Reworking Area Array Devices and the Service & Repair Industry

Patrick McCall, Director of Engineering
PACE Incorporated
Southern Pines, North Carolina, USA

Area array and other components that do not have accessible leads once mounted on a printed circuit board (PCB) are extremely common in today's electronics. The ability to remove and replace them by the service industry is a critical competency. These types of components have gained significant acceptance and with each generation of smaller and more powerful products. Area arrays bring with them special process considerations when rework and repair is required. These include higher levels of process control, more precise control/application of heat, lead free solder considerations, and advances in component construction.

Over the last few years, the cost of the specialized equipment to handle these types of components has come down to where it now very affordable to independent service centers. The procedure/process for reworking area arrays is often very different from reworking standard surface mount components. It is a longer process and of course, there is more opportunity for mistakes. In general, the typical procedure includes the following steps.

Removal of Old Package and Site Preparation:

When removing a package because it has failed, a removal profile can utilize faster ramp rates and therefore shorter times because we are not concerned about the survivability of the package. Proper pre-heating and thorough PCB warming is critical to the success. Removing packages by only applying heat from the top should be avoided as this technique can result in pads being damaged or lifted. Package delamination is also a common problem that is becoming more common as multiple layers of varying materials are being used in the component's construction.

It is extremely important that all the solder joints under the package are liquidus before lifting the package from the PCB. Rework equipment that utilize devices to automatically lift a package after the removal profile is completed are common and generally work well. However, if they are used, the removal profile must be validated through the use of thermocouples to ensure proper temperatures are reached. Should one or a few solder joints not reach solder melt temperatures and the automated head lifts up, pads can and will be pulled off the PCB. Pads and traces for today's area array packages are very delicate and are extremely difficult to repair if damaged. The use of closed loop feedback systems when developing and validating profiles should be standard practice in every rework facility.

Site Preparation

Once a package has been removed, the rework site must be properly prepared for the next installation. Excess solder on the land patterns can usually be removed using one of a variety of conductive desoldering techniques that are readily available. These include the use of desoldering tools and solder wick. Careful attention to the process is important as temperatures over 200 degrees C combined with pressure can easily damage pads or the solder mask.

In general, PCBs that have vias through them are not good candidates for desoldering tools. While the desoldering tip is moved across the land pattern, the vias can become filled with solder. These are not easy to clean out and attempting to do so can result in damage to the via, solder mask or the PCB. The use of solder wick is effective but again, in the hands of an unskilled operator, using solder wick is risky. When using wick, it is important to place the wick on the PCB and to wipe a hot iron tip over the wick. As the solder melts it will "wick" into the copper braid. Do not slide the wick over the array as pads will be lifted and solder mask will be damaged. Wick specially designed for area array clean up is available.

Placement:

Placing area array packages is relatively easy and can be accomplished with commonly available techniques. Placement by hand is a viable option for packages with solder spheres over 1 mm by using a template or the silkscreen around the land pattern as a guide. A vision system can be used to ensure proper alignment and is usually preferred by most operators. Typically, a minimum magnification of 35 to 70 X is required depending on the pitch and size of the solder spheres.

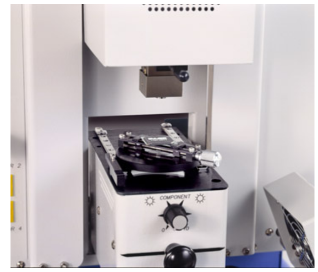

Figure 1

To ensure proper installation, the solder ball must be aligned to at least 50% of the pad on the PCB. In other words, when the device is placed on the PCB the accuracy must be at least 50% of the pad diameter in order to take advantage of the self-aligning properties of array packages. Packages that are aligned less than 50% can actually migrate one full row of the array, which results in a faulty installation.

