Applications Notes on Surface Mount Assembly of Amkor Technology BGA Packages

Paul Mescher

January, 1998

Amkor Technology, Inc.
1900 S. Price Road
Chandler, AZ 85248

1.0 Introduction

Ball Grid Array packages provide a new challenge for the surface mount industry. In all peripheral leaded packages (e.g. Quad Flat Packs or QFPs), the final solder joints can be visually inspected to guarantee quality of both the process and the resulting joint. In contrast, BGA packages, when attached to a printed circuit board (PCB), effectively hide the solder joints between the package body and the board. Only the outside row might be visibly inspected, and that will depend on proximity and location of neighboring components. Thus the guarantee of quality of the surface mount process for BGAs can not be accomplished through visual inspection. Instead it is simple process control, combined with package design, that drive significantly higher soldering yields than are achieved with fine pitch QFP devices. This document is intended to provide some guidance in the control methods and limits required to successfully implement BGA packages into the surface mount assembly process.

2.0 BGA Solder Joint Reliability

In all packaging, it is not good enough to simply create an interconnection that will conduct electricity. That connection must remain intact for some desired period as well, with that period usually defined by some combination of power-on hours and on/off cycles. This is the concept of reliability. As with most SMT packaging, the weakest link in cycling BGA assemblies is the package to card interconnection. The weak link in peripheral packaging can be either in the solder joint or the lead itself, depending on geometry. BGA packages will fail in the solder interconnect at the point of highest stress concentration, which is usually the smallest cross-sectional area (excluding voids). This point will be either at the ball to package interconnection or ball to PCB interconnection, depending on the design geometry used.

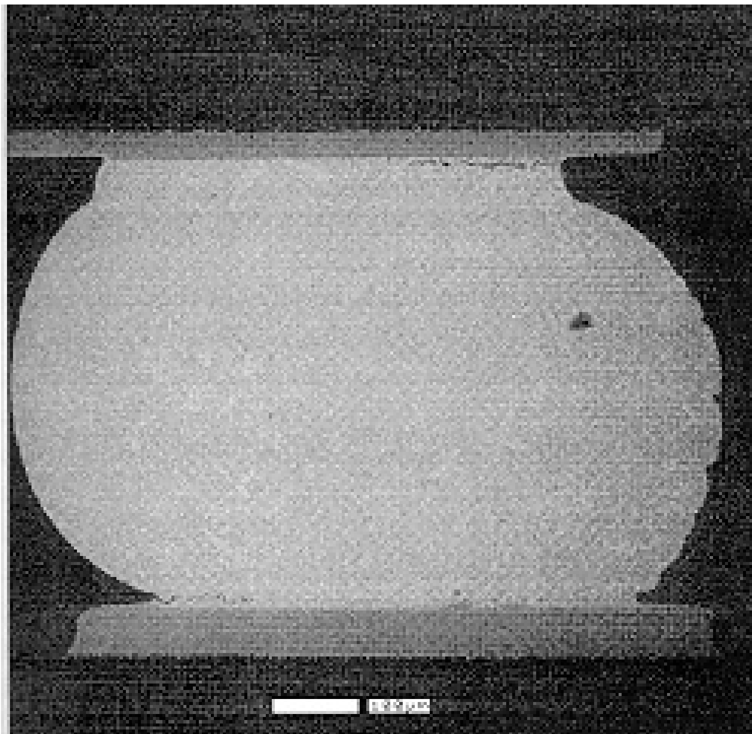

Figure 1 - Fatigue Crack in Eutectic Solder Ball[1]

This behavior results in two advantages for BGA reliability in assemblies. First, as the cross section is controlled by the copper land sizes, which are in turn controlled closely by the manufacturers of substrates and PCB's, the fatigue propagation time across the surface is very consistent. Thus the scatter bands typically associated with QFP solder joint reliability are much narrower with BGA assemblies, allowing much better prediction of field life. The second is that BGA reliability is much less dependent on assembly variables than QFP solder joints. Only process changes that affect the standoff between PCB and package have been shown to alter the thermal cycle life. Other items, such as voids within the solder joint, have been studied and shown to have no negative effect on fatigue life.[2,15]

With an understanding of reliability and its sensitivities (or perhaps more accurately, lack of sensitivities), it is possible to define the assembly process and controls desired to implement BGA products with high yields and high reliability. The general concepts of control for BGA are fairly universal. However, exact process control limits and detailed specifications will depend on the factory performing the assembly operations.

3.0 PCB Design

Control of the BGA assembly process really begins not at the beginning of SMT processing, but at the design of the surface mount pads. The basic pad design is fairly simple. An array of round pads provides the mounting surface. The mounting pads MUST be isolated from any plated through holes (PTHs) required for connection to lower planes. The most common design for this kind of connection is to put the PTH interstitial to the mounting pads and connect to the pad with a wide copper trace. Some individual pad examples are shown below, along with a larger array sample showing interstitial PTH location.

