As originally published in the IPC Proceedings.

Development of a Robust 03015 Process

Robert Alexander Gray
ASM Assembly Systems LLC
Suwanee, GA
grayrobert@asmpt.com

Introduction

Modern consumer electronics are driving the adoption of smaller featured SMT devices such as 0.4 mm or smaller pitch CSP, and 01005’’ / 0402 metric discrete devices. Already roadmaps have been suggested to explore the use of smaller pitch CSP and 03015discrete devices, which are only around 64% the size of 01005 devices. On their own these challenges can be met by using stencils with thinner materials allowing sufficient area ratios to maintain the established safe area ratio guideline of 0.6% or higher.

However when having to process fine feature devices along with larger devices such as connectors and RF shields, which usually require higher paste volumes to overcome co-planarity issues, the area ratio factors encountered in real production are dropping significantly below the conventional rule of thumb of area ratios having to be above 0.6 and in some instances below 0.5 area ratios.

With this forced compromise in area ratio guidelines comes a compromise in process window robustness and subsequent print and even placement process quality.

In order to try and redress this issue, different technologies have emerged in stencil materials and treatments combined with the use of finer grades of solder paste, but the question remains:
“In isolation or by adopting a combination of these technologies is it enough, to establish a robust 03015 process?”

This paper will review major steps considered and taken for the development of a robust 03015 process which was successfully Demonstrated at the company in-house show during Productronica in November 2013, and it will focus on the activities for the solder paste print process. It will include the following topics:

1. What likely end product applications for 03015 components were considered for future use and the implications they have on the process design
2. Component design which drives pad design which in its turn will drive aperture design leading to area ratio calculations to determine workable stencil thicknesses
3. An overview of transfer efficiency and area ratio in paste printing
4. An overview of the key results from an extensive study of different stencil technologies to determine the optimum ones to maximize transfer efficiency for given area ratios
5. A discussion on the need for a safety margin in Area ratio guidelines and the major drivers behind this
6. Efforts to fully characterize the chosen solder paste to define a robust print process window with respect to operating conditions
7. Efforts to optimize the printer settings to define a robust print process window
8. A discussion of the results of the 03015 demonstration that was done several times a day for four days
9. A discussion on the conclusions of this work and suggestions for further work

The new discrete devices 03015 are at this point only just being produced in test quantities for process research and development. It may be some time before they are actually adopted for mainstream use in SMT products – perhaps the work done here will help in the decision whether the current state of the art is ready for them. This paper will discuss the results of the process and research for the 03015 demonstration. The methodologies used however can equally be applied by anyone wishing to use mainstream components for the first time such as 01005 – 0201 discrete, or new finer featured CSP devices than the user is currently used to.

What likely end product applications for 03015 components were considered for future use and the implications they have on the process design

Three main applications were considered as possible early adopters in the future for 03015 devices, if a robust SMT process can be developed:

- SMT sub modules such as RF modules used as components during smartphone PCB assembly: For these the variation in component types is limited and can be described as homogenous assembly. In this case stencil thickness can be reduced to maximize area ratio, therefore eliminating the need to provide higher volume paste deposits for large RF shields or connectors.
- Smart watches: Some smart watches will come from established players who traditionally produced “simple” fitness monitoring devices, but may now wish to increase functionality with GPS, Blue Tooth or better display capability without changing the form factor significantly. This may lead to resistance to use thin stencils as for RF modules, if older designs simply have new functionality added rather than starting design from scratch.
- Smartphones: At the moment main stream smartphones at most use 01005 devices and 0.4 pitch CSP, but there is a technology driver to increased functionality with reduced floor space of the SMT PCB. This market segment will have the most resistance to using thinner stencils which at the moment use 0.08 mm or 0.1 mm thick stencils.

In all cases at this point the use of stepped stencils will not be considered. That is based on the likelihood that component density and spacing will be such that keep-out rules for the steps will be violated.

Component design which drives pad design which in its turn will drive aperture design leading to area ratio calculations to determine workable stencil thicknesses.

If the nominal dimensions of a Metric 03015 device including tolerances are taken the appropriate PCB pad size and spacing can be determined. Based on this we determined to use a pad size of 0.15 mm by 0.12 mm, with a pad-to-pad spacing within the device of 0.1 mm.

This size would be consistent with other device pad sizes. We also used a component edge to component edge gap of 0.15 mm for what we called the “large rectangle” pad.

We further determined that in using such small components, it would be interesting to test tighter component spacing’s. However during placement there is a risk that the component squeezes out paste, potentially leading to paste-touching between adjacent components. To minimize this for this tighter component spacing, we determined to use a smaller pad design of 0.15 mm by 0.1 mm and a further stencil reduction to an aperture size of 0.13 mm by 0.09 mm. This would allow a component placement gap between component edges of 0.1 mm during our trials.

From these dimensions we could determine the smallest occurring area ratios for stencil thicknesses of 0.08 mm, 0.06 mm, and 0.05 mm.

Our testing was done on each stencil thickness, but for this paper we will discuss the use of the 0.05 mm stencil (Fig.1). The smallest pad-aperture design gives an area ratio of 0.53.

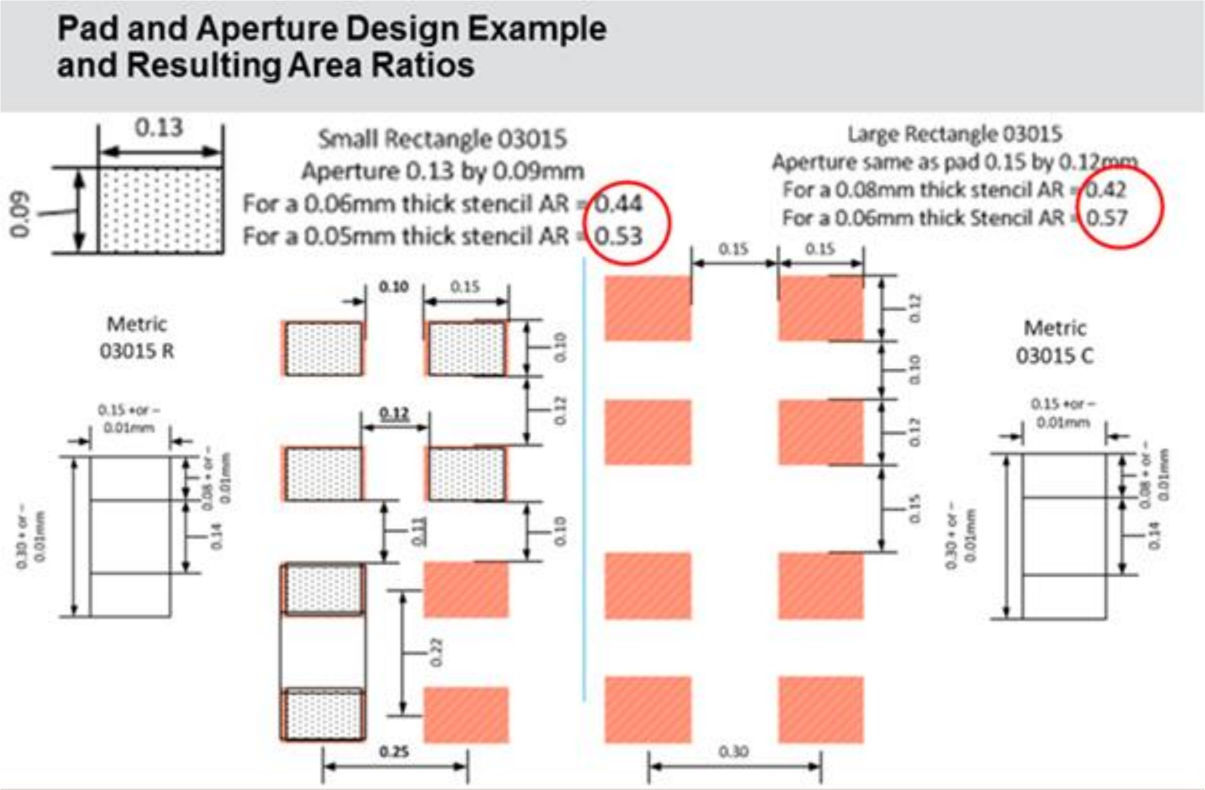


Fig. 1: Pad design: The smallest pad-aperture design gives an area ratio of 0.53

We can also see that the large pad design has an area ratio of 0.42, using a stencil thickness of 0.08 mm (Typically used thickness in smart phone manufacturing).

Our next step is to find a stencil technology that would print consistently at these area ratios.

An overview of transfer efficiency and area ratio in paste printing

The science of area ratio and transfer efficiency is well documented by now. A brief overview will only be given on: how well the print process applies solder paste to the PCB pads is measured by the term “Transfer Efficiency”. This is the percentage of the paste that ends up on the PCB pad, compared to the original theoretical volume of solder paste that was in the aperture prior to the PCB being removed from contact with the stencil after the print stroke.

How well the paste is transferred (measured either by TE-percentage, or a measure of TE-variation, the Standard Deviation, expressed through the statistical term Cp) depends greatly on the “area ratio” (Fig. 2).

The area ratio is the ratio of paste contact area with the stencil aperture walls versus the contact area of the PCB pad. Basically, as the PCB is being dropped from the stencil, the paste wants to stick to both. Which one becomes dominant depends on the contact area and to a lesser extent on surface roughness or “stickiness”.

