BGA (Ball Grid Array)

National Semiconductor
Application Note 1126
December 2000

CONTENTS

INTRODUCTION

The consumer electronics industry is constantly seeking solutions that will resolve challenges to design their solutions smaller and less expensive. The Ball Grid Array (BGA) Package has impacted one of the industry's most limiting design factors, board real estate and its increasing cost. The cost can be indirectly related to the lost revenue caused by a competitive disadvantage when the competition's solution is smaller, better, and contains more functionality. Today's semiconductor, packaging and printed circuit board (PCB) manufacturing technologies make it possible to fit camcorders in the palm of your hand and cell phones and PDA's in your shirt pocket. These applications require PCB densities that include components on the top and bottom sides of the board. The BGA package is one solution that addresses this concern.

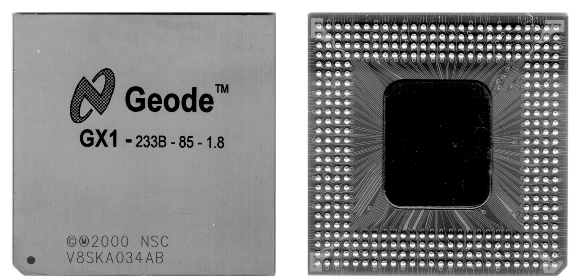
The many benefits of using BGA over similar lead count leaded devices include:

1. Efficient use of board space.
2. Improved thermal and electrical performance and ease of enhancing both.
3. Compatibility with existing surface mount, test and handling equipment.
4. Improved surface mount yields when compared to fine pitch leaded devices.
5. Lower profile (package thickness).
6. Potential lower total cost of ownership compared to leaded devices due to reduced scrap, rework and lack of need for fine pitch assembly equipment.

This application note will focus on general information about the Enhanced Ball Grid Array (EBGA), and Plastic Ball Grid Array (PBGA) packages as well as provide information about their implementation into products and surface mount assembly, see Figure 1.

AN101094-10

PBGA



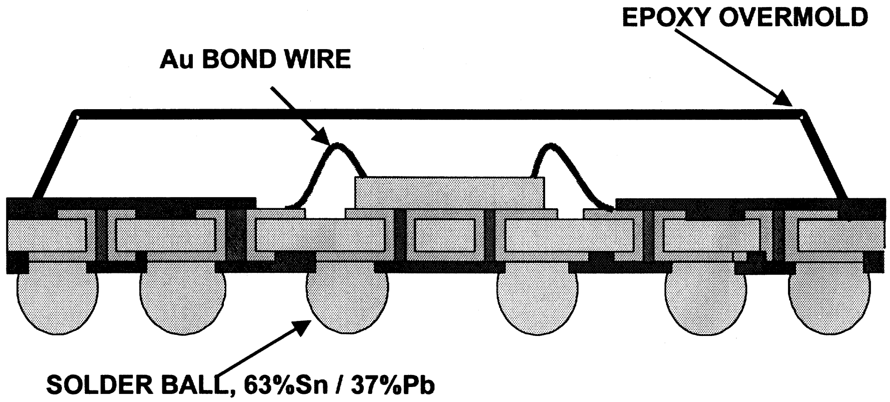
AN101094-9

EBGA

FIGURE 1. PBGA & EBGA

PACKAGE OVERVIEW

PBGA Construction



AN101094-11

FIGURE 2. Cross-Sectional View of PBGA Mounted to PCB

The PBGA package is based on a PCB substrate fabricated of Bismaleimide Triazine (BT) epoxy/glass laminate; see *Figure 2*. This material is used over standard and multi-functional FR4 laminates for its high glass transition temperature of 170 - 215˚ C and heat resistance (230˚ C exposure for 30 minutes with no degradation). The standard core thickness of this two-layer substrate is typically 0.2 mm with 18 µm half ounce (or 0.7 mil) rolled copper on each side. A two-mil (or less) thickness over BT-epoxy/glass dry or multiple-pass wet film solder mask is currently used to ensure that all the substrate vias will be completely tented.