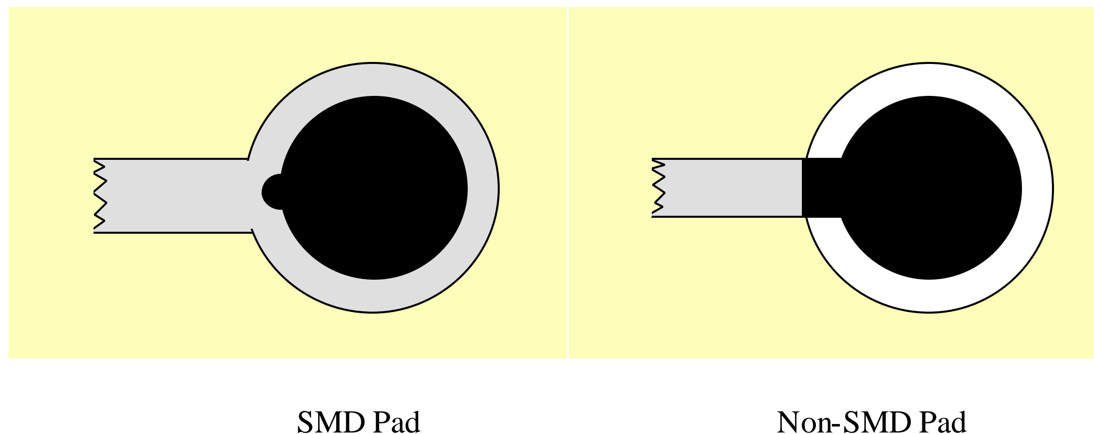

Figure 2 - SMD vs. Non SMD Pad Designs

There are two possibilities for definition of the pad surface: Solder Mask Defined and Non-Solder Mask Defined (Copper Defined). Each has advantages and disadvantages to its use. While circular shaped pads are most common, alternate shapes (e.g. ovals, diamonds) are occasionally seen in use as well. The input trace to the pad can take a variety of forms as well. It is recommended that the trace be at least .010” when using NSMD pads, to reduce the risk of breaking due to local mechanical stresses. Work in this area will continue to add to the body of knowledge as more BGA users come on line.

3.1 Pad Definition Techniques

3.1.1 Solder Mask Defined Pad

In the solder mask defined pad, the copper for the pad area is made larger than the desired land size. The opening in the solder mask is made smaller than the copper land, thus defining the mounting pad. The solder mask defined pad has two primary advantages. First, solder mask size definition is typically better than copper pad definition, as the copper depends on etch control while the soldermask is photoimaged. Second, the overlap of the solder mask onto the copper enhances the copper adhesion to the laminate surface. When using resin systems where adhesion is low, this can be an important consideration.

3.1.2 Non-Solder Mask (Copper) Defined Pad

The non-solder mask (or copper) defined pad has a soldermask opening larger than the copper area. Thus the size of the pad is controlled by the copper etch quality control. This is generally somewhat less accurate than the solder mask photoimage control, and thus the pad size varies more than with the SMD pad. However, because the edges of the copper do not need to extend under the solder mask, the pad can be either made larger, or provided more line routing space between pads. The pattern registration is also as accurate as the copper artwork, which is generally much more accurate than the solder mask pattern. Vision registration on copper fiducials will give exact location of the site. With SMD pads, the misregistration error of the soldermask will also shift the location of the entire site relative to vision fiducials.

3.1.3 Design Recommendations

Industry practice has, in general, trended toward the use of NSMD pads. While there is limited data suggesting a slight fatigue life improvement when matching SMD pads on PCB and package[1], there are very few real applications that approach the limits closely enough for this to be a consideration. The specific advantage in fiducial to pad registration for NSMD pads has proven to be the primary consideration for SMT pad choice.

The size of the pad is typically close or identical to the package pad size. This provides for balanced stress during thermal cycling, helping to maximize fatigue life. Typical Amkor pad sizes are listed in Table 3.