Area Ratio, AR

$$AR = \frac{\text{Area of paste contact on the pad}}{\text{Area of paste contact on the aperture walls}}$$

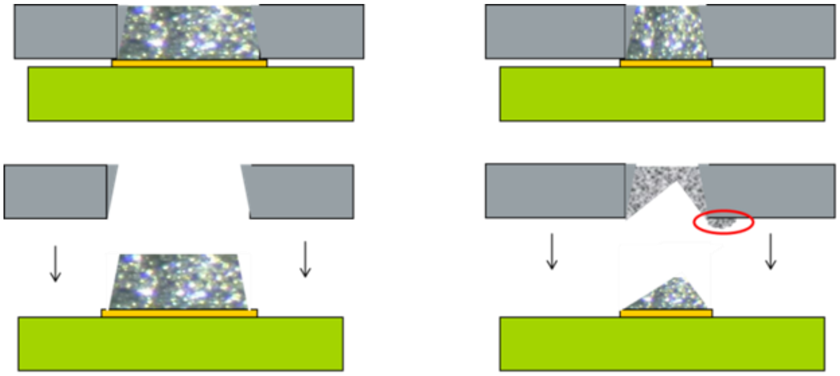
The paste wants to stick to the pad & stick to the aperture walls.
Transfer efficiency is a measure of which one is dominant !

Transfer Efficiency, TE

$$\%\ TE = \left[\frac{\text{Volume of paste deposited}}{\text{Volume of stencil aperture}}\right] \times 100$$

Fig. 2: Formulas for Area Ratio and Transfer Efficiency

With a good area ratio it would be expected that most of the paste would be transferred to the pad, and that there would be very little variation in the volume of transfer from one pad to another of the same shape aperture. As the aperture gets smaller and smaller for the same thickness of stencil, the ratio of contact for the paste to the stencil compared to the pad gets higher and hence the area ratio gets lower (Fig. 3).



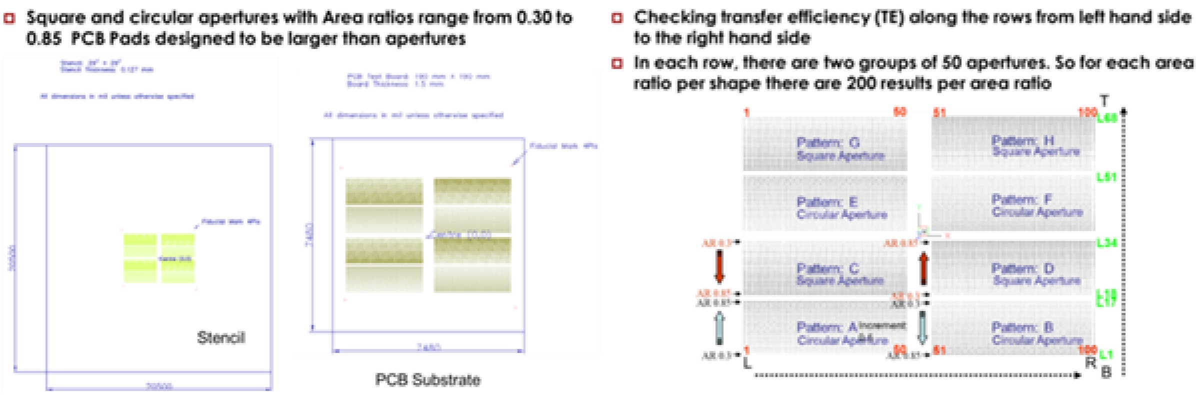
Good area ratio => High transfer efficiency & low volume variation.

Poor area ratio => low transfer efficiency & high volume variation & Paste on the left in the apertures and the bottom of the stencil.

Fig. 3: Illustration of good vs. bad area ratio and paste transfer

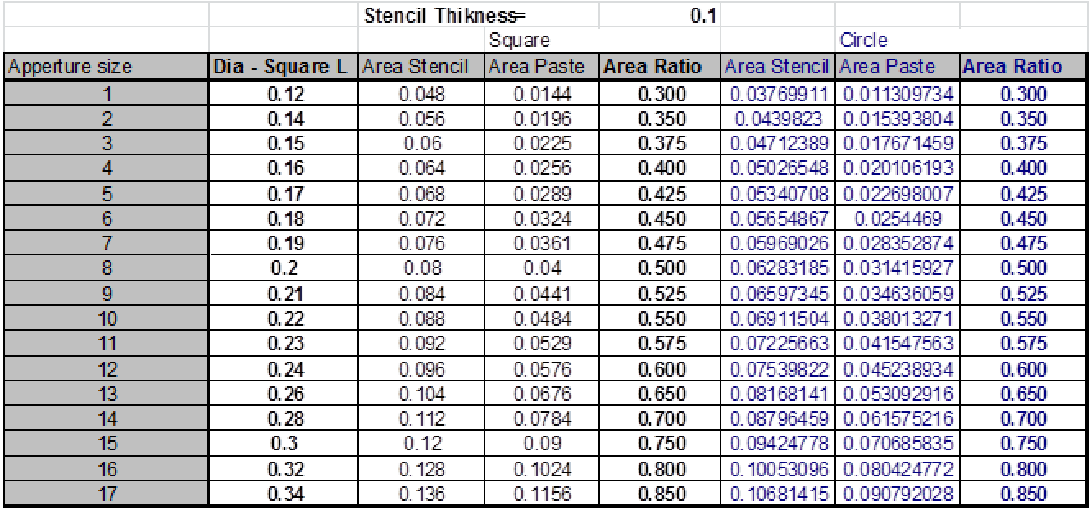
For a poor area ratio it would be expected that the volume percentage of paste transferred to the pad drops, and the variation in the volume of transfer from one pad to another of the same shape aperture gets higher. This poor transfer leaves paste in the apertures of the stencil and on the underside of the stencil around the apertures – both of which can lead to a spiral of deterioration in transfer efficiency.

In order to study transfer efficiency a test PCB and stencil design were developed (Fig. 4).



- Square and circular apertures with Area ratios range from 0.30 to 0.85 PCB Pads designed to be larger than apertures
- Checking transfer efficiency (TE) along the rows from left hand side to the right hand side
- In each row, there are two groups of 50 apertures. So for each area ratio per shape there are 200 results per area ratio

Fig. 4: To study TE a test PCB and stencil design were developed



		Stencil Thikness=		0.1			
			Square			Circle	
Apperture size	Dia - Square L	Area Stencil	Area Paste	Area Ratio	Area Stencil	Area Paste	Area Ratio
1	0.12	0.048	0.0144	0.300	0.03769911	0.011309734	0.300
2	0.14	0.056	0.0196	0.350	0.0439823	0.015393804	0.350
3	0.15	0.06	0.0225	0.375	0.04712389	0.017671459	0.375
4	0.16	0.064	0.0256	0.400	0.05026548	0.020106193	0.400
5	0.17	0.068	0.0289	0.425	0.05340708	0.022698007	0.425
6	0.18	0.072	0.0324	0.450	0.05654867	0.0254469	0.450
7	0.19	0.076	0.0361	0.475	0.05969026	0.028352874	0.475
8	0.2	0.08	0.04	0.500	0.06283185	0.031415927	0.500
9	0.21	0.084	0.0441	0.525	0.06597345	0.034636059	0.525
10	0.22	0.088	0.0484	0.550	0.06911504	0.038013271	0.550
11	0.23	0.092	0.0529	0.575	0.07225663	0.041547563	0.575
12	0.24	0.096	0.0576	0.600	0.07539822	0.045238934	0.600
13	0.26	0.104	0.0676	0.650	0.08168141	0.053092916	0.650
14	0.28	0.112	0.0784	0.700	0.08796459	0.061575216	0.700
15	0.3	0.12	0.09	0.750	0.09424778	0.070685835	0.750
16	0.32	0.128	0.1024	0.800	0.10053096	0.080424772	0.800
17	0.34	0.136	0.1156	0.850	0.10681415	0.090792028	0.850

Fig. 5: Example of aperture sizes for a 0.1 mm thick stencil

Depending on stencil thickness, the aperture size changes to achieve the desired area ratio values as can be seen in figure5.

For this stencil thickness of 0.1mm a 01005 (Inch) component could have an aperture with an area ratio anywhere from 0.525 to as low as 0.425. And a 0.4 mm pitch CSP could be as low as AR 0.5.

These levels are clearly below the traditional recommended threshold of an area ratio larger than 0.6.

A custom designed vacuum block and standard stainless steel squeegee, length 250 mm, were used for printing under the following printing parameters:

Print pressure 8 kg, Stroke speed 100 mm per second, snap-off speed 3 mm per second, snap of distance 5 mm.

The PCB was designed as copper defined pads with solder mask between each pad. See the adjacent picture for a 2-D view of the solder paste deposits from the SPI for this TE PCB (Fig. 6). As can be seen the pads are larger than the aperture sizes.

An automated 3-D solder paste inspection machine with a camera resolution of 20 microns was used to inspect the solder paste deposits and the results were exported for analysis in statistical software.

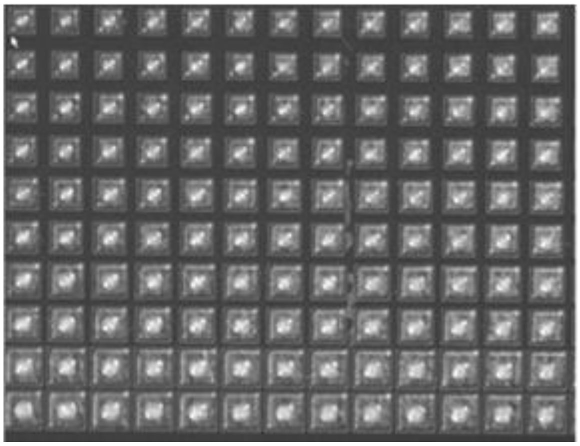


Fig. 6: 2-D view of the solder paste deposits from the SPI for TE PCB

The test procedure for all tests was particular. In order to ensure the same conditions at all times, a proper paste conditioning cycle by kneading and a proper stencil cleaning cycle post-kneading were established. The results given are generally for five sequential prints from each stroke direction, a total of 1000 apertures per area ratio shape, and type four solder paste, unless otherwise indicated.

