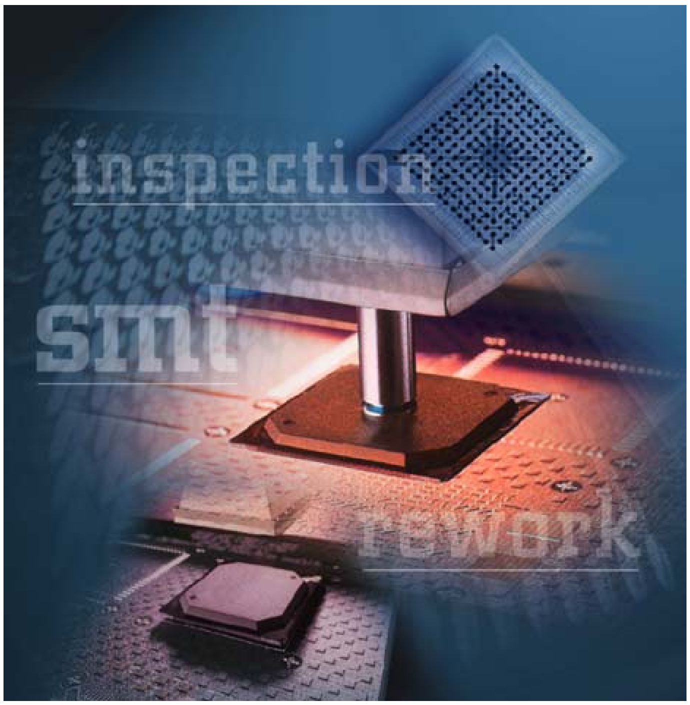
Solder Paste versus Flux Only Attachment for BGA Rework-
Reliability vs Ease of Application?

Bob Wettermann

Abstract:

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. This discussion will focus on the practical aspects of reworking BGAs and other array devices using a variety of both paste and flux only application methods. Data from a recent reliability study comparing the two techniques with a focus on a new stenciling technique is presented.



Introduction

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

Paste application methods

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

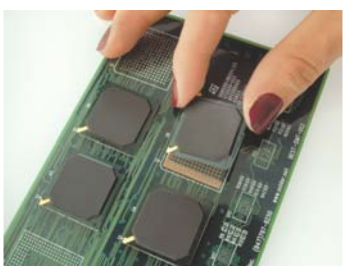


Figure 1 Semi-Permanent BGA Rework Stencil Speeds Rework
(BEST Inc.)

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, 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 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.

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 resisitivity 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 <See Figure 2>

Figure 2 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 to determine the reliability affects of using either the solder paste or flux only attachment technique in the BGA rework process. Furthermore, the study was commissioned to determine what effect the presence of a semi-permanent stencil has on the reliability of a reworked BGA.

Testing
The test methodology for this study included the measuring of resistance values of reworked stencil-printed BGA test samples made with and without the semi-permanent stenciling method after thermal shock cycles. These same tests were carried out using a flux only reworked BGA.

Based on well-proven telecommunications industry testing each of the test cards was exposed to a thermal shock exposure test. Test cards were cycled between –45ºC and +70º C with a 10 minute per segment and 40 minute per cycle dwell. Between each of the 500 cycles there was a transition time of less than 15 seconds. Each of the daisy chain segments were monitored and manually measured for resistance values initially and at intervals up to 500 cycles. A 20% increase in the resistance of any segment constituted a failure.

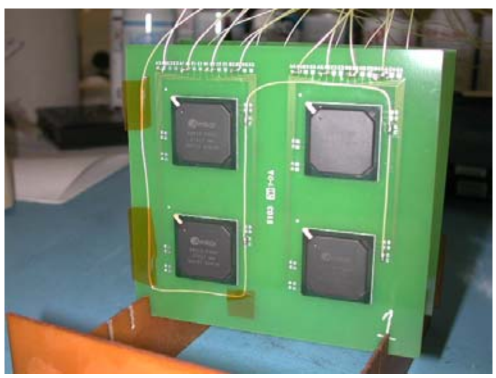


Figure 3- Wired test board

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 Percent Cumulative BGA Failures	Std Paste Application Percent Cumulative BGA Failures
10	0%	0%
50	0%	0%
100	0%	0%
300	0%	0%
500	0%	0%

Table 2

After 500 thermal shock cycles the following results were noted:

Group 1	Semi-Permanent Stencil with Solder Paste	No Failures
Group 3	Standard Reattach with Solder Paste	No Failures
Group 4	Standard Reattach with Flux Only	24 of 80 Circuits Fail 4 out of 5 Cards Fail

With a 95% confidence this data points to a cumulative failure rate of less than 4.6% for both the semi-permanent and standard paste deposition BGA rework methods. Beyond 50 cycles the data was demonstrated to be statistically significant given the sample sizes tested.

