New Process for Advanced Packages (PBGA, CBGA, CSP and New MLF, LLP, LGA)

Paul Wood
Metcal
Menlo Park, CA

Advanced Packages Require a New Rework Process

The latest types of components launched by the leading component manufacturers have increased the need for process control in rework. These packages are referred to by the generic name Land Grid Array (LGA) or the specific package names Leadless Lead-frame Package (LLP) and Micro Lead Frame (MLF). These packages are unique because they contain multiple pad sizes and shapes on the same device. They are also soldered with all terminations on the underside of the package, similar to BGA.

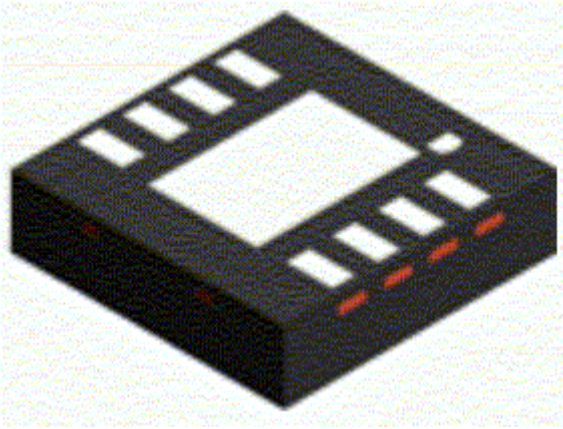
These components lend themselves to normal assembly techniques such as screen-printing the PCB, automatic pick and place, then convection reflow.

The LLP design has a large center pad that is often soldered to PCB as a ground or power connection. The soldered center pads can also act as a heat spreader to the PCB substrate. This helps keep the part cool and controls expansion of the part to the substrate so that the TCE (Thermal coefficients of expansion) tolerances of the part to the PCB board material are minimized.

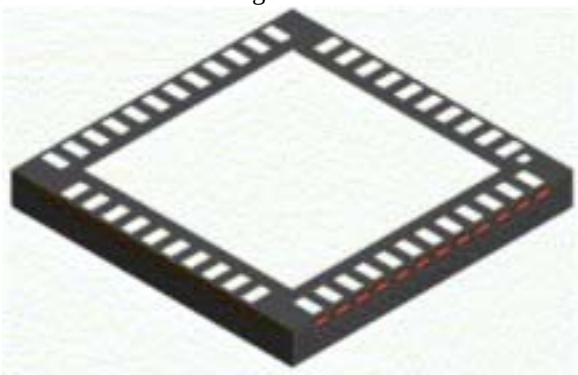
The rework process for LLPs requires all the pads to be soldered to the same height or the potential for small land connections could be an issue. The solder paste volume used for the center and array pads must be controlled to achieve good, reliable joints.

New Style Packages LLP Leadless Lead Frame Package

New, advanced packaging LGA components are more densely packed on the PCB and, therefore, need different processes to be successfully reworked. Because the pad sizes on the component are different, the solder volumes are difficult to control by using just a soldering iron to tin the lands on the PCBs. This is especially true of the sample components illustrated Figure 1.



Dual in-line 8 pin LLP with center connection for ground



LLP 44PIN with center pad

Laminate Conduction Packages

Figure 1 – Sample Components

Because the packages shown above all have different pad sizes on the same device, a new rework method has been developed and proven to be more consistent and reliable. Using the old rework process, where the pads of the PCB are “tinned” with a solder iron, the correct solder level needed to get consistent, equal solder joints was not achieved. Using this method, the larger pads tinned up to a higher height, causing the smaller pads to be lower in height. This is evident when using an x-ray, after reflow, different solder volumes on the pads are seen.

Only the Rework Process is Changed

Printing on a PCB, when fully populated, is difficult due to size and packaging density around the devices. Today, small chip components are often very close to the packages. To alleviate this problem, a method of printing on the package has been developed. This solution prints the standard thickness of paste used in assembly, but on the package. If this method is followed this will control the solder volume for replacement in the rework process.

Note: A good document to reference, in the assembly method of printing and designing stencil thickness and volume, has been written for one range of packages produced by National Semiconductor. See data and design information from National Semiconductor document Design and Process Document AN1187.

Rework Process is as Follows

Remove package from the PCB with a convection rework system, such as Metcal's APR-5000 Array Package Rework System, or a comparable machine.

Clean solder pad array on PCB to remove old unleveled solder using a small blade tip and solder station. Place the solder wick under the blade tip to soak up the solder. Next, clean the PCB with a cleaner and lint free wipe to remove flux residue left from solder wick.

Pick a new part and use the component clamp to secure in stenciling template. Pin 1 location is marked on the plate and under side of the component with a small location pad that is not soldered or printed. Print solder paste on the back of the template, as shown in Figure 2. The template comes with a spatula that matches the aperture size of the print window, so no pads are missed in printing. The spatula has a step etch that aids the printing process and provides very consistent results. One tip is to print twice at 90˚ to ensure the print windows are completely filled.

