As originally published in the IPC APEX EXPO Conference Proceedings.

Rework Challenges for Smart Phones and Tablets

Paul Wood
Advanced Product Applications Manager
APR/ BGA Products
OK International / Metcal
Garden Grove, CA

Introduction

Smart phones are complex, costly devices and therefore need to be reworked correctly the first time.

In order to meet the ever-growing demand for performance, the complexity of mobile devices has increased immensely, with more than a 70% greater number of packages now found inside of them than just a few years ago. For instance, 1080P HD camera and video capabilities are now available on most high end smart phones or tablet computers, making their production more elaborate and expensive.

The printed circuit boards for these devices are no longer considered disposable goods, and their bill of materials start from $150.00, with higher end smart phones going up to $238.00, and tablets well over $300.00.

The implementation of the surface mount devices have become key components for mobile products by offering increased component density and improved performance. For example, the newer style DDR memory integrated components use less power and work at twice the speed of former versions. It is not surprising that most component manufacturers now produce these surface mount devices as small as 1mm square.

Mobile products generally use an epoxy underfill to adhere components to the printed circuit board in order to meet the mechanical strength requirements of a drop test. Reworking glued components is the most difficult application in the electronics industry, and needs to be addressed as a process.

Rework Challenges

The removal of a glued component from a printed circuit board assembly requires a specific order of operations. The first step is to remove the glue fillet (Figure 1) present between the component and the circuit board. Mobile products generally have many types of components glued to the printed circuit board in order to meet the industry standards outlined in JESD22-B111 Board Level Drop Test Method of Components for Handheld Electronic Devices and JESD22-B110 Subassembly Mechanical Shock. The epoxy is applied to the components to prevent the common failure modes of cracks in the laminate, cracks near the intermetallic, and cracks in the bulk solder. The addition of the epoxy increases the robustness of the design and enhances reliability for the user.

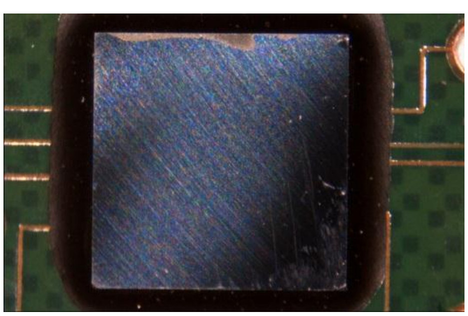

Figure 1: SMD with epoxy fillet

One of the issues associated with underfill epoxy is the glass transition temperature of the material. The glass transition temperature is the temperature at which a sudden change of physical properties occurs. In the case of underfill epoxies, they generally soften at approximately 208°C (exact temperatures dependent on manufacturer’s specifications). Lead-free solder, used to form an intermetallic bond between the component and the pads attached to the circuit melts at 217°C with peak temperatures of 235°-245°C common. The closeness of the epoxy and solder temperatures creates a narrow window of opportunity for removal of the epoxy and removal of the component. Removal of the surface mount device requires the removal of the epoxy fillet from around the sides of the device without damaging or disturbing the adjacent discrete components. Failure to remove the fillet increases the chances of damaging or removing the adjacent components (Figure 2) adding time and cost to the rework process.

