Optimization of Solder Paste Printing for Ultra-High-Density-Interconnect (UHDI) Applications

Mike Butler & Ed Nauss
ITW/EAE
Hopkinton, MA, USA

Tony Lentz
FCT Solder
Windsor, CO, USA

Greg Smith
BlueRing Stencils
Windsor, CO, USA

Abstract

The trend of miniaturization of electronics requires the creation of Ultra-high-density solder joints. Semiconductor manufacturing, flip chip, package-on-package (PoP), system in package (SiP), and miniature components like 0201M (008004 Imperial) may require printing through stencil apertures of 100-150 µm (4-6 mils) or less in size. Creating these miniature solder joints requires an optimized solder paste printing process and the use of IPC Type 6 (5-15 µm) [1] or smaller solder powder sizes.

Solder paste printing performance was studied using various stencil designs, step-stencil thicknesses, print parameters, and solder paste technologies. The test printed circuit board (PCB) included many challenging UHDI components. Printing performance was evaluated using solder paste inspection (SPI). The data was compared, and recommendations were made for successfully printing these ultra-high density soldering applications.

Introduction

The trend of miniaturization in electronics continues to challenge the solder paste print process. As solder joints become smaller, it becomes necessary to use smaller solder powder sizes which allow the solder paste to print and reflow successfully. Decreasing solder powder size carries the challenge of increased surface area of the solder powder. This higher surface area requires the solder paste flux to do more chemical "work" to remove metal oxides and prevent further oxidation during air exposure and reflow of the solder paste. The surface area of the solder powder increases dramatically from IPC Type 3 through Type 6 solder powder sizes [1] (Table 1).

Table 1. Surface area by size for 1 Kg of solder powder.

Solder Powder Size (IPC Type)	Size Range of > 80% (µm)	Middle Surface Area of 1 Kg (m^2)	Amount of Surface Area Over T3
Type 3	25 - 45	22.9	**-**
Type 4	20 - 38	27.7	**1.2x**
Type 5	15 - 25	40.2	**1.7x**
Type 6	5 - 15	80.3	**3.5x**

For the same mass of solder powder, IPC Type 4 solder powder has 1.2 times the surface area of Type 3. IPC Type 5 solder powder has 1.7 times the surface area of Type 3. IPC Type 6 solder powder has 3.5 times the surface area of Type 3. Most modern solder paste fluxes are formulated to work well with Type 3 and 4 solder powders. Some fluxes can also accommodate Type 5 solder powder. The considerably higher surface area of Type 6 solder powder requires changes to the solder paste formulation. An increase in activity level and oxidation protection, as well as rheological modifications are required for solder pastes with Type 6 solder powder.

Cost of the solder powder is an important consideration when deciding to switch to a smaller solder powder size. The relative solder powder costs associated with Types 3 to 6 solder powders are shown below (Table 2).

Table 2. Solder powder size and relative cost.

Type	Relative Powder Cost
3	1
4	1
5	1.1
6	4

Type 3 and 4 solder powders have similar cost while Type 5 costs roughly 10% more. Type 6 solder powders include a significantly higher cost that is 4 times that of Type 3 and 4 solder powders. This cost increase translates to an increased cost of the solder paste.

The main reason for switching to a smaller solder powder size is to improve printability of the solder paste through small stencil apertures. One way to select a solder powder size for the aperture size is called the "5-ball rule" [2]. This rule suggests that a minimum of 5 particles of solder powder ("balls") must fit across the narrowest dimension of a stencil aperture (Figure 1).

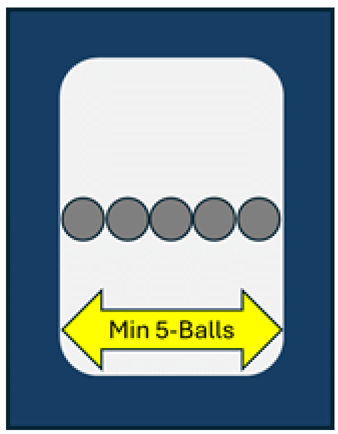


Figure 1. Stencil aperture size and the "5-Ball" bule.

The "5-ball" rule minimum and recommended minimum aperture sizes for each solder powder size is shown below (Table 3).

Table 3. Solder powder size and the smallest aperture size recommended.

Type	Size (µm)	Size (mils)	Smallest Aperture 5-Ball Rule (mils)	Smallest Aperture Recommended (mils)
2	45 - 75	1.8 - 3.0	15.0	16 - 17
3	25 - 45	1.0 - 1.8	9.0	10 - 11
4	20 - 38	0.8 - 1.5	7.5	9 - 10
5	15 - 25	0.6 - 1.0	5.0	6 - 7
6	5 - 15	0.2 - 0.6	3.0	4 - 5
7	2 - 11	0.1 - 0.4	2.0	3 - 4

Type 6 solder powder can theoretically allow a solder paste to print through a 3.0 mil wide stencil aperture, although 4-5 mil aperture sizes are recommended for adequate printability. The "5-ball" rule does not take into consideration the flux portion of the solder paste. Different solder pastes perform differently in printing applications due to variations in flux chemistry and rheology. The metal content of the solder paste also affects rheology and printability of the solder paste. When optimizing a solder paste to print through smaller stencil apertures, the solder powder size, metal concentration, and solder paste rheology need to be balanced. The flux chemistry also needs to be optimized with respect to oxide removal and protection of the solder powder.

Standard Printed Circuit Boards (PCB) have been limited to pad and space sizes of 75 µm. This has restricted the density of the components and size of the PCB. A new PCB assembly technique called High Density Interconnect (HDI) technology has allowed the pad spacing to be reduced from 75 µm to 25 µm. This can result in a 9X increase in density resulting in a reduction of both layer count and overall size. The inner layer circuitry can be 25 µm or less but there are limitations on outer layer pad design. If the pads are masked defined, then the spacing can be set to 25 µm. However, if they are defined by the copper, then it is recommended that the minimum spacing be 50 µm or greater. Mask defined pads have an issue with positional accuracy/repeatability. The process of mask defining a pad is prone to show drift/stretch which changes from PCB to PCB. This solder mask misalignment becomes apparent during the solder paste print process when the PCB is aligned to a steady state stencil image. If the inaccuracy is uniform across the image, then the print machine's alignment system can adjust. However, if the inaccuracy is with individual or groups of pads then the printer alignment will fail. The tolerances and repeatability of this process must be watched closely. The next level, described as Ultra High-Density Interconnects (UHDI), reduces spacing to 12.5 µm. This provides a 36X increase in density when compared to conventional 75 µm technology. The focus initially with UHDI technology is in semiconductor manufacturing where a Ball Grid Array (BGA) can be reduced by two-thirds in size.

