intel®

Ball Grid Array (BGA) Packaging 14

14.1 Introduction

The plastic ball grid array (PBGA) has become one of the most popular packaging alternatives for high I/O devices in the industry. Its advantages over other high leadcount (greater than ~208 leads) packages are many. Having no leads to bend, the PBGA has greatly reduced coplanarity problems and minimized handling issues. During reflow the solder balls are self-centering (up to 50% off the pad), thus reducing placement problems during surface mount. Normally, because of the larger ball pitch (typically 1.27 mm) of a BGA over a QFP or PQFP, the overall package and board assembly yields can be better. From a performance perspective, the thermal and electrical characteristics can be better than that of conventional QFPs or PQFPs. The PBGA has an improved design-to-production cycle time and can also be used in few-chip-package (FCPs) and multi-chip modules (MCMs) configurations. BGAs are available in a variety of types, ranging from plastic overmolded BGAs called PBGAs, to flex tape BGAs (TBGAs), high thermal metal top BGAs with low profiles (HL-PBGAs), and high thermal BGAs (H-PBGAs).

The H-PBGA family includes Intel's latest packaging technology - the Flip Chip (FC)-style, H-PBGA. The FC-style, H-PBGA component uses a Controlled Collapse Chip Connect die packaged in an Organic Land Grid Array (OLGA) substrate. In addition to the typical advantages of PBGA packages, the FC-style H-PBGA provides multiple, low-inductance connections from chip to package, as well as, die size and cost benefits. By providing multiple, low-inductance connections the FC-style, HPBGA offers equivalent or better performance than an extra on-chip metal layer. The FC technology also provides die-size benefits through the elimination of the bond pad ring and better power bussing and metal utilization. The OLGA substrate results in a smaller package, since there is no cavity, and thermal management benefits since the thermal solution can directly contact the die.

14.2 Package Attributes

Table 14-1. PBGA Package Attributes

	PBGA										
Lead Count	196 (15mm)	208 (23mm)	241 (23mm)	256 (17mm)	256 (27mm)	304 (31mm)	324 (27mm)	421 (31mm)	468 (35mm)	492 (35mm)	544 (35mm)
Sq/Rect.	S	S	S	S	S	S	S	S	S	S	S
Pitch (mm)	1.0	1.27	1.27	1.0	1.27	1.27	1.27	1.27	1.27	1.27	1.27
Package Thickness (mm)	1.61	2.33	2.38	1.56	2.13	2.33	2.13	2.38	2.38	2.38	2.38
Weight (gm)	.67	1.56		.70		3.46	2.86	3.87	5.06		
Max. Footprint (mm)	15.20	23.20	23.20	17.20	27.20	31.20	27.20	31.20	35.20	35.20	35.20
Shipping Media:											
Tape & Reel	X	X	X				X	X	X	X	X
Trays	X	X	X	X	X	X	X	X	X	X	X
Desiccant Pack	X	X	X	X	X	X	X	X	X	X	X
Comments/ Footnotes											

Table 14-2. H-PBGA/HL-PBGA Package Attributes

	H-PBGA			HL-PBGA		
Lead Count	495	540	615	304	352	432
Sq/Rect.	R	S	R	S	S	S
Pitch (mm)	1.27	1.27	1.27	1.27	1.27	1.27
Package Thickness (mm)	2.54	2.13	2.54	1.54	1.54	1.54
Weight (gm)	4.52	15.25	4.11		8.90	
Max. Footprint (mm)	27.2 x 31.0	43.0	31.0 x 35.0	31.10	35.10	40.10
Shipping Media:						
Tape & Reel		X			X	
Trays	X	X	X	X	X	X
Desiccant Pack	X	X	X	X	X	X
Comments/ Footnotes	Can be thermally enhanced with heat sinks	Can be thermally enhanced with heat sinks	Can be thermally enhanced with heat sinks		Can be thermally enhanced with heat sinks	

14.3 Package Materials

The PBGA package consists of a wire-bonded die on a substrate made of a two-metal layer copper

clad bismaleimide triazine (BT) laminate. Four-metal layer substrate designs generally contain additional power and/or ground planes to improve electrical and thermal performance. The die and bonds are protected and encapsulated with molding compound. Via holes drilled through the substrate provide routing from the lead fingers to the respective eutectic (63/37 Sn/Pb) solder balls on the underside. Thermal performance can be enhanced by adding heatsink fastened through mechanical means using thermal grease or by using conductive epoxy.

The H-PBGA and HL-PBGA, however, are configured differently to provide for greater thermal and if required, electrical performance. The thermal advantage provided by this design is based first upon attaching the die to the bottom surface of a heatspeader or slug that also forms the topside of the package. Secondly, because the copper heatspreader forms the top of the package, the thermal resistance is extremely low and exposes the package surface to available air flow. If required, this heatslug can be directly coupled to active or passive thermal management devices such as heat sinks or heat pipes. Improved electrical performance is achieved through additional power and/or ground planes.

The FC-style, H-PBGA package consists of a die reflowed onto an oraganic substrate. The substrate consists of four to ten layers of copper with insulating materials in between. The copper layers are connected by vias. BT (Bismaleimide Triazine) resin reinforced with glass fiber forms the core of the organic substrate. Solder bumps (3% Sn, 97% Pb) on the die surface are joined with solder pads (60% Sn, 40% Pb) on the organic substrate in a reflow furnace. These joints form the electrical/mechanical connection between the FC die and the OLGA package. An epoxy underfill fills the gap between die and the substrate. This underfill provides mechanical support and protection for the die-to-package interconnects and also minimizes thermal stress on the die due to CTE (coefficient of thermal expansion) mismatch with the substrate materials. The die backside is exposed allowing the thermal solutions and thermal interface material to have direct contact with the die surface.

See Figure 14-1, Figure 14-2, and Figure 14-3 for description of PBGA, HL-PBGA, and FC-style H-PBGA packages.

Figure 14-1. PBGA Die Up Cross-Section

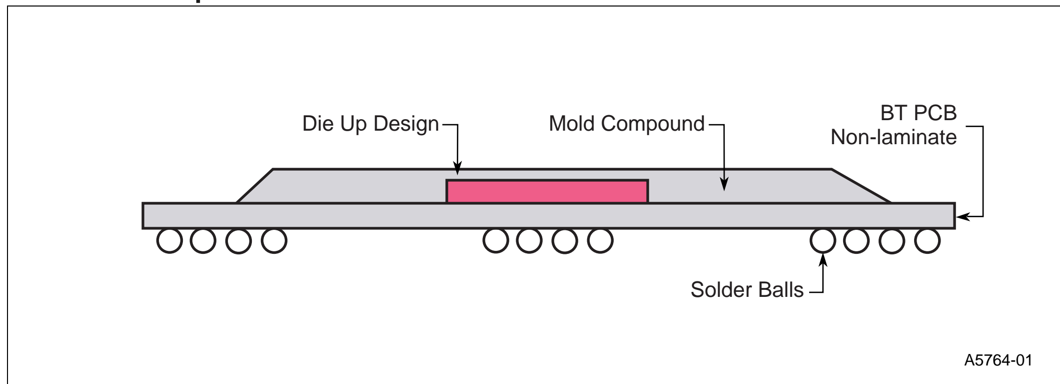


Figure 14-2. HL-PBGA Die Down Cross-Section

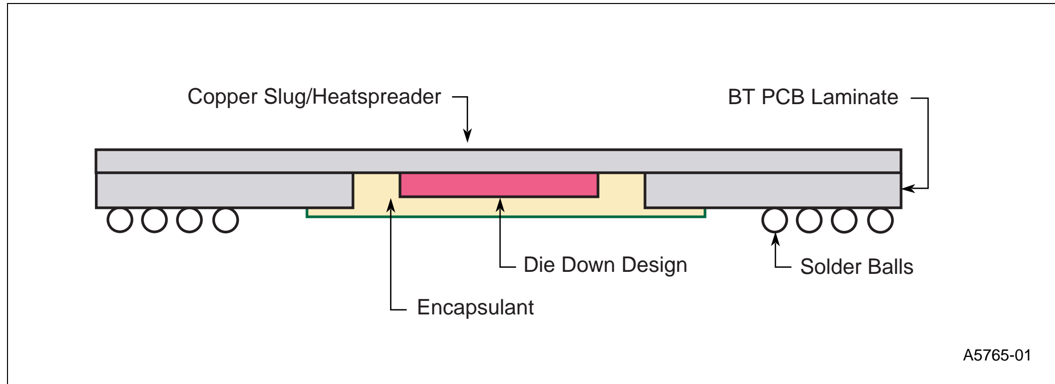
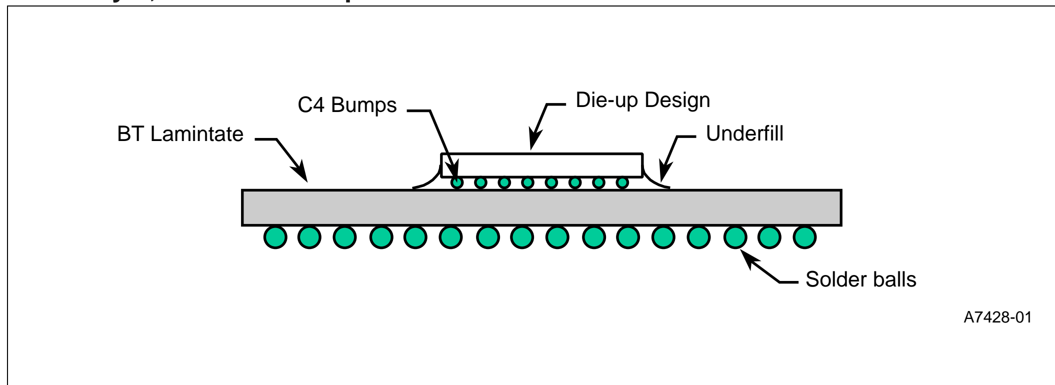


Figure 14-3. FC-Style, H-PBGA Die Up Cross-Section



14.4 Package Dimensions

Table 14-3. Plastic Ball Grid Array Family Attributes

Package Family Attributes	
Category	Plastic Ball Grid Array
Acronym	PBGA, HL-PBGA, H-PBGA
Ball Counts	PBGA: 196, 208, 241, 256, 304, 324, 421, 468, 492, 544. HL-PBGA: 352, 304, 432. H-PBGA: 540. FC-style, H-PBGA: 495, 615.
Ball Material	Solder (63/37)
Ball Pitch	1.0, 1.27 mm
Board Assembly Type	Surface Mount

