STENCIL PRINTING 008004/0201 APERTURE COMPONENTS

Edward C. Nauss and Michael Butler
ITW EAE
Hopkinton, MA
enauss@itweae.com; mbutler@itweae.com

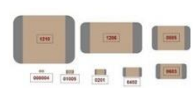
ABSTRACT:
This paper will focus on the application requirements of solder printing small aperture designs, concentrating on 008004 (inch) / 0201 (metric) size components, and the results of a design of experiment printing these challenging apertures. As Moore's law continues to be applied to component miniaturization, the next installment of reduced packaging has arrived in the form of the 008004/0201 for resistors and capacitors. Component size roughly the size of a grain of sand presents specific challenges to the solder printing process. To address these challenges, each aspect of the printing process will need be examined. This includes essential machine requirements, including correct squeegee blades, tooling support, and calibrations, to meet the demanding specifications. The correct match and design of materials will be addressed, focusing on the stencil and substrate design along with solder paste and cleaning solvent requirements. A design of experiment will be reviewed that applies the machine and materials discussed, including the printer and Solder Paste Inspection (SPI) setup and the specific machine parameters used. The results of these DOE's will then be closely examined.

Key words: printing, miniaturization, component, stencil, solder paste, 008004, 0201mm

BACKGROUND:
Moore's Law states that the number of transistors a microchip can contain is expected to double every two years. The same trend can be applied to miniaturization of components. Demands for high functionality from mobile devices, smart watches, wearables, military, medical and audio technology continues to drive the development of component miniaturization. The SMT industry has seen the introduction of the next generation of component miniaturization every 4-7 years with the latest release of the 008004" (0201mm) component size. With a 6% reduction of area and a 1.6% reduction in volume when compared to a 01005" (0402mm) package size, the equipment manufacturers have quickly adapted to this next challenge. However, when introducing micro devices to a manufacturing process, the problematic hurdle that needs to be addressed is the printing process. Focus on the materials, machine and process that ties them together, needs to be addressed in order to successfully introduce 008004" (0201mm) components into a manufacturing process.

Developing a process that can produce a product containing 008004" (0201mm) with a Cp value that is twice a capable can be achieved when each element of the process is working together. This will require the engineer to reevaluate their present process to adapt to the challenges this component will require in order to successfully implement a 008004" (0201mm) component. This may go as far as implementing a clean room environment. The focus of this paper will be on the individual elements of the printing process and the results of testing performed at the ITW EAE process lab in Hopkinton, Massachusetts.

Status of components today:
Referencing **(Figure 1)** most of the market has become comfortable with products that contain components down to 0402" (1005mm). Recently 0201" (0603mm) components are becoming more common in mass production with the 01005" (0402mm) just now entering the mainstream. With each step, we have adapted stencil designs and materials to implement these components into our designs with little or no consideration to the other key elements. Printing 008004" (0201mm) requires us to examine each element of the process prior to fabricating our first substrate to insure each element is designed with this component challenge in mind. The solution to micro component printing is in the aggregate, where each element is using best practices for optimum results. Any deviation or ignoring any element will result in a less than repeatable process.



Millimeter (mm)			Inch (in)		
Component	Length (mm)	Width (mm)	Component	Length (in)	Width (in)
3216	3.2	1.6	1206	0.125	0.063
2012	2.0	1.25	0805	0.079	0.049
1608	1.6	0.8	0603	0.063	0.031
1005	1.0	0.5	0402	0.039	0.020
0603	0.6	0.3	0201	0.024	0.012
0402	0.4	0.2	01005	0.016	0.0079
03015	0.3	0.15	XXXXX*	0.012	0.0059
0201	0.25	0.125	008004	0.0098	0.00049

* : 03015 only offered in resistive package– no Capacitors

Figure 1: Component sizes

Printed Circuit Boards (PCB):

