Freescale Semiconductor
Application Note

Document Number: AN2920
Rev. 2, 12/2008

Manufacturing with the Land Grid Array Package

by *Networking & Multimedia Group*
Freescale Semiconductor, Inc.
Austin, TX

Freescale has introduced the High Coefficient of Thermal Expansion (HCTE) Ceramic Land Grid Array (LGA) to reduce the amounts of lead in finished products and to become Reduction of Hazardous Substances (RoHS) compliant.

This application note describes this HCTE Ceramic LGA and recommends best practices for designing and manufacturing using this package technology.

For assistance or answers to any question on the information that is presented in this document, visit the web site listed on the back cover of this document or send an e-mail to risc10@freescale.com.

Contents

1. Terminology 2
2. What is LGA? 2
3. Benefits of LGA 4
4. Manufacturing with LGA 5
5. LGA Reliability 10
6. Heat Sink Solutions with LGA 12
7. Document Revision History 13

1 Terminology

Term	Meaning
CBGA	Ceramic Ball Grid Array
HCTE	High Coefficient of Thermal Expansion
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
RoHS	Reduction of Hazardous Substances
SMD	Solder Mask Defined
SMT	Surface Mount Technology

2 What is LGA?

The LGA package is a standard flip-chip Ball Grid Array (BGA) shipped with no spheres. Figure 1 shows the top and bottom sides of an LGA device. LGA has been available for hand-held devices in the small plastic package from Freescale and others for several years. Freescale is now introducing the LGA package using a high coefficient of thermal expansion (HCTE) ceramic in larger body sizes.

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 (refer to doc: *CBGAPRES*). 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.

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. Figure 2 illustrates case outlines of a BGA package and an LGA package of the same device.

HCTE flip-chip devices do not require spheres because the coefficient of thermal expansion 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]. Typically, most epoxy-glass or polyimide-glass PCBs have a CTE of 16–22 ppm/ºC.