Table 14-4. Symbol List for Plastic Ball Grid Array Family

Letter or Symbol	Description of Dimensions
A	Overall Height
A_1	Stand Off
A_2	Encapsulant Height
A_3	Die Height with FC Bumps and Underfill
b	Ball Diameter
c	Substrate Thickness
D	Package Body Length
D_1	Encapsulant Length
E	Package Body Width
E_1	Encapsulant Width
F_1	Die Width
F_1	Die Length
e	Ball Pitch
N	Ball Count i.e. Lead Count
S_1	Outer Ball Center to Short Edge of Body
S_2	Outer Ball Center to Long Edge of Body

NOTE:
1. Controlling Dimensions: Millimeter

Figure 14-4. PBGA Package Ball Array Configuration

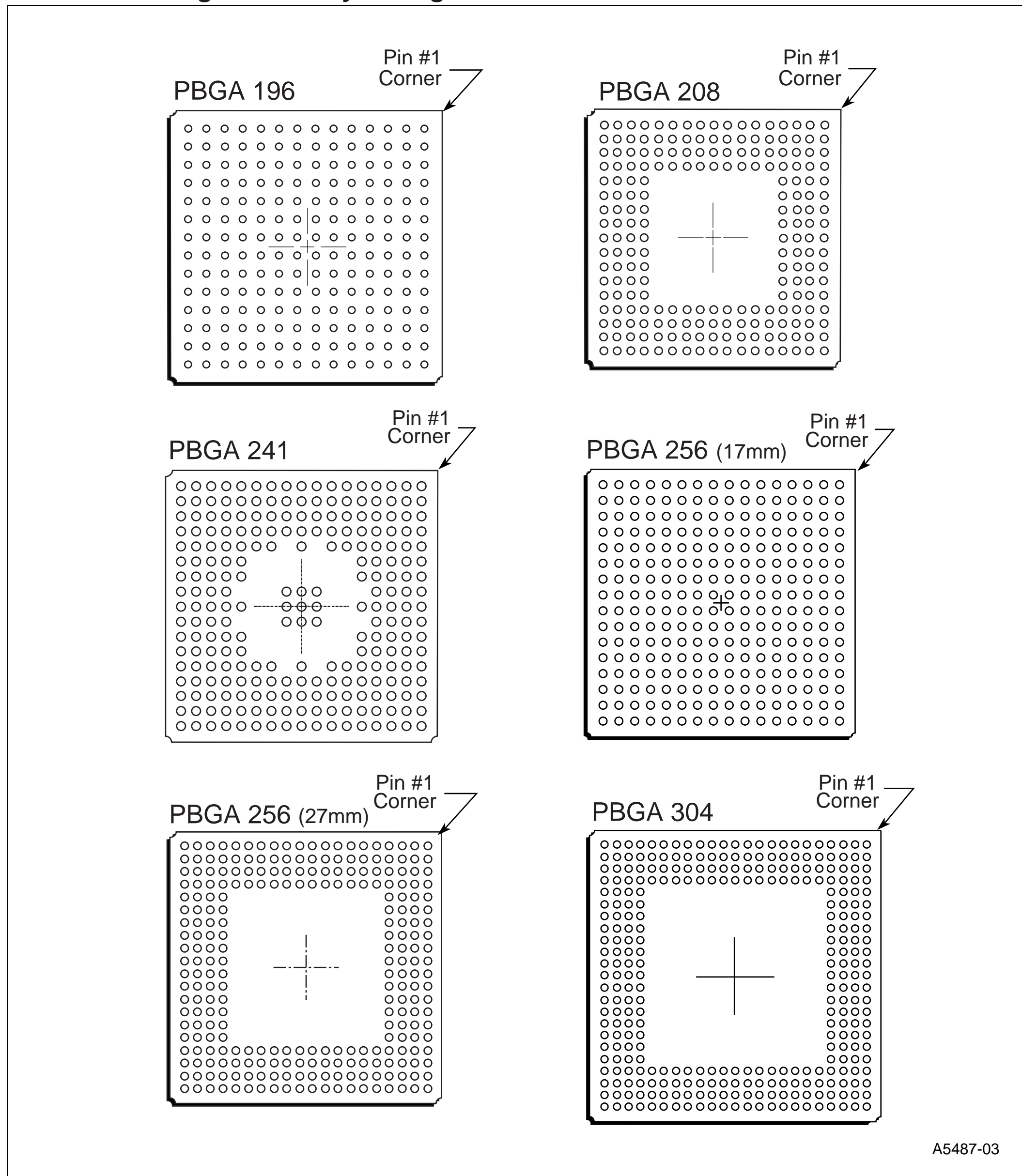


Figure 14-5. PBGA Package Ball Array Configuration Continued

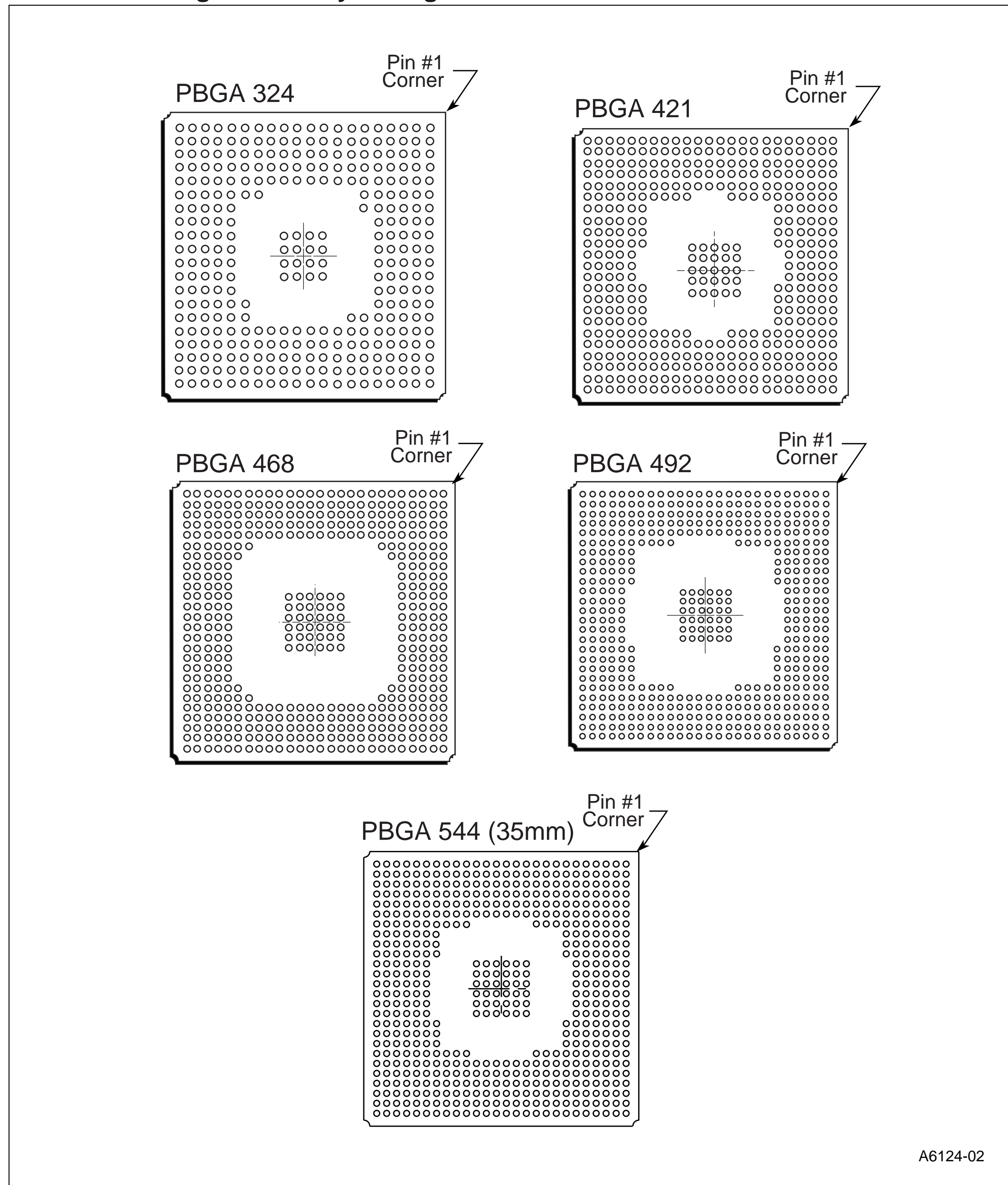


Figure 14-6. 15mm PBGA Outline Drawing

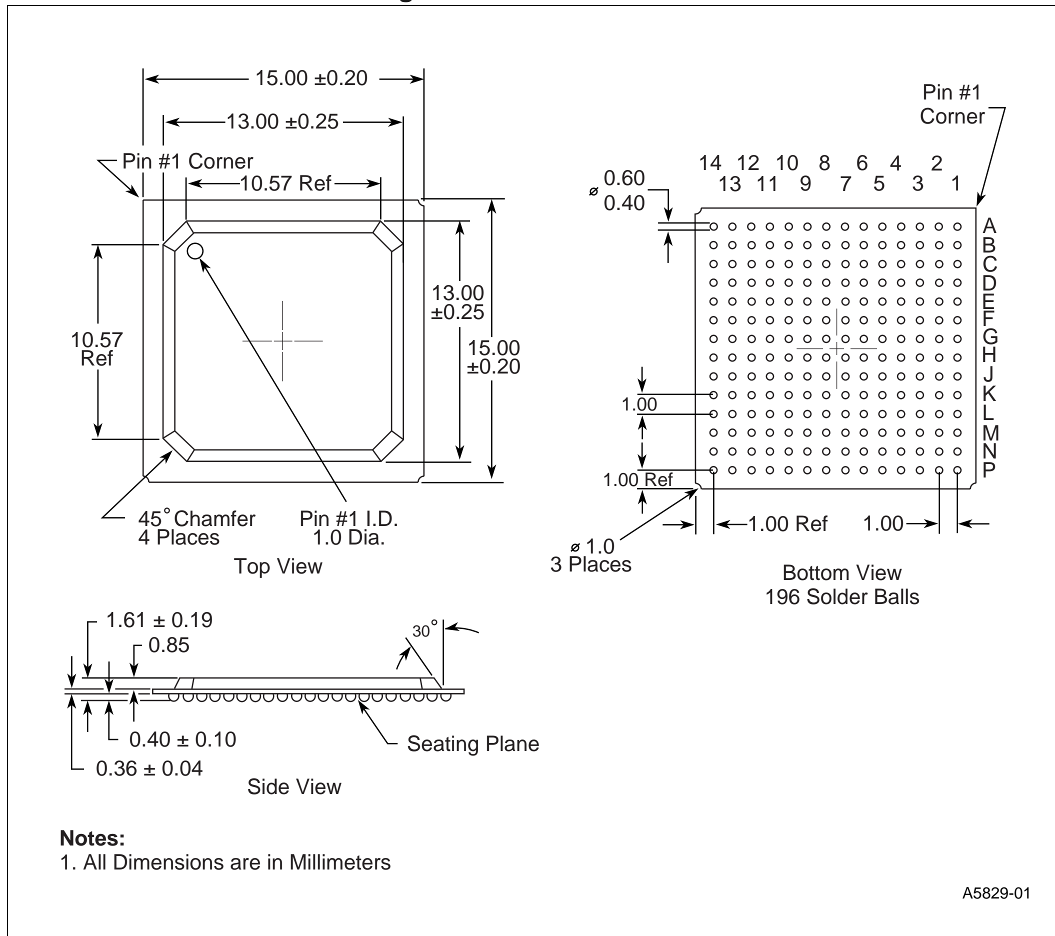


Figure 14-7. PBGA Outline Drawing

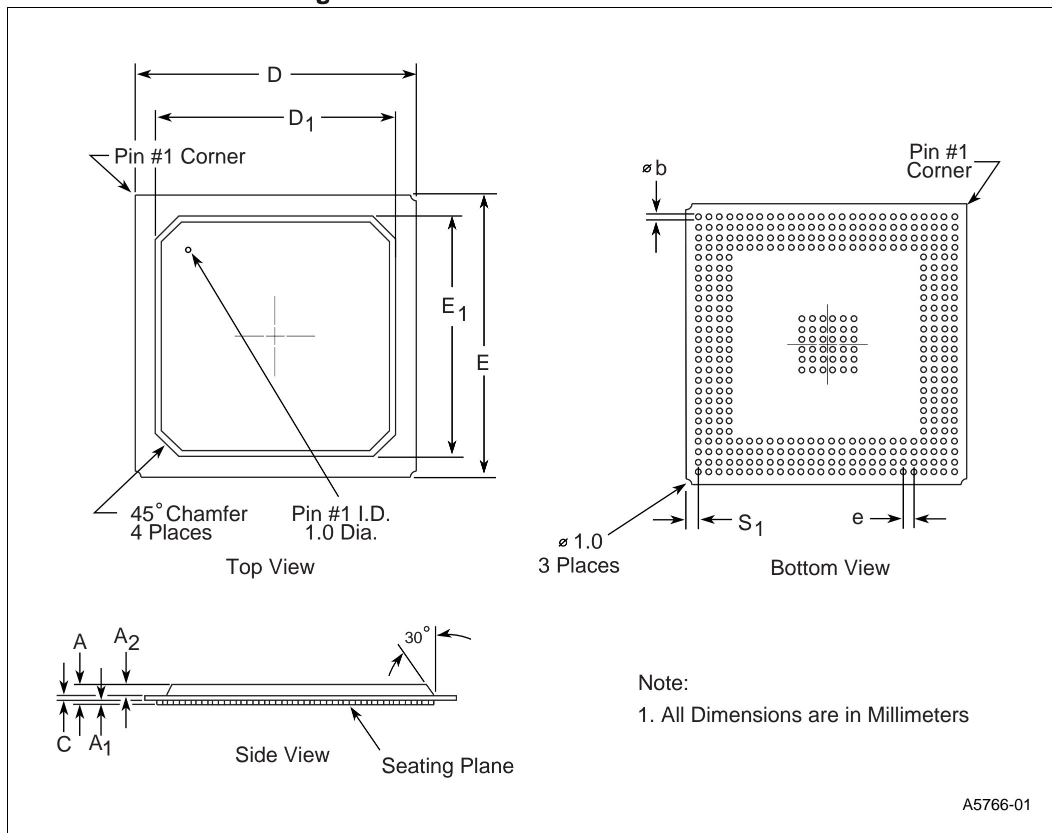


Table 14-5. PBGA Package Family Dimensions

PBGA Package Dimensions												
	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max
N	196		208		241		256 (17mm)		256 (27mm)		304	
A	1.37	1.75	1.94	2.32	2.17	2.59	1.37	1.75	1.94	2.32	2.12	2.54
A_1	.30	.50	0.50	0.70	0.50	0.70	0.30	0.50	0.50	0.70	0.50	0.70