As Process Engineers, most PCB designs are created long before we become involved. Design for manufacturing considerations are most often out weighted by cost requirements. We are often faced with having to design the process to the board rather than the other way around where adjustments and compromises are made to accommodate shortfalls in the PCB design. This is where the Process Engineer needs to take an active role to make sure that success is designed into the PCB to insure repeatable results. So far, applications have been focused on micro components with little to no mixed technology on multi-up panels. The problems with mixed technology, also referred to as the broadband printing issue, is not the focus of this paper though this would need to be addressed before proceeding. Most applications to date have been using PCB thicknesses of 0.030” (0.762mm) or less, so this should be expected. Based on the PCB thickness and the amount of routing, special carriers maybe needed to transfer the product from machine to machine as well as stabilize it during the reflow process. One of the key elements to successful printing which the PCB plays a major role is the capability to form a seal between the stencil and the PCB commonly referred to as gasketing. Gasketing in turn plays directly into the process to transfer the paste from the stencil aperture to the PCB pad efficiently and repeatably. Maintaining a consistent board thickness, by elimination of variance in mask thicknesses from PCB to PCB, is key as this determines the position of the PCB to the bottom stencil surface. Mask encroachment on to the pad surface should be eliminated as this will greatly affect the capability to seal the aperture. Nomenclature and silk screens should be avoided in the PCB design. Issues arise when silkscreen is equal to or greater than the stencil thickness being applied along with being located adjacent to the component. Logos and identification information should be presented on the back of the PCB or located in consideration to cause the least effect on the critical components. The use of barcode labels should be avoided as this is the source of many printing issues with less challenging components. Barcode labels should be applied after the print process is completed if possible. One of the critical mistakes when designing PCBs for micro component printing is to make sure the pad is not significantly below the mask height. When designing a board with a bare copper pad, the pad should be just below or equal to the mask height. If the pad is below the mask, this creates a gap that the paste must now overcome when it is printed into the aperture. Because the paste must flow between the aperture opening on the bottom of the stencil to the pad surface, the paste will be unable to secure a proper adhesion to pad surface. The result is erratic print results that is represented in opens and insufficient volume or pad coverage. In some cases, the PCBs with this issue become non-manufacturable due to erratic results from the printer. To insure the pad is positioned correctly to the mask height and has a flat planar surface, an electroless nickel immersion gold (ENIG) plating should be considered. ENIG plating, consisting of an electroless nickel plating covered with a thin layer of immersion gold, protects the nickel from oxidation has shown to have the best results. In the board design, the decision to use mask defined pads often present issues. Accuracy of the pad locations as well as pads that are sized larger than the specification have been issues when using mask defined pads. Location of the fiducials, especially when implementing a multi-up panel, is critical for the machine vision systems to properly perform alignment. The fiducials should be part of the artwork and be present on the PCB image for best results. Avoid locating the fiducials on the breakout panel as this tends to add to any alignment error. Stretch and step and repeat errors should be avoided as we are dealing with a recommended pad size of 0.005” X 0.006” (0.127 x 0.1524mm) where as much as a 0.001” (0.0254mm) error can have significant consequences. Investment up front in the PCB design and manufacturing will insure success where some of the previously described issues are often difficult if not impossible to overcome.

Squeegee Blades/Enclosed Heads:

A best practice is to separate a set of squeegee blades specifically used for micro-component printing. This insures that the blades being used are undamaged and not worn. The squeegee requirements for micro-component printing is simple, spring steel blades with a squared edge is all that is required. A blade angle of 55 degrees is also recommended where standard blade angles are set to 60 degrees. This change in angle allows more surface area of the blade over the aperture to promote an improved aperture fill. It also improves sheering off the paste at this angle when the blade passes over the aperture to prevent paste drag out and erratic aperture fill. Blade length should match the PCB as closely as possible with a maximum size range within 2 inches of the PCB size in X direction. This will center the squeegee pressure on the PCB as well as prevent long term damage to the stencil. Inspect the blades every time prior to use for cleanliness and for damage. It should be noted that enclosed heads have been used for fine featured printing applications with great success in the market today. The extrusion flow from the pressurized chambers are compatible for repeatable aperture fill for micro-component printing. Some Type 6 pastes have a limited stencil life where enclosed chambers address this issue and minimizes paste waste. For this test we focused on squeegee blades as this represents most of the process applications on the market.

