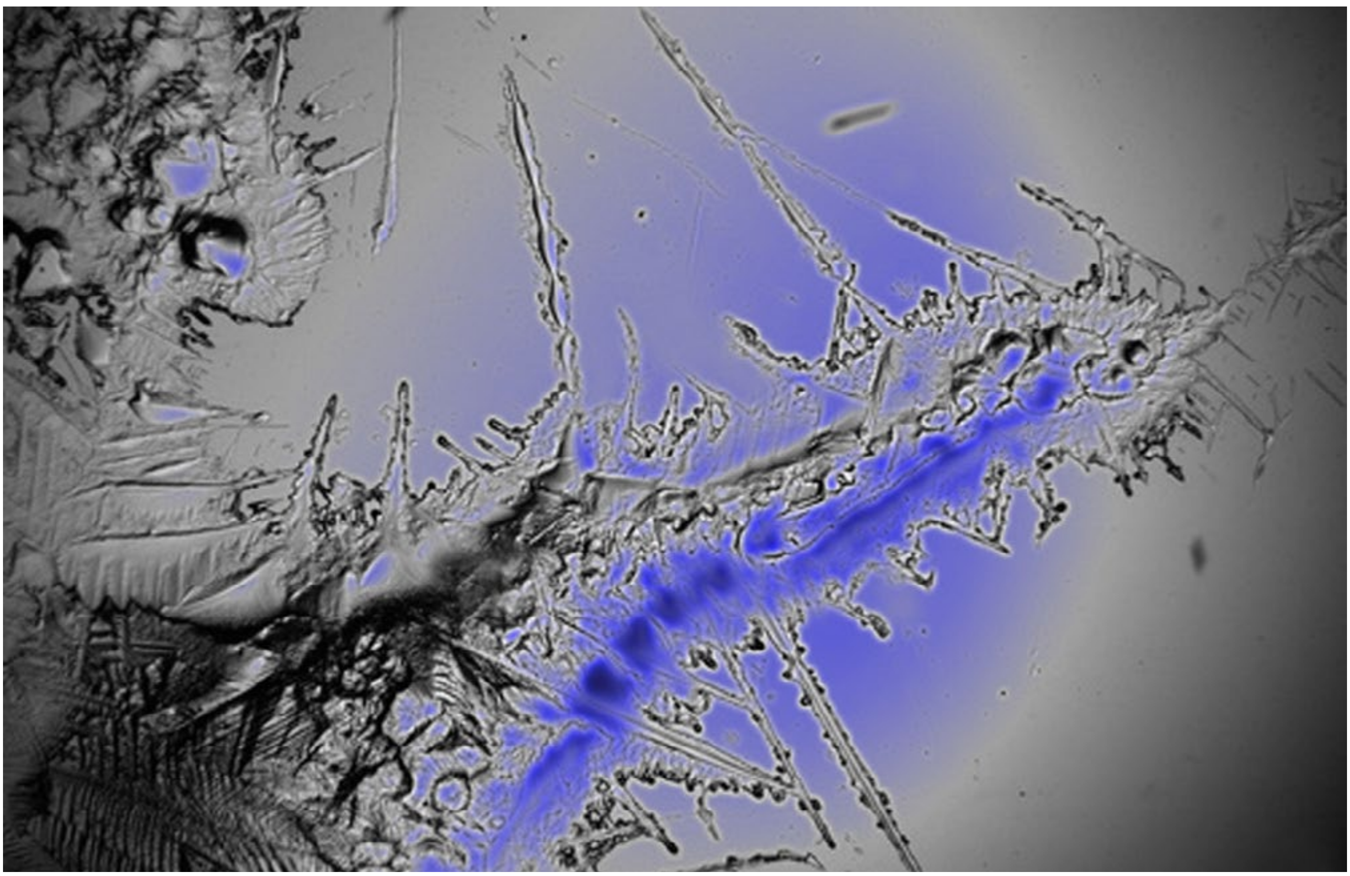
The Theory Behind Tin Whisker Phenomena, Part 1

by Jennie Hwang
H-TECHNOLOGIES GROUP

In my lecture, *Tin Whiskers—All You Should Know,* an audience member expressed the desire to know more about the theory behind tin whiskers and requested that I "spend more time" discussing it. This five-part series, *The Theory Behind Tin Whisker Phenomena,* serves as a response to that request.

Indeed, physical phenomena occur as a result of scientific principles. For tin whiskering, there are reasons and mechanisms behind its occurrence.

Thermodynamics shows us that in the absence of external energy input, nature spontaneously directs to a state of lower energy and stress/strain in a material or a system and the energy tends to be released by making changes over time. Should the fact that whiskers spontaneously grow out of the surface of the tin coating be associated with the change in energy state (stress/strain) in the coating in the direction that lowers its energy state? Basic mechanisms readily release stress/strain without growing whiskers out of the surface. So why do they? Is there a fundamental distinction between the crystal growth within the lattice and the growth out of surface, which protrude like whiskers? Additionally, in order to grow whiskers, there must be a supply of the material (tin atoms). Where and how does the supply come from? And what does it take to allow the supply to move through along a passable path at a rate that is fast enough in a finite time frame?



Before we delve into the practical questions stated above, let's look at well-established findings and observations in the literature, published or unpublished. Studies indicate that whiskers can be mitigated by controlling the plating process in an equivalent way to controlling stresses in materials. The very sharp decrease in internal stress of tin electrodeposits was observed within minutes after plating. This fast stress release can occur regardless of whether initial stress in the deposit is compressive or tensile. In either case, the value of stress drops to very low numbers, but it remains being of the same type as the initial stress form (i.e., high initial tensile stress reduces to much lower stress value, but remains tensile; high compressive stress remains compressive).

It has also been observed that the inclusion of organic elements in tin structure promotes tin whisker growth. Organic inclusion or the level of inclusion is, in turn, affected by the plating chemistry and its control. And the bright tin has exhibited to be most susceptible to whisker formation. It is evidential that recrystallization and grain growth prior to whisker formation have occurred in bright tin deposit, showing large irregular shape grains as the precursor for whiskers.

When comparing between Cu substrate and Ni substrate, Ni substrate tends to retard the whisker formation. This is evidenced by the successful use of Ni barrier layer to mitigate tin whisker in many incidents, albeit not exhaustively. This phenomenon, related to inter-diffusion rate and intermetallic formation, correlates well with the relative diffusion rates between Cu and Sn vs. Ni and Sn. Another observation showed that the external forces exerted to tin plating, such as bending, stretching, torque, scratches, nicks, and exacerbated whisker growth in the physically stressed region.

Still another experiment exhibited that whisker formation involved a shelf time. However, the shelf time varies without a straightforward correlation with temperature, humidity and other environmental conditions. Studies showed that a moderately warm temperature served as the 'greenhouse' that nurtured whiskers, yet a high temperature (e.g., above 150°C) inhibited whisker formation. Furthermore, the highly disparate whisker growth rates have been reported, ranging from 0.03–9 mm/year. It is noted that whiskers can grow even in a vacuum environment.

> "Still another experiment exhibited that whisker formation involved a shelf time. However, the shelf time varies without a straightforward correlation with temperature, humidity and other environmental conditions. Studies showed that a moderately warm temperature served as the 'greenhouse' that nurtured whiskers, yet a high temperature (e.g., above 150°C) inhibited whisker formation."

Against this backdrop, what is the energy driver (or stress/strain gradient) to grow this whisker-like thing protruding from the surface of a coating or metal substrate?

It is interesting to note the fundamental sciences that separate the mechanism between the two most common tin-based materials: solder joint fatigue failure vs. tin whisker. The physical and mechanical behaviors of solder alloys and solder joints have been understood through microstructure examinations in conjunction with mechanical and physical tests. The evolution of microstructure over time and microstructural changes in response to temperature and other external parameters, both environment and in-circuit conditions (if in electronics), provide further understanding of solder's degradation and failure modes. In general, solder joint behavior can be explained through micron-scale mechanisms. However, considering the various phenomena of tin whisker under accelerated conditions, as well as in real-life services, tin whisker calls for a deeper treatment of crystal lattice at atomic level. This goes to the heart of theories of physical metallurgy in crystal nucleation and grain growth from a high energy state to a low energy or stress-free state.

The mechanisms by which tin whiskers occur have been studied over the years through different approaches and concepts, namely, the recrystallization process and stress relief mechanism. And despite the fact that some test results are at variation with the observations, the loosely termed “internal stress” is deemed primarily responsible for the metal whisker formation and growth. As such, the factors that can contribute to internal stress of the tin plating and the conditions that impart additional residual stress to the plating layer during and after plating deposition are the right places to be deliberated.

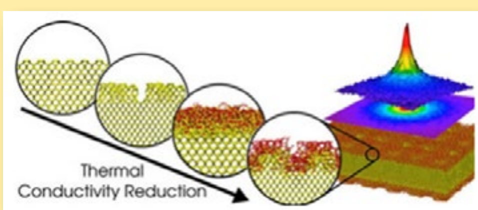
Nonetheless, tin whisker is more than a classical recrystallization and it is more than a classical stress relief phenomena. It is the product of a multi-faceted process.

One plausible theory of tin whisker growth can be postulated through considering the combination and confluence of several key metallurgical processes, which will be outlined in Parts 2–4 of this series. **SMT**

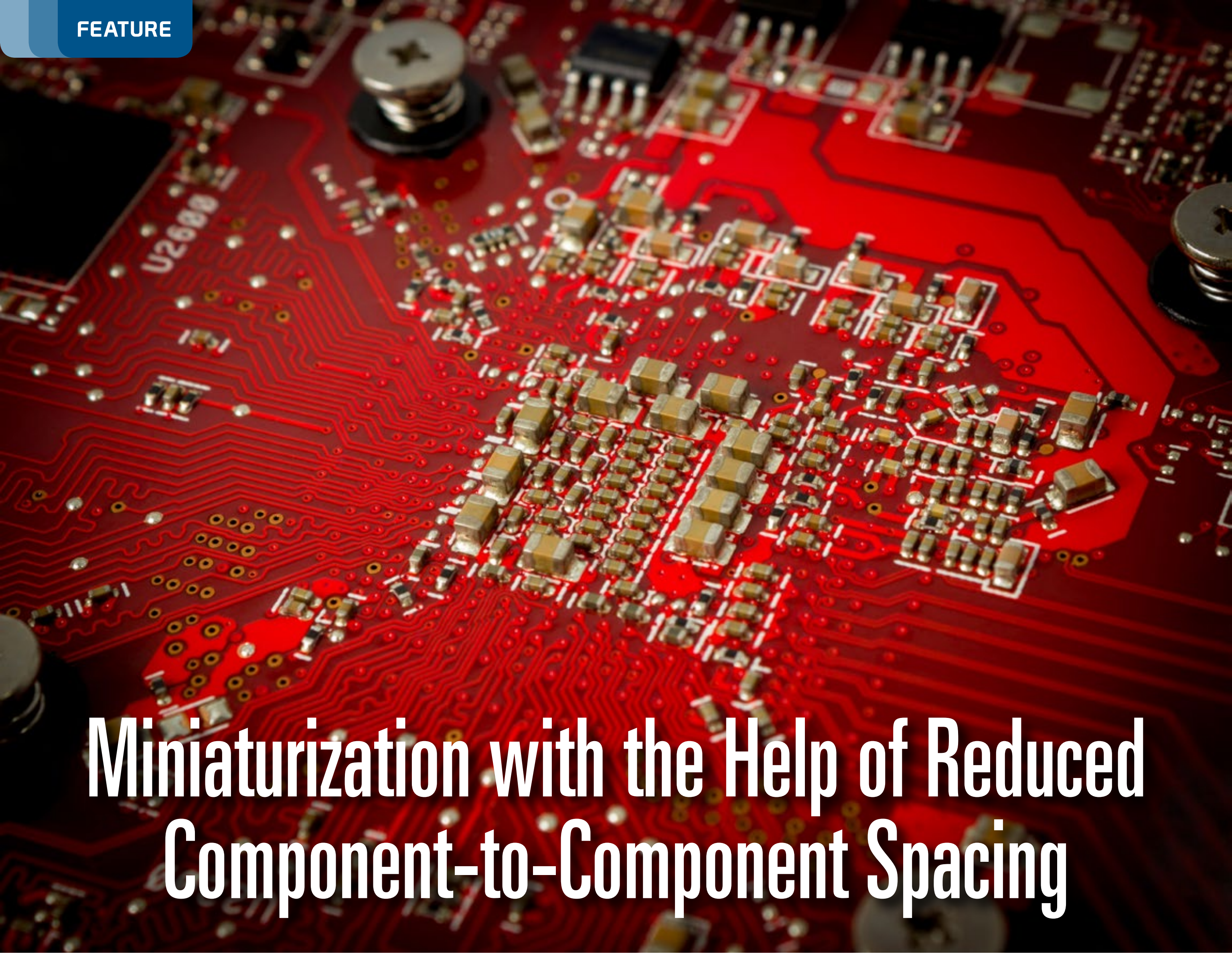
Dr. Hwang, an international businesswoman, speaker, and business and technology advisor, is a pioneer and long-standing contributor to SMT manufacturing since its inception, as well as to the lead-free electronics implementation. Among her many awards and honors, she is inducted to the WIT International Hall of Fame, elected to the National Academy of Engineering, and named an R&D-Stars-to-Watch. Having held senior executive positions with Lockheed Martin Corp., Sherwin Williams Co., SCM Corp, and IEM Corp., she is currently CEO of H-Technologies Group, providing business, technology and manufacturing solutions. She serves as Chairman of Assessment Board of DoD Army Research Laboratory, Commerce Department’s Export Council, various national panels/committees, international leadership positions, and the board of Fortune 500 NYSE companies and civic and university boards. She is the author of 450+ publications and several textbooks, and an international speaker and author on trade, business, education, and social issues. Her formal education includes four academic degrees as well as Harvard Business School Executive Program and Columbia University Corporate Governance Program. For further info, visit JennieHwang.com.
To read past columns, click here.

Flat Silicon Channels Hinder Heat Conduction

A paper published in ACS Nano (“Tuning Thermal Transport in Ultrathin Silicon Membranes by Surface Nanoscale Engineering”) and led by researchers from the Max Planck Institute for Polymer Research (Germany), the Catalan Institute of Nanoscience and Nanotechnology (ICN2) at the campus of the Universitat Autònoma de Barcelona (UAB) (Spain) and the VTT Technical Research Centre of Finland (Finland) describes how the nanometre-scale topology and the chemical composition of the surface control the thermal conductivity of ultrathin silicon membranes. The work was funded by the European Project Membrane-based phonon engineering for energy harvesting (MERGING).



The paper describes how thermal conductivity of ultrathin silicon membranes is controlled to large extent by the structure and the chemical composition of their surface. A detailed understanding of the connections of fabrication and processing to structural and thermal properties of low-dimensional nanostructures is essential to design materials and devices for phononics, nanoscale thermal management, and thermoelectric applications.



Miniaturization with the Help of Reduced Component-to-Component Spacing

by Jonas Sjoberg, Ranilo Aranda, David Geiger, Anwar Mohammed and Murad Kurwa
FLEXTRONICS ADVANCED ENGINEERING GROUP (AEG)

Miniaturization and the integration of a growing number of functions in portable electronic devices require an extremely high packaging density for the active and passive components. There are many ways to increase the packaging density. A few examples would include stacking them with package-on-package (PoP), fine pitch CSPs, 01005 (imperial), and last but not least, reducing component-to-component spacing for active and passive components.

The use of fine-pitch CSP, PoP components and 01005 poses a number of challenges for PCB design, SMT assembly process, and reliability, and by placing them closer together many of these challenges will be magnified. A feasible assembly process must be achieved. The assembly process ranges all the way from screen-printing, placement and reflow soldering in air or nitrogen. Many factors influence the quality of the assembly process and with the reduced pitch and component spacing, the process capabilities for both assembly and PCB fabrication will be tested to its limit and beyond.

In many cases these assemblies also require a rework process either in the manufacturing facility or at repair centers when the product fails in the field during usage. In addition, the correct materials such as PCB material, PCB surface finish, solder paste, dipping flux and PCB design need to be selected to ensure high-yielding, cost-effective and reliable interconnects. Of course, the mechanics of the products makes a big difference as well, but it is very product-de-

pendent. Many of today's products leave little room for designing the mechanics in the most reliable way due to total cost and overall look and size of the products.

This article will discuss different layouts, assembly and material selections to reduce component-to-component spacing down to 100–125 µm (4–5 mil) from today's mainstream of 150–200 µm (6–8 mil) component-to-component spacing.

Introduction

With the neverending drive for smaller, lighter and more advanced features on portable products, the ability to handle miniaturization is becoming a key capability to enable these requirements. Miniaturization can be done in many ways and this article touches on the assembly technologies that can be incorporated in a more or less standard surface mount assembly line with minimal equipment and material upgrades.

Before starting any development work it is critical to understand product and industry requirements and capabilities. If this is not fully understood, no development activities can start. PCB fabrication can be seen as a good example on the importance of understanding requirements and capabilities. If a good quality PCB can't be sourced within the scope of the assembly development project there is no reason to develop an assembly technology process since there is not anything to do the assembly development work on. The key is to ensure that several options for assembly can be achieved and this should be seen as a toolbox of technologies.

For the active components die stacking inside a package is one common way to increase the functionality per unit area on a PCBA, which is very popular for memory devices. However, there can be some drawbacks to creating a stacked die solution. First, this method is a customized solution. If any of the dies to be used changes, the die stack needs to be evaluated to see if changes are needed in the package. For example, a die shrink may occur and this could change the whole package structure. Secondly, if one or more of the dies inside the package fail, the whole unit will have to be scrapped, which would lead to increased cost; this is the well-known compounded yield issue. Lastly, trying to coordinate the many semiconductor suppliers to provide dies to a packaging house for die stacking can be a challenging task and overall responsibility for the complete package yield could in some cases be unclear.

In the PoP process one component is placed on top of another package during a single SMT process to fully utilize the three dimensional aspect of the product. The topside of the bottom component has pads similar to the pads on the PCB for attachment of the top package. Each package is a single unit that can be fully tested as a normal IC package is done today, so the yield would be comparable to the normal yield commonly seen today. Another advantage would be the ability to have second-source options that could be fairly easily inserted into the process. The stacked package can be processed in a traditional SMT environment with a few upgrades that are readily available. Therefore, package stacking enables configurable assemblies and provides greater flexibility in the supply chain. It can be used for memory applications or for a processor with memory, with faster time to market and better management of package testing and compounded yield issues.

Reduced pitch is without doubt one of the bigger challenges for the active components, but it is a very effective way to achieve miniaturization. Today mainstream is 0.4 mm pitch, but 0.3 mm pitch is getting more and more popular. Taking the step to 0.3 mm from 0.4 mm poses a number of challenges mainly related to PCB design, screen printing and getting good quality PCBs. For 0.3 mm pitch, our studies show that screen printing is a big challenge and a dip fluxing process might be needed for some applications. This might sound like a big change, but the process for running an in-line flip chip or package-on-package process is more or less in place since many production lines are already using this process.

With regards to passive components, two very effective ways to achieve miniaturization include using smaller parts such as 0201 and 01005 and reducing component-to-component spacing. Both strategies are very much feasible, but each needs to be carefully considered with the help of analyzing process data and total cost.

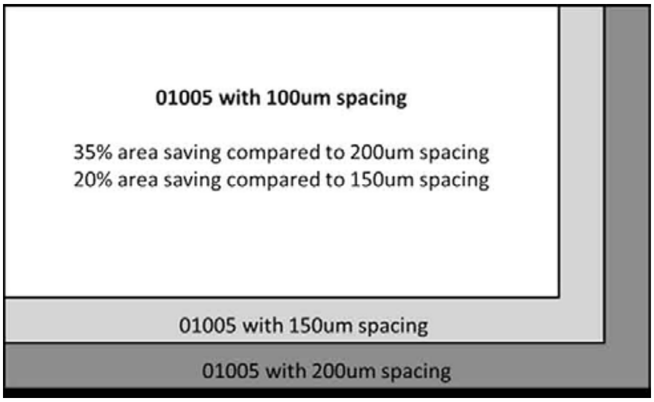


Figure 1: 01005 with component-to-component spacing between 100–200 µm.

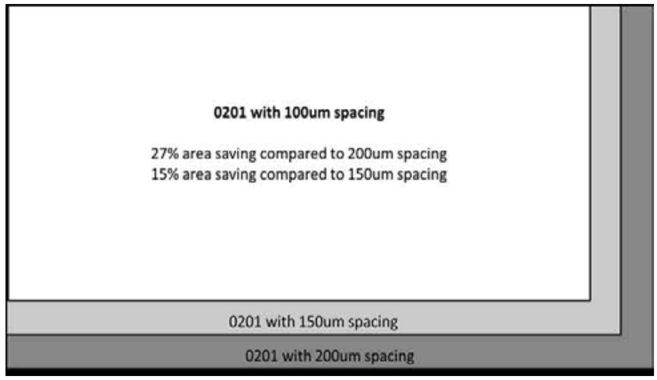


Figure 2: 0201 with component-to-component spacing between 100–200 µm.