Package Type	Pitch	Pad Size	Ball Diameter
PBGA	1.5mm	.63mm	.76mm (.030")
PBGA	1.27mm (.050")	.63mm	.76mm (.030")
PBGA	1.0mm	.45mm	.50mm (.020")
SuperBGA®	1.27mm (.050")	.63mm/.58mm	.76mm (.030")
SuperBGA®	1.0mm	.55mm	.63mm (.025")
*fle*XBGA™	1.0mm	.45mm	.45mm (.018")
*fle*XBGA™	0.8mm	.30mm	.45mm (.018")

Table 3 - Amkor BGA Ball Pad Designs

3.2 PTH Isolation

No matter what shape or definition technique is used for the mounting pad, the isolation of the PTH from the mounting pad is an important feature.[3] If the PTH is contained within the mounting pad, solder can and will wick down the PTH. The amount of solder that wicks depends on many factors, including

PTH finish and coating variations. Because of this, the results are somewhat unpredictable. Some solder joints may be unaffected, while others will be starved to the point of creating opens. The worst result is a partially starved joint with severely reduced cross section. This joint can have significantly lower fatigue life and result in early system failure. Because the quality of the solder joint is guaranteed by control rather than inspection, designs/processes that result in random distributions are generally considered unacceptable, and the PTH-in-pad design is not recommended.

3.3 PCB Quality Issues

The assembly yield of all surface mount components is dependent on several common factors. One of those factors is the quality of the incoming PCB. Once an optimal design has been understood and set, it is then a question of understanding and controlling (where possible) the variability of PCB properties as delivered from the manufacturer.
In specific, two items bear special consideration: planarity and solderability.

The sensitivity to PCB planarity is directly dependent on the package size. Typical PCB manufacturing specifications allow up to .010mm/mm (1%) of warpage. If one envisions a 40mm component, this would equate to nearly 0.4mm of warpage in the area under the package. It is obvious that no large body (>20mm) component, be it peripheral leaded or BGA, would consistently solder to a site with this level of bow. However, the difficulty lies in determining a "right" number for control. In general, attempts to control board suppliers to tighter tolerances have met with resistance, as it is not a normally measured variable. The conventional wisdom seems to be "we know all the boards are inside the .010" limit, so measurement is unnecessary". This is sound process control procedure, given the wide nature of the limit. If the limit is to be tightened, then sampling plans would have to be implemented, scrap rates determined, etc., all of which will (somewhat artificially) drive up the cost.

The same problem exists for solderability. It is impossible to test and guarantee solderability on every BGA site. At the same time, a single pad with wetting problems can result in disaster. If pads were simply non-wetable, the problem of consistent solderability problems might be manageable. The result would simply be multiple opens and high rework rates. However, there is the possibility that the site may be nearly non-wetable, resulting in a small cross-section solder joint that would result in reduced thermal cycle fatigue life. Therefore it is necessary to have the lowest level of solderability defects possible.

The question is, how does one control these PCB variables if they cannot be directly measured? The answer lies in process control data. The best solution is to buy boards from suppliers with consistent quality in both areas. The supplier list can be determined by reviewing defect data and correlating against supplier. Eventually a pared list can be produced that identifies only those suppliers whose boards have proven to be adequate to the task. The data used does not necessarily have to be on BGA's. Depending on the level of data currently collected on peripheral components during assembly, enough information may be available to identify the best suppliers already being used.

It cannot be overemphasized that **total cost of ownership** is paramount in PCB considerations. These built in variabilities will reduce the yield of all SMT components. The reason for additional focus when using BGA components is that because the joints cannot be touched up, any yield detractors will drive up rework. This in turn will drive up overall assembly cost. Simply put, **use of the lowest cost board may not result in the lowest cost overall assembly!** There is an optimum point, subtly different for each user and assembler, where saving pennies on PCB costs adds more cost due to rework and component scrap or salvage. It is good practice to buy the best boards that can be afforded. While this premise sometimes raises concerns about purchasing missions, it is clearly in the best interest of the user' s overall bottom line.

4.0 Surface Mount Processing of BGA Components

There are probably as many different "recipes" for surface mount as there are factories. Preferences for solder pastes, screens, tooling, reflow profiles, and other items depend on many factors learned over long periods of time. There is rarely reason to alter in any significant fashion the existing SMT process in order to introduce BGA. However, it is often the case that better understanding and control of a given process is appropriate to obtain the maximum BGA benefit. The following sections are meant to provide suggestions on what to look for at each of the various steps in the assembly process and on overcoming common concerns.

4.1 Solder Paste Screening

4.1.1 Stencil Design

As with PCB design, having the proper stencil design is much of the battle in paste printing. The usual design is to use circular openings over circular holes. As the majority of the eutectic solder will be provided by the package, the exact solder paste volume is not critical, and thus the aperture sizes can be varied somewhat without affecting the final product quality. However, it is best to use a design that results in good paste release and therefore results in consistent paste deposit shapes and volumes. Maintaining a diameter to stencil thickness ratio of at least 3-to-1 to assures good BGA print quality[4], with larger openings providing better print quality. It is also good to use an opening at least as large as the mounting pad to give a wide placement window. The recommended minimum opening in a .006" thick stencil is .024", in a .008" stencil it would be .024". A typical design might be a .028" opening in a .006" thick stencil, going over a .024" pad on the PCB. The printing of small amounts of paste onto the soldermask surrounding the pad has not proven to be a problem in either yield or reliability.

