Freescale Semiconductor
Application Note

Document Number: AN3241
Rev. 1.0, 10/2009

Land Grid Array (LGA) Package Rework

1 Introduction

This application note describes rework considerations for the Land Grid Array (LGA) style package.

Freescale has introduced radio frequency (RF) modules such as the MC1320x and MC1321x in LGA packages as an alternative package to ball grid array (BGA).

The LGA packages reduce the amounts of lead in finished products and are Reduction of Hazardous Substances (RoHS) compliant, optimized for improved radio-frequency (RF) performance for wireless applications, and/or reduce the overall height of the package by eliminating the stand-off height associated with BGA balls.

For assistance with any questions about the information contained in this note or for more details about the MC1320x and MC1321x devices, visit www.freescale.com/802154. or contact the appropriate product applications team.

Contents

1.1 Acronyms and Abbreviations

BGA	Ball Grid Array
BT	Bismaleimide Triazine
CBGA	Ceramic Ball Grid Array
CTE	Coefficient of Thermal Expansion
EU	European Union
ESD	Electrostatic Discharge
HCTE	High Coefficient of Thermal Expansion
HDI	High Density Interconnect
LGA	Land Grid Array
LTCC	Low Temperature Co-fired Ceramic
MSL*n*	Moisture Sensitivity Level *n*
NSMD	Non-Solder Mask Defined
OSP	Organic Solderability Protectant
PCB	Printed Circuit Board
RF	Radio Frequency
RoHS	Reduction of Hazardous Substances
SMD	Solder Mask Defined
SMT	Surface Mount Technology

2 What is LGA?

The LGA package makes the second level interconnect (from package to motherboard) with a array of solderable surfaces. This may consist of a layout similar to a BGA with no solder spheres. However, it may also have an arbitrary arrangement of solderable surfaces that typically include large planes for grounding or thermal dissipation, smaller lands for signals or shielding grounds, and, in some cases, mechanical reinforcement features for mechanical durability.

Freescale has introduced the LGA package using a high coefficient of thermal expansion (HCTE) ceramic in larger body sizes. Figure 1 shows the top and bottom sides of an LGA device. HCTE LGA and HCTE BGA packages use the identical substrate, high-lead electroplate bumps, die attach procedure, including underfill material, and allow for the same recommended CBGA board assembly process (See Freescale CBGA Customer Presentation). Products from the same line have the same moisture sensitivity level (MSL) and maximum allowable peak reflow temperature regardless of whether it is LGA or BGA.

Freescale's product portfolio also includes LGA packages with organic laminate substrates. These may feature High Density Interconnect (HDI) substrates or Bismaleimide Triazine (BT) substrates. In some cases an array of joints similar to the BGA may be presented. More often, the lands are square, rectangular, or irregular, as seen in these illustrations of the 34 I/O RF Power Amplifier Module.

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.

HCTE flip-chip devices do not require spheres because the coefficient of thermal expansion (CTE) of HCTE substrates matches very closely to that of the typical PCB. The HCTE substrate is a glass-filled, low temperature co-fired ceramic (LTCC) with a CTE of 12.3 ppm/ºC[1]. Likewise, the CTE for the organic alternative substrate materials closely matches the CTE of the mother board materials, ~16 ppm/^{o}C. Typically, most epoxy-glass or polyimide-glass PCBs have a CTE of 16–22 ppm/ºC.

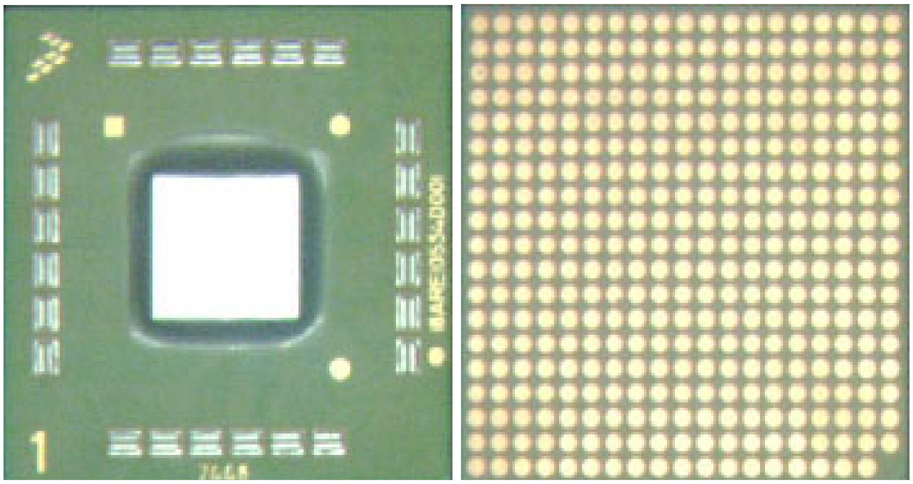


Figure 1. Top and Bottom View of HCTE 360 Pad LGA Device

---

[1] The unit ppm/ºC stands for parts per million per degree Centigrade. Using HCTE as an example, if the temperature of one million millimeters of material is increased 1ºC, that material would expand 12.3 mm.

