POWERED BY Panasonic Electronic Components

Soldering Process Considerations for Land Grid Array Modules

The reflow process is dependant on many parameters; this application note is presented as a guide to soldering LGA modules. Manufacturers are responsible to optimize the process.

Introduction

Land Grid Array is a packaging technology with a square grid of contacts on the underside of a package. The contacts are to be connected to a grid of contacts on the PCB. The contacts are made by using solder-paste.

LGA packaging is related to Ball Grid Array (BGA) although land grid array packages need solder paste before they can be soldered down. BGA packages have solder-balls as their contacts, and are soldered down without solder-paste.

The LGA solder interconnect is formed solely by solder paste applied at board assembly because there are no spheres attached to the LGA. This results in a lower stand-off height of approximately 0.06 mm to 0.10 mm, depending on solder paste volume and printed circuit board (PCB) geometry. • LGA also eliminates risk that customers receive components with missing or damaged spheres due to shipping or handling.

Manufacturing with LGA

Land Design

The solderable area on the mother board should match the nominal solderable area on the LGA package 1:1.

Solder Methods

Critical factors to ensure successful circuit board assembly with LGA devices are the design of the solder paste stencil, the solder paste and reflow profile used, and the PCB pad design. This section recommends stencil attributes that have been known to succeed, such as solder stencil thickness, aperture diameter, paste release characteristics, and practices to ensure consistent solder paste volumes that exceed recommended minimums.

Solder Paste Printing

Use automatic equipment to screen print solder paste onto the PCB and a pick and place system to put the RF module LGA package onto the wet solder paste. Typically, an automatic conveyor will take the board into a reflow furnace to solder the components onto the PCB. Normally, solder paste is applied by automatically dispensing paste through a metal stencil that has been machined to correspond with the pattern of solderable surfaces on the mother board. Panasonic has found good results with a 5 to 6 mil (120 µm to 150 µm) thick stencils and the apertures on the solder paste stencil being the same size as the pads. Industry practice typically uses stencils from 4 mils to 8 mils in thickness with the thinner stencils typically being associated with finer pitch, more closely spaced application. However, local manufacturing experience may find other combinations of stencil thickness and aperture size, give good results.

It is strongly recommended that the process should be controlled to maintain a consistent quantity of solder paste on the same sized lands. If a stencil becomes partially blocked, then the part may be electrically functional. However, a fragile joint will form. This will result in far less mechanical durability for such a joint than the recommended solder joint.

Our used temp. profile for reflow soldering

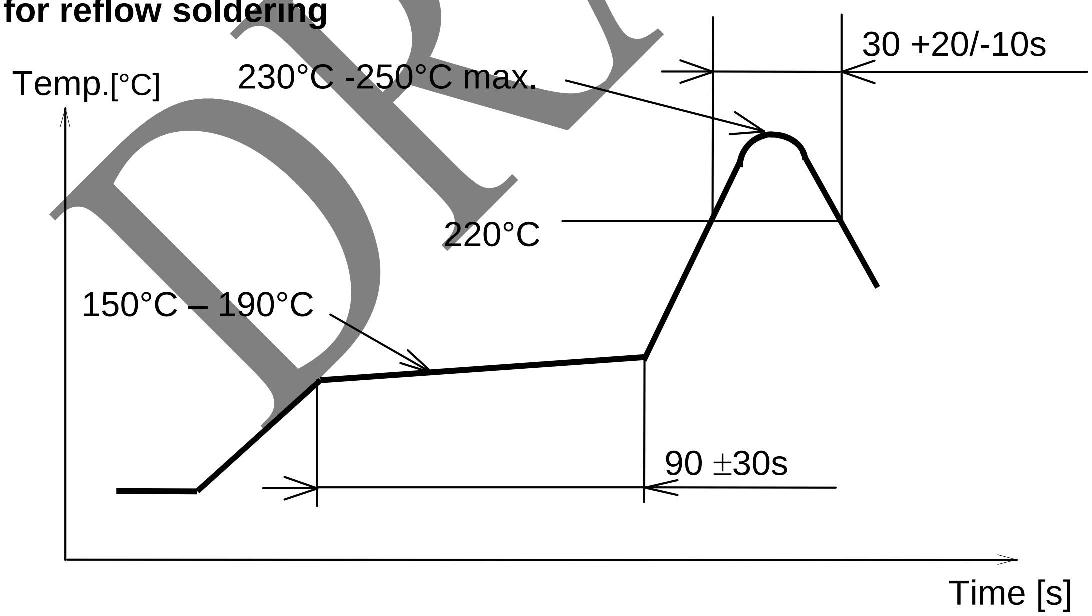


Figure One: Temperature Profile for Lead Free Solder

Reflow for Lead Free Solder Paste

Optimal reflow profile depends on solder paste properties and should be optimized and proven out as part of overall process development. The following guidelines for solder reflow represent good soldering practices to help yield high quality assemblies with minimum rework. It is important to provide a solder reflow profile that matches the solder paste supplier's recommendations. Some fluxes need a long dwell time below the temperature of 180 ºC, while others will be burned up in a long dwell time. Temperatures out of bounds of the solder paste flux recommendation could result in poor solderability of all components on the board. All solder paste suppliers should recommend an ideal reflow profile to give the best solderability. Panasonic has achieved good results with a peak temp of 230 ºC to 250 ºC and a dwell time above 150 ºC for greater than 60 seconds and less than 120 seconds as shown in Figure 1.