A_2	.80	.90	1.12	1.22	1.12	1.22	0.75	0.85	1.12	1.22	1.12	1.22
D	14.80	15.20	22.80	23.20	22.80	23.20	16.80	17.20	26.80	27.20	30.80	31.20
D_1	12.75	13.25	19.25	19.75	19.25	19.75	14.75	15.25	23.75	24.25	25.50	26.70
S_1	1.00 REF		1.34 REF		1.34 REF		1.00 REF		1.44 REF		1.53 REF	
b	.40	.60	0.60	0.90	0.60	0.90	0.40	0.60	0.60	0.90	0.60	0.90
c (2L/4L)	.32/.52	.40/.60	.32/.52	.40/.60	.32/.55	.40/.67	.32/.52	.40/.60	.32/.52	.40/.60	.52	.60
e	1.0		1.27		1.27		1.0		1.27		1.27	

Table 14-6. PBGA Package Family Dimensions Continued

PBGA Package Dimensions										
	Min	**Max**	**Min**	**Max**	**Min**	**Max**	**Min**	**Max**	**Min**	**Max**
N	324		421		468		492		544	
A	1.94	2.32	2.17	2.59	2.17	2.59	2.14	2.52	2.14	2.52
A_1	0.50	0.70	0.50	0.70	0.50	0.70	0.50	0.70	0.50	0.70
A_2	1.12	1.22	1.12	1.22	1.12	1.22	1.12	1.22	1.12	1.22
D	26.80	27.20	30.80	31.20	34.80	35.20	34.80	35.20	34.80	35.20
D_1	23.75	24.25	25.75	26.25	29.75	30.25	29.75	30.25	29.75	30.25
S_1	1.44 REF		1.53 REF		1.63 REF		1.63 REF		1.63 REF	
b	0.60	0.90	0.60	0.90	0.60	0.90	0.60	0.90	0.60	0.90
c (2L/4L)	.32/.55	.40/.67	.52/.55	.60/.67	.52/.55	.60/.67	.52/.55	.60/.67	.55	.67
e	1.27		1.27		1.27		1.27		1.27	