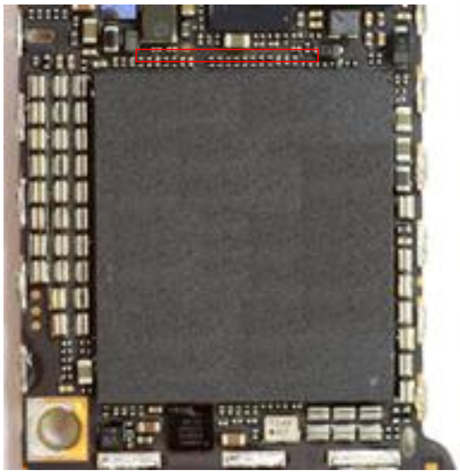

Figure 2: Adjacent components within 10 mils of SMD

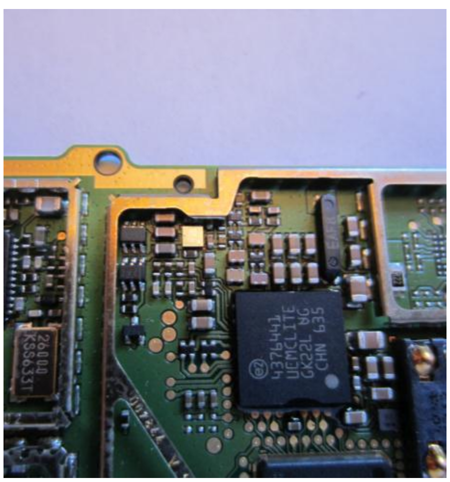

Figure 3: Densely packed components commonly in use for mobile devices

An example of an epoxy fillet removal technique for package on package (PoP) or ball grid arrays (BGA) is the use of a handheld convection tool and dental picks or the use of a soldering iron with very small solder tips. The dental pick or solder tip is heated and then used to carve the epoxy away from the component. This process requires a skilled operator as no automation process exists. To facilitate the ease of carving the epoxy, the dental pick tip must be polished to a mirror finish using extremely fine grit sandpaper. The polished tip when heated will achieve the temperature needed to change the physical properties of the epoxy. The use of a skilled operator is no guarantee of success in removing the epoxy fillet. Care must be taken to ensure that the tool does not damage the surface of the printed circuit board, component to be removed, or adjacent components.

The next step is to remove the component from the circuit board. Components are removed from the printed circuit board using infrared or convection technologies to heat the board and a retractable vacuum nozzle or other mechanical means, like a tweezers nozzle, to physically remove the component. In densely packed mobile devices, ball grid arrays are often mirrored on the printed circuit board. Removal of the top BGA requires precision control of the temperatures. If the solder on the bottom BGA becomes liquidous at 217°C during the top BGA’s removal profile, the solder underneath the bottom BGA expands in volume, forcing the solder to ooze at high pressure through the softened epoxy which has exceeded its glass transition temperature. This can be seen as small solder balls that have appeared out of the BGA (Figure 4).

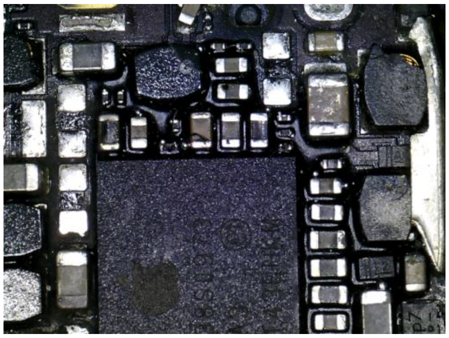

Figure 4: Solder expelled through epoxy during removal

The result of this phenomenon is the appearance of a poorly executed surface mount reflow process, and the possibility of functional failures as shorts. When an X-ray inspection is conducted and compared to original solder joints, a difference in size can be seen between the original printed circuit board and the circuit board after rework. The bottom BGA now has opens as solder has been lost. The solution is to keep the bottom chip below 217°C. 208°C, in general, is the ideal rework temperature for most parts. This often requires the use of a spot heater in the preheater to locally heat and cool components in a vertical position during rework. With the correct temperature applied to the bottom BGA, the top part can be reworked at lead free temperatures. At this point, the choice must be made to scrap the part or recover the component for later reuse.

Some manufacturers at the production level reclaim components, or re-ball and re-use some parts because of cost, e.g. memory, processors, Wi-Fi chips. Energy efficient and faster DDR memory costs up to $35.00 each. Processors can also cost up to $35.00 each. NAND flash memory and Wi-Fi chips can go as high as $41.00 each. Package on package (PoP) with both processor and memory are up to $50.00 combined. The cost of these components can represent a significant portion of the cost of the printed circuit board. If a PCB is damaged due to poor rework operations, some money can be reclaimed from refurbishments of the high cost components. Reusing components after removal is a hidden operation, often kept from most OEM manufacturers. However, contract manufacturer’s recycling parts is often used as a cost savings measure. Removal and reuse of components is common in the development lab environment. Component cost for a prototype may be high and the time needed to acquire a new part may exceed the time available which increases development time and costs. This is true in and out of warranty rework markets. If they are to be reused, the components must be removed safely at correct component temperatures. The pad area must now be prepared for a replacement component.