4.1.2 Paste Selection

There is no "right" paste for BGA assembly. As the balls are solder, the flux activity on the ball surface is assured, so long as the flux reaches the ball surface. Both Type 3 and Type 4 mesh pastes have been reported used with equal success. Thus the selection of paste is generally made to fit the entire component mix being assembled, not driven by the use of BGA packaging. It is necessary, however, to insure that all the thermal and environmental requirements of the paste can be met. There are pastes (typically low solid no-clean) that require very specific temperature profiles in terms of ramp rates and peak temperatures. When using such a paste, it is possible that the required profile may not be achievable on all components, including some BGAs, due to a combination of design constraints and reflow tooling. Such cases are rare, but can occur. The best solution is usually a change of paste to allow a wider process window.

4.1.3 Screener Setup

As with paste selection, the screener setup will depend on the entire component mix being assembled, not just BGA. The only BGA consideration is that the apertures are fairly large, and the use of poly

squeegee blades, especially those of lower durometer, can result in significant scooping of paste from the opening. Metal blades and high durometer poly blades have proven to provide higher and more consistent deposits in almost all cases, with metal being the best. Use of metal or high durometer blades is therefore recommended.

4.1.4 Paste Deposit Inspection

The quality of the paste print is **THE** single most important factor in producing high yield BGA assemblies. Defects detected after paste print require nothing more than a strip and rescreen of the PCB. An escape can turn into a defect downstream requiring rework to repair. Thus the most economic area to intensify process controls when beginning BGA assembly is in the paste screening step.

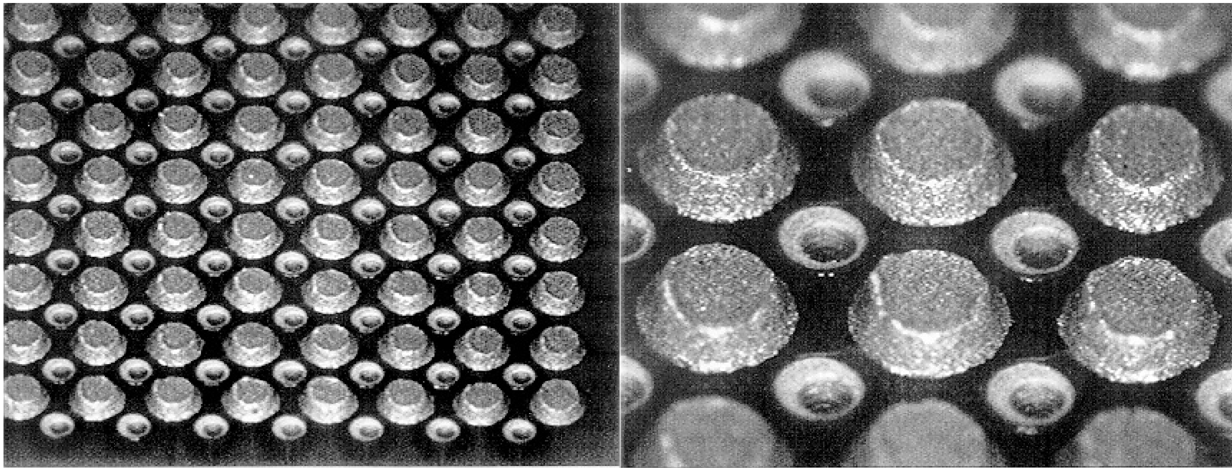

Figure 4 - Typical BGA Paste Deposit (courtesy IBM)

A photo of a typical BGA paste deposit is shown in Figure 4. The preferred shape is a trapezoidal "hockey puck" with a flat top surface. Excessively coned deposits usually indicated paste release problems, and are precursors to plugging and other eventual yield detractors. Because the BGA ball is composed of eutectic solder, the volume of paste is not highly critical to final yield or reliability. However, the presence of adequate **flux** is. The paste is the vehicle to provide the flux necessary to both the PCB and solder ball surfaces to allow soldering. Any situation, such as a plugged stencil hole, that causes paste to be omitted or severely limited on a pad can lead to an open after reflow. Print misregistration or excessive slumping that connects adjacent conductors (either pads or PTH's) is also a source of concern, as a short may be the final result. It is therefore recommended that some sort of paste inspection step be performed, especially during early manufacturing runs. Whether to use visual/microscope inspection, manual measurement, or invest in an automated system depends on the volume to be run and the overall manufacturing philosophy. Once enough data is gathered to feel comfortable with the capabilities of the process, inspection requirements may be altered to fit the overall product requirements for cost and throughput.

