International Rectifier

APPLICATION NOTE

Application Note AN-1028

Design, Integration and Rework Guidelines for BGA and LGA Packages

Table of Contents

Introduction

BGA (Ball Grid Array) and LGA (Land Grid Array) devices are high-performance microelectronic devices designed to provide efficient and reliable operation. To ensure that their performance and reliability are not compromised, it is important to follow closely the guidelines in this application note.

Design considerations

Substrate

To optimize reliability and thermal performance, the printed circuit board (PCB) onto which the device is mounted must be well designed and well fabricated. It should be constructed from glass-reinforced epoxy laminate (FR-4) or polyimide and meet IPC-A-610 specifications.

PCB layout

The layout should maximize thermal conductivity around devices through the use of large copper areas. Figures 1 and 2 show examples of recommended layouts for BGA and LGA devices, with copper areas shown in green and solder mask openings in yellow.

The layout should use large copper power planes and solder mask defined (SMD) openings for device lands. Using SMD pads, instead of non-solder mask defined (NSMD) pads, optimizes thermal and electrical performance while still exceeding all reliability requirements. Figure 3 shows the differences between SMD pads and NSMD pads. For more information on optimizing the thermal and electrical performance of layouts, refer to AN-1029:

www.irf.com/technical-info/appnotes/an-1029.pdf

Layouts should also use vias for both thermal and electrical purposes. To help remove heat, use as many vias as possible under and around the device.

AN-1029 includes recommendations for using vias with BGA devices.

For LGA devices, vias in areas outside land pads (Figure 4) can be of various types: capped, plugged, tented, un-capped and un-plugged. Figure 5 shows capped, plugged and tented vias. Typically, vias have a drill hole of 0.33mm, a wall thickness of 0.020mm (minimum) and capture land of 0.64mm (diameter).

