

Stencil Design Considerations to Improve Drop Test Performance

Jeff Schake
DEK USA, inc.
Rolling Meadows, IL

Brian Roggeman
Universal Instruments Corp.
Conklin, NY

Abstract

Future handheld electronic products will be slimmer than today and deliver more functions, enabled by innovative electronics packaging design using smaller components with greater I/Os assembled in higher density. As solder interconnects between component and circuit board shorten, they also become weaker. This causes us greater concern on the survivability of such delicate electronic interconnects under normal handling impacts and serves motivation for formal study. This investigation will evaluate the influence of stencil printed solder volume on CVBGA97 electronic component lifetime in mechanical stress testing. A stencil aperture design to print a lower limit of solder paste volume has been thoroughly characterized as the first step towards determining the range of print volumes exhibiting the greatest influence on drop, bend, and die shear test performance. In this printing focused piece of work, print volume measurements were found varied across different circuit board pad designs with no change in aperture size. Highest paste volume transfer consistently occurred with solder mask defined pads. Stencil aperture and circuit board pad design variables are discussed in detail.

Introduction

Consumers continually drive electronics packaging engineers to design for portability, higher functionality, and smaller form factor, all while maintaining low manufacturing cost. It's a trend that may never cease, resulting in growing component diversity and assembly complexity. To the contract assembler, this translates to managing smaller component geometries that are mounted to the board in higher densities, which generally requires more accurate processes with tighter tolerances to satisfy end of line yield targets. Mobile electronic applications must also survive harsh handling service conditions, including accidental impacts caused by product mishandling. Progress towards increasing product mobility by shrinking component dimensions results in smaller solder joint sizes and lower standoff height. The expected consequence of this is weaker solder bonds and increased sensitivity to handling failure for the typical Pb-free solder interconnect assemblies in manufacture today. One method to combat this, for example, is implementing a corner and/or edge bond underfill process. However, there may be scope to improve mechanical test performance without the need to underfill by incorporating simple assembly process design modifications in the stencil print process.

The ultimate objective of this 2-part study is to characterize the mechanical attachment strength of assembled CVBGA97 0.4mm solder bump pitch and LGA equivalent packages and to compare failure trends across experimentally controlled solder joint volume changes. Various stencil design strategies are implemented to determine assembly yield and mechanical performance process window. Printing larger solder volume at more drop test sensitive failure prone corner pad locations is also planned for evaluation. This paper addresses the Phase 1 piece addressing the qualification of a stencil print process, involving stencil design strategy, circuit board quality, and influence of these on print deposit volume trends. The large matrix of statistically valid print performance data collected in Phase 1 is evaluated in order to make more meaningful and efficient comparisons in the next phase of planned work. The follow up Phase 2 piece will feature additional mechanical testing that includes drop, bend, and die shear, with performance correlation to printing.

Stencil Design Strategy

When printing into an aperture, solder paste wets two surfaces, namely the aperture wall and the circuit board pad. An important stencil design parameter, known as area ratio, is calculated based on this comparison. It is widely accepted that when the proportion of aperture opening area to aperture wall surface area exceeds 0.66, this area ratio should be capable of printing at least 75% transfer efficiency.[1] This means that less than 25% of the solder paste printed into the aperture will remain there after completion of the print cycle. Larger area ratio values should increase print transfer efficiency, as well as to improve solder paste deposit uniformity. **Figure 1** explains the area ratio concept visually.