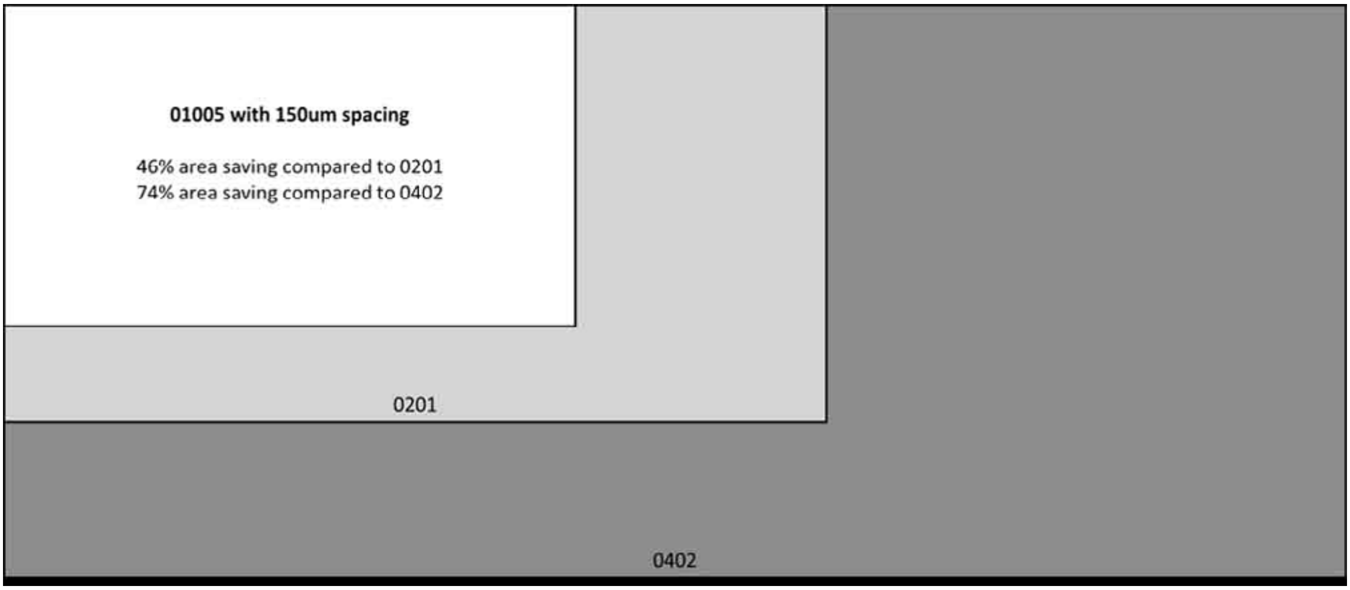


Figure 3: Area saving by using 01005 instead of 0201 and 0402.

Considering the extreme drive for miniaturization and added functionality, it is not surprising that the 01005 packages are becoming increasingly used despite the limited electrical values currently available, the relatively high component cost and the difficulties in the production processes. With reduced component spacing from 0.2 mm–0.1 mm, up to 35% area savings can be achieved (Figures 1,2) and by moving from 0402 to 01005 with 150 µm component spacing a 74% (Figure 3) can be achieved.

To understand what technologies and capabilities that are required going forward it is important to have a well established roadmap process that captures product and industry requirements and availability of components. PCB and materials are equally important. The technology roadmap process typically has four major process steps and each of them is equally important:

- Technology requirement
- Technology roadmap
- Technology development
- Technology deployment

Test Vehicles and Materials

The process development and factory qualification test vehicle for miniaturized assembly technologies (Figure 4) is designed to be similar to a cellular phone using the IPC drop test vehicle JESD22-B111 as a baseline. The company test vehicle is continuously being upgraded based on new findings and technologies; it also serves as a verification vehicle for the “Printed Circuit Board Design Guidelines.”

The PCB panel is 132 mm x 77 mm (Figure 4) and it is made with three identical sections. The surface finish for this board is organic surface preservative (OSP). One side of the PCB is using solder mask defined (SMD) pads and the other side is using non-solder mask defined (NSMD) pads for the active components. The PCB has four layers using halogen-free resin-coated copper (RCC) in the outer layers to enable a better quality for the microvia drilling. Halogen-free FR-4 is used in the inner layers. The total PCB thickness is 0.788 mm (Figure 4).

Due to the high I/O count, small pad sizes and tight spacing the PCB design is very challenging in many aspects.

- ½ oz (20 µm copper), around 33–34 µm (after plating)
- 50 µm copper/copper spacing (inner and outer layers)
- 50 µm solder mask slivers
- 25–40 µm solder mask registration tolerance
- 60 µm uvias
- 0.2 mm uvia capture pads in outer and inner layers
- No silkscreen is used to prevent screen printing issues. Typical silkscreen thickness is around 20 µm (Figure 5)

Copper-filled microvias are used based on previous studies on 0.3–0.4 mm pitch CSPs where we see better screen printing results since the solder paste has a bigger area to adhere against. Furthermore, the so-called microvia induced voids more or less disappear on bumped connections such as CSPs and BGAs. The negative impact of copper filled microvias is increased PCB price.

The PCB includes a number of different component types, including:

- Passive parts: 01005, 0201, 0402, 0603 and 0805
- Active parts: 0.4 mm pitch PoP, 0.4 mm pitch CSP and 0.3 mm pitch CSP

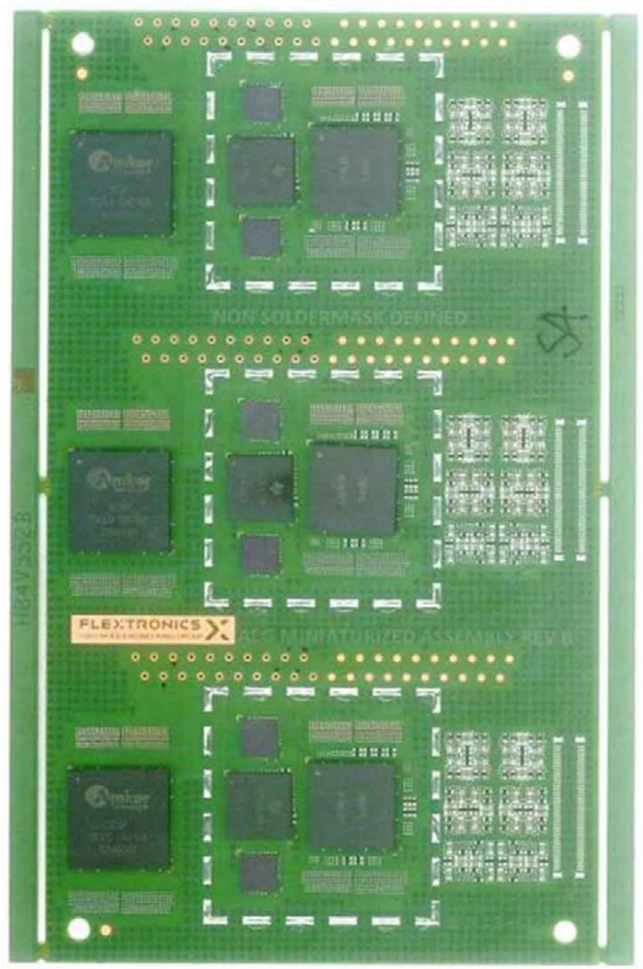


Figure 4: Miniaturized assembly test vehicle.

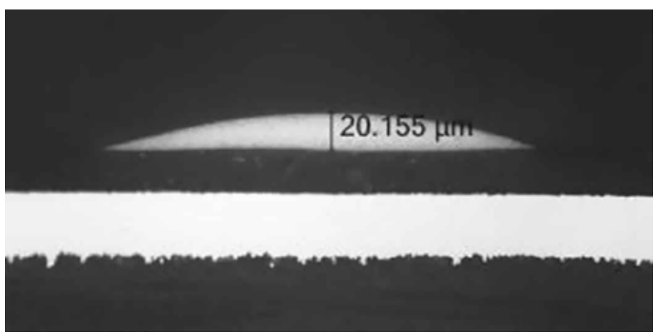


Figure 5: Silkscreen on top of solder mask.

The PCB has different sections with various component spacing within the same component type, and with mixed component types, the spacing ranges between 100–200 µm. Due to the narrow component and pad spacing, solder mask between the pads is only feasible down to 150 µm pad-to-pad spacing.

The main focus on this study have been to reduce the component spacing between passive parts since this is the main challenge due to solder bridges. Passive components next to CSPs, and CSPs next to CSPs, present minor assembly issues and rework concerns.

Solder Paste Selection

A type 4 SAC 305 (Sn96.5Ag3Cu0.5) halogen-free solder paste was used for all our studies related to miniaturized assembly technologies. The selection of solder paste was done through extensive evaluations on printability, voiding, slumping, solder balling, wetting and SIR (surface insulation resistance). For all technologies included on the miniaturized test vehicle the common challenge is solder paste printing and the 0.3 mm pitch CSP and 01005 locations use very similar apertures sizes.

Assembly Details

Assembly trials were conducted under controlled production environment. During the assembly trials, dip fluxing was used for the PoP and 0.3 mm pitch CSPs that were mounted on the board as well. All components were mounted at high speed to simulate a true production environment.

Paste Printing and Solder Paste Inspection

A standard paste printing process was used with a 3 mil (75 µm) and 4 mil (100 µm) thick laser-cut stencil, which was electro-polished from a local supplier in Asia where most of the production of miniaturized products would be produced. Automated solder paste inspection was done on all locations to collect data and automatic stencil cleaning was done after each print mainly due to the 01005 and 0.3 mm pitch

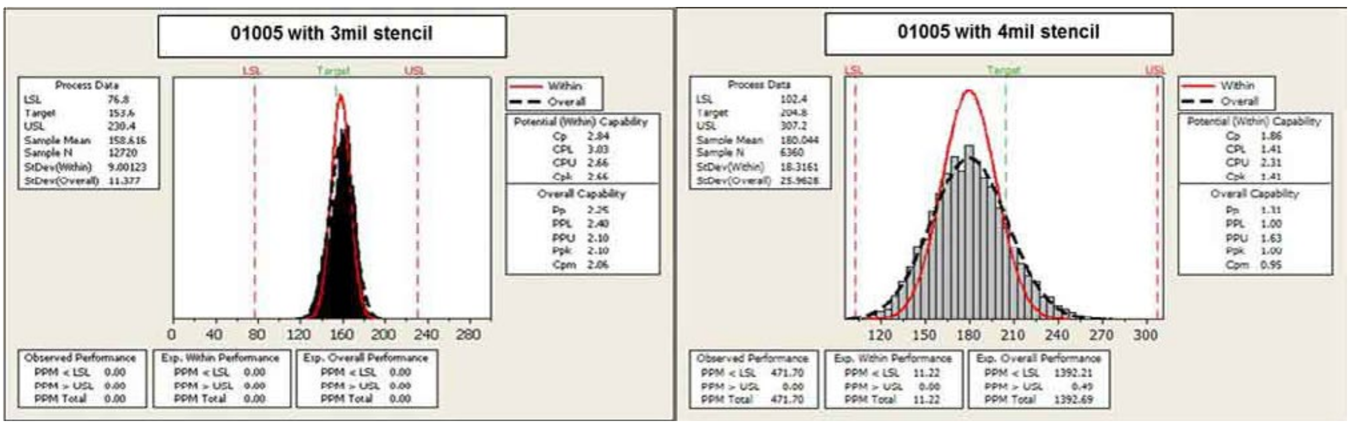


Figure 6: Volume Cp, Cpk, Pp and Ppk with 3 mil and 4 mil stencil thickness.

Stencil Thickness (mil)	AR	Cp	Cpk	Pp	Ppk
3	0.66	2.84	2.66	2.25	2.1
4	0.5	1.86	1.41	1.31	1.00

Table 1: Cp and Cpk with 3 mil and 4 mil stencil thickness.

CSPs. In true production the cleaning frequency could potentially be reduced but this would be product dependent. The solder paste volume criteria were set to 40–120% of the theoretical value.

All the Cp and Cpk calculations used statistical software. True Cp and Cpk using this software are Pp and Ppk since this is the overall calculated capability (Figure 6 and Table 1).

Using a 3 mil (75 µm) stencil gives a much higher Pp and Ppk compared to a 4 mil (100 µm) stencil (Figure 6) and this is expected due to an area ratio (AR) of 0.5 with a 4 mil (100 µm) stencil and 0.66 with a 3 mil (75 µm) stencil.

All solder paste deposits with a 3 mil (75 µm) stencil were within specification showing 0 DPMO print failures. The solder paste printing with a 4 mil (100 µm) stencil showed a DPMO of 1382 on 01005 but these PCBs were still assembled and the solder joints still met IPC 610E standard with regards to solder joint quality.

Pick-and-Place Details

All the assemblies were done on standard fine pitch surface mount machines equipped with an accuracy of 40 µm at 3 Sigma. The flux dipping unit used was the linear dipping unit which comes with replaceable dipping plates. No assembly-related defects were detected during for the 0.3 mm pitch CSP or 0.4/0.4 mm pitch PoP components once the correct process parameters were set.

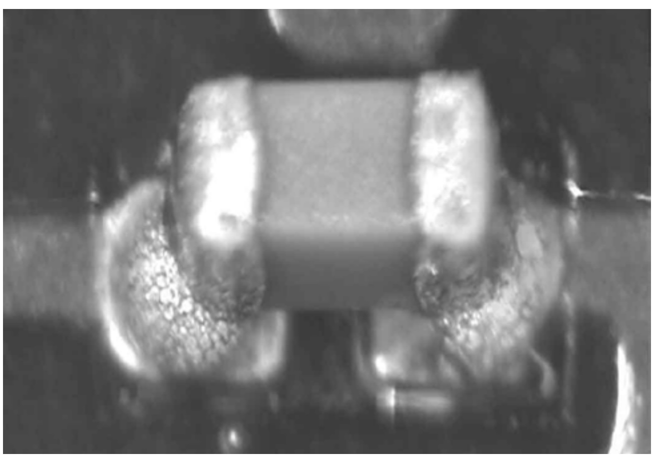


Figure 7: 01005 reflowed in air with a ramp below 1.0°C/s, between 180–217°C.

Reflow

A standard lead-free reflow profile was used. The reflow was done in air and nitrogen, with 65 seconds above 217°C and 245°C peak temperature. Our previous studies on 01005 showed that a fast ramp of 1.0°C/s and above between 180–217°C gives better wetting when reflowed in air but the same basic process was used both for the air and nitrogen reflow in this study. Graping/incomplete wetting is a clear sign of too slow a ramp between 180–217°C (Figure 7).

Rework Process

Rework becomes more challenging with reduced component-to-component spacing, but if the process is set up correctly the amount of rework should be at a minimum. A standard rework process was used for all components on the PCB. For the passive components manual rework was done with help of fine tip tweezers and a hot air blower. For the CSP and PoP a BGA repair machine was used.

Results

The DPMO values (Figure 8) show a very clear breakpoint between 100–125 µm component-to-component spacing and it is also shown that the stencil thickness has a major impact at the smaller component-to-component spacing

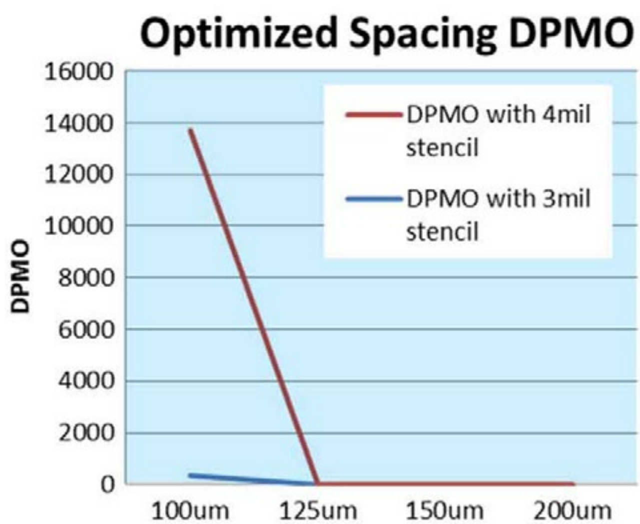


Figure 8: DPMO at different component-to-component spacing with 3 mil and 4 mil stencil thickness.

but that the breakpoint is the same. The usage of nitrogen shows no impact with regards to spacing DPMO.

All defects were related to solder bridges in between passive components and no issues were seen between passive components and CSP/PoP components (Figures 9, 10).

During cross-sectioning and visual inspection of the air reflowed samples it was noticed that some of the 0.4 mm pitch PoP components showed head-on-pillow (Figure 11) and that the solder joint did not wet the whole pad. This was expected and proven in earlier builds, but it was decided to still run in air reflow to be able to validate potential spacing differences between air and nitrogen reflow. The samples reflowed in nitrogen did not show any issues for the 0.3 mm pitch CSP and PoP components.

For the 01005 components the wetting is not as good in air reflow compared to nitrogen reflow, but still considered as acceptable according to IPC-610 E standard (Figures 12, 13).

Conclusions

There are many ways to achieve miniaturization and the key is to have a toolbox of technologies to fulfill various requirements. Depending on the product, several options

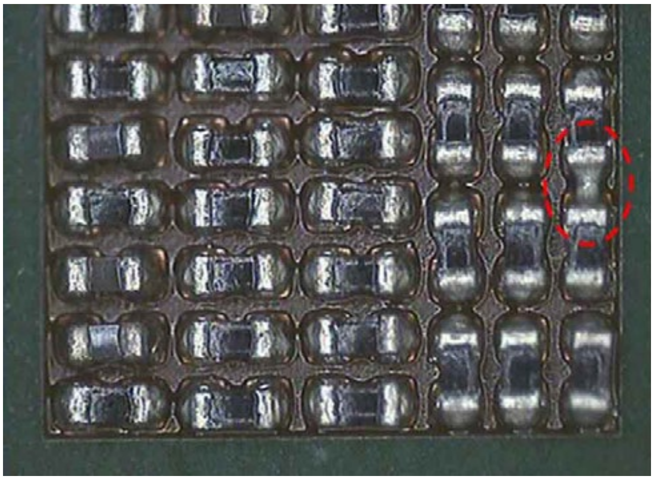


Figure 9: Solder bridge on 01005 at 100 µm spacing.

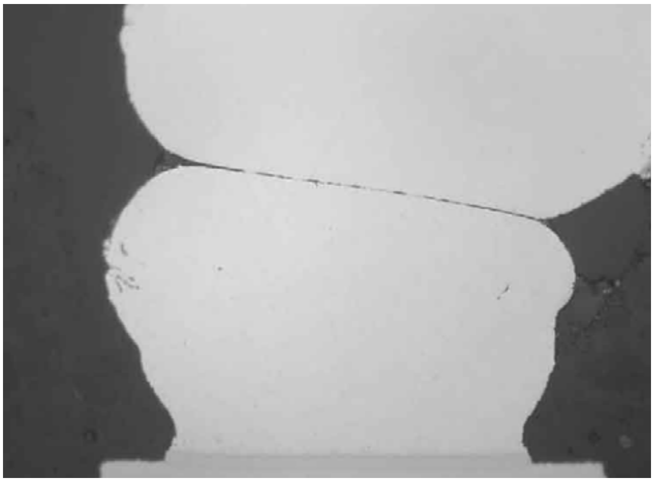


Figure 11: Cross-section of a 0.4 mm pitch top PoP reflowed in air.

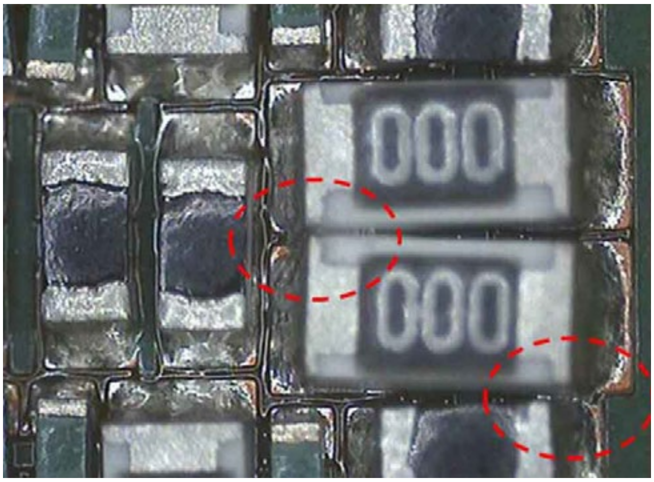


Figure 10: Solder bridges in the mixed spacing section.

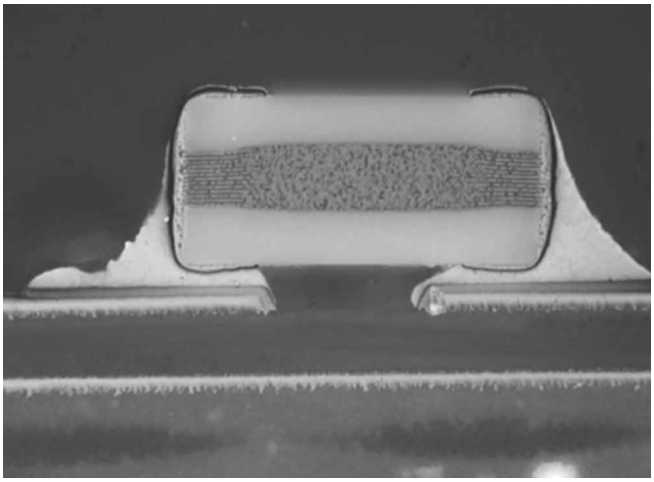


Figure 12: Cross-section of a 01005 component reflowed in air.

