

## SELECTIVE SOLDER PASTE DEPOSITION RELIABILITY TEST RESULTS

Bob Wettermann  
BEST Inc  
3603 Edison Place  
Rolling Meadows IL 60008  
bwet@solder.net

### **ABSTRACT**

*The rapid assimilation of Ball Grid Array (BGA) and other Area Array Package technology in the electronics industry is due to the fact that this package type allows for a greater I/O count in a smaller area while maintaining a pitch that allows for ease of manufacture. While the original assembly process has proven to be fairly trouble-free, replacement of Area Array package devices after the assembly process can be much more difficult. When reworking BGA and area array devices there continues to be discussion related to the technical merits of using solder paste versus flux only. While using flux only during attachment is the BGA rework technique of choice due to its simplicity, the solder paste printing attachment process is supported by data which indicates that it is a more reliable rework process compared to its flux only attachment counterpart. While there have been several studies comparing these two attachment methods, this study highlights the effect of rework technique on the electrical characteristics and reliability of reworked BGAs. Data from a recent reliability study comparing the two techniques with a focus on a new stenciling technique is presented.*

*Key words: BGA, rework, stencil, solder paste*

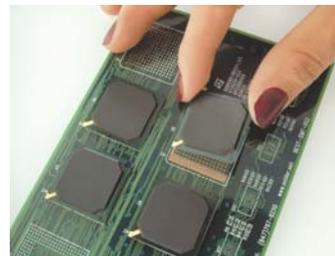
### **REVIEW OF SOLDER PASTE AND FLUX APPLICATION METHODS**

The area array rework process has a real dichotomy between its real world practical aspects and process theory. From a strict process engineering standpoint the proper method for reworking such devices calls for duplicating of the original manufacturing process as closely as possible. Following this guideline implies that the device to be reworked should be placed and reflowed using solder paste and the board/device should be subjected to a very controlled temperature profile such as those found in a modern multiple zone reflow ovens. In addition, the profile printing process should be uniform and controlled as it would be in an automated stencil printing process. Process variables such as squeegee speed, squeegee pressure and snap off velocity should all be monitored and controlled as closely as possible. Instead, what is more often the case, devices are placed using only “tacky flux” smeared across the bottom side of the site being reworked. More often times than not the reworked devices are reflowed using a hot air source with a top side only controlled heating source. Proper rework

Selective Solder Paste Deposition Reliability Test Results  
Bob Wettermann/BEST Inc

procedures, which should mimic the original manufacturing process to ensure the reliability of the reworked device, have taken a backseat to expediency.

There are numerous practical solder paste application methods for reworking area array devices. The most common method used to selectively apply solder paste in reworking such devices is through the use of a metal or one time use plastic film stencil. After the device is removed and the site prepared, the stencil serves to selectively apply solder paste to the various land patterns or sites on the printed circuit board (PCB). These stencils have hole patterns corresponding to the land patterns of the PCB. Solder paste is pushed manually through these apertures using a small squeegee blade with a rework technician attempting to control as many of the paste printing variables of the automated machinery including squeegee pressure, stencil/board release speed and uniform solder paste deposition speed. These variables, even when an experienced rework technician is at the controls, can only be controlled to a limited extent and is a function of the experience level of the operator. However, the problems of warped PCBs can lead to a poor gasket being formed between the bottom of the stencil and the PCB leading to “smearing” of the solder paste patterns. Additional problems in improper perpendicular “lift off” of the squeegee from the site location as well as the lack of a consistent solder paste volume in each of the apertures can lead to shorting between leads, opens and other solder attachment problems. A newer method <Figure 1>of solder