Stencils:

For this experiment we used a 29” x 29” (736 x 736mm) fine grain, laser -cut, Nano-coated, 0.002” (0.0508mm) thick stencil. The aperture size is a square, 0.005” x 0.006” (0.127 x 0.1524mm) that is one to one with no reductions or variations in shape. Based on previous experiments we determined that the 0.002” thick stencil had the best transfer efficiency. When specifying a stencil thickness more often we take in consideration the two ends of the spectrum for paste requirements and find a compromise in-between. Most applications so far using micro-component printing, have had compatible component mixes where there was not a significant difference in requirements. The frame size we used was 29” x 29” (736 x 736mm), however, 23” x 23” (584 x 584mm) stencil may be better suited based on common average board size for 008004” (0201mm) applications and stencil tensioning requirements outlined below. It is recommended using a fine grain stainless steel stencil that is laser cut. Electroform stencils have fallen out of favor with reported issues such as variation on aperture size and foil thickness and stretch being introduced to the image. Recommended for this application is to use high tensioned foils. Stencils have a range for tensioning normally 28 – 40N/cm² (Newton/centimeter). Most stencil tension falls into the lower 30-Newton range. Increasing the tension into the upper 30-Newton range prevents stencil drag. Stencil drag is when you are using a thin stencil foil with a significant amount of aperture openings. The surface tension of the paste that has now adhered to the PCB, pulls at the stencil foil during release, resulting in lower paste transfer efficiency. The higher tension results in a cleaner more balanced release with no transfer issues. Nano-coating is recommended as studies have proven it improves transfer efficiency. Stencil manufactures have improved the application methods for applying Nano-coating to stencils that has improved its manufacturing life. However, aggressive fluxes and repeated aggressive wipes will eventually wear the coating off. Careful handling of thin stencils should be emphasized as they are easily damage. Special care should be used when storing and transferring to and from the machine. Take care when handling blades over the stencil in the machine as a dropped blade could ruin a stencil quickly. Cleaning the stencil using ultrasonic methods after printing is essential for continued stencil life. Type 6 paste is difficult to clean and can become difficult if not impossible to remove if not removed promptly after use.

Solder Paste:

The recommended solder paste for this aperture size is a Type 6 powder size. The specification for Type 6 is a mesh size of +635 mesh size with the ball size range of less than 20µ with an average of 10µ. Type 4 paste is the prevailing powder size presently being used in the SMT market. Significant improvements in powder size yields have eased pricing for Type 4 and Type 5. However, Type 6 pricing has remained constant where comparative pricing can be three times the cost of the Type 4, they are presently using. Squeegee speeds and release parameters are dictated by the paste formulation and flux type. From the printing prospective, Type 6 prints like any other paste, however considerations of the requirements down steam need also be considered. Matching the paste to the Pick and Place as well as the requirement for using nitrogen during reflow must be also considered when using Type 6 paste. As best practice for a 0.005” x 0.006” (0.127 x 0.1524mm) aperture is a Type 6 paste – experiments going forward need to be performed to see if a hybrid Type 5.5 powder size or a Type 5 can be substituted for a Type 6.

