As originally published in the IPC APEX EXPO Conference Proceedings.

Printing of Solder Paste – A Quality Assurance Methodology

Lars Bruno and Tord Johnson
Ericsson AB and MTEK Consulting AB
Katrineholm, Sweden and Österskär, Sweden

Abstract
Solder paste printing is known to be 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.

I. Introduction

This article has its roots in the need to improve capacity and quality levels at an electronic 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 being able 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 printed circuit board 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 printed circuit board 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 larger 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 has several properties that make solder paste printing even more challenging. First of all the boards are large in physical size, it is not uncommon with lengths and widths surpassing 450mm and some even stretch to 500mm. 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 above given challenges put several needs onto a 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 unpractical 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.

The above 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?" The question did in turn generate a project that became a methodology.

II. 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 strategy:

- Verify solder paste inspection repeatability and measurement accuracy.
- Optimize the solder paste printing process.
- Create routines to verify that the solder paste printing process capability remains at a high-level and improves.

It is clear that 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.

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.

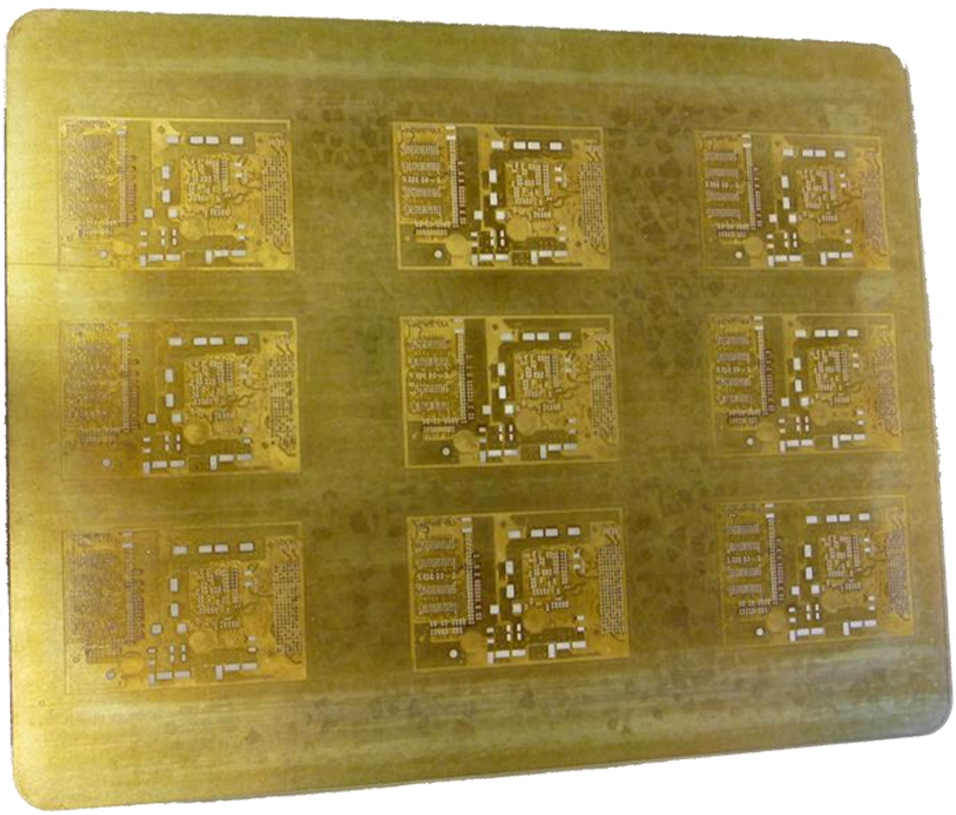

Figure 1 - Reference board for Solder Paste Inspection.

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.

