flex Power Modules

APPLICATION NOTE 317

LGA design and assembly

Abstract

This application note describes how to work with LGA (Land Grid Array) designs and what to consider during assembly.

On our website you can easily filter our products by mounting/LGA design.

This application note is applicable to the following products:

- BMR461
- BMR466
- BMR510
- PMU

Contents

Introduction

LGA (Land Grid Array) is a connection method with a grid of contact pads on the connection side of the PCB. The grid of contacts consists of pads without any pre-deposit of solder. Interconnection between an LGA module and a connecting PCB is achieved by printing solder paste on the connecting PCB which will melt during the reflow process and create a solder interconnection between the module and connecting PCB. Using an LGA design has some benefits over a BGA package and can be seen such as:

- LGA has a lower mounted height compared to BGA.
- The robustness of the LGA module is better since there is a lack of extra solder bumps or solder balls that can be damaged during handling or transport. BGA has solder spheres that can be damaged.
- LGA is compatible with BGA in production since the same assembly procedures can be used.
- LGA soldered with SAC solder has proven to be more robust compared with BGA during temperature cycling.
- The risk of contamination due to the non use of solder in the pads are minimized since gold plated pads are used at Flex.
- LGA can be used with both leaded or lead free assemblies

Design considerations

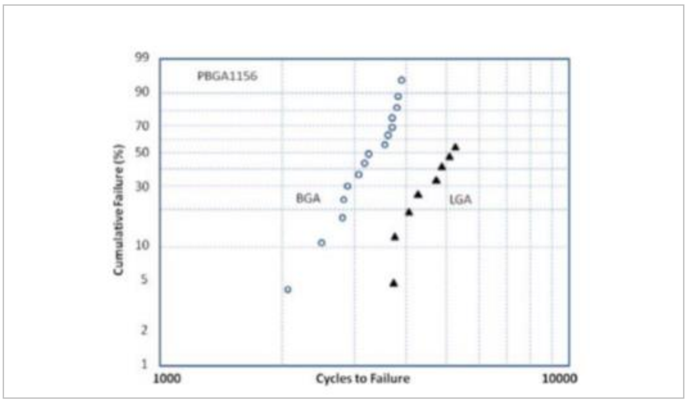
In order to achieve a good soldering result, some considerations should be taken into account when designing the PCB where the module will be mounted.

Robustness of LGA terminations

The main difference between LGA solder joints and most other solder joints is the LGA solder joints´smaller thickness. A solder joint with smaller height is expected to have considerable lower reliability. Comparisons between Sn/Pb BGAs and corresponding Sn/Pb soldered LGAs confirm this and show that the LGAs do have significantly lower reliability than the BGAs.

Picture 1: Weibull plot showing cycles to failure in -40°/125°C thermal cycling – Sn/Pb BGAs and corresponding LGAs

However, when soldering with Sn/Ag/Cu (SAC) solder alloys, the results for LGAs increase and their

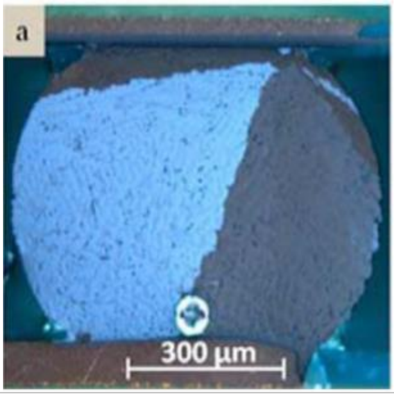


Picture 2: Weibull plot showing cycles to failure in -40°/125°C thermal cycling – SAC305 PBGA1156 and LGA 1156

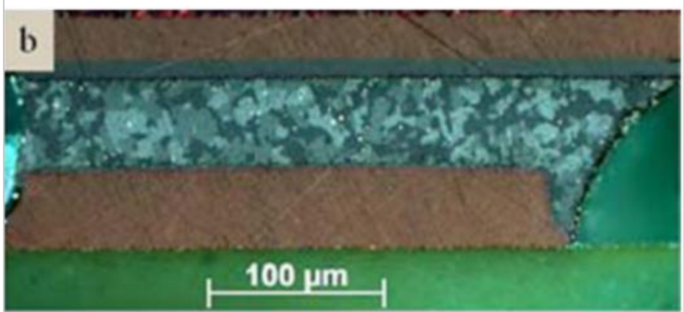
solder joint reliability is often as good, and in some cases even better than BGAs.