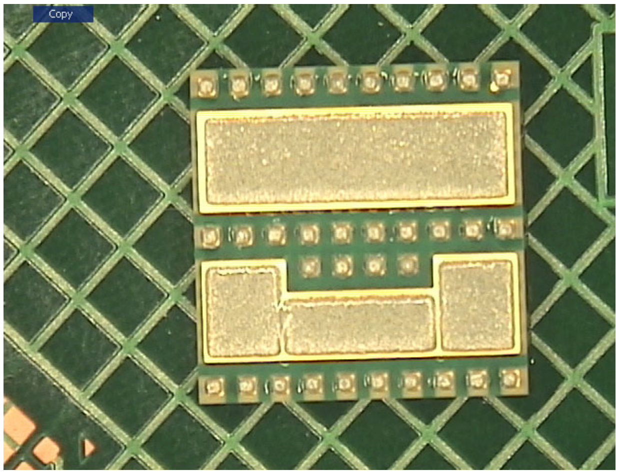


Figure 2. Bottom View of HDI 34 Pad RF Power Amplifier Module

The LGA pad uses the same 0.1 m to 0.9 m of electroless gold plating over electroless nickel as has been used reliably for many years in the traditional BGA configuration. LGA's which use a flip chip first level interconnect (from die to package) typically have a 0.15 m maximum gold thickness. LGA's that have wirebond first level interconnect typically have a 0.5 m to 0.9 m gold thickness. Figure 3 shows an image of a typical LGA pad.

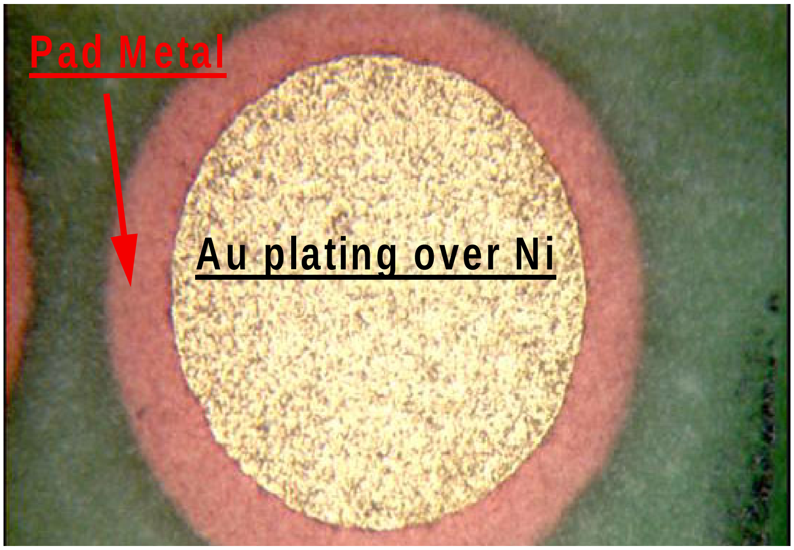


Figure 3. Typical Solder Mask Defined (SMD) LGA Pad

The only RoHS restricted material in Freescale flip-chip HCTE LGA products is lead. These LGA products contain RoHS compliant high-lead bumps between the flip-chip die and ceramic substrate as permitted by the RoHS Directive exemption #10, which reads "Lead in high melting temperature type solders (that is, tin-lead solder alloys containing more than 85% lead) and any lower temperature solder required to be used with high melting temperature solder to complete a viable electrical connection." A modified proposed exemption #10 has been submitted to the European Union (EU) to permit "Lead in solders to complete a viable electrical connection between semiconductor die and carrier within integrated circuit flip chip packages." Freescale LGA devices can ship under either version of exemption #10.

Freescale wirebonded LGA products have no lead in them. Lead-free solders and die attach materials are used to attach the integrated circuit device and any discrete passive components within the package to the substrate.

2.1 Benefits of LGA

Some benefits of the LGA package over a BGA package include:

- LGA devices can be used for either lead containing or lead-free assemblies depending on the surface mount technology (SMT) assembly solder pasted used.
- LGA eliminates risk that customers receive components with missing or damaged spheres due to shipping or handling.
- LGA devices have a lower mounted height than BGA. This can allow for more space above the device for a heat sink solution or for small form-factor applications.
- Board-level reliability significantly exceeds customer requirements when the design and process recommendations are followed.
- The durability of LGA in mechanical drop is typically greater than a BGA that is not underfilled.
- LGA can use the same recommended board assembly process as CBGA.

3 LGA Rework

3.1 Solder Methods

See Freescale Applications Note AN2920 for direction on how to perform assembly of Ceramic substrate based LGA.

3.2 ESD Protection

Proper ESD protection must be taken by the operator when handling electronic devices. Appropriate wrist and foot straps must be worn by the operator when handling these packages and reworking PCB boards.

3.3 Other References

Rework is LGA is very similar to the rework of Quad Flat No-lead (QFN) packages, covered in Freescale Applications Note AN1902.