PCB Support:

The consensus in printing is that tooling support is essential to successful, repeatable print results. The aluminum tooling plate is still the touchstone that all other forms of support are tested against. Since most applications for micro-component printing use PCBs 0.030” (0.762mm) or less, the tooling in combination with vacuum assist to insure the PCB is flat, level and supported will give the best results. The plate should be designed so the PCB fits in a recessed pocket with the PCB surface positioned above the tooling surface. Support wings are also recommended to support the squeegee outside of the print area to prevent long term damage to the stencil. Recommended is a Venturi vacuum system as the Hg (inches of mercury) produced by standard vacuum systems may not be enough to flatten the PCB. When implementing vacuum openings on the plate, take in consideration the thickness of the PCB relative to the hole size to prevent deflection or “dimpling” of the PCB surface. Special attention needs to be focused on the leveling of the bottom of the tooling plate fixture. This will be reflected on how well the PCB gaskets to the stencil. Addressing how to hold the PCB in place during the print process, vacuum is the preferred method to insure a flat surface over top or side clamping for PCBs thickness below 0.030” (0.508mm).

Wiping:

Wiping is the first defense against defects and can have both a positive and negative impact on the process. Determining the correct frequency, wipe sequence, compatible chemistries and materials will impact repeatability and eliminate potential defects. Micro sized apertures require more frequent wiping where a simple experiment can determine the starting point, however, over-wiping with solvent can have the same negative effect as under-wiping. The test involves printing a board and then drive the vision camera under the stencil to inspect the apertures for any paste squeeze out or clogged apertures. Note, that the apertures will contain some paste that was not released based on transfer efficiency and stencil quality. In most cases this paste will be pushed out on the next print sequence and will not require a wipe, please judge accordingly. Continue this process of inspection until defects are starting to form. Subtract 1 board from the total and this can be your starting frequency. If a Solder Paste Inspection machine is available, then based on results, this can be used to determine the correct frequency of wipes.

The recommended sequence is a vacuum/vacuum/dry. The combined vacuum strokes eliminate any paste pulled from the aperture and left behind that was the result of the first pass vacuum. Solvent should be used less frequently as the purpose of this material is to address the flux that can build up around the aperture opening. Recommended frequency for a solvent wipe is every 4-6 wipe cycles. The recommended solvent stroke sequence is a solvent/vacuum/dry where a solvent application always begins the sequence. Consult your paste manufacturer for recommended solvents to ensure that the solvent used is compatible with the paste flux. A quality paper should be used as Type 6 paste can be difficult to clean, where economy paper can have issues with retaining the solder balls and contamination issues could result. [1]

The Printer:

The printer plays the major role in the success of printing 008004" (0201mm) components. It's recommended prior to printing micro apertures to make sure all preventive maintenance and calibrations are up to date. The alignment capability of the machine is vital for dealing with small pads. Advancements in machine vision repeatability and accuracy has kept pace with the introduction of micro-components. However, if your machine was designed back in the 1990's, then it most likely will not have the accuracy resolution to handle these component challenges. Testing the machines vision alignment capability prior to developing the process is recommended so that with the machine verified, issues with alignment can be isolated and solved more quickly. This can be done using a print verification process, using embedded machine software that measures paste deposits for accuracy and repeatability, the results will determine if a vision calibration is warranted. Another key calibration on the printer that is often overlooked is the table leveling to the stencil rails. Since gasketing is paramount when printing micro apertures, this calibration takes into consideration the four corners of the table as it applies to the stencil rails for proper seating between the PCB and stencil. This calibration is overlooked as it was most likely done when the machine was built by the manufacturer and never addressed again after installation. One of the issues with doing this calibration was the difficulty with the procedure used. A feeler gage is employed to measure the distance from the table to the bottom of the adjacent stencil rail. In order to measure the four points, the gage is moved from corner to corner repeatability to dial the distance to within specifications. This process requires the machine to be down significantly often taking hours to complete. A new tool developed by MPM addresses this issue by adjusting all four corners simultaneously. To date, specifications for table to stencil leveling has been in the range of +/- 0.004" (0.1016mm). However, studies have shown best results are achieved when the specification is dropped to +/- 0.001" (0.0254mm). To eliminate any tolerance issues between the table and the tooling plate, the plate can be used as a reference during this calibration. This specification can be achieved using this calibration tool and has played a contributing role in successfully printing micro apertures. The time to complete this calibration has been reduced to roughly 1 hour. Lastly, the printing machine should be completely inspected for cleanliness and clean any paste debris found. Root cause for many issues can be traced to random paste deposits or residual paste that builds up over time.