Figure 14-8. HL-PBGA/H-PBGA Package Ball Array Configuration

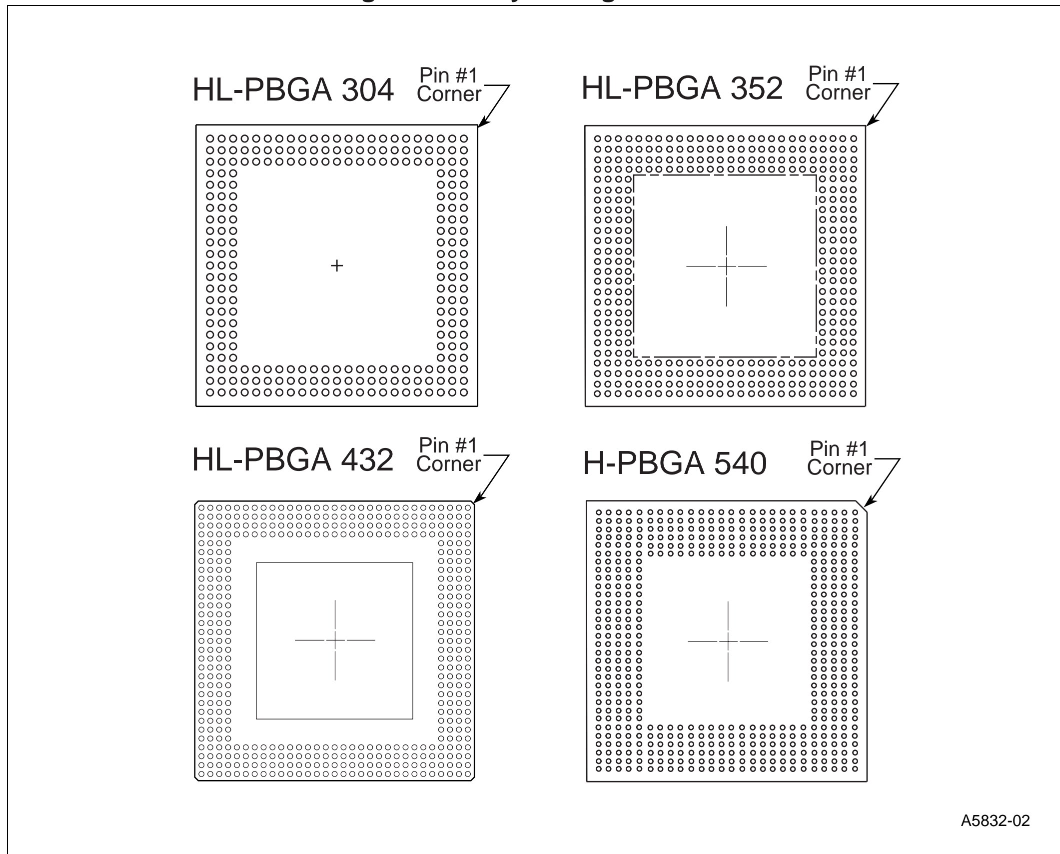


Figure 14-9. HL-PBGA Package Outline and Ball Array Configuration

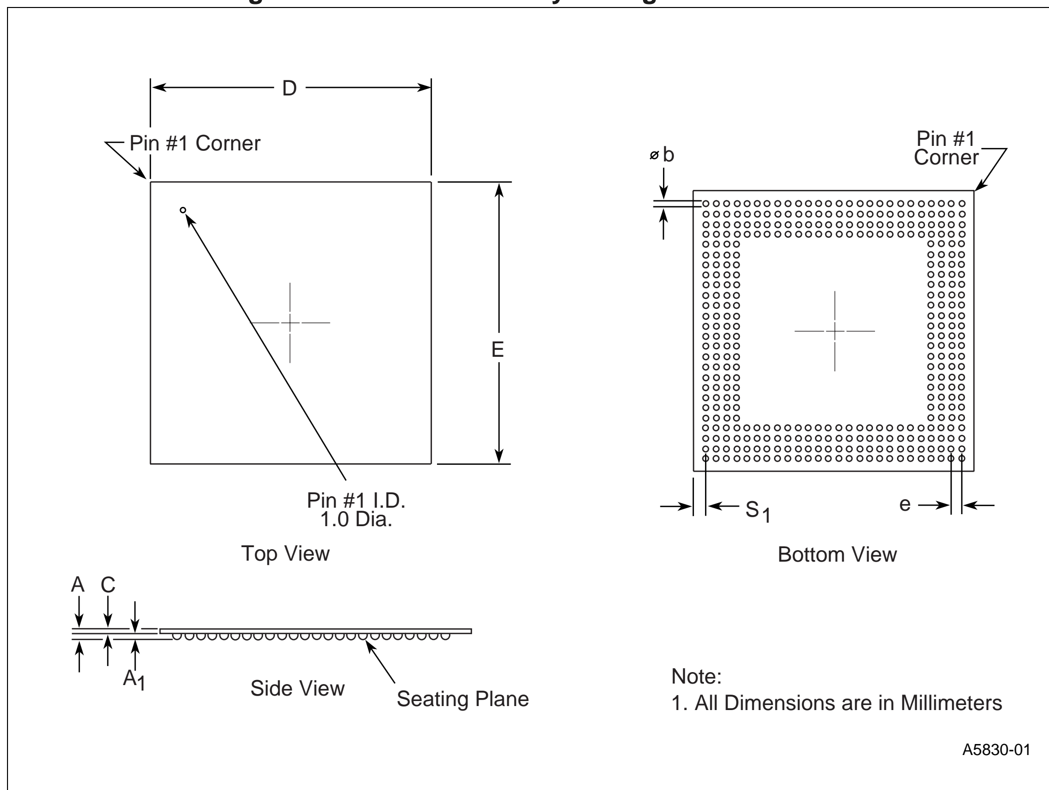


Table 14-7. HL-PBGA Dimensions

	304		352 LD		432	
Symbol	**Min**	**Max**	**Min**	**Max**	**Min**	**Max**
A	1.41	1.67	1.41	1.67	1.41	1.67
A_1	0.56	0.70	0.56	0.70	0.56	0.70
b	0.60	0.90	0.60	0.90	0.60	0.90
c	0.95	0.97	0.85	0.97	0.85	0.97
D	30.90	31.10	34.90	35.10	39.90	40.10
E	30.90	31.10	34.90	35.10	39.90	40.10
e	1.27		1.27		1.27	
S_1	1.53 REF		1.63 REF		0.95 REF	

Figure 14-10. H-PBGA with 25.4 mm square Slug Outline Drawing

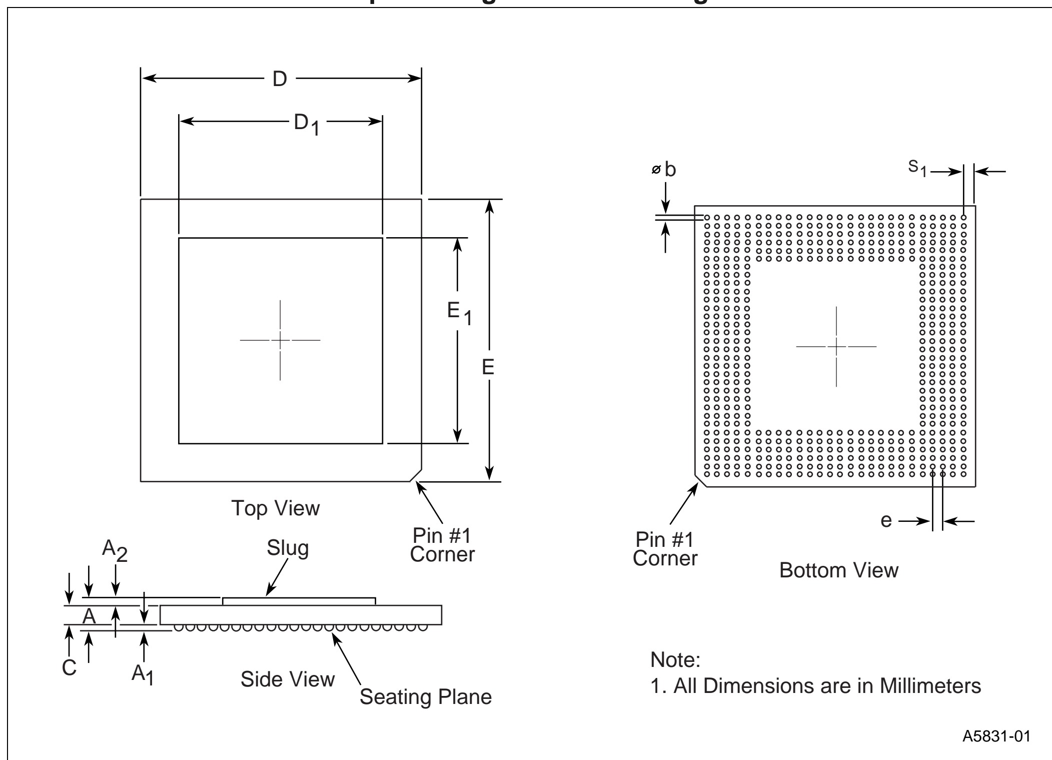


Table 14-8. H-PBGA Package Dimensions

	540	
Symbol	**Min**	**Max**
A	3.59	4.10
A_1	0.40	0.70
A_2	0.95	1.10
b	0.60	0.90
c	2.00	2.30
D	42.30	42.70
D_1	-	27.70
E	42.30	42.70
E_1	-	27.70
e	1.27	
S_1	1.56 REF	
NOTE: Measurement in millimeters		