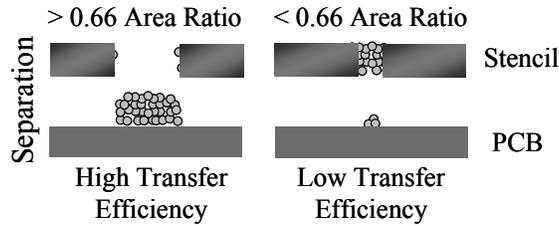


Figure 1 - Area Ratio Concept

Compliance to the minimum 0.66 area ratio rule represents one of three stencil design criteria. Designing enough free space between adjacent apertures is the second factor to account for, since it is important to ensure printed solder deposits do not bridge together. Bridging should be expected if the designed inter-aperture spacing is reduced to approach the stencil thickness value. It is suggested the minimum spacing between apertures should be maintained at least $15\mu\text{m}$ larger than the stencil thickness dimension, based on the authors' experience. Solder paste material, circuit board pad design, printer process parameters, tooling support, and stencil cleanliness can greatly influence capability of stencil printing finely spaced apertures. The third factor to consider in stencil design is the solder paste volume requirement for the specific component type. Compliance to only area ratio and aperture spacing guidelines does not guarantee the correct solder quantity to achieve a proper solder joint shape. A well designed stencil should satisfy these three requirements, however, when this cannot be achieved the tradeoffs should be weighed and allowance for design rule violation only permitted when identified to have the least impact on yield.

The CVBGA97 component used in this study would most likely be matched to a $100\mu\text{m}$ thick stencil when assembled in a production process. However, in our case a $76.2\mu\text{m}$ thick stencil was selected in order to purposely reduce the amount of solder paste volume printed per pad for accomplishing an accelerated mechanical test failure rate. The thinner stencil also allows for a more uniformly distributed small volume paste deposit compared to what is expected from a $100\mu\text{m}$ thick stencil. This is important as we seek to minimize paste deposit variation as a noise variable. Logically, the probability of small print volume solder joints surviving multiple mechanical stress test cycles is low, while larger printed volume solder joints may better tolerate this. A theoretical drop cycle performance curve is plotted in **Figure 2**. The portion of curve where its slope is steepest represents the range of printed paste volume having the greatest sensitivity to drop test performance. The point where the curve begins to flatten, i.e. "knee", defines a threshold solder paste deposit volume corresponding to a minimum value that achieves maximum drop cycle performance. Bend and die shear testing results can be substituted for drop cycle failure to show similar comparisons with correlation to print deposit volume, scheduled to commence in the Phase 2 experiment. This information should steer our industry towards being more capable to design stencils that ensure the assembled part meets or exceeds a required level of mechanical test performance.

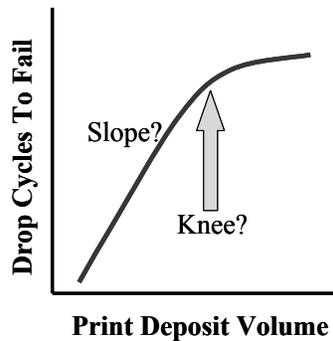


Figure 2 - Drop Performance Sensitivity to Print Volume

A list of candidate aperture sizes using a $76.2\mu\text{m}$ thickness stencil is shown in **Table 1**. The area ratio and spacing between apertures are color coded to simplify interpretation against the first two stencil design criteria described earlier. Red shaded boxes indicate severe stencil design rule violation where print quality is predicted to be poor. Amber represents a design warning where print quality may be on the border of passing or failing. Our initial plan was to select aperture sizes predicted to be on the verge of failing (i.e. amber) and design a stencil to evaluate those in order to establish the boundaries of a print process window. The arrows in the table indicate the aperture dimensions designed on our first test stencil. Phase 2 testing will consider a range of aperture sizes inside these limits.

Table 1 - Aperture Dimensions

| Aperture Size | Area Ratio | Inter-Aperture Spacing |
|---------------|------------|------------------------|
| 150µm | 0.49 | 250µm |
| 175µm | 0.57 | 225µm |
| 200µm | 0.66 | 200µm |
| 225µm | 0.74 | 175µm |
| 250µm | 0.82 | 150µm |
| 275µm | 0.90 | 125µm |
| 300µm | 0.98 | 100µm |
| 325µm | 1.07 | 75µm |
| 350µm | 1.15 | 50µm |
| 375µm | 1.23 | 25µm |
| 400µm | 1.31 | 0µm |

Both square and circle aperture shapes were designed on this laser cut stainless steel stencil. Measurements were performed using a manual coordinate measuring machine to verify aperture size accuracy against gerber design dimensions, shown in **Table 2**. Note the large 325µm square aperture was not measured because that design was not printed due to predicted print quality concerns. More on that will be explained later. The measurements indicate good aperture size accuracy, and show a very slightly tapered wall with bottom side measurements exceeding top side dimensions. This tendency is typical of any laser cut stencil since the laser cuts the stencil by entering the bottom side foil first, and then exiting through the top side. The laser entrance intensity is slightly higher than the laser exit intensity as it loses some energy through the stencil foil. This loss of laser intensity contributes to aperture wall taper.