DESIGN OF EXPERIMENT:
Print Test for 008004 (0201mm)
Overview:

To demonstrate micro-component printing capability, with a focus on the 008004" (0201mm) component using the new SMTA miniaturization test vehicle. Using best practices described above, examine the results to determine Cp, CpK, Pp and PpK results. The goal is to achieve a process capability, Cp, that is equal to or greater than 2.0 as well as a CpK greater than 2.0 that demonstrates that the process in within Six Sigma quality levels. The Pp and PpK numbers will be examined to see how well the process is centered with a goal of equal to or greater than 1.667. The test will use the Edison platform to perform the printing using a standard configuration. A description of the DOE and details on the machine, materials and process used as well as an examination of the results is as follows:

Design of Experiment:

The test consisted of printing a total of 24 PCBs with the first 4 PCBs to be used as kneed boards to normalize the process and get the solder paste to a working viscosity. A wipe will be performed after each print to eliminate any noise from the data. The remaining 20 PCBs will be inspected by a Parmi SPI machine with the corresponding data analyzed through a on board SPC package. Process data for volume and height for the 008004" (0201mm) components will then be gathered and displayed and studied to determine the process capability of printing 008004" (0201mm) components. All equipment used was recertified prior to this test being performed.

Materials:

- Printed Circuit Board (PCB): The PCB used for this experiment is the new SMTA miniaturization test vehicle. **(Figure 2)** The board dimensions are 8.0" (203mm) in X and 5.5" (139.7mm) in Y with a thickness of 0.062" (1.57mm). There are approximately 400 pads per board with 200 positioned at 0 degrees and 200 pads positioned at 90 degrees. The 008004" (0201mm) pads are 0.005" x 0.006" (0.127 x 0.1524mm) with an air gap between pads of 0.0047" (1.1938mm) and a component pitch of 0.00126" (0.032mm)

- Stencil: 29” X 29” frame size, high tension, laser cut with Nano-coating. Aperture sizes were one to one matching the 0.005” x 0.006” (0.127 x 0.1524mm) pad size.
- Blades: 8” (220mm) stainless steel blades, at 55-degree attack angle
- Paste: Type 6 - SAC305 No-clean flux
- Tooling: Dedicated work holder with vacuum – custom made to SMTA PCB.