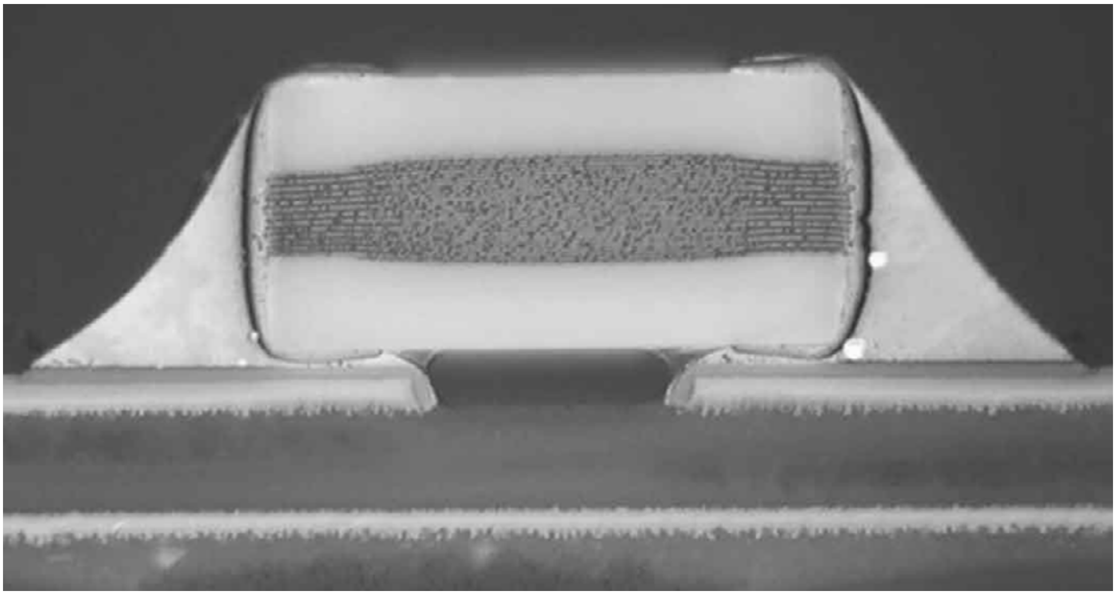


Figure 13: Cross-section of a 01005 component reflowed in nitrogen.

can be considered and the selection should be based on data and not assumptions. It is important to consider the interaction between multiple technologies in all areas during development and deployment since several advanced technologies will, in most cases, be used on the same product. Based on our studies, 125 µm component-to-component is feasible with both a 75 µm and 100 µm stencil thickness. At 100 µm component-to-component spacing a 4 mil (100 µm) stencil is not acceptable, but despite a DMPO of around 370 with a 3 mil (75 µm) stencil, this can in some cases be acceptable.

Acknowledgements

The authors would like to thank Multek and our existing component and machine suppliers in regard to valuable inputs on assembly, printed circuit board design/fabrication and high quality samples for our builds. The authors would also like to thank our failure analytical laboratories in Zhuhai, China and Penang, Malaysia for their support. **SMT**

References

1. Geiger D.A., Sjoberg J., Lee. J., and Aranda. R., “0.3 mm pitch CSP process development and PCB design,” Proceedings of SMTAI 2011, Fort Worth, TX.

2. Geiger D.A., Sjoberg J., Aranda. R., Kurwa M., “Assembly and design challenges for new generation 0.3 mm pitch CSP and 0.4 mm pitch TMV PoP,” Proceedings of SMTAI 2012, Fort Worth, TX and SMTA South East Asia, Penang.

3. Jonas Sjoberg, Ranilo Aranda, Henley Zhou, Jumbo Huang, Jimmy Chen, Sky Xiang, Isaias Daguio, William Uy, JieWei Lu, George Liu, GuoChao Liu, David Geiger, Murad Kurwa, “Development and Deployment of High Density Assembly,” Proceedings of SMTA 2013 China, Shenzhen.

Jonas Sjoberg is a technical consultant with Indium (formerly with Flextronics AEG, Malaysia).

Ranilo Aranda is process engineering manager with Flextronics AEG, Zhuhai, China.

David Geiger is director at Flextronics AEG, California, USA.

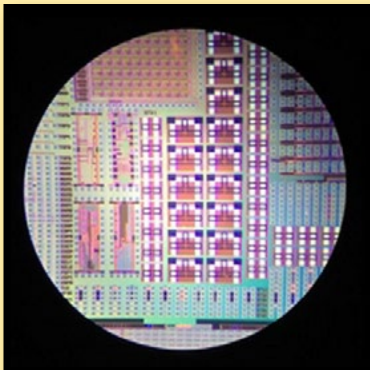
Anwar Mohammed is senior director at Flextronics AEG, California, USA.

Murad Kurwa is senior VP at Flextronics AEG, California, USA.

Ultra-sensitive Sensor Detects Individual Electrons

A Spanish-led team of European researchers at the University of Cambridge has created an electronic device so accurate that it can detect the charge of a single electron in less than one microsecond. It has been dubbed the 'gate sensor' and could be applied in quantum computers of the future to read information stored in the charge or spin of a single electron.

"The device is much more compact and accurate than previous versions and can detect the electrical charge of a single electron in less than one microsecond," M. Fernando González Zalba, leader of this research from the Hitachi Cambridge Laboratory and the Cavendish Laboratory, tells SINC.



Details of the breakthrough have been published in the journal Nature Communications.

"We have called it a gate sensor because, as well as detecting the movement of individual electrons, the device is able to control its flow as if it were an electronic gate which opens and closes," explains González Zalba.

In general, the electrical current which powers our telephones, fridges and other electrical equipment is made up of electrons: minuscule particles carrying an electrical charge travelling in their trillions and whose collective movement makes these appliances work.

However, this is not the case of the latest cutting-edge devices such as ultra-precise biosensors, single electron transistors, molecular circuits and quantum computers. These represent a new technological sector which bases its electronic functionality on the charge of a single electron, a field in which the new gate sensor can offer its advantages.

VIDEO INTERVIEW

iNEMI: A Highly Efficient R&D Consortium

by Real Time with...
IPC APEX EXPO 2015

CEO Bill Bader gives a comprehensive overview of iNEMI, from workshops to R&D projects. iNEMI's 2015 Roadmap covers six market segments and 19 technical areas in the electronics industry. People from 92 member companies collaborate on the most difficult industry problems.

realtimewith.com


TAKE A VIRTUAL TEST DRIVE
AND THE VICTORY LAP IS YOURS

SMT007 Supplier/New Product News Highlights

[Doing Business in India and China]

Electrolube Managing Director Ron Jakeman spoke to I-Connect007 Publisher Barry Matties at CPCA 2015 in Shanghai. The company has evolved over the decades, from producing contact lubricants, to manufacturing globally, with subsidiaries worldwide, including Beijing, China. In this interview, manufacturing, state of the global markets, and even Indian cuisine are discussed.

[Benefits of Soldering with Vacuum Profiles]

Requirements for void-free solder joints are continuously increasing in the field of electronics manufacturing, bringing new challenges that are evolving on a daily basis due to the relentless introduction of new variants of so-called bottom-terminated components (BTCs).

[Are the Robots Taking Over?]

Some have a fear that robots are taking over, but the founders of Universal Robots have a different view. They have focused on producing a versatile robotic arm that frees humans from doing repetitive tasks. Their journey has not been an easy one, having evolved from startup with just a couple of employees.

[Nordson ASYMTEK Restructures Americas Sales Team]

Nordson ASYMTEK, a global leader in dispensing, jetting, and coating equipment and technologies, announces the expansion and restructuring of its Americas sales team. With the changes, Brad Perkins has been promoted to general manager, Americas, for Nordson ASYMTEK. Terry Wilde has taken on the position of regional sales manager, Mexico, and Chris Heesch is district sales manager for the Southwestern USA.

[AIM Expands Distribution to Argentina with HighTech Solutions]

AIM Solder, a leading global manufacturer of solder assembly materials, is pleased to announce the addition of HighTech Solutions SRL as a new stocking distributor of the complete line of AIM solder assembly materials. HighTech Solutions will stock and supply solder bar, flux, wire, paste, preforms, etc. as well as provide technical support to AIM customers within Argentina.

[Electrolube Launches Aromatic-free Conformal Coatings]

Designed for the protection of electronic circuitry, Electrolube's AFA range significantly reduces operational hazards for operators due to its Toluene- and Xylene-free formulation. The range consists of AFA, AFA-F, formulated for use in non-atomised film-coat applicators, and AFA-S, for use in atomised spray applicators and HVLP spray guns.

[Interphase's Revenue Slides 34% in Q4]

Interphase Corporation, a diversified information and communications technology company, today reported financial results for its fourth quarter and full year, ended December 31, 2014. Revenues for the fourth quarter of 2014 were $2.9 million, a decrease of 34% compared to revenues from the fourth quarter of 2013 of $4.3 million.

[Semblant Names Simon McElrea CEO]

Semblant today announced that its Board of Directors has appointed Simon McElrea as CEO. McElrea previously held the position of VP of Marketing, Intellectual Property & Licensing at Energous Corporation, a Californian-based wireless charging company that completed its successful IPO in 2014.

[Saline Lectronics Invests in Juki's New Cube]

Dave Trail, President of Horizon Sales, commented, "This is the third selective solder system that Saline Lectronics has purchased from Juki. Saline's existing Juki units have been operating continuously in a multi-shift environment for a few years now and they needed the Cube to meet increased demand."

[Molex Boosts Fiber Optic Solution Offerings]

Molex Incorporated announced today the expansion of its fiber optic technology platform following the recent acquisition of Oplink Communications. A wholly-owned subsidiary of Koch Industries, Molex will now manage Oplink, a leading provider of optical communication components, intelligent modules and subsystems for a wide variety of customer applications.

FEATURE

Solder Paste Printing: Quality Assurance Methodology

by Lars Bruno, Ericsson AB
and Tord Johnson, MTEK CONSULTING AB

Solder paste printing is known as one of the most difficult processes to quality assure in electronic manufacturing. The challenge increases as the technology development moves toward a mix between large modules and small chip components on large and densely populated printed circuit boards. Having a process for quality assurance of the solder paste print is fast becoming a necessity. This article describes a method to ensure quality secured data from both solder paste printers and inspection machines in electronic assembly manufacturing. This information should be used as feedback in order to improve the solder paste printing process.

Introduction

This article has its roots in the need to improve capacity and quality levels at an electronics manufacturing site. Solder paste printing was identified early on as an area that needed to be secured with many of the new demands put onto the process by recent development in the manufactured products' technology level.

A. The Solder Paste Printing Process

Solder paste printing is one of the most critical processes in electronic manufacturing. The purpose of the process is to apply the correct amount of paste, at the correct position, with the correct form and to do this every time a print is performed. Even though the process can be considered relatively simple, the quality results of the print together with the PCB provide the foundation for the rest of the surface mount process. A good print result is a prerequisite for a good soldering result, while a poor print will lead to additional process issues as the product travels through the manufacturing chain.

The printing process has the following demands and properties:

- Solder paste properties: the viscosity drops when the paste is handled
- Stencil surface friction: must be relatively high to force the paste to roll instead of skid
- Squeegee surface friction: shall be relatively low in order to allow for the paste to roll and release properly when lifted

When the PCB is separated from the stencil, the paste shall stick to the board solder pads and not to the walls inside the stencil apertures. The amount of paste that ends up on the solder pad in relation to the amount of paste that can ideally be filled is known as the transfer efficiency. A transfer efficiency of 80% is commonly stated as an acceptable value but may not always be sufficient or required. Note that during certain situations it is possible to reach a transfer efficiency that is more than 100%.

One of the reasons that the printing process is so sensitive is because it involves mechanical tolerances, software settings, chemical properties and operator knowledge. Some of the most important parameters are:

- Maintenance status of the printer
- Status of the stencil
- The aperture design in the stencil
- Solder paste properties
- Squeegee (enclosed printing head) status
- Program parameters such as squeegee pressure, speed and angle, as well as separation speed between the printed board and the stencil

B. Particular Challenges

In addition to the parameters stated above, the particular type of products produced at the specific manufacturing site referred to in this paper have several properties that make solder paste printing even more challenging. First of all, the boards are large in size; it is not uncommon for them to reach lengths and widths surpassing 450 mm, and some will even stretch to 500 mm. A large print area reduces the process window since it is more difficult for most printers to have reliable results both in the middle of the operational print area and close to the squeegee edges. Secondly, the products contain a large number of components; it is not unusual for these products to have more than 30,000 board solder pads. The large number of pads put demands onto both the printing and inspection process by making it virtually impossible to notice paste deviations through manual inspection. Finally, the boards have many components with different package types, ranging from passive 0201 (imperial) chips and high-end processors to pin-in-paste connectors. This large spread of component packages stresses the solder paste printing since the amount of needed paste differs dramatically, and it is also common that many small components are placed in the vicinity of larger package types. This makes it even more challenging since stencils with step solutions must be carefully evaluated in order not to degenerate the paste deposits for the smallest components.

C. Needs

The aforementioned given challenges put several needs into high-level manufacturing. First, it is clear that it is impossible to control the process sufficiently without secured measurement data. It is also clear that performing this manually becomes both impractical and unreliable. It is therefore a necessity to use automatic inspection machines (i.e., automatic solder paste inspection). Another part that is as equally important as secured data and automatic inspection support is that the data gathered can be utilized by the organization in order to evolve and continuously improve and thereby secure the needed quality level.

This reasoning leads us to a question: How can we ensure that we can trust the measurement equipment that we use in production to secure solder paste deposits? This question did in turn generate a project that became a methodology.

Methodology

The goal of the project was to optimize and to verify the solder paste printing process in a structured way. The implemented project therefore had the following multi-phasal strategy:

- To verify solder paste inspection repeatability and measurement accuracy
- To optimize the solder paste printing process
- To create routines to verify that the solder paste printing process capability both remains at a high level and improves

The methodology is based on three main phases that all need to be fulfilled in order for the needed improvements in production capacity and capability to take place.

A. Securing Solder Print Inspection Measurements

The goal is to control the result of the solder paste printing process (i.e., provide the best conditions to obtain high-quality solder joints). However, waiting with inspection until after reflow is performed is a slow and quite expensive way, in terms of time and rework, to verify the quality of the solder deposits. Instead, it is more useful to have inspection directly after the printing process in order to have immediate feedback and the ability to control the printing process within an acceptable timeframe. There is currently only one effective way to control a large number of solder deposits within an electronic manufacturing and that is to use a solder paste inspection machine (SPI). Of course, using an SPI as the sole instrument to ensure one of the most important process parameters within electronic manufacturing makes it important that one really can rely on the machine to give correct information.

The method therefore started by ensuring that the inspection machine in itself was reliable and had the correct accuracy. This was done by performing a gauge repeatability and reproducibility (GR&R) analysis of the SPI to ensure that it gave the same results each time. There are two important aspects of a GR&R analysis:

- Repeatability: The variation in measurements taken by a single person or instrument on the same or replicated item and under the same conditions
- Reproducibility: The variation induced when different operators, instruments, or laboratories measure the same or replicate specimen

These two aspects were addressed by having two different operators perform the inspection operation at different times in production. Since a GR&R only addresses the precision of a measurement system and not its accuracy, it was also necessary to measure a deposit that is already well known. For such a purpose, it is not preferable to use a board that has been solder paste printed since the paste in itself will change form and characteristics with time, temperature, humidity, etc. Instead, a reference was designed that replicates as many features of a solder paste printed board as possible. The choice fell on a reference that was created from a brass metal sheet that had been etched in two steps. This was made in order to recreate the pattern of a solder mask and copper traces, which would be present on a regular printed circuit board. The pads were then plated with copper to resemble solder paste. To protect the surface treatment from corrosion, the copper was plated with pure tin. This creates a reference board that resembles a printed product and can be used to verify the capability of a solder paste inspection machine. Consequently, the reference board has the following characteristics:

- Resistant: It is made from metal that will not degenerate due to time, normal temperature and humidity levels or other environmental effects
- Reflects an actual printed circuit board: Since the reference is etched in different layers, it simulates shadowing effects and has PCB traces, vias, etc. Thus it resembles the top layer of an actual product
- Entire SPI measurement area is tested: Due to the large size of the reference board it is possible to measure the SPI's performance not only in the center of the machine but also in the outer areas

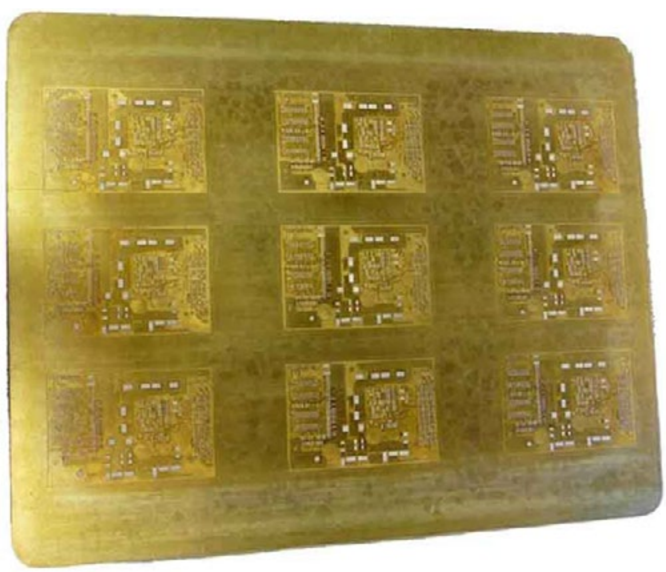


Figure 1: Reference board for solder paste inspection.

The reference board is depicted in Figure 1. The reference has been divided into nine different measurement sections. Each measurement section has the same appearance and they are intended to be copies of one another. However, due to process variations at the manufacturers there are some differences in the amount of metal present at the different locations. It is therefore necessary to measure each deposit separately. Within each section there are ten different measurement points (pads) that have been measured.

Figure 2 shows close-ups of the solder deposits on the reference board. The left shows the deposits from a top view where traces and via hole imitations also can be seen clearly. The right picture shows the deposits from an angle where it is possible to see both the plated copper, which is used to build the main height of the paste, but also the tin surface treatment is clearly visible.

The entire reference board was sent to a measurement laboratory for verification of the height and volume of the reference deposits. The height and volume of the 10 different pads were measured 10 times and an average value was calculated. To gain the correct results the height should be measured using the traces as reference and not utilize the ditch around the pad.

With a verified and measured reference board, it is possible to quality ensure that the SPI accuracy is within reason and that it can be used in production to optimize the printing process.

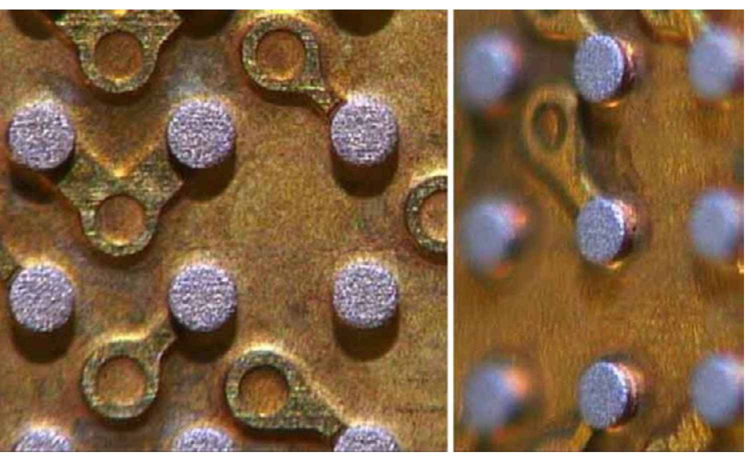


Figure 2: Close-up pictures of the paste deposits on the reference board, from above (left) and at an angle (right).

B. Optimizing the Printing Process

Good control over the solder paste printer is essential in order to achieve production that results in low defect rates. This capability investigation routine explains how a certain kit of material can be utilized in order to control the solder paste printer's different parameters in a controlled manner in order to achieve a reliable and quality secured solder paste print.

The purpose of such a capability investigation is to enable a manufacturing site with a solder paste printer to define and optimize print parameters. This holds true to machine parameters such as speed, pressure, cleaning cycles, etc., but also for indirect parameters that have a large impact upon the print results such as board and stencil support, kneading of the solder paste, pauses in production, squeegee quality, humidity, temperature, maintenance intervals, etc.

When optimizing a solder paste printing process, it is advantageous if the tests performed are related to the type of production that is general or known to soon become general. In this context a product analysis was performed in which different aspects were considered, including: the products physical size, required squeegee lengths, number of apertures, aperture sizes and aperture locations. With these aspects in mind, a PCB test pattern was created and a stencil was designed that mirrored the pattern (Figure 3). Consequently, it will be possible to identify if there are special areas within the solder paste printer that perform worse than other areas. It will also be possible to investigate what size of the apertures that the process can handle in the different areas.

The test pattern on the printed board is designed in such a way that it will be very difficult to achieve acceptable results on all deposits. In fact, the solder paste should have physical difficulties to deposit through the smaller apertures according to the area ratio.

The basic test pattern blocks have been placed in a star formation in order to cover most component placement variants. Additional test

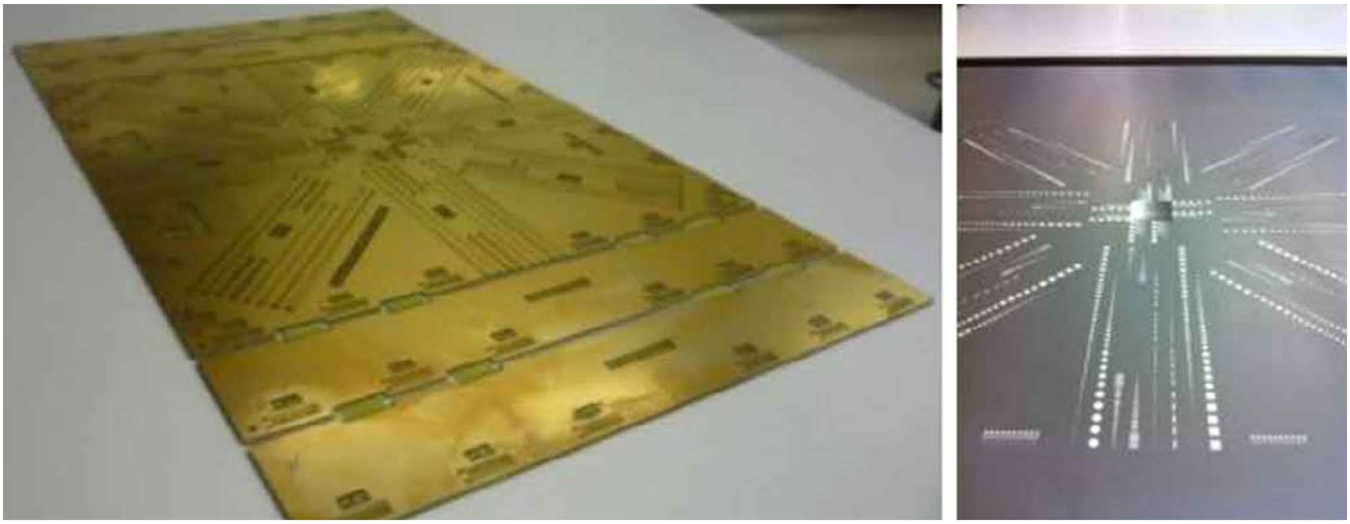


Figure 3: Test pattern on a printed circuit board and its equivalent on the stencil.

