

## **Multiple Methods for Applying Paste Flux or Solder Paste for BGA Rework**

**Bob Wettermann, BEST Inc. MIT**

There are multiple methods, each with its associated benefits for given applications, for printing either solder paste or paste flux for BGA rework. Each of these methods is best-suited for a given situation, board layout and skill level of operators performing the BGA rework. This discussion will layout the various methods and present the specific circumstances for which the specific technique is most well-suited. In addition, the pluses and minuses for each of the approaches will be discussed in detail.

The techniques that are most commonly employed for selective printing of either solder paste or paste flux for BGA rework the use of:

- Solder paste or paste flux for printing using a mini metal stencil,
- solder paste or paste flux for printing using an adhesive-backed flexible film stencil,
- solder paste or paste flux for printing using a stay in place stencil,
- solder paste or paste flux for printing using a BGA rework machine fixture for printing on the solder balls or
- solder paste or paste flux dipping using a dip well.

Each of these methods will be explained and illustrated along with their respective “best uses” for a variety of BGA rework challenges.

### Solder paste or paste flux printing using a mini metal stencil

The selective solder paste or paste flux printing through the use of a miniaturized metal stencil on a BGA location most closely mimics the original manufacturing process. After site prep has been completed, that is remnant solder removal, site cleaning and inspection, the location is ready to re-accept the application of the solder paste or paste flux prior to placement and reflow of the replacement device. A small metal stencil, corresponding to the site locations’ pattern with the same aperture dimensions and thickness of the original SMT stencil, is placed over the pad patterns. There are a variety of ways to help prevent the paste or flux from being smeared on device near the rework location. These methods include taping off the area or building “dams” into the mini stencil in order to help prevent the solder paste or flux from smearing the surrounding area. Sometimes if space is not constrained, the stencil can be oversized which means it can act as a barrier to the paste or flux from going elsewhere. Flaps built into the stencil not only prevent the paste or flux from migrating to unwanted areas of the board but also they bring rigidity to the stencil which helps in aligning the stencil to the board and keeping it coplanar. A mini squeegee, sized slightly larger than the package outline dimension, is used to roll solder paste or flux through the apertures of the stencil. This is a “one and done” process that does not allow for a second printing opportunity. After liftoff of the stencil, a good print will show up as uniform “bricks” of solder paste on each of the pads.

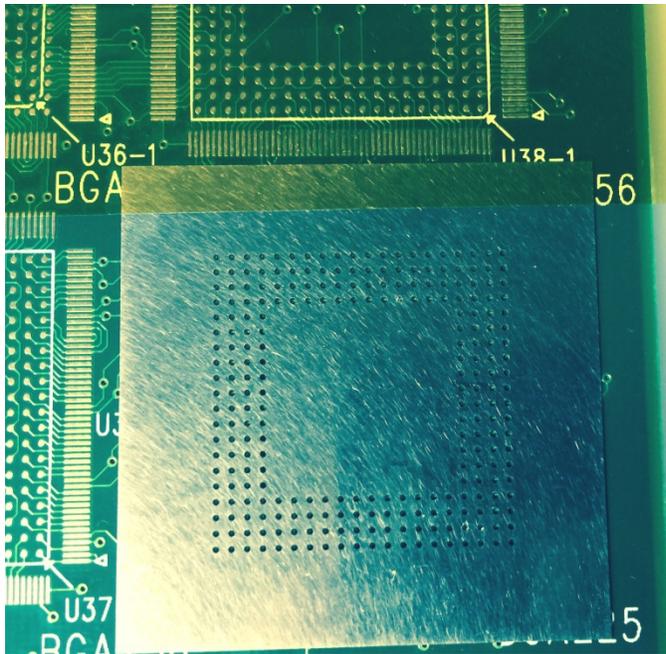


Figure 1- Mini metal stencil taped to PCB

<Pic Figure 2 Mini Metal on the board edited version>

There are several downsides to using this method of printing. The skill level needed to get a good print is at an advanced level. The technician must be able to apply downward pressure on the stencil while at the same time rolling solder paste or flux through the apertures of the stencil. Without coplanarity between the board and the bottom of the stencil, solder paste or flux will smear the underside of the BGA pattern causing the technician to have to start over. If the stencil is also not carefully cleaned in between each of the print cycles the stencil will no longer release uniform “bricks” of paste or flux resulting in less than good print quality. If the technician bends the stencil during the cleaning process the stencil could present the same problems. In many cases these types of stencils take up fairly large footprints in order to accommodate the flaps or the holding fixture which means it is difficult to “shoe horn” this stencil into a modern densely-packed PCB.

