

BGA REWORK

A COMPARITIVE STUDY OF SELECTIVE SOLDER PASTE DEPOSITION FOR AREA ARRAY PACKAGES

Ray Cirimele
BEST, Inc.
Rolling Meadows, Illinois
rcirimele@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. Many prefer to use paste flux only during rework of Area Array devices, there are still instances where selective solder paste deposition is either required or highly beneficial. Just as in the original manufacturing process, solder paste deposition is the biggest yield detractor in the Area Array rework process. An evaluation of the strengths and weaknesses of the current methods for selective deposition of the solder paste revealed that the process has not changed much since its introduction. This paper will introduce a new method of selective solder paste deposition utilizing a “remain-in-place” semi-permanent stencil that is a drastic departure from the existing methods. A comparison of the advantages and disadvantages of the various deposition methods is also provided. This comparison will show that the new method of deposition results in reduced cycle time and increased yield. Although the methods and materials apply to almost all Area Array package types, most references will be to BGA devices as they are the most common.

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

BACKGROUND

There are currently two methods used for the rework (removal and replacement) of BGAs. One method uses solder paste the other uses paste flux. When reworking BGAs that have high temperature solder balls, it is necessary to use solder paste for the interconnect between the package balls and the lands on the printed circuit board (PCB). When reworking BGAs that have eutectic solder balls, either solder paste or paste flux may be used.

There has been much controversy over whether BGA solder joint reliability is better when processed with solder paste versus paste flux. Some studies^{1,2} indicate that the solder joint shape may have a greater impact on the solder joint reliability than the volume. There are still some claims¹ that greater solder volumes (given the correct solder joint geometries) result in greater solder joint strengths and greater reliability, however, the additional amount of solder

volume provided by the solder paste at a typical paste print thickness is usually minimal compared to the volume of solder provided by the eutectic solder balls and has little effect on the joint characteristics.

Regardless of the reliability claims for either solder paste or paste flux, the use of solder paste can improve the yield of the BGA rework process. Coplanarity can be critical during the rework process. Many things can affect the coplanarity. If the BGA has solder balls that are not coplanar it can result in a “open connection” (a ball that has not contacted and

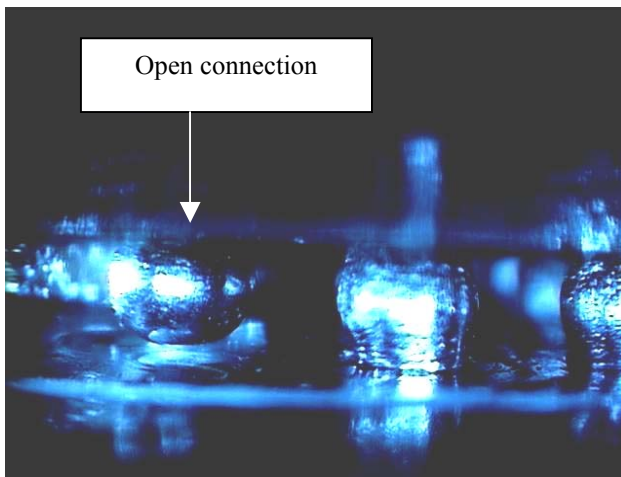


Figure 1. An open connection caused from poor coplanarity and the use of flux instead of paste.

wetted the land while surrounding balls have made contact). Variations in surface finish height, warp of the board or area, and warp or bow of the device itself can create coplanarity problems. The use of solder paste can help to overcome variations in solder ball height. The use of solder paste may also improve wetting as a result of a greater area of intimate contact between the ball and the paste and the land and the paste. Solder paste may provide better “tack” than flux alone. On newer technology PCBs, the utilization of drilled (usually laser) micro-vias in the center of the BGA land requires the use of solder paste to reduce voiding and to prevent loss of the solder ball volume as a result of filling the via.

