TOP 5 BGA REWORK CHALLENGES TO OVERCOME (2016)

Bob Wettermann, MIT BEST Inc. Rolling Meadows, IL, USA bwet@solder.net

ABSTRACT

As BGA component package dimensions continue to get thinner as more of them are being used in handheld device applications. End use device requirements include the need to maintain their interconnections, even when dropped, thus necessitating the use of underfill. The increased demand for higher board densities requires neighboring or mirrored devices during the rework process to be more challenging. Higher reflow temperatures of 'lead free' rework increases the pressure for properly shielding the neighboring components. These developments are causing BGA rework challenges. This summary will discuss the most challenging aspects of BGA rework and options for solutions.

Key words: BGA rework, BGA warping, BGA underfill, mask repair, pad repair

Warped BGAs

One of the challenges in reworking BGAs at this time is the warping of the packages as ever thinner packages lead to a variety of problems. A largely noted issue of a warped BGA is the 'Head in Pillow' (HiP) defect (Figure 1). This defect may or may not be detectable during the X-Ray inspection. In this solder joint defect the solder paste deposit will wet the pad, but does not fully wet the ball. This is due to the ball of the package being 'pulled away' during the device warping. The end result is a solder joint with enough of a connection to have electrical integrity, but not sufficient mechanical strength. Without the strength, these components may fail with a small amount of mechanical or thermal stress. This potentially costly defect is not usually detected in functional testing but rather in the field after the assembly has been exposed to some physical or thermal stresses. Warping can also cause bridging and shorts, at either the outer corners or the center of the device during replacement (Figure 2). This warping phenomenon can cause the center of the BGA to bow upwards and the corners down, or vice versa. Sometimes, this is enough to cause the corners to move closer to the circuit board surface. This puts extra pressure on the corner solder joints. Later in the lifecycle of the device, this wmayill cause shorts or cracks.



Figure 1: Head-in-Pillow Defect



Figure 2: Warpage and z height deflection on BGA

There are several 'fine tunings' of the rework process which will mitigate the impact of warped packages. The main way to lessen the impact of warping includes the adjustment of the reflow profile and solder paste chemistry. The reflow profile will help determine which areas of the device may be subject to thermal stress as well as ensuring the flux activity level does not expire. The solder paste chemistry is an integral step in making sure there is plenty of flux activity if the warping occurs over a prolonged reflow cycle. These variables are important to control during the BGA rework process to minimize warping.

By aiming for a very small temperature differential across the entire BGA package -less than 10°C across the entire package is optimal, the impact of device warpage can be greatly reduced in the rework process. One of the methods for insuring a consistent and small temperature gradient across the entire package is to use a programmable multi zone bottomside heating source (Figure 3). Ideally, a multizoned heating source will heat the underside of the board to 100-120°C (lead-free processing temperatures) with the device location elevated to 140°C if the board and components can withstand this. This slight temperature differential will insure that there is not 'undo thermal stress', which may delaminate or warp the board. The heating nozzle (assuming hot gas is used as the reflow source) will bring the solder balls up to $20-30^{\circ}$ C above liquidus for 60-120 seconds.



Figure 3: Multizone bottom heat of BGA rework system

In addition to the proper removal and reflow profile, the PCB will need to properly supported in order to mitigate any effects of deflections on the PCB during the removal or reflow profiles. Board supports should be such that they prevent undue board deflections. An example of such a board holder can be found in (Figure 4). The lack of support further exacerbates warping which may occur.



Figure 4: PCB support on BGA Rework System

Solder paste chemistry can impact the degree to which the device or PCB is warped during the rework process. Solder pastes, with an activation system that is able to provide sustainable high-temperature fluxing, are capable of creating a homogenous connection. This connection goes beyond the ball and the paste alloy interface, which is important for warped devices to form an intermetallic bond. Those that are not able to sustain high-temperature efficacy are more prone to HiP defects.

Neighboring Device Damage

As the density of components on PCBs has increased, the need to protect neighboring temperature sensitive components in the vicinity of the BGA such as ceramic capacitors, crystals and plastic-bodied components such as connectors has increased. These devices must be protected from exposure to heat during the rework process. If they are not, they can become damaged showing either visible or 'hidden' damage. In addition, these components long term reliability may be impacted by this heat exposure, even if their appearance does not show it. While these components may be able to withstand the peak temperature of 260°C (as defined in J-STD-002) there may be medium and long term impacts to its reliability if the IMC layer grows too thick. Neighboring underfilled parts may have the material 'ooze out' as the softening point temperature is below the liquidus temperature of lead-free solder.

