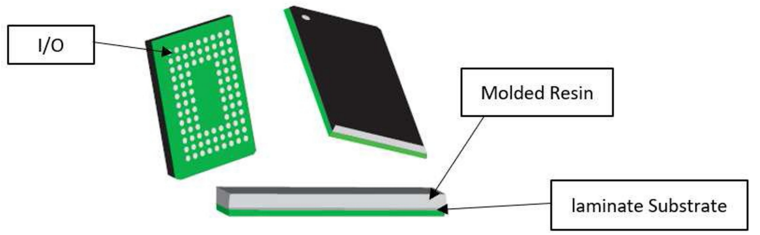
Guidelines for design and board assembly of land grid array packages

Introduction

This application note provides guidelines for the handling and assembly of LGA packages on a printed circuit board. Land grid array(LGA) are molded array packages based on laminate interposer, which have an external pin or I/O on the bottom of the package arranged in columns and rows as shown in Figure 1.

This document presents the general guidelines and suggestions for robust and reliable board assembly of the LGA packages, especially proper board and stencil design, assembly and rework. To develop a specific solution, actual experience and development efforts are required to optimize the assembly process and match the individual device requirements.

Figure 1. LGA package type



1 Packages

LGA is like a BGA but without the solder sphere or like a QFN package but with a laminate interposer. It is a resin molding package with a laminate substrate with Nickel Gold (NiAu) plated terminations and some LGAs have capping layers. Some typical LGA cross-section views are as shown below in Figure 2, Figure 3, Figure 4 and Figure 5.

The external pins or terminations have a land structure and are arranged on the bottom surface and edges of the package. The land pattern could be either in the shape of a circle, square or rectangular. Some packages have large plane or exposure pads for thermal dissipation. It can be mounted on boards just like any other SMD package.

The interconnection is formed only by the solder paste, which results in a lower stand-off suitable for a product that requires a more compact design.

Figure 2. LGA with over mold

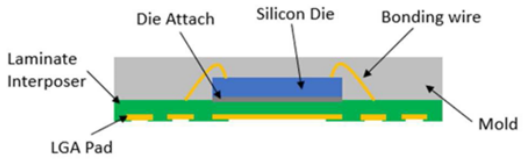


Figure 3. Time-of-Flight LGA

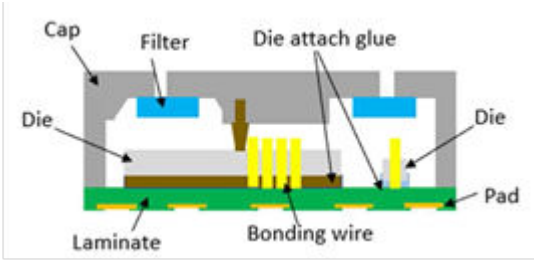


Figure 4. MEMS LGA

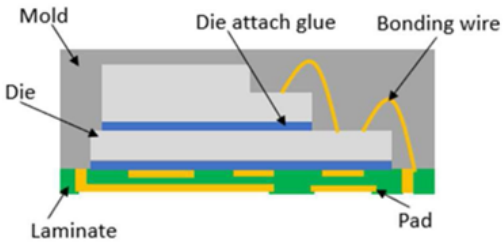
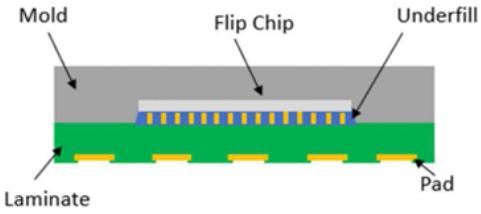


Figure 5. FCCSP LGA



1.1 Package configuration and dimensions

Some typical LGA sizes, example from 2x2 to 3.5x3 mm, with a pitch of 0.35 mm and 0.65 mm, as shown in the table below.

Table 1. LGA packages typical dimensions

LGABody size (mm)	Package family	IO's	Pitch (mm)	Max thickness (mm)	Substrate land pad size (mm x mm)	Land pad design (SMD, NSMD, SSMD)	E-pad size (mm)	Substrate finishing
2 x 2	WF	14	0.35	0.70	0.275 x0.200	SSMD	None	NiAu
2 x 2	WF	12	0.50	0.70	0.275 x0.250	SSMD	None	NiAu
2.3 x 2.3	WF	16	0.40	0.70	0.300 x0.200	SSMD	None	NiAu
2 x 2	VF	14	0.35	0.96	0.275 x0.200	SSMD	None	NiAu
2 x 2	VF	10	0.40	1.00	0.300 x0.200	SSMD	None	NiAu
2 x 2	VF	12	0.50	1.00	0.275 x0.250	SSMD	None	NiAu
2.5 x 2.5	VF	14	0.50	0.86	0.300 x0.250	SSMD	None	NiAu
3 x 2.5	VF	14	0.50	0.86	0.450 x0.250	SSMD	None	NiAu

