

Innovations in Soldering Materials and Optimization of Solder Paste Printing and Inspection Parameters for System-in-Package Assembly

Evan Griffith, Casey Rowland, David Sbiroli, Jonas Sjoberg
Indium Corporation
301 Woods Park Drive
New Hartford, New York, 13323, United States of America
Ph: 315-381-7487
Email: egriffith@indium.com

Abstract

System-in-Package (SiP) assembly has become a popular solution for the challenges of continued miniaturization in the semiconductor industry. The continued and expected growth of the semiconductor industry in the coming years is driving more advanced assembly methods within System-in-Package and advanced packaging in general, and more advanced semiconductor assembly materials are required to meet those challenges such as solder paste specifically designed for SiP assembly, with fine solder powder and specifically formulated flux vehicles.

Fine feature solder paste printing as an assembly method continues to be a challenge for SiP assembly. Small stencil aperture designs have necessitated fine-powder solder pastes, with type 5, type 6, and type 7 being common, as well as there being a need for type 8 for future, more advanced designs. Additionally, as more components are being packed into a SiP, solder paste rheology becomes an important characteristic to consider. Printing transfer efficiency, stencil life, and resistance to slump, are other important paste attributes to consider when evaluating solder paste for SiP assembly. While material properties are important to consider, proper equipment setup for a certain paste formulation is also necessary to maximize throughput and minimize assembly defects. The effects of solder paste printer parameters on fine-feature printing performance will be discussed, such as print speed, separation speed, separation distance, and stencil material.

This paper will first focus on common failure modes in SiP assembly, and the technology advancements in both solder paste formulation and substrate design which help eliminate those failure modes. Then, this paper will discuss how proper stencil printer setup can reduce assembly defects, and will highlight designs-of-experiment outlining which stencil printer parameters are statistically significant in improving process yields of SiP assemblies with fine powder solder pastes, both with respect to pre-reflow inspection and post-reflow component placement. This paper will also discuss the importance of solder paste inspection with respect to assembly defects, and will highlight important parameters to optimize to accurately inspect solder paste deposits below 90 microns.

Key words

I. Introduction

System-in-Package (SiP) assembly is a significant technology driver in the electronics assembly space. Even before the onset of the COVID-19 pandemic, there has been continuing demand for innovative electronic devices across industries, including but not limited to 5G, Internet-of-Things (IoT), artificial intelligence, smart devices, and big

data. From a manufacturer's perspective, the drivers behind this demand are that these devices need to be high-performing and come to the market quickly. SiP devices (and by extension other heterogeneous integration assembled packages) have been established and innovated upon in high-volume manufacturing environments to meet the increase in demand post-COVID-19.

II. Failure Modes in SiP Assembly

As the trend toward miniaturization in SiP applications continues, from current 01005 and 008004 components to 0050025 for next generation packages, the printing performance of solder paste becomes critical. The conventional SMT solder paste printing process using Type 3 or Type 4 powder size has evolved into a more complicated printing process for SiP, using Types 5, 6, 7, or even Type 8 powder size, with much smaller stencil apertures, thinner stencils, and more stringent requirements for allowed paste deposit variability. Solder paste formulation with these powder types needs to be carefully considered; attributes which may be relevant for SMT solder paste may not carry over to solder paste used for SiP assembly. These considerations will be discussed later in this paper.

As packages get smaller and thinner, solder paste manufacturers need to consider attributes of SiP devices when designing an appropriate solder paste. The first of which is that solder interconnections are required to be smaller and tighter. This is driving current soldering materials technology to a different level in order to handle various challenges such as fine aperture printing and small dot dispensing, residue cleaning, and solder joint integrity, with insufficient print deposits being the common failure mode. These smaller print deposits are also being manufactured at a much tighter pitch to accommodate the number of chips fit into a SiP. An incorrectly formulated solder paste for a tight-pitch SiP will inevitably lead to solder bridging issues, one method which certain manufacturers are using to mitigate this issue is by printing multiple solder deposits on the same pad. Warpage is another factor causing yield loss due to SiP substrates becoming much smaller and thinner. In addition, pressure to reduce costs pushes the industry for more cost-effective methods of assembly.