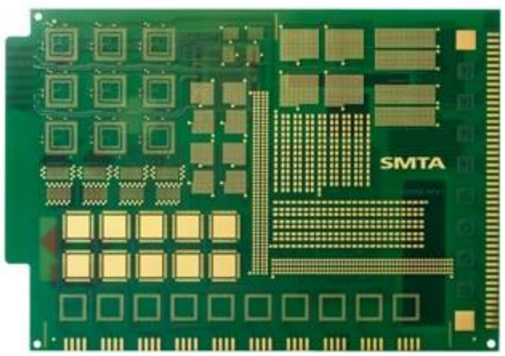


Figure 2: SMT Miniaturization Test Vehicle

Printing machine: the Edison platform was used for this test. **(Figure 3)** The Edison was specifically designed for small to medium sized boards and the printing of micro sized components. The machine vision specifications of a +/- 0.0003” (8µ) repeatability with a Cp of 2.0 @ six sigma and a wet print repeatability of 0.0006” (15µ) with a Cp of @ six sigma. The thin vision camera design reduces the distance the z-axis must travel when loading or releasing a PCB. The system uses a X/Y/Y alignment where the alignment motors have been moved further apart for better resolution. The Z-axis is tuned to the center of the board where when the Z-axes is raised, the PCB and the whole the rail assembly is decoupled from the table to eliminate any stack up intolerances. This ensures that proper gasketing is done and a clean release from the stencil. The stationary wiper is positioned in the front of the machine and the stencil is presented to the wiper via a shuttle eliminating any contamination in the print chamber. The print head uses a single load cell to monitor both squeegees to prevent print direction variation. All calibrations were performed on this machine and was validated using CeTaQ testing procedures.
Solder Paste Inspection: For solder paste inspection we used the Pari Sigma X that was retrofitted with the new high-resolution inspection head. Standard SPI inspection heads do not have the resolution to handle micro component paste deposits. The speed of the scan is slowed from a 100cm/sec to 60 cm/sec to facilitate the micro deposits. The system uses a dual laser optical triangulation and has specification of height accuracy of 2um with a height and volume repeatability of 1%. A gage R+R test was performed prior to performing testing.

Figure 3: MPM Edison Stencil Printer

Machine Parameters:

- Squeegee force: 14 lbs. (1.35 kg)
- Squeegee Speed: 1.5 in/sec (38mm/sec)
- Slow Release Distance/Speed: Distance = 0.100 (2.54mm) Speed = 0.100 (2.54mm)
- Wipe Frequency; Every PCB
- Wipe Sequence: Vacuum / Vacuum / Dry
- Board clamping: Vacuum and side snugging

Results:
Paste Volume and Height for Pads 0 degrees – refer to **(Figure 4)** for volume measurements and **(Figure 5)** for height measurements. The limits for volume were set to 50 % for the lower limit and the upper limit was set to 170%. The distribution curve is centered and shows the average paste volume to be 114.05%, the low end of the volume was 88.74% and the upper volume found was at 139.37. The Cp was calculated at 2.37 and the CpK was calculated at 2.21. The Pp was 2.37 and the PpK was 2.21. The height limits were set to 50% for the lower limit and 150% for the upper limit. The distribution curve is shifted towards the higher end with an average height at 114% with a lower end at 99% and an upper reading of 129%. The Cp was recorded at 3.2 with a CpK of 2.32. The Pp was calculated at 3.28 and PpK at 2.32.

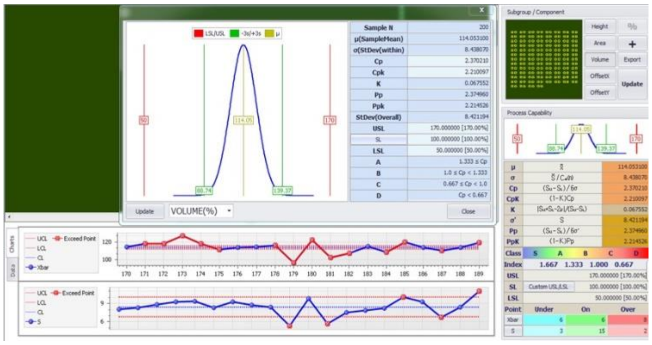


Figure 4: Volume Results 0-Degrees 008004" (0201mm)

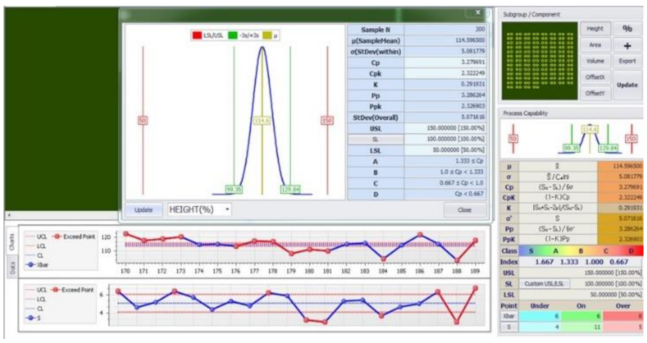


Figure 5: Height Results 0-Degrees 008004" (0201mm)

The results show that volume was treading slightly higher but consistent. The Height was higher than desired but the distribution is tighter, which was reflected in the higher Cp number.

