SeeingTheLight

By Ronald**Schaeffer**

New BGA Solder Mask Repair Technique Using Laser Cut Stencils

The increased replacement of high lead count SMT devices with BGAs and other high ball count area array packages has brought increased challenges to PCB rework and repair. Often solder mask areas surrounding BGA pad areas are damaged when components are removed. In the "dogbone" areas connecting BGA pads to the associated vias, solder mask serves to stop solder from wicking down into the via.

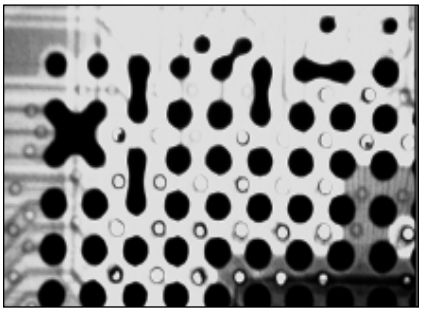

Figure 1. 2D Xray showing reduced ball diameter as solder ball flows down the via (VJ Technologies).

This associated solder mask damage causes the exposed via and circuits to be prone to shorting due to BGA joint starvation. Previously these BGA pad locations had to be repaired using a very time-consuming and tedious hand process using highly skilled PCB repair technicians. A new technique eliminates the need to touch up each location by adding a laser patterned, semi-permanent stencil thereby greatly reducing the time required for BGA mask damage repair.

There are several specifications for both the repair methods and the acceptability criteria post-repair for solder mask that guides the rework technician's work product. The methods used to repair solder mask are defined in IPC 7711 "Rework and Repair Guide." Many times on a practical basis these techniques are augmented and refined by the experience of the repair technician who has many years of experience in repairing solder mask. Once the mask area repair is completed, the PCB acceptability reference documents guiding the reworked area fall under the acceptability criteria of either the IPC-A-610, "Acceptability of Electronic Assemblies" in section 10.0 which describes the acceptability of laminate conditions, or the IPC-A-600 "Acceptability of Printed Boards" in section 2.0 describing externally the PCBs' observable characteristics.

There are several specifications for both the repair methods and the acceptability criteria post-repair for solder mask that guide the rework technician's work product.

The solder mask repair procedure is outlined in IPC 7721 Section 2.4.1. Solder Resist Coating Replacement and can be summarized in the following steps:

1. Clean the area to ensure adequate adhesion, to optimize electrical properties and to minimize corrosive effects.
2. Remove all of the excess solder from the BGA pad and the associated via either with desoldering equipment or solder braid.
3. Scrape or grind off the "dog bone" section connecting the BGA pad to the via.
4. Prepare the replacement mask material.
5. Apply the replacement mask material. Add color agents if specified by the customer to match the aesthetics of the board material.
6. Let the epoxy cure.
7. Examine per the IPC guidelines mentioned above. Confirm that the repair was completed correctly by checking the electrical continuity between the via and the BGA pad.

While this has historically been the procedure for repairing a pad area underneath the BGA and serves as a repeatable, reliable process, it does have some shortcomings. First of all the process is very time-consuming. A highly skilled PCB repair technician (5+ years of experience) can repair a single BGA pad solder mask area in 15 minutes. If a 256 ball count BGA has just a few pad areas requiring solder mask repair this repair task can easily consume one hour per BGA. In many cases this one hour of labor cannot be justified. For fine pitch sizes this work tends to be outsourced to a professional PCB repair facility, as the skill level required to do such a repair is high. Outsourcing this work slows down the turn around time of the rework job.

One of the newest techniques for

>> continues on next page

Ronald D. Schaeffer is Chief Executive Officer of Photomachining. He received a Ph.D. in Chemistry from Lehigh University in 1986. He has been involved in the laser industry for over ten years, holding technical, sales and management positions at Spectra Physics, Lambda Physik, Laser Photonics and Resonetics. His hobbies include music, collecting and repairing antique timepieces, preparing (and enjoying!) fine food, and farming. He can be contacted at RSchaeffer@photomachining.com.