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

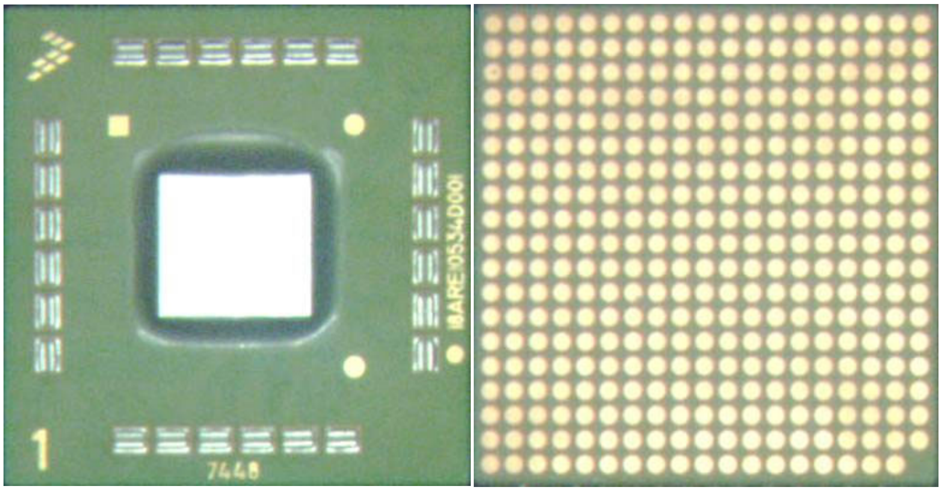

Figure 1. Top and Bottom View of a HCTE 360 Pad LGA Device

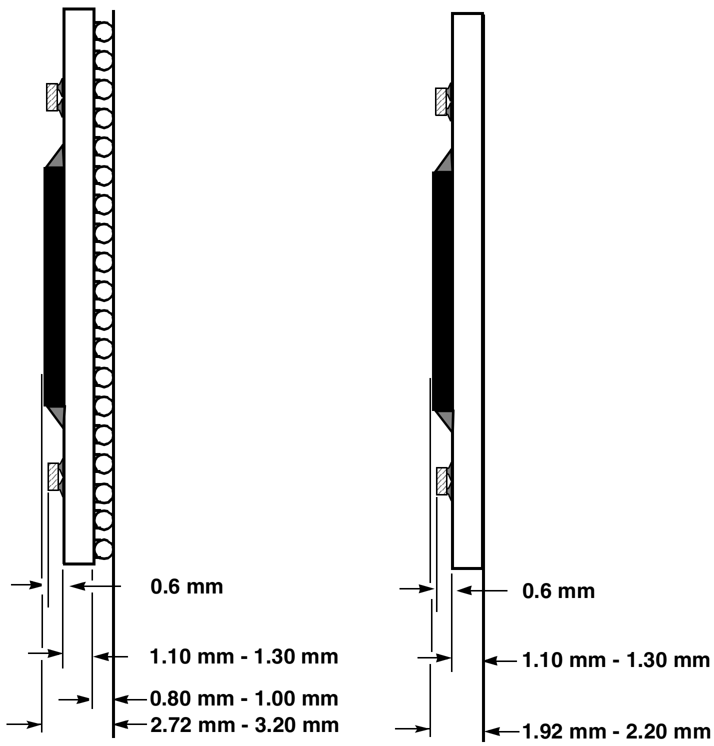

Figure 2. Unmounted Device Profiles (BGA on left, LGA on right)

The LGA pad uses the same 0.1 to 0.5 µm of electroless gold plating over electroless nickel as has been used reliably for many years in the traditional BGA configuration. Figure 3 is an image of a typical LGA pad.

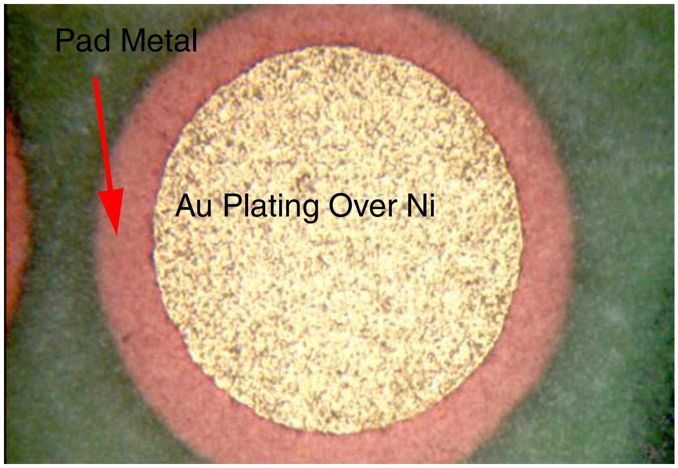

Figure 3. Typical LGA Pad

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 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.

3 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.
- LGA can use the same recommended board assembly process as CBGA (refer to doc: *CBGAPRES*).

These benefits are discussed further in the sections that follow.

4 Manufacturing with LGA

4.1 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 shown to succeed in Freescale and end user tests, such as solder stencil thickness, aperture diameter, paste release characteristics, and practices to ensure consistent solder paste volumes that exceed recommended minimums.

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 in Figure 4.

4.1.1 Solder Paste Stencils

The historically recommended CBGA 0.20 mm (8 mil) thick stencil with 0.72 mm (28.5 mil) diameter apertures provides excellent assembly results with LGA. This combination results in approximately 4800 mil^3 of solder paste applied to the pads. Freescale has tested both laser cut and chemically etched stencils with positive results. Additionally, recent studies resulted in successful assembly with a 0.15 mm (6 mil) thick stencil with aperture diameters ranging from 0.56 to 0.89 mm (22 to 35 mil). Table 1 shows calculated volumes of various stencil thickness versus various apertures. As with BGA assembly, Freescale recommends actual paste volume of 7000 mil^3. However, Freescale and customers have successfully built boards using the minimum allowable paste volume of about 4800 mil^3.

Table 1. Solder Volumes for Various Apertures

Stencil Thickness (mil)	Deposit Base Diameter (mil)								
	28	29	30	31	32	33	34	35	36
5	2925	3137	3357	3585	3820	4063	4312	4570	4835
6	3510	3765	4029	4302	4584	4875	5175	5484	5802
7	4095	4392	4700	5019	5348	5688	6037	6398	6769
8	4680	5020	5372	5736	6112	6500	6900	7312	7736
9	5265	5647	6043	6453	6876	7313	7762	8226	8703
10	5849	6275	6715	7170	7640	8125	8625	9140	9670
11	6434	6902	7386	7887	8404	8938	9487	10054	10636
12	7019	7530	8058	8604	9168	9750	10350	10968	11603
13	7604	8157	8729	9321	9932	10563	11212	11882	12570
14	8189	8785	9401	10038	10696	11375	12075	12796	13537

Notes:

1 0.95 multiplier for typical non-cylindrical deposit shape.