In IR or convection processes the temperature can vary greatly across the PC board depending on the furnace type, size and mass of components, and the location of components on the PCB. Profiles must be carefully tested to determine the hottest and coolest points on the assembly. The hottest and coolest points should fall within recommended temperatures in the reflow profile

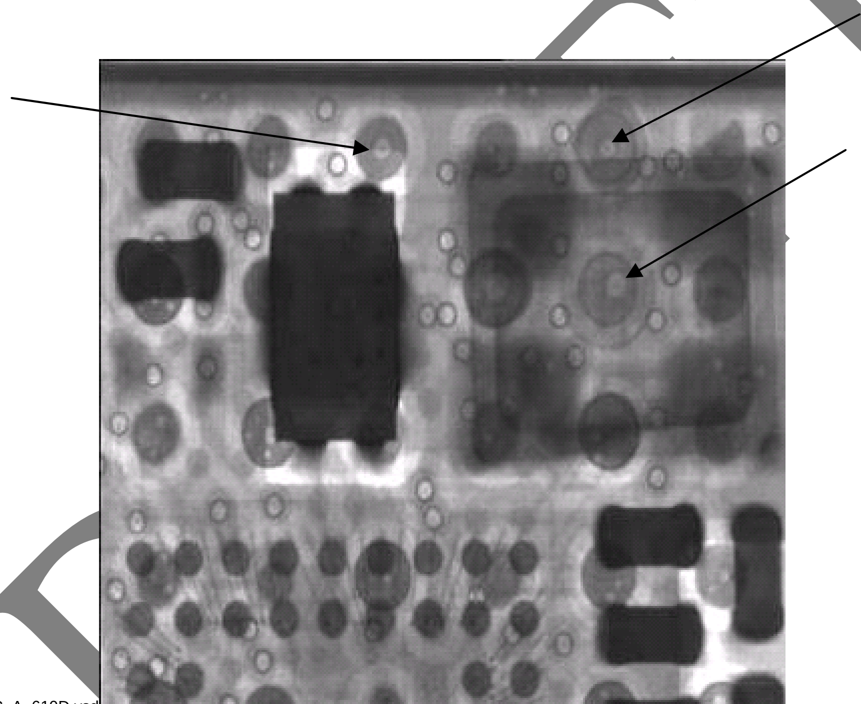
Cleaning Under LGA

Due to the lower stand-off height of the LGA device, no-clean solder pastes are recommended. Full drying of no-clean paste fluxes as a result of the reflow process must be ensured. This may require longer reflow profiles and/or peak temperatures toward the high end of the process window, as recommended by the solder paste vendor. Instances of uncured flux residues after reflow have been encountered with LGA. It is believed that uncured flux residues could lead to corrosion and/or shorting in accelerated testing and possibly the field. The presence and extent of uncured flux residues can be detected by mechanical removal of the module after reflow as part of the overall assembly development process. Cross-sectioning and flat sectioning are also recommended to assess not only residues, but overall joint geometry. Solder flux technologies have improved dramatically in recent years, to the point that most of the industry is using no-clean fluxes. Some of these fluxes require specific reflow profiles. The flux vendor's recommendations should always be followed precisely taking precedent over any the guidelines described in this application note.

Voids:

Expected Void Content and Reliability

The likelyhood of voids can be larger on LGA modules than for modules with BGA or leads. The outgassing flux at an LGA solder joint must migrate a longer distance to the surface of the solder and it has a relatively small surface to the air.
The void content of the module must conform to IPC-A-610D (25% or less voiding area/area). See Figure Two.



Voids_IPC_A_610D.vsd

Figure Two: X-ray Picture Showing Voids Conforming to IPC-A-610D

Consider the following parameters, to reduce void area.

Solderability on module and PCB

Bad solderability is often connected to oxidation and has therefore a major impact on voiding. Flux gets trapped on oxidized surfaces. In general, Ni/Au pads have fewer voids than HASL and OSP.

Solder paste

Using higher activity flux removes oxide rapidly so less flux will be trapped.

Voiding increases with increasing solder paste exposure time, since long exposure time will result in more oxidation and moisture pickup.

Pad size

A large soldering pad means that the outgassing flux must travel more distance to the surface of the solder, and will thereby create more voids.

Solder paste

Smaller powder size and higher metal load means more metal surface to deoxidize and thereby more trapped flux and voiding. Higher metal load also means higher viscosity making it more difficult for outgassed flux to escape from the solder.

Stencil thickness

A thicker solder paste stencil creates more surface area which improves outgassing and decreases voids.

Temperature soldering profile

When the preheat time is too short the flux does not get enough time to react causing it to get trapped in the solder and create voids.

When the reflow time is too long, larger voids are created

When the reflow time is too short, the void area increases.