4.2 Placement

BGA packages have shown to exhibit excellent self-centering properties. Because of this, wide variation in placement location is accommodated during reflow of the solder joints. The general rule for BGA packages is that the placement be "half on pad". This is illustrated graphically below.

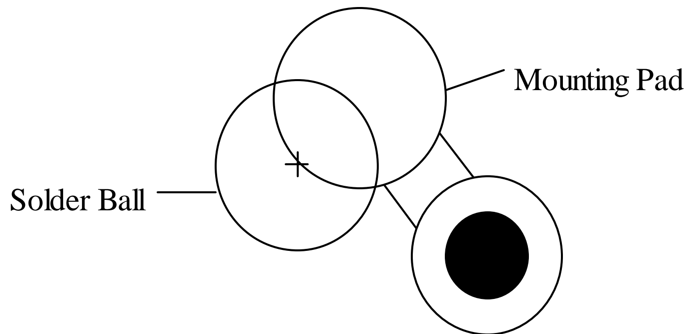

Figure 5 - Ball to Pad Placement Misregistration

The illustration shows an important point. To achieve half on pad, one cannot simply specify half the pad dimension as the accuracy required in X and Y directions. Half the pad dimension is the **radial** accuracy required, or the square root of the sum of the squares of the X and Y misplacement accuracy. For a .024" diameter pad, this translates to ±.0085" in X or Y. Most common placement tools used for flatpack or other non-discrete placement have much better accuracy than necessary, and placement variability is rarely an issue for BGA.

4.3 Reflow

Reflow of BGA packages is in itself nothing new from standard SMT reflow. No "magic profiles" exist to achieve best yields, nor is there a "right" type of furnace or heat to produce the best joint. Both IR and convection furnaces are used for BGA assembly. Convection generally provides more even heating, so may be necessary if thermal spread in IR is too great. Likewise, both air and nitrogen atmospheres can be used. Nitrogen reduces probability of soldering problems, but final joint shape and reliability are not necessarily enchanced by using nitrogen. The profile and other reflow parameters are determined by the choice of solder paste. Every paste manufacturer provides a recommended thermal profile for their products. This guide, or any other internal specifications used for general SMT profiles within a factory, should be applied to BGA packages as well.

There are several special considerations relative to reflow that do come into play with BGA packages. The first is process control. While the profile does not need to be custom for BGA packages, the control of the reflow does. It is not possible to inspect the final solder joints to insure that all of them

have reflowed properly. Thermal profiling is critical, as it is the ONLY method that will insure that all joints on a BGA package are completely reflowed. The thermal profile should be checked for **EVERY** new board design. If there are multiple BGAs on a single board, it is recommended to check the profile of all of the BGA sites. Differences in surrounding components, board cross-section, part density, and BGA part design all will result in varying temperatures under the BGA packages. There will also be a temperature difference between the edge of the part and the center, usually with the center balls lower in temperature. It is therefore recommended that more than one thermocouple per part be used to insure part temperature. The thermocouples should be located within the solder joints themselves for most accurate readings. The simplest method for achieving this is to reflow the thermocouple tip into the ball on the package, snake the wire out between balls, then manually place and reflow the package.

The second limitation that must be taken into account at reflow is moisture sensitivity. Most BGA packages are classed higher than JEDEC Level 1 for moisture sensitivity, meaning that if allowed to absorb moisture beyond a certain point, popcorning will occur during reflow which could lead to premature failure of the part in operation. Most specifications for moisture sensitive components recommend that the maximum temperature during reflow be limited to 220°C. This temperature limit comes from the specific test procedure used to classify the packages. Thus the profiling must insure not only that all solder joints meet a minimum temperature requirement, but also that the package body stays below the 220°C limit as well.

Finally, bowing of the PCB within the furnace is a general SMT concern, but especially for BGA. Depending on the PCB thickness used, significant bowing can occur within a furnace if the PCB is not properly supported. As the PCB bows under the BGA, the distance from package to board becomes variable. The result is opens or very irregular (hourglass) solder joint shapes, especially in the center of the package. Peripheral row packages are much less susceptible to bowing induced defects. This can be especially critical to consider when using very thin PCB's, like those used in PCMCIA cards and portable communication products.

4.4 Cleaning

Cleaning of BGAs has proven to be well within the capabilities of most aqueous or semi-aqueous systems. In general, no special steps need be taken to insure cleaning beyond those typically practiced in the SMT factory. Surfactants can be used if specific problems occur to induce better water penetration and flow. The only unique item to BGA cleaning is insuring dryness. Because of the many close spaces, water tends to be trapped beneath the package. In some instances, a factory standard drying profile does not completely evaporate all the water. In this case, the water can react with the solder balls as it dries, resulting in precipitate formation. These precipitates can act as humidity bridges for short formation between biased points.