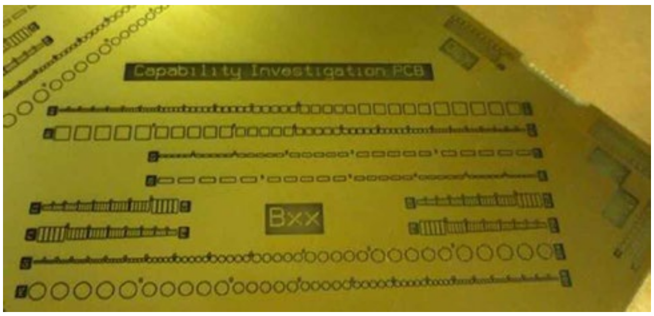


Figure 4: Pattern within an arm of the star.

patterns have also been added around the board edges to evaluate how the printer performs in these areas. The test board has been designed to be able to handle prints of 300, 400 and 500 mm width. The central area of the test board has a pattern that can be utilized to evaluate a possible step-up area on the stencil. The stencil has a thickness of 127 µm and was made from fine-grain steel. Tests have been performed with both etched and laser cut stencil versions.

Within each star arm, the blocks include squares with rounded corners, circles and oblong apertures (Figure 4).

The precise size of the apertures as well as on the PCB is given in Table 1 together with the area ratio and pitch. Each individual size is repeated five times before the next size block starts. Each type of aperture (e.g., square, circle, etc.) is rotated 180° in order to evaluate the size at different positions within the machine.

Before continuing with the discussion of the reference boards and stencils, it is worth revisiting some of the basic rules for designing stencil apertures and how it relates to the pad area. The volume of solder paste that is optimal for a pad is mainly dependent on the type of component that is utilized. The deposited paste volume is theoretically given by the size of the aperture and the thickness of the stencil. As the board and stencil separates from each other during the cycle there will be a collection of different forces acting on the solder paste. Solder paste will either be transferred to the board pad or stick to the stencil aperture walls. Whatever happens is closely related to the following important factors:

- Board-stencil separation speed
- Aperture area and aspect ratios
- Aperture side wall geometry
- Aperture side wall finish

The board-stencil separation speed is set as a machine parameter and is not directly related to the stencil design. The other three, however, are directly related to the material, manufacturing methods and the design of the stencil apertures.

In general, when designing a stencil aperture there is a number of conditions that have to be met in order to get an acceptable process result:

1. Aperture adjusted according to the component and to the pad
2. Aperture adjusted according to paste
3. Aspect ratio larger than 1.5
4. Area ratio larger than 0.66

All of the above conditions generally need to be fulfilled to get both good solder paste release and a correct soldering result.

The aspect ratio is the relationship of the width of the aperture divided by the thickness of the stencil. If it is smaller than 1.5 the traction force from the stencil aperture walls will be too large and the paste will not release completely. Consequently, there will be less paste on the board land than intended and the solder result will not be acceptable. The same kind of consequence will also occur if the area ratio of the aperture is too small. It is possible to calculate the area and aspect ratio by utilizing the following formulas:

$$Aspect\ Ratio = \frac{Width\ of\ Aperture}{Thickness\ of\ Stencil}$$

$$Area\ Ratio = \frac{Area\ of\ Aperture}{Area\ of\ Aperture\ Walls}$$

Aperture Size						
Square		Circle	Rectangle 1		Rectangle 2	
X	Y	D	X	Y	X	Y
0.2	0.2	0.2	0.9	0.225	0.2	0.8
0.25	0.25	0.25	1.1	0.275	0.25	1
0.3	0.3	0.3	1.4	0.35	0.3	1.2
0.35	0.35	0.35	1.8	0.45	0.35	1.4
0.4	0.4	0.4	2.4	0.6	0.4	1.6
0.45	0.45	0.45	3	0.75	0.45	1.8
0.5	0.5	0.5			0.5	2
0.7	0.7	0.7			0.7	2.8
0.9	0.9	0.9				
1.1	1.1	1.1				
1.4	1.4	1.4				
1.8	1.8	1.8				
2.4	2.4	2.4				
3	3	3				

Area Ratio			
Square	Circle	Rectangle 1	Rectangle 2
X	D	X	X
0.39	0.39	0.71	0.63
0.49	0.49	0.87	0.79
0.59	0.59	1.10	0.94
0.69	0.69	1.42	1.10
0.79	0.79	1.89	1.26
0.89	0.89	2.36	1.42
0.98	0.98		1.57
1.38	1.38		2.20
1.77	1.77		
2.17	2.17		
2.76	2.76		
3.54	3.54		
4.72	4.72		
5.91	5.91		

Pad Size						
Square		Circle	Rectangle 1		Rectangle 2	
X	Y	D	X	Y	X	Y
0.25	0.25	0.25	0.95	0.2375	0.25	1
0.3	0.3	0.3	1.15	0.2875	0.3	1.2
0.35	0.35	0.35	1.45	0.3625	0.35	1.4
0.4	0.4	0.4	1.85	0.4625	0.4	1.6
0.45	0.45	0.45	2.45	0.6125	0.45	1.8
0.5	0.5	0.5	3.05	0.7625	0.5	2
0.55	0.55	0.55			0.55	2.2
0.75	0.75	0.75			0.75	3
0.95	0.95	0.95				
1.15	1.15	1.15				
1.45	1.45	1.45				
1.85	1.85	1.85				
2.45	2.45	2.45				
3.05	3.05	3.05				

Pad Pitch			
Square	Circle	Rectangle 1	Rectangle 2
X	D	X	X
0.375	0.375	1.425	0.375
0.45	0.45	1.725	0.45
0.525	0.525	2.175	0.525
0.6	0.6	2.775	0.6
0.675	0.675	3.675	0.675
0.75	0.75	4.575	0.75
0.825	0.825		0.825
1.125	1.125		1.125
1.425	1.425		
1.725	1.725		
2.175	2.175		
2.775	2.775		
3.675	3.675		
4.575	4.575		

Table 1: Aperture sizes, area ratio and pitch between the pads within the test pattern.

Figure 5: Test pattern used at the left/right and front/back sides of the reference board.

As explained in Table 1, for aperture sizes that have a smaller aperture ratio than 0.67 there will most likely be challenges with obtaining sufficient amount of paste on the board pad, since much of the paste will be left on the inside walls of the stencil apertures.

The edge pattern that is present on the board is depicted in Figure 5. On this pattern only rectangular apertures have been utilized in order to evaluate the printer's performance in these sensitive areas.

Row 1 is not expected to provide a satisfactory result due to its poor/low area ratio, the others apertures, however, should give acceptable print results. The pattern presented in Figure 5 is also rotated 180° to make it even more difficult for the printer by placing the smallest apertures at the very edge of the test board.

C. "Ways-of-Working" with Capability Investigations

To be able to reap the benefits from the capability investigations and implement the improvements in operations, it is necessary to have solid routines and standardized ways-of-working. Operator training was therefore an integrated part of the project from the start. Operators were involved in the creation of the reference boards in order to ensure that they would be suitable for usage within the normal production routines. This process also had the benefit of giving operators a deeper understanding of why and how the reference board should be used and treated.

The steps depicted in Figure 6 explain the ways of working with the above described capability investigations.

The first step included performing a setup of the solder paste printer and solder paste inspection machine in order to be able to handle the test. This operation is basically performed just as with a normal production order, with the difference that the reference boards are used

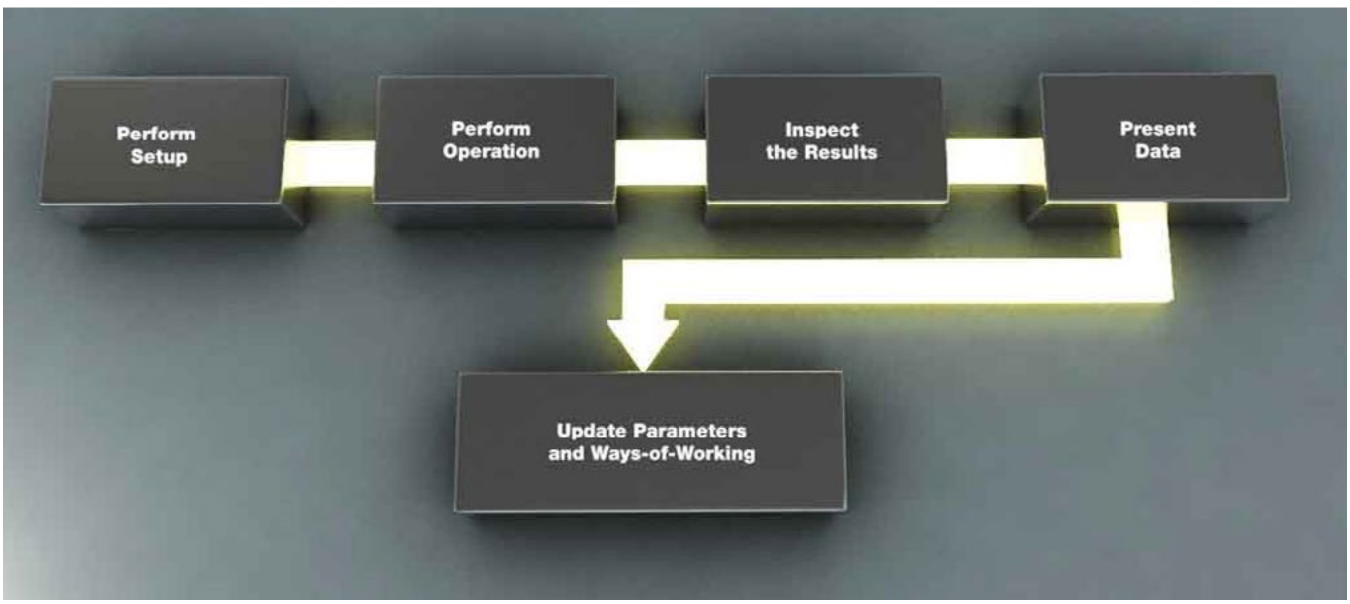


Figure 6: Capability investigation methodology.

instead of normal product boards. The second step includes performing the actual printing/inspection operation. This part is also a standard way of working regarding electronic production. The third step is to inspect the print deposits and evaluate whether they are considered acceptable or not. Once this is complete, the data should be analyzed and presented. With these facts, a decision can be taken to update machine parameters or ways of working around the machine if necessary.

The main usage of this methodology is to perform scheduled verification of the solder paste printing capabilities. When starting with such investigations it is suggested that they are performed frequently, at least once per shift, in order to ensure that there are no short-term reasons for quality deviations. As data is gathered and analyzed and the process is stabilized, the frequency of capability verification can be reduced. After a while, the organization will have gained enough experience regarding the machines to define a suitable frequency for the capability verification.

Utilizing this method also makes it possible for production management to quickly gain an understanding of the quality levels in the solder printing process and make decisions on how to improve it even further. However, as with any change in ways-of-working it is important that management monitors constantly so that the new routines and templates are used in order for the capability verifications to give a sustainable increase in the quality levels.

III. Data and Results

Although the method described here has its main usage in ensuring that the solder paste printing process has the intended capability, it can also be used in a number of additional ways in order to increase knowledge, gather facts and optimize the process even further. Some of the most useful scenarios are described in this section.

A. Process Parameter Optimization

The first and arguably the most important usage of the capability investigation methodology is to optimize the printing process parameters. When performing any optimization of process parameters it is often essential to lock certain variables while varying others in order find the optimum for that particular parameter. For some processes this is an easy task, since there are not so many parameters that affect the final result. For solder paste printing however, this is not the case since the result is affected by material properties, machine status, maintenance, operator handling, the surrounding environment as well as the machine settings. Some of these parameters can be controlled efficiently while others are more difficult.

The method of varying all variables in a controlled way in order to find an optimum is known as a design of experiment (DoE). Performing a complete DoE (i.e., varying all variables for all other possible values) for solder paste printing becomes impossible, or at least impractical, for a production environment due to the large quantity of variables that play a significant role in the result. Instead, a simplified method was used, meaning that all parameters except one was locked and varied. This was performed for several important parameters:

- Print speed
- Squeegee pressure
- Enclosed print-head pressure
- Separation speed
- Board support

The results from the evaluations are depicted in Figures 7–10, together with an explanation of how the tests were performed.

The first goal for the project was to find the basic printing parameters. Since the printer used an enclosed print head to apply the paste instead of squeegees, it was of interest to investigate the internal pressure within the enclosed head. This pressure is mainly used to apply the paste down into the apertures of the stencil. The internal pressure was therefore changed in steps around the suggested value given by the suppliers of the paste and the enclosed print head. For each step, different printing speeds were performed and the printing results were analyzed, with the results plotted in Figure 7. It is clear from these results that there is an optimum pressure between 1–1.4 bar, for any selected printing speed.

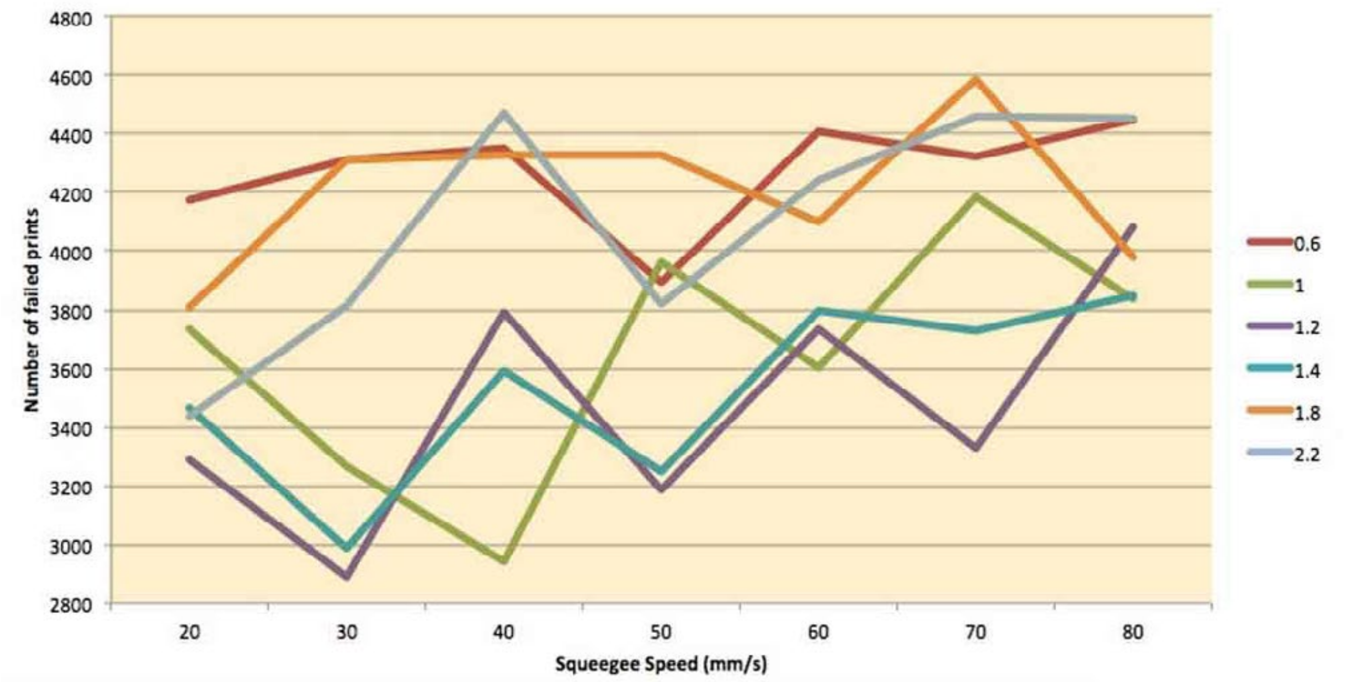


Figure 7: Results for how the amount of successful deposits varies with the squeegee speed and internal print-head pressure.

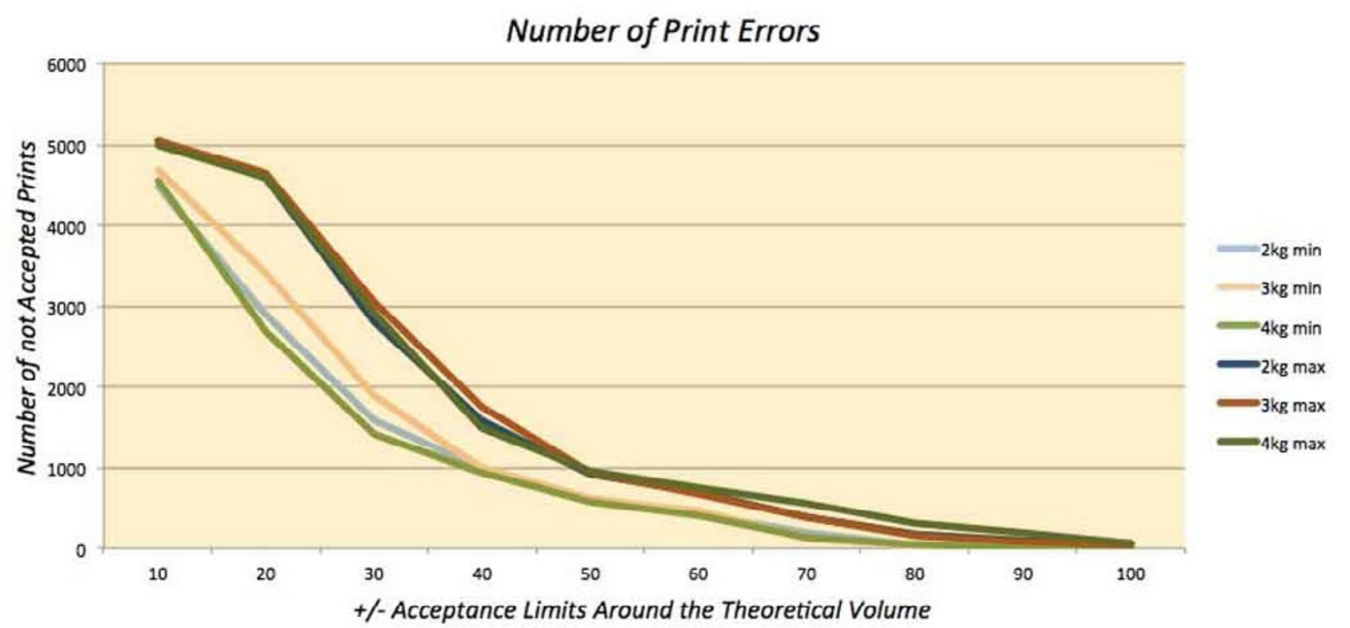


Figure 8: Results for how the paste misprints vary with the squeegee pressure.

The next parameter that was investigated was how the print result varied with the pressure of the squeegees against the stencil. In this case, all the printing parameters were fixed except for the squeegee pressure, the data was analyzed, and the spread of the print errors was plotted against different acceptance levels of the SPI (from ±10% to ±100%), as depicted in Figure 8.

After the basic parameters had been identified, the project continued to optimize the solder paste printing process by also investigating other important parameters. Figure 9 depicts the results of varying the separation speed between the board and the stencil after the printing process is completed. A clear optimum was identified for the project setup of 2 mm/s separation speed.

Another important parameter that was investigated was how the board support affected the printing quality. In production both fixed and variable board support were used for flexibility and cost reasons. The reference board was set up with both types and the separation speed was varied in order to see if there was a significant difference between the different production methods. The results are depicted in Figure 10 and clearly show that the spread between maximum and minimum failure rates has about a 10% difference in favor of a fixed board support.

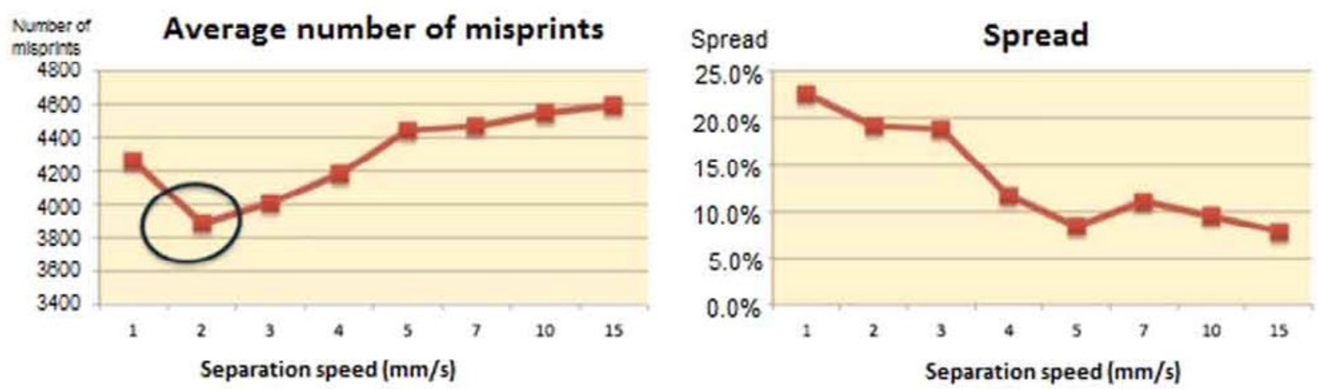


Figure 9: Results for how the misprints vary with the separation speed.