Figure 14-11. FC-Style H-PBGA Package Ball Array Configurations

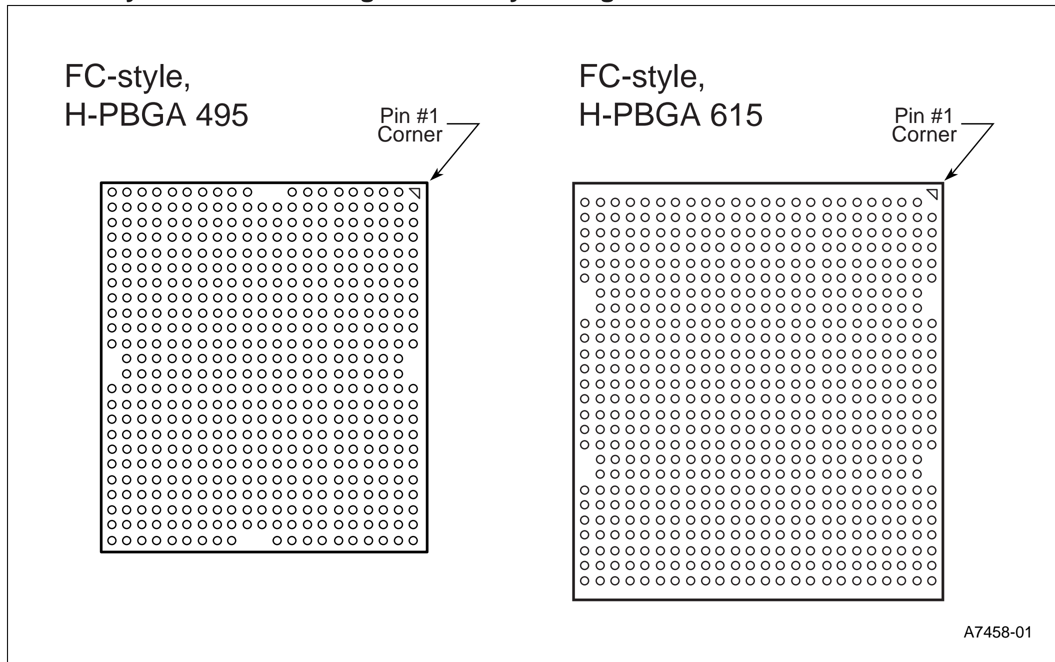


Figure 14-12. FC-style, H-PBGA 495 Outline Drawing

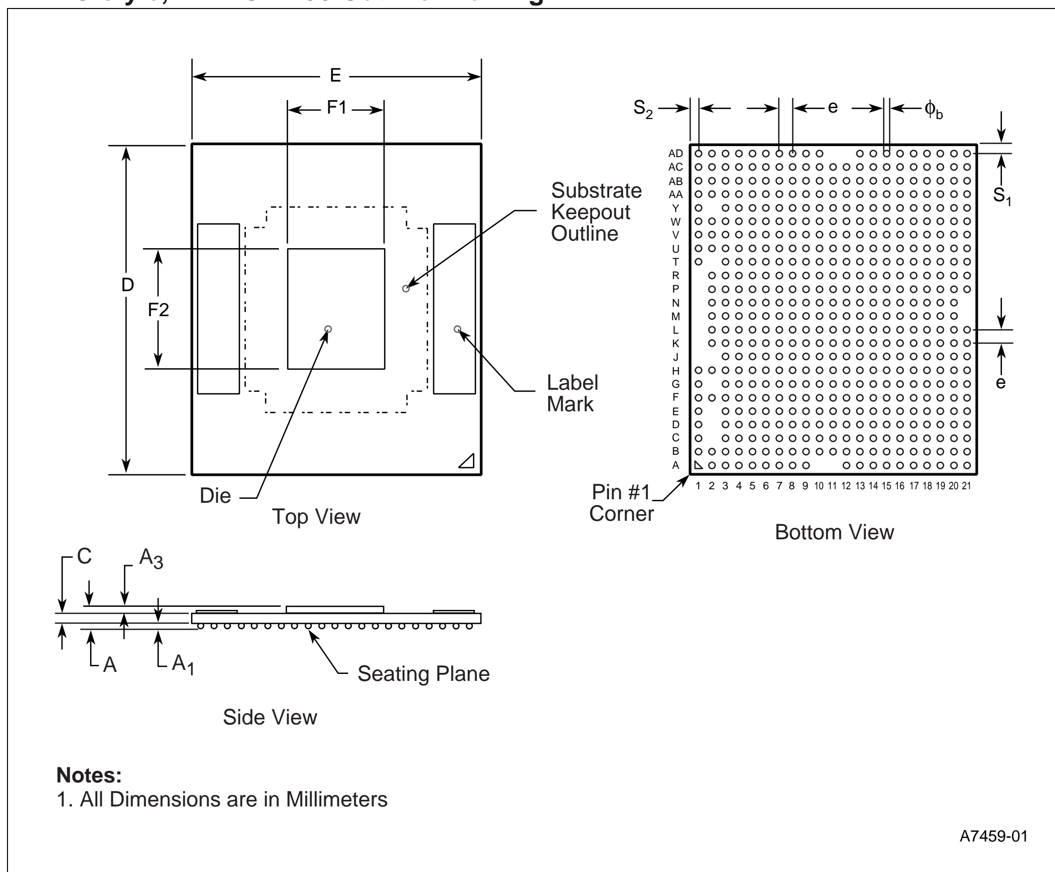


Figure 14-13. FC-style, H-PBGA 615 Outline Drawing

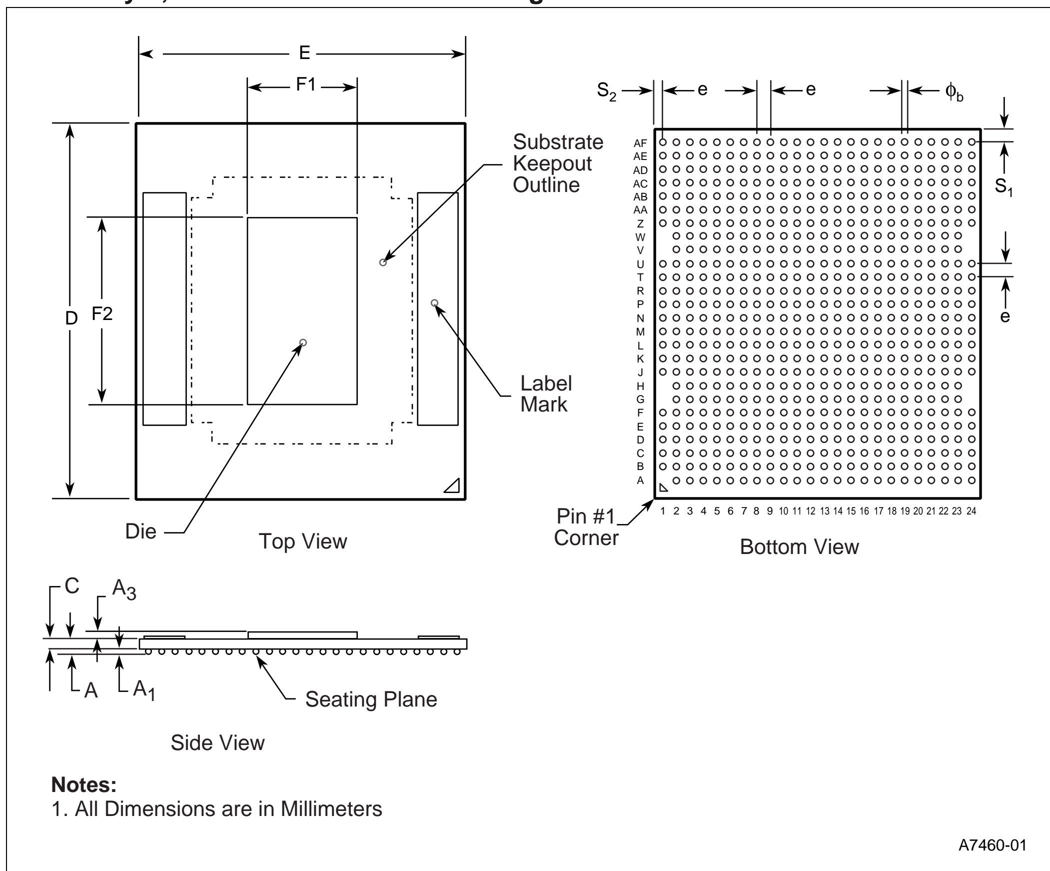


Table 14-9. FC-style, H-PBGA Package Dimensions

	495		615	
Symbol	**Min**	**Max**	**Min**	**Max**
A	2.29	2.79	2.29	2.79
A1	0.5	0.7	0.5	0.7
A3	0.854		0.854	
b	0.60	0.90	0.60	0.90
c	0.93	1.07	1.00	1.20
D	30.9	31.1	34.9	35.1
E	27.0	27.3	30.9	31.1
F1	9.2		10.3	
F2	11.2		17.4	
e	1.27		1.27	
S1	0.895		1.625	
S2	0.900		0.895	
NOTE: Measurement in millimeters				

14.5 Package Usage

BGA packaging can be used for high-performance applications with high thermal and electrical requirements. BGAs fit ICs into a smaller footprint, decreasing pitch spacing, by utilizing an array of solder ball connections. This allows for a higher density of I/O connections than that of conventional QFPs or PGAs. The result is a considerably smaller finished package size.

In general, the ball grid array features shorter electrical path lengths which reduce inductance. Mechanical problems such as fragile leads are absent. The larger spacing between solder lands provide adequate tolerances for more reliable surface mounting. Some heat dissipation can be facilitated through the substrate. These characteristics make the ball grid array package suitable for a wide variety of devices: microprocessors/microcontrollers, ASICs, memory, PC chip sets, and other products. A thin profile and smaller footprint make the BGA an attractive option when board space is a major concern. Small body size BGA packages come close to chip scale package size for use in space constrained applications.

The FC-Style, H-PBGA also allows for Voltage Indentification through an open circuit or a short to Vss on the processer substrate. These opens or shorts are achieved by selectively depopulating some of the balls. Voltage Identification can be used to support automatic selection of power supply voltages.

14.6 Handling: Shipping Media

14.6.1 Mid-temperature Thin Matrix Tray

The BGA packages are shipped in either a tape and reel or a mid-temperature thin matrix tray that complies to the JEDEC standards. Typically, JEDEC trays have the same ‘x’ and ‘y’ outer dimensions and are easily stacked for storage and manufacturing. For tray dimensions please refer to Chapter 10 of this data book. The JEDEC style shipping trays are returnable to Intel for reuse. Chapter 10 contains detailed information on the return addresses for the different types of shipping trays. Intel will pay all shipping costs associated with the return.

14.6.2 Tape and Reel

Tape and reel handling is engineered to contain and protect surface mount components in embossed semi-conductive PVC or polystyrene carrier tapes to aid the high speed board mounting operations found in many high volume board operations. The BGA packages are shipped from Intel in a carrier tape made of antistatic treated plastic. It offers exceptional strength and stability over extended time and wide temperature variations, while at the same time maintaining flexibility for use in automated equipment. The cover tape used is heat sealable, transparent, and antistatic. The loaded carrier tapes will be wound onto a plastic reel. The carrier tape dimensions meet the EIA standards. The tape and reel packaging standards offered by Intel for many of the PBGA/HL-PBGA packages meet the EIA standards, ie, EIA 481-1, 481-2, and 481-3. However, there are some products shipped from Intel in tape and reel that have a package orientation in the tape that is different from the EIA standards. It is advisable that the user of Intel BGA products obtain a product data sheet that shows the tape and reel shipping details to insure the correct cavity orientation is understood.

14.7 Moisture Sensitivity

Most PBGA components are highly sensitive to moisture exposure before the reflow temperature

exposure. Maintaining proper control of moisture uptake in components is critical to the prevention of "popcorning" of the package body or encapsulation material. BGA components, before shipping, are baked dry and enclosed in a sealed desiccant bag with a desiccant pouch and a humidity indicator card. Most BGA components are classified as a level 3 or level 4 for moisture sensitivity as per the IPC/JEDEC Spec J-STD-020, "Moisture/Reflow Sensitivity Calculation of Plastic Surface Mount Devices."

With most surface mount components, if the units are allowed to absorb moisture beyond their out of bag times for their moisture rating, damage may occur during the reflow process. Chapter 8 of this data book provides an in-depth view of package preconditioning methods and moisture sensitivity requirements. Please refer to Chapter 8 for more information regarding how moisture sensitive components are classified.

Prior to opening the shipping bag and attempting solder reflow, the moisture sensitivity of the packages being used should be understood so proper precautions can be taken to insure that a minimal out of bag time is maintained. This will insure that the highest possible package reliability is achieved for the final product. If previously bagged product cannot be mounted before the elapsed out of bag time for that product, the parts can be rebaked as per Chapter 8. Another option is to store the opened units in a nitrogen cabinet or dry box until needed. Placing units in a dry box effectively 'stops the clock.'

It should be understood that packages continue to gain moisture even after board mounting. Components that need to be reworked must be completely processed throught all thermal exposures before the original out of bag limits are reached. If this is not possible, or the time allotment is not ridgely followed, bake-out of the completed boards must be accompliished before subjecting the components to the heat of the rework process. Products being removed from boards that have been returned from the field for failure analysis, must be baked dry before heat expousure. If this step is skipped, massive damage to the component will result, rendering useless any further efforts at determining the cause of failure.