**Figure 1 Semi-Permanent BGA Rework Stencil Attachment**

paste printing, in which a polyimide pressure sensitive tape with hole patterns, serves as a semi-permanent stencil on the PCB, takes away all of the operator variability. Since the stencil becomes a permanent

embodiment of the PCB (like a label), the apertures can be filled with consistent solder paste volumes and there are no issues related to the perpendicular “lift off” especially from warped boards. In addition, the polyimide material between each of the solder balls serve to prevent solder shorts. Limitations of this technique include the inability to clean underneath the BGA or CSP that may reduce the effectiveness of the cleaning operation and the reduced ability to perform optical inspection. Another technique which can be used to selectively apply solder paste to a device rework area is the application of solder paste onto the device rather than the board. This technique utilizes special fixtures and stencils surrounding the part underside while solder paste is printed on to the device prior to placement. Therefore, devices located in high-density areas of the PCB can be reworked while alignment between the solder paste patterns and the solder spheres is insured. Relatively costly proprietary fixture systems are one of the main limitations of this technique. Solder paste can also be applied by a dispensing system which lays somewhat precisely-controlled volumes of solder paste in specifically pre-programmed X-Y locations on a PCB. This technique consistently deposits and controls the paste deposition. Slow cycle rates, long programming times and solder paste chemistry limitations make this a seldom-used technique.

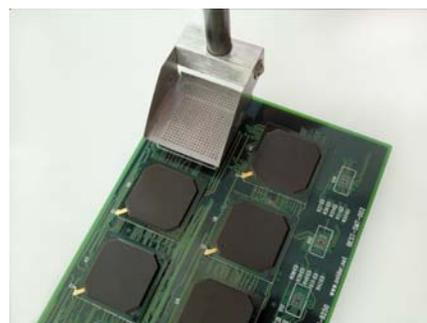
#### **DISPENSE**

The technology of dispensing solder paste has made many advancements and improvements. Most of the advancements have occurred in the fully automated systems and includes changes to the needle configuration, the pump mechanism, and the programming. Although it is possible to manually dispense solder paste with a syringe and needle, it would result in too many inaccuracies and excessive cycle time. The use of semi-automated dispensing systems, which control the quantity of paste deposited by adjusting air and timer settings, provides greater control over the paste quantity, but will not improve accuracy. A fully automated dispensing system usually consists of an X-Y table and a computer controlled dispense head. These systems are programmed with the coordinate data for the locations requiring solder paste and then sequentially dispense the paste at the desired locations. They are very accurate in the paste deposit placement providing very consistent paste deposit quantities, however, one of the drawbacks of using a dispensing system is the requirement to use a solder paste with a lower metal content. Higher metal contents ( $\geq 87\%$ ) usually result in frequent clogging of the nozzle. With this lower metal content, the paste deposit is more prone to slump and separation than higher metal contents. Applying paste to the same locations on many boards is fairly easy due to the programming of the system and fixturing of the PCB. The major drawback to the dispense system is throughput. For larger pin count BGAs, cycle time is limited by the maximum speed of the

pump and may take as long as five minutes when set-up time is included. From a rework point of view, the program and set-up time must be included for every different BGA to be processed. In the subsequent comparison of solder paste deposition methods only methods of stenciling will be reviewed.

#### **METAL STENCILS**

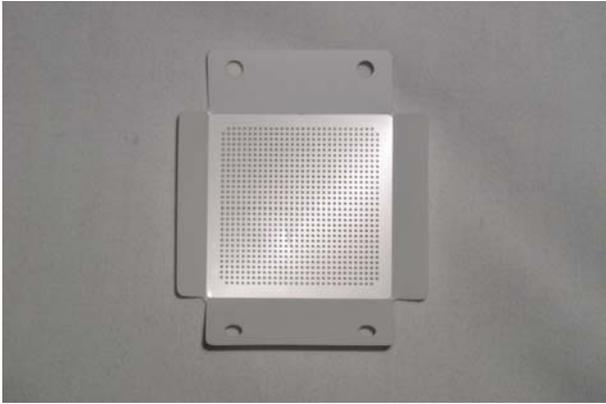
Metal stencils for depositing solder paste have been in use for decades. Metal stencils for selective (a particular device, not the entire PCB) solder paste deposition have been in use almost as long. One of the greatest yield detractors in the SMT manufacturing process is the solder paste application process and the same holds true for the solder paste application process for rework. The process steps for using a component specific stencil for selective solder paste deposition for rework and a full PCB stencil for the original SMT manufacturing are almost identical. The major difference between the original stencil printing process and the selective solder paste printing process used for rework is the amount of process control. During the original solder paste stenciling process there are machine controls for the squeegee speed and pressure, snap-off distance, removal of the stencil from the board, and in some cases stencil alignment. The selective solder paste stenciling process is very dependent on the skill level of the technician to manually control the process. The metal stencil printing process begins with the stencil being aligned with the land patterns on the PCB. Next the stencil must be held in place in a manner that ensures intimate contact with the PCB. A squeegee is then used to roll a bead of solder paste across and down through the apertures of the stencil. Finally, the stencil is lifted from the PCB surface resulting in finely defined solder paste deposits. If the stencil printing operation runs perfectly, the cycle time is very fast. Cleaning the stencil is a critical process step to ensure the stencil will provide acceptable results on the next BGA to be reworked. One variation on the metal stencil process is to print the solder paste directly onto the solder balls instead of the BGA lands on the PCB. This method provides the advantage of printing solder paste where placing a stencil on the board would be impossible due to space considerations.