To determine if a drying profile is adequate, simply blow compressed air under one edge of the package upon exiting the cleaner. A spray of water indicates better drying is required. If this is necessary, usually bumping the drying temperature or slowing down the passage through the drying zone, or a

combination of both, is all that is required. Once a satisfactory setup is achieved, it is usually not necessary to monitor on more than an occasional basis.

4.5 Subsequent Processing

There are additional process steps that may be practiced after BGA attach. In some cases, these processes can affect the product quality. One such process is wave solder. When using the "dogbone" footprint, the PTH is located very close to the mounting pad. During wave solder, this hole is filled with molten solder. The short heat path to the solder ball results in secondary reflow occurring in most if not all BGA solder joints.[5] Secondary reflow can result in opens being induced in previously good joints. To eliminate secondary reflow, the PTH's must be blocked in some manner to prevent the solder from filling the holes. The two most frequent methods are mechanical wave shields and tape. If water based fluxes are used, water soluble tape that dissolves in the cleaner can minimize the impact of taping operations.

Another process that is often used is a second SMT pass with the BGA component on the bottom side (upside down). Tests to date on BGA products have indicated no significant problems with BGA, as long as the same care is exercised as during initial reflow.[6] Surface tension of the solder is generally adequate to hold the part in place. However, with the advent of depopulated arrays on large body parts, it is clear that at some point weight will overcome the surface tension, requiring adhesive to keep the BGA attached to the bottom side.

5.0 Final Joint Quality/Inspection

While the best practice in using BGA packages is process control for quality rather than inspection, it is obviously necessary to understand the methods of characterizing the solder joints in order to establish the process control limits in the first place.

5.1 Visual Inspection

Even though BGA solder joints cannot be visually inspected 100%, some information can be gathered from a simple visual inspection or microscope check. Alignment is easily verified. If a white mark border is accurately printed on the board surface, it is often adequate to compare the part location within the outline. The self alignment of the parts will result in centering on the wrong row/column/both if the part is badly misplaced, and the shift is gross enough to detect by this simple visual check.

Tilting the board on edge to view the edge solder joints can also be used as a rough check. If the thermal profile shows only slight temperature variation between edge and inner joints, and the outer joints are all properly formed and reflowed, it is likely that all joints are equally well reflowed.

5.2 Cross Section

Obviously, one cannot cross-section a shipable part. But as a process determination tool, cross sections can be invaluable. The cross-section of the solder joint will show self-alignment, standoff height variations, voiding, or other irregularities in formed joints. It is also the best tool for failure analysis when an open occurs, as it can show the state of the open joint relative to surrounding joints, which is generally necessary to trace the root cause of the problem.

5.3 X-Ray

X-ray is a powerful tool for understanding all aspects of BGA solder joints. Originally only capable of detecting shorts, today's X-ray systems do an excellent job of real-time imaging of BGA solder joints. Opens and shorts are both easily detected, as are serious voiding problems. Most X-ray tools can detect down to .001"-.002" voids. The manufacturers would have one believe that these voids are a defect to be controlled or eliminated. In truth, studies have shown that 1) voids will always exist in BGA joints[7] and 2) unless they are very large (> 0.5 ball diameter) they do not have an effect on ball fatigue life.[8] Because of this, X-ray is seen primarily as a failure analysis tool for functional parts. If a problem is detected electrically with a BGA, using X-ray to determine if the problem is process or part related can be an excellent method of improving the integrity of yield data. By screening out fails that are not related to the surface mount process, it allows the process engineers to focus only on areas where improvement is necessary.

6.0 Rework of BGA Packages

The SMT yields of BGA packages are very high, but there will still be requirements for rework of components. Component defects, SMT defects, or other functional problems all will require rework of the package from the PCB. In fact, the biggest rework driver is often not a defect at all, but an engineering change. Upgrading components on previously assembled PCBs can involve thousands of components, which is often the vast majority of the total rework events processed.

Rework is the process of removing a component from a PCB, and replacing it with a **new** component. The removed part is NOT immediately reusable. The shape and volume of the solder balls will not be the same as a new package. If module reuse is desired, a separate process for replacement of solder balls should be developed.

6.1 Rework Tooling

There are a number of systems currently on the market for reworking BGA' s. All of the major rework equipment suppliers have designed systems for rework of BGA packages. Most systems utilize hot gas to locally heat the BGA and reflow the solder balls for removal and replacement. Backside heating is a common feature of all BGA rework tools as well. While some systems attempt to backside heat with local hot gas impingement behind the package, this often results in large scale PCB warpage during processing, especially if the card being worked on is large. The recommended method is to use a global preheat that brings the entire PCB up to some specified temperature (usually 80-100°C) prior to using local heat on the package.