III. Solder Paste Formulation Advancements

A. Solder Powder

Solder powder is classified by type according to IPC J-STD-005A, with the relevant powder types for SMT and SiP shown in Table 1 [1]. Solder powder Types 3-5 are commonly used for SMT applications; SiP applications typically require powder types 6 and 7, with applications in the future heading towards Type 8 powder sizes. This guideline results from the standard convention that consistent solder deposits should have roughly six solder particles across a given aperture; smaller apertures therefore require smaller solder particles.

Table 1. Powder Sizes

Powder Type	Powder Size (um)	Minimum Stencil Aperture (um)	Approximate Surface Area Ratio
3	25-45	270	1.0
4	20-38	230	1.2
5	15-25	150	1.9
6	5-15	90	3.7
7	2-11	66	5.6

B. Flux System

In fine-pitch applications, flux systems can be classified as no-clean, water soluble, and ultra-low residue. Ultra-low residue flux is particularly attractive for SiP applications, as cleaning between increasingly small solder deposits becomes naturally more difficult. ULR flux systems also provide the benefit of removing costs associated with cleaning and the chemical processes associated with that cleaning.

Flux homogeneity is also an important advancement which has been made for solder paste applications for SiP devices. Post-processing steps have been developed which remove air bubbles and any additional abnormalities from solder flux, which has the effect of improving the transfer efficiency of the associated solder paste. Any abnormalities in the flux vehicle naturally run the risk of causing defects when the stencil apertures are manufactured on the order of ~100 microns, so ensuring flux homogeneity is key.

C. Paste Rheology

The transition from SMT technology to SiP technology also causes changes with solder paste rheology with respect to particle size. As particle size decreases for fine-feature printing, the viscosity of solder paste increases due to its thixotropic nature. Solder paste rheology is the driving factor behind the ability of solder paste to fill an aperture, the ability for solder deposits to release from the aperture onto the substrate (transfer efficiency), and solder deposit shape. It has been shown that using finer solder powders can improve the transfer efficiency of a solder paste, but many factors need to be considered, such as the shape of the pad and the presence or lack thereof of solder masks [2]. The paste rheology can be further optimized by varying the metal load and flux rheology for better printability in SiP applications. The fine powder size required for good printability, in contrast, places a burden on the rheology improvement effort

D. Slump Behavior

Slump refers to an expansion of a solder paste deposit in the x-y direction caused by gravitational forces after a print. Slump is directly affected by the paste's metal load; a low metal load will cause the paste deposit to slump while a high metal load will give the paste a very high yield stress and prevent it from properly filling and releasing from the

aperture. Metal load optimization is therefore a very important parameter when conducting a printing experiment.

E. Stencil Life

It is important for a solder paste to have a long stencil life is important to ensure print-to-print consistency. High-volume manufacturing environments typically require a solder paste to have a 4-8-hour stencil life. It is important to note that decreasing the size of the solder powder for use in SiP applications can reduce the stencil life of a paste. Smaller powders naturally have more surface area, leading to increased oxidation reactions in the solder paste. Oxidation thickens the solder paste, leading to potentially clogged apertures and the solder paste sticking to the squeegee blades. Flux formulations are made with this oxidation in mind to slow the rate of oxidation. Figure 1 shows an example of a typical response-to-pause experiment which shows the effect of pausing printing for one hour after printing six boards. As shown in the figure, there is no statistically significant change in print performance after an hour pause after the sixth board, showing excellent stencil life at 75um apertures.