LGABody size (mm)	Package family	IO's	Pitch (mm)	Max thickness (mm)	Substrate land pad size (mm x mm)	Land pad design (SMD, NSMD, SSMD)	E-pad size (mm)	Substrate finishing
3 x 3	VF	16	0.50	1.00	0.350 x0.250	SSMD	None	NiAu
3.5 x 3	TF	24	0.43	1.03	0.350 x0.230	SSMD	None	NiAu
4 x 4	L	16	0.65	1.10	0.400 x0.300	SSMD	None	NiAu
5 x 5	VF	45	0.50	0.70	0.350 X0.350	SMD	None	NiAu
8 x 8	VF	56	0.45	0.94	0.400 x0.200	SSMD	4x 4	NiAu
7.3 x 11	F	86	0.90	1.43	0.35 x 0.30, 0.35 x 0.35, 0.90 x 0.90	SMD	24 small pads	NiAu

1.2 Benefit of LGA

1. LGA devices can be used for either leaded or lead-free assemblies depending on the SMT assembly solder paste used.
2. Eliminates the risk of missing or damaged spheres due to shipping or handling.
3. LGA devices have a lower mounted height than BGA. This can permit small form-factor applications.
4. LGA is more robust than BGA in case of mechanical drop.
5. LGA can use the same recommended board assembly process as a BGA.

1.3 LGA packages pads design types

There are typically two types of pad designs for the LGA package as shown below in Figure 6 and Figure 7. The SMD pads have a solder mask opening smaller than the Cu pad while a SSMD has a solder mask covering only the three sides of the Cu pad. The SSMD pad mostly is on the perimeter of the package and this design helps to avoid solder voids during reflow.

Figure 6. Example of package with SMD pad

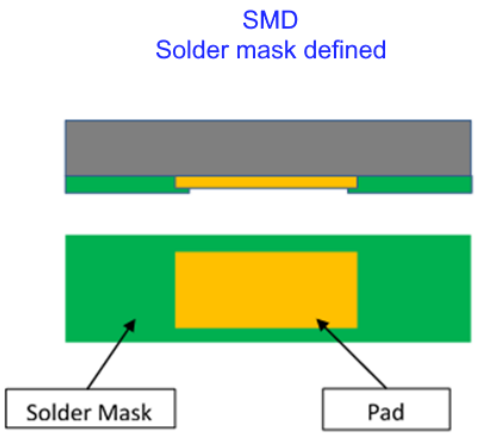
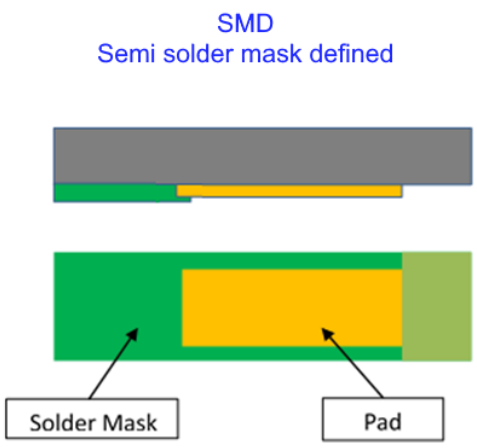


