ADVANCES IN FINE PITCH PRINTING PROCESS TECHNOLOGY

Isaiah Smith
Speedline Technologies, Inc.
Franklin, Massachusetts, USA
iSmith@speedlinetech.com

ABSTRACT

Increasingly fine pitch component patterns on PCBs, and ever-smaller passive components such as 03015's continue to narrow the process window for PCB assemblers. The drive toward ever-finer pitches is driven by a number of factors, particularly miniaturization in mobile devices such as Smartphones, where consumers are demanding greater functionality in compact, hand-held devices. As a result, material handling and component placement becomes more difficult, and solder paste printing is especially impacted.

Thinner stencils, smaller particle size solder paste, and smaller apertures, whereby even aperture shape becomes a factor in volume fill, test the limits of the successful printing process. Achieving sufficient or acceptable paste volume fill of fine-feature apertures has become an issue. Printing machine technology continues to advance to meet the fine-pitch challenge, with greater precision and motion control, closed-loop feedback systems, enclosed media printing, enhanced software tools and machine vision assists, and many other developments and innovations designed to enable successful and repeatable fine pitch solder paste printing for high-density SMT assemblies. This paper details those developments, and illustrates how they are contributing to process improvement through comparative test results.

Key words: printing, fine pitch, aperture, stencil

INTRODUCTION:

The Challenges of Fine Pitch (FP) Printing

In the drive to maintain lowest possible defect levels and highest yields, given that, by most estimates, 70% of all SMT soldering defects originate in the printing process, suppliers of printing machines, stencils, and solder pastes have labored mightily to deal with ever-shrinking board topographies and pad sizes. Printers have become more precise, with greater degrees of control over process parameters including accuracy, paste pressure, print speed, and others, with added advanced features including paste height sensing, vision alignment, solvent-based stencil wiping systems, and even integration with solder paste inspection (SPI) machines with programs for auto-correction of print registration based on M2M SPI feedback. The goal is to keep results defect-free and repeatably so. Solder paste manufacturers continue to refine the chemistry of their formulations and offer a range of ever-finer particle sizes in their pastes. Stencil manufacturers have tried different systems, such as electroformed stencils and nano-coatings to improve the odds and keep the printing process window open as fine-pitch printing requirements have grown finer. These methods have met with success for the most part, but have come with associated higher costs for systems, products, and accessories. Electroformed stencils are expensive, and nano coatings are known to wear off after a limited number of print cycles.

Low-tech Solution?

The challenges of fine pitch printing have inspired some creative thinking in manufacturing engineers; so it's also fair game to explore the possibility of modifying existing materials to achieve acceptable results; after all, obtaining high yields achieves nothing in the end if the cost of doing so negates profitability. In fact, we maintain that under most conditions, electroformed and nano-coated stencils aren't necessary at all, but rather, a different way of thinking about stencil apertures and how to optimize their design to achieve best results with fine pitch given a few 'rules' about the basics of *area ratio* and how it affects solder paste printing, no matter what stencil material, or type of printing machine is used.

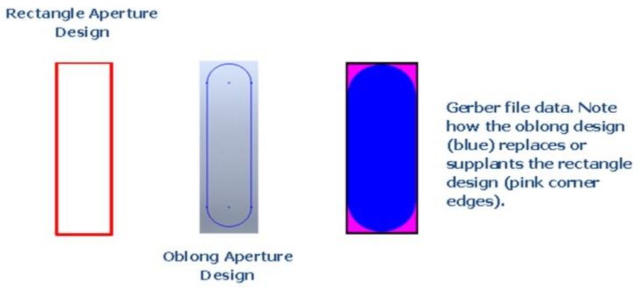


Figure 1. Rounded-off corners on rectangular aperture designs facilitate paste release for Fine Pitch.

We maintain that it quite possible to achieve better yields using a standard laser-cut stencil; but that re-thinking and modifying (or 'editing' as the major stencil houses like to say) the Gerber data for the stencil aperture cut can produce better results with fine pitch.

Area Ratio is Critical

As mentioned earlier, fine pitch printing results are directly tied to stencil aperture area ratio, a relationship between aperture area and stencil thickness. One may not be able to control the size of the fine pitch pads that the stencil is being fabricated to print onto, but printing results can be enhanced by modification of aperture design, keeping a few principles in mind.

It might also be suggested that building and maintaining a good working relationship with one's stencil house is invaluable, not only when creating the stencil design and patterns, but at every step in the stencil's fabrication process, so that a thorough understanding of the possibilities (as well as limitations) is shared and well understood. The knowledgeable counsel of experienced stencil fabrication professionals can help prevent many print and downstream defects and issues that might be prevented at the design stage.