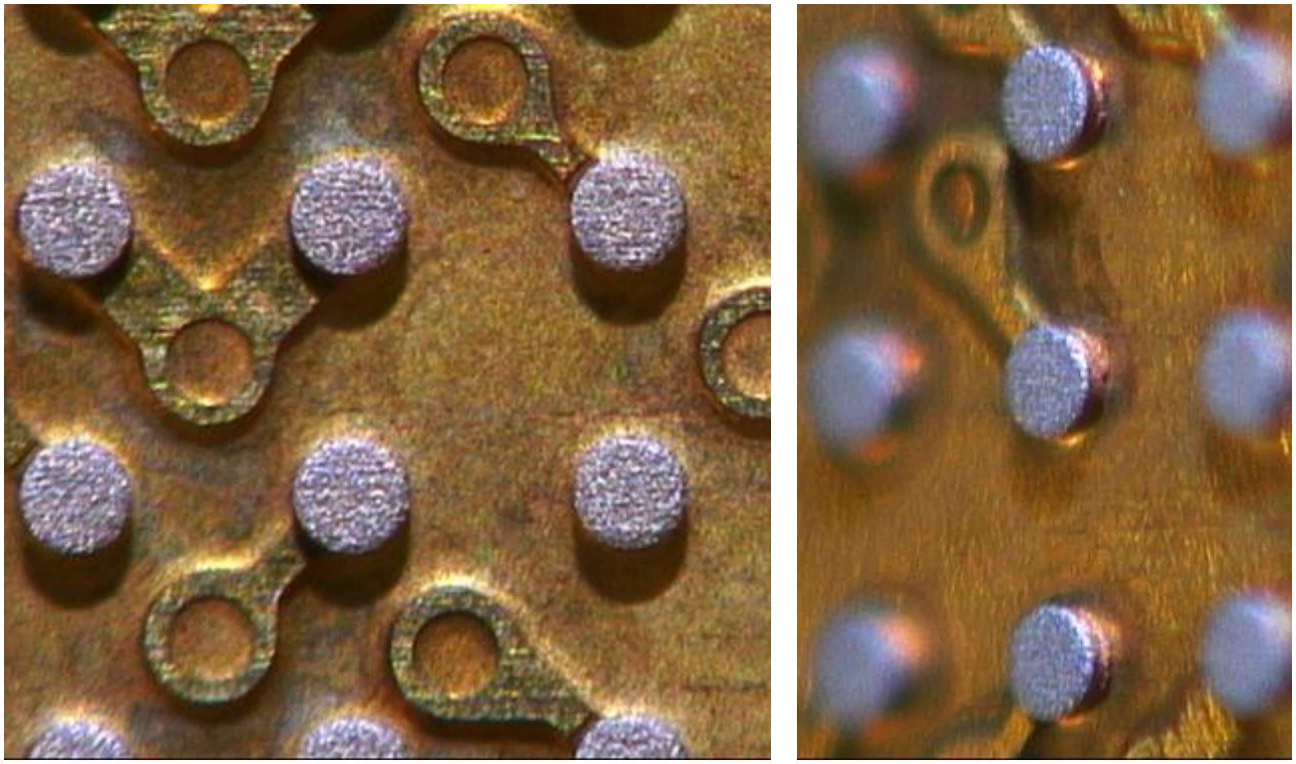

Figure 2 - Close-up pictures of the "paste" deposits on the reference board, from above (left) and at an angle (right).

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 ten different pads were measured ten 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.

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 also have a large impact upon the print results such as board and stencil support, kneeding 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, see 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.

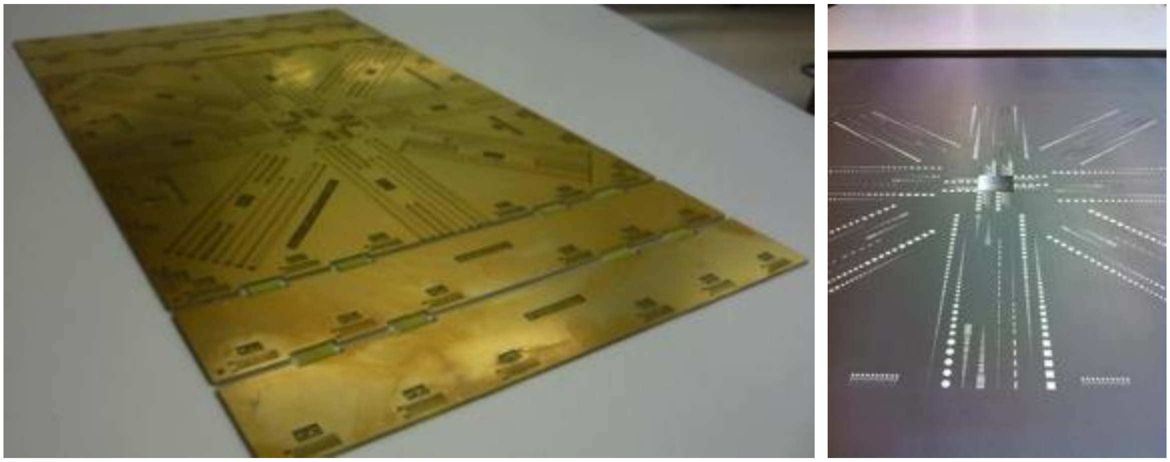

Figure 3 - Test pattern on a printed circuit board and its equivalent on the stencil.

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 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 500mm 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 127µm 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 as depicted in Figure 4.