There are a variety of shielding options in and around the BGA rework area. The increased liquidus temperature of lead- free solder has driven processing temperatures into areas where sensitive components have significant body temperature and time limitations (IPC J-STD-075 is the standard for maximum time/temperature exposure for all non-semiconductor devices). The intermetallic layer thickness, which to some degree represents the 'mechanical strength' of the solder joint, can become too large with extended exposure to times above liquidus. If it becomes too large it can make the solder joint brittle. This in turn can impact the reliability of the solder joint.

In order to prevent these negative impact on neighboring devices in the BGA rework area, proper thermal shielding is needed. A recent study (Wettermann, 2015) indicated that the shielding effectiveness of 'historical' materials, such as KaptonTM tape and stainless steel, are not as effective as more modern materials such as shielding clay gel (Figure 5) and ceramic nonwoven ceramic materials. In order to fully protect a device from thermal damage the study pointed out that the clay gel material is over two (2) times as effective as KaptonTM tape as a thermal shield at a close distance. and (3-4) times as effective at further distances. The ceramic non-woven fiber material is nearly as effective as the gel in terms of its heat withstand properties, but does not need to be cleaned off after use.



Figure 5: Clay Gel protects and shields devices from thermal damage

Trending devices, such as smart phones and tablets, use underfill so that the BGA package can withstand drop testing requirements without damaging the solder joints. The challenge for a rework technician is the pliable nature of the underfill. This typically means the tack properties, even when above the softening point, make for a mess underneath the BGA. Even if the underfilled BGA can be pried off the PCB, the mechanical force exerted can potentially damage the device or board.

Underfill Rework

The softening point of the underfill is less than that of the reflow temperature of the PCB device. This means that under the BGA, as well as any non-protected device, the underfill softens and expands prior to the solder reaching liquidus state. This along with the resulting tackiness presents challenges in the removal of the device. During removal the underfill will 'squirt out' as local pressure pushes out the solder when it reaches reflow temperature. The result is a mess in the area of the underfilled device being reworked. Due to mechanical force, either through prying with a lever arm or a specialty nozzle in removing the underfilled device from the PCB, the board may end up with extensive damage. Due to the mechanical force, either through prying with a lever arm or a specialty rework nozzle in removing the underfilled device from the PCB, the board may end up with extensive damage (Figure 6).

In addition to this prying action causing damage to the PCB there may be damage around and underneath the area of the device in removing the underfill from the PCB. This damage may be caused by the solder wick or nozzle which may scratch or damage the pads of the solder mask when removing remnant underfill. In addition the tack adhesive strength of the underfill may be of such a high value that the pads are pulled off of the board. This phenomena may be most pronounced for pads that are no-connects underneath the BGA.

One of the ways to overcome the extensive damage caused by preparing the BGA site location after device removal is through the use of a high speed milling system. In this approach no direct heat is used to soften the underfill rather a high speed milling operation will "grind off" the cured underfill material as well as the remnant solder balls. This requires precision in this mechanical operation (Figure 6).



Figure 6: Mechanical Grinding off of underfilled BGA results

By not grinding far enough, it will leave too much of the underfill at the BGA device location making the pads non solderable. Care must be taken during this rework method to ensure the mechanical vibration, and stress of rework, does not cause a decrease in the reliability of the PCB.

Extensive Solder Mask Damage

Solder mask damage underneath a BGA location, occurs due to several reasons. This damage is in the form of missing mask or a breakdown in the mask adhesion to the PCB. This can be caused by using solder braid during the site prep process, an uncontrolled heat source to remove the BGA, an abnormally high number of heat cycles applied to the PCB, or by poor initial solder mask adhesion. The resulting problems upon component replacement is solder flowing down the dog bone pattern and 'starving' the solder joint/flow underneath. The poorly adhered solder mask will cause solder shorts, or other soldering anomalies.

Solder mask can be repaired via a variety of techniques, each with its own advantages and disadvantages. A simple technique for "spot" solder mask repair is using a repair 'pen' (IPC 7721 2.4.1) which is typically cured by air drying or a bakeout cycle. Another method, as outlined in IPC 7721 2.4.1, is via the use of liquid solder mask. This can be spread onto the areas that are in need of repair. It is then either heat or UV-cured. Another technique which repairs the mask underneath the BGA, is a stay-in-place stencil. This serves as a reliable way to place the BGA while simultaneously repairing the mask. Lastly, there is the mask repair stenciling technique. This method saves repair time as an ultra-thin stencil defines the area where the mask is repaired. Replacement mask material is then squeegeed into the apertures and then cured. Post curing, the stencil is peeled away, leaving hardened replacement mask in its path.