Stencil technology selection for area ratios down to 0.42; - an overview of the key results from an extensive study of different stencil technologies to determine the optimum ones to maximize transfer efficiency for given area ratios

The principle of solder paste printing via stencil is that the paste which is first filled in the opening (by the squeegee) has in the moment of depositing less adhesion to the sidewalls of the opening than to the surface of the pad. If the adhesion to the side walls exceeds the one to the pad, no print is possible. Since the adhesion to the side walls is a function of their roughness, it is important to select the best combination of material and manufacturing methods for the stencil. Also important is the area ratio which can be calculated when the dimensions of the opening and the stencil thickness are known. Since most of the times the stencil thickness and the opening size are not really negotiable, the only remaining influencing factors are the selection of the stencil material and the method of manufacturing.

In our research the company team studied many different suppliers of stencils and technologies (Fig. 7), including:

- Laser-cut standard stainless steel
- Laser-cut fine grain stainless steel
- The use of three different electro polishing techniques
- E-formed Nickel stencils
- Laser-cut Nickel stencils
- The use of three different Nano treatments

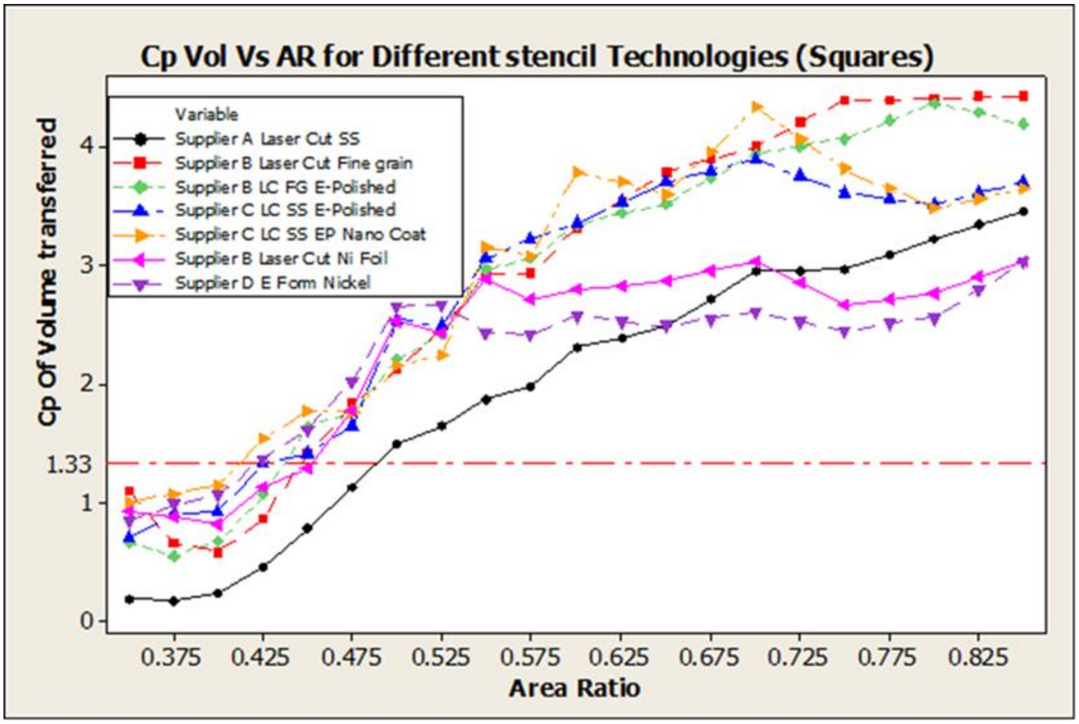


Fig. 7: Results from TE testing of different stencils using a type four paste

On the larger area ratios 0.6 and above: here we can see the nickel stencils had more variation than the others (lower Cp). There may be more variation in stencil thickness and aperture size due to the manufacturing process for Ni stencils.

Among all the stencil technologies studied, the most consistent performance was from laser-cut stencils using fine grain stainless steel with a particular type of Electro polishing after cutting, and with Nano treatment of the bottom surface and aperture walls.

It is also imperative that you choose a solder paste with good printing properties and a fine enough powder size. The solder paste powder type can be selected with the common rule of thumb in mind that five balls of powder should fit through the smallest stencil opening side by side. For our aperture design the conclusion will be to use at least a type 5 paste (Fig. 8).

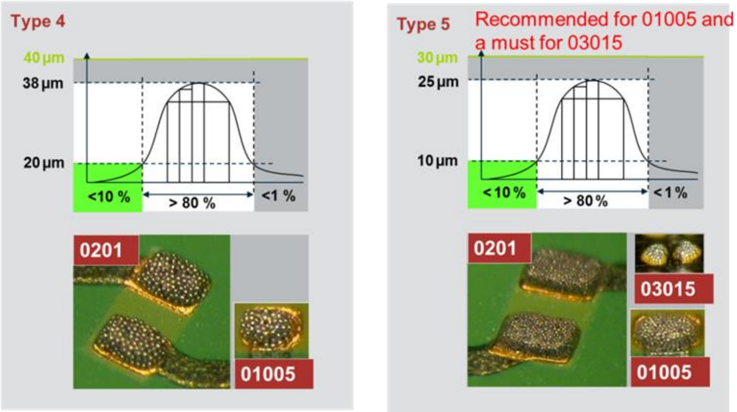


Fig. 8: Comparison of solder paste Type 4 and Type 5

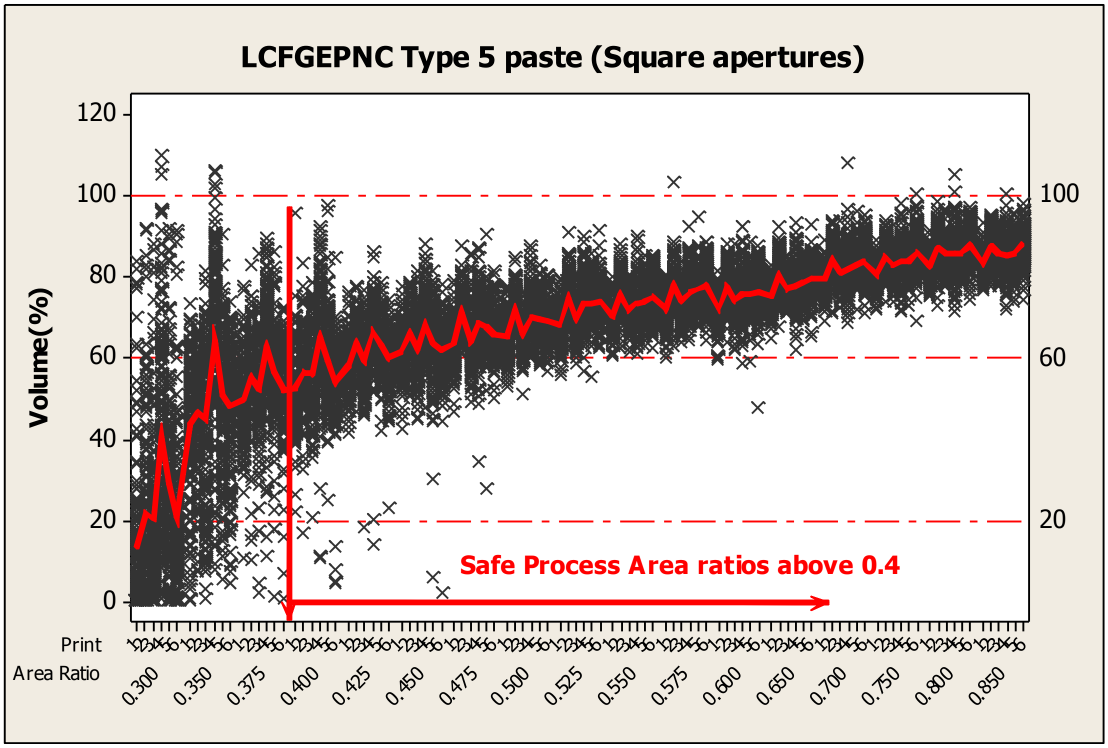


Fig. ***9:*** **Dot plot of printed volume for 200 apertures per area ratio for six prints, one after another using type five solder paste using our best performing combination of stencil technologies**

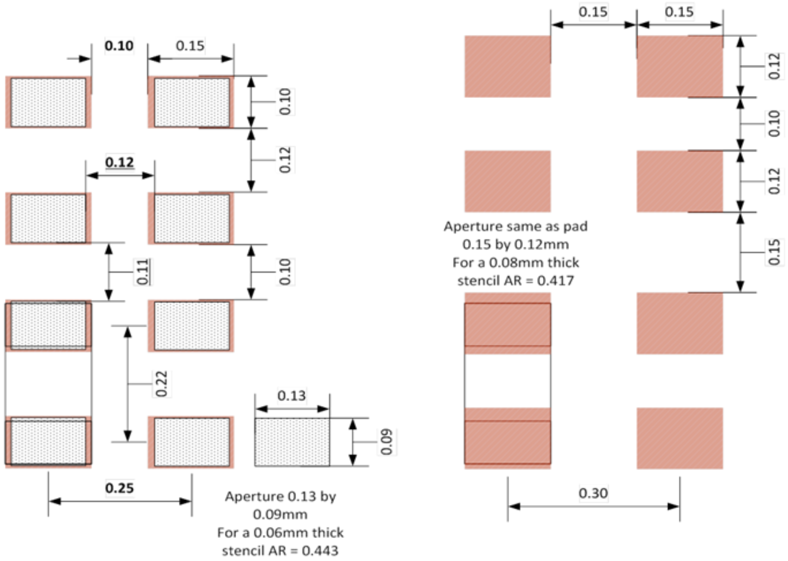


Fig. 10: A safe process should occur at area ratios above 0.4, which indicates that our designed apertures for the 03015 components may provide a good stable process

A discussion on the need for a safety margin in area ratio guidelines and the major drivers behind these sources of variation within the print process

Thermal Expansion: If a stainless steel stencil was cut at 19 degrees centigrade and used in a printer at 25 degrees centigrade, using its coefficient of expansion for every 100 mm of PCB length would result in a mismatch of 0.01038 mm.