Surface insulation resistance (SIR) test measurements were performed and compared on samples both with and without the semi-permanent stencils (as documented in the Telecordia (Bellcore) NEBS standard GR-78, Issue 1, Section 13.1.3). There was no degradation in the SIR test results in either the solder paste or the semi-permanent stencil printed samples.

Discussion

There was an observed reliability difference between the use of solder paste and flux only attachment methods in the rework process (Table 1). This has been observed in other studies 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 indicates that after 500 thermal shock cycles there was no detrimental impact on the reliability of the interconnect in the case of semi-permanent stencil reworked versus standard paste application reworked BGAs (Table 2). Additional testing or more cycles would help to determine if there any differences in reliability between the metal stencil solder paste printing rework process and its semi-permanent counterpart.

Discussion

Flux-A Reduction in Solder Joint Reliability

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. Theses include but are not limited to solder joint size, standoff height underneath the package and the lack of compensation for the lack of part or board co-planarity.

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”.

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. With the reliance of a reduced solder ball volume to compensate for the lack of planarity the likelihood of an open connection or reduced is increased.

Paste- Better Performance for Reworked Parts

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 for the lack of part or board co-planarity in solder attachment.

Increased solder paste volumes due to the use of the solder paste attachment process 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).

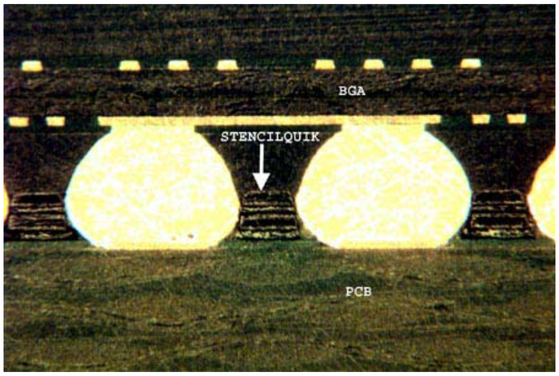


Figure 4-Semi-permanent Stencil, BGA and PCB Cross-Section.
(BEST Inc)

Printed circuit board co-planarity defects can also 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.

Semi-Permanent Stencils-Convenience and Reliability in the Rework Process

The semi-permanent solder paste printing technique allows users to realize the reliability of solder paste printing while experiencing the convenience of the flux only attachment process.

As this study's data indicates there are no differences between the interconnect reliability of a metal stencil and semi-permanent stenciling technique. In the "Top Ten Causes of BGA Assembly Defects" solder paste "skips" or insufficient solder paste volume along with poor part co-planarity are mentioned as the primary causes of BGA placement defects. Greater volumes afforded by the semi-permanent stenciling technique provides for a greater standoff distance and a more reliable joint. In addition, the semi-permanent stenciling technique helps compensate for differences in the part/board co-planarity effects. due to these greater paste print volumes.

The difference in rework process time between a flux only and a semi-permanent solder paste printing rework job is neglible. As IBM points out in its rework guideline the negatives of paste screening on the board for rework are "…there is limited room for local screening, poor print releases when lifting the local site stencil, and the need to clean the entire PCB of the print has to be stripped" (2).

All of these negatives are overcome with the newer "semi-permanent" stencil method as well as the benefit of a more re.

The difference in processing time between a flux only and a semi-permanent solder paste attachment rework job is neglible. 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, poor print releases when lifting the local site stencil, and the need to clean the entire PCB." (2). All of these negatives in the solder paste printing process are overcome with the newer "semi-permanent" stencil method as well as the benefit of a more reliable solder joint.

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 Assemby 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 el.,, 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.
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**