2 Minimum allowable paste volume: 4800 mil^3.

3 Recommended target paste volume: 7000 mil^3.

4 Non-tapered aperture opening.

4.1.2 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°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.

When using a lead-free solder, such as the SnAgCu alloy mentioned previously, preheat the components so the temperature of the package raises to 100°C over a period of no less than 50 seconds. Using either infrared or convection reflow, ensure a peak temperature minimum that allows all solder joints on the assembly to fully reflow and a maximum peak temperature that is below the maximum that the devices or solder material can tolerate. Also, ensure a dwell time of less than three minutes above the solder melt.

Always understand the MSL and peak temperature of all components before attempting lead-free solder temperatures. Most HCTE devices from Freescale are qualified to MSL1 at 260°C.

Freescale has achieved good results with Indalloy® 241 with a peak temp of 235° to 250°C and a dwell time above 217°C for greater than 50 seconds and less than 80 seconds as shown in Figure 4.

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.

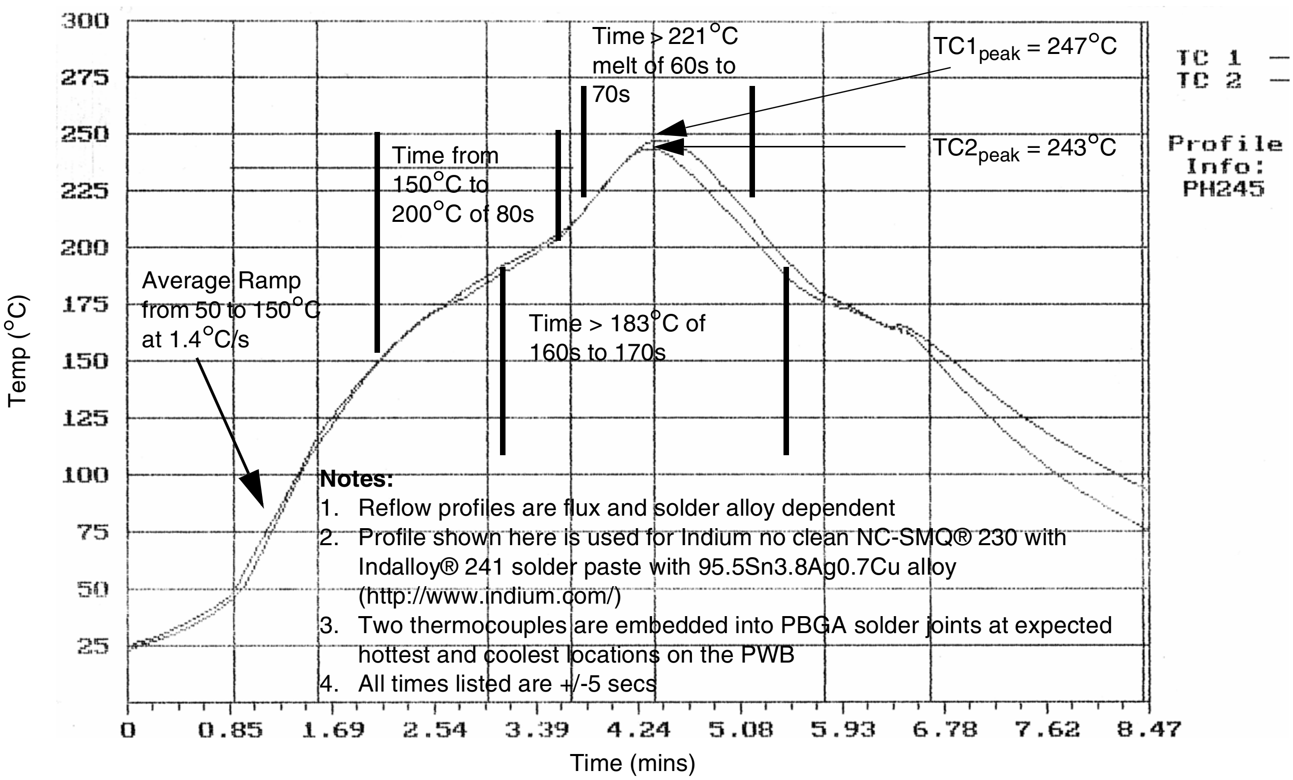

Figure 4. Typical Freescale Pb-Free Board Assembly Reflow Profile

4.1.3 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.