While the solder paste influences the print process, the stencil also plays a role. The stencil metal alloy and laser cutting process affects aperture side wall roughness. The addition of ceramic nano-coatings also improve solder paste release and are recommended to be used in these applications. Stencil aperture area ratio and aspect ratio need to be held above minimum printable requirements, which can include the use of step-downs in foil thickness for ultra-small apertures. To print UHDI solder paste deposits consistently, stencil optimization should address both design and construction based on these considerations.

Printing equipment and parameters play a huge role in the print process for UHDI applications. The calibration of the printer is critical to ensure that the gap between the PCB surface and the stencil surface (co-planarity) is minimized, and consistent across the PCB. This ensures good, repeatable gasketing of the stencil apertures to the PCB pads. The flatness of the PCB is also critical and may require the use of vacuum fixture type support. Print pressure & speed, separation distance & speed, under-stencil cleaning chemistry and cycle, and other parameters need to be optimized to ensure high-quality printing results. Due to the reduced spacing, alignment accuracy and repeatability are key for UHDI applications. The machines base specifications should meet the demands of this technology and calibration must be verified.

It is the intent of this work to detail testing for the optimization of printing Type 6 solder pastes in UHDI applications. Various print parameters and solder paste technologies were evaluated with a challenging test PCB and stencil. Recommendations will be made for successful printing in these miniaturized soldering applications.

Prior Work

E. Nauss [3] studied sealed versus exposed atmosphere printing for Type 6 solder pastes. Two solder pastes were tested with Type 6 solder powder in a challenging print application. One solder paste performed better than the other. The sealed chamber increased the stencil life and the print consistency over time.

C. Ashmore [4] studied solder paste printing for the Metric M0201 assembly process. Type 5 and 6 solder pastes were used. The pad sizes were 125 x 115 µm and 100 x 115 µm with 0.50 and 0.45 aperture area ratios respectively. The Type 6 solder paste provided a fuller print (higher volume) but created more defects than the Type 5 solder paste. The defects were mainly related to high volume and included bridging, and shift/skew of the components.

S. Pei-Lim, et. al. [5] studied challenges in fine feature solder paste printing for SiP applications. Type 6 solder pastes were used with different stencil designs and printing parameters for 01005 Imperial (M0402) and 008004 Imperial (M0201) components. Two water soluble and two no clean solder pastes were used with Type 6 solder powder. Printing down to an area ratio of 0.60 was possible using the correct solder paste rheology. Metal content of the solder pastes also had a significant effect on printability.

S. Joshi [6] reported on Pb-free solder paste development for ultra fine-pitch printing and reflow of M03015 and M0201 metric components. SAC305 with a size range of 5 – 20 µm was used, which is slightly larger than IPC Type 6 (5 – 15 µm). A newly optimized solder paste gave less bridging and better response to pause in printing as compared to older technology. Thixotropy of the new solder paste was optimized for smaller solder powder sizes. Nitrogen was required to achieve acceptable reflow with the solder pastes evaluated.

S. Harter, et. al. [7] studied printing processes for M03015 metric component sizes. Stencil designs, materials, and nano-coatings were varied. SAC305 Type 5.5 (5 – 20 µm) and Type 6 (5 – 15 µm) solder pastes were used. The stencil aperture sizes varied from 120 x 139 µm to 190 x 219 µm. The main effects of the tests showed the following. Type 6 solder paste gave lower printed volume than Type 5.5. 80 µm stencil thickness gave lower printed volume than 60 µm. Bridging on smaller pad spacings was reduced with: thinner stencil, nano-coating, and Type 6 solder paste.

E. Griffith [8] reported on the evolution and application of fine feature solder paste printing for heterogenous application. System in package (SiP) applications typically use 008004 Imperial components, Type 6 and 7 solder pastes, and have small gaps (50 µm) between the pads. Print testing was conducted with three solder paste fluxes mixed with 3 quality levels of Type 6 solder powder, and pad sizes of 6 and 7 mils. Different combinations of flux and solder powder style gave different printing and slump results. This work demonstrated the need to optimize flux, solder powder size, and solder powder quality for SiP applications.

A. Murling, et. al. [9] studied optimum solder powder sizes for 0201M (008004 Imperial) component solder paste printing. The process window for producing consistent uniform solder paste deposits is narrow and the process needs to be well-controlled to maintain adequate volumes. A 60 micron (2.4 mil) thick laser cut nano-coated stencil was used along with Type 5 and Type 6 solder pastes. The Type 5 solder paste was able to be printed through 127 x 152-µ (5 x 6 mil) apertures, however Type 6 solder paste provided lower variation and was closer to the 100% transfer efficiency target.

T. Lentz [10] studied the print and reflow characteristics of various solder pastes with Type 6 SAC305 solder powder. Two of the new solder pastes were able to print with acceptable transfer efficiency through 175 µ (6.9 mil) rounded square apertures with 0.60 area ratio. This was done with standard printer settings using a non-optimized printer. Stencil life, response to pause, and tack life over time were excellent for the new Type 6 solder pastes, however, heat aging and some reflow characteristics were less than ideal. Optimization of these solder pastes for use with Type 6 solder powder is ongoing.

E. Nauss and M. Butler [11] studied printing for 0201M (008004 Imperial) components. The PCB design, flatness and ability to gasket the stencil is critical. Squeegee blades are recommended to be set at a 55-degree angle, while 60 degrees is standard. Blade condition & damage can make it difficult for a "clean sweep" of solder paste from the surface of the stencil. Enclosed flow heads may lengthen working life of Type 6 solder pastes. The recommended stencil design is fine-grain laser cut steel with a ceramic nano-coating. High tension stencils (28-40 N/cm^2) are recommended which may require stainless steel mesh as the mounting material. Type 6 solder pastes are required for 0201M components, and the costs are as much as 3X higher than Type 3 or 4 solder pastes. Issues with Type 6 solder pastes include stencil life, solder balling and graping which need to be addressed. Nitrogen may be required for reflow. Underside board support is key for printing, and pins, blocks or grid-tooling will not be adequate. Dedicated aluminum support plates are recommended. The support plates need to be made with high precision and venturi vacuum to ensure PCB flatness, especially for thin substrates. Calibration of the printer table to the stencil in all 4 corners is recommended to be 25 µ (1 mil) or less. Use of a calibration jig reduces the time needed for calibration and improves accuracy. Underside stencil wiping must use the proper materials and cycles to ensure proper cleaning. This is critical to avoid re-deposition of solder paste and lint onto the PCBs. Testing was conducted on a suitable test PCB with Type 6 solder paste and the printer setup recommended above. SPI Cp and Cpk values for 0201M pads were >2.2, which is acceptable for a repeatable process.