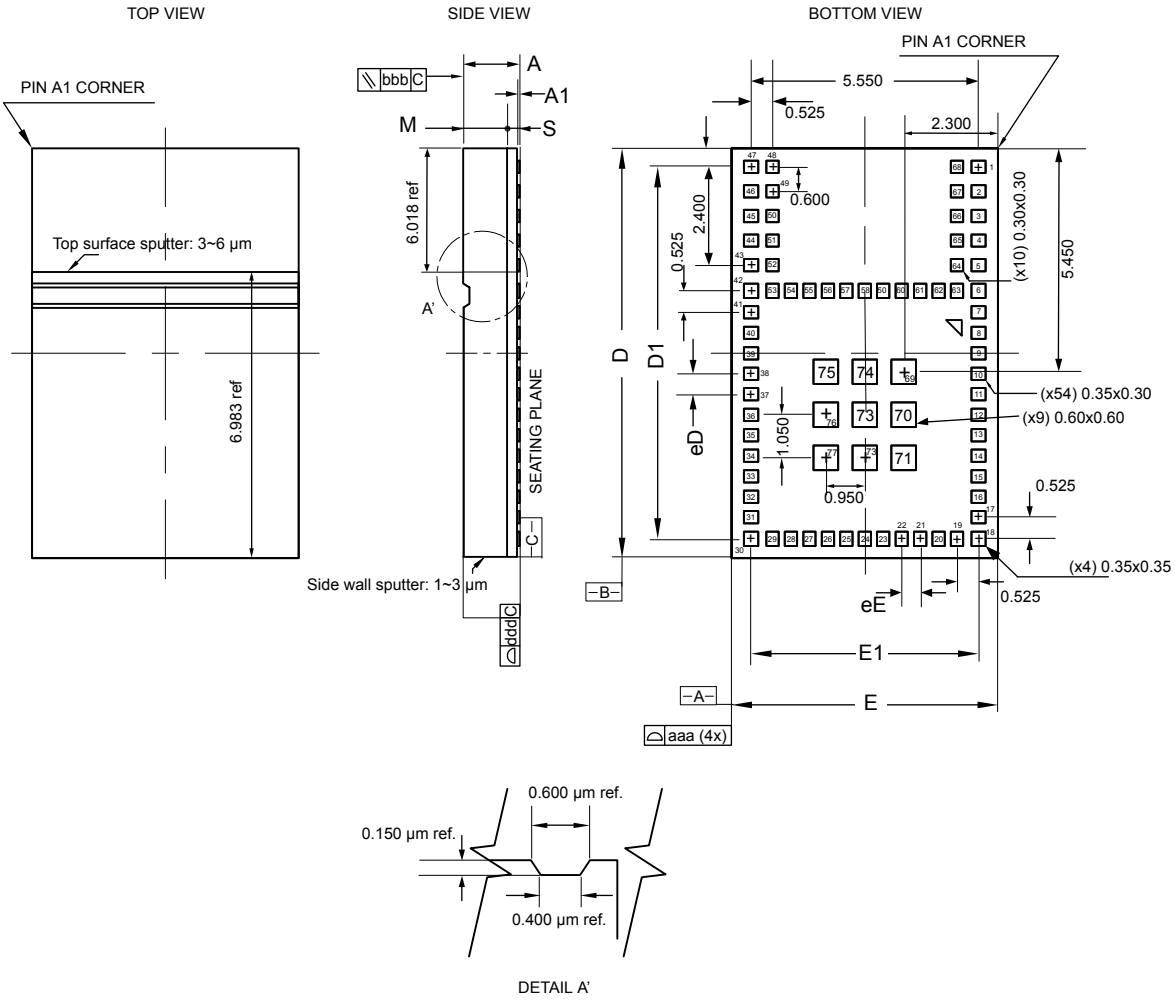
Figure 7. Example of package with SSMD pad



2 Board design

PCB footprint sizes and shapes and stencil designs are critical to surface mount assembly yields and subsequent electrical and mechanical performance of the mounted package. The package design can be obtained from the STMicroelectronics package outline assembly drawing as illustrated in Figure 8.

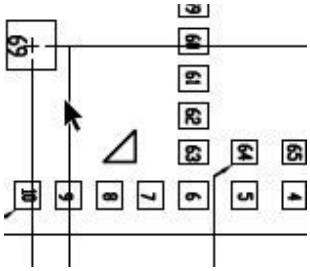
Figure 8. Example of LGA 77-lead, 10 x 6.5 mm package drawing



Note: For LGA packages in SMT the stencil opening is 1:1

The package outline assembly drawing can include a small triangle that may or may not be visible on the chip. This is used for manufacturing and is not relevant for assembly of the device.

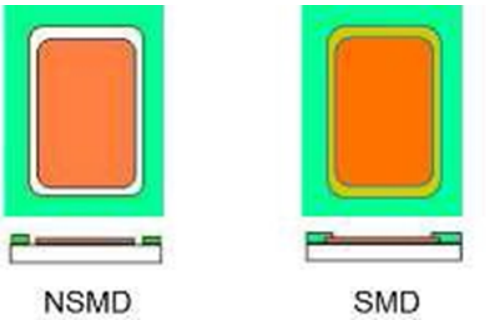
Figure 9. Example of triangle in POA drawing



2.1 Land pattern design

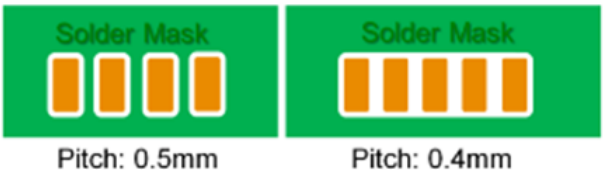
It is recommended to use a non solder mask defined (NSMD) pads, see Figure 10, with opening slightly larger than the land geometry, as copper etching has better dimensional control than solder masking process, because it allow solder to adhere to the sides of the Cu land for better reliability.