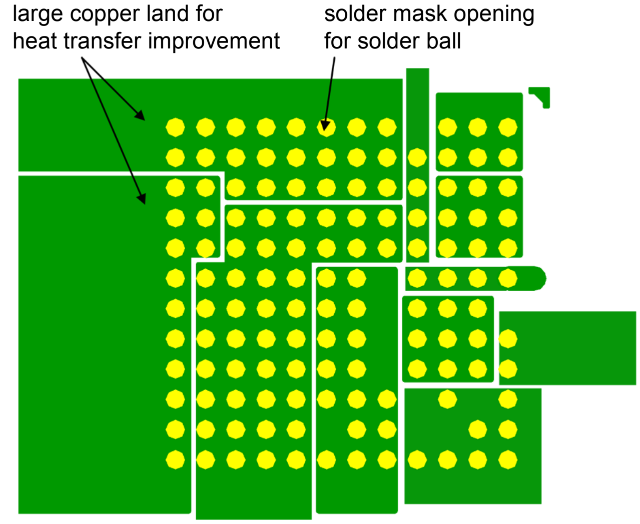

Figure 1 Recommended PCB design – BGA devices

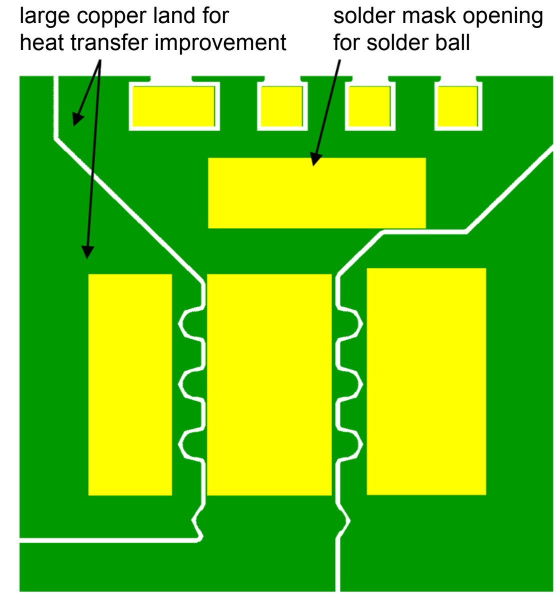

Figure 2 Recommended PCB design – LGA devices

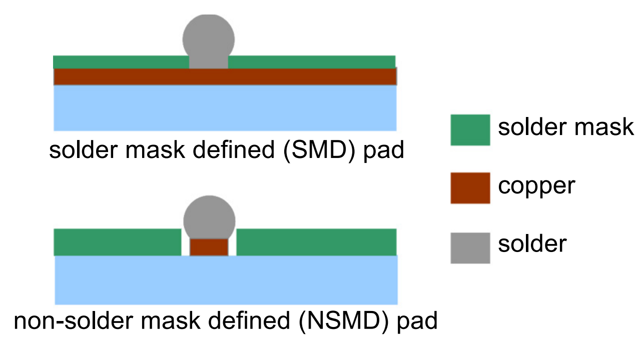

Figure 3 Comparison of SMD and NSMD pads

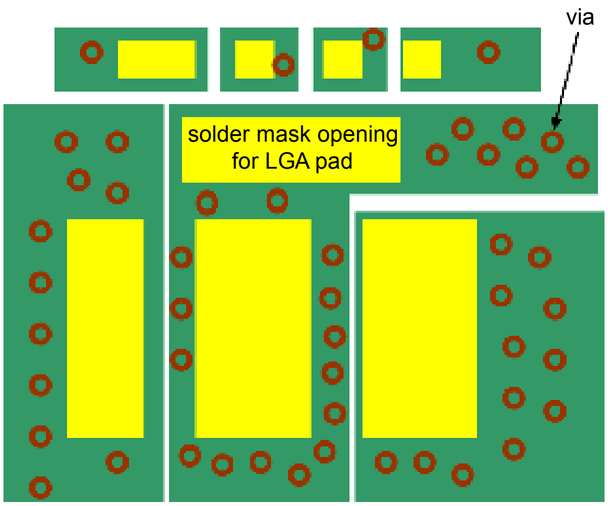

Figure 4 Vias placed around pad openings

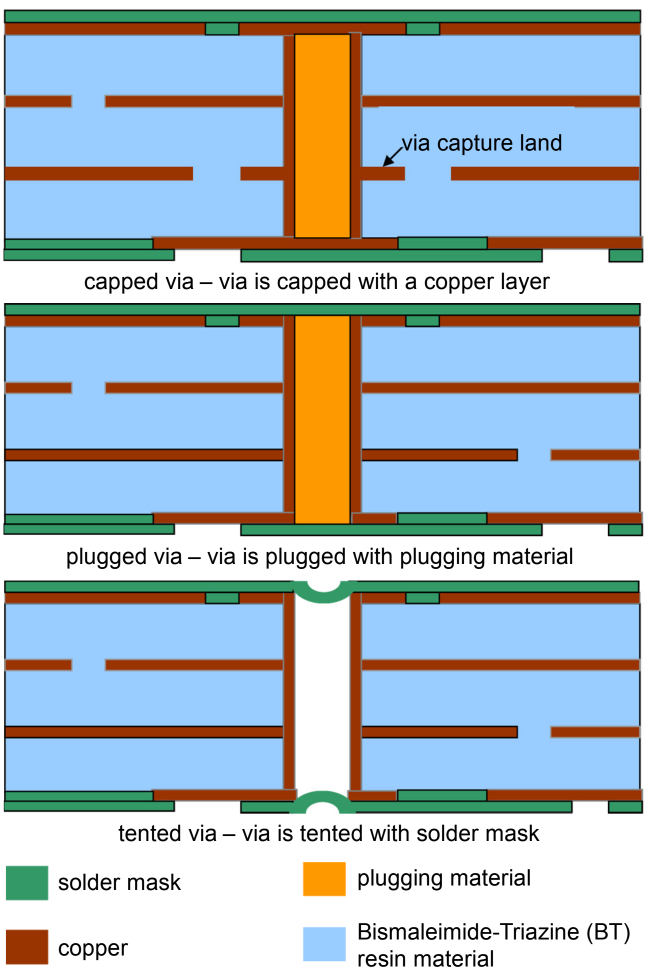

Figure 5 Capped, plugged and tented vias

Any vias within solderable areas (Figure 6) must be capped or plugged to prevent solder flowing through them during reflow. If solder flows through vias, there will be inadequate solder to achieve good joints. Solderable areas must be as flat as possible. If there are low points around the vias, the solder will pool there causing voids in surrounding areas.