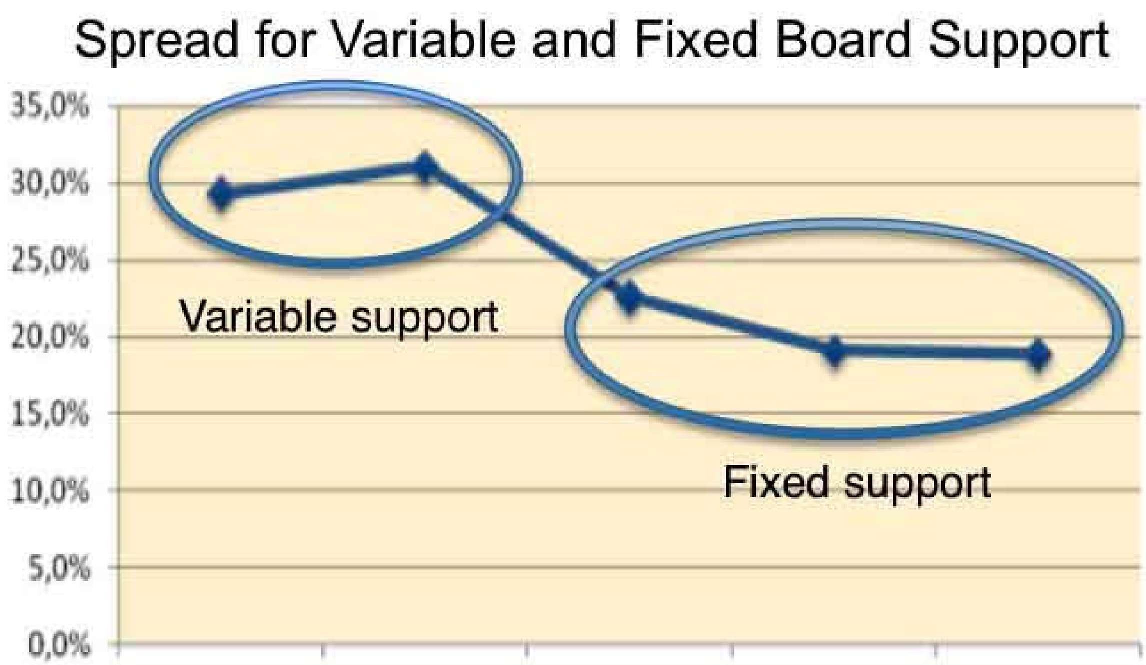


Figure 10: Results for how the spread of the solder paste deposit varies with separation speed and type of board support (40 mm/s print speed, 1.2 bar internal pressure, 2.6 kg squeegee pressure and ±20% acceptance level for the volume).

B. New Materials

Although the methodology was mainly developed to ensure quality in the solder paste printing process, it has also proven to be an efficient and standardized way to evaluate new materials that are introduced into the process. This section describes some of the tests that have been performed in order to secure and evolve the solder paste printing process.

The first test was a solder paste printability test. During a printability test, solder paste is printed through stencil apertures with the same size and shape as the board pads. The volume of all printed solder paste deposits are then measured by a SPI machine and the solder paste deposit volumes from the same size apertures are then sorted out and the volume distributions are compared for different solder pastes.

Figure 11 shows the solder paste volume distributions for two different solder pastes printed through the reference stencil's 0.4 x 0.4 mm square apertures. Both solder pastes had Type 4 particle size, but they had different tackiness, rheology and viscosity properties.

The center of the reference board has an area intended for the step stencil test. Around this area are pads of different sizes and with different distances towards the central area (Figure 12).

The test board pads are squares with rounded corners with side-lengths of 0.25 mm, 0.35 mm, 0.45 mm, 0.7 mm, 1.1 mm, 1.7 mm and 3.0 mm. The pitches between the same pad sizes, in the direction away from the center are given in Table 2.

A test was performed where test stencils with a 1:1 aperture ratio to the test board pads were used. Five laser cut stencils with a base thickness of 5mil were used in this test. Each of the stencils had just one step height and the step heights for the different stencils were 1 mil, 2 mil, 3 mil, 4 mil and 5 mil.

An example of a print result is shown in Figure 14.

After each printed solder paste deposit had been measured by the SPI, the solder paste volume distribution relationship to the distance from the step edge were analyzed. Examples of distributions for the same aperture size at different distances from the step edge and with different step heights are shown in Figure 15.

Another production tool that was investigated was the difference between two manufacturing methods for stencils (laser cut and etched). The two test stencils (as described in the methodology section) were ordered and verified by measuring the area of several different stencil apertures. The stencils were then used to perform ten solder paste prints each and the results were analyzed and are depicted in Figure 16. Manufacturing method one was laser cut and manufacturing method two was etched.

All of the above described tests are the result of the project site's used machines, environment, operators and consumables and the presented results should not be considered as generally true. Instead, they are provided here to simply give an understanding of how

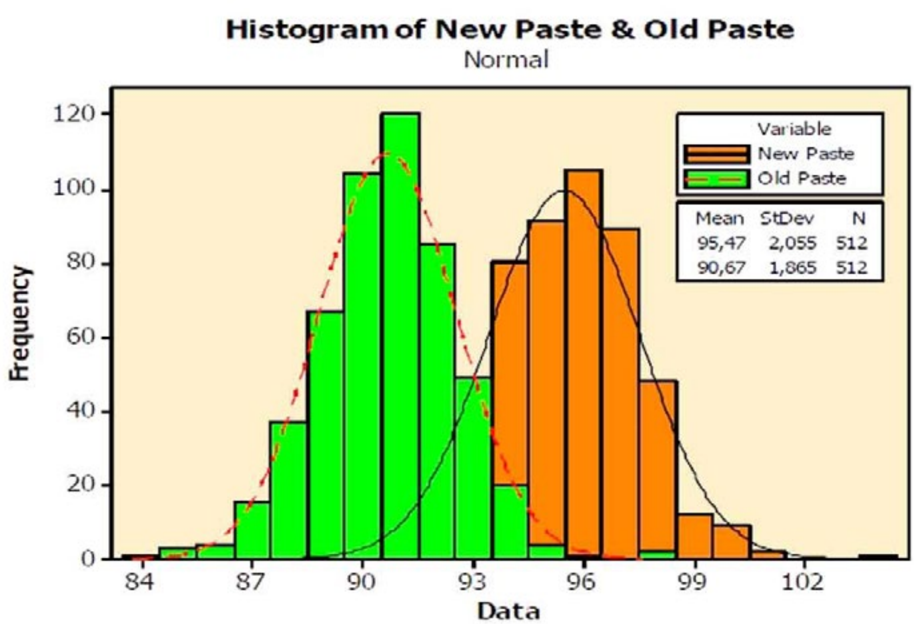


Figure 11: Solder paste volume distributions from two different solder pastes printed through 0.4 x 0.4 mm square stencil apertures.

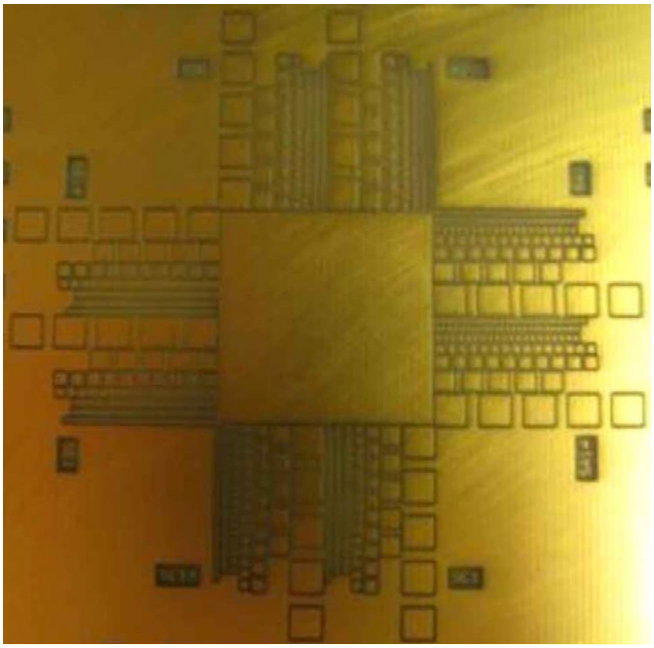


Figure 12: Test board for step stencil test.

Pad size	Pitch
0.25mm	0.45mm
0.35mm	0.60mm
0.45mm	0.75mm
0.7mm	1.125mm
1.1mm	1.725mm
1.7mm	2.775mm
3.0mm	4.575mm

Table 2: Pad size and pitch for the pads close to the center area.

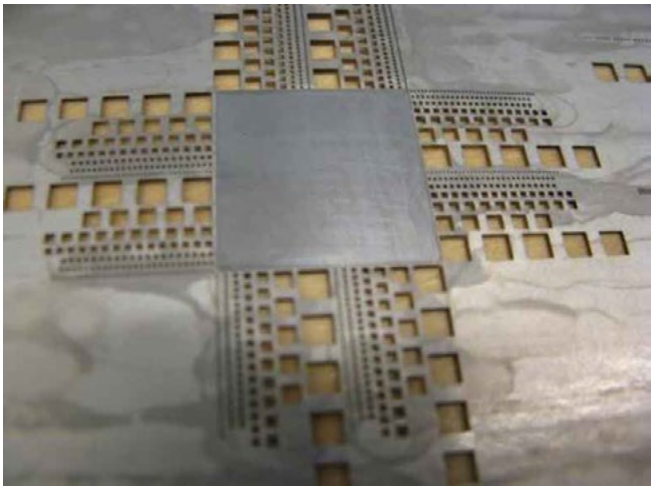


Figure 13: Step stencil with stepped area in center.

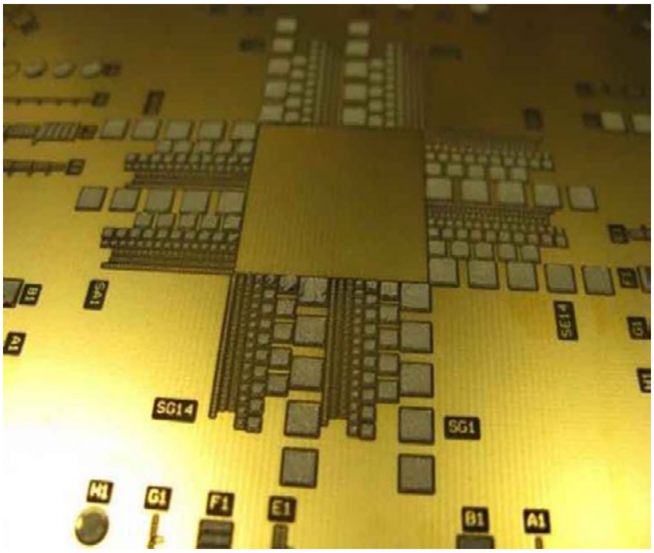


Figure 14: Print result—print direction: from left to right.

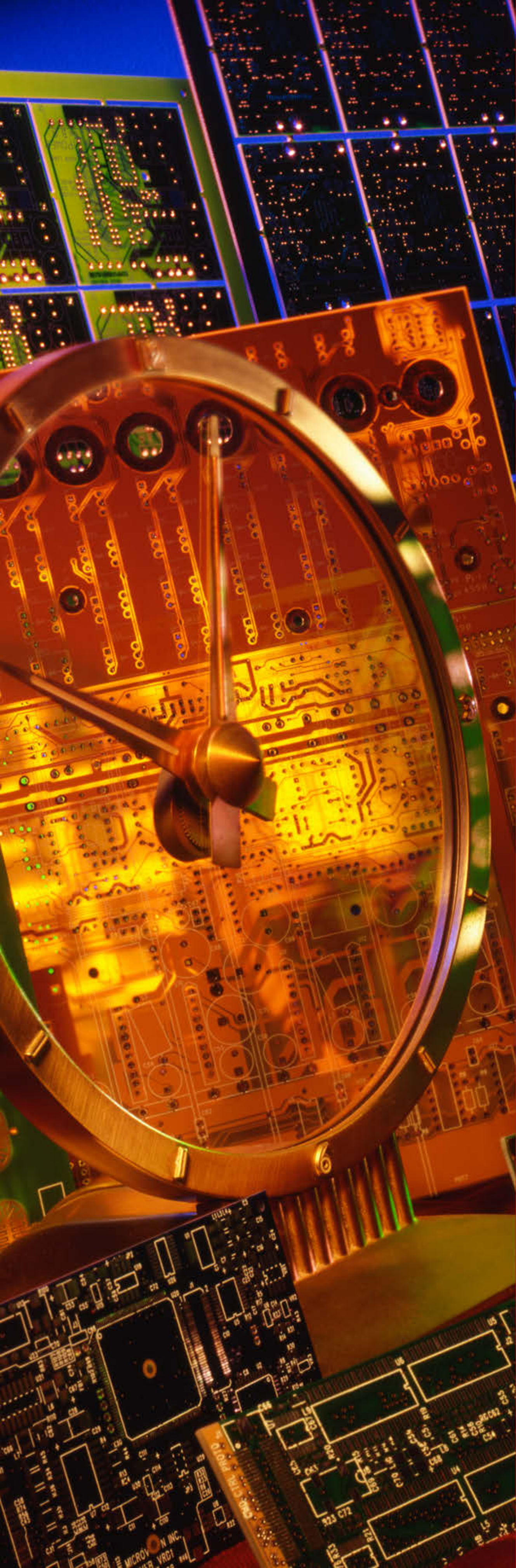
the presented methodology can be used to find the most suitable solutions for a solder paste printing process. It is suggested that each manufacturing site performs their own investigations by utilizing these methods since the presented results here most likely not will be useful to copy.

C. Maintenance, Verification and Troubleshooting

With the presented reference boards it is also possible to understand the printing process even better by also considering:

- Maintenance challenges
- Verification and troubleshooting

The utilization of the presented reference boards can be used for a status-based maintenance (i.e., certain maintenance is performed when needed, instead of being based on a time-schedule). This can be achieved by using the reference boards on a regular basis and analyzing the results to quickly see when a process is deviating from the intended quality levels. By setting routines for this and escalation points,



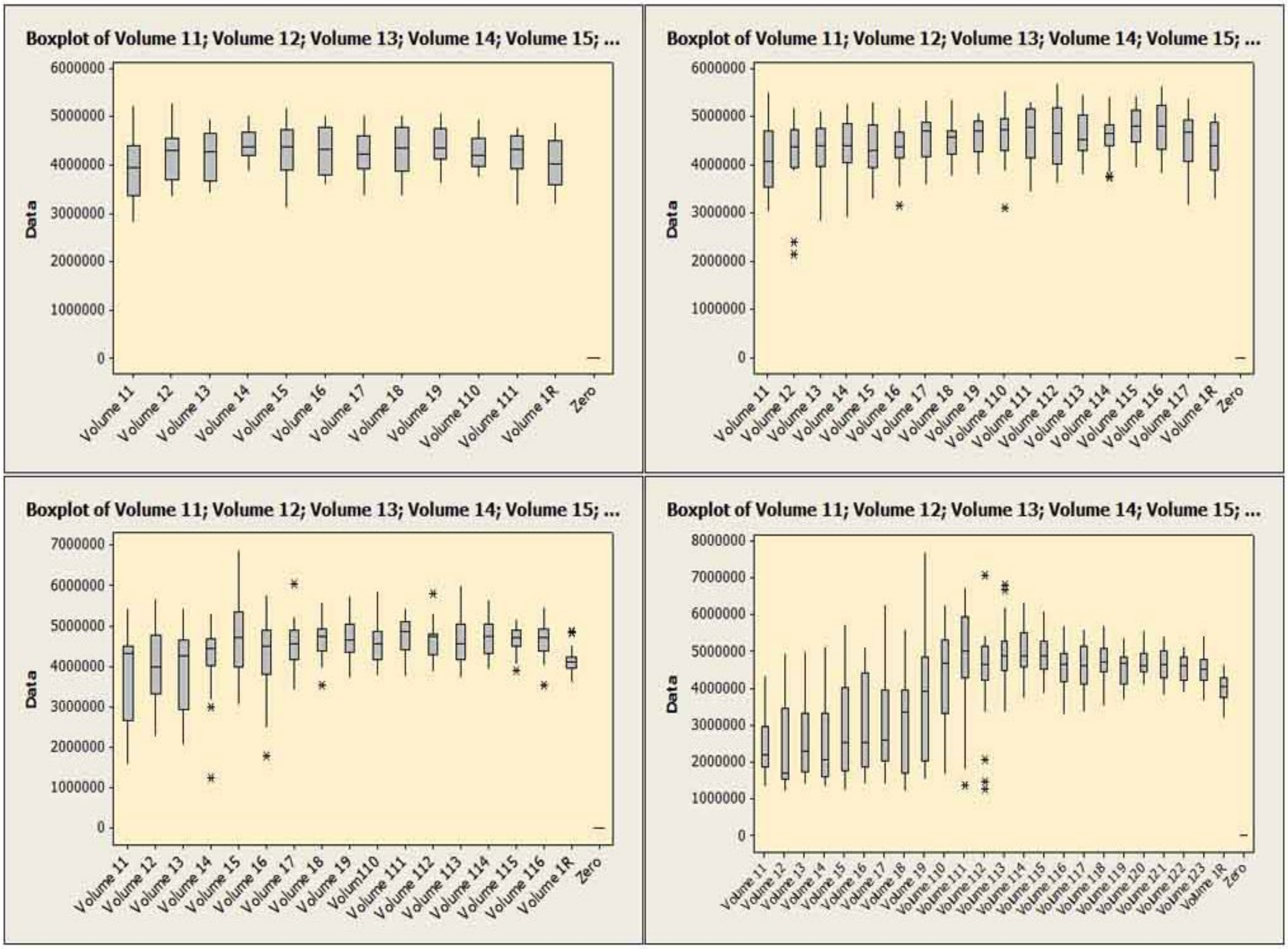


Figure 15: Box plots of solder paste volume distributions (mm3). Volume 11 is the deposit adjacent to the step edge and volume 1R is the deposit furthest away. All distributions above are from 0.25 x 0.25 mm square stencil apertures. 1 mil step (upper left), 2 mil step (upper right), 4 mil step (lower left) and 5 mil step (lower right).

it is possible to implement status-based maintenance.

The reference boards can also be used during verification and troubleshooting scenarios. A typical troubleshooting example is when there is a quality deviation during the manufacturing of a product. The methodology can then be used to verify if the printing and inspection process has changed or if the issues are related more to the product design. Once the issues have been identified it often proves useful to use classical 6-sigma methods like Ichikawa and kaizen events to identify and remedy the root-cause of the quality deviation.

For the implementation of the process verification, the following guideline (Figure 17) can be used in production in order to increase the process control and ensure that the process does not deviate outside the set limits.

IV. Conclusion

This paper has presented a methodology for quality assurance of the solder paste printing process and its adjacent inspection process. The methodology makes it possible to have immediate feedback from the inspection process in order to control the printing process. The methodology utilizes reference boards and

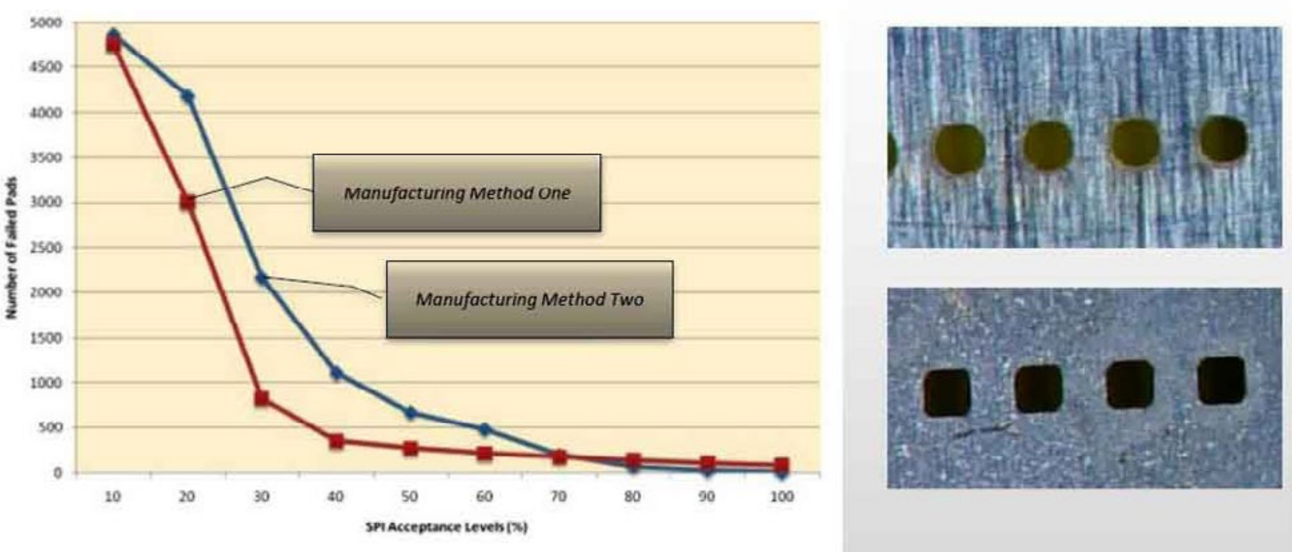


Figure 16: Amount of solder paste misprints when comparing two different stencil manufacturing methods.