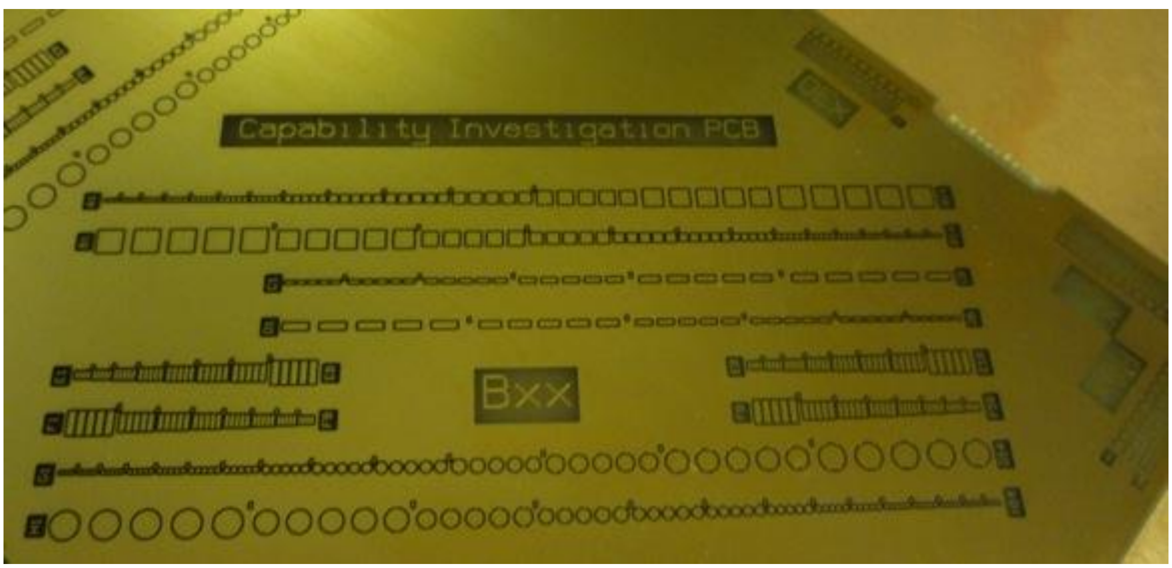

Figure 4 - Pattern within an "arm" of the star pattern.

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. What ever happens is closely related to the following important factors:

- The 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:

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

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

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

Aperture Size							Area Ratio			
Square		Circle	Rectangle 1		Rectangle 2		Square	Circle	Rectangle 1	Rectangle 2
X	Y	D	X	Y	X	Y	X	D	X	X
0.2	0.2	0.2	0.9	0.225	0.2	0.8	0.39	0.39	0.71	0.63
0.25	0.25	0.25	1.1	0.275	0.25	1	0.49	0.49	0.87	0.79
0.3	0.3	0.3	1.4	0.35	0.3	1.2	0.59	0.59	1.10	0.94
0.35	0.35	0.35	1.8	0.45	0.35	1.4	0.69	0.69	1.42	1.10
0.4	0.4	0.4	2.4	0.6	0.4	1.6	0.79	0.79	1.89	1.26
0.45	0.45	0.45	3	0.75	0.45	1.8	0.89	0.89	2.36	1.42
0.5	0.5	0.5			0.5	2	0.98	0.98		1.57
0.7	0.7	0.7			0.7	2.8	1.38	1.38		2.20
0.9	0.9	0.9					1.77	1.77		
1.1	1.1	1.1					2.17	2.17		
1.4	1.4	1.4					2.76	2.76		
1.8	1.8	1.8					3.54	3.54		
2.4	2.4	2.4					4.72	4.72		
3	3	3					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		

As explained above 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.

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

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.