Figure 10. Comparison of NSMD and SMD leads



The solder mask opening should be minimum 0.050 mm, for the registration tolerance of the solder mask. For very fine pitch of 0.4 mm and below, solder mask defined is recommended and openings as “trench” should be considered asthere is not enough space for solder mask web in between pads, see Figure 11, Figure 12 and Figure 13.

Figure 11. Solder mask opening for perimeter lands with pitch 0.5 mm and 0.4 mm



In general, 1 to 1 ratio between the package pad and the PCB signal pad is used.

Figure 12. **Example of joint with NSMD pad**

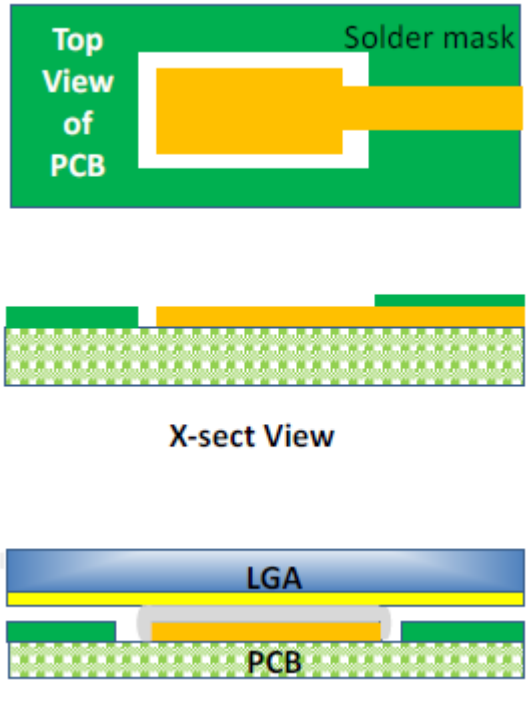
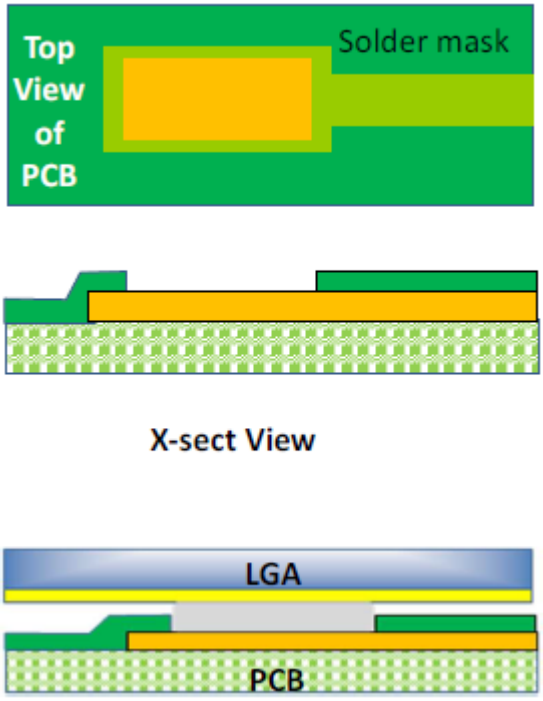


Figure 13. **Example of joint with SMD pad**



2.2 PCB pads design: NSMD vs SMD

-	Pros	Cons
NSMD	Solder “wets” to the side of the copper pad and gives a better solder joint reliability.	Smaller pads have poorer adhesion strength, pads tend to peel off during multiple reworks and reflows.
SMD	Copper pads are stronger as the mask is covering the pads and only the soldering pad is exposed. This improves the adhesion strength between the pad and the laminate.	Less space is required between pads for routing.

2.3 Board finishing

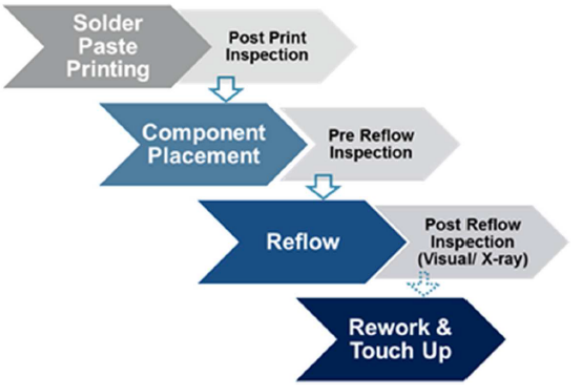
Common board finishing is organic solderability preservative (OSP), electroless nickel immersion gold (ENIG) with good flatness and acceptable land pad surface. HASL is not preferred as the poor flatness gives rise to solder paste printing issue. Immersion Ag is another lead free alternative in terms of flatness but requires special handling

3 Board assembly

Precise process development and experimentation are needed to optimize specific applications or performance.