#### Solder paste or paste flux printing using a flexible film stencil

Selective solder paste or paste flux printing through the use of a flexible film stencil with a repositionable adhesive backing is the modern version of the mini metal rework stencil. After site-prepped pads are ready to accept the application of the solder paste or paste flux via printing, a small flexible film stencil, available in a variety of thicknesses, is aligned over the pad patterns. The stencil is placed on to the board and aligned. Micro alignment can then take place as the repositionable adhesive allows the stencil to be more exactly be aligned underneath a microscope. The film stencil itself can have an oversized pattern which can be rolled up in and around the device area needing replacement in order to prevent the flux or solder paste from contaminating the nearby area. A mini squeegee, sized slightly larger than the package outline dimension, is used to roll solder paste or flux through the

apertures of the stencil. However, unlike the one shot method of the mini metal stencil, the flexible film stencil, because of the adhesive, prevents solder from leaching in between the pads. This means that the solder paste (or flux) can be rolled numerous times back and forth to make sure each of the apertures. After lift off of the stencil, a good print will show up as a uniform “brick” of solder paste on each of the pads just like in the case of the mini metal stencil.

There are several advantages to using this process versus the mini metal stencil. First of all the flexible film stencil more easily fits in to tight spaces as the film can “roll up” to the devices thereby protecting neighboring devices. In addition, the adhesive backing “masks” the paste or flux from migrating underneath the film stencil and smearing the print. In addition, the ability to print many times in order to fill up the stencil apertures allows for a consistent paste volume to be packed in to the aperture wells. These benefits allow the skill level of the rework technician to be less with a film stencil than a stainless steel mini stencil.

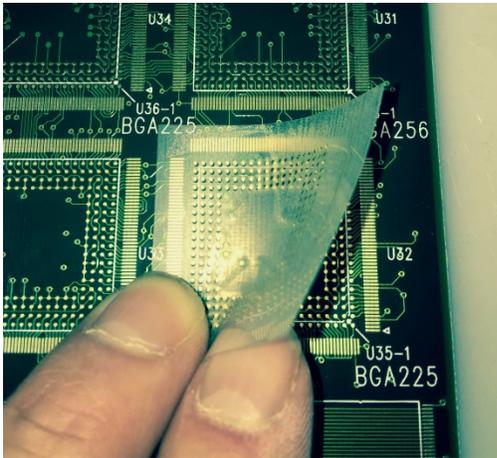


Figure 2 Flexible film stencil

<Figure 1 removable plastic film stencil>

The disadvantage to this printing technique is the greater cost per print when dealing with 50 or more devices. Since these stencils are a single use product, the cost per rework is greater when

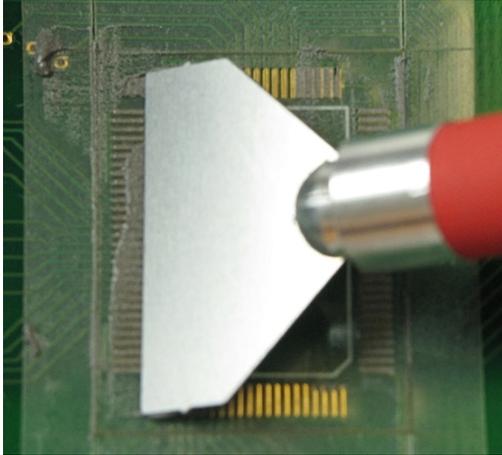


Figure 3- Micro squeegee on flexible film stencil

<Rolling solder paste in to the apertures>

higher volume rework is required. Unlike the single use flexible film stencil, metal mini stencils can be used over and over again.

#### Solder paste or paste flux printing using a stay in place stencil

Another method that can be used is one in which a stay in place, semi-permanent stencil is placed at the rework location. This stencil allows you to tactilely fit the replacement BGA into position without a vision system, it prevents solder shorts by having an insulating medium between the solder balls and it repairs any underlying mask damage. Once the BGA locations has been prepped, the site locations can be printed with paste flux or solder paste using a stay in place Kapton™ stencil. Stencils of various thicknesses are laser cut to the user's preference based on the manufacturer's datasheet dimensions or by extracting the CAD data from the GERBER file. These have a special high temperature adhesive which can withstand the various thermal and chemical processes found in SMT assembly. Like the flexible film stencil, this product has a release liner which is peeled off prior to the stencil being aligned and placed onto the PCB. These stencils are "hidden" underneath the rework location being cutting being fabricated slightly smaller than the outline package dimensions. The stencil also provides the added benefit of being able to prevent most shorting related to device warpage as an insulation between pads is assured. A mini squeegee is used to roll solder paste or paste flux in to the apertures. Like the flexible film stencil the squeegee can be moved back and forth numerous times to insure solder paste aperture "fill". Also, like the flexible film stencil, this Kapton™ acts as a barrier to prevent solder paste from leaching between the pads. There are several other benefits to these type of stencils including mask repair, increased SIR insulation as well as the ability to present thicker prints meaning warpage of the device can be overcome. After solder paste or flux is rolled in to the apertures, a lint free cloth is used to wipe away any excess material from the stencil's surface. The device balls can now be tactilely "fit" in to the apertures making alignment of the device simple.