With the knowledge that the use of solder paste will improve the yield of the BGA rework process, the assumption could be made that most BGA rework was

being accomplished with solder paste. This assumption would be wrong. The reason that more BGAs are reworked with paste flux is “ease of use”. The traditional methods of solder paste deposition require more skill and more attempts to produce an acceptable paste deposit than the methods for flux application. What is needed is a method of depositing solder paste on the first try that is quick and easy.

SOLDER PASTE DEPOSITION METHODS



Figure 2. Various types of stencils.

Although there are many ways to replace the Area Array devices using flux, solder paste, or solder bumps, only the four most common methods of solder paste deposition will be discussed.

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

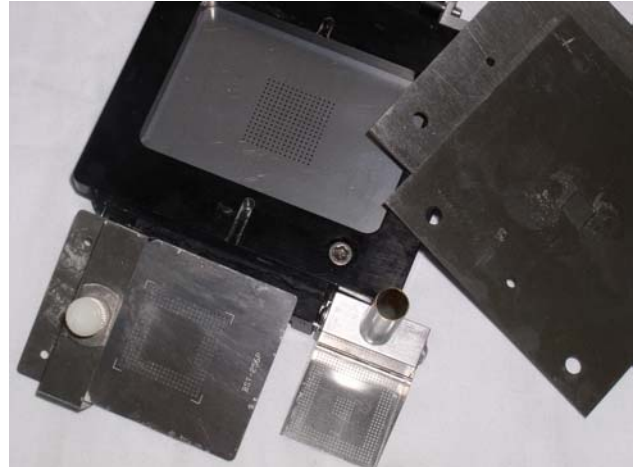


Figure 3. Various metal stencils.

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, then the stencil must be held in place in a manner that ensures intimate contact with the PCB. Next, a squeegee is 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.

Flexible Removable 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

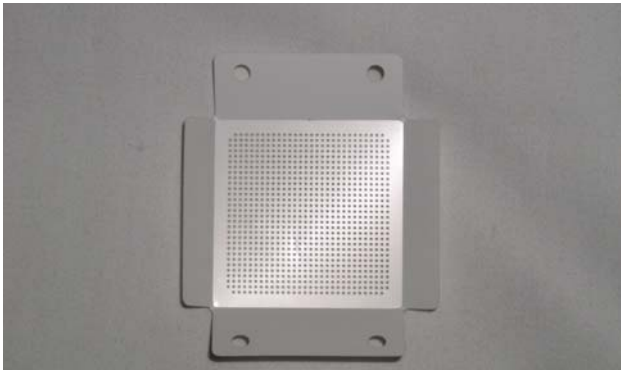


Figure 4. Removable stencil.

step to using the flexible stencil is to remove the 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

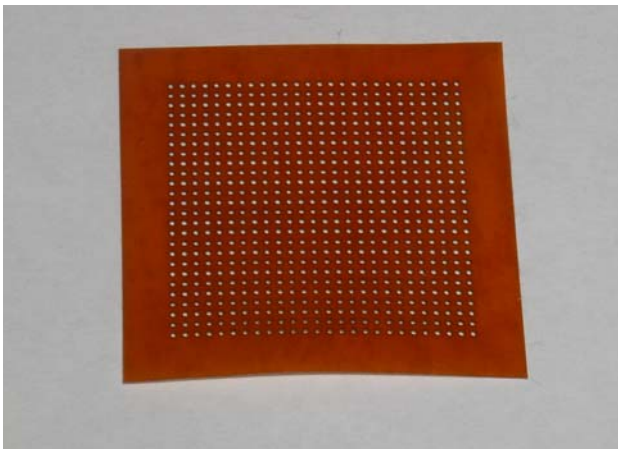


Figure 5. Semi-permanent stencil.

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 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 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.

OPERATIONAL CHARACTERISTICS

Having reviewed the methods and procedures for solder paste printing using the various stencils, it is important to consider what the impact of each stencil type is for every step of the process.