For the PCB there is a similar magnitude of thermal expansion. It may be possible that the PCB is printed at several degrees higher temperature to the stencil, resulting in a similar level of mismatch around 0.01 mm.

So with the printer at 20 degrees centigrade and the PCB at 32 due to failure to cool from first side reflow, we may have a stencil image to PCB pad mismatch of around 0.02mm per 100 mm – a worst case scenario.

PCBs also may have dimensional tolerances per batch of up to a few thousands of mm – after first side reflow this variation may even extend to several tens of thousands of mm (around 0.07mm).

Stencil manufacturing tolerance: Depending on the type of laser-cutting machine and the way the cutting process is controlled, tolerances for individual aperture positions can range from plus or minus 0.02 to 0.03 mm up to plus or minus 0.04 mm for older generation machines. If we assume that for fine feature device stencils the best technology laser cutters and process control is used, we may still end up with an aperture to aperture variation of up to 0.03 mm.

Printer setup and accuracy: When setting up a printer, it is good practice to input an X, Y, and theta offset per stroke direction. Using only the naked eye, it is questionable whether an accuracy of more than 0.05 mm may be achieved. If however high powered microscope or SPI systems are involved in the setup, it may be possible to achieve accuracy of alignment of around plus or minus 0.02 mm (remembering even the SPI has a GRR capability for error).

Once set up there is the inherent alignment capability of the printer to consider. Some machines in use today have specifications of 0.04 mm at 6 sigma. It would be wise when printing such small features to use the best machines with accuracy capabilities of 0.025 mm or less at 6 sigma.

If we take all of these sources of variation into account it may be that even in best cases we can get an accumulated error of:

- Thermal expansion of 0.02 mm over 100 mm
- PCB stretch after first side reflow of 0.035 mm
- Stencil manufacturing variation of 0.03 mm
- Printer setup offset of 0.02 mm, plus machine accuracy of 0.025 mm
- In total a possible aperture to pad mismatch of around 0.125 mm

Imagine this with pads and apertures that are close to dimensions of 0.15 by 0.12 mm – the aperture may be completely off the pad.

This mismatch is on an aperture to aperture case and is not altogether linear along the PCB (due to stencil cutting tolerances).

If we then go on to consider component placement accuracy, it is some wonder that some components may not end up landing on paste. It is to the credit of process engineers everywhere that these possible sources of variation are minimized below these levels to achieve a producible process for 01005 and 0.4 mm pitch CSP use. This little thought exercise allows us to understand the importance of controlling all factors in the process and environment to minimize off-pad printing.

Here are the recommended measures to minimize variation:

- Thermal expansion: Ensure that all stencils are manufactured with tightly controlled room temperatures. Cutting should be performed at twenty degrees centigrade. Also in the SMT department, minimize temperature variation using air conditioning, and if necessary printer-dedicated air conditioners (in this case ensure that the printer door is kept closed to avoid over-chilling of the stencil as the chilling unit blast cold air to try and overcome the open door).

- PCB-stretch after first side reflow: Ensure regular maintenance to the cooling zone of the oven to minimize variation in PCB-stretch here. Also characterize the variation in this area and have the stencil dimensions mapped to match this stretch.

- Stencil manufacturing variation: Take steps to assess stencil suppliers. Only use those proven to be the best to manufacture fine feature stencils.

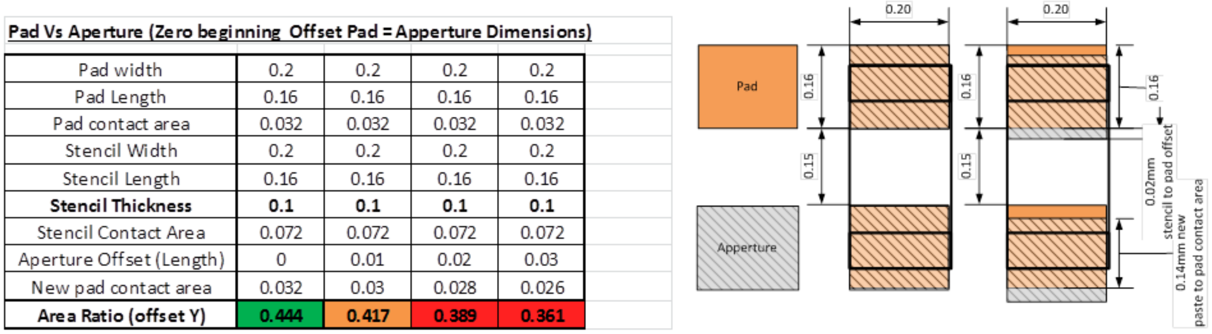
- Printer setup offset of 0.02 mm, plus machine accuracy of 0.025mm: By using the most capable printer models and high magnification optical aids, this source of variation can be minimized.

By carefully optimizing the above factors, occasional aperture to pad mismatch of around 0.05 mm can be achieved (based on the probabilities that all sources of variation accumulate rather than cancel each other out).

The effect of slight off-pad printing to volume variation

It is well known that overprinting (or off-pad printing) can be used to increase solder paste volume. However this is normally used for large component apertures where the amount of overprinting is a magnitude of a few solder balls diameter larger than the average paste particle size. However for these small components, perhaps the situation is a little different?

For type five paste, the average solder ball size can be from 0.015 to 0.025 mm in diameter, for small misalignments of one ball diameter or less. And with copper defined pads it is highly unlikely that the solder balls that are off-pad will be able to reach down and touch the PCB surface. In other words, in this overhang area the solder paste does not touch the PCB but only the stencil walls. So for an aperture the same size as the pad as the aperture moves off the pad, the contact area of paste to stencil stays constant but the contact area of paste to pad decreases (until the offset is larger than one or two ball diameters allowing the paste to drop down and touch the PCB). This has a resultant change to the area ratio calculation (Fig. 11 and Fig.12) for this effect on 01005 devices, and below that the effect on 03015 devices.

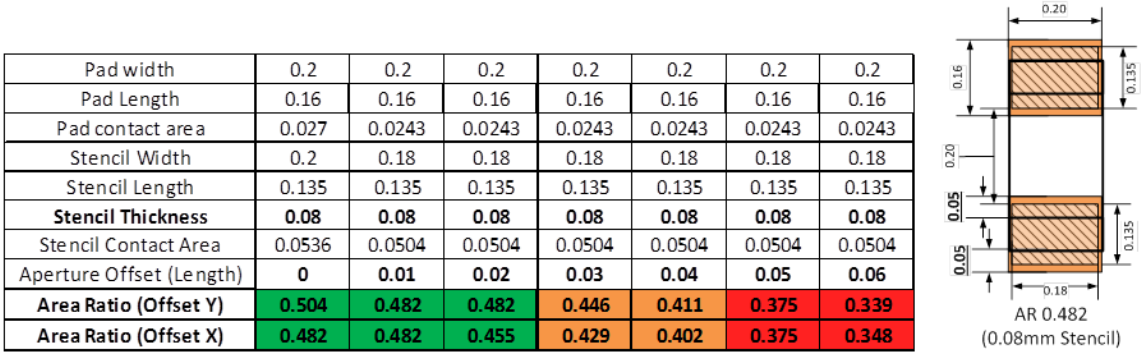


Pad Vs Aperture (Zero beginning Offset Pad = Apperture Dimensions)

Pad width	0.2	0.2	0.2	0.2
Pad Length	0.16	0.16	0.16	0.16
Pad contact area	0.032	0.032	0.032	0.032
Stencil Width	0.2	0.2	0.2	0.2
Stencil Length	0.16	0.16	0.16	0.16
Stencil Thickness	**0.1**	**0.1**	**0.1**	**0.1**
Stencil Contact Area	0.072	0.072	0.072	0.072
Aperture Offset (Length)	0	0.01	0.02	0.03
New pad contact area	0.032	0.03	0.028	0.026
Area Ratio (offset Y)	**0.444**	**0.417**	**0.389**	**0.361**

Fig. 11: Pad Versus. Aperture offset effect on contact areas and hence effective area ratios for an 01005 device and stencil thickness of 0.1mm.

From the 01005 tables in Fig. 11 we can see that for a stencil thickness of 0.1 mm in order to get close to a desired safe area ratio of 0.45, we need to print with an aperture of dimensions 0.2 mm by 0.16 mm and a pad at least that size. Also we can see that for this even a small offset of the aperture to the pad results in a deterioration of the area ratio. Even after just 0.01 mm offset our safety margin is already depleted.