When a faulty component is removed from a printed circuit board, solder and epoxy are left as a byproduct (Figure 5).

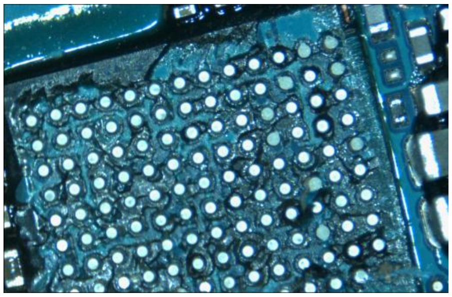

Figure 5: Residual solder and epoxy

Residual solder and epoxy must be removed prior to component replacement. The most common way to remove residual glue is to scrape the epoxy using hand tools. Using this method greatly increases the risk of board damage. An alternative method is when the residual solder and epoxy removal operation can be done manually with a soldering iron, wicking braid, and flux (Figure 6).

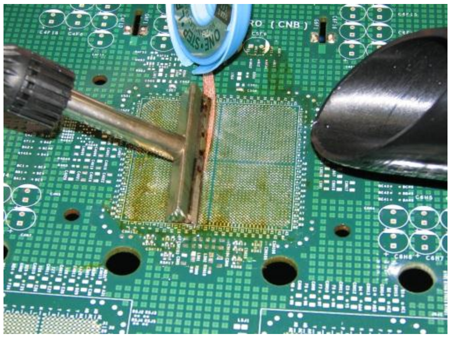

Figure 6: Pad cleaning using company soldering system and blade cartridge

Solder is cleaned with wick or re-tinned with soldering iron hoof style tips to level pads, in most cases, and in some cases increase solder heights on corners to stop bridges in the corners of 0.4mm PoP. This is because components are warped concavely in the middle as a new component, and might require this special technique to achieve success. The risks to the circuit board assembly are inconsistent solder removal resulting in poor adhesion, solder resist damage resulting in opens or shorts, and thermal damage to the PCBA. The alternative is less risk than the most common method. However, both manual processes add additional cost to the process due to the time and resources need to perform the pad cleaning operation and represent a risk to the manufacturing in terms of scrap costs. Cleaning residual solder from ultra-fine pitch 0.35mm and 0.4mm pitch components now require a better solution known as contactless solder removal. Commonly referred to as scavenging, vacuum and hot air are used to melt solder and vacuum up solder without touching the printed circuit board. The advantage of a scavenging site cleaning system is the equipment does not make contact with the circuit board reducing the risk of damaging the solder resist and pads. Scavenging systems require the same thermal profile controls as other rework equipment and benefit from automating the task of moving the vacuum collection nozzle over the site to provide consistent solder removal. With the solder pad cleaned, the operator is ready to replace the component.

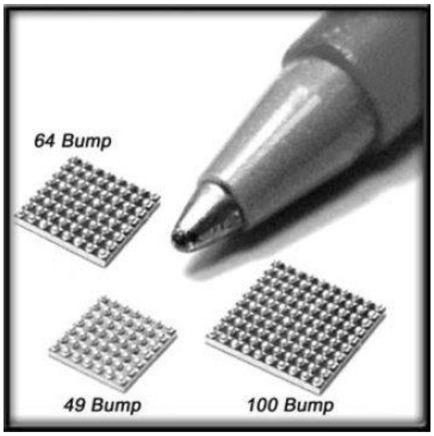

Figure 7: Micro-SMD examples

Replacing components to the printed circuit board requires the use of either infrared or convection technologies to heat the board and a mechanical means of physically placing the component. The size of the components used in mobile devices provides a challenge to the operator. Processors have solder connections between 0.3mm -0.5mm (4-10mils), micro-SMD component sizes range from 1mm-6mm square with 4 to 100 solder balls per package. Common solder ball sizes range from 0.1-0.2 mm (4-8mils). Memory components are typically BGA style and solder connections as low as 0.4mm. Replacing these components often requires the use of a camera to accurately align the part to the solder pad. There are a number of factors to consider when replacing components. The operator must consider whether to apply flux to the printed circuit board or to the component (Figure 8).