A test with SAC305 BGA and its corresponding LGA is shown in picture 2 where the LGA has superior reliability compared to the BGA.

The remarkably good reliability results for SAC soldered LGAs are explained by the difference in microstructure between LGA and BGA solder joints. The lower volume of solder means more undercooling before solidification. The high degree of undercooling for LGA solder joints gives a very fast solidification that most often gives an interlaced grain structure in comparison to the "Beach Ball" microstructure present in the BGA solder joint.



Picture 3: Cross polarized image of a 30 mil SAC305 BGA solder joint demonstrating the characteristic 'Beach Ball' microstructure.

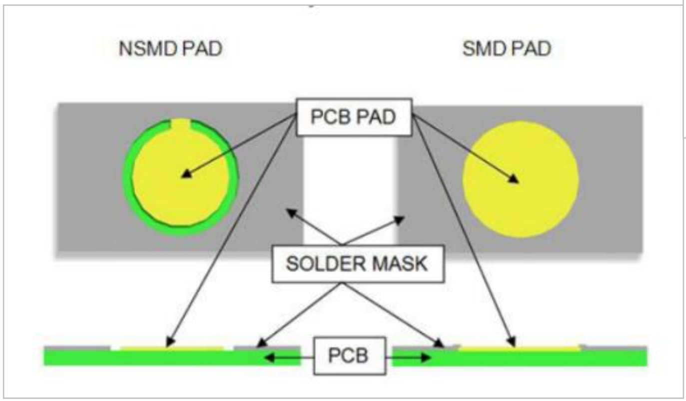


Picture 4: Cross polarized image of a 30 mil SAC305 LGA solder joint exhibiting an interlaed twinned grain structure

Interlaced twinned Sn dendrite microstructure gives harder solder joints, slower creep behavior and a delayed recrystallization.

Land pattern types

A common problem when soldering an LGA module can be the presence of voids in the solder joints created under the escape of gases during the soldering process. This can be minimized using non-solder mask defined (NSMD) pads. NSMD pads have a solder mask opening that is larger than the PCB pads, while solder mask pads (SMD) have a solder mask opening that is smaller than the copper pad. Depending on the application either SMD or NSMD can be used. The SMD land pattern is often recommended to be used if there is a risk for impact failures in for example hand held devices where optimized mechanical durability is important. The NSMD land pattern is recommended where reliability is at risk and solder fatigue failure can occur in applications like base stations, industrial systems, avionics etc.