Pad width	0.2	0.2	0.2	0.2	0.2	0.2	0.2
Pad Length	0.16	0.16	0.16	0.16	0.16	0.16	0.16
Pad contact area	0.027	0.0243	0.0243	0.0243	0.0243	0.0243	0.0243
Stencil Width	0.2	0.18	0.18	0.18	0.18	0.18	0.18
Stencil Length	0.135	0.135	0.135	0.135	0.135	0.135	0.135
Stencil Thickness	**0.08**	**0.08**	**0.08**	**0.08**	**0.08**	**0.08**	**0.08**
Stencil Contact Area	0.0536	0.0504	0.0504	0.0504	0.0504	0.0504	0.0504
Aperture Offset (Length)	**0**	**0.01**	**0.02**	**0.03**	**0.04**	**0.05**	**0.06**
Area Ratio (Offset Y)	**0.504**	**0.482**	**0.482**	**0.446**	**0.411**	**0.375**	**0.339**
Area Ratio (Offset X)	**0.482**	**0.482**	**0.455**	**0.429**	**0.402**	**0.375**	**0.348**

Fig. 12: Pad Versus. Aperture offset effect on contact areas and hence effective area ratios for a 01005 device and stencil thickness of 0.8mm.

However if we reduce the stencil thickness to 0.08 mm and use a reduction in aperture size to pad size as illustrated in Fig. 12 we have a much more tolerant process window. An offset of 0.03 mm is required to begin to deplete the safety margin.

From the 03015 table in Fig. 13 we can see that for a 0.15 mm by 0.12 mm aperture and a stencil thickness of 0.08mm, our area ratio is already very dangerously close to the limit in transfer efficiency, with the best minimum safe area ratio being 0.4 and above leaving no safety margin. However by dropping to a stencil thickness of 0.06 mm (or even better to 0.05 mm thick) we can see that we have a process window safety margin allowing up to 0.02 mm offset before we become concerned.

Pad Vs Apperture offset effect on 03015 device								
(Zero beginning Offset Pad = Apperture Dimensions)								
Pad width	0.15	0.15	0.15	0.15	0.15	0.15	0.15	0.15
Pad Length	0.12	0.12	0.12	0.12	0.12	0.12	0.12	0.12
Pad contact area	0.018	0.018	0.018	0.018	0.018	0.018	0.018	0.018
Stencil Width	0.15	0.15	0.15	0.15	0.15	0.15	0.15	0.15
Stencil Length	0.12	0.12	0.12	0.12	0.12	0.12	0.12	0.12
Stencil Thikness	0.08	0.08	0.08	0.08	0.08	0.08	0.08	0.08
Stencil Contact Area	0.0432	0.0432	0.0432	0.0432	0.0432	0.0432	0.0432	0.0432
Aperture Offset (Length)	0	0.01	0.02	0.03	0.04	0.05	0.06	0.07
New pad contact area	0.018	0.0165	0.015	0.0135	0.012	0.0105	0.009	0.0075
Area Ratio	**0.417**	**0.382**	**0.347**	**0.313**	**0.278**	**0.243**	**0.208**	**0.174**

Pad Vs Apperture offset effect on 03015 device								
(Zero beginning Offset Pad = Apperture Dimensions)								
Pad width	0.15	0.15	0.15	0.15	0.15	0.15	0.15	0.15
Pad Length	0.12	0.12	0.12	0.12	0.12	0.12	0.12	0.12
Pad contact area	0.018	0.018	0.018	0.018	0.018	0.018	0.018	0.018
Stencil Width	0.15	0.15	0.15	0.15	0.15	0.15	0.15	0.15
Stencil Length	0.12	0.12	0.12	0.12	0.12	0.12	0.12	0.12
Stencil Thikness	0.06	0.06	0.06	0.06	0.06	0.06	0.06	0.06
Stencil Contact Area	0.0324	0.0324	0.0324	0.0324	0.0324	0.0324	0.0324	0.0324
Aperture Offset (Length)	0	0.01	0.02	0.03	0.04	0.05	0.06	0.07
New pad contact area	0.018	0.0165	0.015	0.0135	0.012	0.0105	0.009	0.0075
Area Ratio	**0.556**	**0.509**	**0.463**	**0.417**	**0.370**	**0.324**	**0.278**	**0.231**

Fig. 13: The same holds true for 03015 components

This is more in accordance to what we expected could be the likely process variation in a well-controlled process. The fact that this deterioration in area ratio due to aperture to pad offset has such an effect on these small feature apertures is the reason why we must be cautious in setting a safety margin and declaring a safe area ratio for such small feature devices. It is also why we must take all steps to minimize variation in all the possible sources of process variation.

We also prepared a stencil with 03015 apertures and on this stencil we had some apertures directly on the pad as designed and some apertures offset from the edge of the pad by 0.02mm (01005 pads used to allow one aperture to remain on the pad while the other one reduced its contact to the pad when offset)

During this experiment we noted as predicted a drop of mean volume by about 20% and a increase in volume variation at this lower volume from a four sigma process for on the pad and less than three sigma process for the 0.02mm off the pad apertures.

The diagram below Fig 14. shows a picture of each print deposit and it can clearly be seen that for the offset aperture only part of the paste is left on the pad, that part which is off the pad does not stay on the PCB but rather stays with the stencil as predicted.

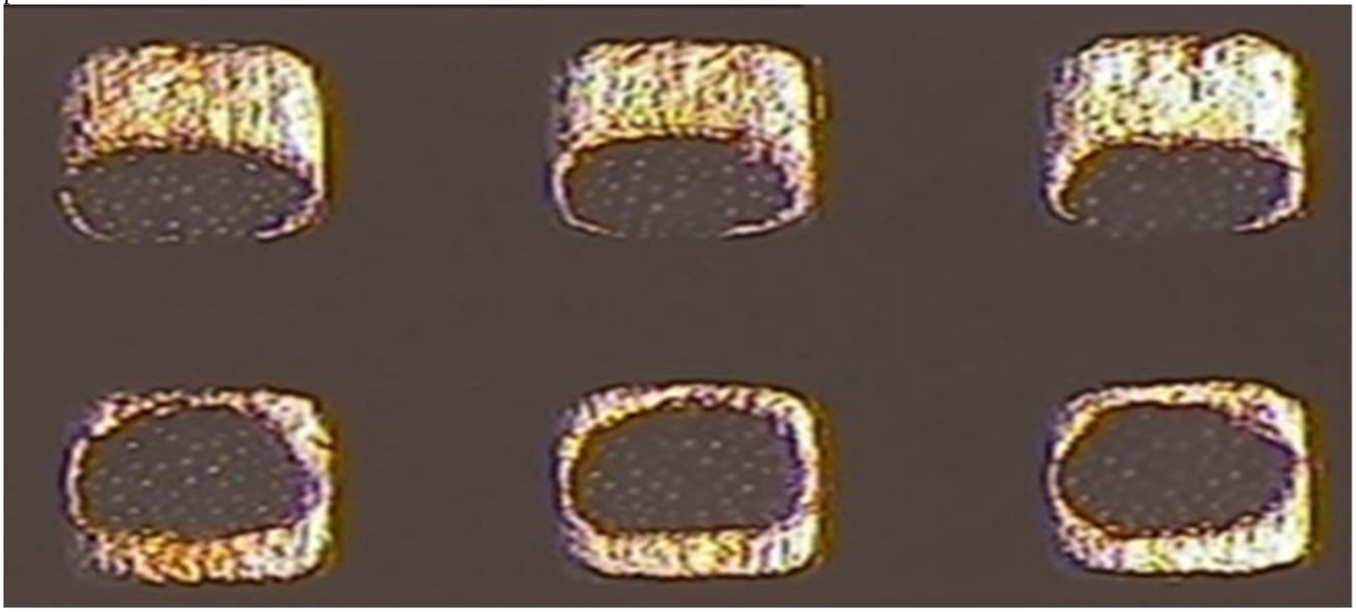


Fig. 14 Same size apertures with a stencil offset showing less paste deposited due to paste overhang reducing effective area ratio.

This illustrates why it is so important to place the aperture accurately on the pad since any offset from the pad results in a large drop in volume and also variation of the solder deposit potentially leading to tombstoning and other defects. Because of this placement of the component must also be accurately on the pad coordinates, offsetting the placement to the paste offset does not match the real paste center and does not solve the problem of more variation in volume.

A test board was manufactured with these pad designs along with stencils using the apertures and technologies discussed (Fig. 15) Sections 4A, 6A, 4B, 6B, 4D, 6D, 4C, and 6C were used for the print placement and reflow demonstration. (4A, 6A, 4C, and 6C having a component edge to component edge gap of 0.15 mm, and 4B, 6B, 4D, and 6D having a component edge to component edge gap of 0.1 mm)

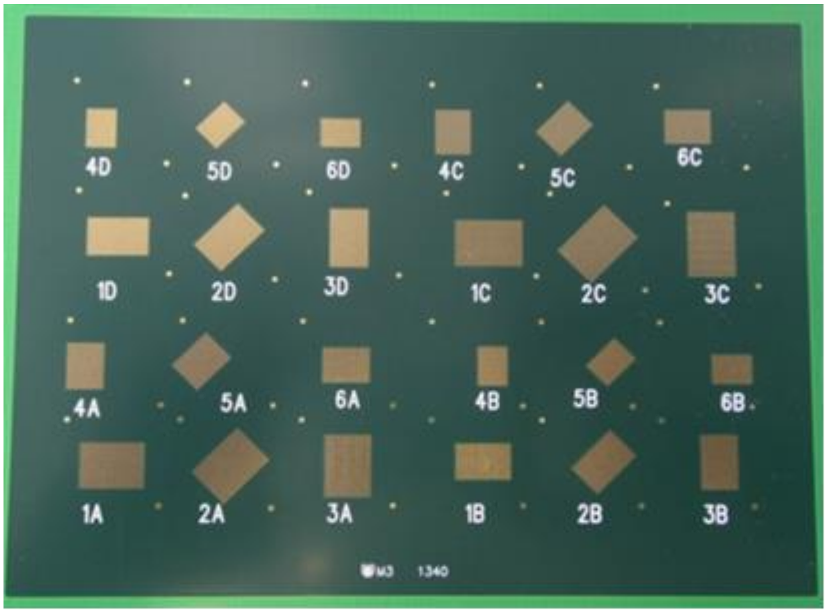


Fig. 15 Test board with the pad designs and sections illustrated (Sections 1, 2, and 3 were 01005 arrays)

Efforts to fully characterize the chosen solder paste to define a robust print process window with respect to operating conditions

The solder paste that we chose to use was the same paste as was used in our stencil benchmarking experimentation but rather than type 4 paste we used type five paste for the 03015 demonstration project (no special effort was taken to determine an optimum paste other than to use type five). In order to optimize the conditions of use during the demonstration we had to understand the behavior of that paste over different conditions such as temperature and print to print dwell times (particularly since during the demonstration we would have up to an hour of idle time between demonstration runs.)