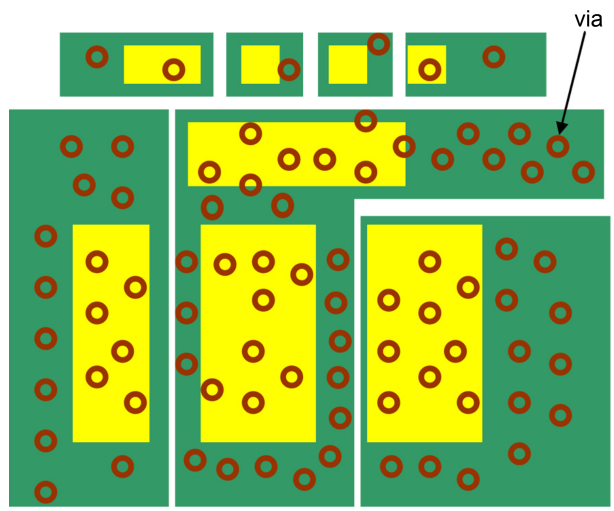

Figure 6 Vias placed in and around pad openings

Solder mask openings

As already stated, land pads should be solder mask defined (SMD).

The solder balls on BGA devices are 0.5mm in diameter and require solder mask openings of 0.4mm in diameter (Figure 7). This optimizes stand-off height and reliability.

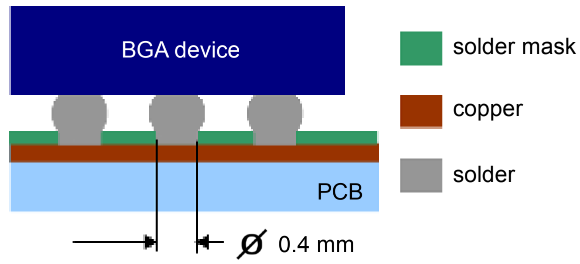

Figure 7 Recommended solder mask openings – BGA devices

For LGA devices, the solder mask openings should be a mirror image of the bottom of the device (Figure 8). The land on the PCB should be the same size as the land on the device. Figure 9 shows the underside of an LGA device. If pad 8 is 5mm x 2mm, for example, the solder mask opening for pad 8 on the PCB should also be 5mm x 2mm. This gives the best reliability.

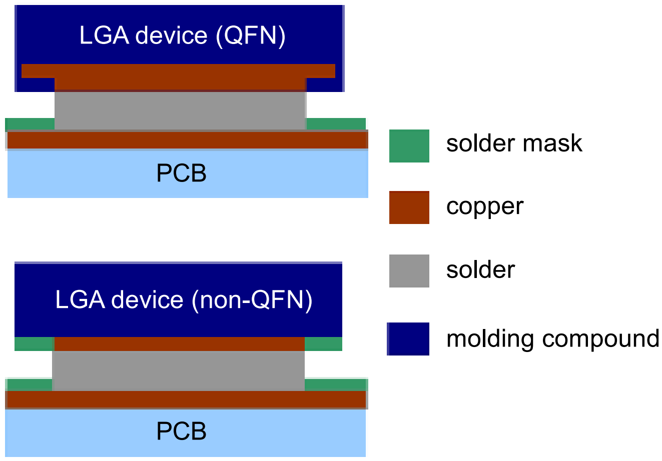

Figure 8 Recommended solder mask openings – LGA devices, QFN (Quad Flat No-Lead) and non-QFN