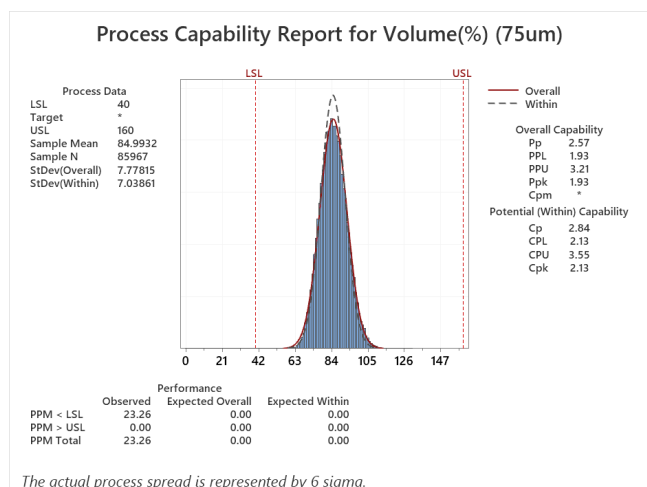
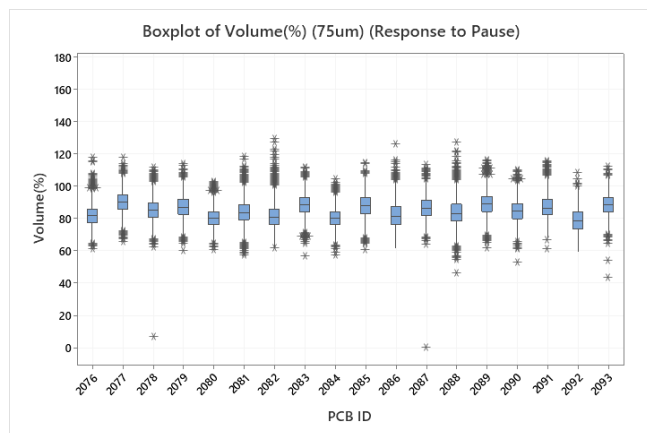


Figure 1. Response To Pause Test of T7 Solder Paste

F. Paste Tackiness

During the solder paste printing process, the paste is required to flow easily during printing, but not to flow at all after. As powder size decreases, the tackiness of a solder paste typically increases. As such, it is important to formulate a solder paste such that the paste is sufficiently non-tacky to be released from the stencil aperture, but tacky enough to hold onto the substrate and the components to be placed subsequently onto the paste after printing [3]. In order to meet the requirement of a SiP application, the rheology of solder paste has to be tailored that application.

Figure 2 shows the comparison of paste tackiness when held at 50% relative humidity for 24 hours. This graph shows the average tackiness of five solder deposits measured at 2-hour intervals. At the beginning of data collection, the tackiness of T7 powder is higher than that of T6, however the values shift over time due to the effect of exposure to the humidity. It is important to note that the values remain consistent through 24 hours, and do not drastically increase or decrease during this time period, ensuring consistent performance through 24 hours.

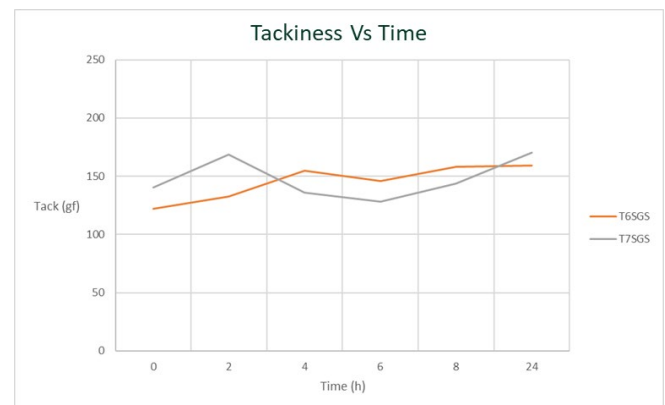


Figure 2. Tackiness over Time of SiP Solder Paste

IV. SiP-Relevant Printing Experiments and Parameters

A. Substrate Advancements

For the forthcoming printing experiments to be discussed, a test vehicle was designed to accurately mimic a typical SiP assembly substrate, with the attributes described in Section II. This test vehicle has ENIG pads spaced in five sections with pad pitches between 50 and 150 um.

As previously stated, as substrates become smaller and thinner for SiP assembly, there is a propensity for warpage, which can affect both final build integrity and initial SPI yields. Figures 3 and 4 show instances of a non-warped and warped test vehicle and the effect it has on SPC results. It's clearly apparent that slight warpage has a strong effect on build integrity.