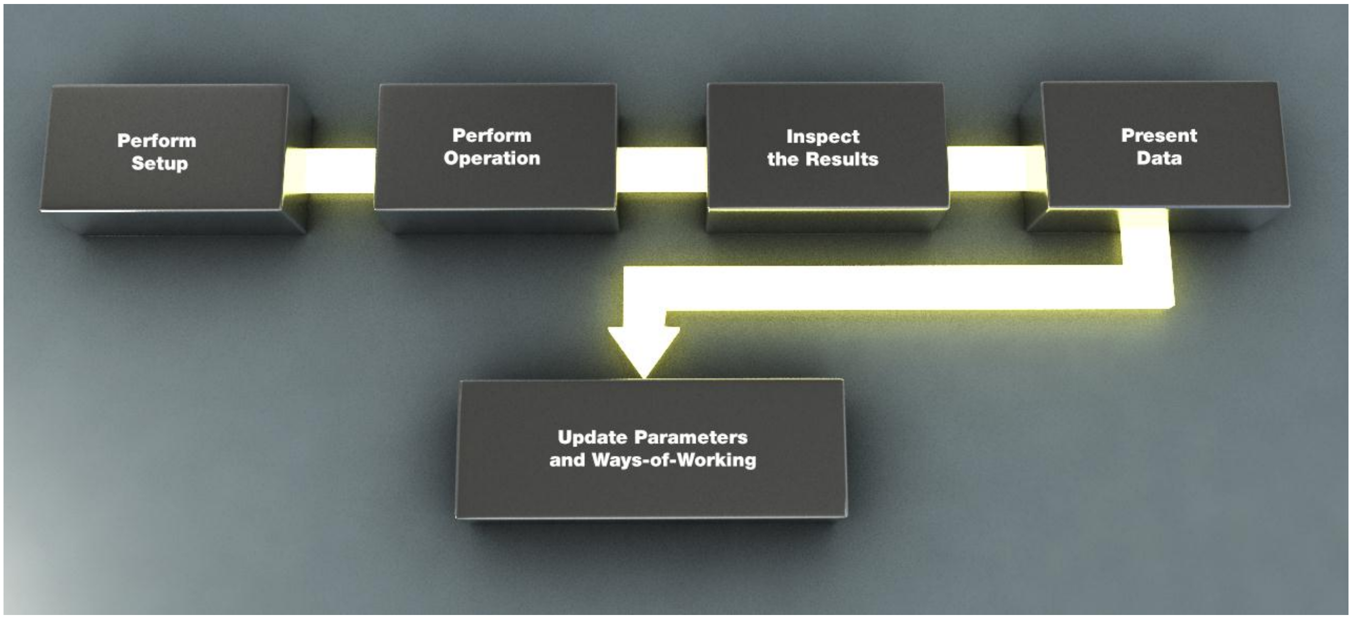

Figure 6 – Capability investigation methodology.

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 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 above 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 Figure 7 to Figure 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 1-1.4bar, for any selected printing speed.

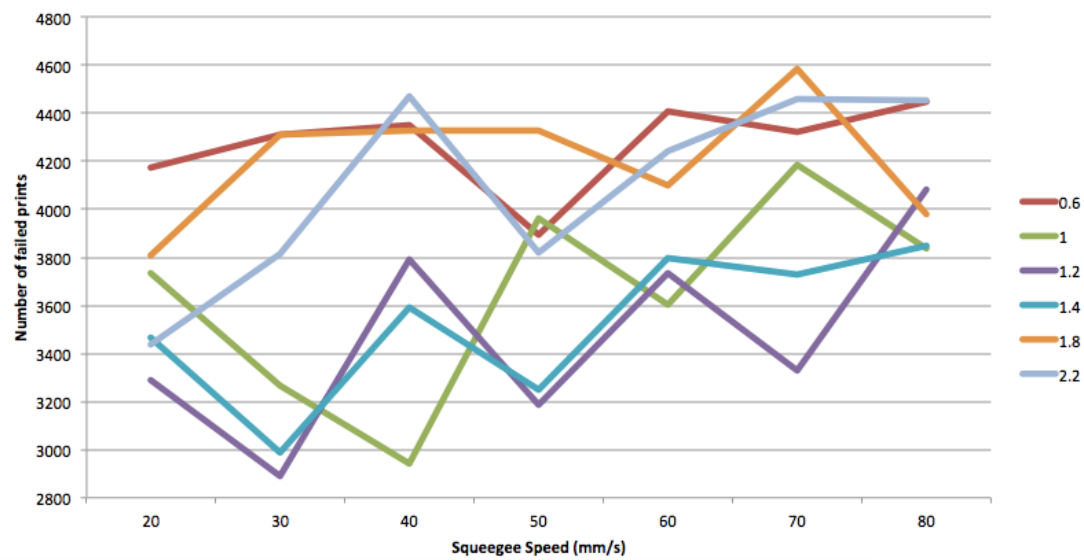

Figure 7 – Results for how the amount of successful deposits varies with the squeegee speed and internal print-head pressure.

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 and 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.

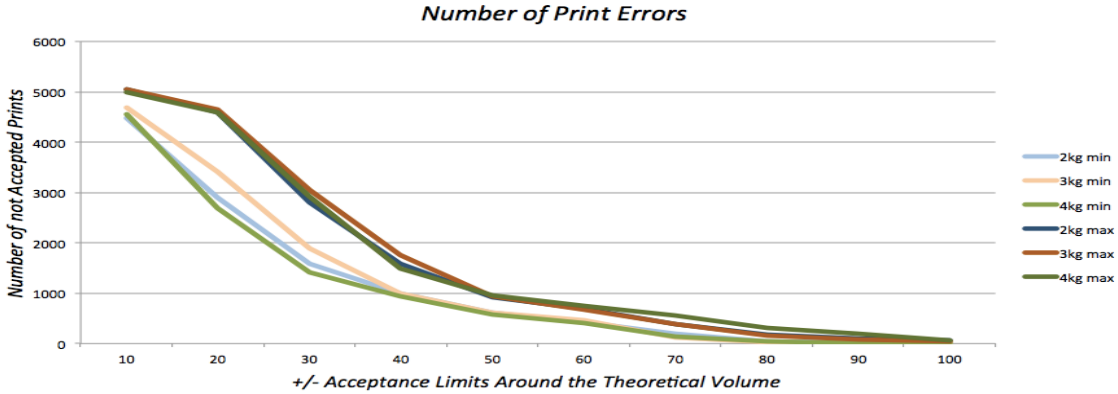

Figure 8 – Results for how the paste misprints vary with the squeegee pressure.