**Figure 2 Metal Stencil Solder Paste Attachment**

### **FLEXIBLE REMOVEABLE STENCILS**

A relatively new material/process for deposition of solder paste is the use of flexible solder paste stencils. These stencils are laser cut from a polymer film with a residue-free adhesive backing that allows for easy removal. The first step to using the flexible stencil is to remove the



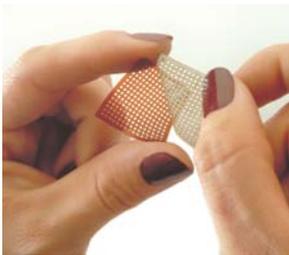
**Figure 3 Flexible Stencil**

paper to expose the adhesive backing. The stencil must then be manually aligned with the land patterns on the PCB and firmly pressed down in place on the PCB surface. A squeegee is then used to roll a bead of solder paste across and down through the apertures of the stencil. The stencil must then be carefully removed from the

PCB surface. There is no cleaning step as the stencil is disposable. Cleaning with any type of solvent would result in some loss of the adhesive.

### **SEMI-PERMANENT STENCILS**

An alternative method of stenciling solder paste on the PCB is to use a stencil that remains in place on the site location and becomes an integral component of the PCB. The stencil material is a polyimide film



**Figure 4 Semi Permanent Stencil**

with a high temperature adhesive covered with a paper backing. This material combination has been used on PCBs for many years now in the form of bar code labels and Kapton® tape. This material is available in the standard formulation or in an ESD safe variety. Like the removable stencil, the semi-permanent stencil is laser cut and can be provided in a number of different configurations. When using the semi-permanent stencil, the paper backing material is removed exposing the high

Selective Solder Paste Deposition Reliability Test Results  
Bob Wettermann/BEST Inc

temperature adhesive. The stencil is then manually aligned with the land patterns on the PCB and pressed into place. Again a squeegee is then used to roll a bead of solder paste across and down through the apertures of the stencil. At this point the paste application process is complete. The stencil is not removed from the PCB and therefore no stencil cleaning step is required.

### **FLUX ONLY APPLICATION METHODS**

There are several techniques which are used to apply “tacky” flux to the area array device being reworked. The most common technique involves using a syringe to dispense the flux onto the pad areas followed by “smearing” of the flux with a brush or gloved finger tip over the entire surface on which the part will be placed. This is a fast, easily understood method to apply flux. However, there is the risk to the long-term reliability of this interconnection as excessive flux residue may reduce the surface resistivity of the PCB surface. Semi-permanent, metal or plastic stencils can be used to apply the flux resulting in flux only being deposited where it is needed. The use of the semi-permanent stencil electrically isolates these pathways as the polyimide wells act as dielectric barriers between each of the pads. Flux only application can also be accomplished by dipping the part into the flux.



**Figure 5 Flux Dipping of BGA (OK International Inc.)**

and placing the part onto the PCB. All of these “flux only” application methods have shortfalls as these methods cannot compensate for the lack of co planarity in either the part or the PCB, relying instead on the solder volume of the device balls only. This in turn leads to a reduction the reliability of the solder joint as more opens can result.