4.1.4 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 $\mu g/in^2$, and bromides at a level of 0.36 $\mu g/in^2$, following a water clean.

4.2 Automated Placement of LGA

A study using a Universal GSM with version 4.5.0 supplemental software showed that the pick and place vision system easily recognized LGA pads with few required modifications to lighting. Side-lit intensities of –2 to –5 allow the pick and place machine to recognize the LGA pads.

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 5 illustrates device misplacement and Figure 6 is an X-ray of a 100% self-aligned soldered down device after 50% misplacement was induced.

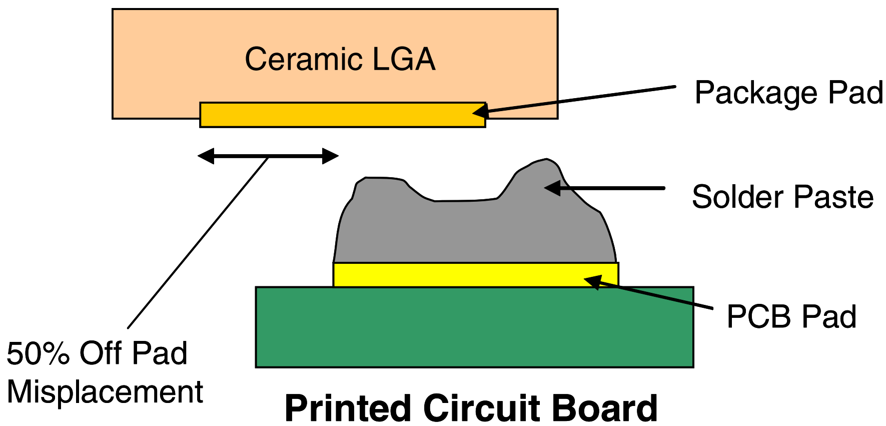

Figure 5. LGA Misplacement of 50%

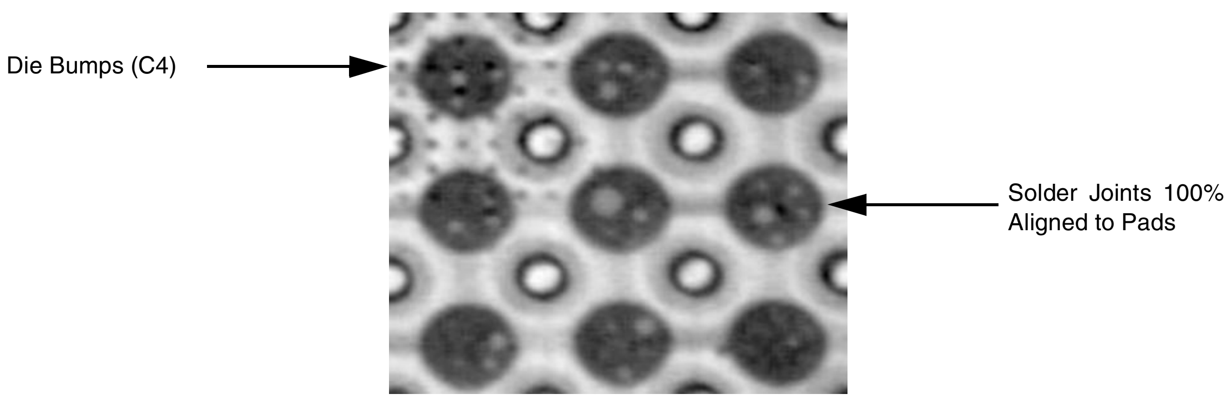

Figure 6. X-ray of Perfectly Self-Aligned LGA After Misplacement

4.3 Daisy-Chain Samples

Daisy-chain samples connect all pins in series and are used to verify processes and connectivity after assembly. Daisy-chain samples are in the same package form-factor as the finished product, but do not usually contain a die or capacitors attached to the substrate. Customers who have never used LGA are encouraged to first test their process with daisy-chain samples. Daisy-chain samples are available for all HCTE LGA products. Contact your sales representative for daisy-chain samples.

Design files can be provided upon request for matching test boards and stencils used by Freescale to generate any test results mentioned in this application note.

4.4 PCB Design Considerations

Freescale's LGA test boards use an organic solderability protectant (OSP) finish. However, all common PCB surface finishes are believed to be compatible with LGA.