Solder mask repair pens are labeled as 'simple-to-use PCB repair tools', which can replace solder mask. The pens can be simply 'drawn' across the areas requiring solder mask. The liquid oozes out of the soft-tipped pen and then can be air or heat-cured. The pen tips themselves are fairly large with respect to modern BGA pads site pitch, making it a challenge to precisely dispense the mask material. Due to the porous nature of the dispensing tip, it has a tendency to pick up debris from the board (including flux residue, remnant mask and cleaning agents). These soaked up debris then can re-contaminate other areas of the board. The skill level for someone making solder mask repairs using this technique will be at the advanced level. However the repair quality will tend to be low.

The most common method used for IPC Class 2 and 3, if allowed, solder mask repairs is via the use of liquid replacement solder mask. In this method, a skilled technician selectively brushes/dabs a small amount of solder mask onto the damaged areas underneath a BGA (first removing the BGA) being careful to not get the material onto the component or solderable areas. Depending on the type of mask used, this mask can then either be heat or UV- cured. On the fine-pitched devices of today the exacting placement of the replacement mask requires a great deal of dexterity, requiring the use of a microscope.

As a time-saving enhancement to the above technique, a stencil can be designed and fabricated using the GERBER files of the board. This will help selectively cover solder mask underneath a specific location underneath the BGA. The stencil is peeled off of its carrier backing, aligned, then placed on to the PCB surface. Replacement mask is squeegeed with a micro squeegee across the surface of the stencil and then cured. After curing, it is removed in an ionizing environment as not to cause any latent static charge damage to the components in the area. This method, while time saving for multiple boards requiring the exact same area of repair or for a very complex large area of repair, requires a repair technician with an advanced skill level.

The last approach to repairing solder mask is through the use of a semi-permanent stay-in-place BGA stencil. This stencil, while having the benefit of being a simple way to place a BGA, also provides isolation between pads and prevents shorting between the IO. This will fix the standoff height between the base of the BGA and the PCB to control collapse height, while serving as a mask "band-aid". Once in place the stencil acts like a solder mask repair stencil. (Figure 7) This allows even the beginning repair technicians to repair damaged solder mask underneath the BGA.



Figure 7: Stay-in-place stencil for solder mask repair

Pad Damage

Another common trouble spot is the potential for pad damage while the BGA is being reworked. Pad damage is usually the result of improper site preparation which can be a result of numerous processing problems. Items on this list are contaminants on the solder wick, improper tip temperature, too much pressure of the soldering iron on the solder wick material, improper solder tip selection or expired flux material. Improper temperature profiles found in the BGA removal process would also be a contributing factor to pad lifting. Finally, there may be cases where the original adherence of the pad to the laminate was insufficient due to problems in fabricating the PCB.

The proper procedure for the preparation of BGA pad sites, after a BGA has been removed is outlined in IPC 7711 procedure 4.1.3. and releveling of the pads is found in 4.2.1. The improper method for pad leveling involves 'scrubbing the deck'. This occurs when the wick is pushed back and

forth along the underside area of the BGA pad site. This can result in the mask being scratched and pads being bent up, or lifted off. The proper technique (Figure 8) calls for the solder braid to be moved in an 'up and down' fashion onto the pads thereby not scratching the surface or 'prying' the pads up.



Figure 8: Proper site preparation technique eliminates mask and pad damage

One way to overcome the problems associated with these contact types of pad prepping processes which rely on operator skill is to use a non-contact method for site preparation. Several higher end rework systems are now equipped with programmable non-contact, scavenging systems. In these systems a heated nozzle comes in to close proximity of the PCB in order to reflow the solder, while at the same time following the contour of the PCB surface and 'vacuuming up' the molten solder from the board. This way results in no damage to the pads or board surface. In addition, the skill level of the rework technician is removed as a variable. The downside of this approach is lessened throughput as both the programming time of the scavenging system and the extended cycle time of the scavenging operation makes this somewhat cumbersome for most rework.