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 2mm/s separation speed.

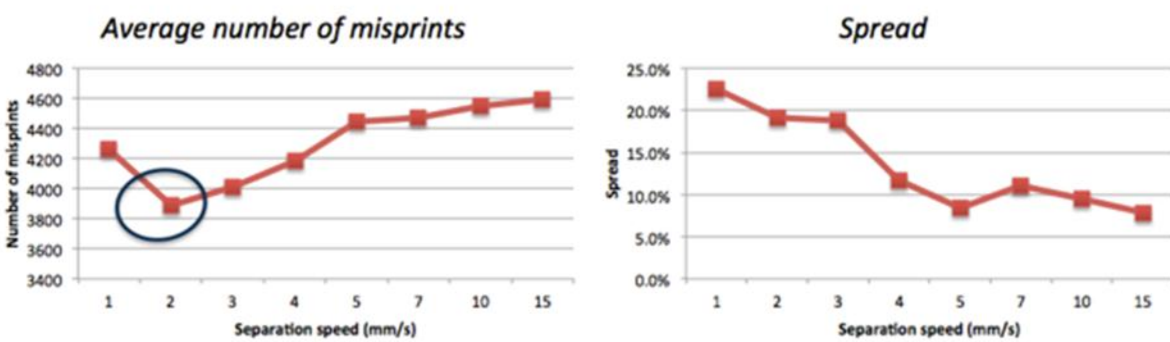

Figure 9 – Results for how the misprints vary with the separation speed.

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.

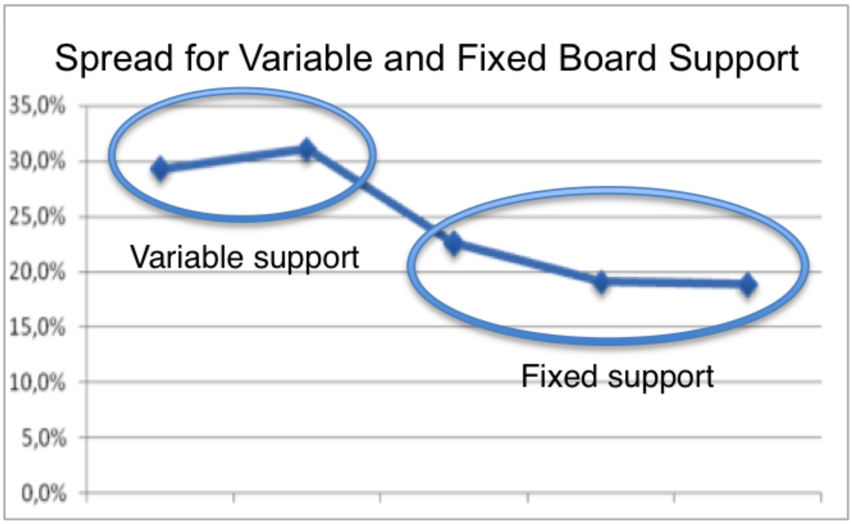

Figure 10 – Results for how the spread of the solder paste deposit varies with separation speed and type of board support (40mm/s print speed, 1.2bar internal pressure, 2.6kg squeegee pressure and ±20% acceptance level for the volume).

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 0.4x0.4mm square apertures. Both solder pastes had Type 4 particle size, but they had different tackiness, rheology and viscosity properties.