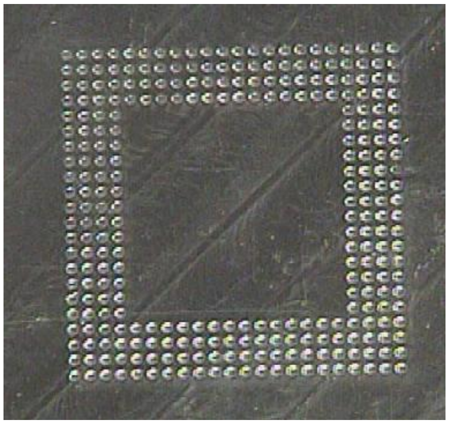

Figure 8: Repeatable flux depositions on solder balls

Liquid flux dries before the solder reflows. Gel flux, in paste form, is used during manufacturing and should be used during rework. Applying flux to the solder balls provides little coverage. Dipping the component into gel flux provides excellent coverage as well as solder ball cleaning. The ideal flux dip is half the height of the solder ball. This transfers the exact amount of flux onto the solder balls and not on to the PCB. After reflow, less contamination is present under the BGA. Also, flux does not onto drip through the via to the other side of the PCB. Other considerations include applying solder paste to the printed circuit board, usually not possible on mobile devices due to the densely packed components; screen printing the solder onto the components is possible on parts down to 0.4mm pitch, or the easiest solution, dipping the components into solder paste. Each type of package presents its own challenge to the rework process. For example, component stencil printing is essential for successful rework of LGA, QFN, BTC, LLF, and PoP devices. Package on package requires the top and bottom components to be reworked simultaneously. These are processors and memory mounted on top of each other and are the most difficult to rework. Screen printing the solder paste onto the solder balls using a stencil is important to achieve the correct stand-off height. Flux dipping the balls of the top package is done to half the ball height (Figure 9).

Figure 9: PoP/BGA paste dip

The PoP device is reflowed with a solder paste reflow profile. The placement of quad flat, no-lead (QFN)/ BTC components is another example of components requiring a repeatable process for rework. QFNs/BTCs, also known as leadless lead frame packages (LLF), have no solder on the package but just a thin plated surface layer of tin, and the distance between the package and the circuit board is within 2mils flat when soldered.

QFNs/BTCs require solder paste either on the PCB or on the part to be replaced. In mobile products printing the PCB is impossible because of high density to adjacent packages; the stencil would never fit to the PCB for printing. Stencils are made to print the package with 4 or 5 mills of solder paste. The center pad is a ground pad and is usually split up into 4 or 8 quadrants; this helps stop the package swiveling during the reflow process and ensures solder coverage to be equal. Most manufacturers require 60-80% solder contact to the PCB ground pad in the center. This ground pad keeps the package cool and stops overheating, as heat is transferred to the PCB via the center ground pad. Special 0.4mm pitch stencils have been developed to better align the package to stencil print locations. QFN/BTC with a pitch of 0.5mm have been in mobile devices since 2005 and are much easier to rework. QFNs/BTCs with a 0.4mm pitch are the most difficult and challenging. These components often have two rows of connections on the perimeter. Each type of component requires careful consideration in regards to the techniques needed to place the part on the circuit board.

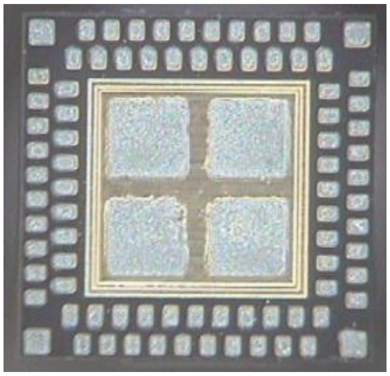

Figure 10: Multi-row QFN/BTC

Once the part is placed on the circuit board, the new components require the development of thermal profiles. Thermal profile repeatability is very important. The thermal profile should start with a linear ramp rate of 0.5°C-2.0°C/second which allows the gradual evaporation of volatile flux constituents and prevents defects such as solder balling/beading and bridging as a result of hot slump. It also prevents unnecessary depletion of fluxing capacity when using higher temperatures.