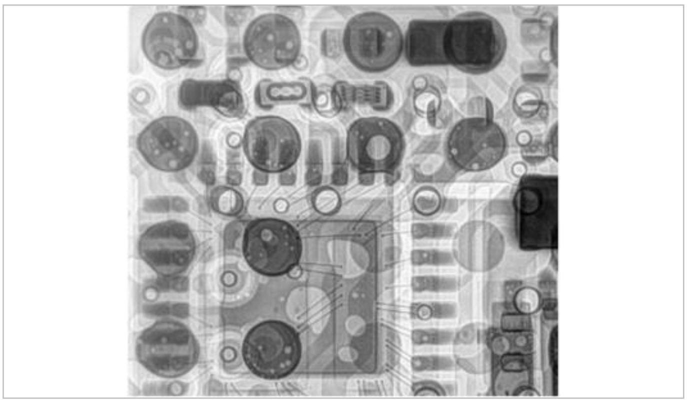
Picture 5: NSMD & SMD pad in comparison

Solder stencil openings

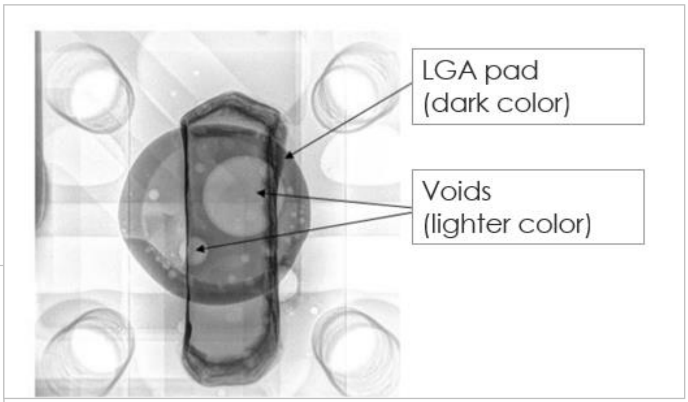
For LGA devices, the solder mask openings should be a mirror image of the bottom of the device. The land on the PCB should be the same size as the land on the device. Manufacturing sites may change the size of the paste opening in order to produce optimum yield in surface mount assembly, based on stencil thickness, local processes etc.

Voids in LGA pads

As mentioned before, LGA pads are subject to voids created by outgassing during the soldering process. Flex Power Modules has tested and documented the voids in an LGA based module to view a typical soldered LGA module.



Picture 6: X-ray picture of module



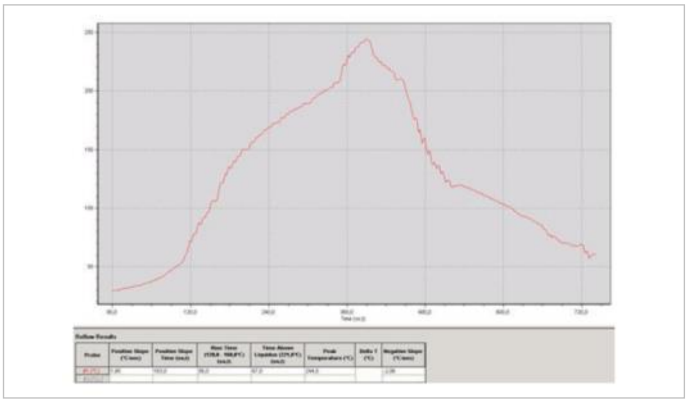
Picture 7: Close-up of the LGA pad with voids

The following conditions were used during the trial:

- Solder paste: Nihon Almit LFM-48W TM-Hp (lead free Sn/Ag/Cu solder paste)
- Stencil thickness: 0.1 mm
- LGA pads: 1 mm size and with filled micro-vias.
- Reflow profile according to picture 8 on the next page

Assembly considertions

In order to obtain the best possible results when soldering the module onto the mother board some assembly guidelines should be considered.



Picture 8: Reflow profile

The module is photographed using an x-ray to be able to see through the components and focus on the solder interconnection between the LGA pads of the module and the test board.

The whole module can be seen in picture 6 and on this module 31 circular solder pads and on square solder pad is the solder interconnection between module and test board. The pads have a darkish color and the voids have a grey/white color with mostly circular shapes present inside the pads. A close-up of one of the pads can be seen in picture 7 where the white/grey rings are voids.

Voids are inevitable in a LGA solder interconnection and the example above shows a normal distribution of voids in the solder after reflow. In some places the voids are close to 40% of the pad area but most pads have a void area of 5-10%. This example of a soldered LGA module is considered to have a robust interconnection towards a test/customer board.

Preparations

The boards are baked prior to packing in a moisture barrier bag to minimize the moisture in the modules which during the reflow process could cause outgassing from the PCB and contribute to excessive voiding in the soldering joints. If the products have been unpacked and stored in an uncontrolled environment and no longer can be considered dry, the modules must be baked according to JEDEC standard J-STD-033. Please read the moisture caution label on the moisture barrier bag for more information regarding baking and storage.