3.4 PCB Preparation

During the package removal process the PCB will be heated. To the extent that moisture will be catastrophically driven from other components on the PCB and within the PCB there is a risk of product damage. In order to prevent such “popcorn” failures of components due to adsorbed moisture it is recommended that the PCBs should have had strict control for storage in Nitrogen Cabinets or a prebake (e.g. 125^{o} C for 16 hour for boards with SMT components or 95^{o} C for 16 hours for boards with temperature sensitive components) to remove the moisture from the PCB prior to removal of the LGA.

4 Package Removal

In general, a rework station should have a split vision system, an XY table for alignment and a hot air reflow system with top and bottom heaters for component removal.

To fully remove the faulty component from the board, hot air is applied from the top and bottom heaters. It is important that the whole board is at least somewhat elevated in temperature to minimize warping due to CTE mismatch between the area of the LGA which must be heated to a temperature above the liquidity of the solder and the balance of the board. A target temperature for the entire board is approximately 125^{o} C.

An air nozzle of the correct size should be use to conduct the hot air to the LGA component such that the vacuum pick tool can properly remove the component. The temperatures for the heaters should be set to achieve the targeted board temperature of 125^{o} C and then increase the spot temperature at the component being reworked above the solder liquidity and below 245^{o} C so copper pealing does not occur. When the solder is molten, use a vacuum pick tool to remove the component. Figure 4 shows a typical set-up.

Load PCB on XY Table Select Appropriate Nozzle Removal

Figure 4. Typical Package Rework Station and Process

Alternatively, a shell-type tool may apply heat directly to the LGA package. Top and bottom pre-heat is still required. The process window for a shell-type tool is very sensitive due to the fast heat up and potential for tilt if one side of the package melts before the other side. A typical set up in this configuration is shown in Figure 5 and Figure 6.

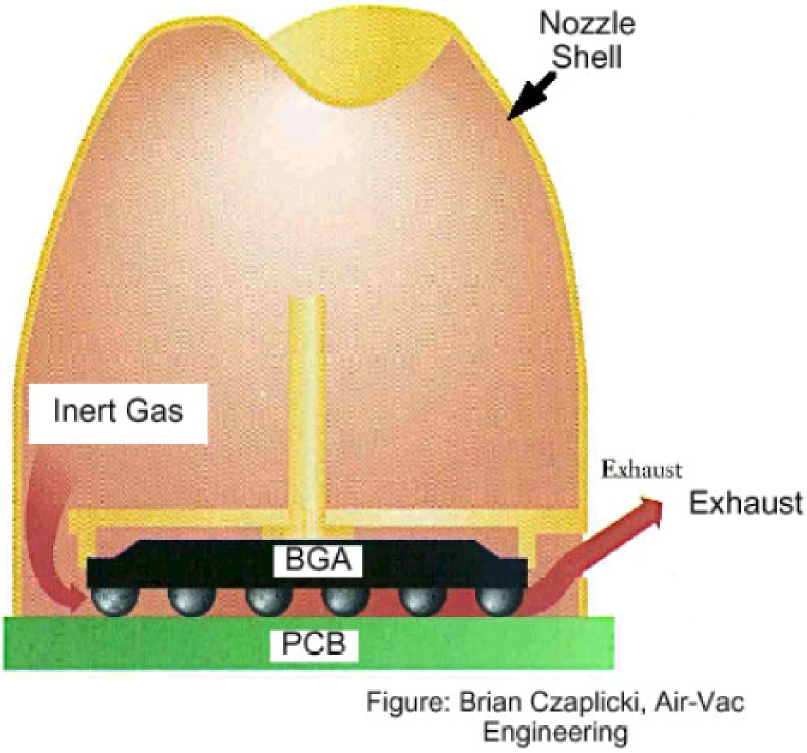


Figure 5. Patented Air-Vac Engineering Heat Nozzle

Many assembly sites have extensive in-house knowledge on rework and their experts should be consulted for further guidance on how to remove the package.