When designing aperture cuts, it is very important to keep the principles of area ratio in mind. For example, when calculating area ratio, which incidentally is 0.66 for laser-cut stencils, best results are obtained when one can 'aim for the happy median' of that number; for example, a number too far to the left (or right) of 0.66 could result in defects further downstream in the manufacturing process.

When ordering a stencil it is important to request the appropriate stencil thickness to ensure proper paste release from the stencil. The aspect ratio for rectangular apertures should be 1.5, and an area ratio of 0.66. It is best to evaluate the smallest apertures on the stencil to determine the proper stencil thickness.

One may use the following formulas to select an appropriate stencil thickness:

Aspect Ratio = Aperture Width (W) / Stencil Thickness (T) [should be 1,5]

Area Ratio = Area of Pad (LxW) / Area of Aperture Walls (2 x (LxW) x T) [should be 0.66]

It is more common to use the area ratio as a guide to paste-transfer efficiency. The area ratio is the area beneath the stencil aperture opening, divided by the area of the inside aperture wall. For a rectangular aperture, the area ratio = $[(L \times W)/2(L+W)T)]$, where L and W are the aperture length and width, respectively, and T is stencil thickness. For a square aperture, area ratio = $S/4T$, where S is the side of the square. For a circular aperture, area ratio = $D/4T$, where D is the diameter of the circle. The generally accepted rule is to achieve area ratios of >0.66 for paste transfer. This was true until the advent of electroformed stencils. It is common with this technology to print apertures with area ratios down to 0.50. [1]

The definition of aperture area ratio for is the area of the stencil aperture opening divided by the area of the aperture side walls. A simple calculation shows that area ratio (AR) reduces to diameter (D) of a circle divided by 4 times stencil thickness (t): $AR = D/4t$. It should be noted that results for circular apertures are the same values as are calculated for square apertures, with D now equal to the side of the square. (formula for calculating area ratio). Here are some stencil aperture design recommendations that I have found work well for me as a process engineer:

Rule #1: Round off the Corners

Just because the SMT pad is square doesn't mean that your aperture has to match it. The key is volume of paste deposited, and you want it to be consistent every time. Solder paste tends to remain in sharp corners of square apertures, not depositing on the pad as it should and thereby causing clogs. This of course results in print defects (insufficiencies or missing) and then soldering defects if not detected; starved solder joints, or no solder joints result downstream. The high surface are of the inside of the aperture at the corners, versus the pad surface area, will cause the paste to adhere to the aperture walls in those corners and not transfer to the pad after a print stroke.

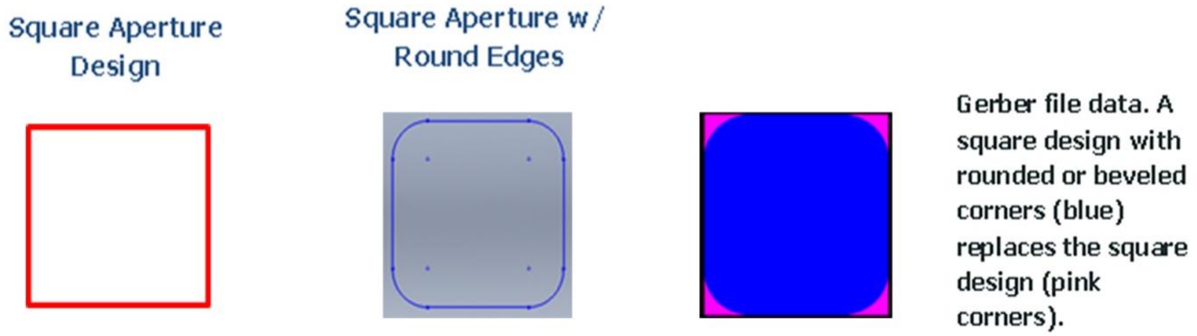
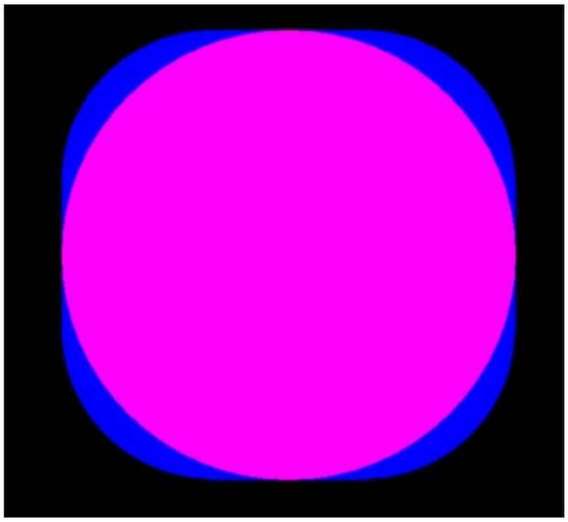


Figure 2. Rounded corners on square aperture designs.