3.1 Assembly process flow

Figure 14. Typical assembly process flow



3.2 Stencil design for perimeter pads

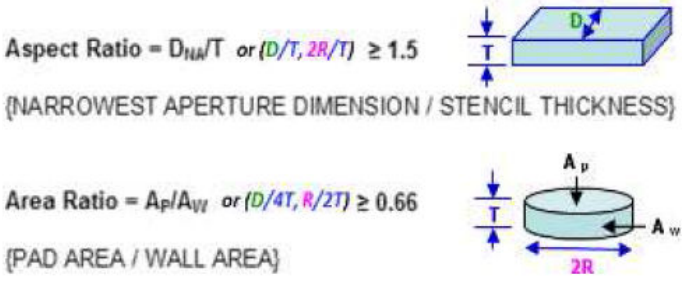
It is crucial to use a good quality stencil to achieve a robust solder paste printing process.

Nickel electroform or laser cut electro-polished stainless steel (or nickel) is recommended.

Taper aperture walls 5° tapering are recommended to facilitate paste release. Suggested stencil thickness (T) is 100 µm for 0.4 mm pitch and stencil aperture should follow the rules for aspect and area ratio.

For the best solder paste transfer, an area ratio of

> 0.66 (laser cut) and > 0.55 for electro form or an aspect ratio > 1.5 is required



One of the most important parts of SMT is to have a robust and consistent solder paste printing process. To achieve this requires process monitoring of paste volume and stencil condition.

Overprinting may cause solder bridging and inconsistent solder paste volume results in opens after reflowing due to poor coplanarity. Marginal pass solder paste volume causes reliability issue in the field as the solder paste is the only connection between LGA pad and PCB pad.

The stencil aperture is typically designed to match the PCB perimeter pad size (i.e., 1:1) depending on area ratio for proper disposition of solder paste. As for smaller pitch < 0.40 mm for a circle pad, A square aperture is recommended for round pads and a pitch smaller than 0.40 mm. This produces a slightly larger volume of solder paste for easy release

3.3 Solder paste printing

Typical lead-free solder paste is SAC alloy. Solder paste type 4 (or type 3) is recommended to optimize the deposition efficiency as per below table.

Table 2. Solder paste powder size

Parameter		Type	
Solder paste type		3	**4**
Powder size (µm)		45-25	36-25
Pitch (mm)	0.5 to 1.0	√	√
	0.35 and 0.4	-	√

There are two different types of soldering flux for solder paste. Water soluble and no clean flux. No clean solder paste with very little residue after soldering is recommended. The left-over residue is not corrosive and des not damages the PCB.

Precaution must be taken for water soluble flux as it tends to corrode after a prolonged time if it is not well cleaned (esp. very small stand-off).

3.4 Solder paste inspection

Inspection of the printed solder paste is recommended before placing parts. A 2D (X, Y) inspection or an automated 3D (X, Y, Z) volume inspection or a statistical measurement process control plan is part of the assembly process to assure a repeatable solder deposit and a high yield assembly quality.

3.5 Component placement

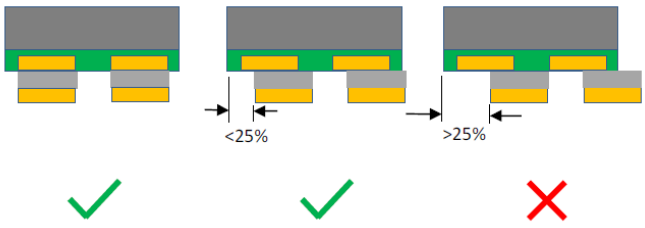
Precise and accurate component placement is required (tolerance of ±50 µm and below) to have uniform solder coverage, especially with the high density I/O configurations.

Misalignment can easily cause solder bridging and other problems like skewing especially for 0.40 mm pitch packages.

Pick and place equipment is recommended to have placement force control and force being minimized where possible:

The guideline of the mounting before pre-reflow:

Figure 15. Placement and self-alignment during reflow

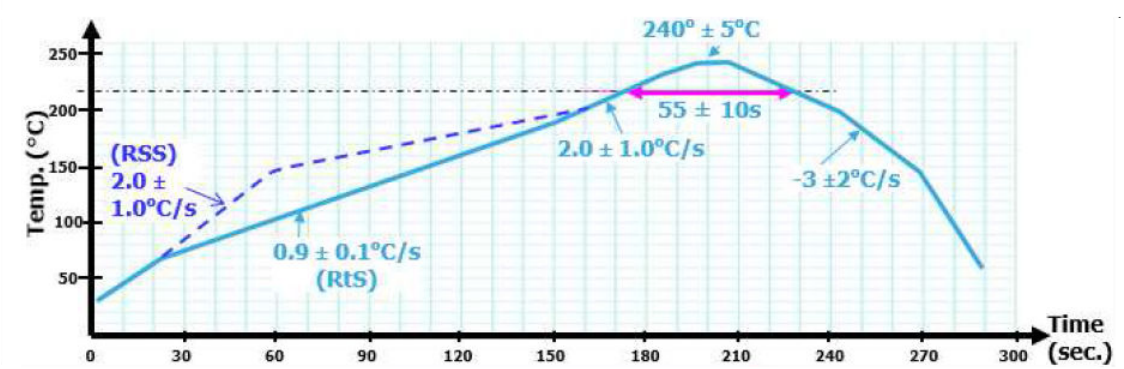


3.6 Convection reflow

There are no special requirements necessary when reflowing LGA components. As with all SMT components, it is important that profiles be checked on all new board designs. In addition, if there are multiple packages on the board, the profile must be checked at different locations on the board. The solder paste manufacturer's temperature profile is used to optimize flux activity and minimize voiding.

The recommended lead-free reflow profile as illustrated on Figure 16

Figure 16. **Typical Lead-free reflow profile for LGA**



1. *Ramp-to-spike profile is recommended for Pb-free assembly with better wetting and less thermal exposure than traditional ram-soak-spite profile.*
2. *Thermocouple is attached through bottom side of board undermeath the center of LGA.*
3. *Perfom optimization for actual board/panel and refer to solder paste supplier's recommendation*

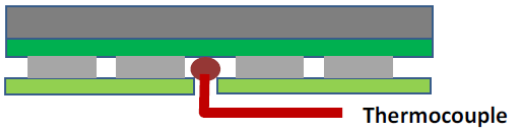
Table 3. **Typical Lead-free reflow profile for LGA**

Profile	Ramp-to-spike (straight line profile)	Ramp soak spike
Temperature gradient (in preheat)	(T = 70-150 °C): 0.9 ± 0.1 °C/s	(T = 70-150 °C): 2.0 ± 0.5 °C/s
Soak/Dwell(1)	N/A or (T = 150-200 °C): 60 ± 20 s	Soak (T = 150-200 °C): 70 ± 30 s
Temperature gradient (in pre-flow)	(T = 200-225 °C): 2.0 ± 1.0 °C/s	
Peak temperature (in reflow)	240 ± 5 °C	
Time above 220 °C	55 ± 10 s	
Temparuture gradient (in cooling)	-3 ± 2 °C/s	
Time from 50 to 220 °C	150 to 230 s	

1. *Refer to solder paste recommention*

For an example of thermocouple insertion on profile board for larger LGA, see Figure 17.
For a smaller package, like 2x2, the thermocouple can be attached to the side of the package.

Figure 17. **Insertion of thermocouple for large LGA**



3.7 Double-sided process

The double-sided process follows the same procedure as the single-sided process: mount and reflow the packages on one side and simply turnover the board and repeat the process.

4 Inspection, defects and analysis

After surface mount assembly, X-ray should be used for sample monitoring of the solder attachment process; this identifies defects such as solder bridging, insufficient solder and voids.

Currently there is no specific standard for solder voids after reflow. As a general guideline for solder joints it is recommended to use a 25% maximum voids criteria.

Figure 18. X-Ray inspection, acceptable joints

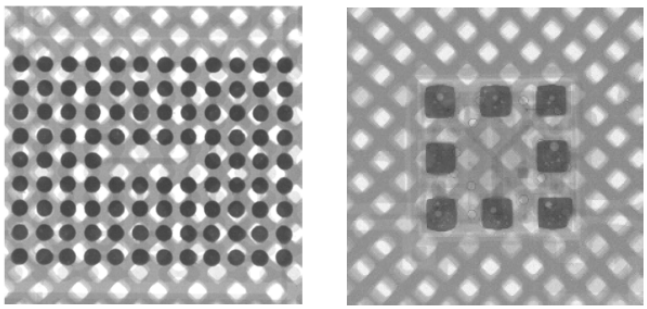
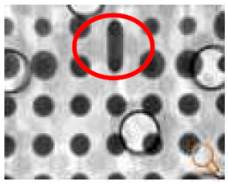
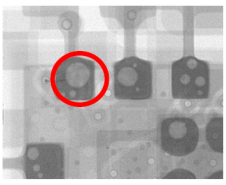


Table 4. Typical defects with causes and remedy

Type of defect	Causes	Remedy
	• Too much solder paste volume. • Stencil not properly clean after printing.	• Optimize the aperture opening. • Optimize the cleaning parameters.
Void >25% 	• Reflow profile not optimized. • Too much solvent in flux content of solder paste.	• Increase the soaking time of reflow profile to reduce the solvent before reflow.