As expected, optimum printing temperatures for paste are around the 18 to 22 degree centigrade range. For our demonstration we would be in a well-controlled air conditioned room which maintains such temperatures.

But also we had to understand the effect on transfer efficiency of this particular paste with respect to the “energy imparted in the paste”. This is best considered as the effect on paste viscosity over time after the initial paste conditioning normalization exercise used in our experimentation (Fig. 16). Please see below a graphic which shows the CP of transferred paste volume for an area ratio of 0.425 aperture for the following conditions:

1. Print immediately after the ten strokes knead conditioning cycle (in this instance the paste has a lot of energy imparted and has the lowest viscosity due to the “hyper activation” of such extensive knead strokes).
2. Ten minutes time after kneading before printing
3. Twenty minutes time after kneading before printing
4. Thirty minutes time after kneading before printing

5. Forty five minutes time after kneading before printing
6. Forty five minutes time (Found to be a time when this particular paste has a large change in printability) followed by a small kneading cycle to avoid hyper activation then printing.

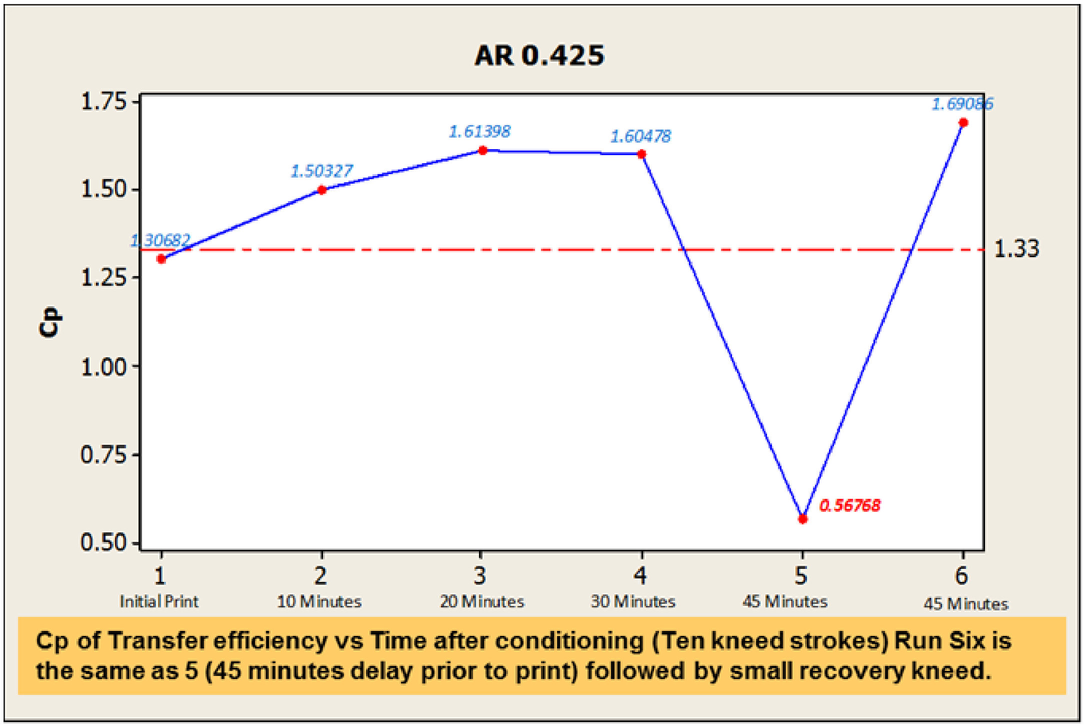


Fig. 16 Cp of transferred paste volume for an area ratio of 0.425 apertures for the above conditions, the same holds true for the smaller 03015 apertures.

From this chart we can see the small effect of "hyper activation" which was part of our initial TE testing pre-conditioning routine, so we learned to avoid such hyper activation to minimize this slight effect. But more importantly for this particular paste we could see that between thirty minutes and forty-five minutes print delay time, this particular paste experiences a large change in printability.

From this it was determined that during the demonstration the best way to avoid this effect was that every thirty minutes after printing we would print one PCB to maintain the paste viscosity and therefore printability in a stable region. Note not all pastes experience such effects at this delay time but it is wise to understand the behavior of whatever paste is to be used.

Efforts to optimize the printer settings to define a robust print process window
An experiment was undertaken to determine the optimum settings for three main parameters in the printer setup, and one noise factor was studied (delay time after the small knead conditioning cycle and first print). A Taguchi L9 test was performed to study and to adapt a consistent and sustained printing process for the chosen stencil technology and paste type. Below you can see such a test plan, featuring conditions used and applied process steps when finding the solder paste printing recipe settings (See details in Table 1).

Table 1. General conditions and settings for the DOE.

GENERAL CONDITIONS		
Preparations	Conditions	
Solder paste room temperature	22 degree Celsius, +/- 3 degrees	Relative humidity 50%, +/- 20%
Kneading solder paste	1 times front and back print	TE board used for set up
Stencil cleaning	1st cycle, Wet, dry, vacuum, 2 times	
Compressed Air	Air blast	
Stencil cleaning	2nd cycle, Wet, dry, vacuum, 2 times	
Print process	Print 1 board per setting	Taguchi Table
Inspection	Auto. Solder Paste Inspection 3D	Cp, Standard deviation

PROCESS STEPS		
Process	Conditions	
Change recipe settings	Taguchi L9 table	Speed, pressure, snap off, dwell time
Exercise solder paste	Preparation table	Kneading
Perform stencil cleaning	Preparation table	3 steps
Wait dwell time	L9 table	Time
Print board	L9 table	Stroke Speed, Blade print pressure, PCB snap off from stencil speed, dwell time.
Inspect result	3D SPI	Cp, Standard deviation

TAGUCHI L9 TEST TABLE									
Board ID	L9.1	L9.2	L9.3	L9.4	L9.5	L9.6	L9.7	L9.8	L9.9
Stroke Speed mm/s	60	60	60	100	100	100	140	140	140
Stroke Pressure N	60	80	100	60	80	100	60	80	100
Separation Speed mm/s	1	5	10	5	10	1	10	1	5
Dwell Time minutes	1	5	10	10	1	5	5	10	1

We carried out the L9 test to achieve lowest variation for printing process with selected solder paste type 5. We used the company designed TE board and the chosen stencil technology combination (Fig. 17).