Ease of Alignment

All three of the stencil types discussed are relatively easy to align and place. All are aligned and handled manually. Magnification may be used to aid in alignment of fine pitch stencils. In the event of misalignment, the metal stencil is usually easier to realign than the removable or semi-permanent stencils. Even with the adhesive, both the semi-permanent stencil can be removed from the board surface and realigned. A significant advantage of the removable and semi-permanent stencils is that they can be easily trimmed to fit into very tight areas that would normally prevent paste printing with a metal stencil.

Non-Planar Surfaces

If the rework location is not coplanar due to PCB warp or solder resist irregularities stencil gasketing can become more difficult when utilizing metal stencils. Metal stencils (typically made from stainless steel) do not easily conform to the contours of the rework location. The poor gasket (seal) on the metal stencils will allow solder paste to work it's way underneath the stencil creating the potential for solder shorts. The material and the adhesive on both the removable and the semi-permanent stencil ensures that the stencils conform with the contours of the rework area regardless of surface irregularities.

Multiple Squeegee Passes

When working with any BGA rework stencil, it may be difficult to fill all the apertures of the stencil with one pass of the squeegee blade. Making multiple passes with the squeegee blade on a metal stencil could result in the solder paste being squeezed under the stencil unless the stencil is held firmly in place against the land patterns. The "near perfect" gasketing of the removable and semi-permanent stencils allow virtually as many passes as necessary without compromising the paste print.

Maintain Print Alignment

To achieve a good paste print, the metal stencils and the removable stencils must be removed from the PCB surface quickly in one motion. This removal must be done while maintaining the stencil parallel to the PCB surface and

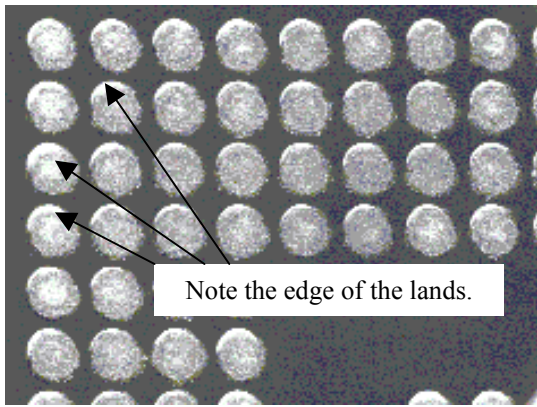


Figure 6. Paste alignment shifted from improper stencil removal.

raising it straight up. Those are a lot of variables to control in a very manual process. There is one system for metal stencils that assists in the lifting of the stencil from the PCB surface, however, removing the stencil without smearing the solder paste deposits requires some skill in every removal process. The semi-permanent stencil remains in place after the squeegee step. Since the stencil never gets removed from the PCB, it eliminates the potential for smearing the solder paste deposit.

Paste Release

As with full size production stencils, achieving adequate solder paste release from the stencil apertures is critical. Design of the stencil plays a big role in the paste release characteristics, however, even well designed stencils can have some problems with complete paste release. Some of the stencil variables that affect solder paste release are the topography (or smoothness) of the aperture walls. Many metal stencils undergo a polishing step for the inner walls of the apertures in order to improve the release. Another feature that can improve the paste release characteristics is a taper or trapezoidal shape of the aperture walls. Aspect ratio of the apertures can have the greatest effect. If any one of these features of the stencil is not correct for the specified

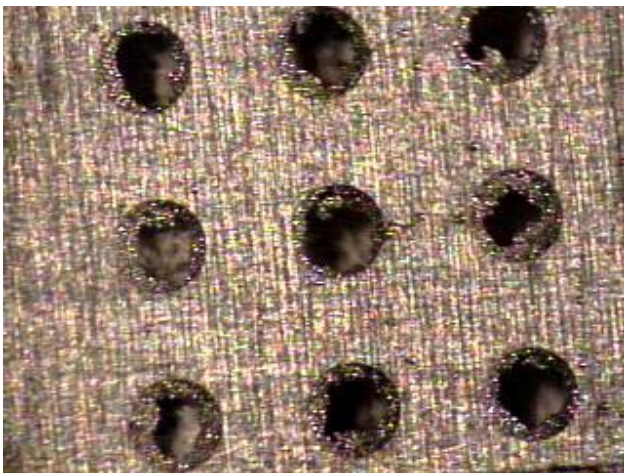


Figure 7. Improper paste release leaving clogged apertures.

location, the stencil can become clogged resulting in the paste deposit becoming smeared or incomplete. It then becomes necessary to remove the paste deposit and clean the area thoroughly and attempt the paste print step again. Since the semi-permanent stencil remains in place on the PCB, there is no opportunity to encounter paste release problems.