5 Cleaning after reflow

No-clean solder paste generally doesn't require cleaning. However, depending on the application, soldering flux activity, cleaning after solder reflow may be required, such as by ultrasonic agitation or spray cleaning, which is commonly used in PCB process.

Be cautious when using ultrasonic cleaning as it could weaken the soldering joint.

6 Board level reliability

STMicroelectronics has evaluated several LGA for board mounting and solder joints reliability, especially versus thermal cycling with good performance.

See examples in Figure 19 to Figure 22 for OLGA2.7x2.3-8L_0.85 mm pitch and UFLGA 5x5-79 with 0.5 mm pitch:

Figure 19. Drop test for OLGA 2.7x2.3-8L (JESD22-B111)

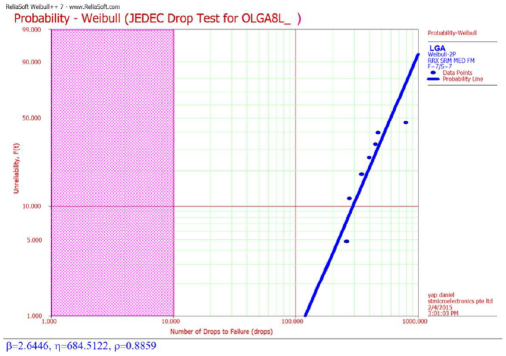


Figure 20. Failure analysis after drop test

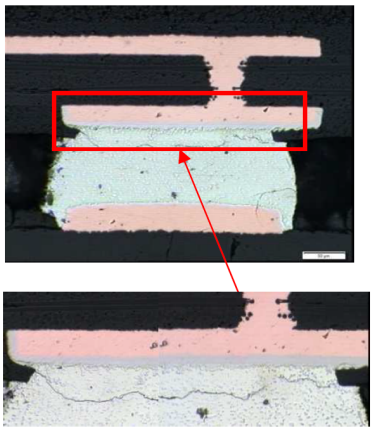


Figure 21. Thermal cycling for UFLGA5x5-79 (JESD22-A104D)

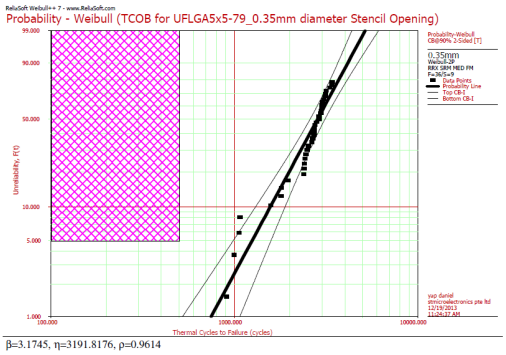
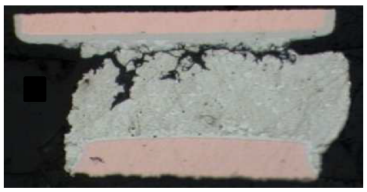


Figure 22. Failure analysis after thermal cycling



7 Rework guidelines

7.1 Rework of LGA

Reworking of LGA is carried out due to:

1. Damage
2. Misalignment after reflowing
3. Solder bridging
4. Malfunction of the package

7.2 Package removal and site dressing

1. Bake the PCB (e.g., 125C for 4 hrs) to remove the moisture (if necessary)
2. Package can be removed manually or semi auto method.
3. For manual method, apply heat on the LG package with a hot air gun and cover neighborhood component with heat shielding material to prevent damage.
4. Use a vacuum pick up nozzle or a tweezer to remove the device from the board after the solder has melted.
5. For semi-automated method, PCB is clamped on a rework equipment conveyor. Heat is applied from the equipment rework nozzle to melt the package solder.
6. After the solder has melted, a vacuum pickup nozzle is lowered to remove the package automatically.
7. Remove solder residue from the PCB pad with solder wick and solder iron before replacing with a new LGA.
8. Clean the solder pad with IPA to remove flux residue after de-soldering process.

7.3 Manual method for placing new LGA

1. Use a mini stencil to apply solder paste onto the PCB LGA pad.
2. Mini stencil must have the same thickness, aperture opening and pattern as the normal stencil.
3. Inspect the printed pad to ensure even and sufficient solder paste before component placement.
4. If the neighboring parts is too close to the LGA components and mini stencil method is not applicable, then solder paste can be applied to the package LGA pad instead.
5. The solder paste is applied to the LGA pad using rework jig and mini stencil.
6. The LGA package is then subjected to reflow to form the solder joint.
7. Apply a thin layer of no clean flux on the PCB LGA pad.
8. Mount LGA package on PCB by "Pick & Place" machine.
9. Use hot air nozzle to melt the solder to form the joint between the package and PCB pad.
10. Proceed X-ray to inspect solder joint quality.