Instead of using a rectangle aperture cut design, use an oblong aperture cut. Getting rid of the sharp corner edges will reduce the potential of paste clogging in the corners. Wetting out to the corners of the pad during reflow should not be a problem with good solderability (pad) and an optimized reflow process, lead-free notwithstanding.

Rule #2: BGA Pads – The Beveled Rectangle

The opposite principle actually applies to a BGA pad, which is, understandably, circular. However if your circular pad (and aperture) are very small in diameter, the shape itself can negatively affect print registration, especially for very fine pitch packages of chip-scale size and micro-BGAs. The emergence of ever-more-compact smart phones and mobile/wireless devices is driving the expanding use of 0.3 millimeter (mm) ultra-fine pitch micro CSPs (chip scale packages), micro BGAs (ball grid arrays), and other active devices. For example, with a two-layer CSP, 0.3 mm pitch means that the distance is 11.8 mil between the center of a given solder ball and the center of the next solder ball.



Using a square BGA aperture design with rounded edges facilitates better print registration yields.

Added to that, under older design guidelines, PCB design and layout engineers would design a solder-ball-joint pad with the diameter of 20 percent less than the diameter of a BGA/CSP solder ball, i.e., using the formula A = B = 80% x ball-diameter where the size of the aperture is actually 80% of the diameter of the pad size [2]. So we're dealing with very tiny apertures and for fine-pitch or micro BGAs, my experience has shown that best results are obtained by turning the circle into more of a square with rounded or beveled edges – a beveled rectangle, if you will. This will produce better print registration yields. The squaring of the shape (**Figure 3**) is not enough to cause bridging even if it falls slightly outside of the land area. 125- to 150-µm-thick stencil foils are the standard for this application, but remember that to maximize yields, good release, a consistent volume of solder paste, and optimized aperture shapes are critical, especially as ball pitches decrease.

EXPERIMENTAL RESULTS

A stencil for printing fine pitch SMT pads was created using the abovementioned guidelines for aperture design. Print tests were run and the results recorded, using data obtained from a Solder Paste Inspection (SPI) machine. The histograms for each component footprint follow. In all of these examples, we are basically comparing the average (mean) volume percentage, area percentage and standard deviation.

Process Program Parameters:

- Board Size: 10" X 8"
- Force: 19 lbs.
- Speed: 1.5 Inches/Sec
- Squeegee Blades: 12"
- Solder Paste: Indium 3.2 (PB-Free) Type 4
- Separation Profile: N/A
- Wiper Profile: Vacuum, Solvent and Dry Wipe after each print
- Stencil Thickness: 4mil (Laser Cut)

CONCLUSION

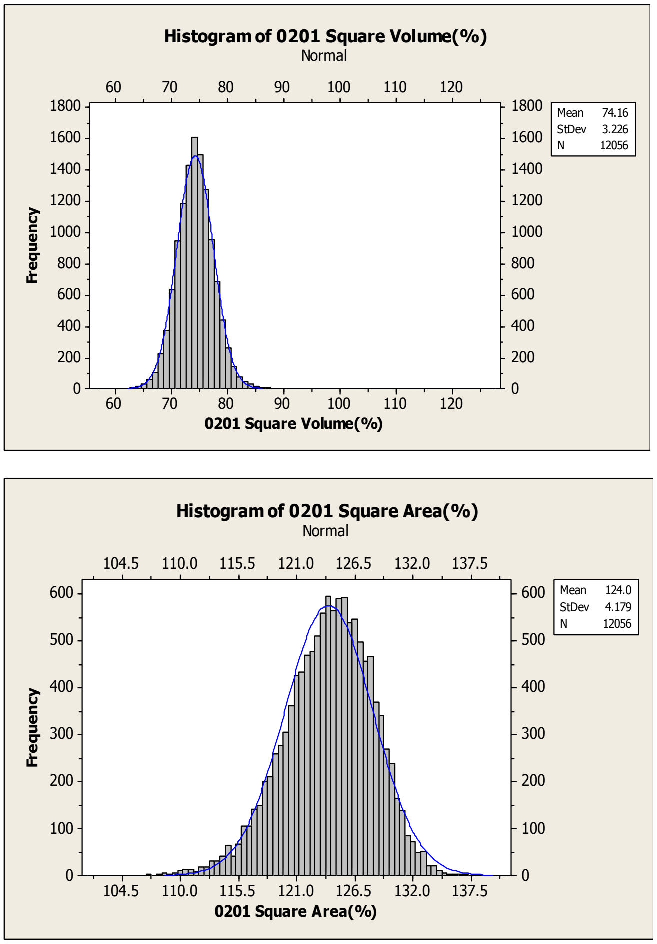
In almost every example, following the aforementioned stencil aperture design guidelines, the edited stencil apertures exhibited higher print volume and area percentages, and also exhibited a lower standard deviation, which indicates that the modified stencil is more repeatable in terms of results (good volume percentage) for these fine-pitch printing applications.