The silicon chip containing an integrated circuit is die bonded to the top side of the substrate using die attach adhesive typical of that found in leaded devices. The chip is then gold wire-bonded to wire bond pads on the substrate. Traces from the wire bond pads take the signals to vias that carry them to the bottom side of the substrate, and then to circular solder pads.

The bottom side solder pads are laid out on a square or rectangular grid with a constant 1.27 mm pitch. This pitch, as well as a 1.0 mm pitch, is prescribed by the JEDEC registration for the square PBGA (MO-151).

An over mold is then performed to completely cover the chip, wires and substrate wire bond pads. Typical feature dimensions common to most PBGA configurations, as discussed above, are summarized in *Table 1*.

TABLE 1. Typical PBGA configuration

Package	316 PBGA	352 PBGA	388 PBGA
Dimension (X x Y x Z)	27 x 27 x 2.13 mm	35 x 35 x 2.33 mm	35 x 35 x 2.33 mm
Ball Pitch	1.27 mm	1.27 mm	1.27 mm
Ball Diameter	0.75 mm	0.75 mm	0.75 mm
Ball Composition	63% Sn/37% Pb	63% Sn/37% Pb	63% Sn/37% Pb
Weight	2.45 grams	4.05 grams	4.05 grams
Substrate Material	Bismaleimide Triazine (BT)	Bismaleimide Triazine (BT)	Bismaleimide Triazine (BT)
Coplanarity (JEDEC)	0.2 mm	0.2 mm	0.2 mm
Moisture Sensitivity	MSL 3	MSL 3	MSL3

EBGA Construction

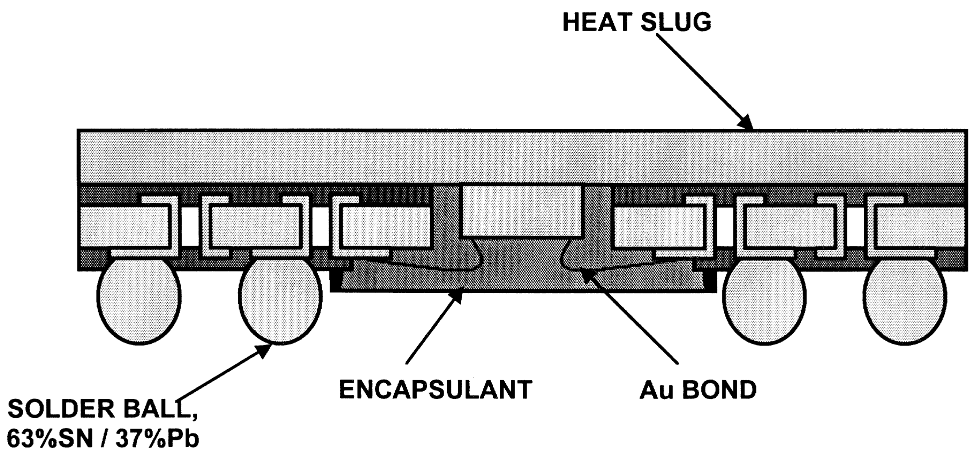


FIGURE 3. Cross-Sectional View of EBGA Mounted to PCB

The EBGA package is configured differently from the PBGA to provide a greater thermal and, if required, electrical performance; see *Figure 3*. The thermal advantage for this package is provided by attaching the die to the bottom of a heat spreader or slug that also forms the topside of the package. Also, because the heat spreader is the top of the package, the package surface is exposed to airflow and the thermal resistance is very low. The heat spreader or slug is laminated to a printed circuit board (PCB) substrate fabricated of Bismaleimide Triazine (BT) epoxy/glass laminate. The silicon chip containing an integrated circuit is die bonded to the heat spreader using die attach adhesive typical of that found in leaded devices. The chip is then gold wire-bonded to wire bond pads on the circuitized substrate. Traces from the wire bond pads take the signals to circular solder pads.