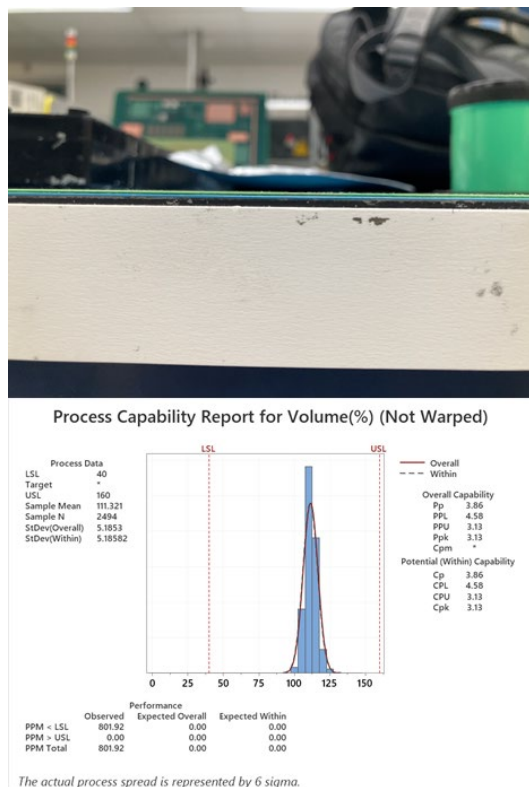


Figure 3. PCR of a Flat Test Vehicle

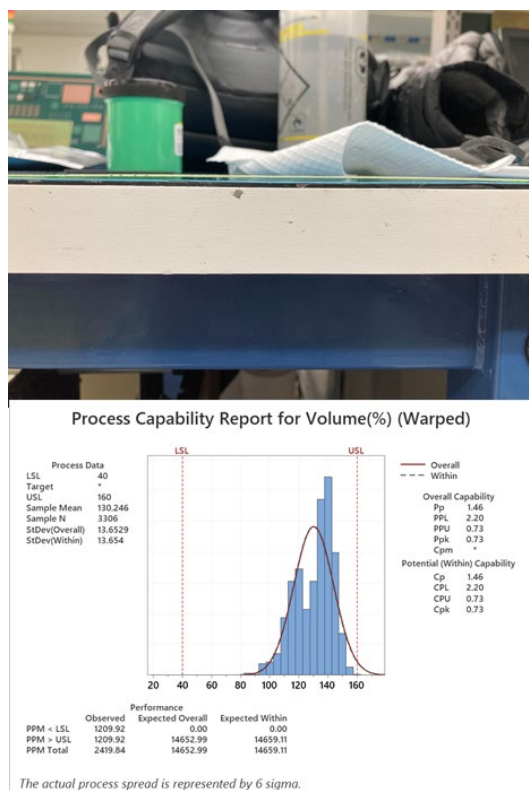


Figure 4. PCR of a Warped Test Vehicle

In addition to mimicking thin substrates, the test vehicle developed also mimics the printing patterns seen at large OSATS. To mitigate the risk of bridging in our test vehicle, we modified the stencil openings to print circular paste deposits onto pads etched to 01005 size (200x220um) and 008004 size (100x150um). An example of a print on this test vehicle is shown in Figure 5. This setup gives flexibility with testing, as it allows for many different sizes of deposits to be analyzed as well as the flexibility to test for bridging in extreme cases.

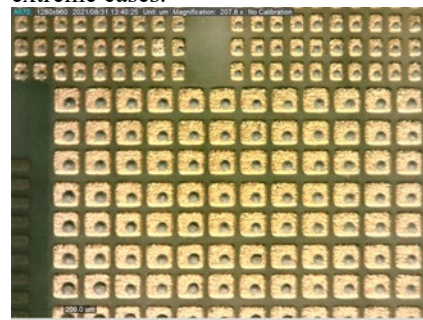


Figure 5. 90um Deposit on 01005 Pads

B. Stencil Technology

In order to properly test solder paste formulated for SiP applications, a study was conducted in order to find the optimal stencil material at fine feature print deposits. Typically, electroform stencils and laser-cut stencils with nano-coating are two common processes used for fine-feature stencil manufacturing. The laser cut stencil process involves (1) processing Gerber data, and (2) cutting image [3, 4, 5]. Electroforming is an additive process [4, 5]. A mandrel is used as a base for the photoresist application and for resolution of the image. The mandrel is then placed in a bath where Ni is plated. The opening will be formed around the photoresist until the desired thickness of the stencil has been achieved.