### **PURPOSE**

The purpose of this study was twofold. First the effect on BGA rework reliability as a function of the type of rework attachment technique was investigated. Secondly, this study was commissioned in order to determine the effect of electrical properties post BGA rework using a variety of different attachment methods and techniques.

### **THERMAL SHOCK TESTING**

A portion of the test PCBs were exposed to thermal shock tests in order to check for open circuit conditions of

reworked BGAs both with and without the use of a semi-permanent stencil. In this series of tests, the resistance of daisy chain circuit patterns running between the BGA and test board after exposure to thermal shock was measured. Samples were prepared in order to compare two separate test groups. Each test group consisted of (5) sample PCBs with (4) BGA devices per PCB. Two different groups of samples were prepared as follows:

#### Group 1 Semi-Permanent Stencil using a Solder Paste

##### Reattach Method

#### Group 2 Mini Metal Stencil Reattach Method using

##### Solder Paste

A second grouping, used to compare the thermal shock test results of flux only versus selective solder paste deposition, were prepared as follows:

#### Group 3 Flux Only Reattach Method

#### Group 4 Mini Metal Stencil Reattach Method using

##### Solder Paste

### **METHOD OF TEST SAMPLE PREPARATION**

Samples were prepared using BGA daisy chain dummy components from Amkor with 484 I/O and pads with a 1.0 mm pitch. The daisy chain patterned PCBs were a single-sided FR4 material with 1 oz. copper and a HASL finish. Each PCB had pad sizes of .025 inches with a 1mm pitch. Four separate daisy chain circuit patterns for each BGA with a total of 4 BGA's were populated on each test vehicle.

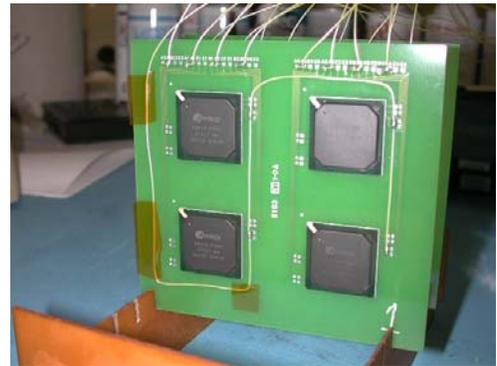
The Group 1 thermal cycling test group had the BGAs reworked using the semi-permanent stencil rework technique while Group 2 samples were prepared used standard reattach methods. For assembly of Group 1, a polyimide film stencil (0.008" polyimide with high temperature pressure sensitive adhesive) was attached to the test PCBs with the BGA pads exposed through openings in the film. Solder paste (Alpha Omnix 5000) was then applied using a .010 inch stainless steel handheld squeegee to each aperture opening with proper care being paid to the fill of each of the aperture openings. The film stencil was left in place. A balled dummy BGA (.025" diameter / Sn63/Pb37 solder alloy ball) was placed by hand on the stenciled card and reflowed using a standard developed profile.

For Group 2 and 4 test samples, the same steps as above were used except for the method of selective solder paste application. The solder paste was applied by using a .008 inch stainless steel "mini" rework stencil and applying Selective Solder Paste Deposition Reliability Test Results  
Bob Wettermann/BEST Inc

solder paste to the aperture openings. After releasing the stencil from the board, the stencil was then removed leaving the solder paste on the board. Similarly a balled "test" BGA was then placed on the circuit board.

Similarly Group 3 test samples were prepared as previously described except "tacky" flux (Alpha Ultra Print 78) was used as the attachment method. The flux was applied with a cotton tip applicator to the corresponding land patterns on the PCB.

These groups were exposed to a rigorous thermal shock exposure test in order to accelerate any mechanical failure mechanisms. The samples were exposed from -45°C through room temperature up to +70°C. The transition time between each temperature was less than 15 seconds with a 10 minute per segment dwell time. Resistance values of each daisy chain segment were manually measured and recorded at the outset as well as at the end of 10, 50, 100, 300 and 500 thermal cycles. A 20% increase in resistance in any segment from the initial measurement was determined to be the failure criteria.