7.4 Semi-automated method

1. This is a better method for reworking LGA.
2. There are many rework equipments in the markets which are capable to perform the rework. See example in Figure 19.
3. Execute Section 7.3: Manual method for placing new LGA step 1 to 3
4. Then the PCB is clamped on the rework equipment conveyor/table.
5. New LGA is picked up by a nozzle and the LGA soldering pad is aligned to the PCB pad.
6. Heat is applied to the PCB until the solder melting point is reached. The LGA is then slowly lowered to perform the soldering.
7. If the neighboring parts is too close to the LGA components and mini stencil method is not applicable, solder paste can be applied to the LGA pad instead. Repeat Section 7.3: Manual method for placing new LGA step 5 to 7.
8. LGA package with solder is then picked up by a nozzle and aligned to soldering pad on the PCB

9. Use hot air nozzle with controlled temperature to melt the solder . Bottom board side heating (150 °C) is often needed to avoid warpage and overheating of component.
   Heat is applied to the PCB until the solder melting point is reached. LGA is then slowly lowered to perform the soldering.
10. The quality of the solder joint is then inspected by X-ray machine.

Figure 23. Example of rework station

8 LGA handling

8.1 Handling ESD package

Integrated circuits and components are ESD sensitive, so proper precautions are required for handling and mounting processes as electrostatic discharge could cause damage and failure to the devices. Ensure to have proper ESD control during the handling and mounting process using industry standard requirements such as JEDS625-B guideline, for proper handling and ESD controls.

8.2 Handling moisture sensitive package

Most LGA packages are classified level 3 as per JEDEC J-STD 020E standard. The humidity is easily trapped by LGA packages in laminate (substrate) mold compound glue and underfill materials inducing vapor pressure during reflow. The moisture pressure inside non-hermetic packages may cause internal materials /delamination or crack or bond damage.

In the most severe case, the stress can extend to external package cracks, commonly known as “popcorn” that bulge and then crack. Humidity inside the package can induce some package deformations during the reflow which contribute to stress on the solder joints and could lead to quality and reliabilities risk.

Recommendation:

Implement tight control to minimize the moisture absorption level after opening the dry-pack, observe and follow the moisture sensitively guidelines on package dry-pack label and in JEDEC standard J-STD-033.

E.g., use nitrogen or dry/desiccator cabinets to store packages before boards soldering.

Parts can be re-baked after few days in open air in order not to exceed the moisture sensitivity guidelines.

Baking is required before assembly if the following condition occurs:-

1. Package is exposed to floor life more than specific requirements
2. Humidity indicator card show more than the specific percent level
3. Packages are not stored according to J-STD-033B standard

9 Reference

Table 5. **Reference list**

Reference	Description
JEDEC J-STD 020E	“Moisture/Reflow sensitivity classification for non-hermetic surface mount devices”, Jan 2015
JEDEC J-STD-033D	“Handling, packing, shippingand use of moisture/reflow sensitive surface mount devices”, Apr 2018
JEDEC JESD 625-B	“Requirementsf or handling electrostatic discharge sensitive (ESDS)- devices”, Dec 2011
IPC7093	“Design and assembly process implementation for bottom termination components”, March 2011
IPC-A-610J	“Requirements for soldered electrical and electronic assemblies”, May2016
IPC7351B	“Generic requirements for surface mount design and land pattern standard”, June 2010
JEDEC JESD22A104F	"JEDEC standard thermal cycling", Nov 2020
JEDEC JESD 22B111	"JEDEC standard board level drop test", July 2003
IPC-7525	“Stencil design guideline”, February 2007

10 Acronym

Table 6. **Acronym list**

symbol	Description
BGA	Ball grid array
Cu	Copper
ESD	Electrostatic discharge
I/O	Input and Output
LGA	Land grid array
MSL	Moisture sensitive level
NSMD	Non solder mask define
PCB	Printed circuit board
PCBA	Printed circuit board assembly
SMD	Solder mask define
TFLGA	Thin profile fine pitch LGA
VFLGA	Verythin profile fine pitch LGA
WFLGA	Veryvery thin profile fine pitch LGA

Revision history

Table 7. **Document revision history**

Date	Version	Changes
01-Feb-2023	1	Initial release.
09-May-2024	2	Updated: • Table 1. LGA packages typical dimensions • Section 2: Board design

Contents

List of tables

List of figures