A. Nobari and S. St-Laurent [12] investigated characteristics of Pb-free solder powders for solder paste applications. Type 5 and Type 6 SAC305 solder powders were aged and studied for reflow performance and surface oxide formation. Type 6 solder powder degraded in solder performance with aging much faster than Type 5. Elevated humidity accelerated the decline in solder balling performance for both powder sizes. Elevated temperature did not change solder ball performance. Two flux formulas were tested with Type 6 solder powder, and one was better at limiting solder balling after aging than the other. Oxygen content of the solder powder increases within 7 days with elevated humidity (80% RH) but not with lower humidity levels. The oxides that form on SAC305 solder powder is comprised mainly of SnO.

Experimental Methodology

The overall test process used for print testing is listed below.

A. Prepare the printer for testing by calibration of the table height (Z-axis) to the stencil plane.
    a. The gap from table to stencil was minimized to below 50µ (2 mils) across the area of the table.
B. Install the test stencil and solder paste A and run 4 knead strokes.
C. Print a PCB, measure the alignment of the stencil to the PCBA through SPI.
D. Use SPI data to optimize the alignment using printer offsets in X, Y and theta rotation.

E. Repeat steps C and D until the alignment is within 5-10 µ and minimized theta rotation.
F. Print 30 PCBAs with solder paste A and measure the printed solder paste using SPI.
   a. Underside cleaning was performed after each print using a Vac/Wet/Dry cycle and a commercially available cleaning agent.
   b. The PCBs were cleaned using a commercially available cleaning agent in an ultrasonic bath, and then re-used.
G. Remove solder paste A, clean the stencil and printer hardware.
H. Install the cleaned stencil with solder paste B run 4 knead strokes.
I. Optimize the print alignment using the same process as steps C, D, and E.
J. Print 30 PCBAs with solder paste B and measure the printed solder paste using SPI.
   a. Underside cleaning was performed after each print using a Vac/Wet/Dry cycle and a commercially available cleaning agent.
   b. The PCBs were cleaned using a commercially available cleaning agent in an ultrasonic bath, and then re-used.
K. Export the data from the SPI for each solder paste and analyze using statistical analysis software.

A test PCB with a variety of component sizes down to 0201M (008004 imperial) and 0.3 mm pitch BGA pad layouts was selected for testing [11]. This test board is pictured below (Figure 2).

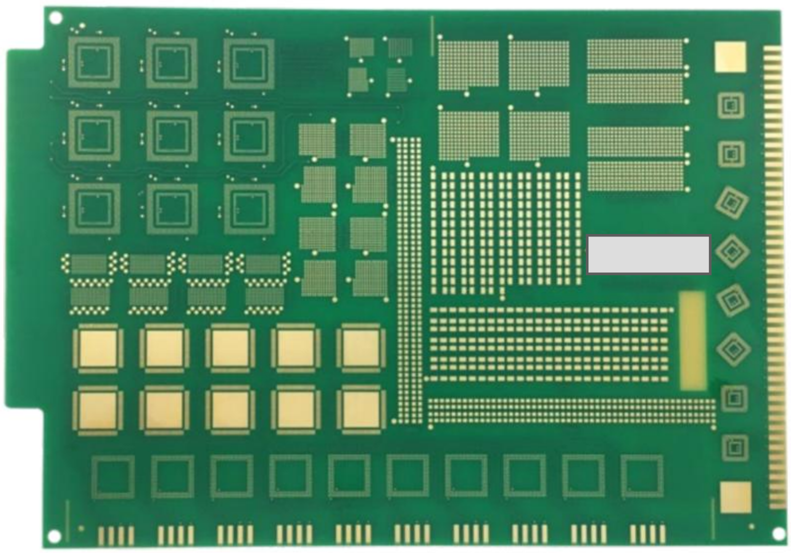


Figure 2. Test PCB to measure print performance.

The PCB dimensions are 203 mm (8.0") in X and 139.7 mm (5.5") in Y with a thickness of 1.57 mm (0.062"). This test PCB is double sided. The surface finish is electroless nickel immersion gold (ENIG). The component pads of interest on the test PCB are detailed below.

- BGA packages
  - 0.3 mm pitch with a total of 8 patterns per board.
  - 368 pads per device for a total of 2944 pads per board.
  - Pad sizes are 152 µm (0.006") round.
  - Pads are solder mask defined.
- Individual discrete components
  - 0201M (008004I) NE and SE locations.
    - Pad size is 120 µm x 145 µm (0.0047" x 0.0057") with 400 components with 800 pads per board.
    - 200 components positioned at 0 degrees and 200 positioned at 90 degrees.
    - The air gap between pads is 120 µm (0.0047") and the component pitch is 400 µm (0.0157").
    - Pads are copper defined.
  - 0201M (008004I) NW and SW locations.
    - Pad size is 130 µm x 160 µm (0.0051" x 0.0063") with 400 components with 800 pads per board.
    - 200 components positioned at 0 degrees and 200 positioned at 90 degrees.
    - The air gap between pads is 120 µm (0.0047") and the component pitch is 400 µm (0.0157").

- Pads are copper defined.

The stencil is detailed as follows: 29” x 29” space saver frame, high tension, laser cut fine gain stainless steel, 50 µm (2.0 mil) thick, with a ceramic nano-coating. An image of the stencil design file is shown below along with a close up of selected components (Figure 3).