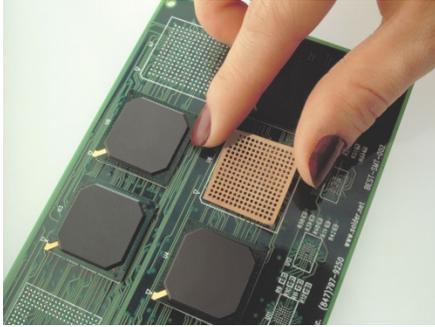


Figure 4 – Stay in Place Stencil

#### STEP 2

There are numerous benefits to using this type of stenciling approach for BGA rework. One of the most useful benefits of this method is not having the requirement of a costly rework. The solder balls of the BGA can be tactily “fit” in to the apertures of the stencil making placement simple. Another advantage to this approach is its ability to act as a mask thereby repairing any mask damage underneath the BGA location. The adhesive-backed Kapton™ stencil acts like solder mask in terms of performance. In addition, this stencil helps to prevent bridging by having a “dam” between each of the apertures. Finally, the stencil helps limit the collapse height of the solder balls thereby preventing shorting from

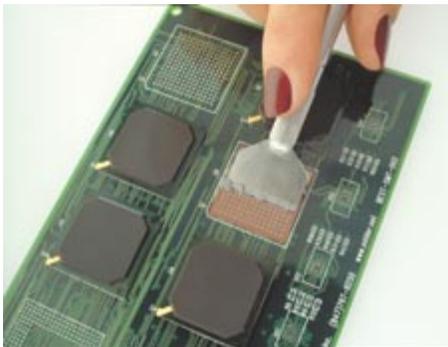


Figure 5- Using a mini squeegee to roll solder paste in to stay in place stencil

taking place.

#### STEP 3 StencilQuik

Some of the downsides of the stay in place stencil method are the inability to inspect underneath the BGA and the possibility that the user will not allow any new materials on the PCB without extensive testing. The material of the stay in place stencil-both the Kapton™ and the adhesive system- becomes part of the printed circuit board assembly. There are some strict medical and military hardware assembly guidelines in place that may not allow for this material to be placed on to the PCB and not be used. Due to the nature of the stencil taking up the gap between the bottom of the device and the

board, the use of this process does not allow for either visual or endoscopic inspection after rework (there is no impact on the x-ray image).

Solder paste or paste flux printing using a rework system fixture for printing on solder balls

One method that is utilized for printing solder paste or paste flux onto the balls of a BGA for rework involves the use of a specialized fixture for holding and printing selectively the bottom of solder balls. For this method a high end rework system with the availability of a split prism alignment system and a pick and place nozzle is required. The fixture (Figure 6) is custom-designed by the machine builder to fit onto the rework station. This custom tooling that holds both the component in place as well as holding the printing stencil for selectively printing onto the solder balls is engineered by the machine builder. This tooling “nests” or holds the part during the printing process. The holding pocket dimensions are gleaned from either a sample part or the mechanical drawing of the part. The stencil is designed using the part mechanicals found on the data sheet. The rework technician prints either solder paste or paste flux in a single pass using a micro squeegee on to the balls. A little over 50 percent of the ball diameter is exposed by the fixture. Once the part has been printed, the entire part is placed in to the machine where it can be picked, aligned and reflowed.