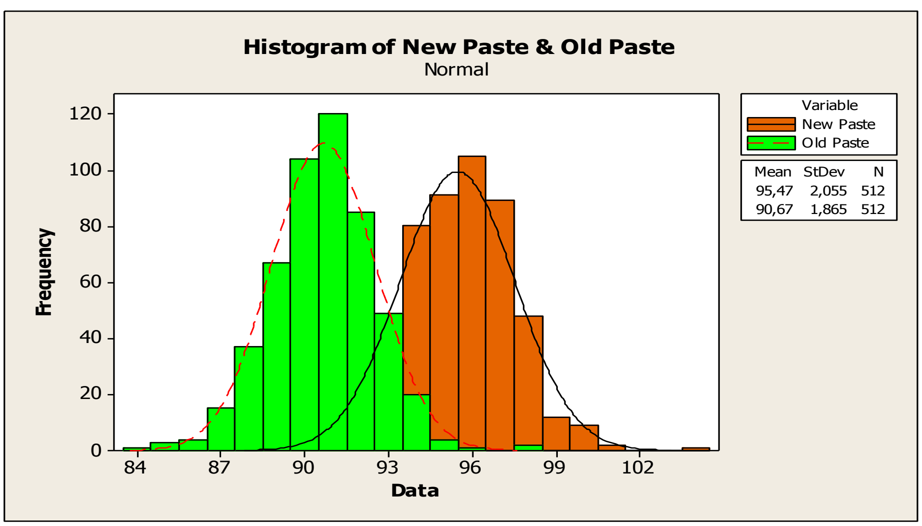

Figure 11 - Solder paste volume distributions from two different solder pastes printed through 0.4x0.4mm square stencil apertures.

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, see Figure 12.

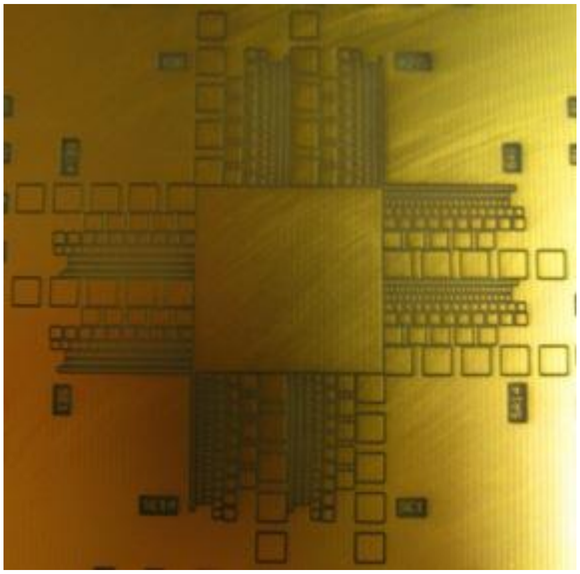

Figure 12 - Test board for step stencil test.

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

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

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

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 1mil, 2mil, 3mil, 4mil and 5mil.

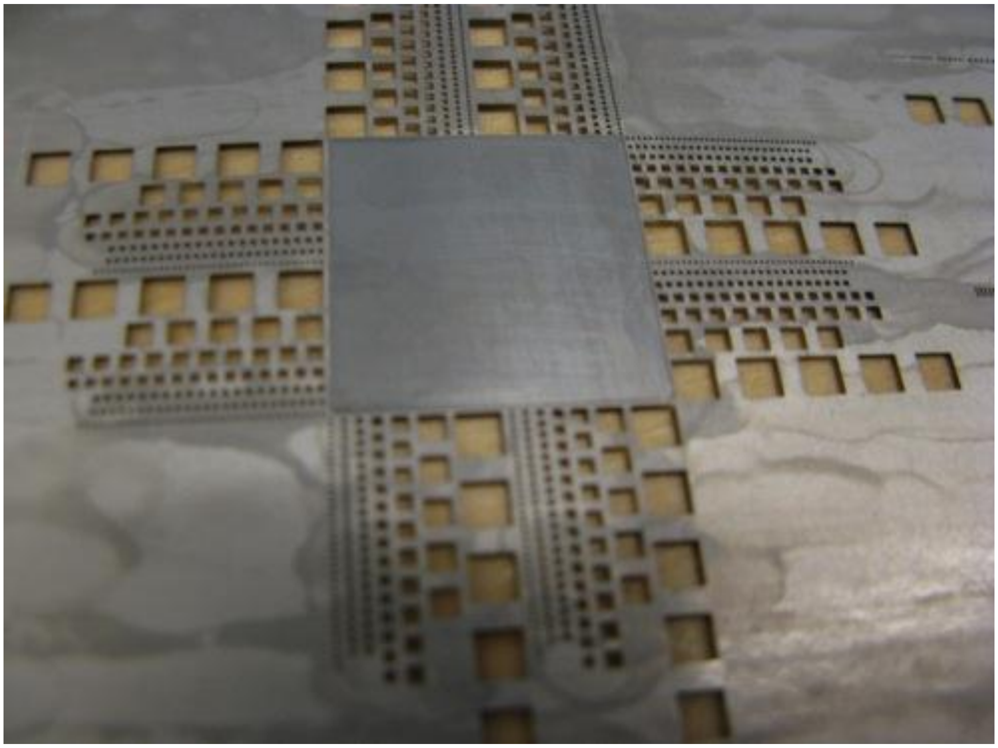

Figure 13 - Step stencil with stepped area in center.