14.8 Designing Boards For BGAs

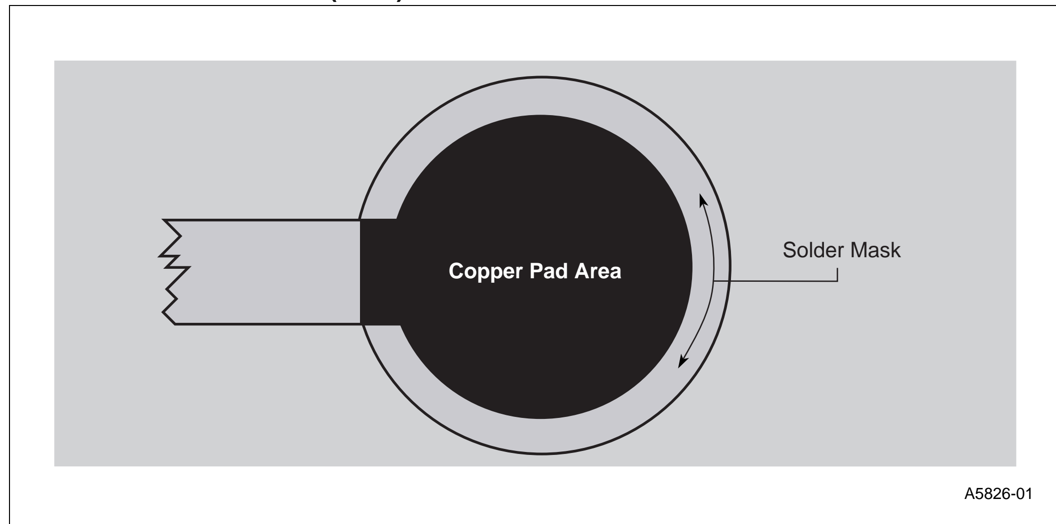
Most BGA packages use Solder Mask Defined pads on the package side of the solder ball. PCB pad size is typically close to or identical to the package pad size. This provides for balanced stress during thermal cycling, which helps to maximize fatigue life.

14.8.1 Solder Mask Defined (SMD) Pad

In the solder mask defined (SMD) pad shown in Figure 14-14, the copper for the pad area is larger than the desired land size. The opening in the solder mask is made smaller than the copper land, thus defining the mounting pad. A couple of points to consider with solder mask defined pad are:

- There is an advantage in that the overlap of the solder mask onto the copper enhances the copper adhesion to the laminate surface. When using resin systems where adhesion is low, this is an important consideration.
- One disadvanatage of SMD pads is that the fatigue life has shown to be lower then NSMD pads through long term reliaiblity testing. Because of this issue, the solder mask angle at the pad edge has been thinned on many new package designs to minimize the mask impingment on the solder ball.

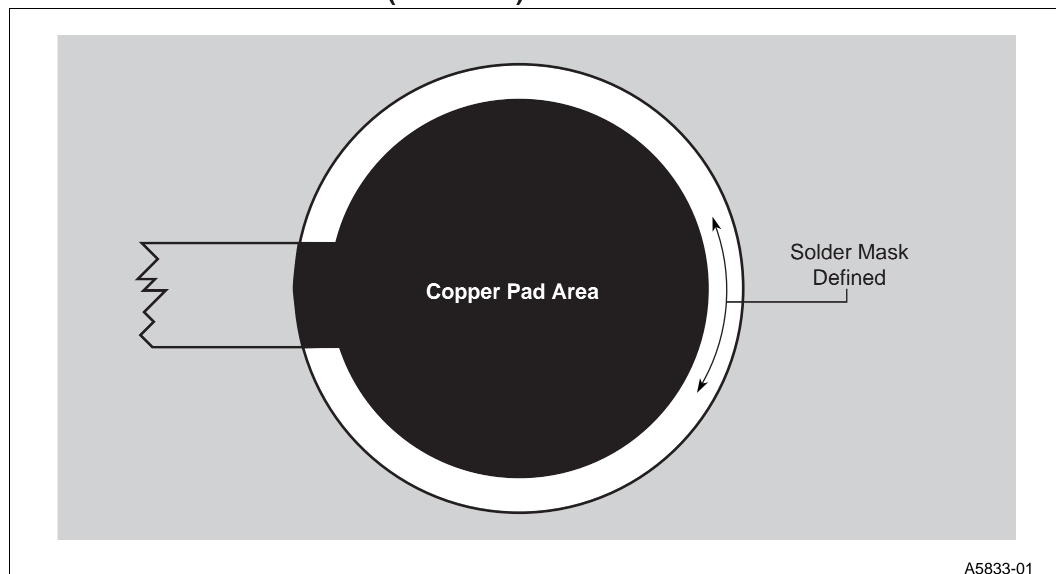
Figure 14-14. Solder Mask Defined (SMD) Pad



14.8.2 Non-Solder Mask (i.e. Metal or Copper) Defined Pad

The non-solder mask (sometimes called metal or copper) defined pad Figure 14-15, has a solder mask opening larger than the copper area. Pad size is controlled by the copper etch quality control. This is generally less accurate than the solder mask photo image control. Non-solder mask pad size varies more than with the SMD pad. However, because the edges of the copper do not need to extend under the solder mask, the pad can be either made larger, or provided more line routing space between pads. Pattern registration is also as accurate as the copper artwork, which is generally much more accurate than the solder mask pattern. Vision registration on copper fiducials (reference points) will give exact location of the site. With SMD pads, the misrepresentation error of the solder mask will also shift the location of the entire site relative to vision fiducials.

Figure 14-15. Non Solder Mask Defined (Non-SMD) Pad



14.8.3 BGA Package Considerations

The following subsections address the BGA package layout and includes guidelines for pad size, vias and routing. It is important to implement a keep-out zone around BGAs for rework purposes. The keep-out zone distance is determined by the type of rework equipment to be used. See Section 14.9.8 for more information.

14.8.3.1 Routing and Pad Size

Many perimeter style BGA package typically contains four or five rows of solder balls; as such, it is possible to route one or two traces between pads to route signals in a four-layer board.

- When routing one trace between pads, the manufacturers preferred line width and spacing technology becomes a limiting factor.
- When routing two traces between pads, 5 mil traces and 5 mil spaces are required for a 24 mil pad size. For a 20 mil pad size, 6 mil traces and 6 mil spacing can be used. Figure 14-15 shows a routing example for 20 mil ball pads. Either pad size is acceptable, so the decision is primarily determined by the manufacturers preferred line width and spacing technology.
- When larger trace widths are desired, another alternative is to route 5/5 or 6/6 (mil space/mil trace) within the BGA pads, and then neck up to the larger trace widths once you have cleared the BGA component

Using the routing scheme shown in Figure 14-16, the first two ball rows are routed on the top signal layer and the inner two rows are routed on the bottom side of the package substrate. In this case vias are required between the BGA pads. Using the routing scheme shown in Figure 14-10, the first three ball rows are routed on the top signal layer and the inner row is routed on the boards bottom side. In this case the vias are not required in between the BGA pads. Vias are discussed in Section 14.8.3.3.

Figure 14-16. Board Side BGA Routing Example One

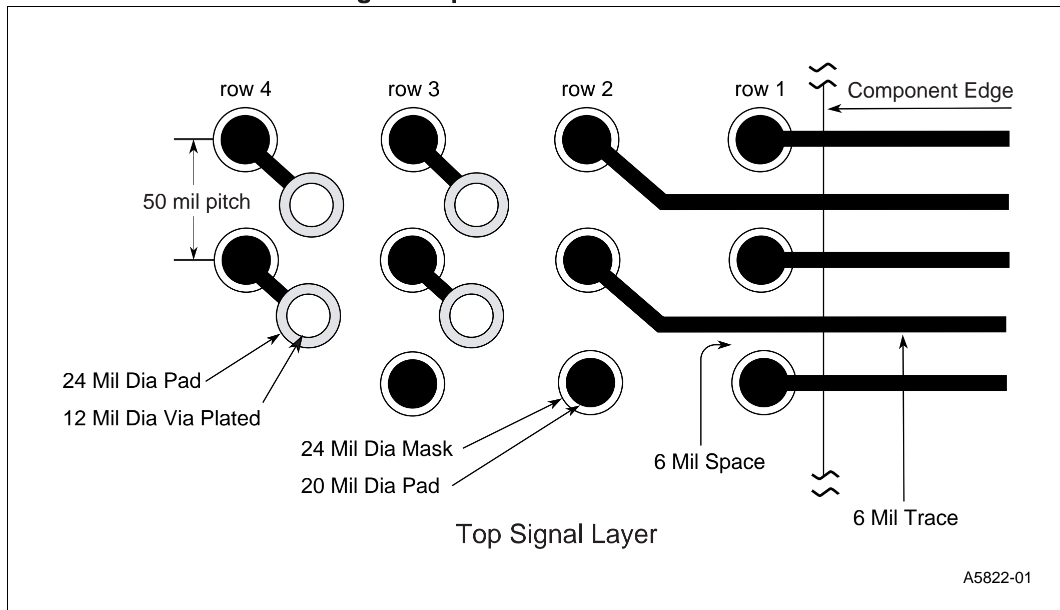
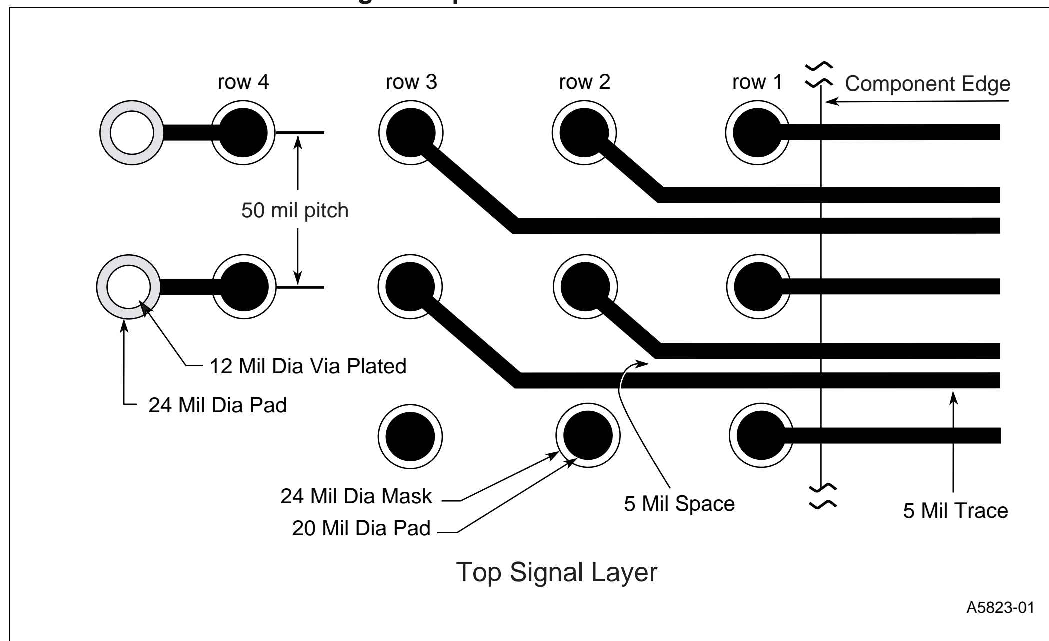