The bottom side solder pads are laid out on a square or rectangular grid with a constant 1.27 mm pitch. This pitch, as well as a 1.0 mm pitch, is prescribed by the JEDEC registration for the square EBGA (MO-151).

An encapsulation is then performed to completely cover the chip, wires and substrate wire bond pads. Typical feature dimensions common to most EBGA configurations, as discussed above, are summarized in *Table 2*.

TABLE 2. Typical EBGA configuration

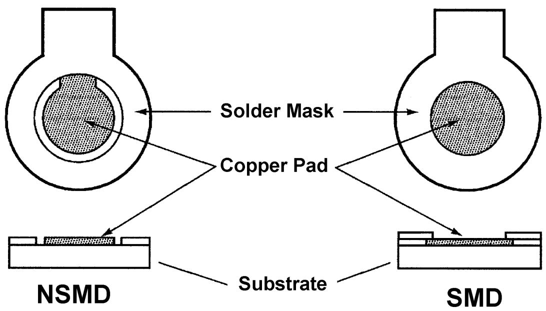
Package	215 EBGA	352 EBGA	432 EBGA
Dimension (X x Y x Z)	27 x 27 x 1.72 mm	35 x 35 x 1.72 mm	40 x 40 x 1.72 mm
Ball Pitch	1.27 mm	1.27 mm	1.27 mm
Ball Diameter	0.75 mm	0.75 mm	0.75 mm
Ball Composition	63% Sn/37% Pb	63% Sn/37% Pb	63% Sn/37% Pb
Weight	5.06 grams	8.26 grams	10.88 grams
Substrate Material	Bismaleimide Triazine (BT)	Bismaleimide Triazine (BT)	Bismaleimide Triazine (BT)
Coplanarity (JEDEC)	0.2 mm	0.2 mm	0.2 mm
Moisture Sensitivity	MSL 3	MSL 3	MSL3

PACKAGE HANDLING/SHIPPING MEDIA

The BGA packages are shipped in a high temperature range thin matrix tray that complies with the JEDEC standards. Typical JEDEC trays have the same 'x' and 'y' outer dimensions and are easily stacked for storage and manufacturing.

DESIGN RECOMMENDATIONS

Solder Pad Geometry



AN101094-1

FIGURE 4. NSMD and SMD Pad Definition

Two types of land patterns are used for surface mount packages: 1) Non-Solder Mask Defined pads (NSMD) and, 2) Solder Mask Defined pads (SMD). NSMD has an opening that is larger than the pad, whereas SMD pads have a solder mask opening that is smaller than the metal pad. *Figure 4* illustrates the two different types of pad geometry.

NSMD is preferred because the copper etch process has tighter control than the solder masking process.

NSMD pads require a ±0.075 mm clearance around the copper pad and solder mask this avoids overlap between the solder joint and solder mask and accounts for mask registration tolerances.

SMD pad definition can introduce stress concentration points near the solder mask on the PCB side. Extreme environmental conditions such as large temperature variations may cause fatigue that leads to cracked solder joints and reliability problems.

For optimal reliability, National recommends a 1:1 ratio between the package pad and the PCB pad for the BGA wherever trace routing is not a constraint. The ratio may be reduced to 1:0.8 if absolutely necessary. See *Table 3*.

TABLE 3. Guidelines for Pad Design

PBGA 352 ball with 1.27 mm pitch	NSMD	SMD
Solder Ball Diameter	0.75 mm	0.75 mm
PCB Pad Diameter	0.64 mm	0.78 mm
Solder Mask Opening Diameter	0.78 mm	0.64 mm

Escape Routing Guidelines

A typical PBGA substrate has four or five rows of solder balls around the periphery of the package, which provides interconnection to the PCB. The number of lines routed (**N**) between the pads on the PCB is defined by the pad size and the line width and spacing capabilities of the PCB manufacturer. The following relation ship is used to define **N**:

$$N = \frac{P - D - S}{L + S}$$

AN101094-2

P = Pad Pitch
D = Pad Diameter
L = Line Width
S = Line Space

For NSMD pads, exposure of underlying copper traces is forbidden, so the diameter and tolerance of the solder mask opening define **D**.