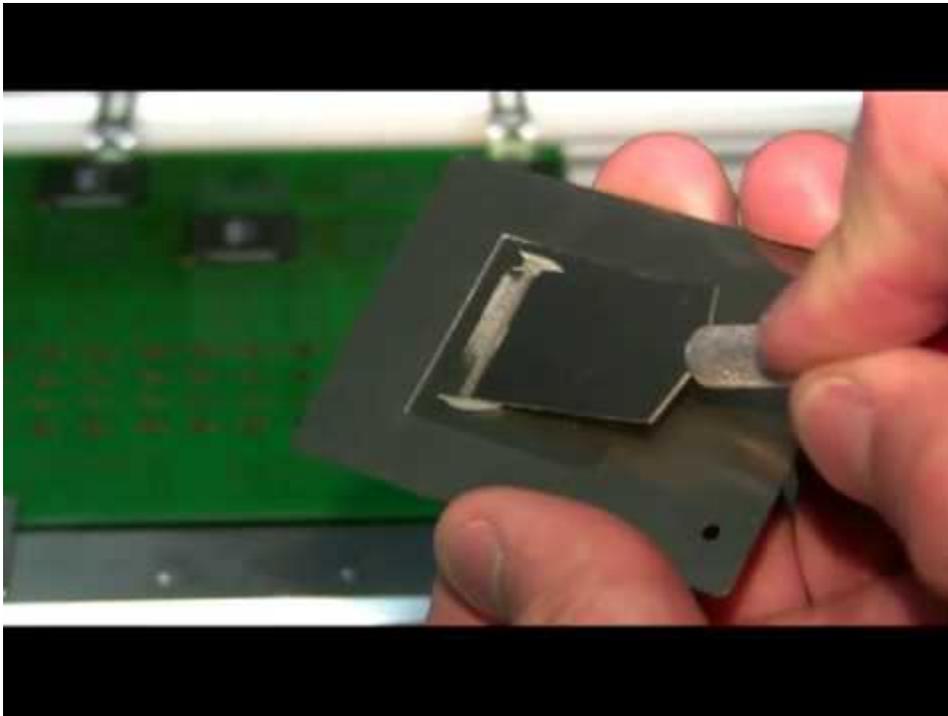


Figure 6- Using a rework system fixture for printing on solder balls

Solder Paste Printing Onto Component –Courtesy Metcal

There are several advantages as well as disadvantages in using this technique. One of the advantages to using this selective printing method on to the solder balls is that there is no need to protect neighboring

devices from being smeared with paste or flux. Only the necessary solder paste or paste flux volume is applied to the balls. This results in minimal clean up of any flux residue. Most importantly this technique allows you to rework devices which are tightly packed on the circuit board. While being a very appropriate technique for tightly spaced BGAs, this technique does have some drawbacks. The fixtures and stencils for this process are relatively expensive for low volume rework applications on a per-print basis. Thus this method is more appropriate for higher volume rework. In addition, it usually takes several weeks to generate any tooling for a “new” component.

Also, the dexterity and patience required of the rework technician is high meaning that the technicians being able to perform this rework is limited.

#### Solder paste or paste flux dipping using a dip well

The final method that will be analyzed for application of solder paste or paste flux for BGA rework is the dip method. In this method solder paste or paste flux is selectively applied to the bottom of the solder balls. This means that solder paste or flux is only applied to 50-60 % of the ball diameter. The new or reballed component, using a controlled dipping source such as a robot, rework station or pick and place machine, is used to pick up the component and then dip it into a dipping well. The dipping well is sized to accept the bottom footprint of the solder balls and its depth is such that approximately 60% of the ball diameter. After making sure the dipping well is properly filled, the part is “bottomed out” in the dipping well. The extraction rate and the viscosity of the material controls how much of the material ends up on the bottom of the solder ball. The solder paste should be formulated such that it is meant for dipping. Sometimes colorants are added to the paste or flux so that visual inspection is easier.

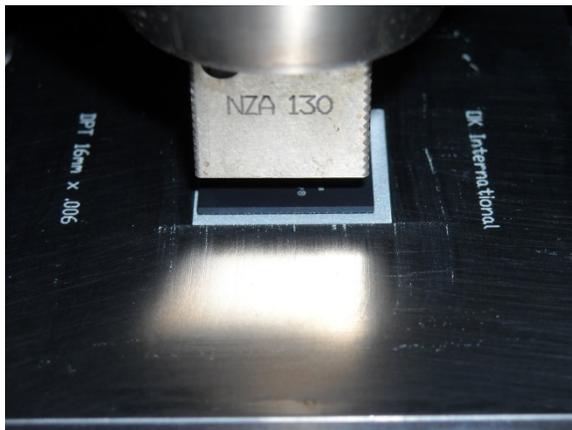


Figure 7 – Solder paste or paste flux dipping technique

<Part solder paste dipping>

There are several advantages to using this method of solder paste or flux application. First of all, like the ball printing technique previously described, there is only selective application of either solder paste or

paste flux on the solder balls. This means the flux residue on the PCB is limited making the potential for any residual active flux species to be minimal. This reduces the risk of any reliability fallout in the electronics assembly operating life. Also, like the previous technique described, the dipping method allows for devices that are in the middle of a very dense circuit board to be reworked. Also, with only a few dip well sizes accommodating the most common solder ball diameters, the tooling expense for this technique is minimal. Once the process is “dialed in”, the skill level of the operator is somewhat taken out of the equation as no micro printing technique expertise is required. Downsides of this technique include the necessity for a high end rework system, making sure the environmental variables are kept steady in order to retain the rheological properties of the paste flux or paste and the difficulty of inspecting the balls for proper coverage once “dipped”.

While there are numerous other BGA paste or flux selective application techniques, these are the most-heavily used in the Class II and Class III assembly areas.