Table 2 - Stencil Aperture Measurements

| Shape | Gerber Size | Top Meas. | Bot. Meas. | Avg. Meas. |
|--------|-------------|-----------|------------|------------|
| Circle | 175µm | 172µm | 180µm | 176µm |
| Circle | 325µm | 323µm | 328µm | 325.5µm |
| Square | 175µm | 168µm | 182.5µm | 175.25µm |

Circuit Board Design

The test circuit board design is the small form factor that is typically used in drop testing consisting of a simple double sided 2 layer FR4 construction with organic solderability preservative coated copper pads. Further details are listed in **Table 3**. A photo of the circuit board is shown in **Figure 3** with red outlines identifying the locations of the 15 CVBGA components to be evaluated. The circuit board pads for these components are designed as both non solder mask defined (NSMD) and solder mask defined (SMD). Exposed solderable pad diameter is designed to be either 203.2µm or 250µm.

Table 3 - Circuit Board & Component Specifications

| | |
|-------------------------|--------------------|
| PCB Size | 132mm x 77mm x 1mm |
| Construction | FR4, 2 Layer |
| Metallization | Cu/OSP |
| Pad Design | SMD, NSMD |
| Component Type | 97 I/O CVBGA |
| Component Size | 5mm Square |
| Pad Pitch | 0.4mm |
| Solder Ball Dia. | LGA, 250µm |

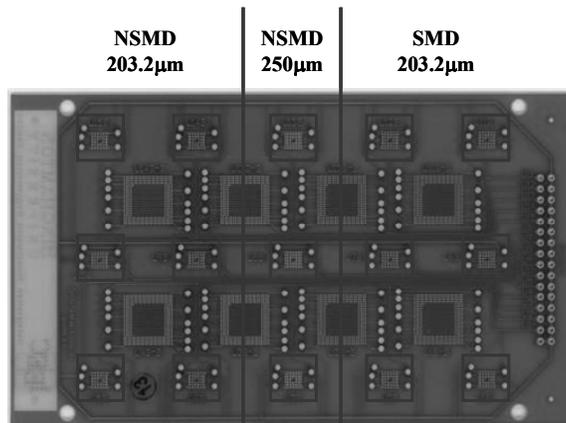


Figure 3 - Drop Test Board

As was done with the stencil, the circuit board pads were measured to verify dimensional accuracy against the gerber design values. The results listed in **Table 4** suggest copper features were over-etched and solder mask openings oversized. From a stencil printing perspective the consequence of this size error is considered more severe for the non-solder mask defined pads where the stencil aperture has less pad to effectively gasket onto, and may impair paste transfer efficiency.

Table 4 - Pad Measurements

| Pad Definition | Gerber Size | Measured Size |
|----------------|---------------|---------------------------|
| NSMD | 203.2 μ m | 178 μ m (-25 μ m) |
| SMD | 203.2 μ m | 213 μ m (+10 μ m) |
| NSMD | 250 μ m | 225 μ m (-25 μ m) |