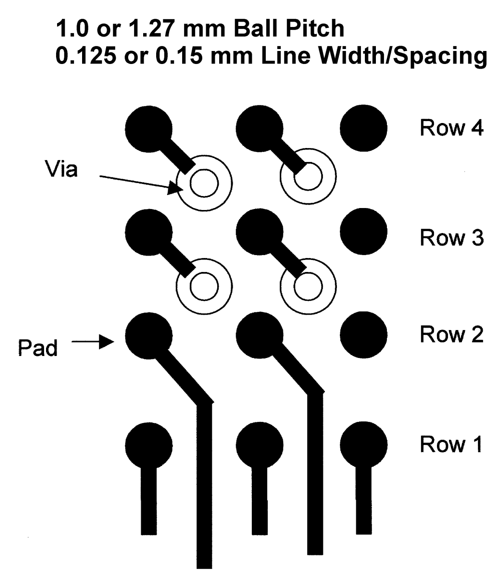
The number of routing lines as a function of pad pitch for various PCB line space/width geometries is shown in *Table 4*. Routing assumes a four-layer board (2 signal and 2 ground) with NSMD pads on the PCB. The package pad to PCB pad ratio is 1:1.

TABLE 4. Recommended Number of Routing Line

	Pad pitch 1.27 mm
L/S = 0.15 mm	1
L/S = 0.125 mm	2
L/S = 0.10 mm	2

Either a 1.0 or 1.27 mm pitch PBGA with four rows of solder balls can be routed to a four layer PCB (*Figure 5*) using a 0.15 mm (6 mil) line/space. The first two ball rows can be routed to one signal layer while the third and fourth ball rows can be routed to a second signal layer.

Routing possibilities become more complicated for a four-layer board, if there are five rows of solder balls. For a 1.27 mm ball pitch PBGA, a 0.125 mm (5 mil) PCB line/space design will be necessary for routing (*Figure 6*). A 1.0 mm PBGA will require a 0.10 mm (4 mil) line/space to successfully route 5 rows of solder balls to a four-layer PCB (*Figure 7*). For both packages, the first three ball rows are routed to one signal layer while the fourth and fifth ball rows are routed to a second signal layer.