As with CBGA, a 0.72 mm (28.5 mil) diameter pad is recommended for a 1.27 mm pitch LGA. Studies are currently underway to determine pad geometries using 1.0 mm pitch LGA products that are expected to be introduced in the future. A copper pad diameter in the range of 0.50 mm to 0.58 mm (20 mil to 23 mil) is expected to provide excellent results for 1.0 mm pitch LGA products.

A copper defined or non-soldermask defined (NSMD) pad is the most common and is believed to provide the most consistent solderable surface. A soldermask opening that is 0.83 mm to 0.88 mm (32.5 mil to 34.5 mil) in diameter would be used with the 0.72 mm (28.5 mil) copper pad diameter for the NSMD pad. In some cases, where a stronger bond between pad and motherboard may be required for improved drop/bend/vibration test performance, an soldermask defined (SMD) pad, where the copper diameter is larger than that of the soldermask, may be advantageous.

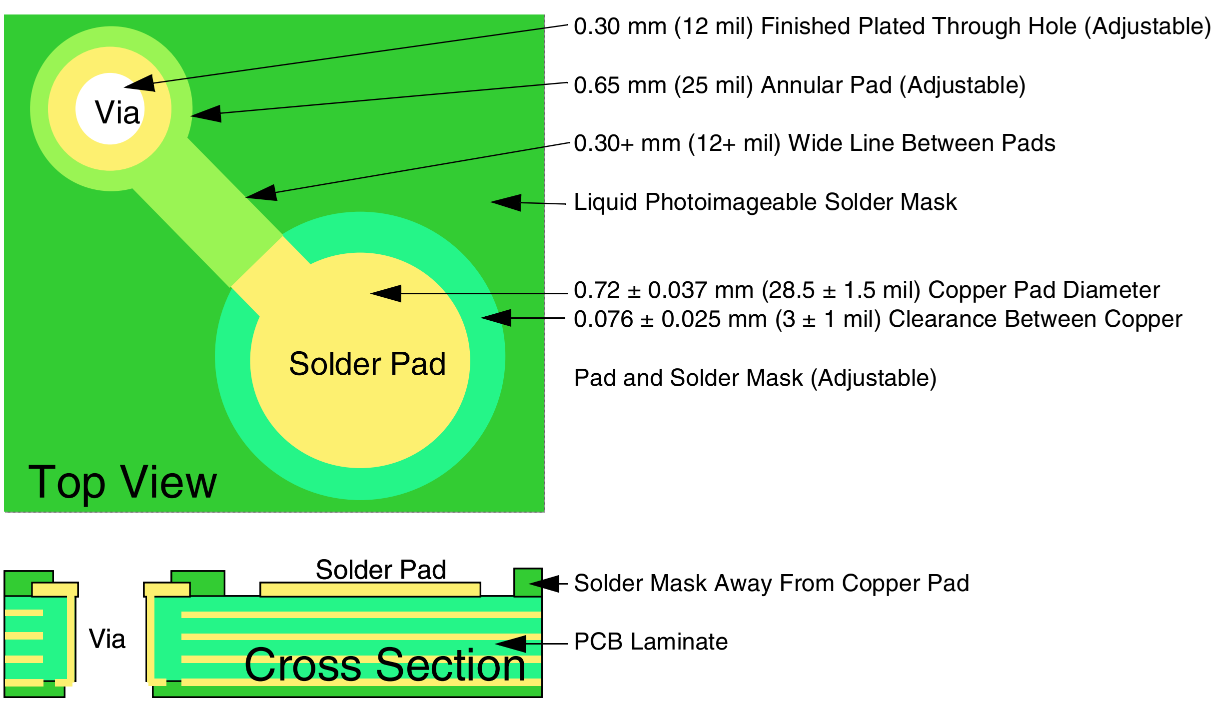

Figure 7. Recommended NSMD Pad for CBGA and LGA

Freescale recommends a minimum trace width of 0.30 mm at the point the trace attaches through the LGA pad, as shown in Figure 7. Freescale also recommends that there be adequate filleting where the trace meets the pad. Trace cracking has been observed near the pad when trace widths are less than the recommended width.

5 LGA Reliability

5.1 Solder Joint Reliability

Solder joint reliability studies to date indicate that LGA or BGA greatly exceed typical industry reliability requirements when the recommendations in this document are followed. Telecommunications and computing customers typically require 3000 failure-free 0 to 100°C thermal cycles. One Freescale study on 25 mm body, 360 HCTE LGA using a 1.57 mm thick, OSP finish four-layer board, showed at least 3000 failure-free cycles with both leaded and lead-free solder paste. This study used the standard recommended stencil geometry. Results are shown in the Weibull plot in Figure 8.