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

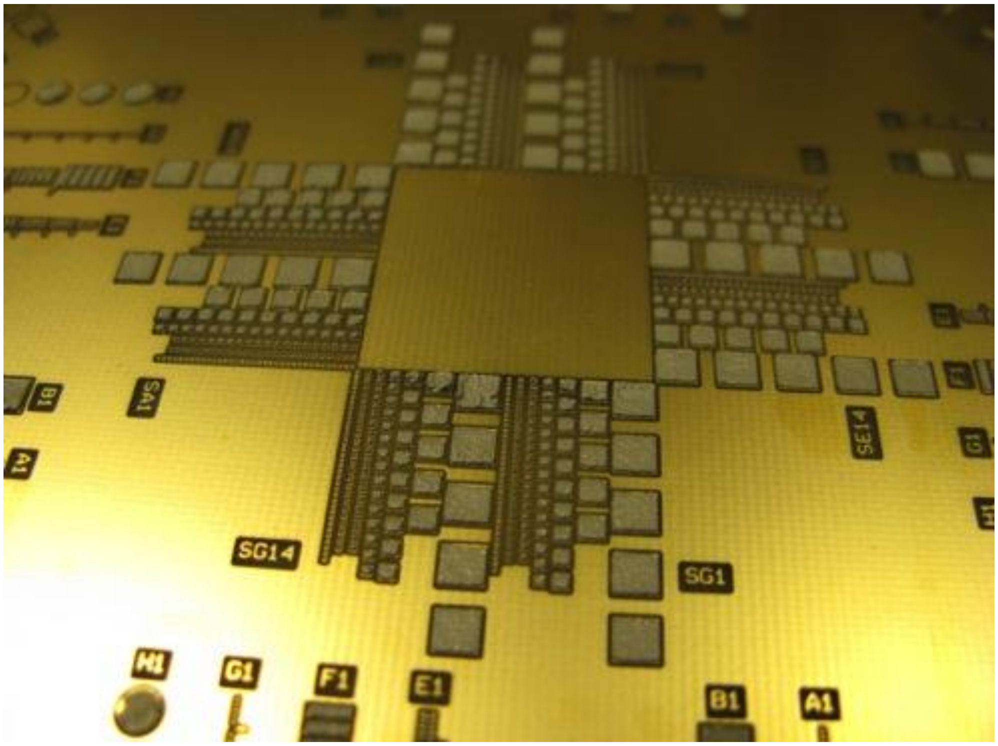

Figure 14 - Print result – print direction: from left to right.

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.

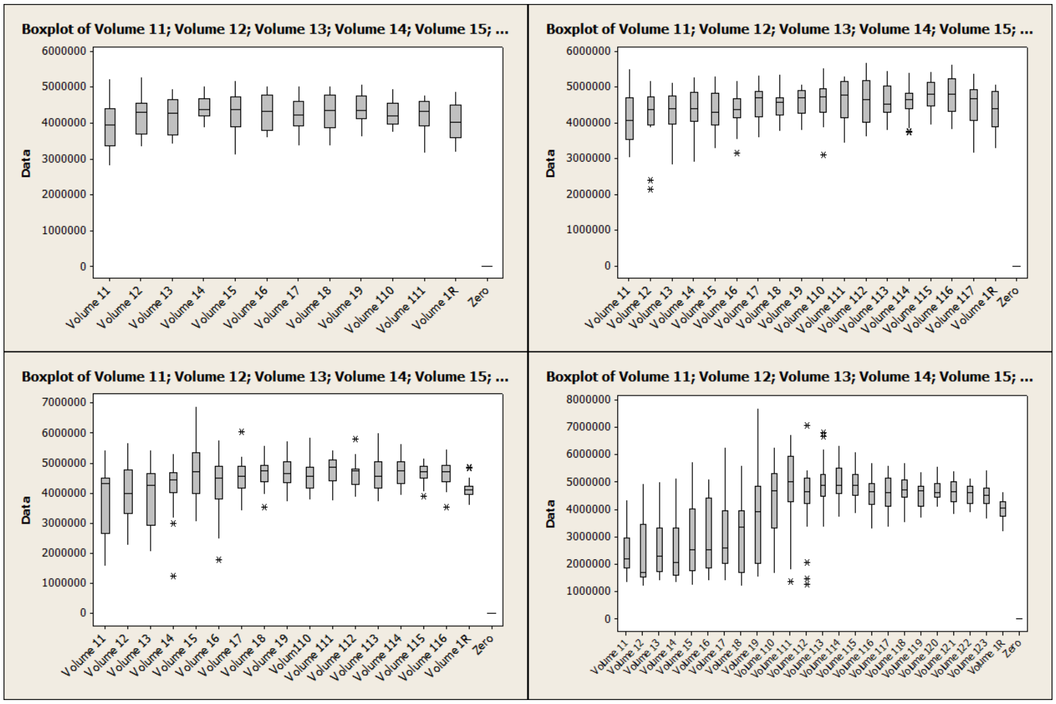

Figure 15 - Box-plots of solder paste volume distributions (µm^3). "Volume 11" is the deposit adjacent to the step edge and volume 1R is the deposit furthest away. All distributions above are from 0.25x0.25mm square stencil apertures. 1mil step (upper left), 2mil step (upper right), 4mil step (lower left) and 5mil step (lower right).