To determine the optimal stencil material for SiP applications, a solder paste printing study was conducted. Three stencils of different thicknesses were chosen: a 30um laser-cut stencil, a 35um electroform stencil, and a 40um laser-cut stencil. An experiment was conducted where 20 test boards were printed with each stencil, and the resulting process control variables were analyzed. Since the stencils were of different thicknesses, apertures with similar area ratios were compared to each other to measure the transfer efficiency of each stencil. The results of this study are shown in Figure 6.

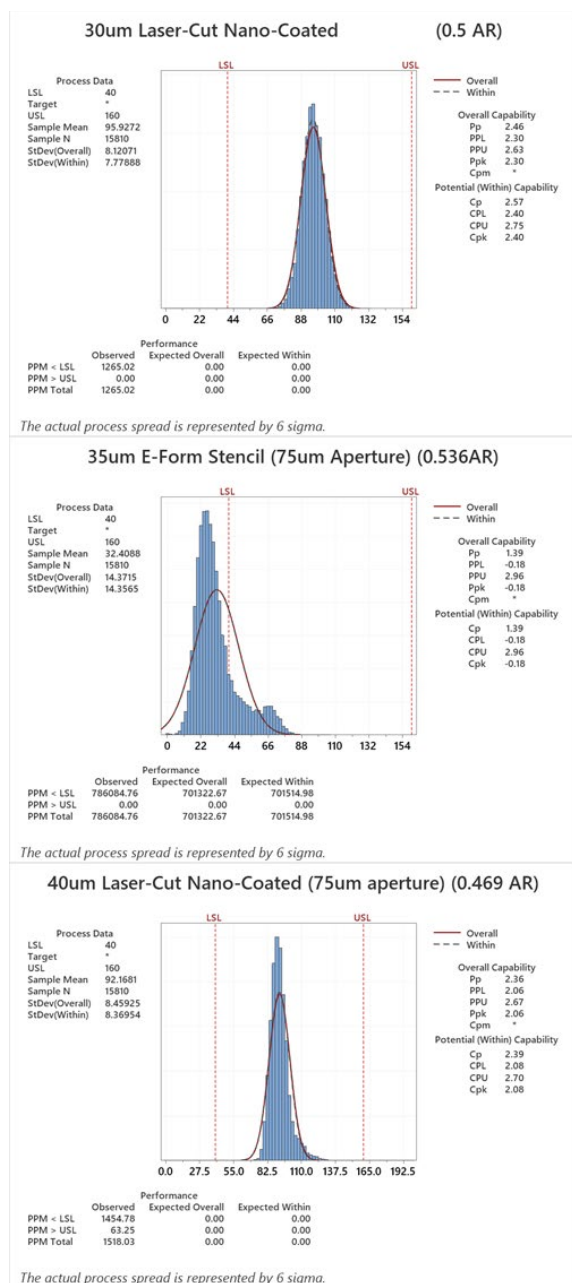


Figure 6. Stencil Comparison at Similar Area Ratios

The results of this study show that at similar area ratios, the two laser-cut stencils outperform the electroform stencil, even though the area ratio for the 40um stencil is much lower than that of the electroform stencil. Naturally, the thinnest stencil has the best performance, although the SPC numbers of the 40um laser-cut stencil are only slightly worse. The results of this study show that 60um aperture printing is feasible with laser-cut stencils, and that the SiP printing process is made more feasible with laser-cut stencils.

C. Board Support and Clamps

When setting up a stencil printer, it is imperative to have implement the proper board supports and board clamps to

ensure consistent printing performance without board shift. This is especially important with SiP printing due to the small and thin substrates in use. Vacuum tooling, where the PCB board, once loaded in the printer, is proven to eliminate bridging [size pei paper] as compared to magnetic pins due to the fact that the board is much more rigid with the former support, and there is minimal gap between the stencil and PCB. Foil-less clamps are used in tandem with vacuum tooling and achieve good printing performance as well.

An experiment was conducted to test the effect that changing a vacuum tooling plate has on the print performance of a T7 solder paste. In this experiment, six boards were printed for each stencil and vacuum tooling combination; in this case, an electroform and laser-cut stencil were used, in addition to two vacuum tooling blocks, the only difference between the two being a 1mm height difference. These blocks will be referred to as “New” and “Old,” with “New” being 1mm shorter than “Old.” SPC numbers were measured, and the results are shown in Figure 7.