Flux/Paste Application:

Flux must be used to have successful package installation. It may be used by itself or it may be combined with solder in the form of solder paste. Applying the proper amount of flux to the rework site is critical. Too little flux and the solder will not flow correctly, too much flux and out-gassing can occur resulting in voids within the solder joint or flux will be left over which can cause resistivity problems as well as have corrosive effects on the solder joint after reflow. There are a variety of methods for applying flux. See Figure 2.

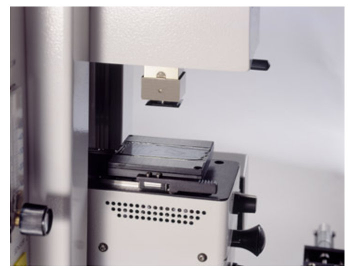

Figure 3

Solder paste can be applied using a variety of techniques. Some of the more common techniques used in rework include using spot stencils to apply it directly to the PCB and using component stencils to apply it to the component. It is often easiest to screen paste onto the component directly and to lift the stenciled component off of the stencil using the rework unit's pick-up mechanism. See Figure 3.

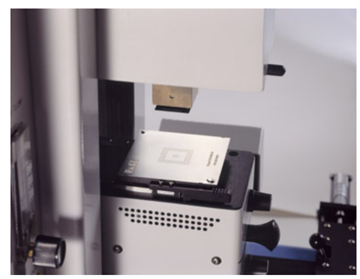

Figure 4

Heating options

A variety of heating types are available on a wide variety of systems. Most common are systems that use hot air or N2 to transfer the heat to the rework site. These systems all use a "nozzle" of some type to direct and contain the hot air to the rework site. The cost of ownership on convective systems is higher as nozzles must be purchased to match the size of each component to be reworked. Convective heaters are occasionally used on the bottom of the PCB. However, since bottom heaters are used to warm the entire PCB, the air is generally not contained using a nozzle so energy requirements are high and they are not efficient.

The second most commonly used heating technology is infra-red heating. IR heat is very stable, consistent and also penetrates materials better than air. IR heat is a form of radiant heat which will efficiently travel from the emitter to the target so containment nozzles are not needed. It is common to see systems that use IR heating on the bottom side and convective heating on the top side. When IR is used on the top side heater, it can be focused or directed using baffles or by shielding the PCB around the rework site using metal or metal tape.

Heat Application & Profiling

When installing Area array type components, a four or more zone profile best. The zones are typically: Pre-heat, Soak, Reflow, and Cool Down. The zones are intended to raise the temperature at the rework slowly, and precisely to minimize the thermal stress that can be generated within the PCB, component, and at their interconnection. Heat should always be applied from the top and the bottom of the installation site.

In the “Pre-heat” phase, bottom side heating ensures homogenous temperatures across the board. This keeps the PCB from warping, twisting, or flexing, during the process, which is essential for maintaining planarity of the installation site.

During “Soak”, the bottom side heater should continue to operate while heat is added from the top heater. The combination of top and bottom side heat application allows the installation site and package to reach a temperature of between 140 and 170degrees C and to stabilize. The stabilization should be maintained for 30 to 60 seconds, allowing the flux to fully activate and driving off any volatiles in the flux.

During “Reflow”, higher temperatures are applied from the top heater in addition from the bottom heater. It is important to apply heat slowly and evenly so the entire package warms to the reflow temperature uniformly. Temperature differences across a package of as little as 7 to 10 degrees C can cause damage. Reflow should always be achieved with the lowest temperatures possible, insuring the safety and reliability of the package as well as PCB.

Profiling can be accomplished using a site containing a previously installed package, or by performing an actual installation. Either method can be used to develop a reliable profile. Thermocouples of some sort should always be used to verify thermal conditions. Most systems have software features that make the profiling process easier or may develop the profile based on a few inputs.

Inspection

Once the package has been installed, it should be inspected. A wide variety of inspection methods are available. Usually for area array packages, X-ray systems are the most commonly used (See Figure 5), followed by video microscopes that look under the component at a 90 degree angle. After visual inspection is completed, electrical testing is usually used to verify the installation.

Figure 5