Figure 2 – Attaching the Component on the Stenciling Template

Solder paste education in the rework area may also be needed. Many operators need to be educated about cleanliness in operation and not leaving paste tubs open. A paste pallet is a good solution. The pallet is a small stainless plate that is used as a mixing and paste preparation plate. To use, take a small amount of paste out of the tub and reseal the pot. Using only a small amount will keep the pot of paste in good condition. Never use the back of a PCB for this, as the spatula will remove solder resist and fiber in to the joint. Also, never use solder paste from syringes, as this dispense paste at a different viscosity and will slump easily. Standard type 4 fine pitch paste works well on 20 mil pitch components.

Once paste has been applied, place template on rework machine and pick package out of template with paste on underside of package.

Inspect, on rework machine, with the vision system to look on the underside of the part. If all pads are printed correctly then alignment can be done and the component placed on the PCB. If not, the package will need to be cleaned and reprinted or a new part printed. One major advantage is that no paste is placed onto the PCB until the printed part is ready, eliminating the process of re-cleaning the PCB.

When placing the part, it is important to control this part of the process, so that the part is gently placed onto the PCB. If the part is placed with too much force, bridges may form. Usually, the solder paste is 5 mils and can be just placed or lowered a few mils onto the PCB, however, do not push too hard onto the PCB.

Once the component is placed on the PCB, the reflow stage needs to be done, similar to a full conveyer reflow oven. Because solder paste is used, a good solder profile needs to be performed to create good solder joints with no defects such as voids or opens. If a multi-zone reflow profile is not performed, some voids may appear in the large center pads.

The best method to assure proper soldering is to duplicate the oven profile. If this method is not available and live interactive software is available with the convection rework machine, then thermocouple the PCB next to the component and create a profile.

Use multi-zone reflow to control the solder paste reflow similar to that of production reflow solder ovens. This involves multiple preheating of air, including top and bottom control, the same as a full convection reflow oven.

1. Preheat, soaking, ramp, reflow, and cooling zone
2. This will solder these packages to the same standard as a production assembled part.

Figure 3 and 4 shows a BGA Stenciling Template with a component in the template. This apparatus was originally made for LGA components and then applied to standard BGA and CSP parts. By printing on the part, the solder level can be increased to that which is used in production assembly, thus increasing the quality of rework and reliability. This method is ideal for LLP and similar parts, with their different size pads, because they can be reworked easily and reliably.

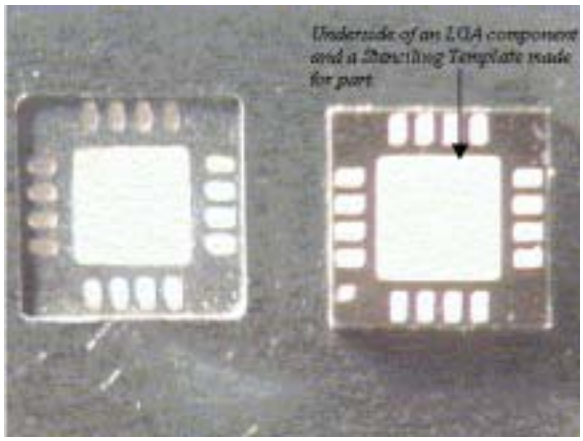


Figure 3 – Underside of LGA Component

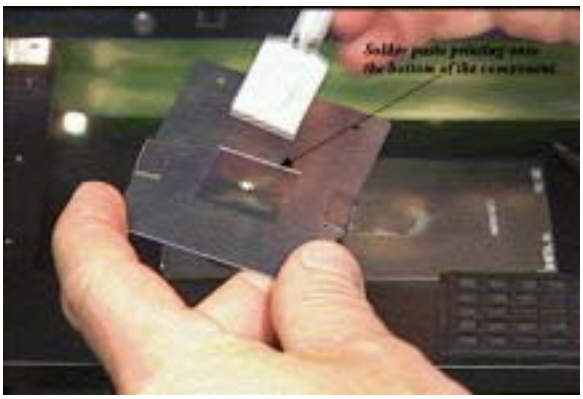


Figure 3 – Solder paste printing onto the bottom of the component

Placement of Printed Part

Placement must be done with a prism to overlay the component image to the PCB pads. Hand placement is not recommended due to the lack of accurate depth control needed to place the part without squashing the solder paste. (See Figure 5.)