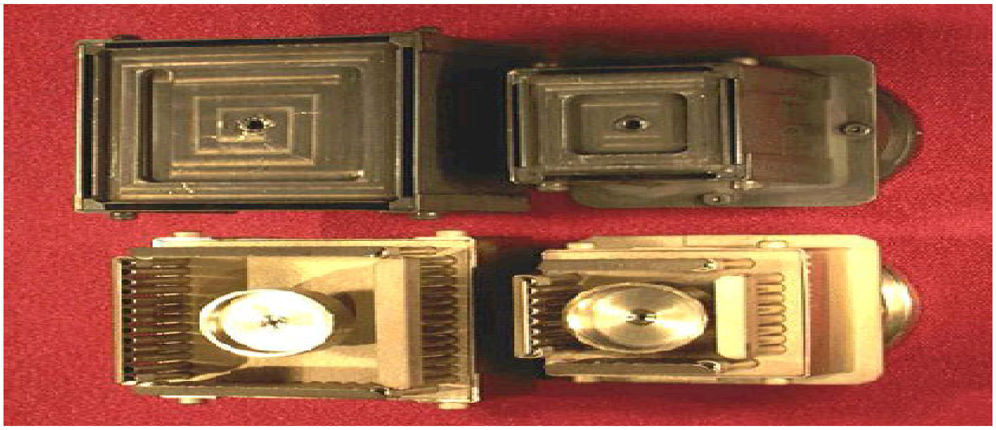


Figure 6. Shell Type Preheat Tool/Vacuum Pick-up Tool

4.1 Package Disposal

Freescale follows standard component level qualification standards for packages and these include three solder reflows survivability. A package that has been attached to PCB and then removed has seen two solder reflows and if the PCB is double sided, then the package has seen three solder reflows. Thus the package is at or near the end of the tested and qualified range of known survivability. The removed LGA package should be properly disposed of so that they will not mix in with new LGA components.

4.2 Site Preparation

Once the LGA component is removed, the site should be cleaned and dressed to prepare for the new component placement. A de-soldering station can be used for solder dressing. However, using a vacuum tool to remove excess solder while the PCB is still hot from the part removal eliminates a further temperature cycle on the board. A solder wicking braid may also be used to remove excess solder. This is typically a manual operation that puts a premium on operator skill and experience. The applied temperature should not exceed 245^o C. Otherwise, the copper pad on the PCB may peel off.

4.3 Solder Paste Printing

Unless otherwise indicated, Freescale studies discussed in this document use Indium no clean NC-SMQ® 230 flux and Indalloy® 241 solder paste made up of 95.5Sn/3.8Ag/0.7Cu. Devices were soldered to boards using the reflow profile shown in Section 4.6, “Reflow Profile”.

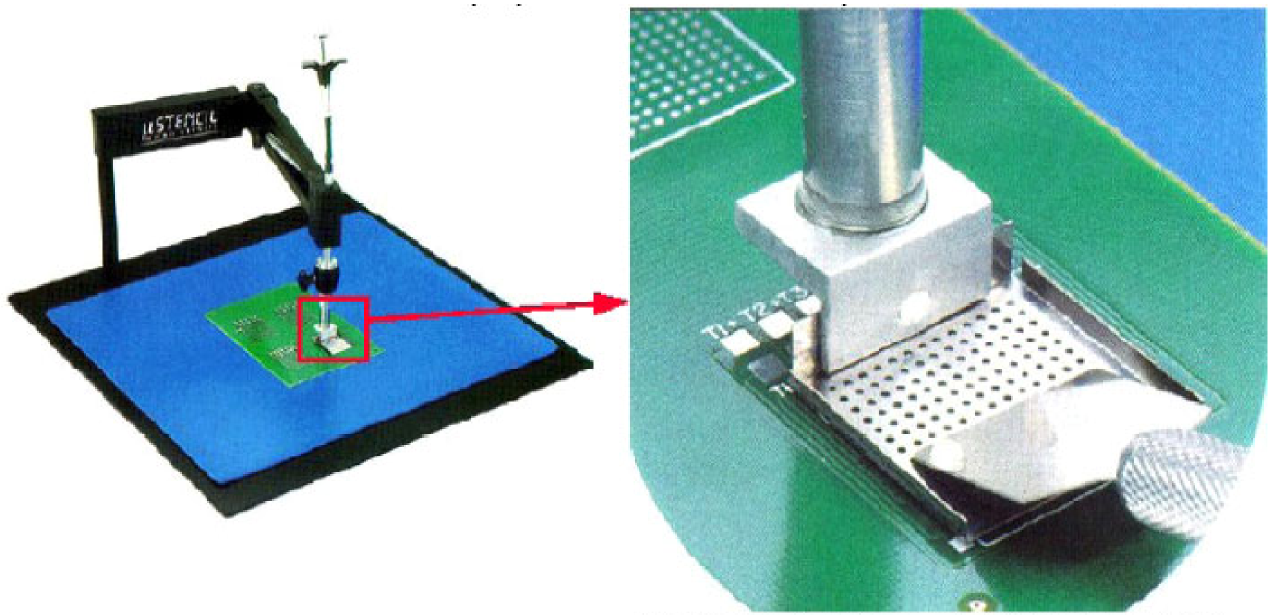
Flux should be applied uniformly but sparingly to the part for pre-tinned parts. Alternative fluxes to the NC-SMQ® 230 flux should be compatible with the production cleaning strategy. Flux is not used in systems that involve direct application of solder paste to the PCB during rework.

4.3.1 Manual Dispense (Not Recommended)

The operator may use a needle to dispense paste directly to the solderable areas on the PCB if there is insufficient paste remaining on the board. This is ***not*** recommended.

4.3.2 Mini-Stencil

The user may fabricate a mini-stencil with the same stencil thickness, aperture opening and pattern as the normal production stencil that was used to originally place the component. The mini-stencil is placed on the position where the package will be placed and aligned to match the interconnect areas on the PCB. The operator uses a mini-squeegee blade to deposit solder paste on the mini-stencil and spread the paste into the mini-stencil openings. The printed pads should be inspected to ensure even and sufficient solder paste before component placement. See Figure 7.