Paste Volume and Height for Pads 90 degrees – refer to **(Figure 6)** for volume measurements and **(Figure 7)** for height measurements. The limits for volume were set to 50 % for the lower limit and the upper limit was set to 170%. The distribution curve is centered and shows the average paste volume to be 108.71% where the low end of the volume was 82.31% and the upper volume was at 135.11%. The Cp was calculated at 2.27 and the CpK was calculated at 2.24. The Pp was 2.27 and the PpK was 2.28. The height limits were set to 50% for the lower limit and 150% for the upper limit. The distribution curve is centered with an average height at 105.84% with the lower end at 94% and an upper reading of 117%. The Cp was recorded at 4.22 with a CpK of 3.81. The Pp was calculated at 4.31 and PpK at 3.82.

Figure 6: Volume Results 90-Degrees 008004" (0201mm)

Figure 7: Height Results 90-Degrees 008004" (0201mm)

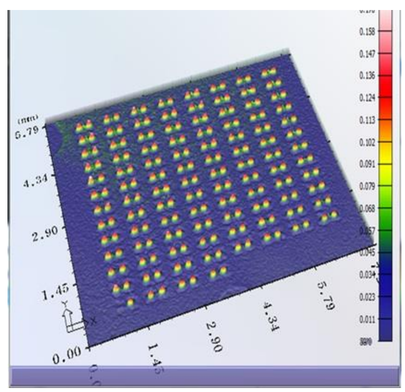
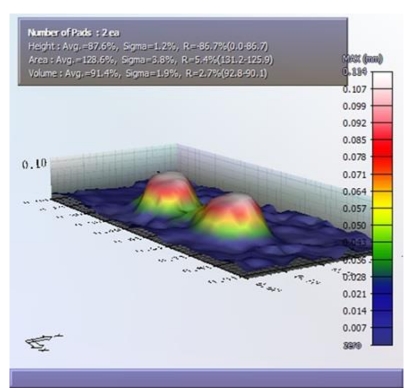


Figure 8: Overview of paste deposits 008004" (0201mm)



Figures 9: Individual Paste Deposits 008004" (0201mm)

The results show a tighter more centered curve for 90 degrees for both volume and height. The volume is shifted slightly towards the center, which is closer to the preferred position. The height for the 90-degree components shows a very tight curve with a Cp of 4.22, which is the most substantial improvement from the 0-degree components. closer to the preferred position. **(Figure 10)** shows a summary of the results.

008004" pads (Zero Degrees)

- Volume
  - Cp 2.370210
  - CpK 2.210097
  - Pp 2.374960
  - PpK 2.214526
- Height
  - Cp 3.279691
  - CpK 2.322249
  - Pp 3.286264
  - PpK 2.326903

008004" pads (90 Degree)

- Volume
  - Cp 2.272988
  - CpK 2.224115
  - Pp 2.277543
  - PpK 2.228572
- Height
  - Cp 4.322438
  - CpK 3.817346
  - Pp 4.331100
  - PpK 3.824996

Figure 10: Test Results Summary

CONCLUSION:

The results of the print test show that 008004" (0201mm) components can be repeatably printed and can be done where the process window is twice the capability. The results also show that pads that are oriented at 90-degrees gave the best results, however, the 0-degrees orientation was well within specification. The ability to print micro sized components is obtainable when each of the elements are addressed with the machine, the materials and the process working together. No individual aspect outweighs the other where working together, the wheels of the process turn in sync in order to achieve the desired results.

Future Work: Further investigation needs to be done, to see the effects of printing 008004" (0201mm) components with different powder sizes and understanding the associated costs versus capability, compared to the type 6 powder. Another test would involve comparing the results of squeegee blades vs enclosed print head, to see if there are any advantages between these two applications methods.

REFERENCES:

[1] Study completed Shea Engineering Services