Figure 3(Left): Reverse strategy for BGA apertures, i.e., squaring the circle slightly.

REFERENCES

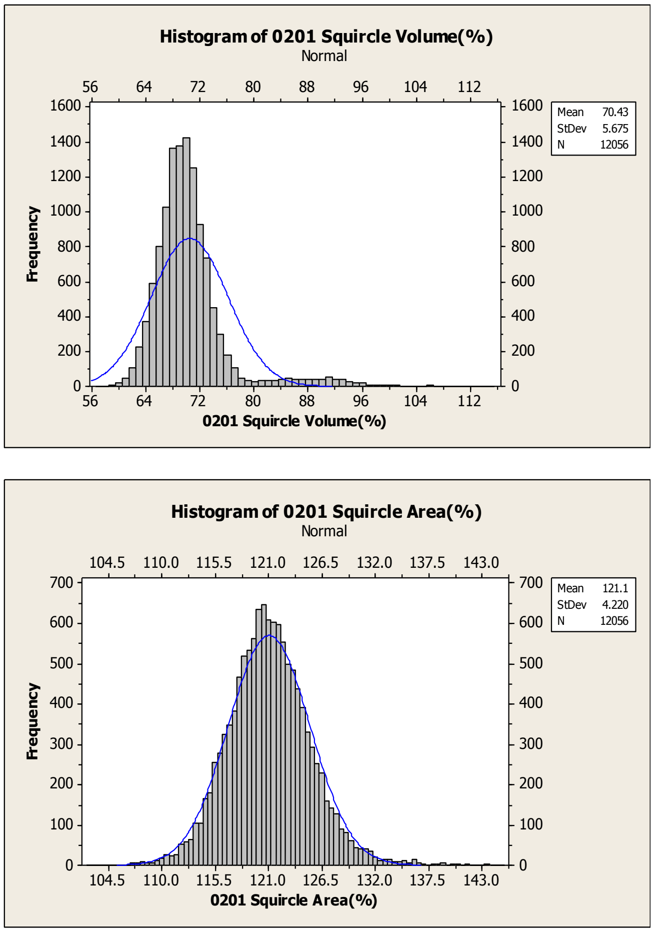
[1] "Choosing a Stencil" by William E. Coleman, Ph.D., and Michael R. Burgess, PhotoStencil, SMT magazine.

[2] "Taking on the 0.3mm Ultra-Fine Pitch Device Challenge in PCB Design" by Michael Yu, Nexlogic Technologies.



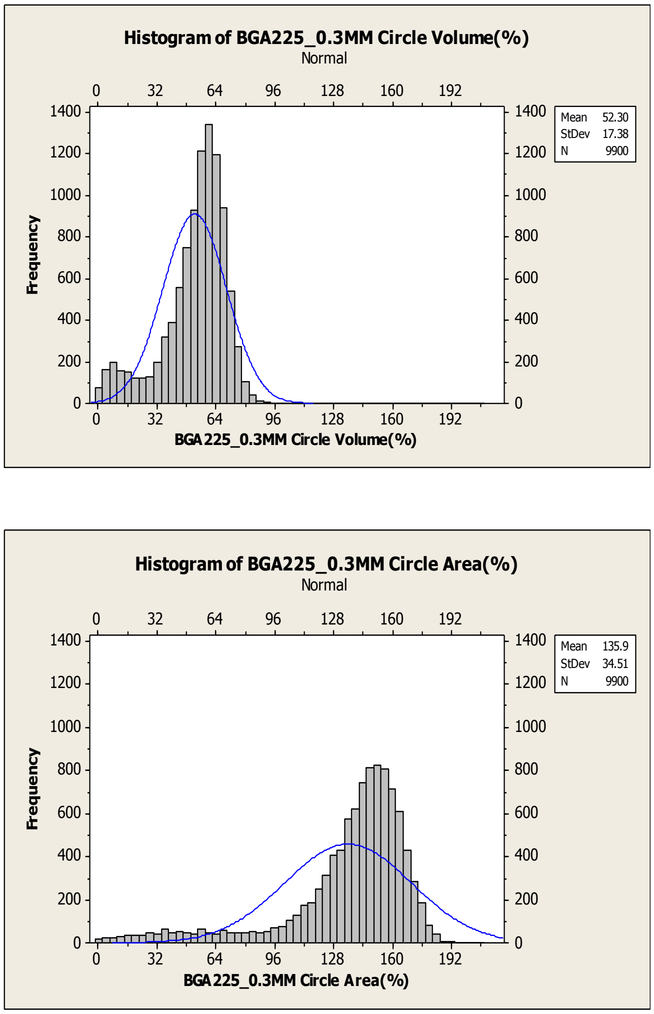
Figures 4a, b: 0201 Square Aperture:

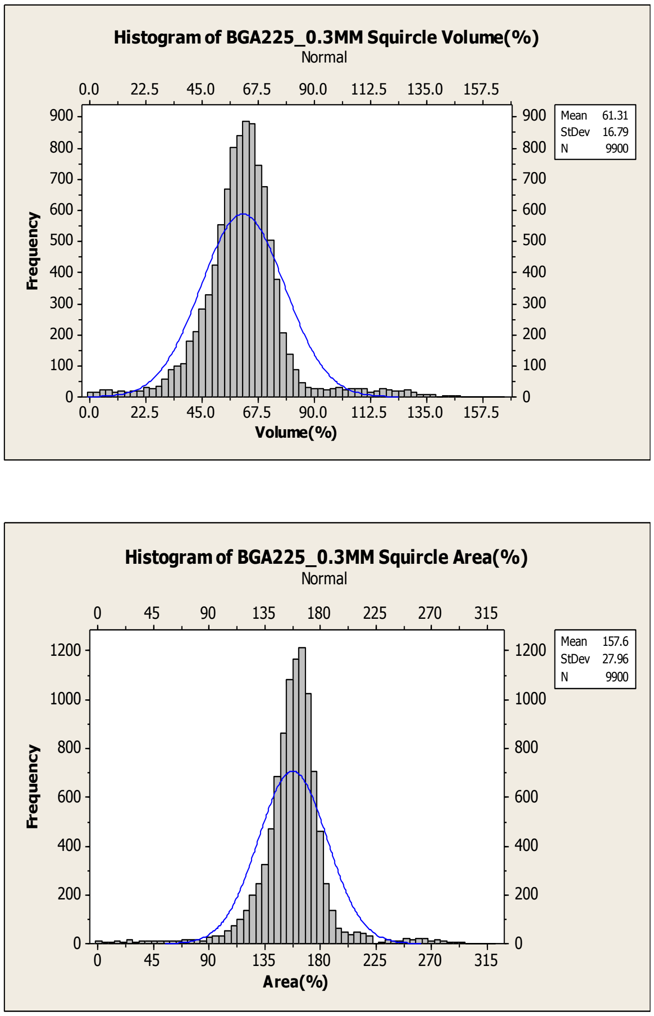
- Pad Size: 0.012”W X 0.018”L
- Area ratio: 0.9



Figures 5a, b: 0201 Square with Rounded Corners: The goal with rounded corners is to reduce volume to avoid tombstone defects and eliminate the possibility of clogged apertures.