Another finding upon inspection of our boards was the level of obvious solder mask mis-registration occurring on some of them. **Figure 4** shows the two left side photos of pads exhibiting good solder mask registration, while the two photos on the right exhibit the solder mask openings being shifted in “x” and “y” by 50 μ m respectively. This level of solder mask layer shift is actually not uncommon for a typical PCB manufacturing process, where layer to layer tolerances can be 2x the amount shown here. These particular photographed pads have been printed with solder paste and cleaned off several times before, which explains the occurrence of satellite solder ball particles getting trapped between pad and solder mask opening. Also included in the photos are red colored dashed circles which represent properly scaled 175 μ m diameter solder paste deposit outlines. The position of the red circle is either “aligned to metal” or “aligned to solder mask”, and this is done to explain how solder mask mis-registration can impact printed solder paste alignment to pad. When the stencil printing machine is programmed to use metal defined fiducials (NSMD) on the circuit board, then the expected solder paste deposit location is “aligned to metal”. If solder mask defined fiducials (SMD) are used instead, then the solder paste should follow the deposit outlines shown for “aligned to solder mask”. The point raised here is the significance of circuit board fiducial design, as NSMD and SMD options can have profoundly different effects on printed paste alignment results for circuit boards exhibiting poor solder mask registration to pads.

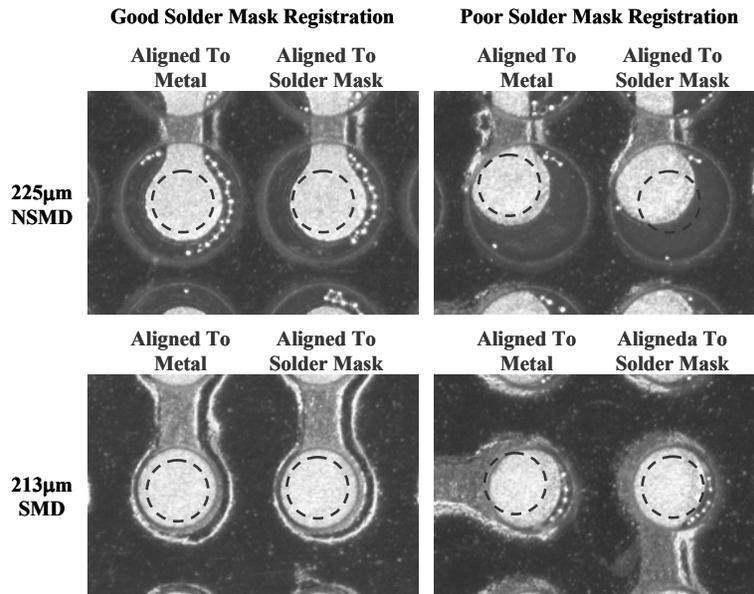


Figure 4 - Solder Mask Registration Tolerance Differences

Solder Paste Printing

The stencil print process evaluation consisted of printing only one circuit board (several times over and over again) and using a dedicated 3D solder paste inspection system to measure individual paste deposits. The board chosen for print study displayed good solder mask registration tolerance. By only using one circuit board, this eliminates including board to board differences that may contribute to increasing the variation in print deposit volume. Ten consecutive on-contact prints were performed using 175 μ m and 325 μ m diameter circular apertures as well as 175 μ m size square apertures. The stencil was not manually or automatically cleaned between any of the ten prints, however, the stencil was fully cleaned between sets of ten prints when transitioning to a different aperture size. The 3D paste inspection system measured all solder deposits on prints number 3 and 9, with both corresponding to printing the board from the front to the rear of the machine. A statistically adequate sample of paste deposits can be produced from a single print, so it was decided to analyze data from two prints to

ensure a measure of print deposit volume repeatability between several prints was established. Print 3 was chosen for measurement because it represents the first print using a fully wetted squeegee with lubricated apertures, and print 9 was selected because it uses the same squeegee several prints later and eliminates the possibility of print direction affecting results. Further details about the printing machine setup are listed in **Table 5**.