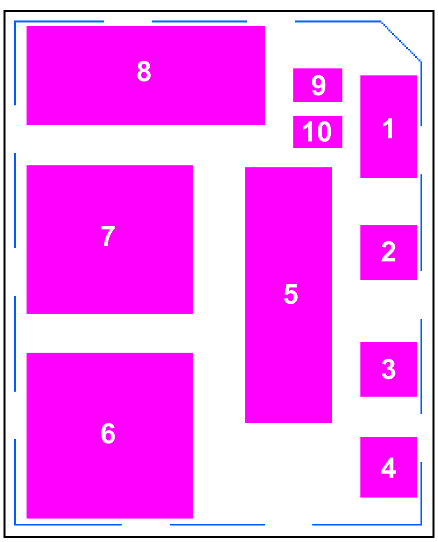

Figure 9 Bottom view of an LGA device

Terminations

For lead-free devices, nickel/gold or tin terminations are recommended. For standard devices, most terminations are acceptable; gold, tin or HASL (hot-air solder leveling) are recommended.

The underlying copper layer should be 1.0–2.0 oz. When using gold terminations, the gold layer should be 0.5–1.2 microns and a nickel layer of at least 5.0 microns should be used as a barrier. The gold layer should not be thicker than 1.2 microns.

When using a tin finish, both gray and white tin (α-tin and β-tin) are acceptable.

Assembly considerations

Storage and baking

BGA and LGA devices are supplied in tape-and-reel form and packed in sealed, nitrogen-purged, antistatic bags. Devices in unopened bags or stored in a non-condensing nitrogen environment have a shelf life of at least 12 months from the bag seal date. Devices have a Moisture Sensitivity Level (MSL) of 3 or above.

The devices should be handled in accordance with the IPC/JEDEC J-STD-033A standard. If they are exposed to the ambient environment before reflow soldering, they should be baked and dried following the instructions in Tables 4-1 and 4-2 in the standard. This ensures that any moisture absorbed during exposure is expelled before board mounting.

Stencil design

For BGA devices, a stencil with a thickness of 0.15mm is recommended, with openings of 0.4mm in diameter. Figure 10 shows a typical stencil design; the red areas represent the stencil openings for solder balls.

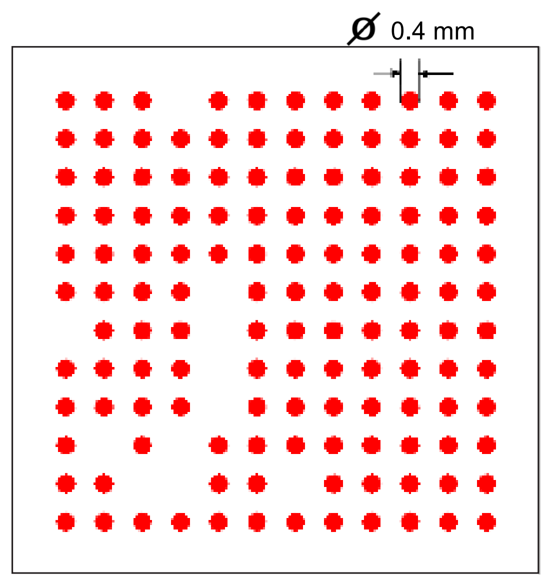

Figure 10 Typical stencil design – BGA devices

For LGA devices, a stencil with a thickness of 0.15mm is recommended, with openings that are 0.05mm less in each dimension than the corresponding pads on the device (Figure 11). Refer to the device datasheet for specific stencil recommendations.