Stand Off

There are some BGA devices with eutectic solder balls that tend to exhibit a great deal of ball collapse after reflow. This can be caused from design (land size) or BGA weight. Many times this excessive collapse can result in solder shorts (usually in the corners). Generally, greater stand-off height in conjunction with proper solder volumes and shape can provide better solder joint reliability. The semi-permanent stencil will prevent excessive collapse of the solder balls and can be used to provide a minimum stand-off height.

Solder Shorts

Solder shorts can occur on BGA devices for a number of reasons. Most of these reasons have to do with the solder paste deposition or excessive collapse of the solder balls causing an increase in the ball diameter and a reduction in the ball spacing. One clear advantage that the semi-permanent stencil has over the other stencils is that it acts as a physical barrier to prevent solder shorts from occurring. The non-wettable polyimide film holds the solder paste in the stencil apertures, and once the solder becomes molten it coalesces and cannot migrate over the stencil.

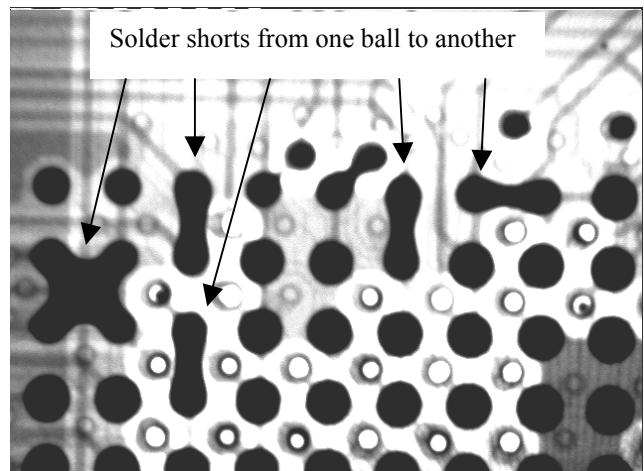


Figure 8. Solder shorts.

Ease of BGA Placement

The use of metal stencils and the removable stencils does not have any effect on the placement of the BGA devices.

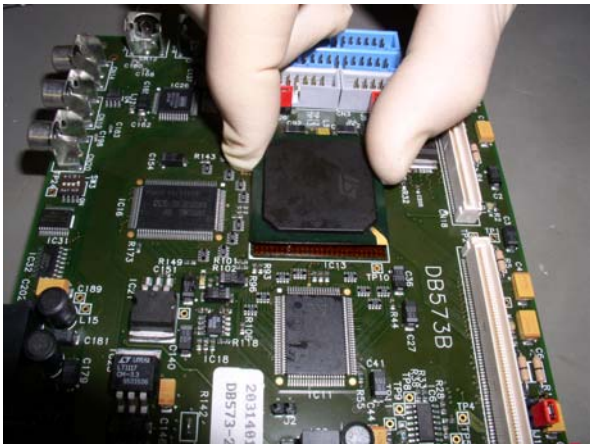


Figure 9. Manual placement of a BGA.

A semi-permanent stencil can be used as an alignment tool for the placement of the BGA devices. If the border of the polyimide material is evenly distributed on all four sides of the device it is placed correctly. This allows for manual placement of the BGA device. With minimal experience, a technician can “feel” the device settle into the apertures on the semi-permanent stencil.