Table 5 - Printing Machine Parameters

| | |
|-------------------------|-----------------------------------|
| Printer | Fully Automatic, High Performance |
| Squeegee | 170mm, 45° Metal Blade |
| Tooling | Dedicated Vacuum, Foiless Clamps |
| Print Speed | 25mm/sec |
| Print Pressure | 5kg |
| Separation Speed | 1mm/sec |
| Stencil | Laser Cut, 76.2µm thick |
| Solder Paste | SAC305, Type 4, No Clean |

A collection of print photos is shown in **Figure 5**, organized in rows by aperture type and in columns by circuit board pad type. All photos are taken of the ninth consecutive print. Even without the 3D inspection data to reference, we can make several unmistakable conclusions. The most obvious is the tendency for solder paste deposits to bridge heavily using the 325µm diameter circular apertures. Bridging was a concern noted from review of stencil design rules in **Table 1**, but the resulting defect severity shown was far worse than what we predicted. For this circuit board the 325µm aperture size is clearly too large. An additional 10µm to 15µm spacing between adjacent apertures would certainly reduce or even completely eliminate solder paste bridging, and this will be further tested and reported in Phase 2. The 325µm size square aperture was not attempted to print as bridging would likely be worse compared to the equivalent size circle aperture since the challenging inter-aperture space between square geometry apertures follows a longer boundary. Another observation that leaps out is the surprising effect, perhaps, that pad type has on resulting paste volume. Keep in mind that all pad types are printed with the same aperture size, yet it appears the SMD pads yield significantly larger print deposits. The size of the pad is also noted to have an influence on printed volume for the two sets of NSMD pads, with the larger pad exhibiting more solder volume transfer. It is difficult to judge differences in paste deposit volume attributed to changes in aperture geometry as well as to compare differences in deposit uniformity using these 2D views. For these more difficult to examine comparisons, the 3D paste inspection tool is very well suited to characterize.

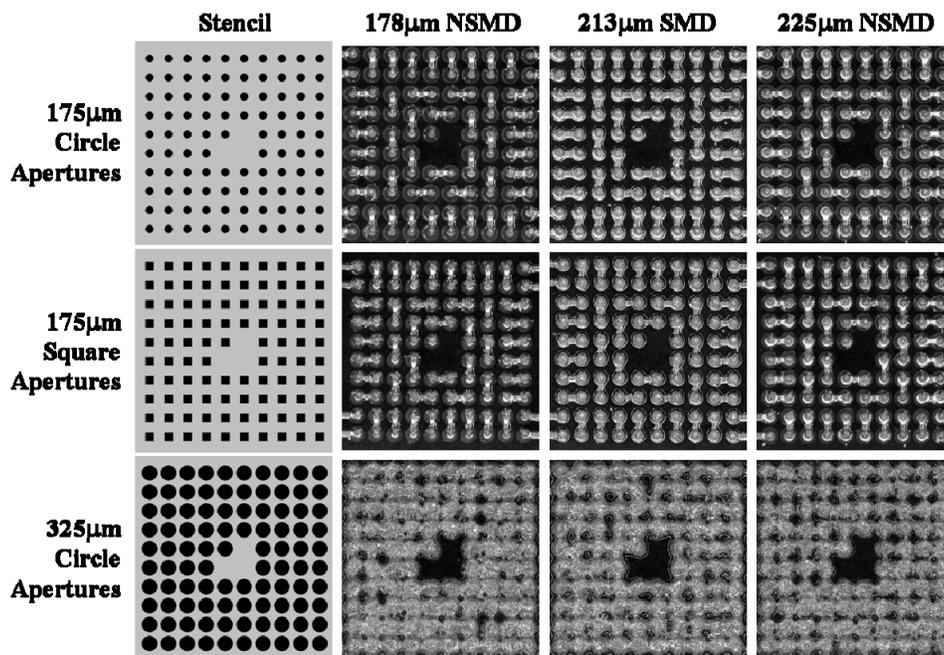


Figure 5 - Print 9 Photos

Figure 6 and **Figure 7** summarize the quantified measured paste volume results for 175µm size circle and square apertures respectively, comparing the three pad types in print 3 and print 9. The bar chart plots support the print photos showing a higher solder print volume transfer on the SMD pads for both prints using both circle and square apertures. Also, pad size does appear to influence average paste transfer, at least for the NSMD pads compared here, where the larger pad yields

higher print volume. Further analysis of the print behavior on the NSMD pads reveals the tendency for the larger pad to have better paste volume repeatability, identified by the lower values of standard deviation data occurring. The paste volume difference between printing circle and square apertures is consistent with theory that squares should deliver higher values. The repeatability of the print generally appears more sensitive to pad type for the square geometry and is optimum with the SMD pad. The differences between print 3 and print 9 are subtle, with print 9 giving slightly more paste volume transfer and marginally reduced repeatability as we would anticipate. The consistency of trending shown suggests our print process is in control.