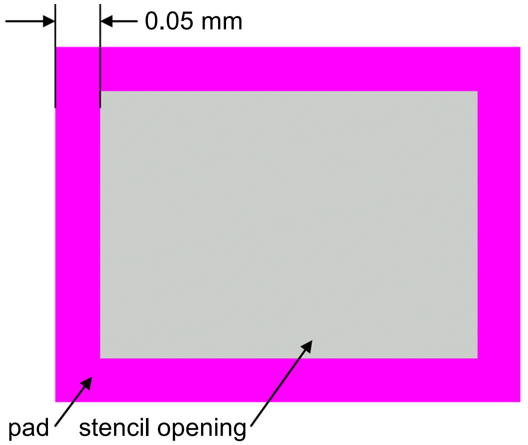

Figure 11 Relative sizes of pad and opening – LGA devices

Solder pastes

For lead-free devices, a tin-silver-copper solder paste must be used (Sn96.5Ag3Cu0.5, Sn95.5Ag4Cu0.5 or similar). For standard devices, a eutectic tin-lead solder paste (Sn63Pb37or similar) must be used.

An automatic or manual stencil/screen printer can be used to distribute the solder paste onto the PCB pads.

Device placement

A placement accuracy of +/- 0.1mm is required.

Reflow

Table 1 and Figure 12 show typical reflow profiles and conditions for lead-free and eutectic tin-lead solders. Follow the solder paste manufacturer's guidelines and place the profile probe on the PCB close to the device. Figure 13 shows example reflow profiles for lead-free (Sn96.5Ag3Cu0.5) and tin-lead (Sn63Pb37) solders.

Table 1 Solder reflow profile conditions

Profile feature		Lead-free solder*	Eutectic tin-lead solder*
Pre-heat	Minimum temp (ts_{min})	150 °C	100 °C
	Maximum temp (ts_{max})	200 °C	150 °C
	Time (ts_{min} to ts_{max})	60–80s	60–120s
Reflow	Liquidus temp (T_L)	217 °C	183 °C
	Time (t_L)	60–150s	60–150s
Peak temperature (T_P)		260 °C	225 °C
Time to reach within 5 °C of T_P		20–30s	10–30s
Average ramp-up rate (ts_{max} to T_P)		3 °C/s	3 °C/s
Ramp down rate		6 °C/s	6 °C/s
Time from 25 °C to T_P		8 minutes	6 minutes
Do not exceed		260 °C	225 °C

* see Figure 12

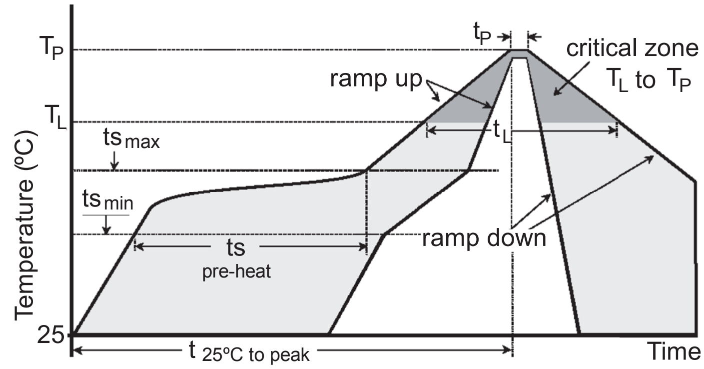

Figure 12 Typical reflow profile

For IR lead-free devices, peak reflow temperature should not exceed 260°C. For IR standard devices, peak reflow temperature should not exceed 225°C.

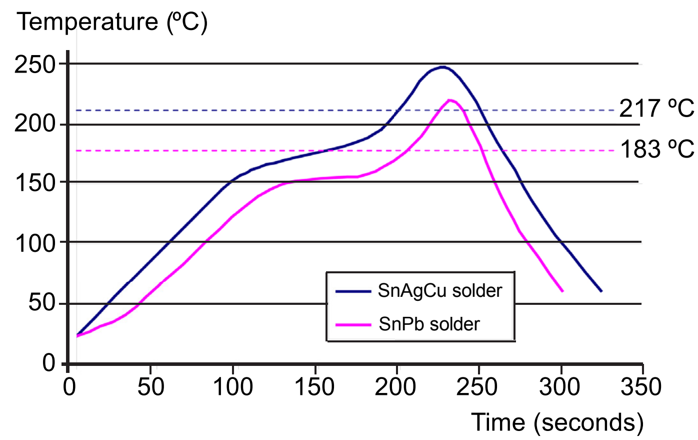

Figure 13 Typical solder reflow profile

Inspection

As with any bottom-terminated package, the best way to inspect BGA and LGA devices after reflow is through a combination of visual inspection of the peripheral solder joints (if applicable) and X-ray imaging of the connections directly under the package.