Equipment Selection

There is a wide range of equipment available that was designed specifically for the rework of BGAs. The available equipment varies greatly in both cost and features. Most of the “high end” equipment features some type of vision system that can superimpose the solder balls onto the land patterns of the PCB to ensure correct alignment of the BGA prior to placement. There is no need for vision systems to assist in the alignment of the device when using the semi-permanent stencils. These same “high end” systems also feature computer controlled profiles to provide the correct time/temperature settings to enable the BGA to evenly achieve reflow. If the thermal profile is not adjusted



Figure 10. “High end” BGA rework system.
(VJ Technologies)

properly and it allows one side of the BGA to reflow before the other sides, the surface tension of the molten solder balls

will pull that side down to the PCB and lever the far side up away from the PCB. This anomaly is similar to the mechanism that causes tombstoning or drawbridging. When utilizing the semi-permanent stencil, there is no need for vision systems to assist in the alignment of the device. Since the stencil material itself will prevent excessive collapse, it reduces the effect of uneven heating.

Many OEM’s and EMS companies utilize nothing more than manual placement and a hot air gun to perform BGA rework. Although there is a tremendous cycle time reduction when performing BGA rework with a hot air gun, it is very dependant on the operator for control of the process and obviously does not even come close to the control that any of the more sophisticated systems provide.



Figure 11. Hot air gun being used to solder a BGA.

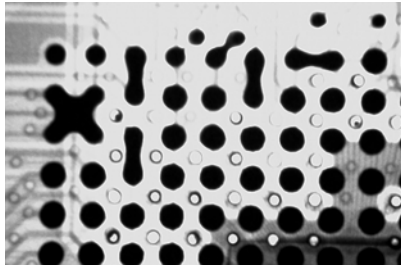
Because of the lack of control, BGA rework should not be attempted with a hot air gun. If there are no other alternatives and the rework must be performed with a hot air gun, the use of semi-permanent stencils should dramatically improve the yield of the rework process.

SPECIAL CONSIDERATIONS

Solder Resist Damage

BGA rework is frequently necessary during evaluation of prototypes or engineering models. This rework may be required to correct design flaws in the BGA or the PCB. Production of prototypes and engineering models are a great opportunity to evaluate the design for manufacture. During rework of these assemblies and even full scale production assemblies, certain design issues become apparent that have a much greater effect on the rework process than the original manufacturing process. An example would be a lack of solder resist on the vias and/or the dogbones of the BGA land patterns. During the BGA rework process, after the removal of the BGA the remaining residual solder on the BGA lands must be removed. There are some non-contact solder extraction systems that can usually do this safely without causing any damage to the solder resist, but in most cases the solder is removed through the use of a soldering iron and wicking braid. Depending on the adhesion of the solder resist, the undercut of the resist, and the width of the resist at the solder dams, it can be easily damaged and removed along with the solder during the wicking process. If this occurs, it allows the eutectic solder balls on the BGA device to wet or flow down the conductor and into the via

resulting in an open at the ball to land interconnect due to reduced solder volume. The typical repair for missing solder resist is to manually place and cure the resist. Depending on the ball count of the device and the extent of the missing resist, this repair can take over an hour due to the labor intensive nature of the repair. When using the semi-permanent stencil for rework of BGAs with missing solder resist, no repair of the missing solder resist is necessary as the adhesion of the stencil to the PCB surface will prevent the flow of solder from the land to the via.



Reduced ball diameter as solder ball flows down the via.

Figure 12. X-ray image showing result of resist damage.

Solder Volume

Based on previous works done on the relationship between BGA solder joint quantity and solder joint strength and reliability, greater solder quantities generally result in greater joint strength. Some studies indicate that solder joint shape may have as much to do with joint strength as the solder volume. Greater stand-off heights with a fillet shaped like an hour glass may provide better reliability than a traditional rounded collapsed ball shape^{1,2}. If this is the case, it is possible to use semi-permanent stencils with a much greater thickness to help achieve this. Normally, it is not possible to print a significant thickness of solder paste because of the effect it would have on the Area Aspect Ratio (AAR) of the stencil aperture³. The AAR is defined as the area of the aperture divided by the aperture wall area. For round apertures the AAR is defined by the following equation:

$$\text{Area Aspect Ratio} = \frac{\text{aperture diameter}}{4 \times \text{stencil thickness}}$$

It becomes very difficult to print paste at AARs below 0.8. As an example, an aperture that has a 0.5mm diameter on a stencil with a thickness of 0.15mm would have an AAR of 1.25. This would be fairly easy to print. On the other hand, an aperture with a diameter of 0.38mm on a stencil with a thickness of 0.20mm would have an AAR of 0.48 making it virtually impossible to achieve a good print with good paste release. With a semi-permanent stencil it is now possible to print paste at a much greater thickness while taking the AAR variable out of the equation.