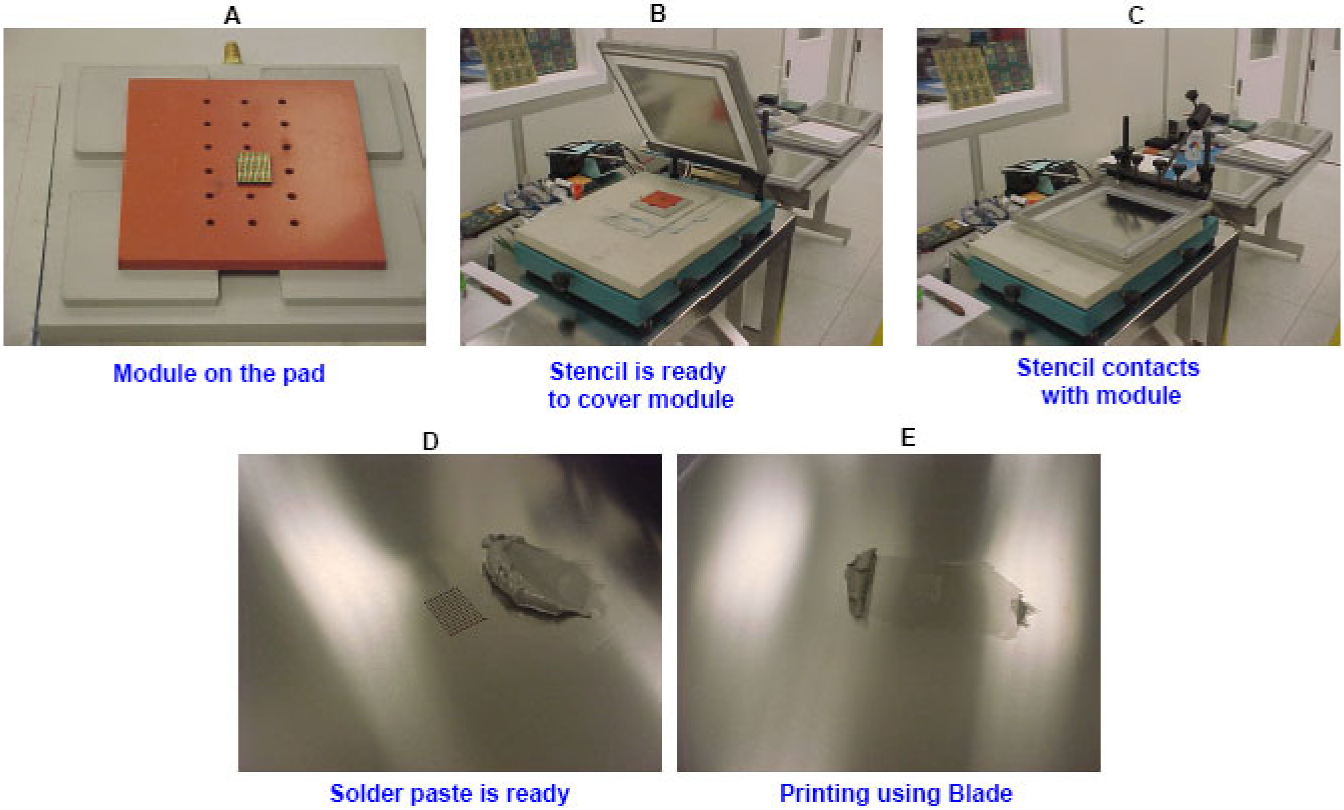


(Photos Courtesy of OK Industries)

Figure 7. Mini-stencil and Mini-squeegee

4.3.3 Locally Pre-tinned Components

The user may prepare a stencil for off-line tinning. A unit is placed in a fixture and a stencil which matches the production stencil in thickness, aperture openings and patterns is used to deposit solder paste on the component. Depending on the volume of pre-tinned units desired, this fixture/stencil operation may be a one-up operation or involve a fixture with multiple sites so multiple units may have paste deposited at one time. Note that some adjustments to the aperture openings for large heat sink lands may be necessary so the post-reflow height of the solder paste is similar between the heat sink and the signal lands. It is Freescale's experience that exactly matching the conventional assembly paste print patterns on the large lands will lead to too little solder on the pre-tinned heat sink areas. The parts with the solder paste are then put through a solder reflow pass (See Section 4.5, "Solder Reflow Profile for Lead-Free Paste"), cleaned, and set aside to use exclusively for rework.



Module on the pad

Stencil is ready to cover module

Stencil contacts with module

Solder paste is ready

Printing using Blade

Figure 8. Pre-tinning an LGA

Alternatively, the customer may obtain a clean ceramic plate. Stencil print the production solder paste pattern on the plate and place a component on the solder paste pattern. Reflow the assembly per the profile in Figure 12. Remove the part from the ceramic plate (since the solder will not wet to the ceramic) and clean the part. The ceramic plate may be cleaned and re-used.