CAUTION

LEVEL

This bag contains

MOISTURE-SENSITIVE DEVICES

If Blank, see adjacent bar code label

1. Calculated shelf life in sealed bag: 24 months at < 40 °C and < 90% relative humidity (RH)
2. Peak package body temperature: ______ °C
   If Blank, see adjacent bar code label
3. After bag is opened, devices that will be subjected to reflow solder or other high temperature process must
   a) Mounted within: ______ hours of factory
   If Blank, see adjacent bar code label
   conditions ≤ 30 °C/60%
   b) stored at <10% RH
4. Devices require bake, before mounting, if:
   a) Humidity Indicator Card is > 10% when read at 23 ± 5 °C
   b) 3a or 3b not met.
5. If baking is required, devices may be baked for 48 hours at 125 ± 5 °C

Note: If device containers cannot be subjected to high temperature or shorter bake times are desired, reference IPC/JEDEC J-STD-033 for bake procedure

Bag Seal Date: ______
If Blank, see adjacent bar code label

Note: Level and body temperature defined by IPC/JEDEC J-STD-020

Picture 9: MSL label

To avoid contamination on the soldering pads extra care has to be taken when handling the boards. Clean soldering surfaces do not generate as much gases when the flux reduces the metal oxides or reacts with contaminants during the soldering process.

The module should be handled in an ESD safe way to prevent damages induced by electro static discharges.

Solder paste

To obtain a high quality soldering, a reliable solder paste is a key factor in producing high yield assemblies.

A high flux activity solder paste with good wetting abilities will clean the pad surfaces fast and hence decrease the gases that are created during the reduction of metal oxides. This type of solder paste will give the gases more time to escape before the solder solidifies.

When looking at voiding properties a huge difference between different solder pastes have been spotted during several tests of different solder paste materials.

In order to minimize the risk of solder bridges, a solder paste with good slumping characteristics shall be used.

Another important factor is the handling and storage of the solder paste. Contamination and age are two factors that will degrade the performance of the solder paste and the storage conditions should be followed according to the manufacturer data sheets.

Screen printing

When designing a screen printing stencil the recommended mounting mode for the specific component should be followed. A typical stencil thickness ranges from 4 to 8 mils (100-200 µm) in accordance with the industry standards. The thinner the thickness of the stencil the finer the pitch of the specific component is a typical design rule. This is a guideline and usually the manufacturing site has experience in what combination of stencil thickness and module size that gives the best result.

If problems with solder bridges and solder splatter should occur, a feasible solution could be to reduce the stencil apertures (normally by 1 to 2 mil).

The printed deposits shall have a smooth, flat or rounded surface. Irregular surface topography may lead to air entrapment between the LGA pad and the paste deposit. Adjust the printer settings to achieve a clean deposit if needed.

The use of automatic equipment will give the best result when screen printing the solder paste onto the PCB. A consistent thickness of the solder paste on the PCB pads will ensure a robust solder connection. If a stencil becomes partially blocked a fragile joint will form although the part may be electrical functional but the mechanical durability in the solder joint will be degraded.

Reflow soldering

A ramp-soak-ramp reflow process is generally preferred over a direct-ramp process because the extended soak period and additional time over liquidus tend to reduce the size and number of voids in the solder joints due to gas entrapment.

In order to minimize the risk for solder bridges or solder splatter, do not use too high temperature increase ramp rate in the reflow process.

For SAC alloy soldering, it is an advantage to cool with a rather high cooling ramp rate. If the maximum recommended cooling ramp rate of -4° C/s is used the solder joints will undercool more and the probability increases to get so called interlaced twinned tin dendrite solder joint microstructure which improves the reliability.

Location at customer board

Robustness of the mounted LGA package is highly depending on the location on the mother board. Areas with influence of mechanical stress and high temperature will decrease the robustness of the terminations between the LGA package and motherboard.

To reduce stress in solder joints and improve mechanical durability, a rigid PCB is essential and the thicker the PCB the more rigid which will have a positive effect on the robustness of termination.

A recommended clearing distance between module and nearby components is minimum 1 mm to facilitate rework. The larger clearance the better. There are a lot of specialized equipment and different types of hot air nozzles available, and if using a nozzle that goes down towards the motherboard's surface a clearance area of approximately 3 mm is needed.

Rework

Removing the LGA module from the motherboard

When removing an LGA module from the motherboard for rework some steps need to be taken.