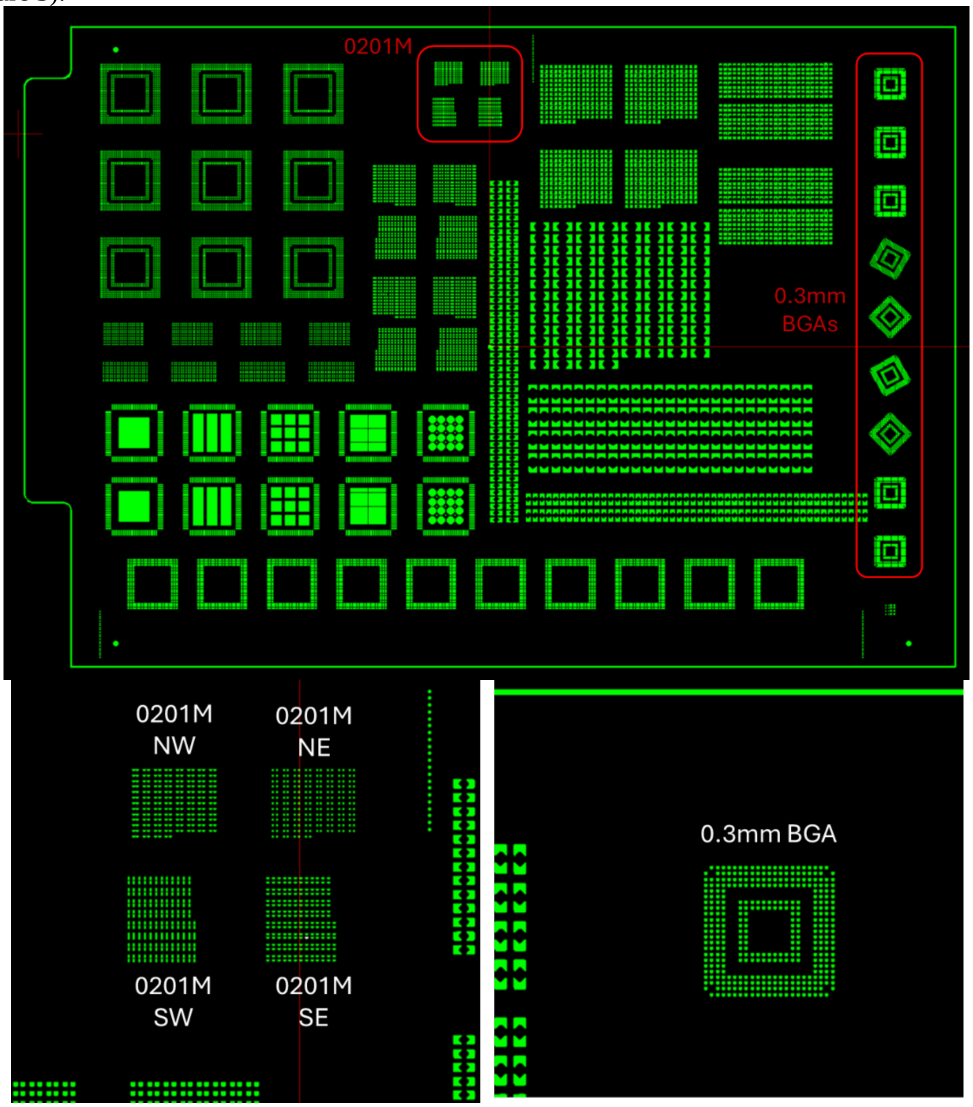


Figure 3. Test stencil and smallest apertures.

The 4 different 0201M locations are coded as NW (Northwest), NE (Northeast), SW (Southwest) and SE (Southeast). The smallest stencil aperture sizes and spacings are listed in Table 4 below. The rectangular apertures show the dimensions in width which is perpendicular to the print direction, and the tall direction is parallel to the print direction.

Table 4. Smallest stencil aperture sizes and spacing between apertures.

Component	Aperture Size	Area Ratio	Narrowest Space Between Apertures
0201M (008004I) NE	152 x 152 µm (6.0 x 6.0 mils)	0.75	147 µm (5.8 mils)
0201M (008004I) NW	180 µm wide x 150 µm tall (7.1 x 5.9 mils)	0.81	120 µm (4.7 mils)
0201M (008004I) SE	160 µm wide x 130 µm tall (6.3 x 5.1 mils)	0.70	170 µm (6.7 mils)
0201M (008004I) SW	150 µm wide x 180 µm tall (5.9 x 7.1 mils)	0.81	120 µm (4.7 mils)
0.3 mm BGAs	152 x 152 µm (6.0 x 6.0 mils)	0.75	147 µm (5.8 mils)

The two solder pastes used were a commercially available no-clean SAC305 Type 6 (solder paste A), and a new no-clean SAC305 Type 6 (solder paste B). Both solder pastes are rosin-based, low activity, halide and halogen free materials with a ROL0 classification. These solder pastes were manufactured by different sources and have different flux technologies.

The printing machine used is a MPM Momentum II.

The specifications for the printing machine and hardware are as follows:

- Wet print accuracy: 17 microns @ 6 sigma, CpK $\geqslant$ 2.0
- Alignment repeatability: $\pm$11 microns @ 6 sigma, CpK $\geqslant$2.0
- Calibrated and verified
- Options:
    - Edge PCB clamping
    - Paste height monitor
- Tooling: Blocks were placed to fully support the PCBAs, and substrates were held in position utilizing side-snugging.
- Blades: 220 mm (8") stainless steel blades with a 55-degree attack angle

The print parameters were optimized for each solder paste prior to testing. The parameters used are shown in Table 5 below. The difference in print parameters between the two solder pastes are highlighted in orange.

Table 5. Print parameters.

Parameter	Value (Solder Paste A)	Value (Solder Paste B)
Squeegee force	6.8 kg	7.7 kg
Print speed	38.1 mm/sec	38.1 mm/sec
Blade gap	-2.0 mm	-2.0 mm
Post print lift height	12.5 mm	5.1 mm
Post print lift speed	80.0 mm/sec	80.0 mm/sec
Separation distance	2.54 mm	2.54 mm
Separation speed	1.27 mm/sec	1.27 mm/sec

The SPI system used is a Parmi SigmaX.

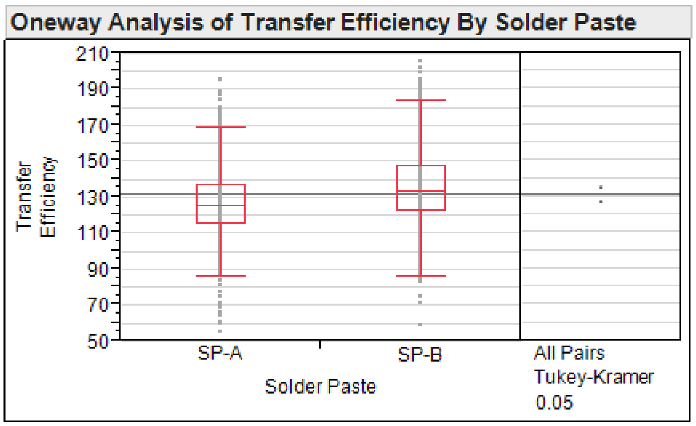
The SPI specifications are as follows.

- X – Y Resolution (µm): 7x7
- Height, Area, and Volume Repeatability: 3 Sigma < 1 µm, on certified target
- Height Accuracy: 2 µm, on a certified target

Results and Discussion

0201M Print Data

The printability of the solder pastes for the 0201M (008004I) components were of particular interest. The overall transfer efficiency (TE%) for each solder paste with all locations averaged is shown below (Figure 4).



Excluded Rows 2065

Means Comparisons

Comparisons for all pairs using Tukey-Kramer HSD

Connecting Letters Report

Level			Mean
SP-B	A		135.5
SP-A		B	127.5

Levels not connected by same letter are significantly different.

Figure 4. Transfer efficiency (%) for each solder paste in all 0201M locations.

Tukey Kramer HSD (honest significant difference) means comparison with a 95% confidence level shows that solder paste B gave an overall higher TE% than solder paste A. Each solder paste printed differently within the 4 different 0201M locations. These locations are coded as NW (Northwest), NE (Northeast), SW (Southwest) and SE (Southeast). The TE% for each solder paste is broken out by location below (Figure 5).