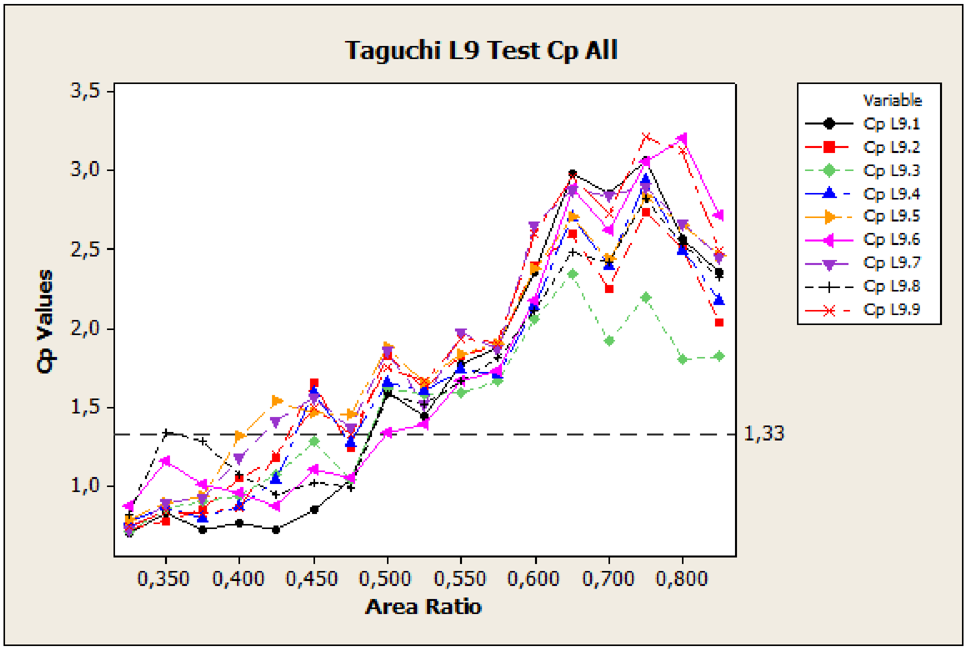


Fig. 17 L9 Test, All Cp results for each area ratio

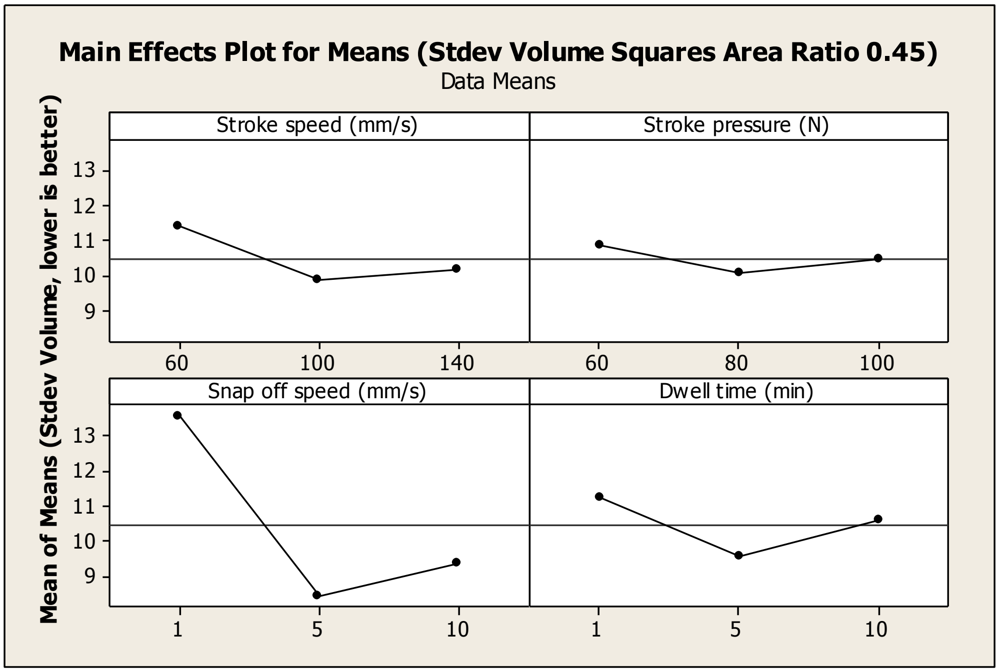


Fig. 18 L9 Test main effects plot for the means.

Table 2 Settings and values from the L9 test

<table>
<tr><th colspan="10">TEST TABLE</th></tr>
<tr><td>Board ID</td><td>L9.1</td><td>L9.2</td><td>L9.3</td><td>L9.4</td><td>L9.5</td><td>L9.6</td><td>L9.7</td><td>L9.8</td><td>L9.9</td></tr>
<tr><td>Stroke Speed mm/s</td><td>60</td><td>60</td><td>60</td><td>100</td><td>100</td><td>100</td><td>140</td><td>140</td><td>140</td></tr>
<tr><td>Stroke Pressure N</td><td>60</td><td>80</td><td>100</td><td>60</td><td>80</td><td>100</td><td>60</td><td>80</td><td>100</td></tr>
<tr><td>Separation Speed mm/s</td><td>1</td><td>5</td><td>10</td><td>5</td><td>10</td><td>1</td><td>10</td><td>1</td><td>5</td></tr>
<tr><td>Dwell Time minutes</td><td>1</td><td>5</td><td>10</td><td>10</td><td>1</td><td>5</td><td>5</td><td>10</td><td>1</td></tr>
<tr><th colspan="10">ANALYSIS</th></tr>
<tr><td colspan="10">Stdev versus ABCD</td></tr>
<tr><td>Predicted values</td><td colspan="4">S/N ratio -20,175</td><td colspan="5">Mean 9,95</td></tr>
<tr><td>Factor levels for predictions</td><td colspan="2">A</td><td colspan="2">B</td><td colspan="3">C</td><td colspan="2">D</td></tr>
<tr><td>Design levels</td><td colspan="2">1</td><td colspan="2">1</td><td colspan="3">2</td><td colspan="2">1</td></tr>
<tr><td>Actual values</td><td colspan="2">60</td><td colspan="2">60</td><td colspan="3">5</td><td colspan="2">1</td></tr>
</table>

The test shows us that for our preferred minimum area ratio of 0.45, for the small aperture design the main effect is snap-off speed. With a stroke speed of 60 mm/s, stroke pressure of 60 N, separation speed of 5 mm/s and dwell time 1 minute, we will achieve statistically stable, predictable and consistent printing process. After a confirmation run to verify these values these would be the parameters used during the 03015 print demonstration.

A discussion of the results of the 03015 demonstration that was done several times a day for four days

For the given aperture designs we had achieved a four sigma process for paste volume transferred on the small aperture and a five sigma process for the larger aperture and pad design (using the 0.05mm thick stencil).

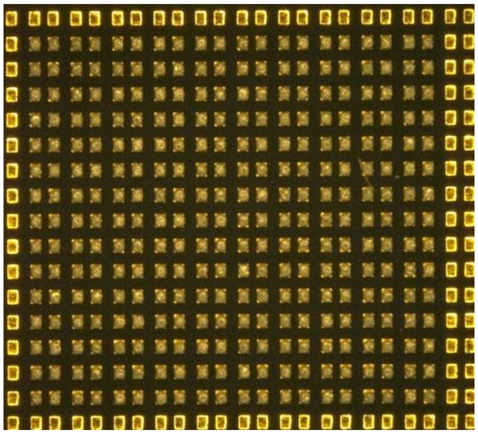
During the days of the demonstration in front of the eyes of many of the world's SMT process engineers, the team;-

Ran sixty PCBs, **placed 36,000 03015 components** with **Zero placement** defects and a **component pick-up rate of 99.95%.**

At printing, the company team:

- **Printed 72,000 apertures** and had **one print defect found at SPI**. This print defect was a squashed solder ball coined in such a way as to partially block one aperture
- **A 15 DPMO process at printing**
- **Five Defects Per Million Opportunities DPMO SMT process overall**.

Below in Fig 19 and 20 you can find some pictures of the printed pads for the 0.15 mm by 0.12 mm pad combined with the 0.13 mm by 0.09 mm reduced size aperture.



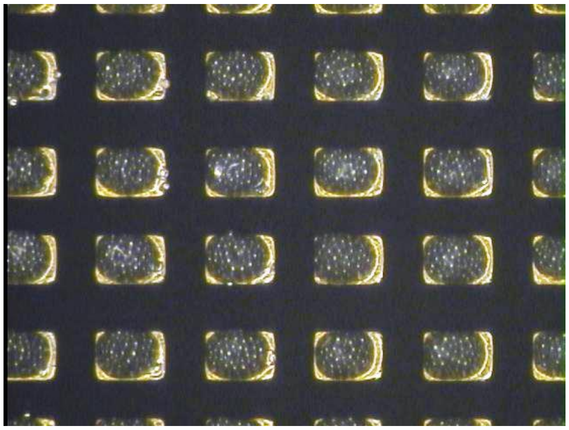


Fig. 19: Printed pads for the 0.15 mm by 0.1 mm pad combined with the 0.13 mm by 0.09 mm reduced size aperture
Fig. 20: More printed pads for the 0.15 mm by 0.1 mm pad combined with the 0.13 mm by 0.09 mm reduced size aperture (Higher magnification)

Note the precision of stencil alignment and aperture shape and position. Also you can see below that using the smaller aperture design a component gap of 0.1 mm does not lead to paste squeeze-out causing the paste to touch adjacent area deposits, and you can see the excellent placement accuracies achieved during the demonstration.

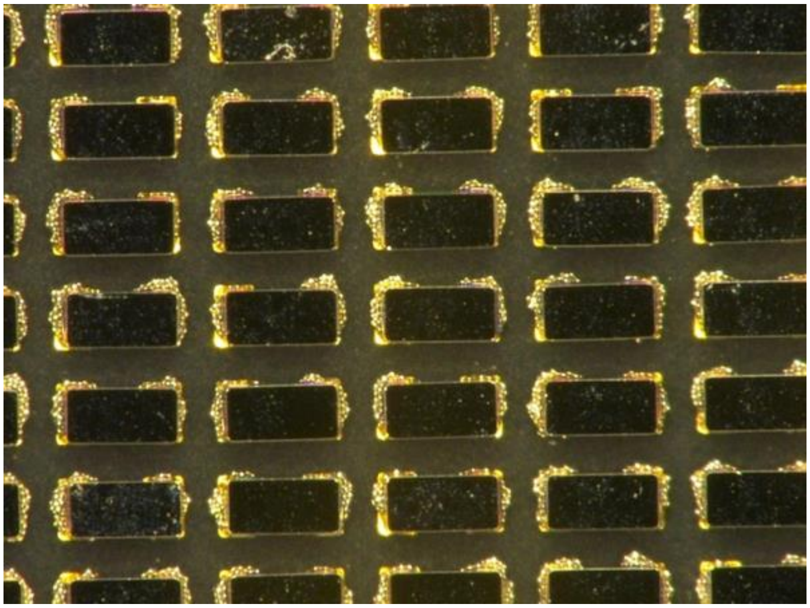


Fig. 21: Printed pads for the 0.15 mm by 0.12 mm pad combined with the 0.13 mm by 0.09 mm reduced size aperture

Here are some important factors that need to be considered for successful 03015 printing:

- Full characterization of the chosen paste: this includes DOE to find optimum squeegee speed and force, snap-off speed, paste viscosity over time and use
- Extreme accuracy of stencil alignment: since the smallest aperture dimension is 0.09 mm, any misalignment of the stencil to the PCB can place the aperture partially off pad during printing. This will reduce the effective area ratio (Paste contact with pad) leading to rapid deterioration in print consistency
- Extreme accuracy of stencil aperture cutting: just like stencil alignment any aperture out of position can lead to rapid deterioration in print consistency.