- Pad Size: 0.012”W X 0.018”L
- Area ratio: 0.9



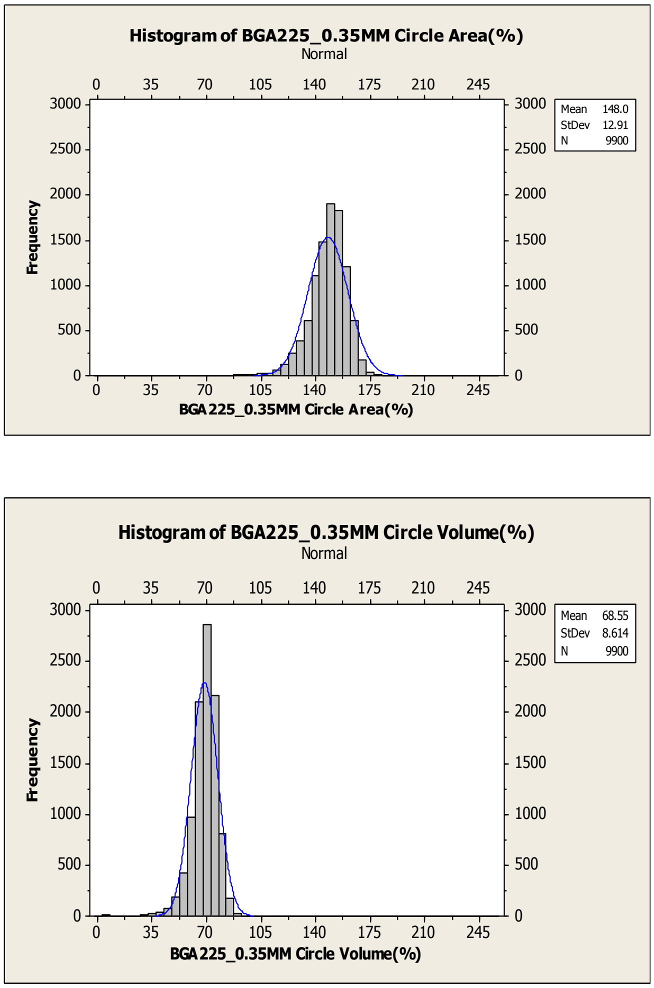


Figures 6a, b: BGA 225, 0.3mm Pitch with Circle Apertures:

- Pad Size: 0.006" Dia.
- Area ratio: 0.375

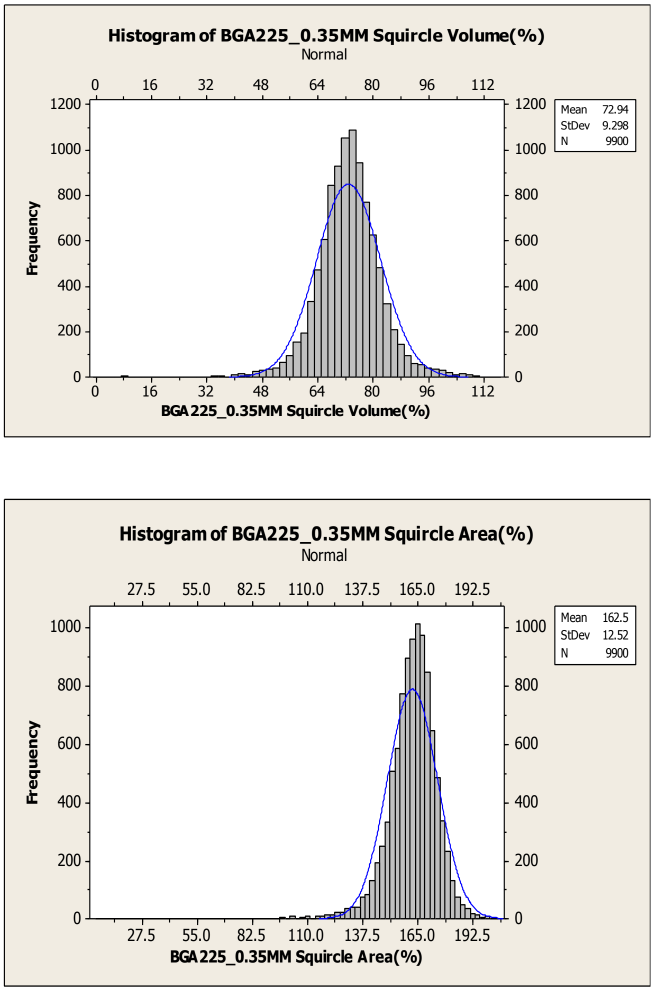
Figures 7a, b: BGA 225, 0.3mm Pitch with Square/Rounded Corner Apertures: Volume is a little higher but the process is more repeatable to hit the average (mean).

- Pad Size: 0.006" Sq.
- Area ratio: 0.375



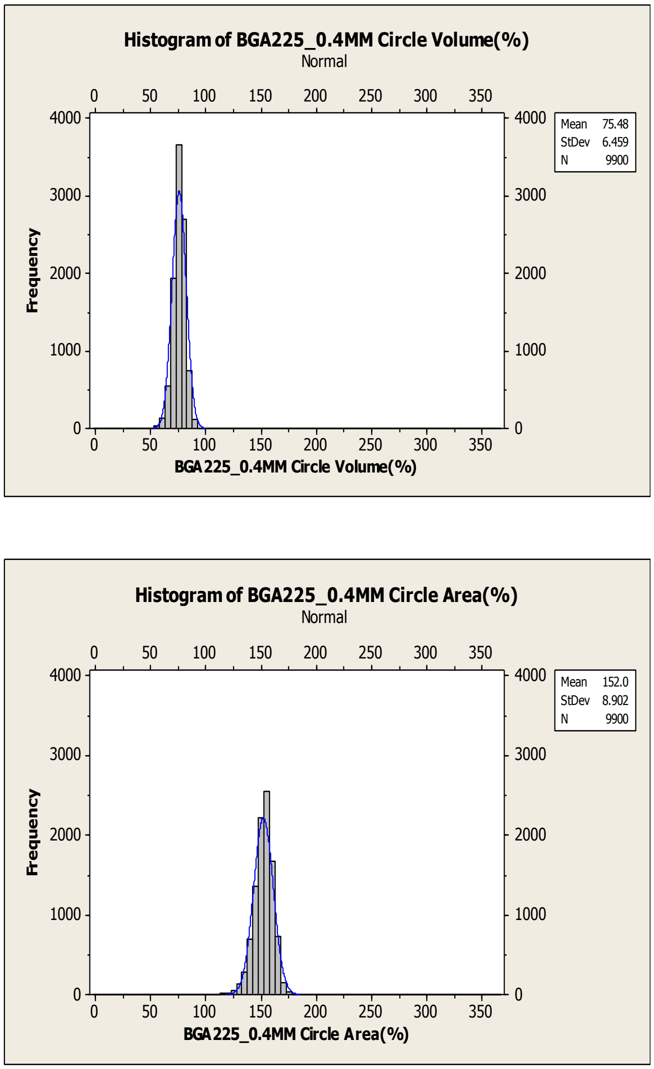
Figures 8a, b: BGA 225, 0.35mm Pitch with Circle Apertures:

- Pad Size: 0.007” Dia.
- Area ratio: 0.4375



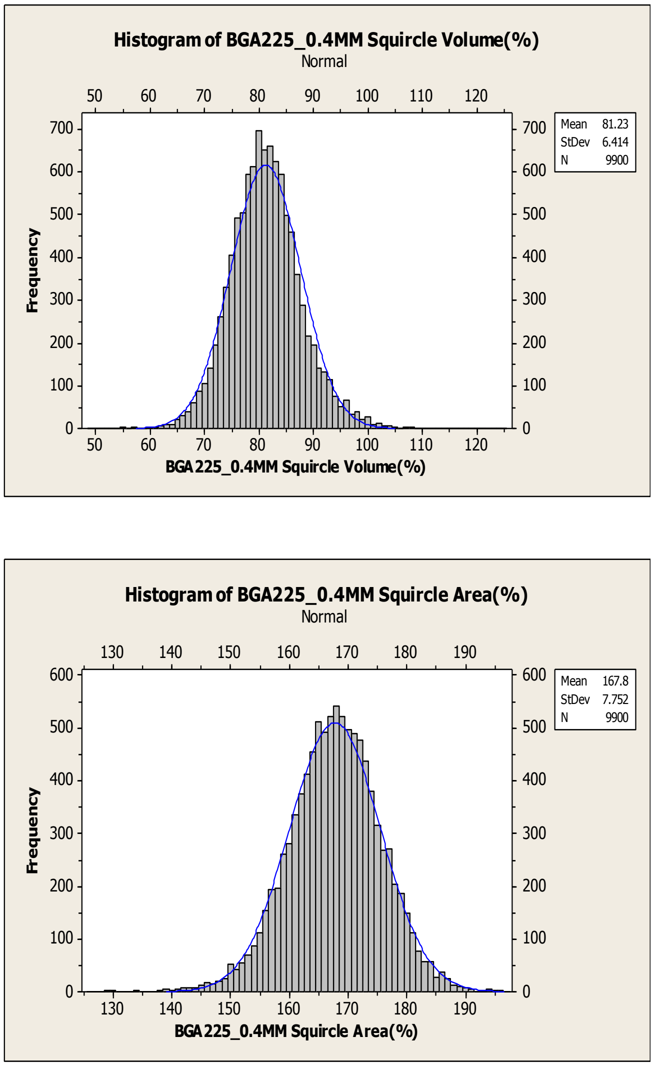
Figures 9a, b: BGA 225, 0.35mm Pitch with Square/Rounded Corners Apertures: Higher volume % due to better print registration with ‘squircle’ apertures.