Stencil	Tooling	Pad Size	Volume %	Cpk	Ppk
EForm	New	90	63.74	0.449	0.434
EForm	New	105	62.23	0.728	0.684
Laser	New	90	91.11	1	0.916
Laser	New	105	73.51	0.991	0.821
EForm	Old	90	56.78	0.444	0.397
EForm	Old	105	70.3	0.824	0.739
Laser	Old	90	95.3	1.461	1.433
Laser	Old	105	81.73	1.288	1.243

Figure 7. Results of Tooling Experiment

The results show a significant improvement on both 90 and 105um apertures with the taller tooling block, with respect to more paste being deposited and the process being much more consistent. The stencil variable also reflects what was outlined in the previous section, with Laser-cut stencils outperforming electroform stencils. What’s interesting about these results is that the 1mm height difference between the tooling block has a significant effect on the print parameter, perhaps due to the tooling block not reaching enough to the underside of the stencil. When dealing with SiP assembly processes, even minor height differences in tooling blocks can have a large effect on results.

D. Significant Printing Parameters

Another experiment was conducted with a properly formulated paste to determine which print parameters have the strongest effect on a fine-feature solder paste printing process. For this experiment, three print parameters were selected: print speed (25, 25, and 45 mm/s), separation speed (5, 12, and 20 mm/s), and separation distance (10, and 0 [which represents an instant drop] mm/s). Two boards were printed per iteration and the results were analyzed using Minitab ANOVA software. Main Effects Plots and Interaction Plots will be shown for the effects of these parameters on Deposit Volume (%) and Cpk.

Figure 8 shows the Main Effects Plot of these three parameters on deposit volume. From the pareto chart, print speed is the only significant variable which effects the volume of a print deposit, which is expected because as you increase the print speed, there is less time for the paste to fill an aperture. Interaction plots are not shown for this output variable because no variable interactions are significant.

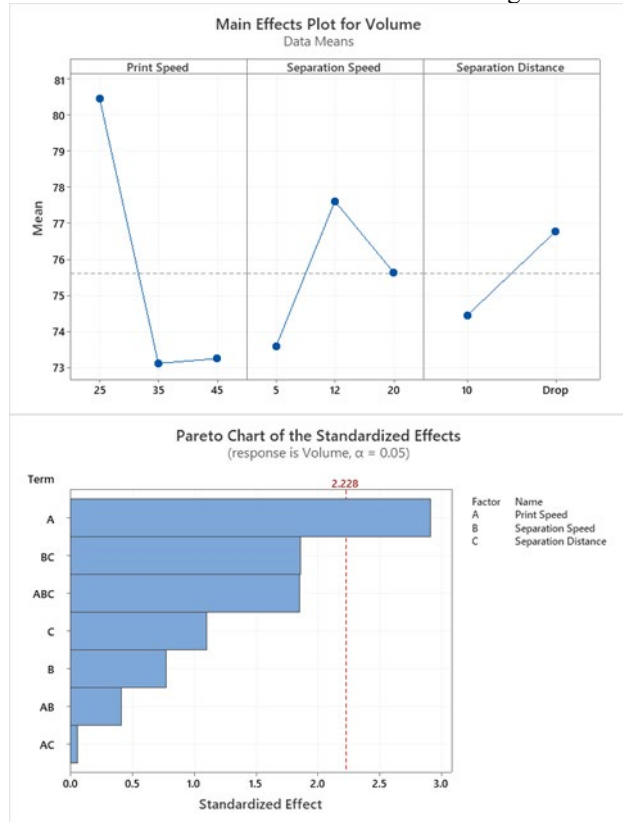


Figure 8. Significant Print Variables Affecting Solder Volume

Figure 9 shows the Main Effects and Interaction Plots for the effect of changing these parameters on the overall print consistency (Cpk). From this chart, the most significant effects on the entire print process are the interaction between separation speed and separation distance, and the print speed. Like for volume %, having a slower separation speed increases the Cpk of the process. Looking at separation speed and separation distance, Cpk improves when you increase the separation speed, as well as the separation distance from 10 mm/s to a drop. However, moving dropping the board with a large separation distance seems to worsen the process. It may be that moving 10mm at 20mm/s is actually faster than just dropping the board; this setting may be printer-dependent and should be investigated when setting up a print process. From these results, for a fine-feature, fine powder process, the best parameters seem to be having a slower print speed with a fast separation speed. These parameters, while being statistically the most relevant for SiP printing, will need to be fine-tuned for a particular paste.