A subsequent study using varied stenciled geometries and proprietary solder paste between Freescale and a leading contract manufacturer yielded results of up to 12,000 failure-free cycles before the study was terminated, as shown in Figure 9.

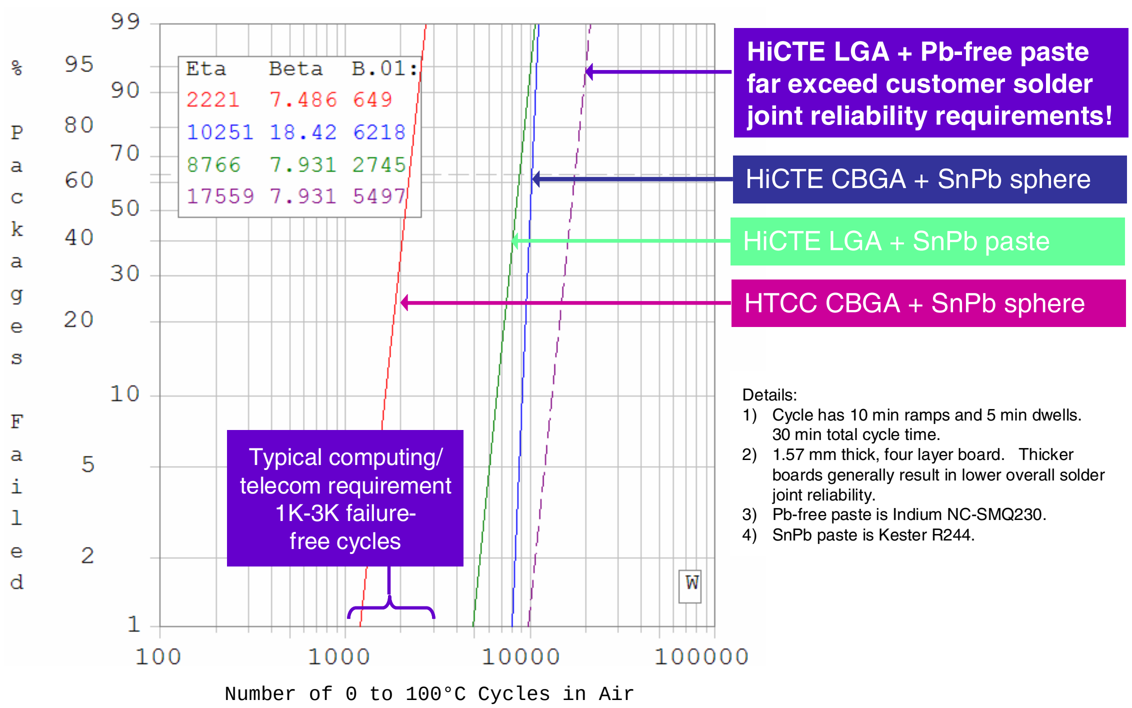

Figure 8. Weibull Plot of Typical Conditions

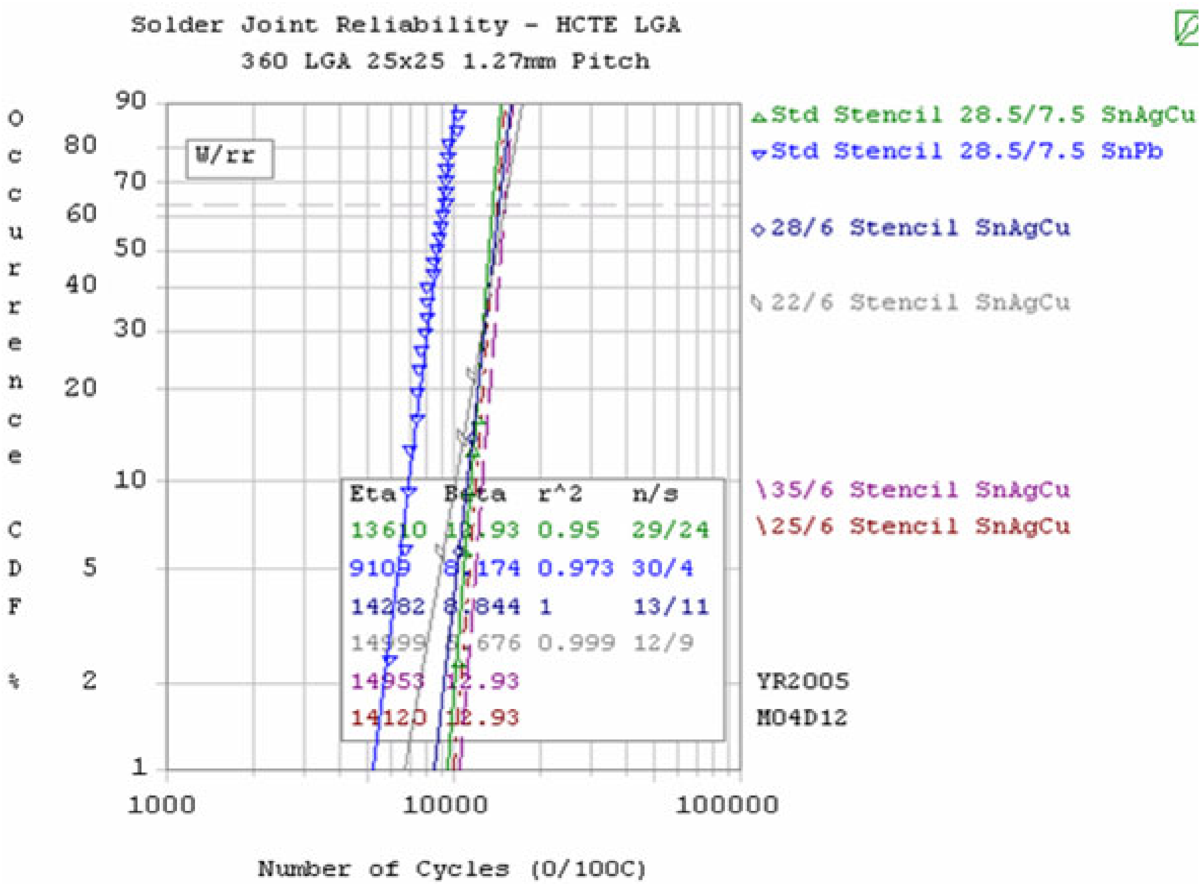

Figure 9. Weibull Plot of HCTE 360 Pad LGA with Optimized Recommendations

More studies are underway with more severe thermal cycling (-40º to 125ºC) that may be required for military and aerospace applications.

5.2 Solder Joint Voiding

As with BGA, solder joint voids do occur in LGA. Voids in LGA can be larger due to geometry and greater ratio of flux to solder. IPC-A-610D specifies a greater than 25% voided area is a defect for BGA, however, it does not specify the defect limit for LGA. This specification applies only to collapsing spheres, which LGA does not have. Thermal cycling with LGA solder joint voiding of greater than 25% repeatedly shows excellent solder joint reliability. All studies referenced in this document use LGA devices that, when soldered to the test board, contained some joints with greater than 25% voiding.

The most significant factor in solder joint voiding is the solder paste formulation. Consult your solder paste supplier for what they believe to be a low voiding solder paste. Other influences to void formation are reflow profile, solder paste volume, PCB surface finish, and oxidation, as discussed in previous sections. Actions to reduce solder joint voids include using a nitrogen reflow atmosphere with less than 100 ppm oxygen and extending pre-heat times and/or time above liquidus in reflow.

After extensive thermal cycling, solder joint fractures appear similar to BGA joint fails. Fractures can occur through bulk solder at or near the package and/or PCB pad. Figure 10 shows a void in a cross-section of an LGA joint. This joint has been cycled to failure with a fracture propagating though bulk solder and through the void.

Studies are underway with various voiding levels to better understand the effect of voids on long term reliability.