6.2 Rework Processing

The rework process itself is relatively simple, but can be time consuming, especially for inexperienced operators. The biggest challenge of BGA rework is the requirement of a controlled process. Often SMT rework is accomplished with very coarse methods like hand placement and soldering iron reflow. Such approaches will not work for BGA packages. If the process used to rework the package is not understood and well controlled (similar to initial attach), the result will likely be additional defects and a subsequent additional rework. The characterization of the rework process and acceptance of process controls rather than technician skill for quality control is generally the biggest single difficulty in implementing BGA rework.

To perform BGA rework, there are essentially four steps; removal, site preparation, flux or solder paste application, and replacement/reflow.

6.2.1 Component Removal

The removal of the BGA package is accomplished using a hot gas tool. Generally, a custom head is used that is sized to the part being removed. Proper sizing of the gas head reduces the thermal impact on adjacent packages during the removal process. Determination of the correct tool settings are highly

dependent on the tool being used, the package being removed, and the PCB the package is attached to. However, determining the settings is a straightforward exercise in profiling, similar to initial reflow. By using a profile assembly with thermocouples mounted in solder joints (see section 4.3), tool settings can be determined that assure that all solder joints reflow. The part is then generally removed by a vacuum nozzle within or integral to the hot gas head. (When profiling, it is wise to shut the vacuum off so the part is not removed to avoid destroying the profile card.) It is necessary to control the pressure of the head down onto the part during removal. If pressure is applied after the solder balls are melted, the solder will be pressed between the "plates" of substrate and PCB, resulting in bridging that will have to be removed manually. Two possible solutions are to establish the head height prior to reflow and control the stroke, or to place shims under the edges of the part to prevent collapse.

Another consideration during removal is the possible affects on neighboring components. If there are other SMT packages near to the package being reworked, it is recommended that the solder joint temperature on them be monitored as well. If the temperature on the adjacent joints gets to near reflow temperatures, then some simple shielding may be necessary to reduce direct impingement from the rework gas flow.

As mentioned previously, preheating of the PCB assembly is good practice in rework of BGA packages. There are several advantages. First, it can reduce heating time required using the rework head. In cases where one PCB assembly can be preheated while another is reworked, cycle time can be greatly reduced. Second, more uniform profiles are achieved. One of the challenges of profiling is the temperature spread between center and edge solder joints. Preheating reduces the spread by bringing the baseline temperature closer to the reflow temperature. Another advantage of preheating is the reduction in PCB warpage during processing. Warpage is caused by the higher local temperature at the site than the surrounding area. By raising the PCB assembly temperature, this mismatch, and therefore warpage, is reduced. Finally, preheat reduces problems of adjacent components, as it allows shorter gas heat times or lower gas temperatures to be used, thus reducing risk of affecting neighboring solder joints.

6.2.2 Site Preparation

Once the part is removed, the remaining site needs to be prepared to accept the new component. The removal process generally leaves varying amounts and shapes of solder on the mounting pads. To insure maximum probability of success, it is recommended that all sites be made as uniform as possible prior to reattach of the new package. Solder removal can be accomplished in a variety of methods, from solder wick to custom solder vacuum tools. No matter the method, the desired result is the same - similar mounting surfaces across all mounting pads.

6.2.3 Flux/Paste Application

In order to solder the new package to the site, the addition of flux is necessary. Like initial attach, solder paste can be used as the flux carrier. In this case, the simplest method is to simply use a small,

custom stencil to apply the paste locally. The system manufactured by Mini Micro Stencil is a good example of an inexpensive local screening solution.

Unlike initial attach, it is often simpler to simply apply flux to the site rather than paste. Flux has the advantage of being able to be simply applied with a small brush, sponge, or squeegee. The flux should be of the tacky variety to hold the part in place once it is placed prior to reflow. The other considerations for flux application are flux volume control and preheat. Flooding the area can provide a path for balls to bridge, so a thin, consistent coat is preferred over a mass area glob. If no-clean flux is used, overfluxing can result in contamination problems. Since many no-clean fluxes do not become inert until they are heated, excess flux that is spread outside the zone affected by rework heating will remain active, and can quickly become a corrosion or migration problem. Limiting the flux to just the component area or use of water soluble materials is the solution to such problems. Finally, the preheat must not drive off the flux or reduce it' s tack properties prior to placement of the new part.