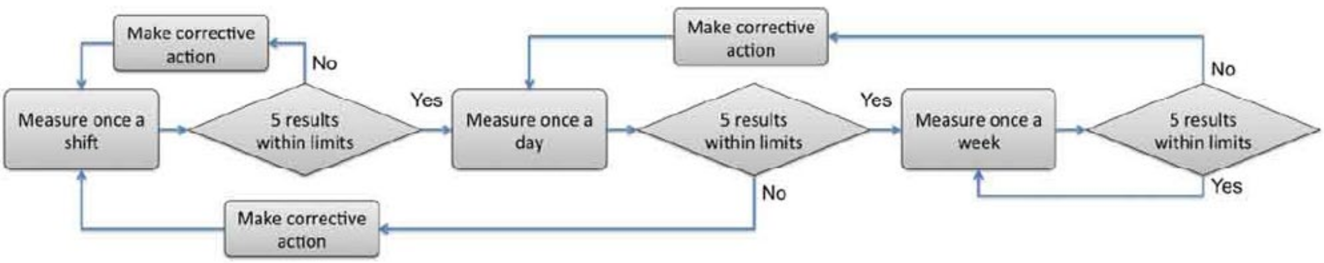


Figure 17: Workflow for implementing verification in a production environment.

stencils to create a standardized way of comparing results. This standardization makes it possible to:

- Perform capability investigation
- Optimize process parameters
- Evaluate new materials and consumables
- Perform efficient troubleshooting
- Pursue status-based maintenance

It also ensures repeatability and accuracy of the measurement equipment and makes it possible to survey the optimized process stability over time and environmental changes. **SMT**

IIV. Acknowledgement

The authors would like to thank the following people that have contributed to the development and test of the methods used in this paper and paper review:

1. Ericsson Katrineholm: Peter Sonevang, Leif Karlsson, Roger Zwosta, Staffan Bengtsson, Randy Le.
2. Ericsson Kumla: Leena Korhonen, Kalevi Lehikoinen.
3. Ericsson Kista: Benny Gustafson, Anne-Kathrine Knoph.
4. MTEK Consulting: Mattias Andersson.

Lars Bruno is a product engineer at Ericsson.

Tord Johnson is the COO of MTEK Consulting AB.

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

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

ITAR

Electronics Industry News
Market Highlights

Optical Components Market Growth Picking up Again in 2015

LightCounting's analysis shows that optical components market has been growing at 8–12% per year on average over the past five years, but slowed during the second half of 2014. Recent reports from leading suppliers indicate that market growth picked up in Q1 2015, and LightCounting expects that the market will reach a new record in Q2 2015.

Smart Grid Sensors Market to Hit $350M in 2021

The market for smart grid sensors is set to rapidly expand in the coming decade, with revenues growing nearly ten-fold, from 2014–2021, according to IHS Inc. Based on information in a new study from IHS Technology, the market for smart grid sensors is centered in North America.

State of U.S. Manufacturing After the Recession

Since the economic recovery started in 2009, the U.S. manufacturing sector generally maintained a 12% share of GDP over the five years ending in 2013. The sector's performance was impressive when compared with other major advanced countries that experienced similar declines and early recoveries in manufacturing production.

Flexible Batteries Market Attracts New Investments

After years of slow progress from small companies, flexible batteries now have the attention of big brands such as Apple, Samsung, LG Chemical, Nokia and STMicroelectronics, who are set to drive the flexible battery market from US$ 6.9 million in 2015 to over US$ 400 million in 2025 according to IDTechEx.

Next-Gen Smart Lighting Systems Revenues to Hit $1B by 2020

In a new report, "Smart Lighting Market Opportunities: Smart Bulbs and the Rise of Local Lighting Intelligence," NanoMarkets concludes that the market for these new systems will reach just over $1 billion by 2020.

Gartner Reveals Vertical Industry Trends in SEA's Thriving Markets

"Some might argue that given recent political, financial and climate challenges, Southeast Asia is a risky proposition," said Venecia Liu, research vice president at Gartner. "However, Gartner believes that Southeast Asia's economic development and growing consumer demand mean that its growth potential outweighs the risk."

Flexible Display Market to Hit $30B by 2025

In the newly published "Flexible, Curved, and Bendable Display Technologies and Market Forecast 2015 Report," (Second Edition), Touch Display Research analyzed more than 10 display technologies and more than 14 applications of flexible displays, and the report shows how the flexible display market is accelerating.

Smart Wearable Healthcare Devices Market to See Significant Growth

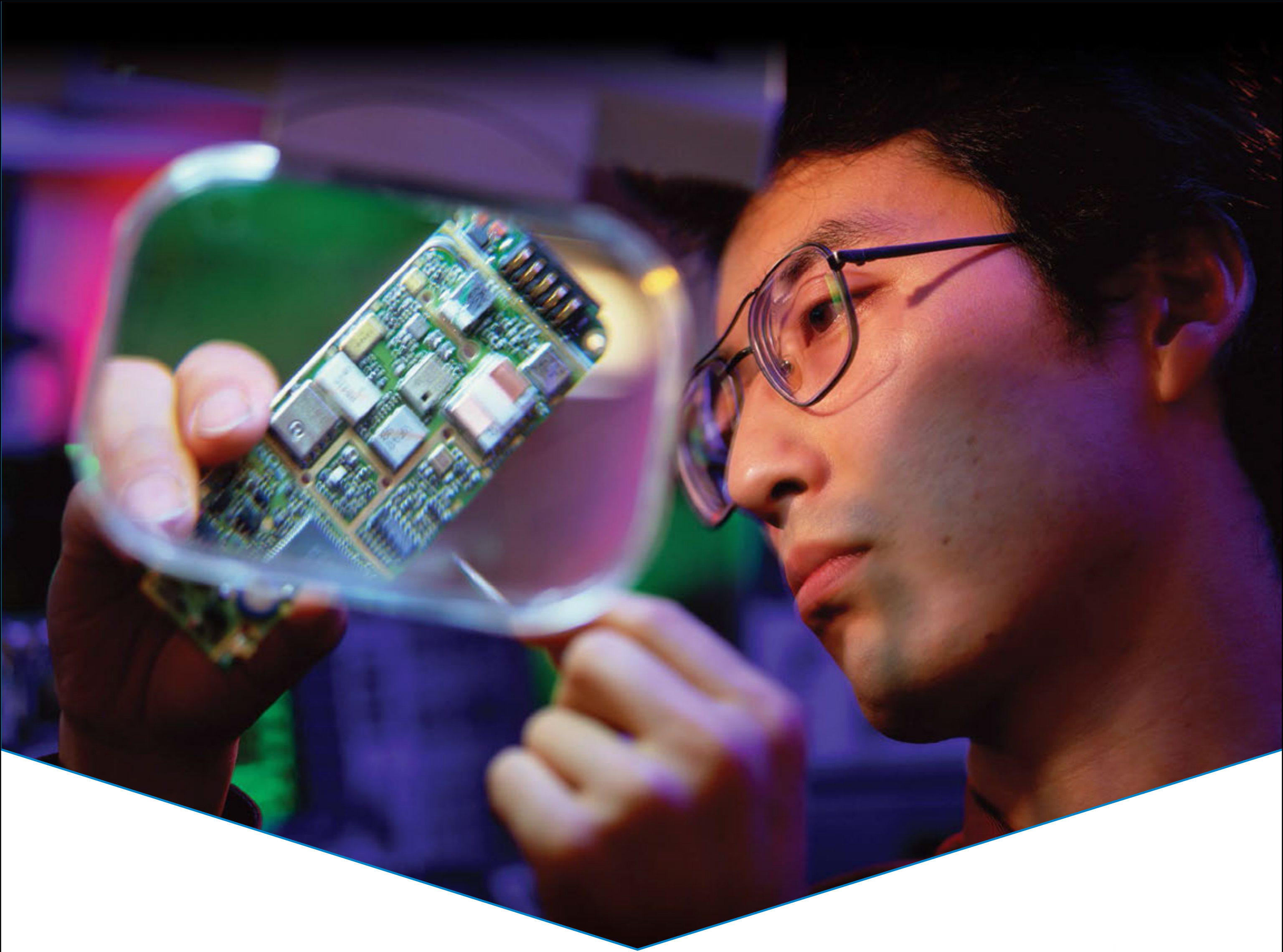
The Technavio report emphasizes the increased use of smart wearable devices for tele-home healthcare. Monitoring many patients in hospitals can be a tedious process because of the lack of adequate resources. Smart wearable devices enhance treatment and monitoring processes by saving time and reducing healthcare costs.

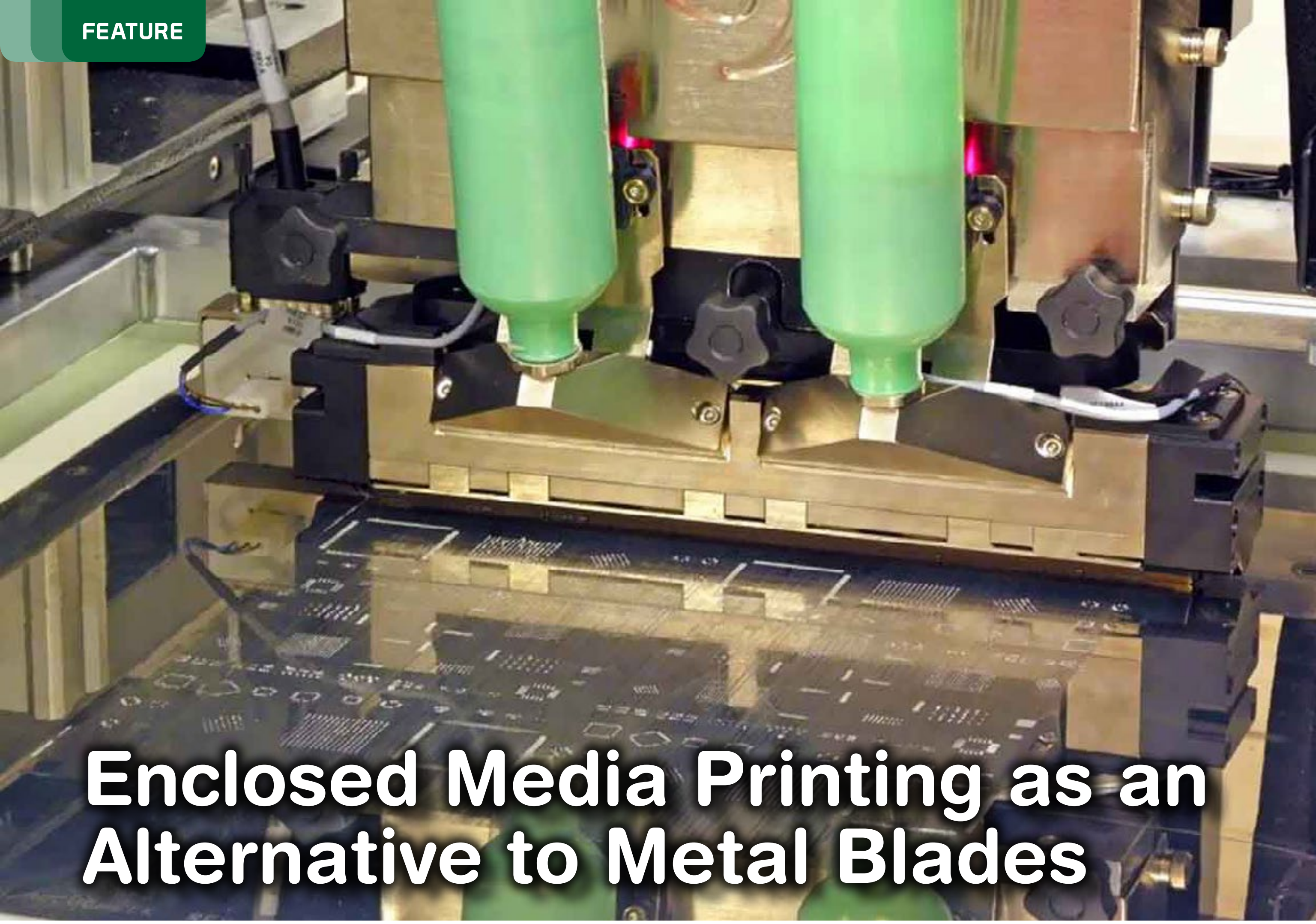
IoT Data to Exceed 1.6ZB in 2020

A new report from ABI Research estimates that the volume of data captured by IoT-connected devices exceeded 200 exabytes in 2014. The annual total is forecast to grow seven-fold by the decade's end, surpassing 1,600 exabytes—or 1.6 zettabytes—in 2020.

Tech Transfer Provided $1.8T to US Economy

The report, "The Economic Contribution of University/Non-profit Inventions in the United States: 1996–2013," estimates that, during this 18-year time period, academia-industry patent licensing bolstered U.S. gross industry output by up to $1.18 trillion, U.S. gross domestic product (GDP) by up to $518 billion, and supported up to 3,824,000 U.S. jobs.





Enclosed Media Printing as an Alternative to Metal Blades

by Michael L. Martel
SPEEDLINE TECHNOLOGIES INC.

Fine pitch/fine feature solder paste printing in PCB assembly has become increasingly difficult as board geometries have become ever more compact. The printing process itself, traditionally the source of 70% of all assembly defects, finds its process window narrowing. The technology of metal blade squeegees, with the aid of new materials, understanding, and settings such as blade angle, has kept pace with all but the smallest applications.

Enclosed media print head technology has existed, and has been under increasing development, as an alternative to metal squeegee blade printing. Until recently, the performance of enclosed print heads had been comparable to the very best metal squeegees, but advances in enclosed print media technology have now made it a superior alternative to squeegee blades in virtually all applications.

Introduction

Solder paste printing through stencils has long been achieved using metal blades, or squeegees, which replaced polymer squeegee blades due to performance issues years ago. The move from screens to stencils, and then to smaller apertures and fine pitch land patterns, necessitated the change to metal, which offered superior printing performance characteristics.

The evolution of PCBs in terms of the miniaturization of assemblies, components, and ever-finer feature print patterns has not slowed, and as a result continues to present ever-increasing challenges to the makers of assembly equipment and solder paste printing technology, narrowing the process window. Fine pitch and fine feature printing applications, e.g., 200µ–.50 area ratio (AR) and 150µ–.375 AR, have been pushing blade printing technology to its limits.

Skilled printer operators using superior metal blade alloys, varying angle of attack, etc., can achieve acceptable results against the most challenging printing applications, but there are

shortcomings that include considerable material waste, which translates into significant cost, less fill volume as apertures become smaller, and unacceptable variation in the consistency of results. Newly developed enclosed print media technologies have been developed as an alternative to metal blade printing. Due to recent advances, enclosed media printing delivers better results overall when measured against ordinary-use metal blades, and they excel in material savings since the print head encloses the solder paste or other media from the surrounding atmosphere, providing steady and uniform results for fine feature apertures filling due to tight process control. For these and other reasons that will be outlined in this paper, they constitute an attractive alternative to metal blade printing.

Aperture Fill is Essential

When printing fine pitch or fine features especially, good aperture fill volume is the key to well-formed solder joints without insufficiencies. Aperture fill results from downward pressure on the paste by the squeegee, forcing solder paste into the apertures. Pressure is generated by compression of the bead of paste, from the squeegee blade's angle of attack as it moves across the stencil and rolls the bead of solder paste before it. Variations in the blade angle (some creatively) can, with some tweaking, optimize the print process. But there are always concerns relative to the uniformity of pressure across the blade, or the quantity of paste in a given spot, which can also be affected by high-volume requirements (through-hole, paste-in-hole) on the same stencil adjacent to fine pitch apertures. With squeegee blade printing, gauging the prospect of successful aperture fill has always been subject to the vagaries of area ratio. The more disproportionately greater the surface area of the stencil wall to the pad area, the better the chance that, when the stencil peels away, the paste will remain with the stencil.

"When printing fine pitch or fine features especially, good aperture fill volume is the key to well-formed solder joints without insufficiencies. Aperture fill results from downward pressure on the paste by the squeegee, forcing solder paste into the apertures. Pressure is generated by compression of the bead of paste, from the squeegee blade's angle of attack as it moves across the stencil and rolls the bead of solder paste before it."

With fine feature devices, the primary printing defect is poor or insufficient aperture fill. Without proper or complete fill, the solder paste may not adhere to the pad and pull away/remain in the aperture; or it may result in a 'starved' solder joint (or incomplete joint) if undetected by the downstream SPI machine. Some of the causes of insufficient aperture fill are the same causes identified for other print defects (e.g., pause in printing/raised paste viscosity; squeegee speed too high or too low; squeegee pressure too low; not enough paste on stencil; and others).

Proper or optimum aperture fill for fine feature printing is not only a function of mechanical setup and squeegee blade material and parameters, but also a function of having a precisely-controlled volume of solder paste at all times on the stencil. Control is a keyword here; the more fine-featured the printing application, the greater the degree of precise control over the process that needs to be exercised.

In the words of George Babka (Assembléon):

In a majority of operations, operators scoop solder paste onto the stencil without precisely measuring the amount applied. A few companies specify the amount of solder paste to be placed on the stencil at the beginning of a production run (i.e., one full 350g jar), but most do not. Yet even these former companies fall short of full control of the volume of solder paste on the stencil when setting up the paste dispenser.

Most paste dispense processes are suited to a larger volume of paste (75–150g), and dispensed at infrequent intervals (every 40–50 prints). As a result, the roll size can still change significantly during the course of the shift…a 100g change in solder roll size will cause approximately a 7% change in maximum paste filling pressure. This is a significant change. It is enough to dramatically alter the solder paste print quality, especially on small apertures…A larger solder paste roll will have higher applied pressure, as well as a longer time that the pressure is applied. Too large of a roll size for a given speed applies too much pressure to the aperture for too long of a time, increasing the potential for bridging of fine pitch devices…True solder paste volume control can be accomplished in one of two ways: via an enclosed print head, or to dispense more accurately and more often…[1]

The smaller the pad/aperture, the more dramatic the effects of poor aperture filling, due to the more precise (and smaller) volume of paste required to create a successful solder joint. The absence of a specific small volume of paste from a large pad print, with no deleterious effect on the solder joint ultimately formed, could, for a fine feature component, be the entire amount of paste required for that connection.

Enclosed Media Print Head Technology

Enclosed media print head technology was designed to provide the same required pressurization of the solder paste as the squeegee blade's angle of attack, but uniformly across the length of the print head while printing, regardless of the amount of paste in the enclosed chamber. The key to consistency and uniformity in printing results, where the stencil or print application requirements were anything but uniform, was determined to be direct, fast-response control of paste pressure. This would have to be closed-loop controlled so that pressurization could respond to such demands, for example, as are made by through-hole fills that require a very high volume of paste, rather than a pattern of fine pitch pads. Additionally, with optimum aperture fill, the more likely the paste will cover the entire pad area, and that its tack will be sufficient to keep it on the pad when the stencil peels off.

Efforts to improve the printing performance of enclosed media resulted in a change in the way that the paste in the paste chamber is pressurized. Rather than indirectly or pneumatically, pressurization is accomplished mechanically by a motor-driven plunger or piston that is separated from the paste by a membrane. The plunger applies pressure directly to the

Device Type	Enclosed Media Volume	Enclosed Media Std. Deviation
0201	5% Greater	4% Less
BGA 80	9% Greater	18% Less
150µ - .35 Area Ratio	54% Greater	26% Less

Significantly greater volume with finer featured devices

Less standard deviation than blades as apertures grow smaller

Table 1: Volume and standard deviation comparison with squeegee blades.

volume of paste in the chamber, and a series of transducers sense the pressure and are part of a closed-loop system of very tight pressure control, which maintains chamber pressure actively at +/- 0.1 psi from set point.

In comparative printing tests against squeegee blades, these expectations proved correct, and surprisingly, the finer the feature, the better the results that were obtained with enclosed media. For example, whereas with metal squeegee blades the finer the aperture the less percentage of desired volume could be depended upon, with enclosed media, the trend was just the opposite, with print volumes over 50% for 150 micron apertures with a 0.375 AR.

Table 1 shows the results of comparative tests between printing with an enclosed media print head and metal blade squeegees. Note that the finer the aperture, the more the quality and consistency of squeegee blade printing deteriorates compared to the performance of enclosed media.

Print Head Design and Operation

The enclosed media print head maintains solder paste in a closed print chamber that can be pressurized to force the solder paste out of the chamber and onto the substrate. Conventional squeegee blades are replaced with metal blades that function as both scraper and seal. These blades are aligned in a leading edge configuration at 45 degrees, as opposed to a typical trailing edge squeegee configuration. The inwardly-inclined blades are 10mm apart at the point of contact with the stencil. When the head is lowered onto the stencil, the stencil foil will close the entire print chamber during operation and allow a positive pressure to be generated inside. Silicone dams seal the opposite ends of the chamber.