Paste Flux Application

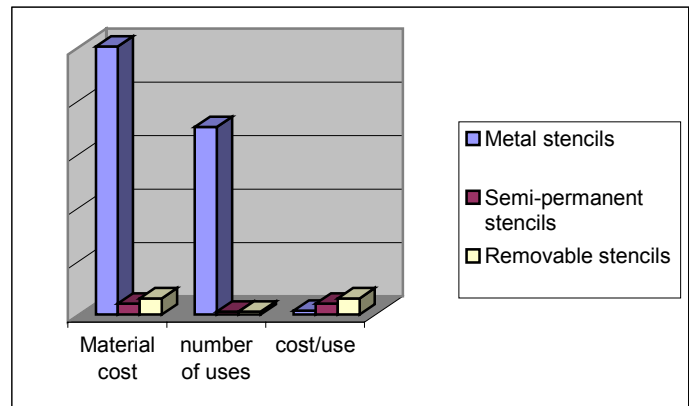
In the event that the selected method for BGA attachment utilizes flux instead of solder paste, the semi-permanent stencil may still prove to be advantageous. The usual application method for flux on the BGA land patterns is to generously apply with an applicator to the lands. During

this application the flux is spread evenly over the lands and the solder resist in one continuous layer. The logical choice would be to apply the flux only to the areas that require it... the lands. Even with the large quantities of flux being applied to the entire general area of the BGA lands, the temperatures seen at the PCB surface will be sufficient to activate all flux constituents (when using a no-clean flux) and ensure that the residues left behind are benign. By using a semi-permanent stencil to apply the flux only where it is needed, and the fact that the stencil will restrict the migration of the flux, it should be possible to use flux with a higher activity level without the concerns that would normally accompany such a process.

COST COMPARISON

Material Cost of the Stencil

Stainless steel stencil costs are much greater than the costs for either the removable or the semi-permanent stencils. This is because the stainless steel material is much more expensive than the base material of the other two stencil types. Even with a significant cost difference, metal stencils can remain an economically viable method as the metal stencils can be used many times before being disposed of. How many times a metal stencil can be used is dependant on the degree of care and maintenance used on a particular stencil. Stainless steel stencils are easily damaged by rough or careless handling because they are not fixtured in a heavy frame like the full size production stencils. Any bending or denting of the stencil surface can result in the stencil not making the proper gasket between the PCB and the stencil.



Lack of proper cleaning can degrade the inner surfaces of the apertures making good paste release an impossibility. Although it is hard to define an accurate lifespan (measured in uses) for metal stencils, a good average is approximately 70 uses before the stencil must be discarded. If this was the only factor used to determine the cost effectiveness of the various stencils, the metal stencils would be less costly by at least a factor of two, but there are many other factors to consider.

Cleaning Cost

Any stencil that is designed to be used more than once must be cleaned. Proper cleaning of metal stencils is critical. If the stencil is not cleaned properly it could result in uneven solder paste deposits or solder shorts as solder paste is transferred from the bottom of the stencil to the PCB

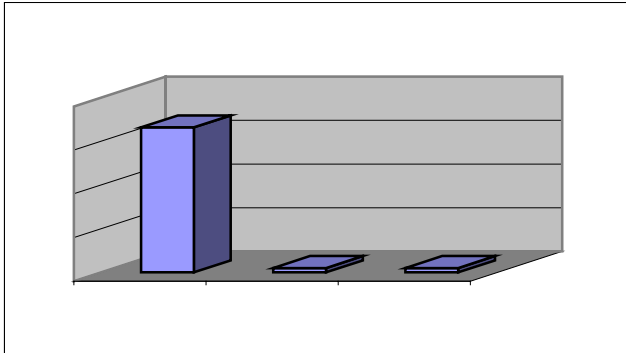


Figure 14. Cleaning costs.