Figure 14-17. Board Side BGA Routing Example Two



14.8.3.2 Bottom Layer Routing

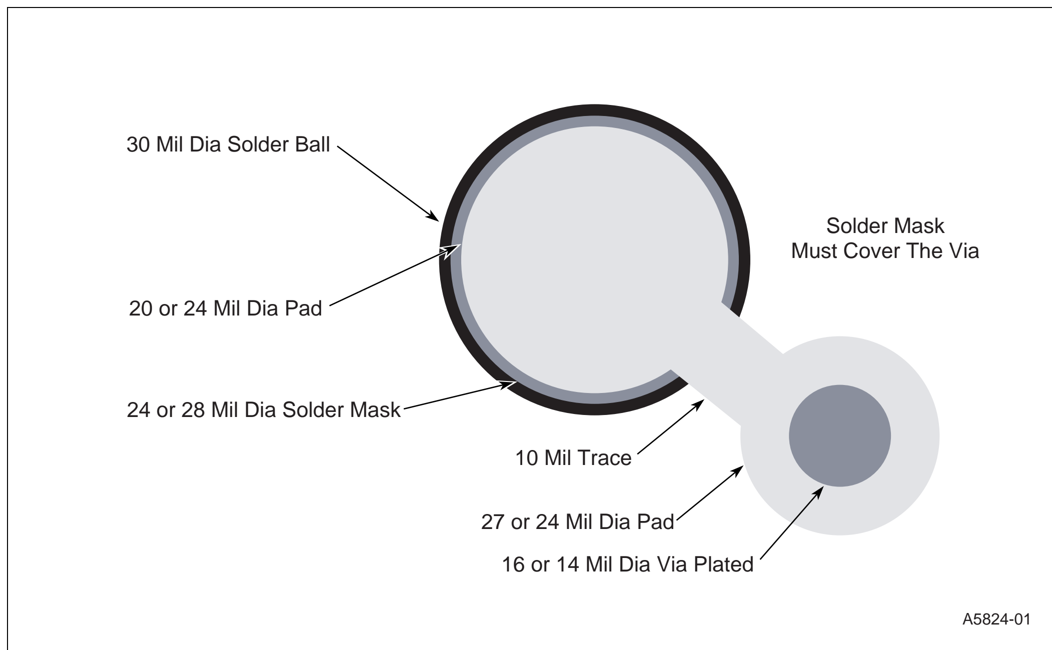
Figure 14-16 shows a routing example for the solder side of a four-layer board. The first two rows (R1, R2) are routed on the component side, while the inner two rows (R3, R4) are routed on the board's solder side. This example shows 6 mil trace widths.

14.8.3.3 Plated Through Hole (PTH) Isolation

Regardless of the technique used for the mounting pads shape or definition, isolation of the plated through hole (PTH) from the mounting pad is important. If the PTH is contained within the mounting pad, solder will wick down the PTH. The amount of solder that wicks depends on many factors, including PTH finish and coating variations. Because of this, the results are somewhat unpredictable. Some solder joints may be unaffected, while others will be starved to the point of creating opens. The worst result is a partially starved joint with severely reduced cross section. This joint can have significantly lower fatigue life and result in early system failure. Because the quality of the solder joint is guaranteed by control rather than inspection, designs/processes that result in random distributions are generally considered unacceptable, and the PTH-in-pad design is not suggested. All vias located between the BGA ball pads must be covered with solder mask. It is suggested that, at minimum, vias on the top side be covered with solder mask. The bottom side can also be covered.

Figure 14-14 shows the connection between the BGA ball pad and a via. This connection is often referred to as the dogbone footprint.

Figure 14-18. BGA Pads and Vias



14.8.3.4 PCB Quality and Co-planarity

The assembly yields of mounting BGA components is influenced by the PCB properties and process control procedures followed by the manufacturer of the PCB. PCB co-planarity requirements are directly related to the package size. Typical PCB manufacturing specifications allow up to 0.01 mm (1%) of warpage. For a 35 mm component, this would equate to nearly 0.35 mm of warpage in the area under the package footprint. It is obvious that no large body (>20 mm) component, peripheral leaded or BGA, would consistently solder to a site with this amount of bow. Most responsible PCB vendors take precautions to be well below the 0.010 mm specification that is recommended in the industry standard specifications.

14.8.3.5 Solderability Testing of BGA Components

Standard solderability test methods for through-hole and other leaded devices aren't suitable for testing BGA components. A 'dip and look' process will strip much of the solder ball off the substrate surface, leaving little to examine for determining if the solder surface was wettable. A wetting balance will only be able to sample specific solder balls on the substrate, which can be an acceptable method for sampling, but would take a considerable length of time to do the whole surface of high lead count devices, and would still only be representative of one unit. The untestablility of production BGA packages has led most suppliers to further develop their ball attach processes to insure minimal solder oxidation occurs. In some instances, this means doing all test and burn-in operations before ball attach. Where this is impossible, it means establishing stringent control over all operations after ball attach that would impact solderability. These operations, which usually include electrical testing, burn-in, inspection, bake and bag, shipping, storage, etc., are characterized to minimize oxidation and solder ball damage. However, with today's fluxes and controls on reflow furnaces, it is very unlikely that board mount problems can be traced to solder ball oxidation. If proper solder type, viscosity, screen print, and reflow practices are followed, very high board mount yields are being obtained even with product that has been stored for a couple of years on warehouse shelves. Most product is used within one year from the manufacturing date, which is also the limit for the desiccant in the moisture barrier bags. If solderability problems persist, a careful evaluation of the PCB solder pads should be done to insure that there is proper wetting occurring on both sides

of the connection. Soderability problems on BGAs are often traced to oxidation of the pads on the boards.

14.9 Package To Board Assembly

14.9.1 Fluxing

Most BGA assembly is done with a solder paste that contains flux, however, there are some companies reporting adequate results from mounting BGAs when using low residue, no clean or an aqueous clean flux. To obtain high yields and reliable joints this may require monitoring the surface insulation resistance of the board to insure there are no foreign contaminates on the board or slder ball surface that will show up as reliability problems later on.

Fluxing without paste does come with disadvantages; the ball reflowed with only flux will have a smaller solder volume than the initially attached ball. Therefore, the package will result in a slightly (usually 0.001-0.002mils) lower standoff height from the PCB. The thermal cycle reliability of BGA component could be slightly compromised because of the reduced standoff height, especially if solder balls are under the die area as in full array BGA types. The self centering ability decreases because of the smaller solder volume. The formation of the fillet between the ball and the board can be hindered if the ball is mis-shapen or there is some minor imperfection with the pad area.

14.9.2 Solder Paste

Assembly with solder paste has advantages over just fluxing the pads. The paste is the vehicle to provide the flux necessary to both the PCB and solder ball surfaces to enable proper soldering of the component to the board. A no-clean or aqueous clean solder paste with 63Pb/37Sn is commonly used in mounting the PBGA. Typically the choice of solder paste determines the profile and reflow parameters. Most paste manufacturers provide a suggested thermal profile for their products which should be referenced prior to developing a reflow process.

Since the BGA balls consist of solder, the flux activity on the ball surface is assured, so long as the paste reaches the ball surface. The selection of paste is generally made to fit the entire component mix being assembled, not driven by the use of BGA packages. It is necessary, however, to ensure that all the thermal and environmental requirements of the paste can be met.

On the average, the BGA packages do not need any specialized solder paste. However, most solder suppliers have developed a BGA paste that has minimal voiding during reflow. This paste may also be used for non BGA components as well. It is advisable that a time limit of 45 minutes or less be maintained from screen pasting to reflow (30 minutes is optimum) to avoid the paste drying out and affecting solderablity or contribuing to voids in the solder balls. Some manufacturers ratings (of 2 to 6 hours) for air exposure is judged by using 500 gram jars, not small dots of solder sitting on a board which dry out considerably faster.

14.9.2.1 Paste Deposit Inspection

The quality of the paste print is the single most important factor in producing high yield BGA assemblies. Defects detected after paste print require a strip and rescreen of the PCB. Any deviation can turn into a defect downstream requiring rework and repair. Thus the most economic area to intensify process controls when beginning BGA assembly is in the paste screening step.

Excessive volume of paste will have some negative effects on the self centering properties of the BGA (see Section 14.9.4.1) and could cause yield or reliability issues (shorts or bridges). However,

adequate paste thickness is required to compensate for board warp, poor component coplanarity, and to achieve acceptable board reliability. Another factor in yield or reliability is the presence of adequate flux. Any situation, such as a plugged stencil hole, that causes flux or paste to be omitted or severely limited on a pad can lead to an open connection after reflow. Print misregistration or excessive slumping that connects adjacent conductors (either pads or PTHs) is also a source of concern, as a short may be the final result. Therefore, it, is usually beneficial to perform a paste inspection step, especially during early manufacturing runs. Whether to use visual/microscope inspection, manual measurement, or invest in an automated system depends on the volume to be run and the overall manufacturing philosophy.