**Figure 6- Wired test board**

Samples for surface insulation resistance tests were prepared in much the same way as the thermal shock test using similarly layed out PCBs. Samples were made up with BGA daisy chain BGAs from Amkor with 484 I/O and pads at a 1.0 mm pitch. The daisy chain test PCBs were a single sided FR4 material with 1 oz. copper with a HASL finish. PCB pad pitch for the BGA was .025" with a 1mm pitch. Two patterns of the daisy chain patterns of the BGA run adjacent to a third common pattern. Two groups of samples were prepared for comparison of SIR measurements. Insulation resistance measurements were made from each of the individual patterns to the common. One group of samples was reworked using the semi-permanent and "mini" metal rework stencils while the second group had the BGAs attached using the "tacky" flux only technique.

In the first group of prepared samples "tacky" flux was used to attach the BGA. For assembly of the first board, the polyimide film stencil was placed on the test boards with the BGA pads exposed through openings in the film. Flux was then applied to each aperture opening. The balled dummy BGA was then hand-placed on the

stenciled card and reflowed. Board two preparation followed the same steps as board one except for the application of the flux. The flux was applied to the circuit board land patterns with a cotton tip applicator. The BGA was then placed on to the circuit board using the rework systems' split vision alignment system. Both card types were reflowed using the same standard profile with a maximum temperature of 220 °C.

In the second grouping of SIR test boards, selective solder paste attachment methods were used. For assembly of board two, the polyimide film stencil was placed on the test cards with the BGA pads exposed through openings in the film. Solder Paste (Alpha Omnix 5000) was then applied to each aperture opening, filling the aperture opening. Excess solder paste was removed from the top surface of the stencil. A balled dummy BGA was then reflowed using the already-developed profile.

For the preparation of board one in the second grouping, the same steps as above were followed except for the application of the solder paste. The solder paste was applied by using a stainless steel handheld squeegee and rolling solder paste into the aperture openings of the miniature .010 inch thick stencil.

Surface insulation resistance measurements as documented in the Telecordia (Bellcore) NEBS standard GR-78, Issue 1, Section 13.1.3 were performed. Initial room ambient measurements were made and were followed by a 24 hour unbiased soak at 35° C and 85 % RH. Measurements were made at this time. These measurements were followed by a 96 hour soak at 35° deg. C / 85 % RH with a 50 Vdc bias between the patterns and common. Final measurements were made while under soak with the bias removed.



Figure 7- Test boards in Thermal Cycle Chamber

**THERMAL SHOCK TEST RESULTS**

**Standard Paste vs Std Flux Attachment Reliability Test Data**

Number of Cycles	Std Flux Application Percent Cumulative BGA Failures	Std Paste Application Percent Cumulative BGA Failures
10	0%	0%
50	0%	0%
100	5.1%	0%
300	6.3%	0%
500	8.8%	0%

Table 1

**Semi-Permanent Stencil vs Standard Paste Attachment Reliability Test Data**

Number of Cycles	Semi-Permanent Stencil Paste Application	Std Paste Application Percent Cumulative BGA Failures
10	0%	0%
50	0%	0%
100	0%	0%
300	0%	0%
500	0%	0%

Table 2

**SIR Test Results-Flux Only Attachment**

**Initial Measurements**

Semi-Permanent Stencil Technique  $1.0 \times 10^{12}$  ohms  
 Flux Only Technique  $2.9 \times 10^{10}$  ohms

**After Soak**

Semi-Permanent Stencil Technique  $9.3 \times 10^9$  ohms  
 Flux Only Technique  $7.7 \times 10^8$  ohms

**After Soak and Bias**

Semi-Permanent Stencil Technique  $1.1 \times 10^{10}$  ohms  
 Flux Only Technique  $2.8 \times 10^9$  ohms

**SIR Test Results-Solder Paste Attachment**

**Initial Measurements**

Semi-Permanent Stencil Technique  $2.0 \times 10^{10}$  ohms  
 "Mini" Metal Stencil Technique  $8.5 \times 10^{10}$  ohms