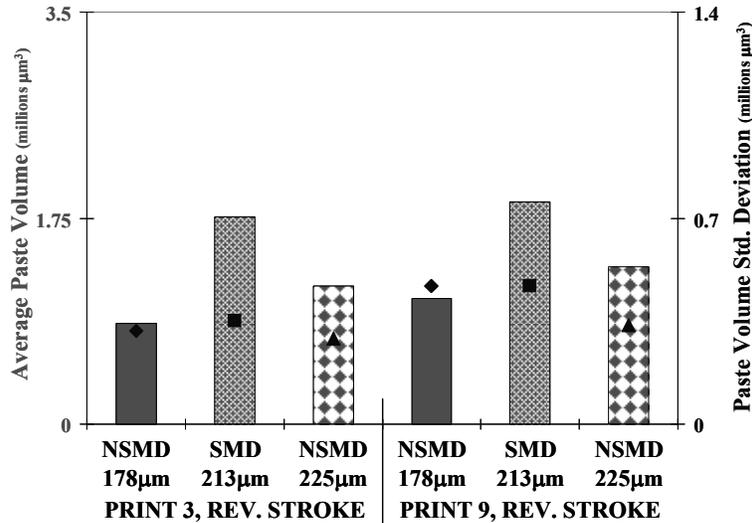


Figure 6 - 175µm Circle Aperture Trends

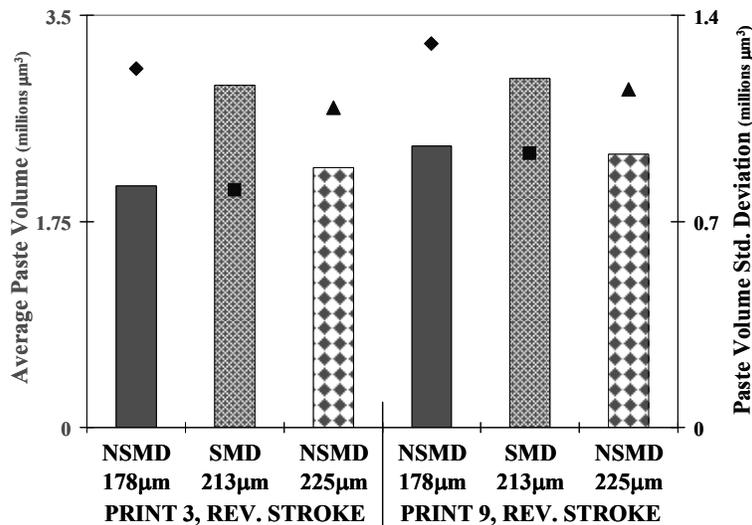


Figure 7 - 175µm Square Aperture Trends

Discussion

Prediction of stencil print volume typically considers the stencil design exclusively and is calculated by subtracting a paste transfer loss factor (attributed to area ratio) from the capacity of the aperture. While this may be suitable to forecast results on blank boards, this experiment has demonstrated that circuit board pad dimensions can have profound influence on print deposit volume. Consider the comparison between the small 178µm NSMD pad and the 213µm SMD pad for circular apertures in **Figure 6**. The SMD pad exhibits twice the average paste volume with no change in aperture size! Of course, this represents an extreme example of paste volume sensitivity to pad design and it is possible this tendency may be diminished for less challenging aperture area ratios. We believe the reason for improved paste volume transfer on the SMD pad type is attributed to the larger amount of space available to be occupied by printed solder compared to the NSMD pads. This can be more easily understood when referenced against **Figure 8**, which shows cross-sections of the same aperture design aligned and placed in contact against the three different pad types. For the SMD pad shown in B the solder mask