The decision to use paste versus flux usually comes down to a question of exact duplication of the initial assembly process. The ball reflowed with only flux will have slightly smaller solder volume than the initially attached ball (due to the solder added with the paste), and therefore will result in a slightly (usually about .001") lower standoff height from the PCB. As the thermal cycle reliability of BGA components is directly related to standoff height, the life of the flux only reworked solder joint will be slightly lower than the joint created with paste. If this is a concern in the use application, then a paste addition method should be used during the rework process.

6.2.4 Replacement and Reflow

Placement is generally accomplished by the reflow tool using split prism optics. The part is aligned over the site and placed down on it. The pressure is released prior to heating. If the part is held under pressure during reflow, the balls will be compressed and liquid solder squeezed between the substrate and card, resulting in bridging similar to the problem described during removal.

Once the part is placed, the hot gas is applied to reflow the solder joints. All the stipulations that applied to both initial attach profile and part removal apply to the replacement profile. The most important are; all joints reach some minimum (user defined) reflow temperature, part body temperature be limited to <220°C, and neighboring component solder joints not be detrimentally affected. As with initial attach reflow, the development and control of the thermal profile is the single best guarantee of final quality. The profile should be checked on each new PCB design for each package type and adjusted as necessary. There is NEVER a "one-size-fits-all" machine setup for BGA rework!

7.0 Summary

The conclusions of the work on BGA assembly to date are overwhelming. Every SMT user who has invested the time to understand the processes and accept the lack of inspectability has embraced BGA' s as far superior to the ultra-fine pitch alternatives. Even the additional challenges in rework are not a serious deterrent when the overall improvement of throughput and quality are placed on the opposite scale. Hopefully this document has provided an introduction to the path that will lead to BGA success. Welcome to the future of SMT......... with BGA!

For additional information on BGA surface mount concerns, contact Paul Mescher at (602) 821-5000.

8.0 References

1. S. Bolton, A. Mawer, and E. Mammo, "Influence of Plastic Ball Grid Array Design/Materials Upon Solder Joint Reliability", *Intl. Journal of Microcircuits and Electronic Packaging*, vol. 18, no. 2, pp. 109-121, 1995.
2. A. Mawer, S. Bolton, and E. Mammo, "Plastic Ball Grid Array Solder Joint Reliability Considerations", *Proceedings of the 1994 Surface Mount International Conference*, pp. 239-251, 1994.
3. A. Holiday, et al, "Designed Experiment to Evaluate Assembly Defect Drivers for Plastic BGA Packages", *Proceedings of the 1995 Surface Mount International Conference*, pp. 305-312.
4. J. Mearig, "An Overview of Manufacturing BGA Technology", *NEPCON West Proceedings*, pp. 295-299, 1995.
5. M. Petrucci, C. Ramirez, and J. Brown, "High Volume SMT Assembly of High Pin Count PBGA Devices", *Proceedings of the 1995 Surface Mount International Conference*, pp. 297-304.
6. T. Ejim, et al, "Designed Experiment to Determine Attachment Reliability Drivers for PBGA Packages", *Proceedings of the 1995 Surface Mount International Conference*, pp. 385-392.
7. W. O' Hara and N.C. Lee, "Voiding in BGA", *Proceedings of the 1995 Surface Mount International Conference*, pp. 279-285.
8. R. Johnson, A. Mawer, T. McGuiggan, B. Nelson, M. Petrucci, and D. Rosckes, "A Feasibility Study of Ball Grid Array Packaging", *NEPCON East Proceedings*, pp. 413-425, 1993.
9. P. Mescher and G. Phelan, "A Practical Comparison of Surface Mount Assembly for Ball Grid Array Components", *Proceedings of the 1994 Surface Mount International Conference*, pp. 164-168, 1994.
10. S. Fauser, C. Ramirez, and L. Hollinger, "High Pin Count PBGA Assembly: Solder Defect Modes and Root Cause Analysis", *Proceedings of the 1994 Surface Mount International Conference*, pp. 169-174, 1994.
11. C. Ramirez and S. Fauser, "Fatigue Life Comparison of the Perimeter and Full Plastic Ball Grid Array", *Proceedings of the 1994 Surface Mount International Conference*, pp. 258-266, 1994.
12. P. Mescher and G. Phelan, "Ball Grid Array Packages: 500 I/O at Less Than 1ppm per Lead? You' re Kidding!", Workshop presented at 1994 Surface Mount International Conference, 1994.
13. J. Lau, ed., Ball Grid Array Technology, McGraw-Hill, 1995.
14. T. Chung and P. Mescher, "Rework of Ball Grid Array Assemblies", *NEPCON West Proceedings*, pp. 334-345, 1995.
15. D. Banks, T. Burnette, Y. Cho, W. DeMarco, and A. Mawer, "The Effects of Voiding on Plastic Ball Grid Array Reliability", *Proceedings of the 1996 Surface Mount International Conference*, pp. 121-126, 1996.