4.3.4 Purchase Pre-tinned Components

The user may approach a local subcontractor or Freescale to purchase parts that have been pre-tinned.

4.4 Place the Component

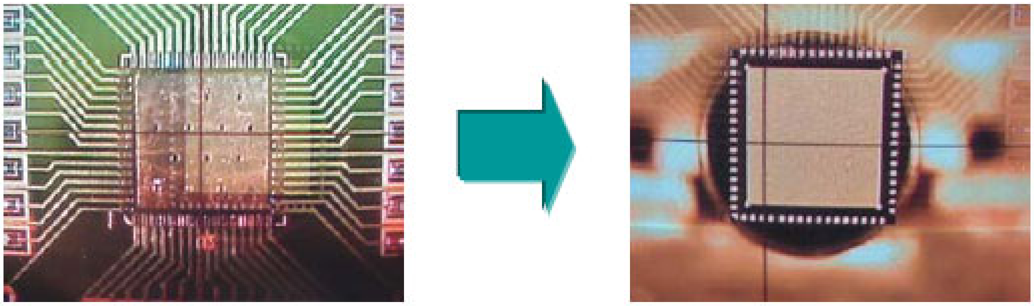
Any rework station should have good look-up or look-down capability with video or optical vision.

4.4.1 Manual Placement (Not Recommended)

The operator picks up the replacement component and places it on the PCB, taking care to align the part as closely as possible so the lands on the component and the PCB are properly aligned. With fine pitch components there is clearly a risk of solder short circuits with this method. Pre-tinned parts, where the paste has already been through one reflow presents a lower risk of solder shorts than solutions that have “wet” solder paste.

4.4.2 Semi-automatic Placement (Recommended)

A vacuum nozzle is used to pick the new package up. The split vision system displays images of both the LGA lands and the footprint on the PCB. The two superimposed images are aligned manually by adjusting the XY table. Once the PCB and package are aligned, the package is placed down on the PCB (See Figure 9.



PCB Image Captured by Camera   Superimpose LGA on PCB

Figure 9. Semi-Automatic Component Placement

4.4.3 LGA Self Alignment

Array LGA and BGA have been shown to be equally tolerant of up to 50% off-pad misplacement. Both package types exhibit self-alignment in any direction including X-axis shift, Y-axis shift, and rotational misplacement. Figure 10 shows device misplacement and Figure 11 shows a 100% self-aligned soldered down device after 50% misplacement was induced.

The best experience with self-alignment has been seen with parts that feature arrays of lands. Parts with irregular solderable features on the bottom of the package and large ground planes do ***not*** show a strong self-alignment capability. For those packages, it is clear that there is no substitute for careful, precise placement of the component on the PCB.