covers the periphery of the pad, generating a standoff for the aperture to gasket against during the print and increasing the space available for solder to be printed. This bonus volume allows the SMD pad to print more paste. Examination of the NSMD pads in A and C shows the aperture gasketing directly to the pad itself in both cases. The smaller NSMD pad in A is close in dimension to the aperture, which may explain the tendency for this one to print lower volume with poorer repeatability performance. The schematic shows the aperture in A to be in perfect alignment to the pad, but in reality typical process tolerances will produce slightly offset alignment (i.e. **Figure 4**) that will jeopardize the integrity of the stencil to pad seal and cause more randomness in the resulting formed print deposit. The aperture seal against the larger NSMD pad in C allows a more generous alignment mismatch tolerance that is more likely to maintain a proper gasket and print more consistent print volume. Keep in mind the aperture size modeled in **Figure 8** is a small dimension and larger designs will be tested as well for drop cycle performance, and this will have consequences on the aperture to pad gasket. For example, by expanding the aperture size slightly for the small NSMD pad in A, the stencil will seal against the solder mask instead of pad, while for the larger NSMD pad in C the stencil will continue to contact the pad directly. It will be interesting to track changes in print volume and repeatability as aperture size is incremented larger over these pad types as we move forward with our testing.

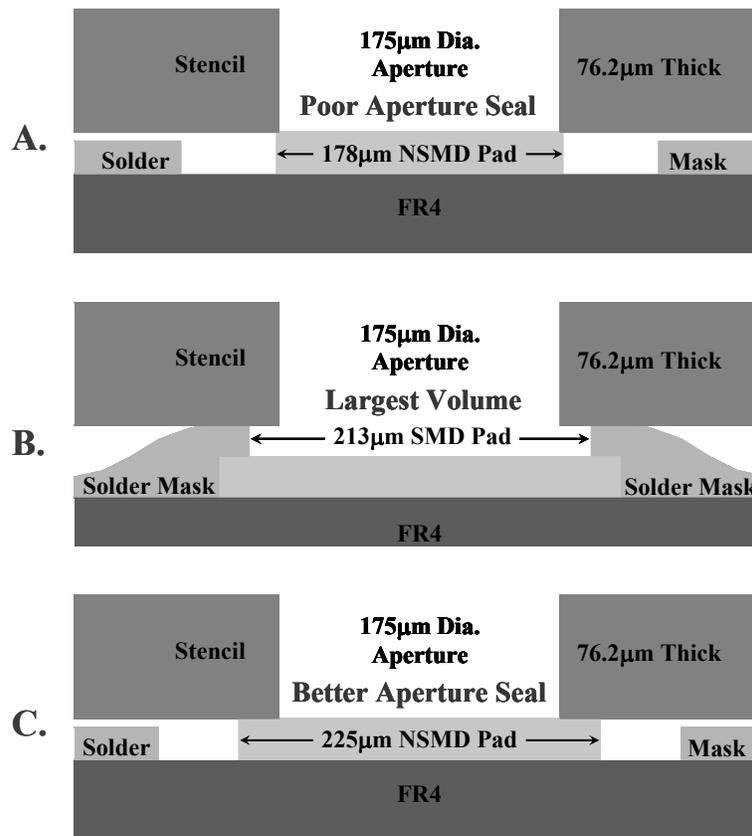


Figure 8 - Cross Sections of Aperture on Pad

Conclusion

This investigation has shown that stencil print volume and repeatability performance is not exclusively attached to compliance with aperture design rules, but is also strongly influenced by the circuit board pad design. It is incorrect to assume a common aperture size will print the same volume on different pad types. Furthermore, the manufacturing tolerances associated with the circuit board can also have consequences on print registration and aperture gasketing quality. Additional print qualification testing will commence using larger aperture dimensions to build up a profile of print volume and repeatability metrics that will ultimately be compared against results of drop, bend, and die shear testing of assembled CVBGA97 components.

References

[1] IPC-7525A “Stencil Design Guidelines”, February 2007.