AN101094-3

FIGURE 5. Routing for Four Rows of Solder Balls

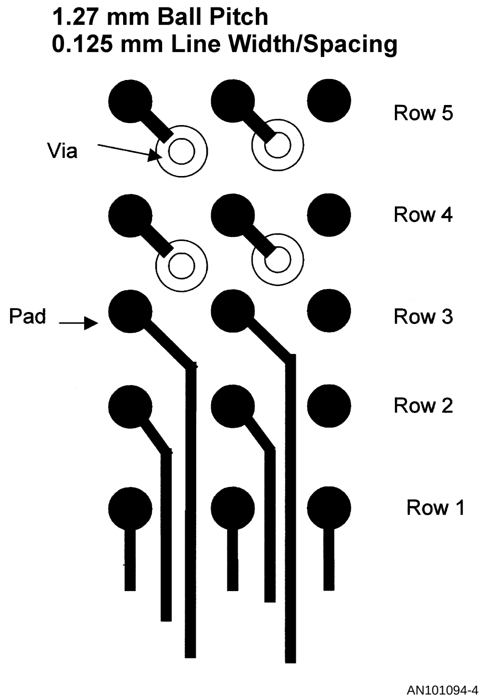


FIGURE 6. Routing for Five Rows of Solder Balls

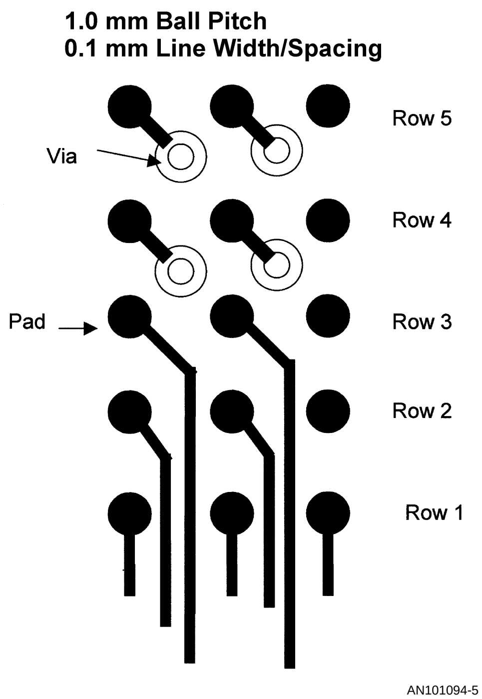


FIGURE 7. Routing for Five Rows of Solder Balls

Via Density

Via density has a significant impact on the routability of PBGA, especially as the solder ball pitch decreases to 0.8 mm. *Figure 8* shows the routability of a 0.8 mm ball pitch PBGA using 0.1 mm line space and width. Mechanically drilled holes below 0.30 mm (12 mil) in diameter, begin to add substantial cost to the PCB, with 0.20 mm (8 mil) being the minimum diameter that can be typically drilled in high-volume PCB manufacturing. As traditional PCB material and processes have been pushed to the limit and are unsatisfactory in meeting the wiring densities, new PCB manufacturing technologies have emerged. *Table 5* list trends for PCB used in portable computers.

TABLE 5. Feature Trends for Portable Computer PCB's

	1997	2000	2002
Number of Layers	5 - 8	7 - 9	7 - 10
Inner Layer Thickness (mm)	0.1	0.7	0.5
Via Diameter (mm)	0.28	0.125	0.1
Land Diameter (mm)	0.48	0.25	0.25
Line Space/Width (mm)	0.1	0.075	0.05

To increase wiring density and improve electrical performance, thin layers are fabricated onto a PCB core with small micro-vias. These micro-via technologies are referred to as build-up-board (BUB) technologies and are cost-effective alternatives to PCB processes for manufacturing board substrates.

There are many variations of BUB boards, though a generic structure consists of two-layers of un-reinforced epoxy dielectric coated on to a rigid (core). Micro-vias are typically formed using either laser or photolithography processes. The fabrication of BUB boards requires new processes and materials compared to standard PCB technologies. *Table 6* shows some micro-via geometries available to facilitate the routing of BGA packages.

The cost trade-offs for deciding on a micro-via PCB design compared to a standard PCB design is a function of the via forming process, materials used, and the number of vias. Consult your PCB manufacturer for a detailed cost analysis.

TABLE 6. Available Micro-Via Geometries

Line Width (mm)	0.07 - 0.1
Line Space (mm)	0.075 - 0.1
Micro-via Hole Size (mm)	0.075 - 0.1
Micro-via Pad (mm)	0.2 - 0.3
Drilled Through Hole (core) (µm)	250
Drilled Through Hole Pad (µm)	500
Buried Via Hole (µm)	200
Buried Via Capture Pad (µm)	450

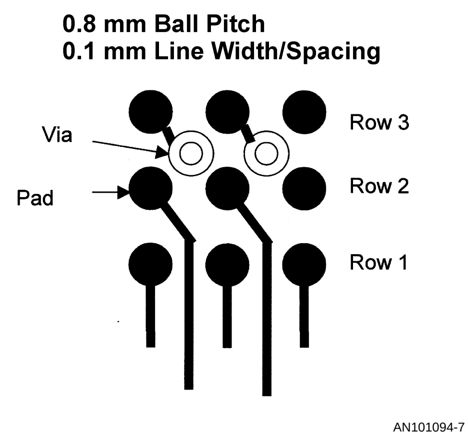


FIGURE 8. Routing for Three Rows of Solder Balls

ASSEMBLY RECOMMENDATIONS

Process Flow & Set-up Recommendation:

The BGA surface mount assembly operations include:

- PCB plating requirements
- Screen printing the solder paste on the PCB
- Monitoring the solder paste volume (uniformity)
- Package placement using standard SMT placement equipment
- X-ray pre reflow check - paste bridging
- Reflow and cleaning (dependent upon the flux type)
- X-ray post reflow check - solder bridging & voids

PCB Plating Recommendations:

A uniform PCB plating thickness is key for high assembly yield.