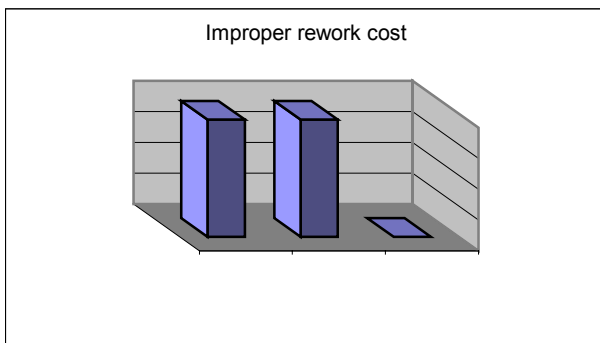
surface. Although it is possible to wash the metal stencils in small baths, most often they are washed by hand. This can be a labor intensive task as special emphasis on cleaning the apertures is time consuming.

BGA Alignment Costs

When using a metal stencil or a removable stencil, it is critical to have some type of vision system to place the BGA. Without the use of a vision system, there is too much risk of smearing the solder paste deposits. To avoid solder shorts, the alignment should allow for no greater than 25% pad overhang. Even with automated equipment and a highly skilled technician, the alignment process can still be time-consuming. The use of semi-permanent stencils can eliminate much of the time taken during the alignment process by allowing the technician to manually place the BGA into the apertures of the semi-permanent stencil.

Cost of Improper Rework

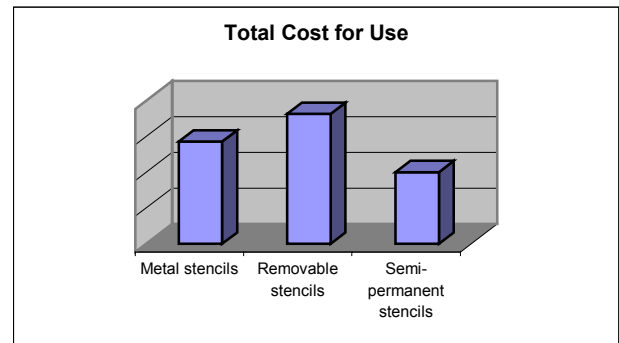
BGA rework utilizing metal stencils or removable stencils can fail for a number of reasons. The failure may be caused



by solder shorts from the stenciling process or excessive collapse or by an open caused from incomplete solder paste deposition. In either case, any failures must be removed and replaced and the BGA must be reballed. Even if this occurs

infrequently, (once every twenty times) the expense can add up.

When taking all factors into consideration, it is clear that the use of semi-permanent stencils is more cost-effective than metal or removable stencils. With a relatively low material



cost and the improvements the semi-permanent stencils bring to the process it surpasses other stenciling methods.

CONCLUSION

Although stencils have been used in the electronics industry for many years, the use has been plagued by the impact of the removal of the stencil from the substrate. Through the use of compatible materials it has become possible to omit the stencil removal process step and thereby remove many of the potential opportunities for error. The improvements to the process include:

- Ease of use
- No paste release problems
- No smearing during removal
- Prevention of solder shorts
- Repairs damaged solder resist

The fact that by leaving the stencil in place benefits the process of BGA replacement by opening the process window is an added bonus.

REFERENCES

- [1] 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".
- [2] Xingsheng Liu, Materials Science and Engineering Department and Power Electronics Packaging Laboratory Virginia Polytechnic Institute and State University, March 27, 2001.
- [3] Greg Caswell and Julian Partridge, "BGA to CSP to flip chip-Manufacturing Issues", Journal of the Microelectronics & Packaging Society Vol. 8, No. 2 (2001)