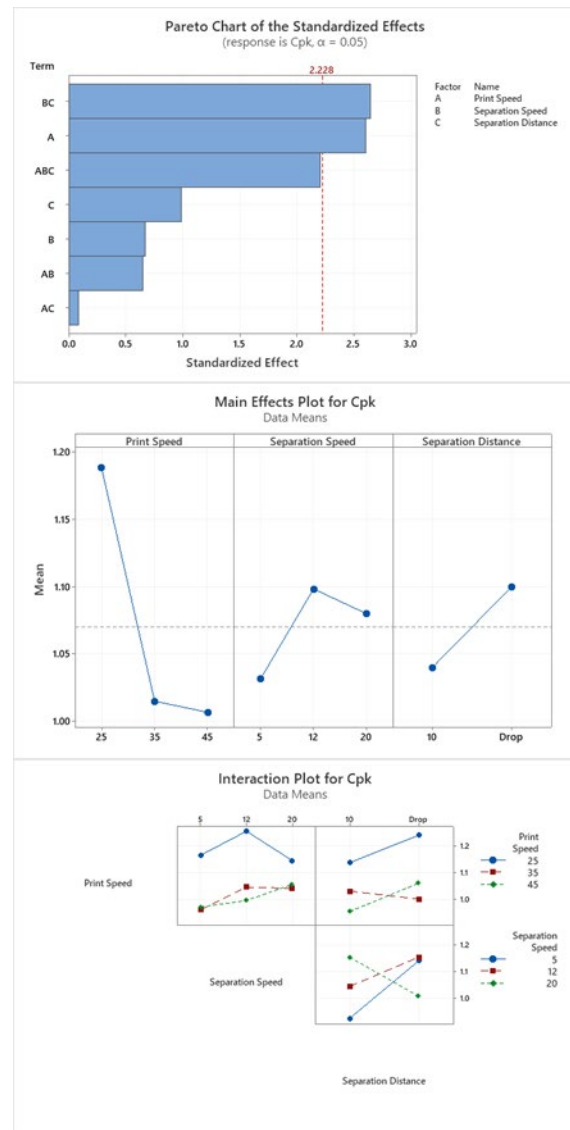


Figure 9. Significant Print Variables Affecting Cpk

V. Solder Paste Inspection

A. Solder Paste Inspection Overview

It is important to note that with the thinner stencils and smaller solder prints in use for SiP assembly, solder paste inspection (SPI) becomes much more difficult to optimize, as machine cameras need to be able to accurately measure small solder deposits while dealing with glare. The two main parameters that are often adjusted to obtain the most accurate printing data on an SPI are threshold height and extended region of interest (ROI). Threshold height is the height which the SPI sets as the upper limit for solder deposits; a deposit exceeding this height runs the risk of being measured as having excessive volume percentage. The extended ROIs are the areas around each pad that the SPI actually inspects when

measuring a solder deposit. These are usually set to be squares, but can be modified based on the pad dimensions. An SPI comes with recommended settings to use for both threshold height and extended ROIs for a given stencil height. However, with fine feature printing on stencils that are manufactured below 75 μm , these recommended settings can cause issues with reading prints that are much more excessive than in actuality. This issue was a major reason for designing the test vehicle described earlier in this paper; in order to remove any effect that solder masks, which at fine features are at a similar relative height to the solder deposit have on SPI readings. From the previous figures in this paper, the optimization that occurred with this new test vehicle fixed most excessive print readings which were prevalent on previous test vehicles.

B. Experiment Setup

With a properly designed test vehicle for SiP applications in hand which eliminated the effect of thick solder masks around the solder deposit, and experiment was conducted to determine the optimal ROIs for an individual pad. As previously described, this test vehicle and stencil design was thought up as to deposit solder paste in the center of a metallized pad, either an 01005 or an 008004. A print experiment was conducted in order to study the effect of generating the ROI completely within the pad versus around the outside of the pad, as shown in Figure 10. Six boards with a paste formulated for SiP assembly (T7) were printed and statistical process control variables were collected.