The pressure inside the chamber is generated by a plunger, or piston, that is motor-driven downwards into the upper part of the chamber to generate pressure. A flexible membrane separates the plunger from the solder paste. Three pressure transducers measure the internal chamber pressure and feed this information back to a control system that drives the plunger to the required height to actively maintain the desired pressure to +/- 0.1 psi from set point. With every print cycle, more solder paste transfers out of the head onto the substrate while the

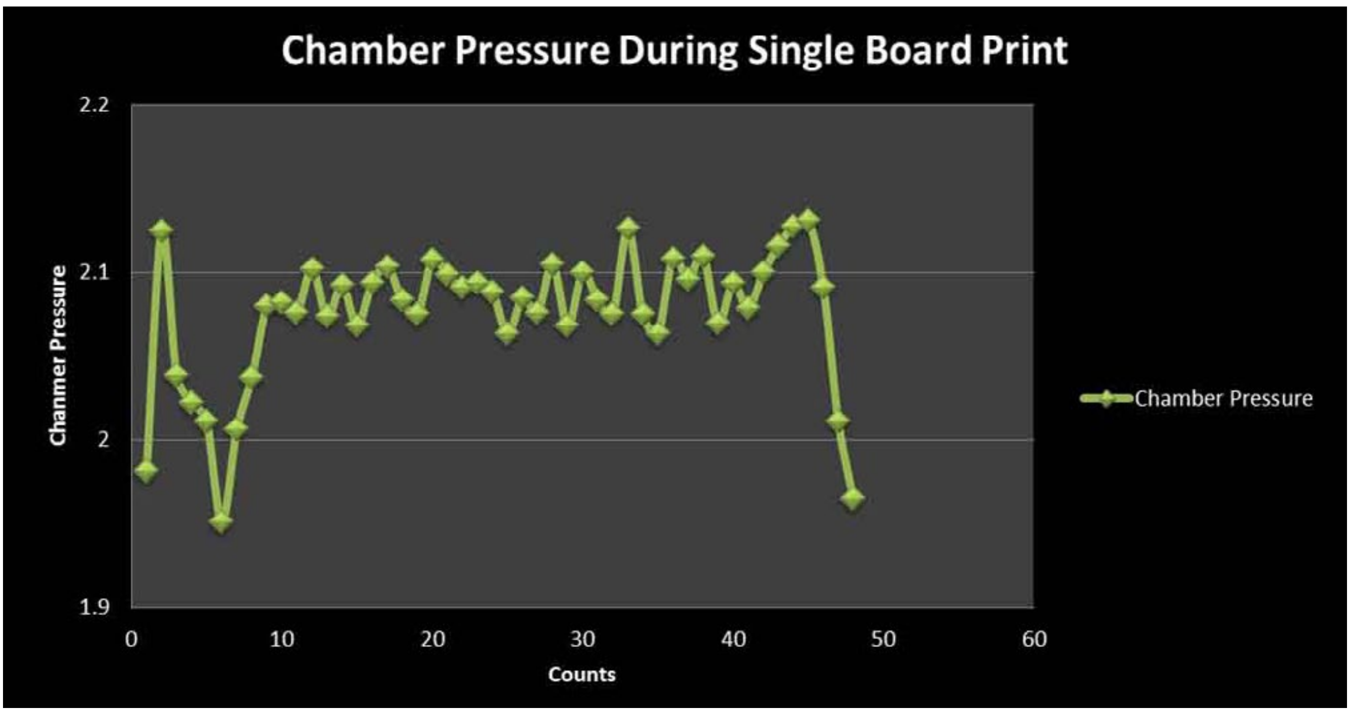
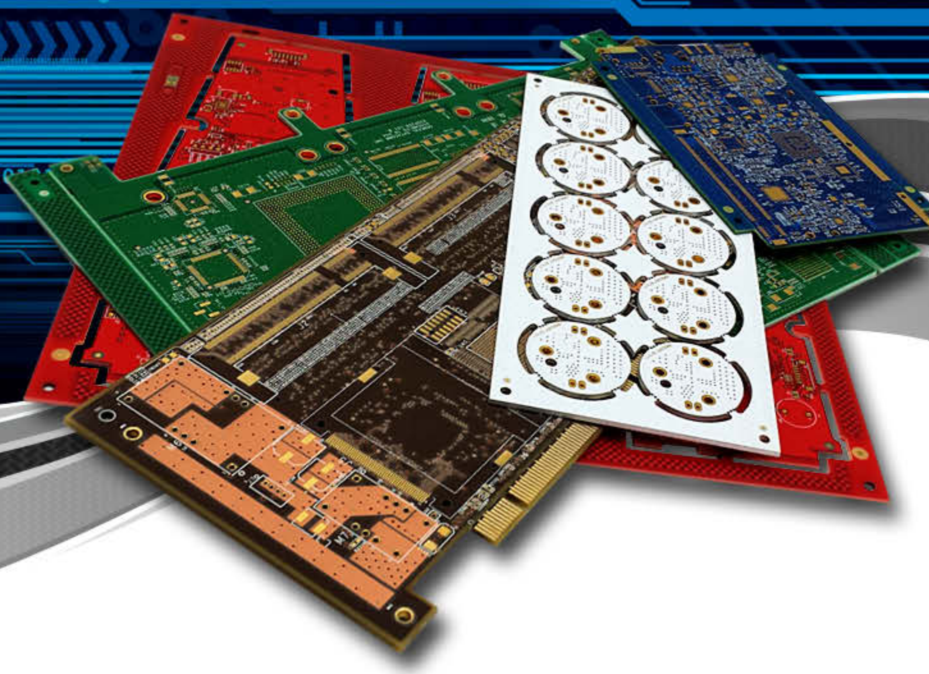


Figure 1: Chamber pressure consistency plot.

The Absolute Best Value in High Technology Printed Circuit Boards



Since 1979, Eagle Electronics Inc. has provided our customers with the highest quality Printed Circuit Boards at fair and competitive prices. From our modern 50,000 sq. ft. facility, we continue to meet and exceed our customer's ever increasing expectations and requirements.

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

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

click here to learn more

www.eagle-elec.com

plunger travels further down into the chamber.

A low-level sensor detects when the plunger reaches the lowest position, thus triggering the refilling of the head with solder paste. The paste is provided from two industry-standard cartridges that are mounted onto the front of the head. Air pressure is applied to the cartridges to force the paste out of the cartridges and into the chamber. The plunger is kept at its upper position during this fill routine. Once the pressure transducers read a defined fill pressure inside the chamber, the air pressure to the cartridges is relaxed and the plunger moves back down onto the membrane. The enclosed media print head is ready to continue printing.

Figure 1 is a plot of chamber pressure during a simulated production run. Interboard pressure data illustrates how tight, closed-loop chamber pressure control is maintained consistently board to board regardless of paste consumption requirements (large apertures & small).

"A low-level sensor detects when the plunger reaches the lowest position, thus triggering the refilling of the head with solder paste. The paste is provided from two industry-standard cartridges that are mounted onto the front of the head. Air pressure is applied to the cartridges to force the paste out of the cartridges and into the chamber. The plunger is kept at its upper position during this fill routine."

Operation

Prior to the first print, the head chamber must be filled completely with solder paste. The filling operation is supported by a dedicated software routine that ensures that the machine is in the correct status for this operation. During the fill routine, the plunger is positioned at a fixed 'fill position' above the membrane, for the purpose of limiting the expansion of the membrane upwards while solder paste is being forced out of the cartridges and into the chamber. At the end of the fill routine, the plunger is moved to the relaxed distance relative to the fill position.

The enclosed media print head differs from similar types of print heads in that pressure is applied to the solder paste mechanically, not through air (pneumatic) pressure. The print head consists primarily of a paste chamber that is filled with solder paste from standard industry paste cartridges. These replaceable cartridges are mounted onto the print head assembly. Air pressure is used (software-controlled) to drive the paste out of the cartridges, through a manifold, and into the paste chamber. Pneumatic pressure is employed only to fill the chamber; it plays no role in pressurizing the paste for printing. Print pressure is supplied through a plunger (piston).

Three robust, highly sensitive pressure transducers ensure tight control of pressure inside the head during filling, as well as during printing. These sensors are mounted directly inside the paste chamber and constantly monitor the chamber to ensure a constant 'full' status. The chamber is considered full when the fill pressure is averaged across the selected transducers. During printing, the head is designed to keep the chamber full, and is thus programmed to initiate an "in production fill" between cycles. A sensor reliably senses a paste empty condition in the cartridges. Blade assemblies on either end (front and rear) of the print head control the paste distribution and excess removal from the stencil. Blade assemblies consist simply of specially-designed blades attached to a round shaft.

Materials Savings

Customer data indicates that the materials and solder paste savings achievable with enclosed media can be in excess of 50%. Solder paste is saved because it does not dry out, does not need to be discarded at the end of a shift, and there is overall less throwaway and loss through cleaning. Every scenario will be different, but the one represented in Figure 2 (actual example) realized a payback in a little more than 28 weeks. Clearly, higher volumes of

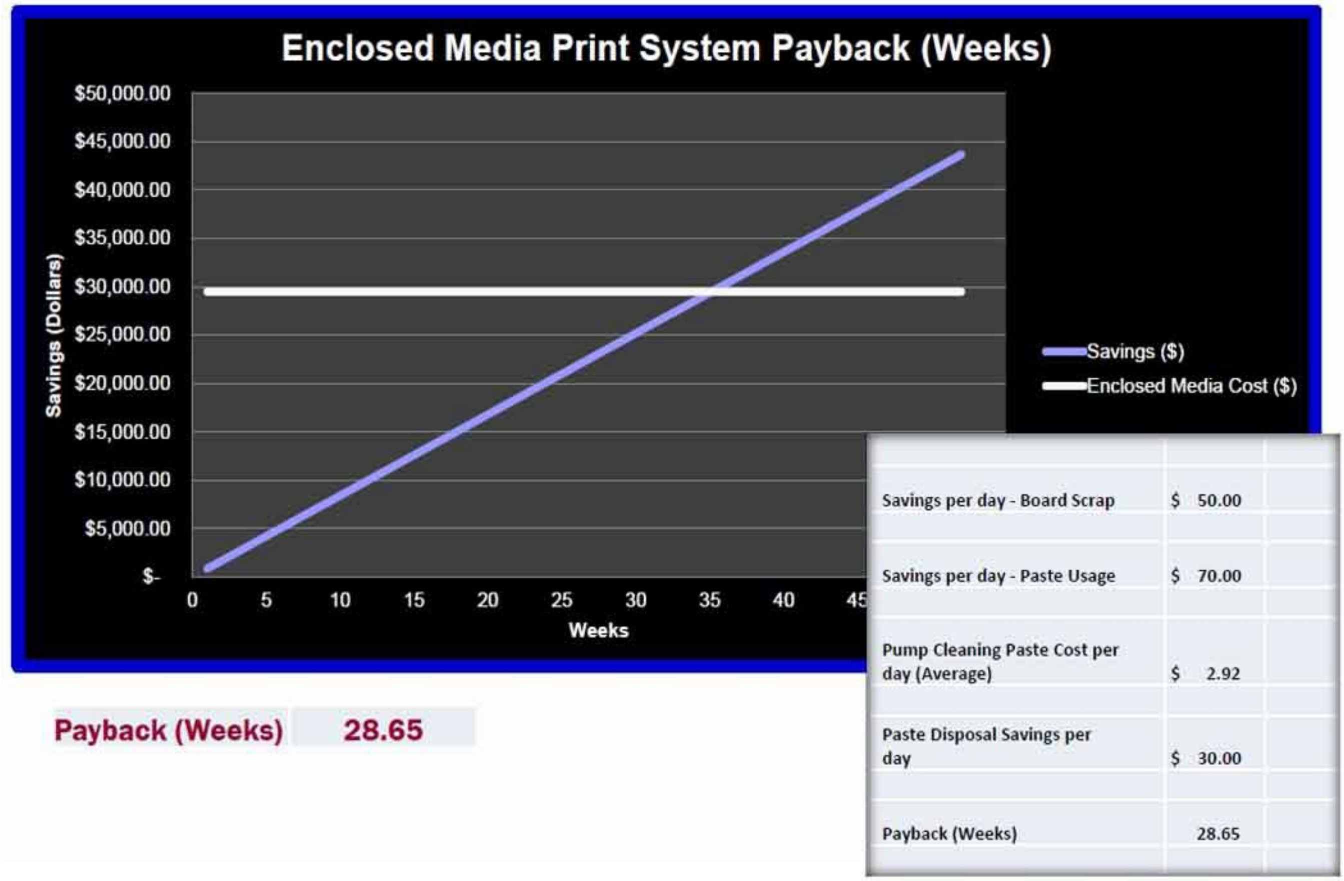


Figure 2: Enclosed media payback, in weeks.

production, coupled with a higher cost solder paste, could easily shorten that time. Nonetheless, ROI for the enclosed media print head is relatively quick, across the board, based on material savings alone.

Conclusion

Enclosed media printing technology is a suitable replacement for metal squeegee blades particularly for demanding fine pitch applications and mixed low and high volume paste consumption applications. The ability of enclosed media printing to successfully meet the volume fill requirements of ever-shrinking aperture sizes and lopsided aspect ratios is sufficient justification. Additionally, the paste savings over squeegee blade printing are significant, so much so that in the current economic climate these cumulative savings can contribute to a relatively quick payback on the equipment investment and have a measurable effect on the bottom line. **SMT**

References

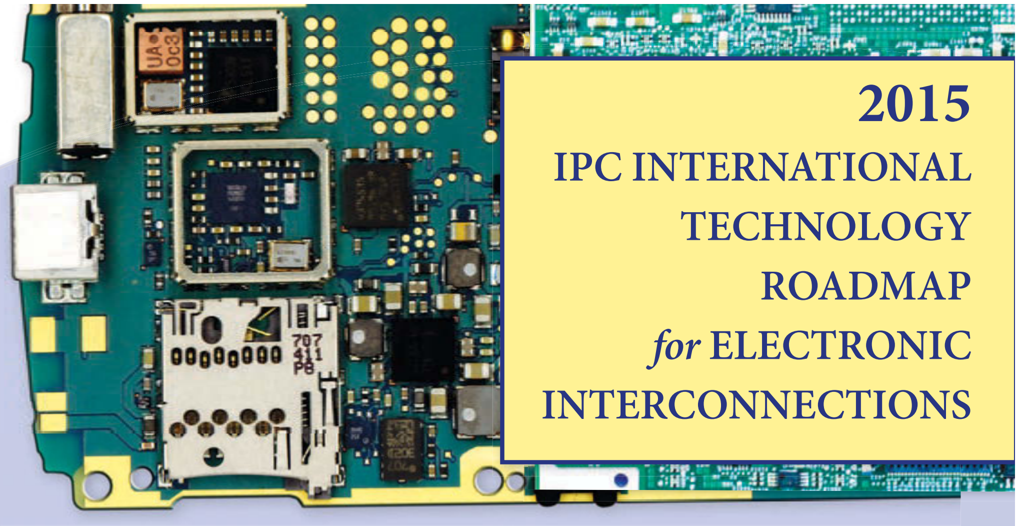
1. "Minimizing Solder Paste Printing Variation," George Babka, Assembléon America Inc., Circuits Assembly Magazine, Nov 2009, Vol. 20 Issue 11.

2. "Effect Of Squeegee Blade On Solder Paste Print Quality," Rita Mohanty, Bill Claiborne. Speedline Technologies, Franklin, Massachusetts, and Frank Andres, Cookson Electronics, South Plainfield, NJ. June 2010.

3. "Troubleshooting the Stencil Printing Process," Chrys Shea, Shea Engineering Services, SMTA Houston Chapter, slide presentation.

Michael Martel is in marketing with Speedline Technologies Inc.

IPC®



2015 IPC INTERNATIONAL TECHNOLOGY ROADMAP *for* ELECTRONIC INTERCONNECTIONS

Can You Compete in the Electronics Industry?

The *IPC International Technology Roadmap* provides valuable marketing and forecasting information and the technological requirements now and for about five years into the future to design, build, and assemble electronic interconnect products, with a heavy, historically based emphasis on printed boards and printed board assemblies that answer questions like:

- What materials will be needed in the coming years?
- What kind of equipment will need to be purchased?
- What type of training protocols will need to be adapted?

Gain valuable insights into the technology needs of tomorrow and put the informational resources of the *IPC International Technology Roadmap* to work for your long-range and strategic planning activities.

What's New and Valuable in 2015 and Beyond

- **Laser-aided manufacturing — for yield improvements**
- **Equipment development**
- **Demand for improved vision systems required**

What Do You Need to Prepare for?

Tremendous increase in clock speed in low-cost/high volume applications predicted over next 5 years.

Much smaller discrete components will continue shift towards emplace embedded and away from formed embedded.

Global material concerns are increasing.

Are you capable of succeeding?

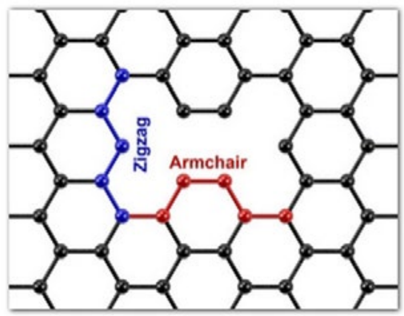
With data compiled from all over the world, this comprehensive roadmap is a must-have for any organization. Get the knowledge and research at your fingertips and start making informed business decisions for today and tomorrow. Order your copy now!

www.ipc.org/roadmap

'Holey' Graphene for Energy Storage

Engineers at the University of California, San Diego have discovered a method to increase the amount of electric charge that can be stored in graphene, a two-dimensional form of carbon. The research, published recently online in the journal Nano Letters, may provide a better understanding of how to improve the energy storage ability of capacitors for potential applications in cars, wind turbines, and solar power.

Capacitors charge and discharge very fast, and are more useful for quick large bursts of energy, such as in camera flashes and power plants. Their ability to rapidly charge and discharge is an advantage over the long charge time of batteries. However, the problem with capacitors is that they store less energy than batteries.



Zigzag and armchair defects in graphene.

How can the energy storage of a capacitor be improved? One approach by researchers in the lab of Mechanical Engineering Professor Prabhakar Bandaru, at the Jacobs School of Engineering at UCSD, was to introduce more charge into a capacitor electrode using graphene as a model material for their tests. The principle is that increased charge leads to increased capacitance, which translates to increased energy storage.

The team used a method called argon-ion based plasma processing, in which graphene samples are bombarded with positively-charged argon ions. During this process, carbon atoms are knocked out of the graphene layers and leave behind holes containing positive charges—these are the charged defects. Exposing the graphene samples to argon plasma increased the capacitance of the materials three-fold.

VIDEO INTERVIEW

The Value of Solder Paste Print On-Demand

by Real Time with...
IPC APEX EXPO 2015

Mycronic's Nico Coenen and Mattias Jonsson sit down with I-Connect007 Guest Editor Steve Williams to take an in-depth look at the value of solder paste print on demand. One of the benefits is that the technology allows the option of optimizing the paste volume on each and every component, for each pad.

CLICK TO VIEW

realtimewith.com

Mil/Aero007
News Highlights

Insitu: Moving Mountains in a Mountain Community

Nearly 800 employees build and support Insitu's UAS line of ScanEagles, Integrators and ground stations. Boeing's cameras captured the first images inside of Insitu's new production facility, Eagle Point. Watch the video for a peek inside and learn more about where high-tech meets mountain height.

IPC Survey Assesses Pb, Pb-free Issue in Military Electronics

IPC launched a "fast facts" survey today to assess the current state of the supply chain for military electronics suppliers and the issues surrounding the trend toward lead-free electronics. All manufacturers of printed circuit boards (PCBs) and electronic assemblies for military or aerospace applications are invited to participate.

Plexus Secures Accreditation from U.S. DoD

Plexus Corp. today announced that its wholly owned subsidiary, Plexus Aerospace, Defense and Security Services LLC, has received accreditation as a Microelectronics Trusted Source by the Defense Microelectronics Activity (DMEA).

Navitas to Provide Circuit Card Assemblies to US Navy

Navitas Systems LLC has been awarded an Indefinite Delivery, Indefinite Quantity (IDIQ) contract by the Department of the Navy, Naval Surface Warfare Center (NSWC), Crane Division.

BAE Systems Installs Second ACE Soldering System

BAE Systems has invested in a second KISS-103IL inline selective soldering system from ACE Production Technologies Inc. The KISS-103IL selective soldering system has been installed at the BAE Systems facility located in Endicott, New York and is the second selective soldering machine BAE Systems has ordered from ACE Production Technologies.

Kitron AS Secures Contract from Lockheed Martin

Kitron AS, a subsidiary of Kitron, has received a contract from Lockheed Martin Mission Systems and Training for production of Integrated Backplane Assembly, for deliveries to the F-35 low rate initial production program, LRIP 9 and 10.

Sypris Electronics' Revenue Down on U.S. DoD Funding Uncertainties

Revenue for our electronics group was $7.1 million in the fourth quarter of 2014 compared to $10.0 million in the prior year, reflecting lower product sales to overseas customers and budgetary and funding uncertainties within the U.S. Department of Defense.

TT Electronics-IMS Earns Cubic Global Defense Award

TT Electronics was recognized for exceptional quality and customer service by a leading provider of realistic training systems and highly specialized support services for national military and security forces.

Sparton Expands Offering; Acquires Stealth.com

"The addition of the Stealth.com business meets the criteria of our growth strategy by further expanding our engineered components and products offering in the ruggedized electronics space," stated Cary B. Wood, president and CEO of Sparton. "Stealth.com complements our existing offerings under Aydin Displays and KEP Marine, allowing us to serve a wider breadth of customers with a greater variety of products and services."

Nortech Systems to Move to New Corporate Headquarters

Nortech Systems, Inc., a leading provider of full-service electronic manufacturing services, will be moving to a new corporate headquarters by early July 2015. The company signed a lease for a 19,000-square-foot facility in Maple Grove, Minnesota.

here at Hunter Technology,

we make it happen!

For over forty-five years now, we have been "making it happen" for our customers. By having an unsurpassed committment to quality. By providing the EMS services that you need. And mostly by being there for you, every step of the way. Here's what our customers are saying:

"I didn't believe that NPI and production support could peacefully co-exist. Hunter has proven me wrong."

AMD

"Hunter's engineering support is second to none. I used to think the tier 1's could only offer that level of expertise and capability. I stand corrected."

OCLARO

"I engaged Hunter with the hope of resurrecting an engineering bone pile. My expectations were greatly exceeded. Hunter is now my largest EMS Supplier."

ZONARE

"With the most challenging technology, I take it to Hunter because I know it will get done right."

NORTHROP GRUMMAN

We couldn't make anything happen without our great staff. Here are few words from the people that make it happen:

Call us today to find out what we can make happen for you!

HUNTER TECHNOLOGY CORP
Reliable. Solutions. Delivered.

+1 (408) 957-1300
www.hunter-technology.com

Selecting an Automatic Pick-and-Place Machine, Part 3

by Robert Voigt
DDM NOVASTAR

In this month's installment in this series of columns aimed at helping buyers analyze and select SMT equipment for printed circuit board assembly, I will address features that affect decisions on the selection and purchase of an appropriate automatic pick-and-place machine. As with any complex machine, there will be tradeoffs between cost and capabilities, some of which specifically relate to production accuracy and yield. We will address:

1. Mechanical positioning methods
2. Machine construction
3. Solder paste fluid dispensing
4. Component feeders

To review, when starting the evaluation process, there are two defining factors to keep in mind which determine what category fits your machine needs. The first principal factor is CPH (components per hour), and the secondary factor is machine capability. While it's constructive to start by understanding how production rates affect the type and performance of a pick-and-place machine, please refer to the prior two installments, in the February and March issues, for those ranges.