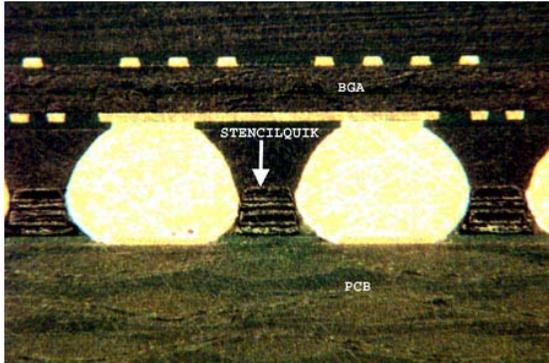
**After Soak**

Semi-Permanent Stencil Technique  $2.1 \times 10^9$  ohms  
 "Mini" Metal Stencil Technique  $4.0 \times 10^9$  ohms

**After Soak and Bias**

Semi-Permanent Stencil Technique  $2.6 \times 10^9$  ohms  
 "Mini" Metal Stencil Technique  $5.4 \times 10^9$  ohms

An average of the 8 measurements for each board was computed.



**Figure 8-Semi-permanent Stencil, BGA and PCB Cross-Section.**

In the case of the flux reattachment technique surface insulation resistance values were two orders of magnitude higher for the samples prepared using the semi-permanent stencil. Each of the individual measurements showed the same trend as reflected in the averages reported in these results. The surface insulation resistance values were marginally lower for the samples prepared using the semi-permanent paste stenciling technique.

There was an observed reliability difference between the use of solder paste and flux only attachment methods in the rework process (Table 1). In the standard flux only reattachment process 24 out of the 80 circuits had failures. This has been observed in other studies (7,8) and is likely the result of additional solder volume in the solder paste printing process which compensates for the lack of board and part coplanarity and/or the increased standoff height of the part from the board. The data also indicated that after (500) thermal shock cycles that there was no detrimental impact on the reliability of the interconnect regardless of the paste stenciling technique used (Table 2).

## **DISCUSSION**

The enhanced surface insulation resistance properties of the semi-permanent stencil placed BGAs is a result of the properties of the stencil. Each of the aperture “wells” of the semi-permanent stencil entraps the flux as the adhesive does not allow the flux to seep between each of its neighboring wells. Flux residue does not have the means to migrate along the surface of the PCB and create a potentially conductive pathway, thereby enhancing the insulation resistance between the neighboring solder balls on the surface of the PCB.

There are several key points to keep in mind with respect to the use of flux in the rework process which can cause the reliability of the reworked joint to be compromised. Selective Solder Paste Deposition Reliability Test Results  
Bob Wettermann/BEST Inc

Theses include but are not limited to solder joint size, standoff height underneath the package and the lack of compensation for the non-planar parts and boards.

The size of the solder joint will be smaller after reworking the part with flux compared to a paste print attached part. Decreased solder volumes of BGA joints have been shown to have reduced mechanical reliability compared to those with greater volumes. In fact Motorola lists in its application notes this lack of solder paste volume or “skips” as one of the “Top 10 Causes of BGA Assembly Defects” (1).

The packages’ standoff height from the bottom of the part to the surface of the PCB is reduced when using the flux only attachment technique. The resultant reduction in heat dissipation causes the part to heat up locally and realize a greater resultant thermal stress thereby reducing the expected solder joint life.

Another reason that the use of flux only in the BGA rework process reduces the longevity of the interconnect life is that it fails to make up for the differences in planarity. If either the part or the PCB becomes warped due to unevenness in heating either surface can be non-planar. The resulting reduction on solder ball volume post reflow in the flux only reattachment technique leads to the greater likelihood of an “open” connection.

There are several reasons for the expected better performance of paste printed reworked BGAs compared to their flux only attached counterparts including: increased solder volume and the compensation effects which can make up for the lack of part or board coplanarity in solder attachment.

Increased solder paste volumes due to the use of the solder paste attachment have several advantages in the BGA rework process. Several studies have indicated that a greater standoff distance between the part and the PCB leads to more efficient localized heat dissipation as a part goes through its thermal cycles (7). Intel’s rework guidelines of PBGAs calls for a 8 mil thick rework stencil due to the increased paste print volumes’ effect on expected reliability gains. Motorola’s part application guideline also warns against the dangers of using too thin of a rework stencil “as lower solder paste volumes can affect rework yields” (1).