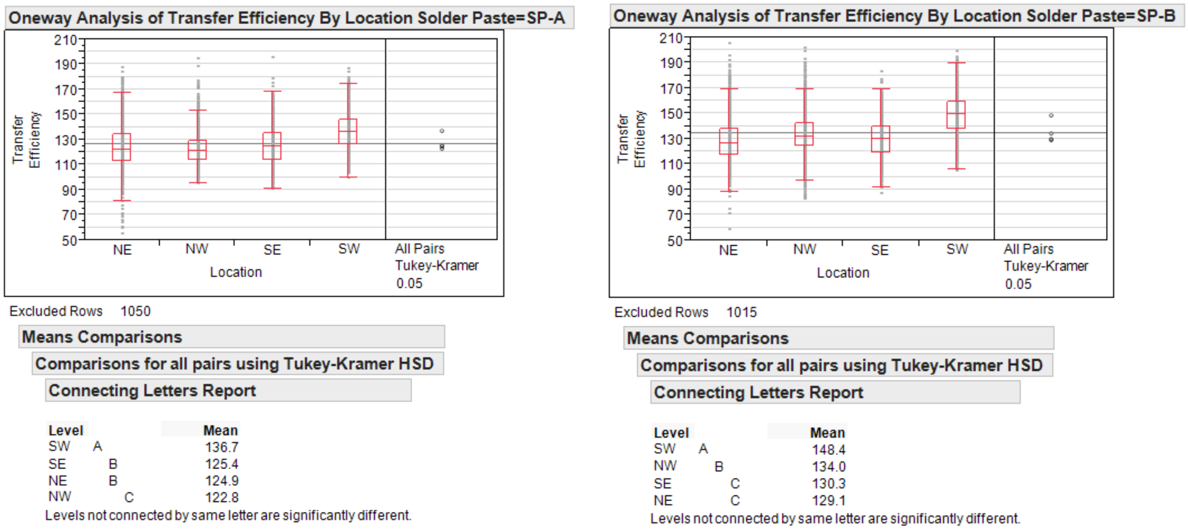


Figure 5. Transfer efficiency (%) for each solder paste separated by 0201M location. Solder Paste A (Left), Solder Paste B (Right).

Tukey Kramer HSD means comparison shows that the solder pastes printed a little differently in the different locations. Solder paste A gave the highest TE% in the SW location, followed by the SE and NE locations which printed similarly, and the NW location gave the lowest TE%. Solder paste B also gave the highest TE% in the SW location, followed by the NW location, and the SE and NE locations gave the lowest TE%.

The SW and NW pads are the same size at 130 µm x 160 µm (0.0051” x 0.0063”) but different orientations. The SW pads long edge is parallel to the print direction, and the NW pads long edge is perpendicular to the print direction. The stencil aperture designs were identical for these pads (180 µm x 150 µm (7.1 x 5.9 mils)) but also oriented in different directions mimicking the orientation of the pads. Each solder paste shows a TE% about14 higher for the SW orientation over the NW orientation. It is likely that during the print stroke, the solder paste fills the aperture more completely when the long edge is parallel to the print direction. This has been observed with other components like quad-flat packs, in previous print testing.

The SE and NE pads have the same size at 120 µm x 145 µm (0.0047” x 0.0057”) but different orientations. The SE pads long edge is parallel to the print direction and the NE pads long edge is perpendicular to the print direction. The stencil designs are different for these pads. The SE location has stencil aperture sizes of 160 µm x 130 µm (6.3 x 5.1 mils) with the long edge oriented parallel to the print direction. The NE location has stencil aperture sizes of 152 x 152 µm (6.0 x 6.0 mils). These SE and NE pads did not show the same difference in TE% as the SW and NW pads. The TE% is nearly identical for both the SE and NE pads for each solder paste.

The distribution data for TE% in the 0201M locations is broken out by solder paste below (Figure 6).

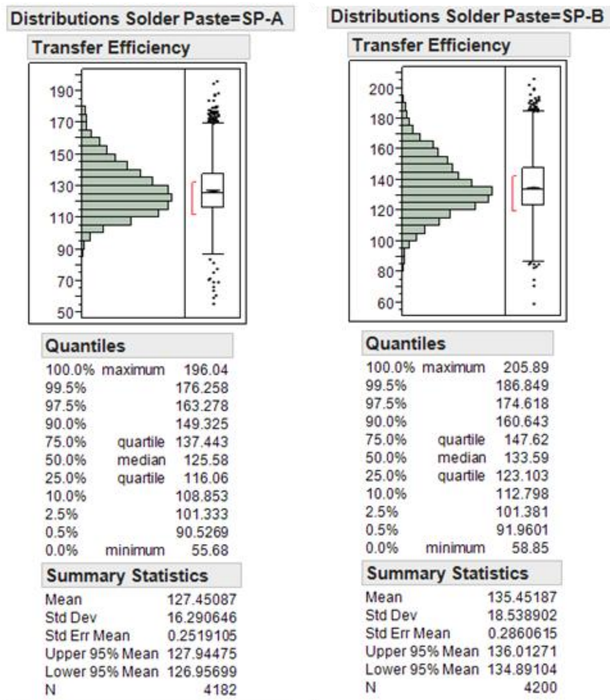


Figure 6. Distribution data for transfer efficiency (%) in all 0201M locations for each Solder Paste. Solder Paste A (Left), Solder Paste B (Right).

The mean TE% for solder paste B is higher than solder paste A, but the standard deviation and standard error are also higher for solder paste B. The distribution data is otherwise similar for these two solder pastes.

The coefficient of variation (CV) is one way to measure the process capability. CV is calculated by dividing the mean TE% by the standard deviation of TE% and converting the result into a percentage. A process with a CV of less than 10% is considered capable, a CV of 10-15% is considered marginal, and a CV of over 15% is not considered capable. The overall CV for printing solder paste A in all 0201M components is 12.7%. The overall CV for printing solder paste B in all 0201M components is 13.7%. Both CVs for these solder pastes fall into the marginal CV category.

The distribution data for TE% for solder pastes A and B broken out by 0201M location is shown below (Figures 7 and 8).