- PCB with Organic Solderability Preservative coating (OSP) finish is recommended.
- For PCBs with electroless, nickel-immersion, gold finish, the gold thickness recommendation is 0.15 µm ± 0.05 µm to avoid solder joint embrittlement and ensure solderability to the nickel.

Solder Paste Printing

Solder paste deposition using a stencil-printing process involves the transfer of the solder paste through pre-defined apertures with the application of pressure. Stencil parameters such as aperture area ratio and the fabrication process have a significant impact on paste deposition. Inspection of the stencil prior to placement of the BGA package is highly recommended to improve board assembly yields.
Three typical stencil fabrication methods include:

- Chem-etch
- Laser cut
- Metal additive processes

  Nickel-plated electro polished chem-etch or Laser cut with tapered aperture walls (5˚ tapering) is recommended to facilitate paste release. The recommended aperture size is 0.1 mm larger than the pad size to allow 0.05 mm overprinting on each side.

Paste Recommendations

No clean type 3 or 4 paste is recommended.

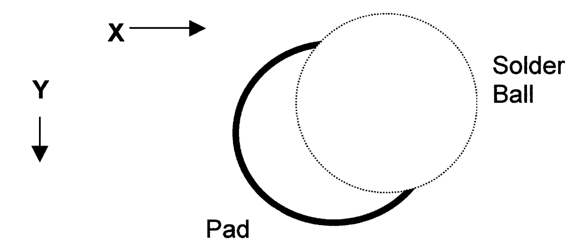
Component Placement

BGA packages are placed using standard pick and place equipment with a placement accuracy of ±0.10 mm. Component pick and place systems are composed of a vision system that recognizes and positions the component and a mechanical system which physically performs the pick and place operation. Two commonly used types of vision systems are: (1) a vision system that locates a package silhouette and (2) a vision system that locates individual bumps on the interconnect pattern. The latter type renders more accurate place but tends to be more expensive and time consuming. Both methods are acceptable since the parts align due to a self-centering feature of the BGA solder joint during solder reflow.

BGAs have excellent self-alignment during solder reflow if a minimum 50% of the ball is aligned with the pad. The 50% accuracy is in both the X and Y direction as determined by the following relation,

50% misalignment for a 0.63 mm and 0.35 mm diameter pads are ±0.22 mm and ±0.12 mm, respectively. As stated above, standard pick and place equipment can accurately place components within this degree of accuracy.