Printed circuit board co-planarity defects can be overcome by increased solder paste volume. Thinner PCB substrates, localized ground plains or uneven heating of the PCB during the rework process can all lead to board warpage. Large BGA package sizes, as well as poor quality control in solder ball dimensions can also cause board co-planarity discrepancies along the part underside. Poor co-planarity in either the PCB or the part surface can lead to opens or uneven collapse and even no connects after reflow. Increased solder paste volumes overcome

these height differentials in the rework process leading to more reliable interconnects.

### **CONCLUSION**

The conclusions of this study confirm other works which point to the use of solder paste attachment as a superior rework technique compared to the use of flux only attachment in the BGA rework process. If expediency is the primary goal and flux only attachment is the technique of choice, then the use of the semi-permanent stencil will enhance the electrical properties of the interconnections.

As the results of this study indicate there are no differences between the interconnect reliability of a metal stencil and semi-permanent paste printed BGA. The semi-permanent stenciling technique helps compensate for differences in the part/board co-planarity. Greater volumes afforded by the semi-permanent stenciling technique provide for a greater standoff distance and a potentially more reliable joint. More exhaustive testing would help determine if there any differences in long-term reliability between the metal stencil solder paste printing rework process and its semi-permanent counterpart.

There are negligible differences in processing time between the flux only and a semi-permanent solder paste attachment rework technique. As IBM points out in its application guidelines, the negative aspects of standard paste screening on the board for rework are “(that)...there is limited room for local screening (and) poor print releases when lifting the local site stencil” (2). These negatives in the solder paste printing process are overcome with the “semi-permanent” stencil method, do not impact BGA rework throughput all while providing for a more reliable solder joint than the flux only attachment alternative.

### **References**

- [ 1 ] Digital DNA Motorola BGA Application Guide, Motorola 2003.
  - [ 2 ] CBGA Surface Mount Assembly and Rework User's Guide, IBM 2002.
  - [ 3 ] “Chip Scale Package Rework Considerations: Using Solder vs Gel Flux”, Paul Wood , Intel Application Series, 2002.
  - [ 4 ] “Solder Ball Endurance”, Terence Collier, Advanced Packaging, September 2003
  - [ 5 ] “Flip-Chip / CSP Assembly Reliability and Solder Volume Effects”, Jean-Paul Clech, SMI Conference Proceedings, pp 315-324.
  - [ 6 ] “Solder joint Reliability of Sn-Ag-Cu BGA Components Attached with Eutectic PB-SN Solder Paste”, Hua et al., Journal of SMT, pp34-42, Volume 16 Issue 1, 2003.
  - [ 7 ] ”BGA Reliability Characterization Project Temperature Cycling Tests Final Report”, David Love and David Towne, High Density Packaging User Group International, January 1999.
- Selective Solder Paste Deposition Reliability Test Results  
Bob Wettermann/BEST Inc

[ 8 ] “Investigation of the Manufacturing Challenges of 2577 I/O Flip Chip Ball Grid Arrays”, Thomas Cipelewski and M Meilunas, IPC APEX SMEMA Council Designers Summit 04

[ 9 ] IPC 7095-Design and Assembly Process Implementation for BGAs, Device Manufacturers Interface Committee of IPC

[ 10 ] Bingzhi Su, Saeed Hareb, and Y.C. Lee, Mirng-Ji Lii and Mark E Thurston, “Solder Joint Reliability Modeling for a 540-I/O Plastic Ball-Grid-Array Assembly”.

[ 11 ] Xingsheng Liu, Materials Science and Engineering Department and Power Electronics Packaging Laboratory Virginia Polytechnic Institute and State University, March 27, 2001.

[ 12 ] Greg Caswell and Julian Partridge, “BGA to CSP to flip chip-Manufacturing Issues”, Journal of the Microelectronics & Packaging Society Vol. 8, No. 2 (2001)

[ 13 ] AG Communications Thermal Shock Test Study StencilQuik™