Next, an optional soak stage between 160°C and 180°C may be implemented for a few reasons. This includes minimizing the ΔT across the board to <10°C and allows for full volatization of the flux solvents to reduce flux induced voiding. Finally, the recommended peak temperature is 225°C-235°C and the total time above liquidous (TAL) should be less than 120 seconds above 217°C. A rapid cool down of 4°C/second is desired to form a fine grain structure. Slow cooling will form a large grain structure, which typically exhibits poor fatigue resistance. If excessive cooling 4°C is used, both the components and the solder joint can be stressed due to a high coefficient of thermal expansion (CTE) mismatch. Fixtures can be used to improve process yields. Fixtures are very important in mobile device rework due to the size and shape of many mobile device PCBs. A fixture will ensure consistency by locating the circuit board in relation to the heating source (Figure 11).

The fixture should be low mass and made of a material which does not act as a heat sink. A metal fixture will absorb thermal energy during use and will alter the profile. Proper circuit board profiling and the right fixture will ensure a repeatable process.

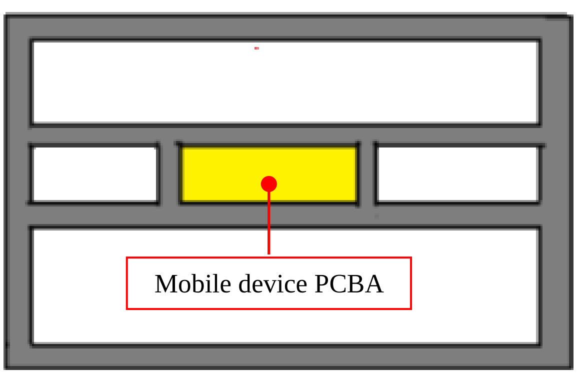

Figure 11: Mobile device fixture

After the part has been reflowed, the circuit board is tested. Testing at this stage ensures the reworked component functions properly before applying the epoxy. Typically, the circuit board will be subjected to detailed inspection and a functional test. The detailed inspection insures that the component was placed and reflowed on the circuit board and that there is no observable physical damage to the board. Boards may be subjected to detailed visual inspection under magnification or subjected to X-ray analysis. The circuit board is functionally tested. This testing ensures the circuit board and the reworked part are functioning to specification and there are no opens or bridges as a result of the rework process. Once testing has been completed, the unit is ready for the final step in the rework process.

Finally, the last step in the process is to dispense epoxy underfill. Typically, the operator will use the epoxy defined in the manufacturing process. Generally, underfill is dispensed along a corner or in a line along the edge of the surface mount device (Figure 12). The device is heated to 125° to 160° in accordance with the manufacturer's recommendations and taking into account the circuit board properties. Capillary action will absorb the material and distribute it underneath the part filling the space between the component and the circuit board. The board is maintained at a constant temperature until the epoxy has cured. Typical cure time to achieve the optimal strength is approximately 5 minutes. However, the cure time can be longer depending on the epoxy properties.

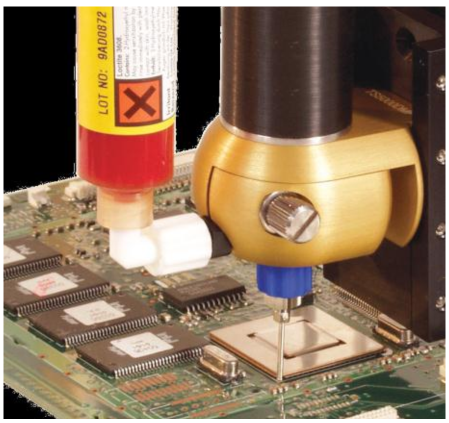

Figure 12: Dispensing epoxy underfill

Considerations

After attending the rework presentation and reviewing this paper, attendees will understand the challenges and process needed for successful repair of surface mount devices and the associated devices present on a printed circuit board assembly. Attendees will understand the challenges inherent in each stage of the process and understand the need for process control to manage the risk of rework. A robust and repeatable rework process is essential for high rework yields. Proper preparation and application of solder and flux will add to the robustness and repeatability of the process. Reclamation of expensive components can be worthwhile if safely removed. The paper pays particular attention to the challenges associated with underfill epoxy and the lack of a robust, repeatable process and the need for a better solution for pad cleaning and preparation. Additionally, the industry lacks a reliable approach to removing epoxy from printed circuit boards which needs to be addressed.