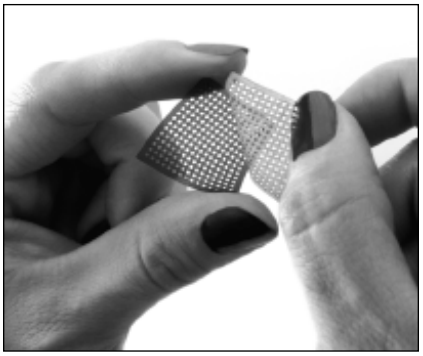

Figure 2. Semi-permanent stencil being pulled off its release liner (BEST, Inc.).

>> continued from previous page

repairing damaged solder mask underneath BGA areas addresses these shortcomings. This procedure involves the use of a semi-permanent stencil. In this method the stencil remains in place on the site location and becomes an integral component of the PCB. Like the metal stencils, the semi-permanent stencil is laser cut and can be provided in a number of different configurations. The construction of this semi-permanent stencil is similar to that of standard PCB labels in that once the release liner is removed, a high temperature adhesive is exposed. The stencil is then manually aligned with the land patterns on the PCB and pressed into place to activate the adhesive. 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. 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.

The application for placement of such a stencil on a BGA site involves the following steps:

1. Clean the area to ensure adequate adhesion, optimized electrical properties and minimized corrosion.
2. Place the semi permanent stencil over the BGA land patterns. The "new" solder mask is now in place.
3. Inspect.

There are numerous advantages to this technique including:

1. Solder mask damaged during the removal of the BGA no longer requires long repair times thereby increasing BGA rework throughput. In some cases an hour or more of repair time is saved.
2. Reduction in the skill level required for mask repair. Now a technician with little PCB repair knowledge can repair the damaged mask areas underneath a BGA.
3. There are none of the paste release problems associated with the metal or paper stencils including "smearing" of the paste or improper printing due to a lack of stencil cleanliness. No longer do BGA rework areas need to be reprinted numerous times.
4. The semi-permanent stencil prevents solder shorts as an insulative barrier surrounds (without touching) each of the solder spheres. This ensures a more reliable rework process.
5. A more reliable rework process is the result of using the semi-permanent stencil technique, as the surface insulation resistance values, especially using a no-clean flux, are an order of magnitude greater.
6. The process of BGA replacement is simplified to the point where a low level operator can hand place BGAs with a pitch of 1.00mm or greater. This reduces the time required for rework and eliminates the need for high-end rework systems.
7. The stencil acts as a standoff preventing uneven collapse of the solder spheres and potential shorting, a problem pronounced in ceramic BGA packages.
8. A faster, more cost-effective rework process for BGAs as rework time is cut by a factor of 50% or more

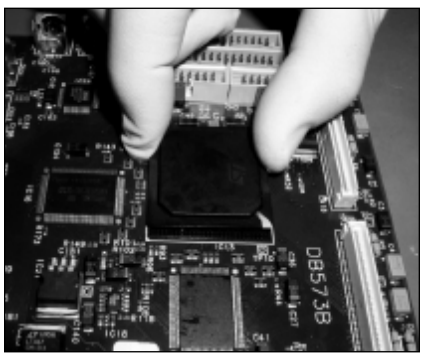

Figure 3. Manual placement of a BGA (photo courtesy of BEST, Inc.).

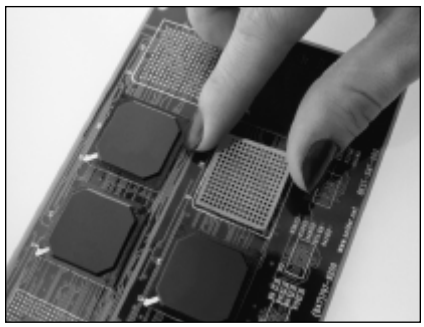

Figure 4. Semi-permanent stencil being placed onto a BGA site (BEST, Inc.).

This new semi permanent BGA stenciling technique eliminates the need to touch up each location by adding a semi-permanent stencil. This greatly reduces the time required for BGA mask damage repair touchup thereby increasing throughput in the rework and repair of PCB solder mask damage. The masks themselves are easily made using laser technology, which has both the economic viability and flexibility needed to make large volumes of many different repair patterns. **CT**