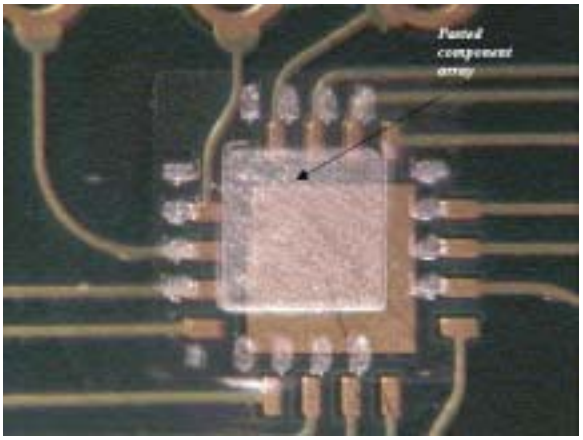


Figure 5 - Image of Pasted LLP and PCB Land Array

This method is also a good solution for CSPs to achieve the same stand off height as a production soldered part.

A CSP soldered with 12 mil solder balls will only achieve a stand off height of 4mils with flux process. If it is printed using stenciling templates, the stand of height will raise to the same height as a production soldered part that is 6 to 7 mils height. (See Figure 6.)

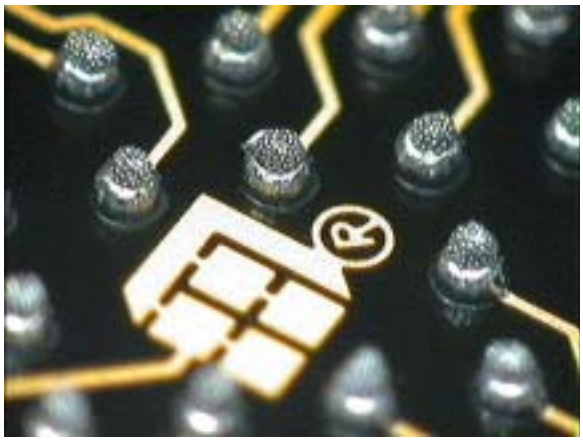


Figure 6 - Printed µ BGA Package

Reflow of Packages is the Same as Production Soldering

Care must be taken to ensure the parts reach the correct maximum temperature and only stay above the melting point for the specified time. This can vary from 30 to 60seconds depending on paste manufacturer. If parts are reflowed for to long, grainy joints can occur and cause reliability problems. Also, if the reflow is too fast, a void can occur, also affecting reliability.

An example of voids can be seen on an LLP 44 pin, Figure 7, where the reflow was too fast and the paste did not have time to settle. This can also be caused when a small part does not have the mass of a BGA, for example. Therefore, less reflow pressure is needed along with a low flow rate in the reflow head.

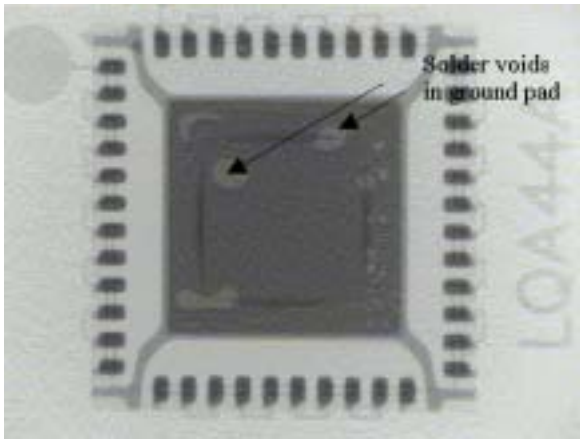


Figure 8 - An example of voids

When conducting reflow, the typical 3 zones and cooling reflow profile will look like the chart in Figure 9.

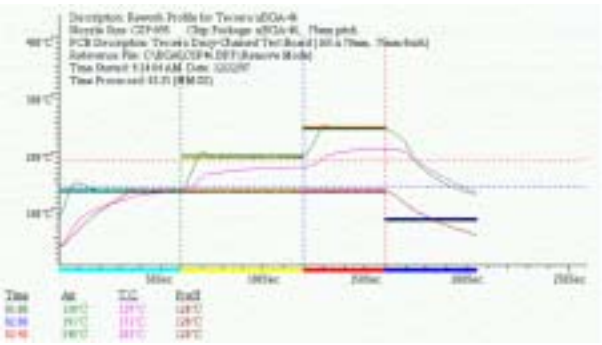


Figure 9 – Typical 3 Zones and Cooling Reflow Profile

Cooling Considerations

Cooling is an important stage to get the PCB temperature down, to maintain control of the time above reflow.

If hot plates or IR under PCB heating are used, time above reflow is more difficult to control because the plates or heaters will still radiate heat under PCB rework site after the heating cycle has finished. The use of convection allows heating to be rapidly changed from hot to cool and helps cool the PCB quicker thus minimizing the time above reflow. This is more important in lead free applications where the reflow temperature is up in the 225 C to 235 C from 183 C. The time above melting is usually only 15 to 30 seconds.