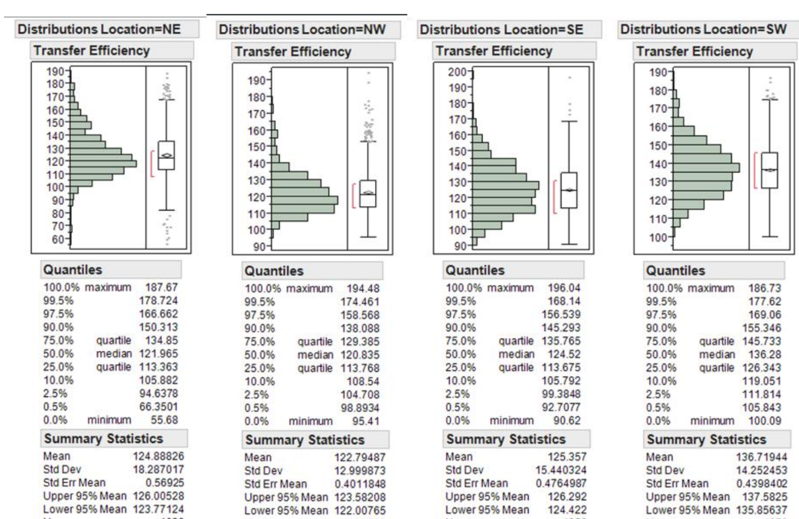


Figure 7. Distribution data for rransfer efficiency (%) for Solder Paste A separated by 0201M location.

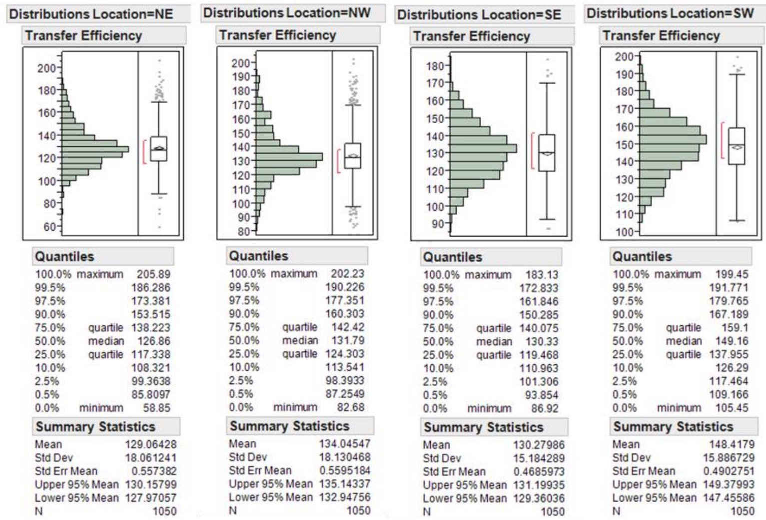


Figure 8. Distribution data for transfer efficiency (%) for Solder Paste B separated by 0201M location.

The CVs for the 0201M components and each solder paste are separated by location below (Table 6).

Table 6. Coefficient of Variation (CV) for each solder paste at each 0201M location.

0201M Location	CV% for Solder Paste A	CV% for Solder Paste B
NE	14.6	14.0
NW	10.6	13.5
SE	12.3	11.6
SW	10.4	10.7

The CV for each of these locations and both solder pastes fall into the marginal CV category. The NE location has aperture sizes of 152 x 152 µm (6.0 x 6.0 mils) and an area ratio of 0.75. The NE location gave the highest CVs for each solder paste, with the CV for solder paste A being a little higher than solder paste B. These rounded square ("squircle") shaped apertures created more randomness (higher standard deviation) in the printed solder paste deposits than the other locations which have rectangular apertures.

The SE location gave relatively moderate CVs for both solder pastes with the CV for solder paste A being a little higher than solder paste B. This location has apertures of 160 µm wide x 130 µm tall (6.3 x 5.1 mils) with an area ratio of 0.70 which is the lowest of the 0201M locations. The print direction is parallel to the smaller (130 µm) dimension of the apertures. Intuitively it would seem that as area ratio decreases, CV would increase, but this is not the case for the NE and SE locations. The SE rounded rectangular apertures (lower AR) gave lower CV than the NE squircle apertures (higher AR).

The NW location gave a relatively low CV for solder paste A and a relatively high CV for solder paste B. These apertures are 180 µm wide x 150 µm tall (7.1 x 5.9 mils) and the print direction is parallel to the smaller (150 µm) dimension of the aperture. The SW location has the same aperture design, but the apertures are rotated 90 degrees, with the print direction parallel to the larger (180 µm) dimension of the aperture. The SW location gave slightly lower CV for solder paste A than for solder paste B, and the overall lowest CV for the 0201M apertures.

The transfer efficiency varied from print to print for each solder paste as shown below (Figure 9). All 0201M locations are averaged into each box plot.

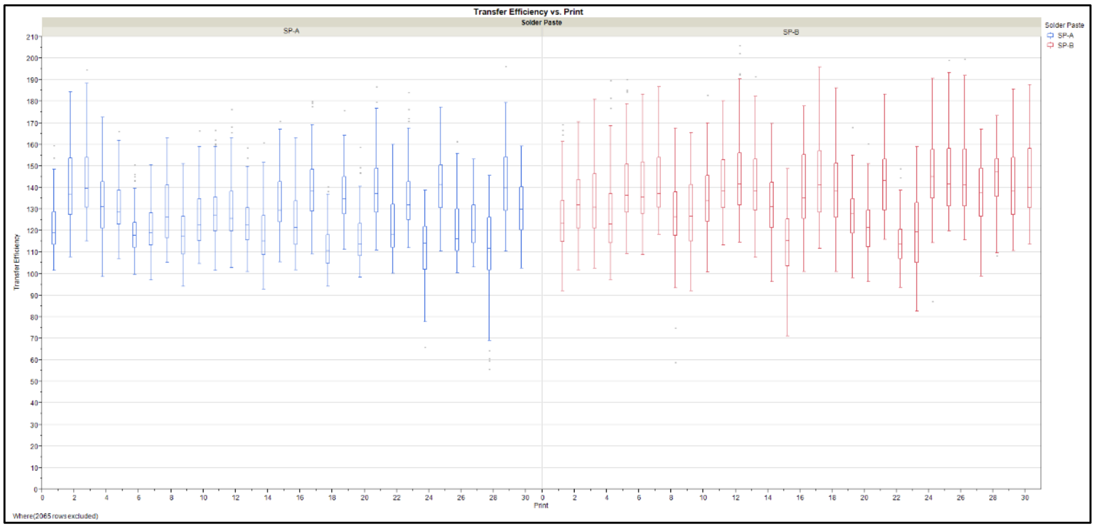
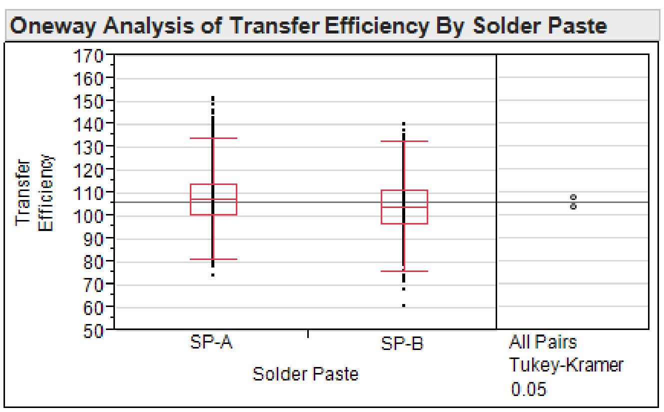


Figure 9. Transfer efficiency (%) by print # for each solder paste in all 0201M locations. Solder Paste A (Left-Blue) and Solder Paste B (Right-Red)

The TE% for each solder paste varies from print to print in the 0201M locations. There does not seem to be a consistent pattern to the TE% variation. This variation could be due to normal variations in stencil to PCBA alignment, gasketing, aperture fill, solder paste "snap off", etc.