A sufficient fixture is needed to fix the motherboard during the process to keep it stable.

For thick and large motherboards a powerful reflow station needs to be used due to the excessive amount of copper that needs to be heated up. A larger heater from underneath might be needed to obtain sufficient heat. A hot air reflow system with top and bottom heaters is recommended.

The motherboard should be preheated to approximately 125°C to avoid high thermal gradients which can lead to problems when solder does not reach its liquidus temperature.

An appropriate nozzle must be used to conduct the hot air to the LGA component so that the LGA module can be removed from the mother board.

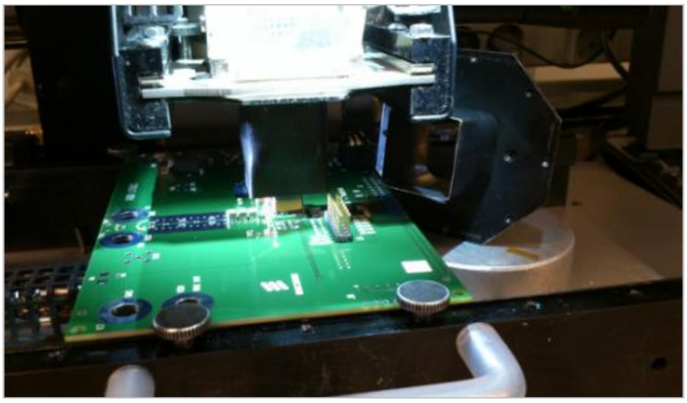
For the reflow station use the reflow profile recommended for the mother board and module (maximum 245-260 degrees). Mark pin number 1 both on the module and motherboard before removal of the module to minimize the risk for incorrect mounting when soldering the new module onto the mother board.

Specific components sensitive to heat can be protected by heat resistant tape.

In some cases where a large choke covers a large area of the module and picking up of the module is done by lifting the choke it is necessary to secure the choke before lifting to avoid the choke legs getting loose from the module PCB.

This can be done by adding glue between the choke and the PCB and should be applied at minimum two parallel sides of the choke. Use a glue that will harden when heated and this before reaching the melting temperature of the solder.

Removal of the module is preferably done by a vacuum pick up tool or can be done by hand using a pincher when the solder has melted. If the module will be repaired after removal be sure not to cover any components or soldering areas with glue. If the module will be scrapped after being removed, glue can be applied on all sides of the choke to secure it to the PCB of the module



Picture 10: Reflow station with air nozzle

After removal of the module

After the removal of the module from the motherboard, clean the pads on the motherboard carefully using a soldering iron and a soldering wick (braided copper). Afterwards clean the area with alcohol. Be careful when removing the soldering wick from the PCB pad so that the PCB pad does not get damaged.

Re-solder module to the motherboard

Use preferably a hand dispenser with a syringe and needle to dispense the solder paste on the pads of the motherboard. Be careful not to dispense too much solder which could create voids in the solder joint. Place the new module by hand and use the same reflow profile in the reflow oven when re-soldering the module to the motherboard.

Inspection of solder joints

Visual inspection of the solder joints of an LGA package is not possible due to its low height and array of solder joints."

The use of an X-ray or high precision camera system capable of viewing parallel to the x-y plane is necessary to inspect the solder joints after reflow or a rework of the module.

Flex Power Modules

Flex Power Modules, a business line of Flex, is a leading manufacturer and solution provider of scalable DC/DC power converters primarily serving the data processing, communications, industrial and transportation markets. Offering a wide range of both isolated and non-isolated solutions, its digitally-enabled DC/DC converters include PMBus compatibility supported by the powerful Flex Power Designer.

EMEA (Headquarters) | Torshamnsgatan 28 A, 16440 Kista, Sweden

APAC | 33 Fuhua Road, Jiading District, Shanghai, China 201818

Americas | 6201 America Center Drive, San Jose, CA 95002, USA

pm.info@flex.com
flexpowermodules.com
flexpowerdesigner.com
youtube.com/flexintl
twitter.com/flexpowermodule
flexpowermodules.com/wechat
linkedin.com/showcase/flex-power-modules