- PCB dimensional variation: different PCB technologies have different tolerances in PCB dimensional variation, due to stretch; just like stencil alignment this PCB stretch variation can lead to rapid deterioration in print consistency
- Good PCB support during printing and snap-off (release from the stencil): generally this is best done using custom designed vacuum tooling

Here are some important factors that need to be considered for successful 03015 placement:

- **Keeping boards still and steady:**Even the slightest vibration causes errors when you place 03015 components. High-quality board clamps on the conveyor systems of the placement machines and company pin support keep each board safe and steady. In this case, a special pin placement unit automatically places the support pins in their pre-programmed positions

- **High precision feeders to ensure precise component supply:** With precise direct drives and automatic pickup position correction, high precision feeders ensure that each 03015 component is picked up with consistent reliability, in this case the standard 8mm feeder was used.

- **Gentle vacuum:** Placement nozzles that meet the special requirements of 03015 and other super-small components in terms of size, material and shape. Additionally machines that clean and switch out these nozzles automatically during long term production maintain a robust process. With perfect interaction of precision in feeder, placement head and nozzle technology, each 03015 can be optimally picked up without being touched

- **Perfect process control:** Component sensors that validate each pickup, and high-resolution vision systems use their adjustable lighting capabilities to ensure the quality and alignment of each component individually for a no compromise vision measurement leads to exceptional level of placement reliability

- **Optimized placement sequence:** Software tools that automatically ensure the optimum placement sequence (to avoid that large components get in the way or shields are placed before the components they cover, etc.) ensure optimum placement optimisation without compromises due to shadowing from adjacent tall components..

- **Small size requires extra-special care:** To be able to place super-small components on equally small pads without damaging the component or squishing the solder paste, the force, speed and timing of the placement must be determined with great granularity. Special sensors can make sure that even PCB bulges or thickness variations are properly taken into account

A discussion on the conclusions of this work and suggestions for further work

In this paper we have presented results from the 0.05 mm thick stencil, and from the demonstration we would say that with the appropriate care taken to select good stencil technology from a competent supplier and the steps taken here to fully optimize and characterize the process once this particular 03015 device is available, then a stable process using a 0.05 mm thick stencil may be possible. Of course further work will be needed in this case but for module manufacturers this may be of interest. One condition may be that after SPI, if a solder defect is found, then that sub module on the PCB may not be placed avoiding difficult rework and waste of components.

Our other work on the 0.06 mm thick stencil indicates that it may also be suitable for module manufacturing under the same conditions. However the question that has to be asked is: is this enough paste height for the component variety on smart watches? And can the larger PCB cost of the smart watch support scrapping sub-PCB if there are print defects (even if components are not placed)?

For the 0.08 mm stencil, our results were marginal (though not quoted here) for the larger aperture and pad (0.15 mm by 0.12 mm). At this stage is not felt that the process is robust enough. Perhaps some optimization of the paste rheology may improve the situation and may be worthy of further work. Also increasing the pad and aperture size to 0.16 mm by 0.14 mm may achieve an acceptable area ratio in order to achieve a better process capability. But that work has still to be done, and may also have an effect on other defect types such as mid-chip solder balls, tomb-stoning on three- or five-sided terminated devices (we placed and reflowed the production 03015 device which is a single sided termination).

During the demonstration we did not use ultrasonic activated Squeegee blades for 03015 device printing, however our testing does indicate this technology can extend the safe process window for the transfer efficiency performance using our TE test vehicle, see below figure 22 showing the print performance using ultrasonic activated blades and type five paste for comparison with Fig 9 (The same conditions without ultrasonic activated blades.)

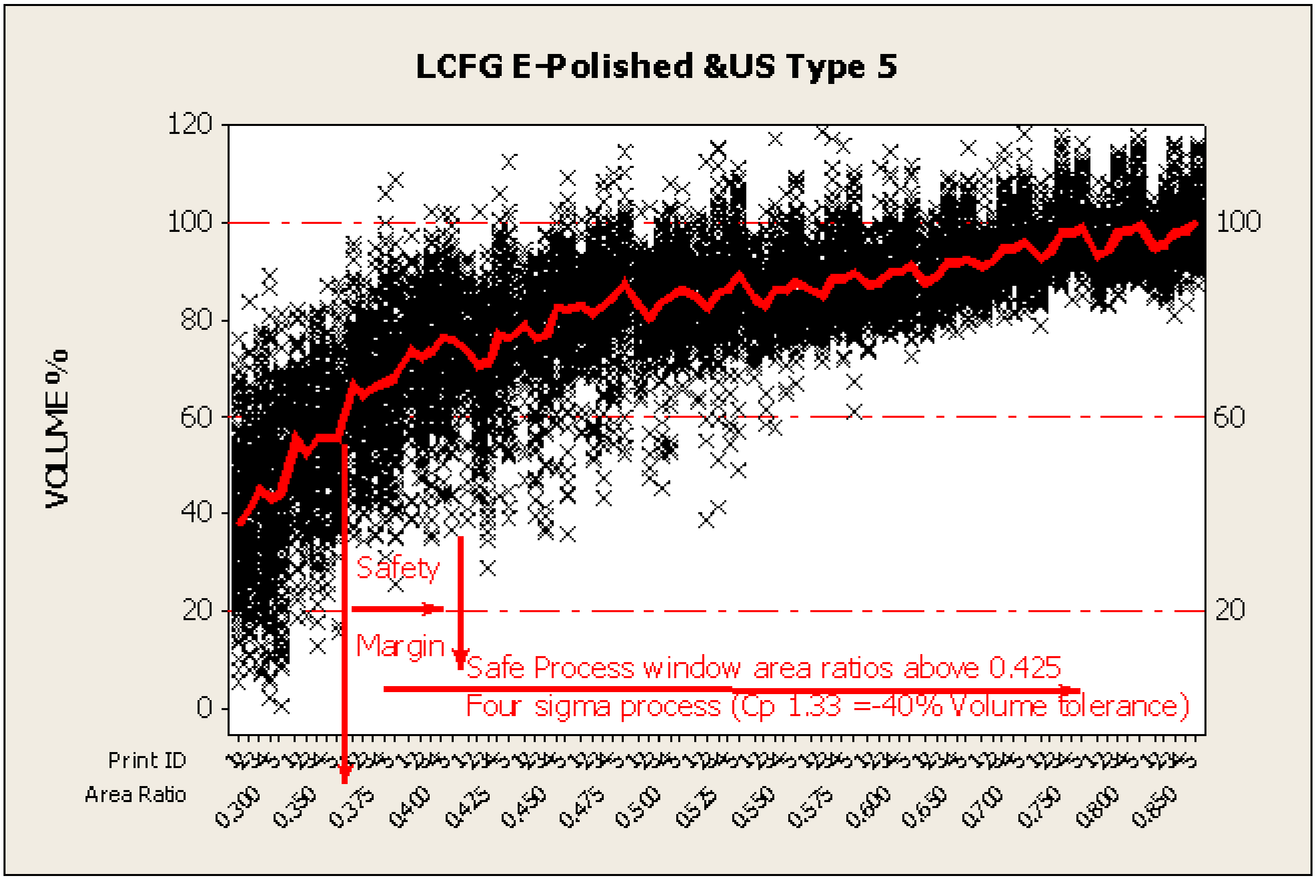


Fig. 22: TE Performance of laser cut Fine grain E-Polished stencil with ultrasonically activated squeegees

The results of the ultrasonic blade trials indicate that their use does shift the safe process window threshold to the left in the TE curve and volume variation, this indicates further work in this area is worthwhile to measure the same effects on the smaller 03015 apertures.

Concerning placement, the demonstration has successfully demonstrated the capability to place such small 03015 devices on a standard machine at full speed. This demonstration was done with component spacings of 0.15 mm and 0.1 mm component edge to component edge. This was in line with the original objective of this demonstration, to consider the future implementation of these devices in production of such objects as smartphones where there will be resistance to reducing stencil thicknesses.

In other studies the team has successfully done placement trials with component spacing less than 0.1 mm edge to edge. Of course appropriate reductions in stencil geometry are required to ensure no paste “shorting” between these small gaps after placement.

More work needs to be done to study the behavior of the other types of 03015 devices and their needs for different solder volumes and optimum pad dimensions to minimize tomb stoning and other defects (fig. 23).

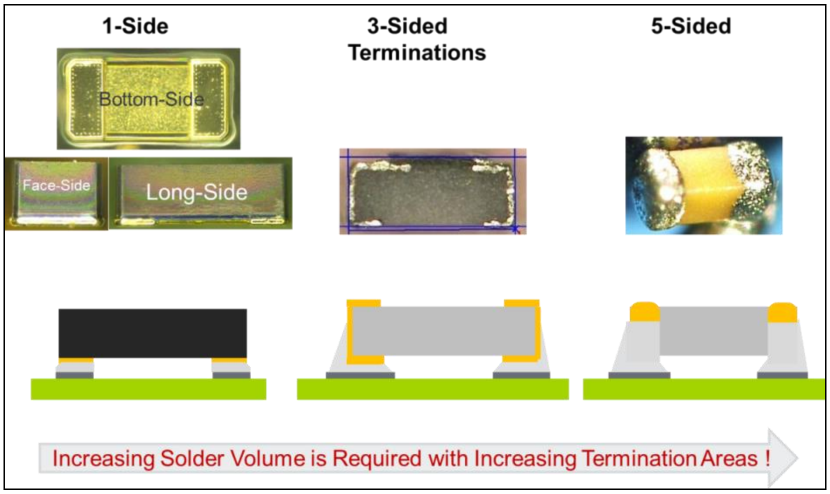


Fig. 23: The three main differences for 03015 (1-, 3-, or 5-sided termination)