$$\frac{D}{2} = (X^2 + Y^2)^{1/2}$$



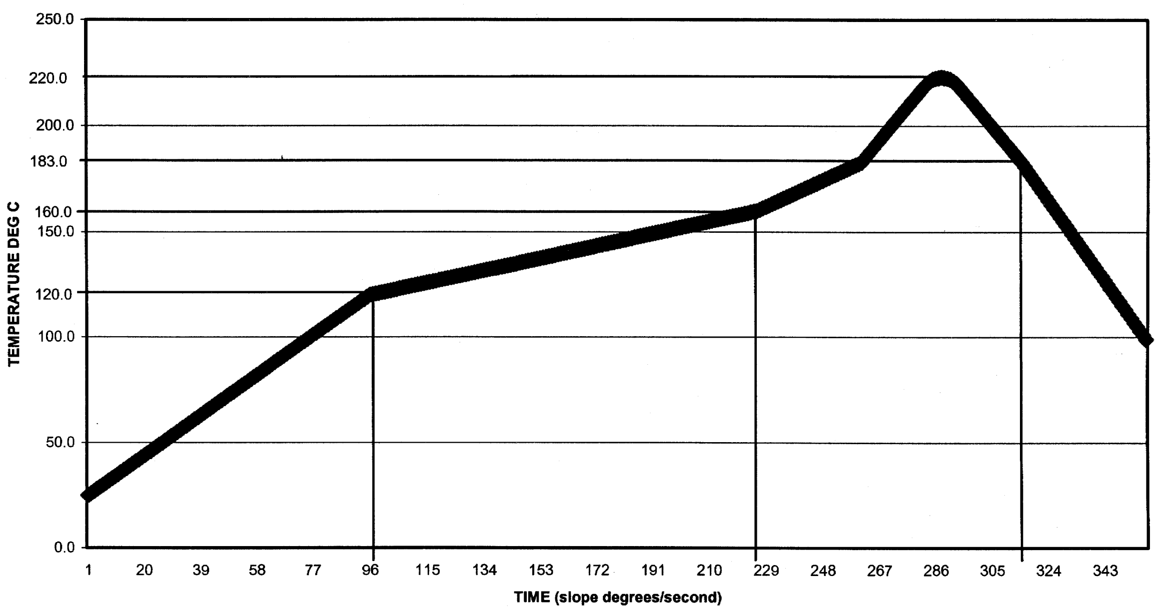
AN101094-8

Reflow and Cleaning

The BGA may be assembled using standard IR or IR convection SMT reflow processes without any special considerations. As with other packages, the thermal profile for specific board locations must be determined. Nitrogen purge is recommended during solder for no-clean fluxes. The BGA is qualified for up to three reflow cycles at 225˚ C peak (J-STD-020). The actual temperature used for the BGA is a function of:

- Board density
- Board geometries
- Component location on the board
- Size of surrounding components
- Component mass
- Furnace loading
- Board finish
- Solder paste types

It is recommended that the temperature profile be set-up at the ball location of the BGA as well as several other locations on the PCB.



AN101094-13

		Convection / IR
Ramp Up °C/sec (Note 2)	Maximum	3 °C/sec
	Recommended	**1 °C/sec**(Note 1)
	Minimum	(Note 1)
Dwell Time 120° C to 160° C (Note 2)	Maximum	(Note 1)
	Recommended	**130 seconds**
	Minimum	(Note 1)
Dwell Time 160° C to 183° C (Note 2)	Maximum	(Note 1)
	Recommended	**35 seconds**
	Minimum	(Note 1)
Dwell Time ≥ 183° C (Note 2)	Maximum	85 seconds
	Recommended	**50 seconds** (Note 1)
	Minimum	(Note 1)
Peak Temperature (Note 2)	Maximum	225° C (Note 1)
	Recommended	**220° C** (Note 1)
	Minimum	
Dwell Time Max. (within 5° C of peak temperature)	Maximum	10 seconds
	Recommended	**5 seconds**
	Minimum	1 second (Note 1)
Ramp Down °C/sec (Note 2)	Maximum	4 °C/sec
	Recommended	**2 °C/sec**
	Minimum	(Note 1)

Note 1: Will vary depending on board density, geometry, package types, PCB finish, and solder paste types.

Note 2: All Temperatures are measured at the solder joint plane.

FIGURE 9. General Reflow Profile Guidelines for PBGA & EBGA

Solder Joint Inspection

After surface mount assembly, transmission X-ray should be used for sample monitoring of the solder attachment process. This identifies defects such as solder bridging, shorts, opens and voids. Note: Voids, up to 25% of the solder joint area, typically do not have an impact on solder joint reliability.

Replacement/ Rework

The quality of the rework is controlled by directing the thermal energy through the component body to solder without over-heating the adjacent components.

Heating should occur in an encapsulated, inert, gas-purged environment where the temperature gradients do not exceed ±5˚ C across the heating zone using a convective bottom side pre-heater to maximize temperature uniformity. Interchangeable nozzles designed with different geometries will accommodate different applications to direct the airflow path NOTE: Industry standard SMT rework systems include these elements.