0.3mm BGA Print Data

The printability of the solder pastes for the 0.3 mm pitch BGA components was also of particular interest. The overall transfer efficiency for each solder paste is shown below (Figure 10).



Excluded Rows 8382

Means Comparisons

Comparisons for all pairs using Tukey-Kramer HSD

Connecting Letters Report

Level			Mean
SP-A	A		107.7
SP-B		B	104.1

Levels not connected by same letter are significantly different.

Figure 10. Transfer efficiency (%) for each solder paste in All 0.3 mm BGA locations.

Solder paste A gave an overall higher TE% than solder paste B in the 0.3 mm BGA locations. This is the opposite of the TE% results for the 0201M locations, which showed higher TE% for solder paste B. The BGA pads are solder mask defined while the 0201M pads are copper defined. This difference in pad styles likely affected the print characteristics of these solder pastes.

The distribution data for TE% in the 0.3 mm BGA locations is broken out by solder paste below (Figure 11).

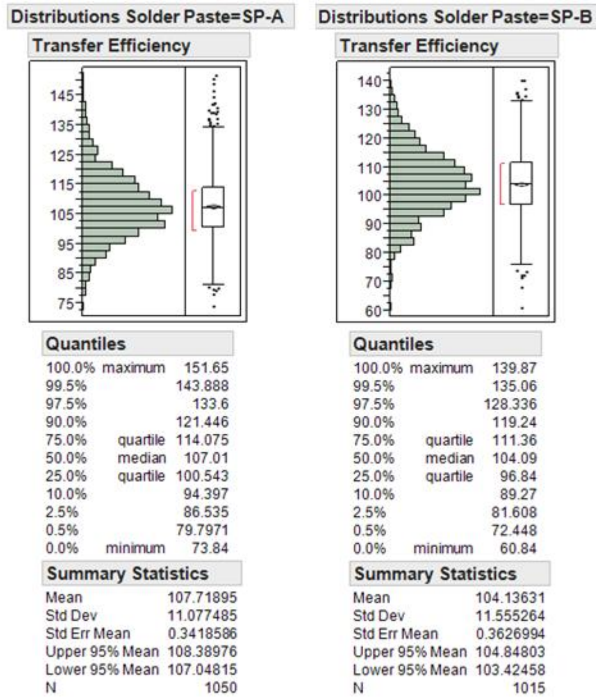


Figure 11. Distribution data for transfer efficiency (%) in all 0.3 mm BGA locations for each solder paste. Solder Paste A (Left), Solder Paste B (Right).

The CV for printing solder paste A in the 0.3 mm BGA locations is 10.3%, and the CV for printing solder paste B is 11.1%. Solder paste B has a higher CV than solder paste A, and both CVs are at the low end of the marginal category (CV of 10-15%). The overall CVs for each paste in the 0.3 mm BGA locations is significantly lower than the 0201M locations (12.7% SP-A and 13.7% SP-B).

The print-to-print variation for each solder paste in the 0.3 mm BGA locations is shown below (Figure 12).

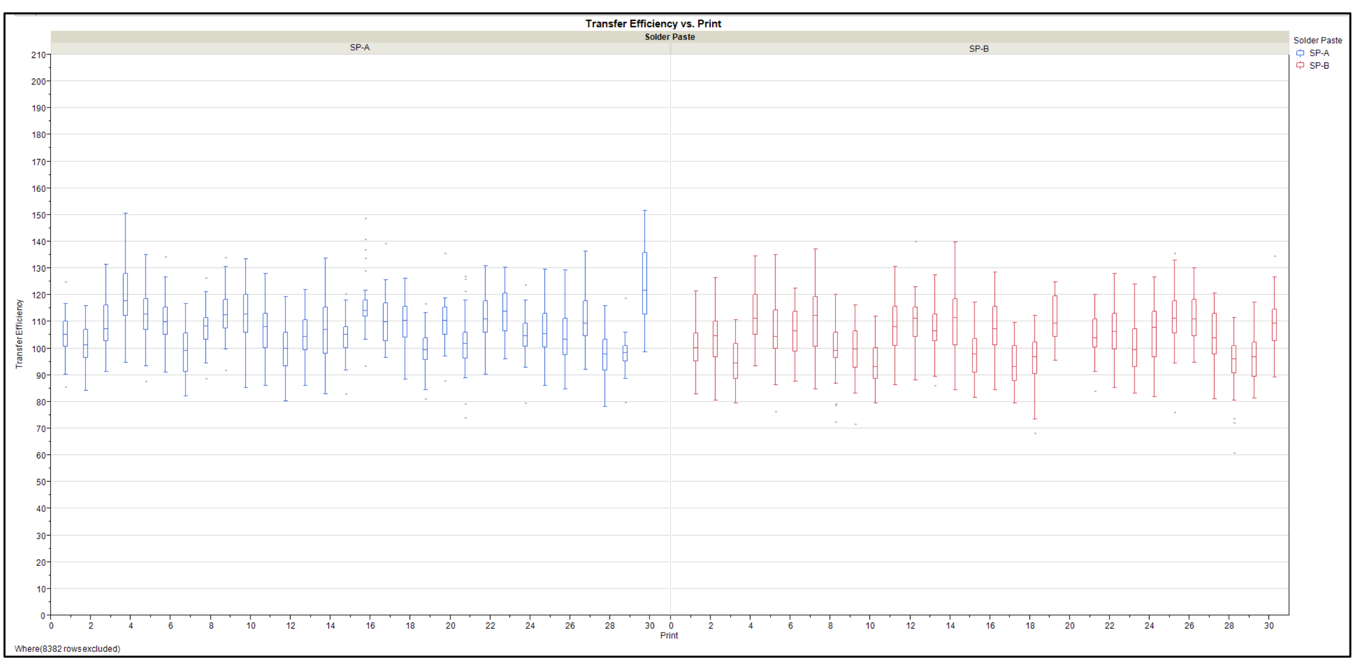


Figure 12. Transfer efficiency (%) by print # for each solder paste in All 0.3 mm BGA Locations. Solder Paste A (Left-Blue) and Solder Paste B (Right-Red)

The 0.3 mm BGAs gave less variation of TE% from print to print (Figure 14) than for the 0201M locations (Figure 11).