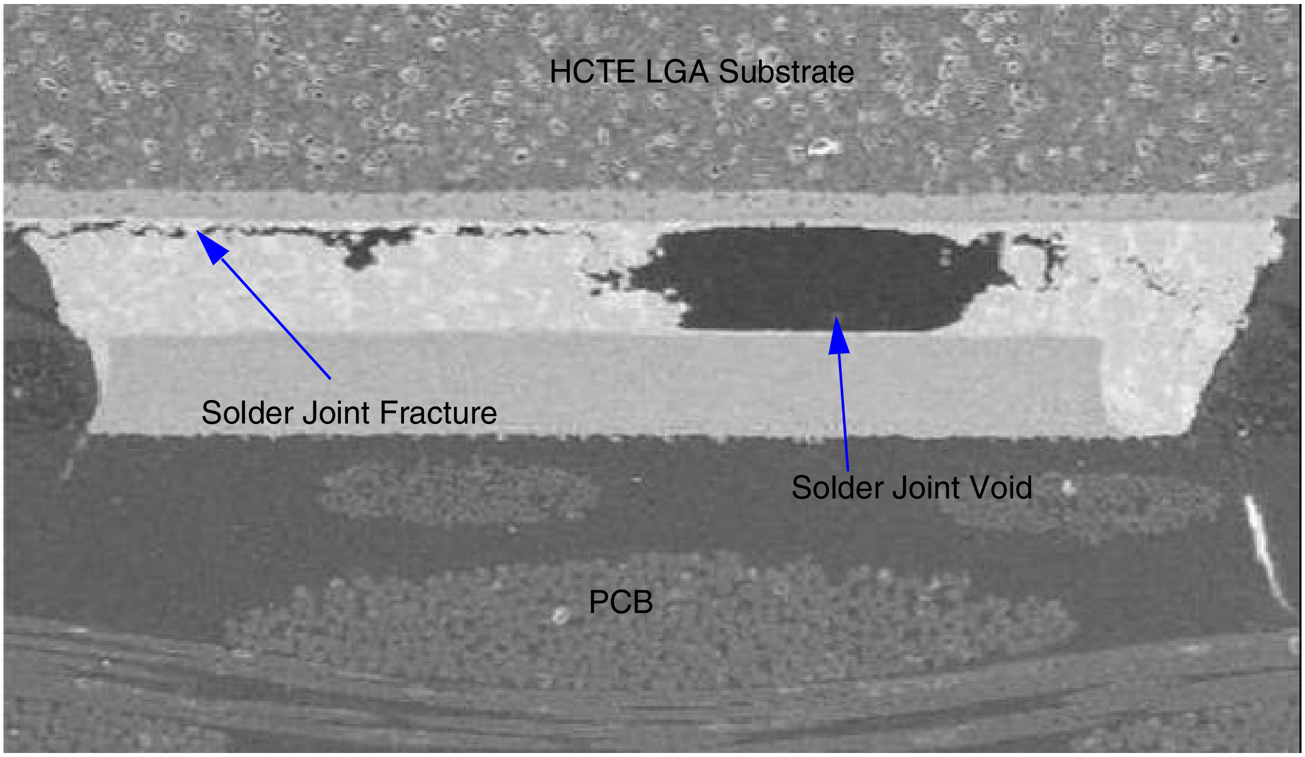

Figure 10. Solder Joint Fracture of LGA

6 Heat Sink Solutions with LGA

Heat sink solutions that attach through the circuit board are recommended. Please note that when converting from BGA to LGA, the slightly lower stand-off height with LGA must be accounted for in heat sink mounting. As with BGA, ensure the heat sink provides a uniformly distributed force.

There may not be enough room between an LGA package and the circuit board to accommodate a chip clip-on heat sink due to the lower stand-off height of LGA. Investigation is ongoing for viable chip clip-on heat sink solutions.

7 Document Revision History

Table 2 provides a revision history for this application note.

Table 2. Document Revision History

Rev. No.	Date	Substantive Change(s)
2	12/2008	Added “High Coefficient of Thermal Expansion (HCTE) Ceramic: before “Land Grid Array” and “HCTE Ceramic” before “LGA” on page 1. Applied new template to include new back page info.
1	10/2005	Initial release.

THIS PAGE INTENTIONALLY LEFT BLANK

How to Reach Us:

Home Page:
www.freescale.com

Web Support:
http://www.freescale.com/support

USA/Europe or Locations Not Listed:
Freescale Semiconductor, Inc.
Technical Information Center, EL516
2100 East Elliot Road
Tempe, Arizona 85284
1-800-521-6274 or
+1-480-768-2130
www.freescale.com/support

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)
www.freescale.com/support

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 China Ltd.
Exchange Building 23F
No. 118 Jianguo Road
Chaoyang District
Beijing 100022
China
+86 10 5879 8000
support.asia@freescale.com

For Literature Requests Only:
Freescale Semiconductor
Literature Distribution Center
P.O. Box 5405
Denver, Colorado 80217
1-800 441-2447 or
+1-303-675-2140
Fax: +1-303-675-2150
LDCForFreescaleSemiconductor
@hibbertgroup.com

Document Number: AN2920
Rev. 2
12/2008