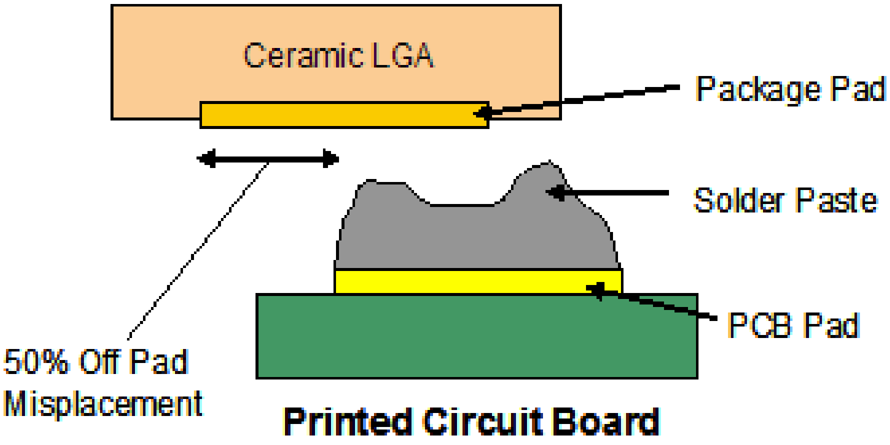


Figure 10. LGA Misplacement of 50%

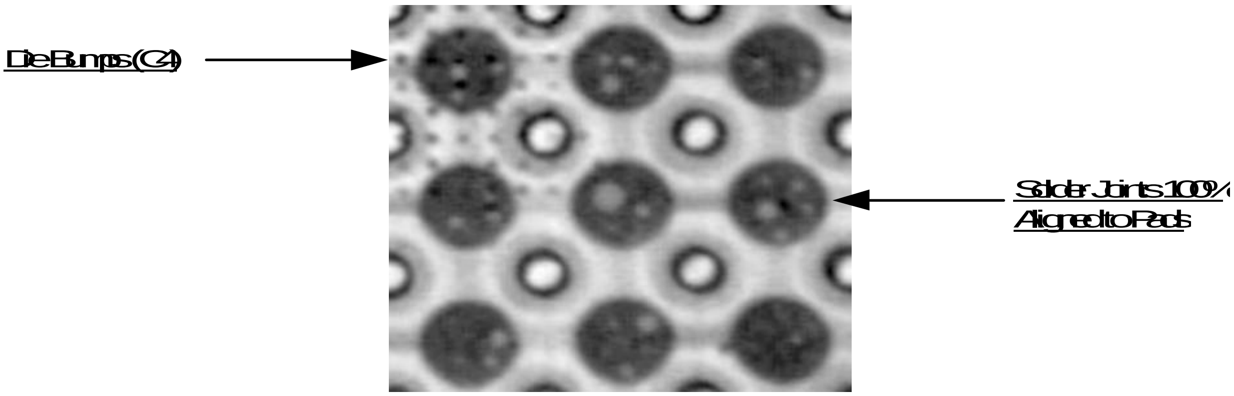


Figure 11. X-ray of perfectly Self-aligned LGA After Misplacement

4.5 Solder Reflow Profile for Lead-Free Paste

Optimal reflow profile depends on solder paste properties and should be optimized and proven out as part of an overall process development. The following guidelines 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^o C, while others will be burned up in a long dwell. 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.

Freescale has achieved good results with Indalloy® 241 with a peak temp of 235^o C to 250^o C and a dwell time above 221^o C for greater than 50 seconds and less than 80 seconds as shown in Figure 12.

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 assembly. 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. To monitor the process, thermocouples must be carefully attached with very small amounts of thermally conductive grease or epoxy directly to the solder joint interface between the package and board.

4.6 Reflow Profile

Experience with specific products and production equipment sets may lead users of LGA to have slightly different profiles that are optimized to their local conditions.

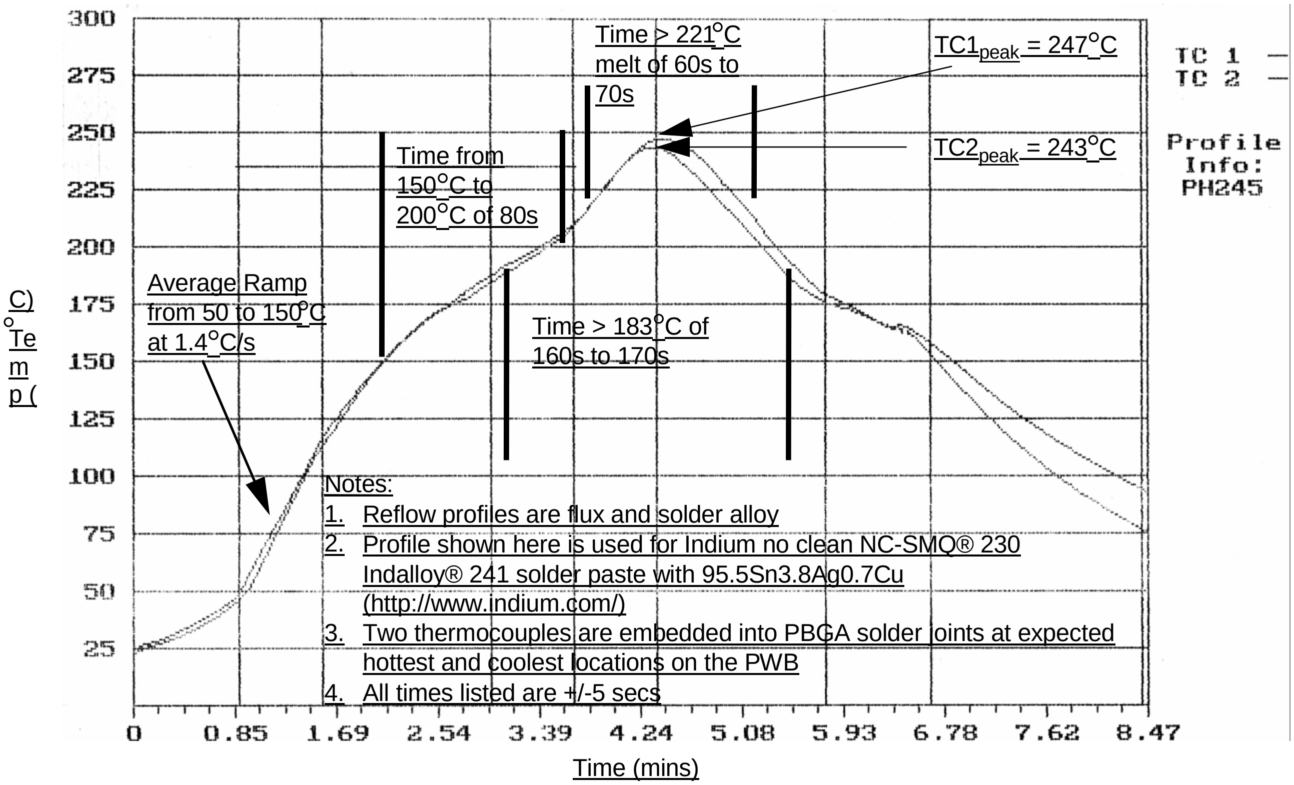


Figure 12. Typical Freescale Pb-Free Board Assembly Reflow Profile (Example is for BGA, LGA Uses Same)

4.6.1 Reflow Atmosphere

Assembly and reliability studies were conducted in a furnace with an air atmosphere. This setup produces excellent results. However, there are advantages in using a nitrogen atmosphere, such as more complete wetting and a reduction in solder joint voids.