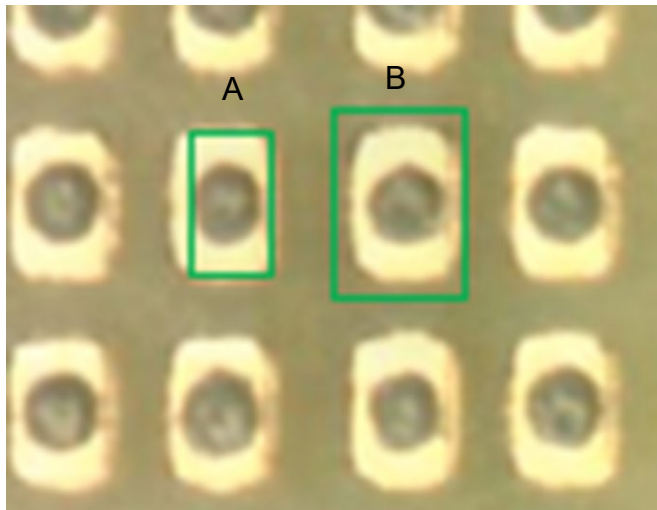


Figure 10. ROI Experiment Setup

C. Results

The results of this experiment are shown in Figure 11. SPC values were calculated using Minitab software with a reference specification of 40%-160%, common in device assembly sites. As shown in the table, Program A, where the region of interest is completely within the soldering pad in

question, offers much more consistent results than Program B. This suggests that even the small height difference between the raised metallized pad and the PCB body affects SPI readings at these small aperture sizes. Another observation is that as you reach smaller aperture sizes in Program B, the Area% variable, which measures the pixels in the SPI program which are picked up by the SPI camera, increases to 150%. This is not representative of reality, as 150% contains quite a lot of noise for the data to be meaningful.

Program	Aperture Size (μm)	Vol%	SD	Cpk/Ppk	Area %	Area Ratio
A	60	88.29%	15.2	1.06	125.80%	0.5
	75	88.35%	11.4	1.41	106.24%	0.625
	90	84.93%	9.03	1.66	102.43%	0.75
	105	96.21%	10.25	1.83	113.70%	0.875
B	60	115.15%	18.15	0.82	156%	0.5
	75	104.91%	13.3	1.38	126.37%	0.625
	90	93.81%	11.76	1.52	111.31%	0.75
	105	109.47%	12.27	1.37	126.71%	0.875

Figure 11. Results of ROI Variation Experiment

We see as we approach 60 μm apertures that the process control and Area% variables become worse. At such fine features, much finer observations and optimization needs to be done in order to accurately measure those small pads.

VI. Conclusion

In summary, the failure modes for SiP assembly are prevalent, but can be avoided with properly formulated solder paste. Improvements to solder flux vehicles, powder technology, and substrate design can greatly improve the transfer efficiency on stencil apertures below 80 microns. Additionally, seemingly minor tooling setups and SPI with respect to printing can greatly affect solder deposits, leading to lead failures which could easily be avoided.

Acknowledgment

The authors of this paper would like to acknowledge their colleagues at Indium Corporation who helped prepare the samples for these tests and assisted with experiment execution, including Dr. Yan Liu, Sze Pei Lim, and Dakota Monk.

References

- [1] S. P. Lim, K. Thum, A. Mackie, "Fine Feature Solder Paste Printing for SIP Applications," *CSTIC*, March 2017.
- [2] E. Briggs, "Meeting Future Stencil Printing Challenges with Ultrafine Powder Solder Pastes," *International Conference on Soldering and Reliability*, May 2014.
- [3] N.C. Lee, "Reflow Soldering Processes and Troubleshooting: SMT, BGA, CSP and Flip Chip Technologies", Boston, MA, 2002
- [4] M. D. Herbst, "Metal Mask Stencils for Ultra Fine Pitch Printing", in *Proc. of Surface Mount International*, San Jose, CA, pp. 101–109 (29 August–2 September 1993).
- [5] Metal Etching Technology: Technical information (November 1993).