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.

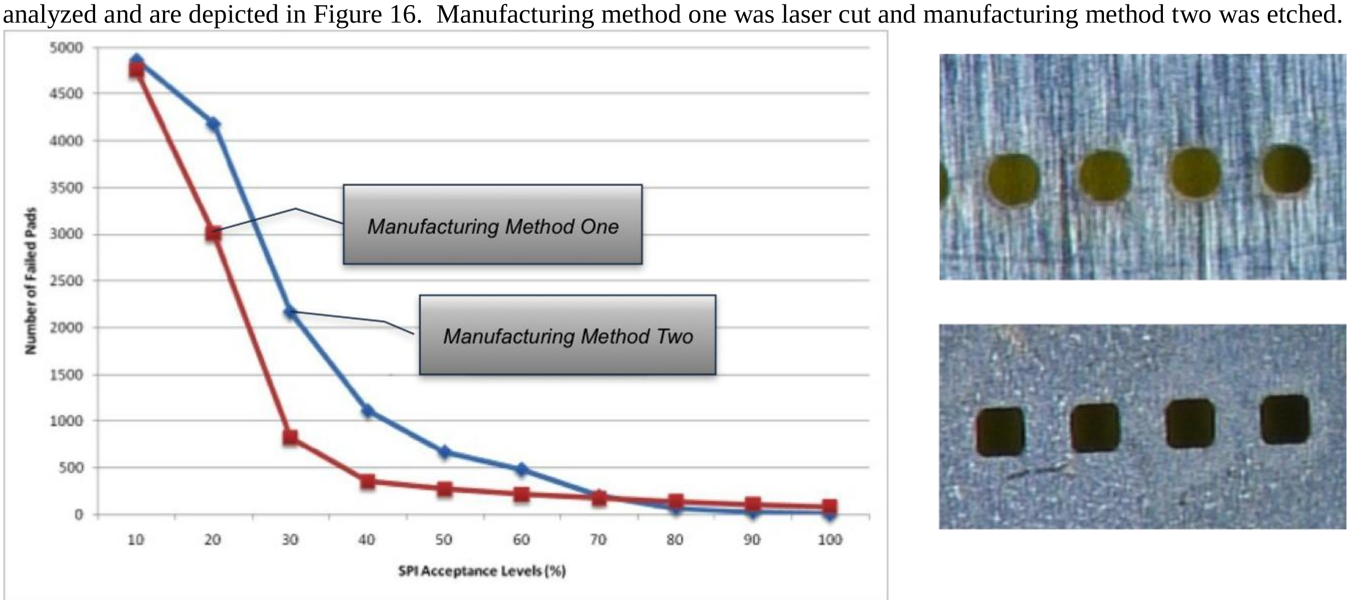

Figure 16 - Amount of solder paste misprints when comparing two different stencil manufacturing methods.

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 how 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 analyze the results to quickly see when a process is deviating from the intended quality levels. By setting routines for this and escalation points, 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.

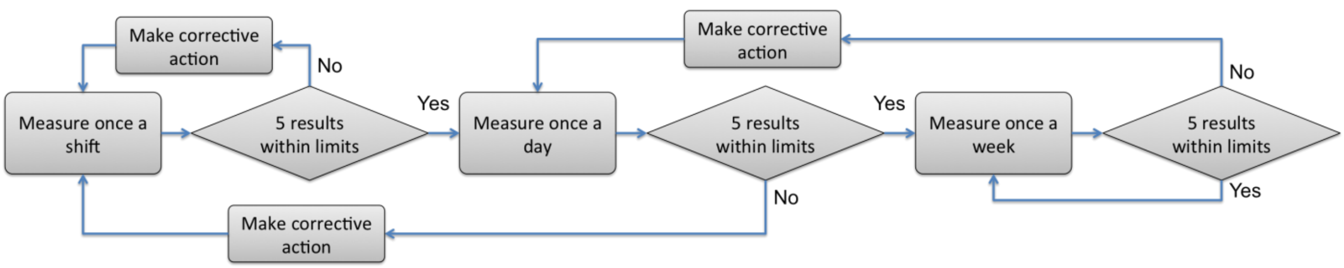

Figure 17 – Workflow for how to implement verification in a production environment.

IV. Conclusion / Summary

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 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.

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:

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