4.6.2 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 LGA 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.

Freescale has investigated water soluble solder pastes that do require cleaning in combination with LGA. Using an ion chromatograph, it has been shown that assembly cleanliness is very acceptable, with chlorides detected at 2.11 g/in^2, and bromides at a level of 0.36 g/in^2, following a water clean.

5 LGA Reliability

5.1 Solder Joint Reliability in Temperature Cycling

See Freescale Applications Note AN2920, Manufacturing with the Land Grid Array Package for more details.

5.2 Mechanical Durability

Performance of packages in drop testing depends critically on the thickness of the PCB, the position of the components on the PCB, how the PCB is braced within the application, the customer test conditions and the design of the LGA footprint. Consult with the Application Team for any specific package performance information that may be available.

5.3 Application Space

The application should drive the PCB land construction details. NSMD lands on the PCB are known to give better temperature cycling reliability (See AN2920 for details). However, Freescale has found that SMD pads improved the mechanical durability compared to NSMD pads in drop testing by 10% as measured by first strain to failure in a single impact 4 point bend test. The user should select a PCB land configuration based on their own assessments of their product, bearing in mind the general application space. Freescale has observed a greater sensitivity to mechanical durability in drop testing for some market segments such as cell phones, portable music players, flash memory, etc. Other business segments such as computers, servers, and base stations emphasize good performance with respect to temperature cycling fatigue.

6 References

- AN2920 Manufacturing With the Land Grid Array Package
- AN1902 Quad Flat Pack No-lead (QFN)
- Freescale CBGA Customer Presentation (http://www.freescale.com/files/32bit/doc/package_info/CBGAPRES.pdf)

For assistance or answers to any questions on the information presented in this note, contact the appropriate Freescale product applications team.

NOTES

Archived Archived Archived Archived Archived Archived Archived

How to Reach Us:

Home Page:
www.freescale.com

E-mail:
support@freescale.com

USA/Europe or Locations Not Listed:
Freescale Semiconductor
Technical Information Center, CH370
1300 N. Alma School Road
Chandler, Arizona 85224
+1-800-521-6274 or +1-480-768-2130
support@freescale.com

Europe, Middle East, and Africa:
Freescale Halbleiter Deutschland GmbH
Technical Information Center
Schatzbogen 7
81829 Muenchen, Germany
+44 1296 380 456 (English)
+46 8 52200080 (English)
+49 89 92103 559 (German)
+33 1 69 35 48 48 (French)
support@freescale.com

Japan:
Freescale Semiconductor Japan Ltd.
Headquarters
ARCO Tower 15F
1-8-1, Shimo-Meguro, Meguro-ku,
Tokyo 153-0064, Japan
0120 191014 or +81 3 5437 9125
support.japan@freescale.com

Asia/Pacific:
Freescale Semiconductor Hong Kong Ltd.
Technical Information Center
2 Dai King Street
Tai Po Industrial Estate
Tai Po, N.T., Hong Kong
+800 2666 8080
support.asia@freescale.com

***For Literature Requests Only*:**
Freescale Semiconductor Literature Distribution Center
P.O. Box 5405
Denver, Colorado 80217
1-800-521-6274 or 303-675-2140
Fax: 303-675-2150
LDCForFreescaleSemiconductor@hibbertgroup.com

Information in this document is provided solely to enable system and software implementers to use Freescale Semiconductor products. There are no express or implied copyright licenses granted hereunder to design or fabricate any integrated circuits or integrated circuits based on the information in this document.

Freescale Semiconductor reserves the right to make changes without further notice to any products herein. Freescale Semiconductor makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does Freescale Semiconductor assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. “Typical” parameters that may be provided in Freescale Semiconductor data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including “Typicals”, must be validated for each customer application by customer’s technical experts. Freescale Semiconductor does not convey any license under its patent rights nor the rights of others. Freescale Semiconductor products are not designed, intended, or authorized for use as components in systems intended for surgical implant into the body, or other applications intended to support or sustain life, or for any other application in which the failure of the Freescale Semiconductor product could create a situation where personal injury or death may occur. Should Buyer purchase or use Freescale Semiconductor products for any such unintended or unauthorized application, Buyer shall indemnify and hold Freescale Semiconductor and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that Freescale Semiconductor was negligent regarding the design or manufacture of the part.

Freescale™ and the Freescale logo are trademarks of Freescale Semiconductor, Inc. All other product or service names are the property of their respective owners.

© Freescale Semiconductor, Inc. 2009. All rights reserved.

Document Number: AN3241
Rev. 1.0
10/2009