Machine capability is the second defining factor in helping choose the correct auto pick-and-place machine for your needs. In this installment, we will address three aspects of machine capability that have a direct impact on final board quality and production yield.

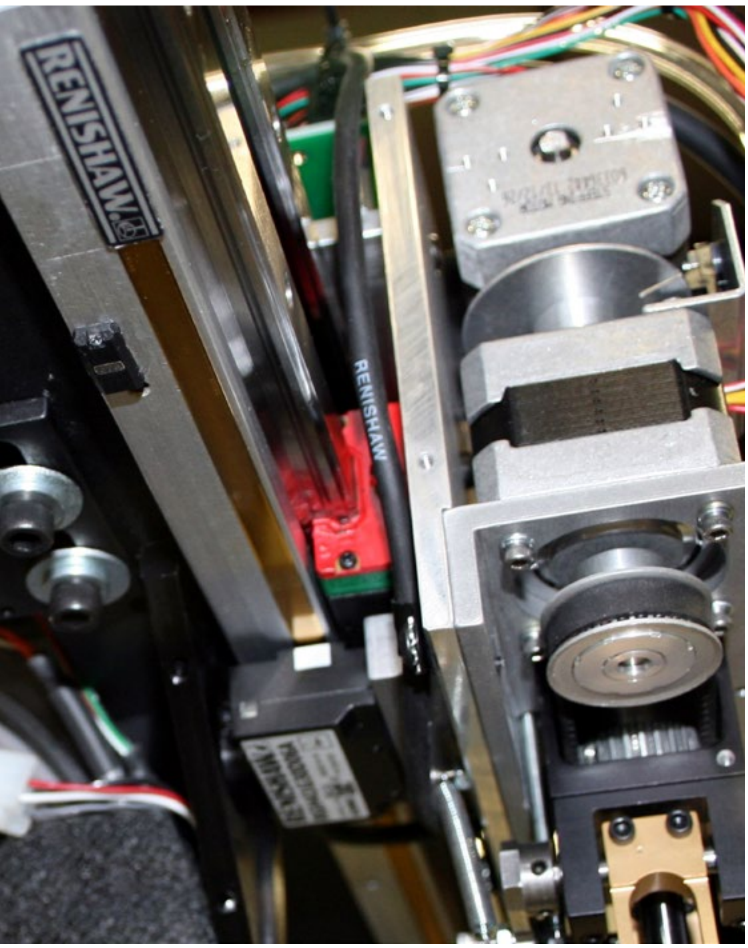


Figure 1: Underside view of a Renishaw linear scale and encoder, one of the most accurate brands for mechanical positioning.

Accuracy and Repeatability

For production machines, we typically recommend looking for a machine with accuracy of +/- .001" and fine pitch capability, down to 12 mils, on a repeated basis. Be aware that less expensive machines rarely meet this spec. There are several types of positioning systems employed, differences in construction methods, and a variety of component feeders, all of which have an impact on quality and yield.

Component Positioning Systems

After each component is picked up and centered in the tool by one of the methods described in Part 2, it must then be positioned ac-

International Conference on Soldering and Reliability

May 19 – 21, 2015
Hilton Markham Suites
Markham, Ontario, Canada

SMTA Soldering and Reliability

www.smta.org/icsr

Conference Overview

Join us for the 9th annual International Conference on Soldering and Reliability. Materials as different as graphene, nanofibers and coatings will be discussed. Common topics like stencils for disparate-sized components, tin whiskers, head on pillow and specific reliability issues will be highlighted. RoHS initial implementation may be well behind us, but the implications are still resounding through our industry. There will be two sessions on lead free solders, with one concentrating on bismuth-containing possibilities. It will be an information-packed two days!

Workshops: May 19

Conference: May 20 – 21

Toronto Expo & Tech Forum: May 21

Keynote Address:
Electronic Industry Collaboration – Critical 21st Century Supply Chain Tools

Bill Bader
CEO, iNEMI

Co-located Toronto Expo & Tech Forum

Your local electronics assembly and manufacturing trade show!

- Free technical sessions
- Free lunch
- Networking

Technical Committee:

Bev Christian, Ph.D., *BlackBerry*
Martin Anselm, Ph.D., *Rochester Institute of Technology (RIT)*
Peter Arrowsmith, Ph.D., *Ops A La Carte*
Matthew Kelly, P.Eng, MBA, *IBM Corporation*
Polina Snugovsky, Ph.D., *Celestica, Inc.*
Tim Luke, *Alpha*

Supporting Partner:

ReMAP Refined Manufacturing Acceleration Process
AAPF Accélération de l'amélioration des processus de fabrication

Media Partners:

SMT — an I-Connect007 publication
CIRCUITS ASSEMBLY

curately on the board, in an X-Y position. Following are three methods commonly used for positioning, along with pros and cons for each.

1. Positioning with no feedback system (open loop system)
2. Positioning with rotary encoders (closed loop system)
3. Positioning with linear encoders (closed loop system)

Method 1: No positioning feedback loop

In this system, the motor drives the part to a location on the board defined in the program by the number of steps in each X-Y axis, but there is no way to tell if it actually ends up in the right place. These systems use stepper motors for positioning.

Pros: Low cost.

Cons: Unreliable accuracy; not recommended for high quality production.

Method 2: Positioning with rotary encoder

In this method, an encoder is mounted directly on the motor shaft and delivers position feedback to the control system; however, it only reports the motor position, and not the actual position of the X-Y axis. This is dependent upon the remainder of the mechanical components that make up the machine. These machines can use stepper or servo motors (which are usually associated with cost).

Pros: Low cost; this system is widely used on entry-level machines.

Cons: Typical positioning accuracy of +/- .005".

Method 3: Positioning with linear encoder

In this method, linear scales are mounted to the machine's X-Y axes table and an encoder is mounted on the traveling beam that will be carrying the components. This method will report its actual position back to the control system and

Figure 2: View of a pick-and-place machine frame being welded together.

make corrections to the position programmed, if needed, to within a few microns of the actual X-Y location for the component placement (typically 12,800 increments, or steps, for each inch of travel). The best machines in this category use servo motors.

Pros: Very high accuracy, to within +/- .0005"; very repeatable.

Cons: More costly, but necessary for high-value production.

NOTE: The quality of the encoder (the position feedback sensor), is an important element in the whole system and does affect accuracy.

Machine Construction

When selecting a pick-and-place machine, be aware that its construction will dictate its effective CPH range and footprint, including considerations for the number of component feeders it can accommodate.

1. All-welded steel: The most accurate machine will have a frame that is constructed of a solid welded structural steel tube. This provides significant stability necessary for accurate positioning and high-speed movement of X-Y axes. This construction method is recommended for any production environments, and it will remain stable without requiring ongoing calibration.

2. Bolt-together frame: Extruded aluminum or formed sheet metal frame will come with a lower initial accuracy than a welded frame and will need to run more slowly because it can't handle the rapid inertia shifts of X-Y axes movement. Further, it will likely go out of calibration frequently, which will adversely impact labor time, downtime and yield. (Lower cost usually reflects a weaker construction.)

Solder Paste/Fluid Dispensing

Any pick-and-place machine should be capable of offering fluid dispensing systems. Most common liquids include solder pastes, adhesives, lubricants, epoxies, fluxes, glue, sealants, and more. This is a valuable option when building prototypes or one-off PCB assemblies that do not warrant the cost of a dedicated printer stencil or foil.

Component Feeders

If the machine's production will be dedicated to jobs with a small number of components it's very easy to identify the number and type of feeders. However, that is not usually the case with contract assembly shops, since they don't know what type of board and how many different components the next job will require. Some OEMs also need flexibility for a wide range of board configurations, especially if they intend to use the same machine for prototypes and several different production boards.

So it is useful in those cases to consider a machine with the greatest number of feeder positions and options that can accommodate the footprint your space can handle.

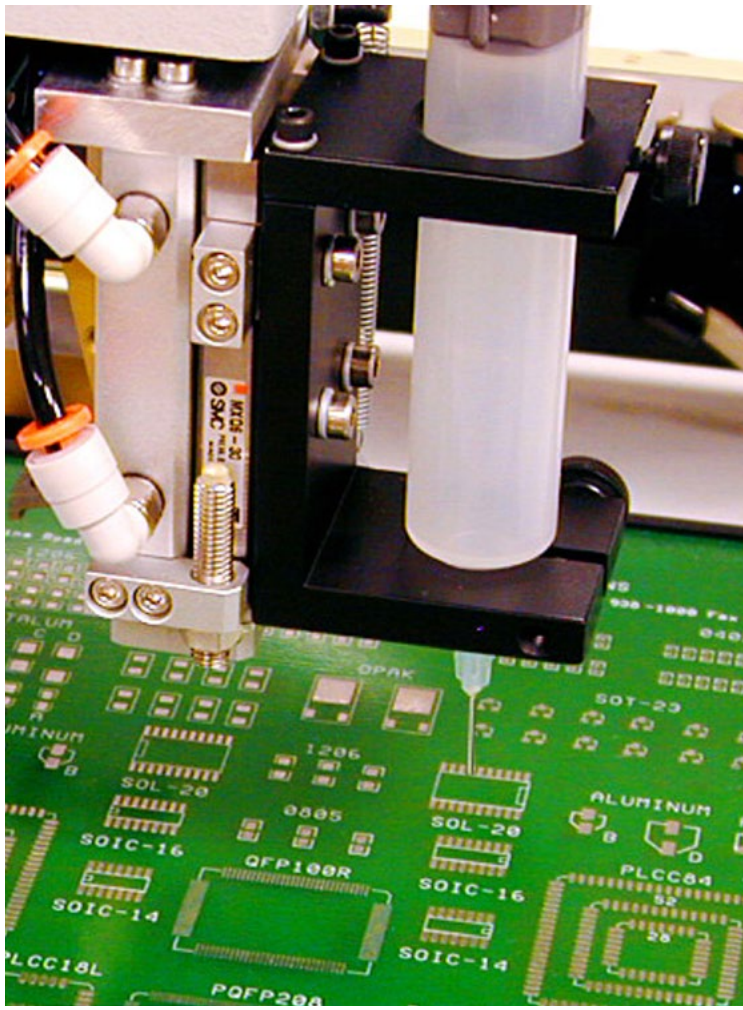


Figure 3: Fluid/solder paste dispenser, typically mounted to the head.

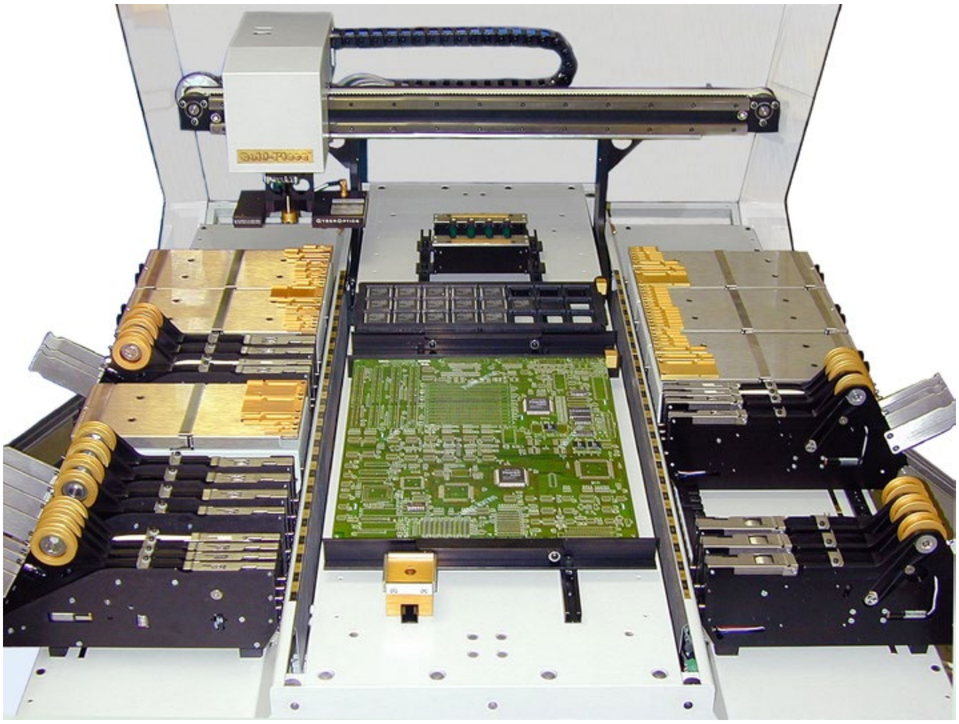


Figure 4: Example of a work deck with multiple feeder types and flexible configurations.

Types of feeders include:

1. Cut strip holders: usually associated with low-volume.

2. Matrix tray holders: used for components not available on tape.

3. Tube feeders: dispense components supplied in tubes.

4. Electric tape (and reel) feeders: usually more costly initially, but offer the best long-term investment. Electric tape feeders are available as single units in a variety of sizes, and cover the range of 0201 components up to 56mm large components. Many manufacturers now offer a multiple feeder (known as bank feeder). These are usually available for 8mm tape, and can come with up to 8–12mm feeder lanes per unit.

Since components are packaged in many forms (e.g., discrete components on tape, quad packs, matrix trays, tubes, cut strips, etc.), your choice of feeders would depend on production, but also on any size restrictions you may have. A good starting point is to purchase the most feeders you can get in the footprint you have available.

Vendor Support

When evaluating any type of SMT machine, consider factory support to be one of the most important assets of your purchase. The best way to learn how a company treats its customers is by word of mouth. Talk to several customers to find out how happy they are with the machine, the seller, and the support they provide. Where is the manufacturing plant? Can they help troubleshoot alignment issues over the phone? Do they offer field service? Do they have spare parts in stock for immediate shipment? While there isn't much of a used market for manual, machine-assisted or enhanced manual pick-and-place machines, it is still a good idea to ask your supplier about their older machines in the field, if down the road spare parts will be available, and about their capability to customize a spare part if the machine becomes obsolescent. Ask what the expected life-cycle of the product is. The industry standard is seven years. Remember, there is a difference between a true manufacturer and an equipment supplier or distributor.

In the next installment, Pick-and-Place, Part 4, I will cover software interface, utilities/programming, optimization, CAD translation and offline software. **SMT**

Robert Voigt is VP of global sales at DDM Novastar Inc. To reach Voigt, click here.

2015 Programs

May 13–14
IPC Technical Education
Fort Worth, TX, USA
Professional development courses for engineering staff and managers:
- DFX-Design For Excellence (DFM, DFA, DFT and more)
- Best Practices in Fabrication
- Advanced Troubleshooting
- SMT Problem Solving

June 9
ITI & IPC Conference on Emerging & Critical Environmental Product Requirements
Fort Lee, NJ, USA

June 9–10
IPC Technical Education
Chicago, IL, USA
Professional development courses for engineering staff and managers:
- DFX-Design For Excellence (DFM, DFA, DFT and more)
- Best Practices in Fabrication
- Advanced Troubleshooting
- SMT Problem Solving

June 10
ITI & IPC Conference on Emerging & Critical Environmental Product Requirements
Des Plaines, IL, USA

June 12
ITI & IPC Conference on Emerging & Critical Environmental Product Requirements
Milpitas, CA, USA (San Jose area)

September 27–October 1
IPC Fall Standards Development Committee Meetings
Rosemont, IL, USA
Co-located with SMTA International

September 28
IPC EMS Management Meeting
Rosemont, IL, USA

October 13
IPC Conference on Government Regulation
Essen, Germany
Discussion with international experts on regulatory issues

October 13–15
IPC Europe Forum: Innovation for Reliability
Essen, Germany
Practical applications for meeting reliability challenges like tin whiskers, with special focus on military-aerospace and automotive sectors

October 26–27
IPC Technical Education
Minneapolis, MN, USA
Professional development courses for engineering staff and managers:
- DFX-Design For Excellence (DFM, DFA, DFT and more)
- Best Practices in Fabrication
- Advanced Troubleshooting
- SMT Problem Solving

October 28–29
IPC Flexible Circuits-HDI Conference
Minneapolis, MN, USA
Presentations will address Flex and HDI challenges in methodology, materials, and technology.

November 2–6
IPC EMS Program Management Training and Certification
Chicago, IL, USA

November 4
PCB Carolina 2015
Raleigh, NC, USA

December 2–3
IPC Technical Education
Raleigh, NC, USA
Professional development courses for engineering staff and managers:
- DFX-Design For Excellence (DFM, DFA, DFT and more)
- Best Practices in Fabrication
- Advanced Troubleshooting
- SMT Problem Solving

December 2–4
International Printed Circuit and APEX South China Fair (HKPCA & IPC Show)
Shenzhen, China

Questions? Contact IPC registration staff at registration@ipc.org or +1 847-597-2861

Recent Highlights from SMT007

1 Sparton Acquires Hunter Technology Corporation

Sparton Corporation announced that its wholly owned subsidiary, Sparton Hunter Corporation, completed a merger with Hunter Technology Corporation in a $55 million all-cash transaction. The merger is subject to certain and conditional post-closing adjustments.

2 SMTA Capital Chapter Welcomes New Officers

The SMTA Capital Chapter is pleased to announce the officers for the 2015 term. Brian A. Costanzo of DRS Signal Solutions (President); Karen Walters Ebner of Raytheon Systems Company (Vice President); Bert Horner of the Test Connection Inc. (Secretary); Fernando Rueda of Kyzen Corporation (Treasurer); Tom O'Connor of DfR Solutions (VP of Membership); Jessica Kreis of ZESTRON Americas (VP of Communications); and Martin Anselm of Rochester Institute of Technology (Board Liaison for the SMTA Capital Chapter).

3 LACROIX Electronics Joins LoRa Alliance

LACROIX Electronics is now a member of the LoRa alliance in order to meet increasing customer demands on wireless communication for long distance and low power consumption. For several years, LACROIX Electronics has developed significant expertise in wireless and LoRa is now a complementary technology to radio communication industrialization services.

4 Yokogawa to Transfer PCB Factory to OKI

OKI announced an agreement with Yokogawa Electric Corporation on the transfer of Ome Factory of Yokogawa Manufacturing Corporation, a PCB manufacturing and assembly facility, to OKI Printed Circuits, an OKI Group company responsible for OKI's printed circuit board business, effective April 1.

5 Flextronics Joins Daintree Networks' Partner Program

Daintree Networks, the leader in open networked wireless control and operation solutions for smart buildings, today announced that Flextronics, a leading innovative supply chain solutions company, has officially joined its ControlScope Connected Partner Program.

6 STI Electronics Earns IPC's Validations Services QML

IPC's Validation Services Program has awarded an IPC J-STD-001, IPC-A-610, and IPC J-STD-001 Space Addendum Qualified Manufacturers Listing (QML) to STI Electronics Inc. (STI), an electronics manufacturer and IPC Training Center in Madison, Alabama.

7 Bellerophon Taps Flextronics to Manufacture Mark2

Bellerophon Therapeutics Inc. has selected Flextronics International Ltd. as its manufacturing partner for the INOpulse Mark2, the company's next-generation pulsatile nitric oxide delivery device. Bellerophon and Flextronics have entered into an agreement under which Flextronics will manufacture, repair and service the Mark2 devices to be used in Bellerophon's INOpulse clinical development programs.

8 IMI Outperforms EMS industry's Growth Rate in 2014

"The company's remarkable growth is a result of various drivers—a wider and more diversified geographic footprint, a steady increase in demand from original equipment manufacturer customers, and a deep focus on product innovation and customer service. More importantly, we have improved our use of manufacturing technology and design intelligence in our products and operations," said Jaime Augusto Zobel de Ayala, chairman of IMI.

9 SMTC Suffers from 15.6% Revenue Drop in FY2014

Revenue for the year was $228.6 million, a 15.6% decrease from 2013. The majority of the decrease in revenue was due to reduced volumes with two customers. The gross profit percentage for 2014 was 8.3%, compared to 5.6% in the prior year, as a result of cost cutting and improvements in manufacturing efficiencies.

10 NOTE Posts Sales Growth in 2014; Sees Positive 2015

"In 2014, we advanced against the competition on a fairly stable European market. After a strong third quarter with 17% sales growth, we anticipated stable but weaker sales performance in Q4. Sales in Q4 were down 3% year on year, for the full year, sales were up by over 6%," commented by NOTE's CEO, Peter Laveson.

EVENTS

For the 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.

Michigan Expo & Tech Forum
May 5, 2015
Livonia, Michigan, USA

Oregon Expo & Tech Forum
May 5, 2015
Beaverton, Oregon, USA

Puget Sound Expo & Tech Forum
May 7, 2015
Bellevue, Washington, USA

Wisconsin Expo & Tech Forum
May 12, 2015
Milwaukee, Wisconsin, USA

IPC Technical Education
May 13–14, 2015
Fort Worth, Texas, USA

International Conference on Soldering & Reliability 2015
May 19–21, 2015
Markham, Ontario, Canada

Toronto SMTA Expo & Tech Forum
May 21, 2015
Markham, Ontario, Canada

Huntsville Expo & Tech Forum
June 4, 2015
Huntsville, Alabama, USA

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements
June 9, 2015
Fort Lee, New Jersey, USA

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements
June 12, 2015
Milpitas, California, USA

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

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

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

EDITORIAL:
MANAGING EDITOR: **LISA LUCKE**
(209) 304-4011; lisa@iconnect007.com,

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

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **SHELLY STEIN** shelly@iconnect007.com
MAGAZINE LAYOUT: **RON MEOGROSSI**
AD DESIGN: **SHELLY STEIN, MIKE RADOGNA**
INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**
COVER PHOTO: **DENIS DRYASHKIN**

SMT® (Surface Mount Technology) is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138

May 2015, Volume 30, Number 5 • SMT© (Surface Mount Technology©) is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *SMT Magazine:*

JUNE:
Test & Inspection

JULY:
Supply Chain Management