Removal of the BGA

Removing the BGA from the PCB involves heating the solder joints above the liquidus temperature of eutectic (63Sn-37Pb) solder using a vacuum gas nozzle. Baking the PCB at 125˚ C for 4 hours is recommended PRIOR to any rework. Doing this removes any residual moisture from the system, preventing moisture induced cracking or PCB delamination during the demount process.

A 1.27 mm (50 mil) keep-out zone for adjacent components is recommended for standard rework processing. If the adjacent components are closer than 1.27 mm, custom tools are required for the removal and rework of the package.

It is recommended that the reflow profile used to reflow the BGA be as close to the PCB mount profile as possible. Preheat the PCB area, through the bottom side of the board, to 100˚ C before heating the BGA to ensure a controlled process. Once the liquidus temperature is reached, nozzle vacuum is automatically activated and the component is removed. After removing the package, the pads may be heated with the nozzle to reflow any residual solder, which may be removed using Teflon tipped vacuum wand.

Site Preparation

Once the BGA is removed, the site must be cleaned in preparation for package attachment. The best results are achieved with a low-temperature, blade-style conductive tool matching the footprint area of the BGA in conjunction with a de-soldering braid. No-clean flux is needed throughout the entire rework process. Care must be taken to avoid burn, lift-off, or damage of the PCB attachment area.

Solder Paste Deposition

Because the BGA is a land area type package, solder paste is required to insure proper solder joint formation after rework. A 100 µm thick mini-stencil is recommended to deposit the solder paste patterns prior to replacement of the BGA.

Component Placement

Most BGA rework stations will have a pick and place feature for accurate placement and alignment. Manual pick and place, with only eyeball alignment, is not recommended. It is difficult to achieve consistent placement accuracy.

Component Reflow

It is recommended that the reflow profile used to reflow the BGA be as close to the PCB mount profile as possible. Preheat the PCB area, through the bottom side of the board, to 100˚ C before heating the BGA to ensure a controlled process. Once the liquidus temperature is reached, the solder will reflow and the BGA will self align. NOTE: Insure the replacement BGA meets the Moisture Sensitivity Level requirements to prevent moisture-induced problems.

APPENDICES

Solder Joint Reliability

Board level reliability has been demonstrated with the PBGA under various conditions. For 361 ball PBGA with 1.27 mm pitch, initial failure occurs at 2500 temperature cycles from −40 to 100˚ C (10˚ C/minute ramp and 16 minute dwell). The table lists the number of cycles to the first failure and 50% failures with various package and board designs. The use of NSMD on both the package and the board clearly has an advantage for promoting board reliability. See *Table 7*.

TABLE 7. Board Reliability Results for 361 ball PBGA, −40 to 100˚ C temperature cycle.

Package			Board		Cycles to Falure	
	Pad Type	Pad Opening (mm)	Pad Type	Pad Opening (mm)	First	50%
1	NSMD	0.51	NSMD	0.51	5500	6000
2	SMD	0.56	SMD	0.56	2500	2900
3	SMD	0.56	NSMD	0.51	2100	2800
4	NSMD	0.51	SMD	0.56	3200	4000

National Semiconductor Corporation
Americas
Tel: 1-800-272-9959
Fax: 1-800-737-7018
Email: support@nsc.com
www.national.com

National Semiconductor Europe
Fax: +49 (0) 180-530 85 86
Email: europe.support@nsc.com
Deutsch Tel: +49 (0) 69 9508 6208
English Tel: +44 (0) 870 24 0 2171
Français Tel: +33 (0) 1 41 91 8790

National Semiconductor Asia Pacific Customer Response Group
Tel: 65-2544466
Fax: 65-2504466
Email: ap.support@nsc.com

National Semiconductor Japan Ltd.
Tel: 81-3-5639-7560
Fax: 81-3-5639-7507