- Pad Size: 0.007” Sq.
- Area ratio: 0.4375



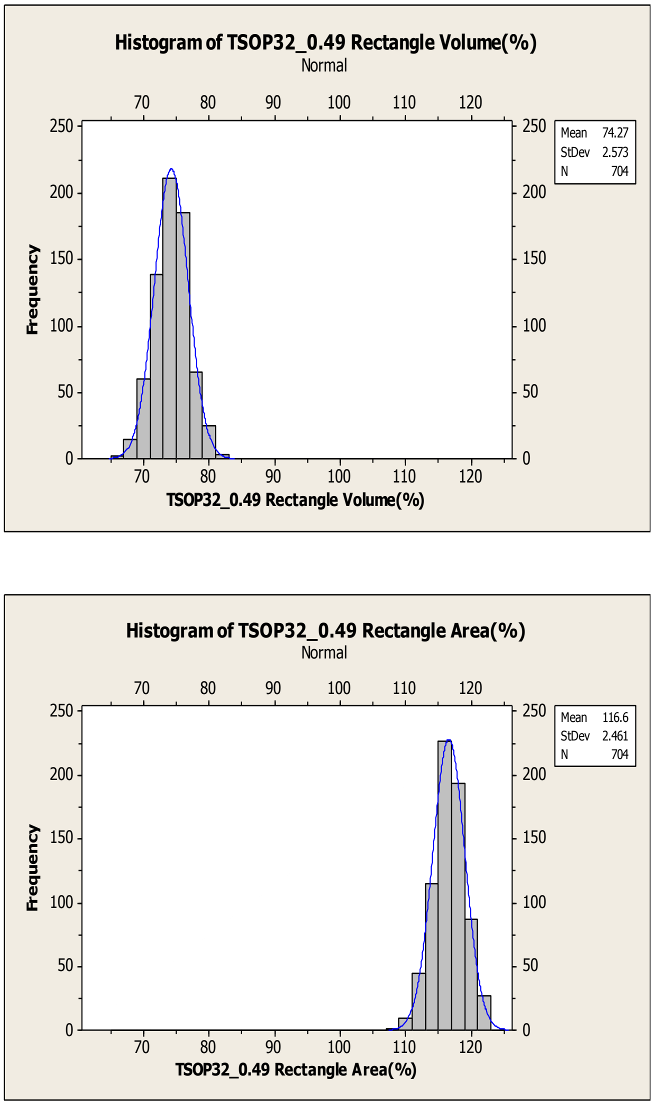
Figures 10a, b: BGA 225, 0.40mm Pitch with Circle Apertures:

- Pad Size: 0.008”Dia.
- Area ratio: 0.5



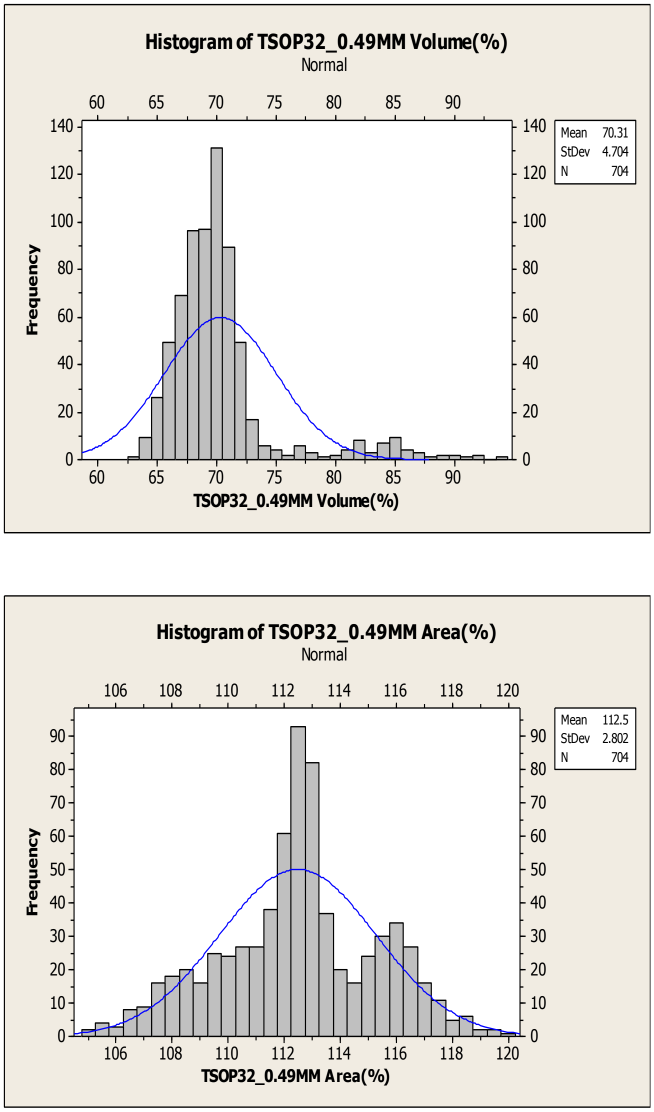
Figures 11a, b: BGA 225, 0.40mm Pitch with Square/Rounded Corners Apertures: Higher Average Volume % and better standard deviation = More repeatable prints.

- Pad Size: 0.008” Sq.
- Area ratio: 0.5



Figures 12a, b: TSOP 32, 0.5MM Pitch with Rectangle Apertures:

- Pad Size: 0.060"W 0.012"L
- Area ratio: 1.25



Figures 13a, b: TSOP 32, 0.5MM Pitch with Oblong Apertures: The goal with oblong apertures is to reduce the potential for shorts; hence the smaller average volume and area coverage.

- Pad Size: 0.060"W 0.012"L
- Area ratio: 1.25