Print Data Comparing Both Components with the Same Stencil Design

The 0201M NE location has the same stencil design as the 0.3 mm BGA locations with 152 x 152 micron (6x6 mil) rounded square ("squircle") apertures. The pad styles are different for these two components. The 0201M NE pads are copper defined 120 µm x 145 µm (0.0047" x 0.0057") rectangles with a pad area of 17,400 µm^2. The 0.3mm BGA pads are solder mask defined 152 µm (0.006") rounds with a pad area of 18,145 µm^2. The TE% for the 152x152 micron (6 x 6 mil) apertures on the copper defined and solder mask defined (SMD) pads broken out by solder paste is shown below (Figure 13).

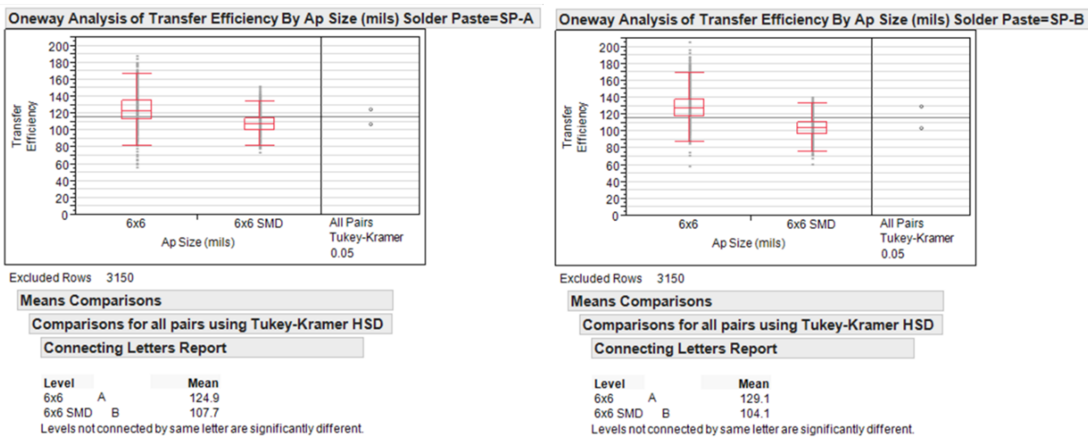


Figure 13. Transfer Efficiency (%) for the 152x152 µm (6x6 mil) Stencil Apertures on the Copper Defined (6x6) Pads and the Solder Mask Defined (6x6 SMD) Pads. Solder Paste A (Left) and Solder Paste B (Right).

The mean TE% and CVs are much higher for the 0201M NE location with copper defined pads than the 0.3 mm BGA locations with solder mask defined pads. The mean TE% and CV for each solder paste and each of these component locations is in Table 7 below.

Table 7. CV by solder paste and the 0201M NE location and All 0.3 mm BGA locations.

Location	Mean TE% for Solder Paste A	CV% for Solder Paste A	Mean TE% for Solder Paste B	CV% for Solder Paste B
0201M NE (Cu defined)	124.9	14.6	129.1	14.0
0.3 mm BGA (SM defined)	107.7	10.3	104.1	11.1

It is evident that the copper defined rectangular pads (0201M) provide for higher TE% but also higher variation than the round solder mask defined pads (0.3 mm BGA), when the same stencil designs are used for both.

Conclusions

0201M Print Data

- Solder paste B gave a higher TE% than solder paste A.
- Solder paste A had a lower CV than solder paste B, although both were in the moderately capable CV range.
- The SW location gave the highest TE% of the locations for both solder pastes and the lowest CVs. The SW pads and stencil apertures were oriented with the long edge parallel to the print direction. This orientation allowed higher solder paste volumes than for the opposite orientation (NW). The SW location CVs were near the capable end of the moderate CV range.
- The NE location gave near the lowest TE% for both solder pastes and the highest CVs. The stencil aperture was a "squircle" which in general gave significantly lower solder paste volumes than the other aperture designs. The CVs were near the high or "not capable" end of the moderate CV range.
- 30 prints over the course of approximately 1 hour showed good print consistency and repeatability.

0.3 mm BGA Print Data

- Solder paste A gave a slightly higher TE% and slightly lower CV than solder paste B. Both CVs were at the low end of the marginal CV range.
- 30 print testing showed less variation for the 0.3mm BGAs than for the 0201M locations.

Comparing Both Components with the Same Stencil Design

- The 0201M NE location gave significantly higher TE% and higher CVs than the 0.3mm BGAs for both solder pastes.
- The copper defined 0201M rectangular pads have higher TE% but less consistency than the solder mask defined 0.3mm BGA pads.

Acknowledgements

We give thanks to ITW/EAE for the use of the applications lab in Millford, MA. We also give thanks to BlueRing Stencils who provided the stencil used in this work.

References

[1] Solder Paste Task Group (5-24b), "Requirements for Solder Pastes", *IPC J-STD-005B*, March 2024.
[2] Stencil Design Task Group (5-21e), "Stencil Design Guidelines", *IPC-7525C*, November 2021.
[3] E. Nauss, "Sealed Versus Exposed Atmosphere Solder Printing Comparison for Type-6 Powder Solder Paste", *Proceedings of IPC Apex Expo*, 2024.
[4] C. Ashmore, "Investigating the Metric 0201 Assembly Process", *Proceedings of IPC Apex Expo*, 2019.
[5] S. Pei-Lim, K. Thum, A. Mackie, "Challenges in Fine Feature Solder Paste Printing for SiP Applications", *Proceedings of IMAPS International Symposium on Microelectronics 2016*.
[6] S. Joshi, "Lead-free Solder Paste Development for Ultra Fine-pitch Printing and Reflow of 03015 and 0201 Metric Chip Components", *Proceedings of SMTA International, 2016*.
[7] S. Harter, J. Franke, C. Lantzsch, "Evaluation of the Stencil Printing for Highly Miniaturized SMT Components with 03015MM in Size", *Proceedings of SMTA International, 2015*.
[8] E. Griffith, "Evolution and Applications of Fine Feature Solder Paste Printing for Heterogeneous Integration", *Proceedings of SMTA International, 2021*.
[9] A. Murling, et. al., "Fine Powder Investigation For Optimum Imperial 008004 Solder Paste Printing: Part I", *Proceedings of SMTA International, 2020*.
[10] T. Lentz, "Meeting the Challenges of Ultra-Fine Feature Printing And Reflow Through Optimization Of Pb-Free Solder Paste", *Proceedings of SMTA International, 2024*.
[11] E. Nauss, M. Butler, "Stencil Printer 008004/0201 Aperture Components", *Proceedings of SMTA International, 2019*.
[12] A. Nobari, S. St-Laurent, "Investigation of Characteristics of Lead-Free Powders for Solder Paste Application", *Proceedings of IPC Apex Expo, 2017*.