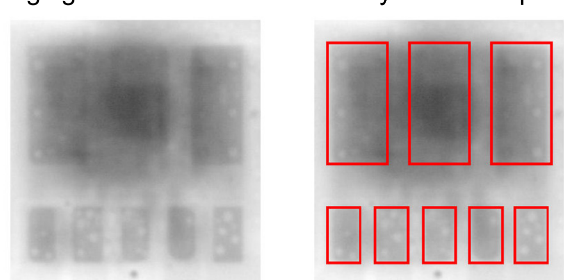

Figure 14 X-rays of LGA

Figure 14 is a typical X-ray image of a board-mounted LGA device, which shows the solder joints, device alignment and solder voiding level. Regarding solder joint voiding, most customers use 25–30% as the acceptable limit, often citing industry standards such as IPC-A-610 or IPC-7093. However, having tested board-mounted devices deliberately voided up to 45%, International Rectifier has been unable to detect any deterioration in electrical or thermal performance in application compared with devices voided to 5–10%.

Post-reflow cleaning

When using no-clean solder pastes, post reflow cleaning is not necessary. However, if flux cleaning is required, acceptable methods include ultrasonic systems, alcohol-based solvents and properly controlled water-based systems. Most solvents suitable for use with other components can safely be used with BGA and LGA devices. Surfactants can be added to improve water penetration and flow. An adequate drying profile must be used to ensure that no water is trapped under devices after cleaning. Follow all recommendations from the suppliers of both solvent and solder paste.

Rework guidelines

Modern rework stations for BGA and LGA devices often use two heating stages. The first heats the substrate, either with a conventional hot-plate or a hot-air system. The second stage uses a hot-air system for localized heating, often with the option of unheated air for faster cooling of the solder interconnections on the replaced device; this improves the solder grain structure.

The device placement mechanism or arm usually has a hot-air de-soldering gun as part of the pick head, equipped with a vacuum cup and thermocouple. Once the solder reflow temperature has been reached, the vacuum is automatically engaged to allow the device to be removed from the substrate. This reduces the risk of causing damage by premature removal.

International Rectifier does not recommend reusing devices removed from a substrate. Dispose of the old device and use a new replacement.

To replace a BGA or LGA device:

Note: If you usually bake to remove residual moisture before rework, insert your normal procedure here.

1. Heat the site to approximately 100°C (150°C for lead-free assembly) using the substrate heating stage.

Note: Pb devices are qualified for a maximum reflow peak temperature of 225°C (260°C for PbF devices). To avoid overheating the device or substrate, adjust the settings on your equipment to achieve a maximum air temperature of 300°C.

2. Lower the placement arm to bring the de-soldering tool into contact with the device. When the device and the solder interconnects reach reflow temperature, lift the placement arm to remove the device from the substrate. Discard the device.
3. Clear residual solder from the site using a blade-type de-soldering tool and de-soldering braid. Clear residual flux using a flux-reducing agent. Take care in cleaning the site: damage to the solder-resist may produce undesirable results.
4. When the site is ready, apply new solder paste with a micro-stencil and squeegee.
5. Position a new device on the vacuum tip of the placement head and lower the placement arm until the device is in contact with the solder paste.
6. Switch off the vacuum on the placement head and retract the placement arm, leaving the device in place.
7. Heat the site to approximately 100°C (150°C for lead-free assembly) using the substrate heating stage.
8. Use the de-soldering tool to heat both device and solder interconnects to reflow temperature, waiting until all the solder has reflowed.
9. Retract the arm, leaving the device in place. Cool as quickly as possible.

Standards

Joint Industry Standard, Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices, IPC/JEDEC J-STD-033A, July 2002.