The proper BGA removal profile needs to be developed to make sure the removal of the BGA will not unnecessarily cause pad lifting for removal of the BGA. The process for developing such a profile is similar to that of profiling a PCB for assembly, only done in reverse and on a smaller scale. First, a solder sample board is analyzed in order to determine the proper location of the various thermocouples. Typically one is embedded in the die of the BGA, One each in two corner balls, one or more on the neighboring parts and one or more others scattered throughout the BGA ball pattern depending in the idiosyncrasies of the PCB's thermal characteristics. Attachment of the thermocouples includes drilling and gluing them into the board, the solder balls, and the actual die itself. After this the technician 'dials in' a reflow profile based on experience. Solder profiles are developed and the temperatures at these locations are measured. From there, adjustments to the profile, removal of neighboring parts, or proper heat shielding will fully optimize the profile for minimized reflow time and damage to the neighboring components, while gaining the most

consistent collapse of the solder balls. Bottom side heat is especially necessary with thermally massive boards or for those processed using lead-free solder. This will prevent thermal damage to the PCB if the rework process is being done in higher liquidus temperature solders such as leadfree. Multizone bottomside heaters are sometimes necessary for higher layer count boards and to maintain a smaller temperature differential over the entire component area. Finally, the proper board support will help prevent warping of the board.

If the pads have been turned up by more than one pad thickness or have been partially or completely ripped off of the board there are two basic methods of repair as outlined in the IPC 7721 board level repair guidelines. In one method a 2-part epoxy is used while in the other a pre-adhered dry film adhesive is used.

The 2-part epoxy (IPC 7721 2.6-epoxy mixing) method for pad repair is outlined in the IPC-7721 Procedure 4.7.1 Surface Mount Pad Repair, Epoxy Method. It is the most reliable method for pad repair. This requires a higher degree of skill and patience by the technician. In this method the site is prepped for the new pad by removing the existing pad or pad/trace combination with a sharp knife. After cleaning the area, the proper replacement pad or pad/trace is soldered to the existing board using a lap solder joint. The 2-part epoxy is mixed per the manufacturers recommendations on a flat plate where the mixture can be seen (a glass plate is easiest). Using a wooden orangewood stick, a thin layer of this 2-part mixture is then placed underneath the replacement pad or pad/trace area and clamped using a nonstick surface clamp to hold the replacement pad or pad/trace to the PCB during the curing process. Curing per the epoxy manufacturer's direction, which is usually accelerated to some extent by elevated temperature conditions, will then complete the process. Prior to implementing the final step of overcoating with solder mask over the trace area and/or up around the periphery of the pads to gain some adhesive strength, the continuity between the various conductors involved in the repair process should be confirmed.

The dry film method for pad repair is outlined in IPC7721 Procedure 4.7.2 Surface Mount Pad Repair, Dry Film Method. It is the cleanest of the two approaches and requires a high degree of dexterity to perform. The pad and trace preparation process as described previously is used in this method. However, when cutting out the replacement trace and film, great care needs to be taken in making sure that the replacement cut out dry film is adhered everywhere on the bottomside of the foil. If a trace section needs to be lap soldered to that section on the board, the technician needs to make sure that the overlapped section has the dry film on the board. Once it is affixed in position, the replacement materials will have the heated tip of a curing iron sized over the bonding area. While being adhered, the curing temperature needs to be confirmed, as both the correct temperature and pressure needs to be applied in order to get the cure right. Cleaning , testing, and inspection are the same as in the case of the epoxy repair described above.

CONCLUSION

The above discussion is a snapshot in time with respect to the top BGA rework challenges. These challenges, in no particular order, were that of: Excessive BGA warping due to thinning packages, shielding of neighboring devices in an era of increasing component density, underfilled BGA device removal and replacement, as well as the repair of solder mask. In addition to these current challenges others including reworking ever-finer devices pitches and integrating lower reflow temperature solders are rework trials that will be added to this list soon.

REFERENCES

[1] Gregory L. Tonkay, Robert H. Storer, Ronald M. Sallade, and David J. Leandri, "Critical Variables of Solder Paste Stencil Printing for Micro-BGA and Fine-Pitch QFP", IEEE Transactions on Electronics Packaging Manufacturing, Volume 27, No 2 April 2004.

[2] Garcia, Omar, et el, "Selective Reflow Rework Process", Apex Proceedings, 2015.

[3] Wettermann, Robert and Gaynor, Adam "Effectiveness of Different Materials as Heat Shields during Reflow/Rework", Apex proceedings, 2015.

[4] Luetzlow, Norbert and Jahal, Kiuldip "Quantifying Performance of Mechancial and Chemical Processes for Pretreatment Prior to Solder Mask Application, Apex proceedings, 2013.