PHOTO STENCIL
PRINT BETTER.

SMT

Rework Challenges, Rework Solutions

By William E. Coleman, Ph.D., Photo Stencil

Although the goal of every process is zero defects, the reality is that rework-and-repair is necessary sometimes, especially given the cost of scrapping high-dollar assemblies. Fortunately, a variety of tools and methods are available for reworking defective PCB assemblies (PCBAs), even those with densely packed layouts and ultra-fine-pitch components. Specialized holding fixtures, miniature stencils, and related tools often make the rework process a cost-effective alternative to scrapping an assembly.

Several steps comprise the rework process in which a defective component, or a component with assembly errors, is removed from the PCB and replaced by a good device. The usual procedure is to remove the defective component, clean the pads on the PCB, print fresh solder paste on the pads, place the new component into the solder paste bricks, and reflow the component by heating locally. Typically, a small stencil and squeegee blade are used to print solder paste on the PCB's component pads. In the early days of SMT, before small printing repair stencils were readily available, process engineers would take an obsolete stencil and cut out a portion that matched the component to be reworked. However, stencil suppliers now provide a more convenient solution. Miniature stencils, with very small footprints and tri-fold upside walls to keep solder paste confined, are used to print solder paste directly on pads. Specially designed small squeegee blades fit the footprint between the fold-up sides.

This process works well with relatively large devices like 0.65-mm-pitch QFPs and 1.27-mm-pitch BGAs when there is sufficient clearance around the component. The trend today, however,

is to use smaller components packed closer together on the PCB. Circuit boards may contain QFNs, with package sizes ranging from 3 to 12 mm, I/O count ranging from 12 to 80, and pitches ranging from 0.4 to 0.65 mm. Another example is the micro-BGA, with package sizes ranging from 1.5 to 25 mm, pitches ranging from 0.4 to 0.65 mm, and I/O counts ranging from 6 to 2025.

An alternative repair process is to print solder paste directly on the QFN leads and ground plane, or on the BGA solder balls.[1] This repair system incorporates a universal holding fixture that can be used for any BGA or QFN device. Custom-designed stencils, spacer shims, and package-holding fixtures, manufactured by laser cutting or electroforming metal foils, support the procedure. Here, the BGA or QFN is tightly gasketed to the stencil during printing. In the BGA case, the solder ball gaskets to the stencil and protrudes partially into the stencil. This helps the BGA balls self-center to the stencil apertures. Once the paste is printed on the component, the rework technician lifts the device out of the universal holding fixture with a vacuum pick and places it on the PCB in the proper location. This is achievable even when neighboring components are in close proximity to the repair part.

A BGA package soldered to the PCB may have a short or open rendering the PCB useless, but the package itself may still be perfectly good and functional. The process engineer must remove and replace this package with a new one, since in the removal process the solder balls on the BGA package melt. Some BGA packages, however, are very expensive, in which case it is desirable to reball the BGA for reuse. This is achieved by removing all the solder balls from the BGA package, cleaning the BGA pad sites, applying flux to the pad sites, and placing new solder balls on the BGA package.

One method to handle the reballing process is to use a universal BGA reballing tool. This comprises a fixture that holds and aligns a ball drop stencil, BGA package holding tools, and shims that provide the proper spacing between the stencil and BGA package for proper ball drop and clearance from flux on the BGA pads. The ball drop stencil and holding tool must be custom designed for each unique BGA package. When properly spaced, the top of the stencil will be even with the top of the solder ball after it is dropped into the stencil aperture. Solder balls are applied to the top of the ball drop stencil and brushed over the surface until all stencil apertures are filled with solder balls. At this point, excess balls are poured off the stencil surface. Finally, the entire package is sent through reflow, where the solder balls melt and form truncated spheres with uniform ball height. The stencil material must have non-solder wetting properties to prevent solder adhesion to the stencil during reflow.

Conclusion

Although rework is never a desired step in the production process, it is often a cost-effective alternative to scrapping a high-dollar assembly. Dense layouts and fine-pitch packages seem to present a daunting challenge, but there are several innovative tools and methods available that provide a reasonable and effective means to attack the problem.

REFERENCES:

[1.] "BGA and QFN Repair Process," W. E. Coleman, Paper S08:01, APEX 2008.

William E. Coleman, Ph.D., is an *SMT* Editorial Advisory Board member and vice president of technology at Photo Stencil. He is Chairman of IPC's Stencil Design Guidelines Subcommittee 5-21e, and a member of SMTA and IEEE. Coleman has presented Technical Stencil Design reviews to SMTA chapters globally. He may be contacted at (719) 535-8528; bcoleman@photostencil.com.