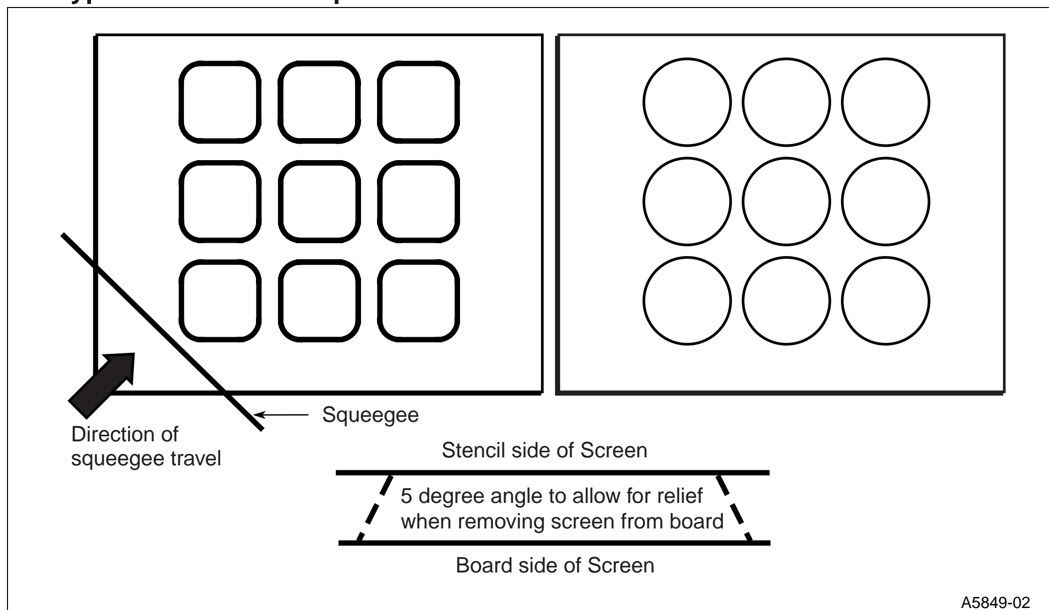
14.9.3 Solder Stencils

The stencil thickness, as well as the etched pattern geometry, determines the precise volume of solder alloy deposited onto the device land pattern. Stencil alignment accuracy and consistent solder volume transfer is critical for uniform reflow-solder processing. Stencils are usually made of brass or stainless steel, with stainless steel being more durable. Hole designs are dependant on the solder ball size, squeegee type, board layout, and the paste used. There appears to be no single hole style that is used by everyone. There are companies that are using square, diamond, round and oval shapped holes. Round holes are definitely the dominate design. Many companies are promoting a stencil with a rounded corner, square hole with five degree tapered opening has been shown to be a good hole design to use for BGAs with 20 mil or smaller solder balls. Thickness of the stencils are usually in the 6 to 8 mil (.15 to .20 mm) range. A squeegee durometer of 95 or harder should be used. The blade angle and speed must be fine tuned to ensure even paste transfer. Ensuring proper stencil application is the most important factor with regards to reflow yields further on in the process. The paste materials tend to dry out when not properly environmentally controlled (see14.9.2 for more explanation).

Maintaining a diameter to stencil thickness ratio of at least 3 to 1 can provide good BGA print characteristics, with larger openings providing better print quality. It is also beneficial to use an opening at least as large as the mounting pad to give a wide placement window. A typical design might be a 0.028 opening in a 0.006 thick stencil, going over a 0.024 pad on the PCB with 30 mil solder balls.

The printing of small amounts of paste onto the solder mask surrounding the pad has not proven to be a problem in either yield or reliability.

Figure 14-19. Typical BGA Paste Deposit

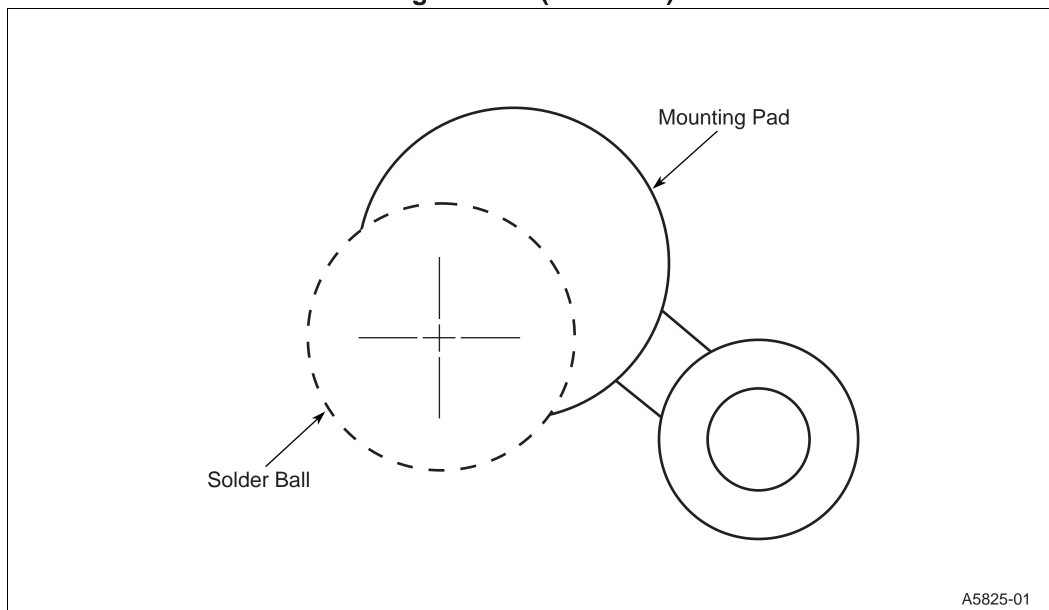


14.9.4 Placement and Alignment

14.9.4.1 Placement

BGA packages have shown excellent self-centering properties. Because of this, wide variation in placement location is accommodated during reflow of the solder joints. The general rule for BGA packages is that the placement be at least 50% on pad. This is illustrated graphically in Figure 14-16. The self centering characteristics of the BGA are attributed to surface tension which will pull the component onto the pad during peak reflow temperatures.

Figure 14-20. Ball to Pad Placement Misregistration (A4470-01)



Most common placement tools used for flatpack or other non-discrete packages have much better accuracy than necessary, and placement variability is rarely an issue for BGAs.

14.9.4.2 Alignment

The pick and place accuracy governs the package placement and rotational (theta) alignment. This is equipment/process dependent. Slightly misaligned parts (less than 50% off the pad) typically automatically self-align during reflow. Self centering on the pads is greatly reduced for grossly misaligned packages (greater than 50% off the pad) and may develop electrical shorts, as a result of solder bridges, if they are subjected to reflow.

14.9.4.3 Pick-n-Place Machines

The main areas of concern for Pick-n-Place machines are:

- Component body alignment
- Component ball alignment.
- Inspection of component balls before placement.

Most alignment inspections can be done from either the top or the bottom. As there is no defining mark on the bottom of the BGA package, the only way to ensure Pin 1 is located in proper position is from the top by using the Pin one mark as the orientation indicator. Proper placement is then done:

- From the top by alignment marks on the board (which must be done on the stencil, see Figure 14-17).
- By aligning off the bottom of the component by using an up-down vision system to accurately place the balls on the solder paste.

Because the top surface of some of the BGA packages (like the HL-PBGA types) are highly reflective, some top side vision systems do better with their inspection process if they use a diffuse lighting source instead of polarized source. A round fluorescent tube type light near the package or a light filter over the polarized fixture seems to enhance the operation of some systems.

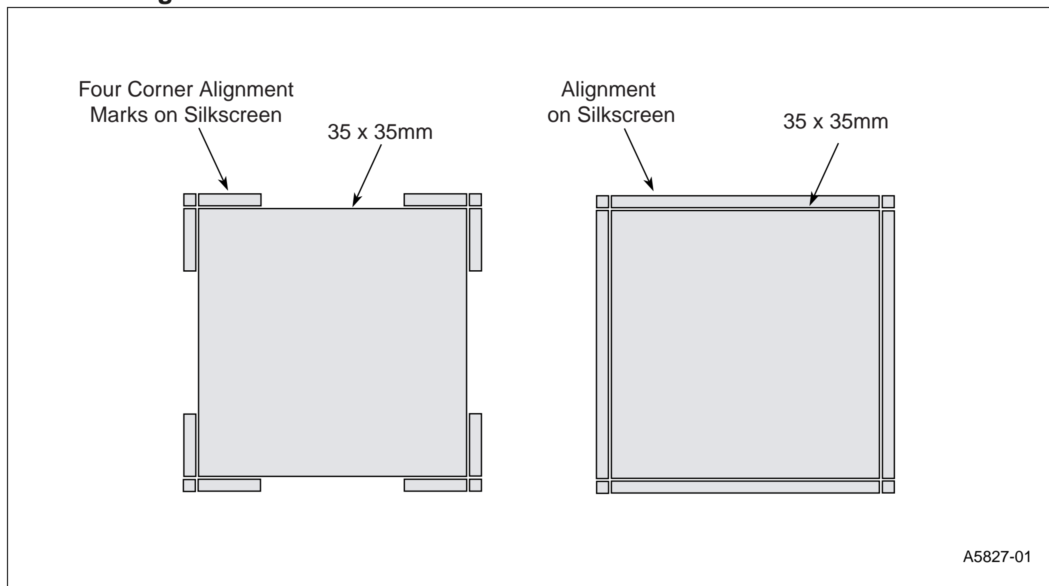
14.9.4.4 Outline for Machine Placement

Variability of the body dimension specification is 0.10 mm (~4mils) worst case. The same data indicates that the ball location variability is 0.075 mm (~3mils) worst case. The impact on alignment during the placement (vision) process is:

- If the package edge is used to determine the true center of the component, alignment of the balls is within 0.075 mm of true position.
- If only the package edge is used, alignment of the balls is within 0.10 mm + 0.075 mm (0.175mm, ~6mils) of true position.

Given that the allowable misalignment of the balls is 50% or less of the pad width (e.g.; 12mils for a 24mil pad), a manufacturable process can be achieved using only the package edge (outline) and alignment marks on the stencil for machine placement, see Figure 14-16.

Figure 14-21. BGA Alignment



14.9.5 Solder Reflow

Except for semi-accurate placement of packages, there are no special requirements necessary when reflowing BGA components. As with all SMT components, it is important that profiles be checked on all new board designs. In addition, if there are multiple PBGAs on the board, the profile should be checked at the different PBGA locations on the board. Component temperatures may vary because of nearby surrounding components, location of the part on the board, and areas of package densities. Temperatures may also be different at the edge of the package than in the center. Chapter 9 provides a more in-depth look at how to manage these types of concerns. Table 14-10 provides specific parameters to be followed during SMT preheat, preflow, and reflow processes. Proper desiccant handling procedures should be followed prior to reflow to insure optimal reliability of the final product. See Chapter 8 for details.

When doing second pass wave solder of mixed technology components on the same board as BGAs, insure that the solder wave profile is very tightly controlled. A high temperature on the wave solder process can warp the board and break the joints on the topside of the BGA component.

14.9.6 PCB Cleaning

Cleaning can be done with aqueous, semi-aqueous, or solvent based systems, or not done at all. With the need to eliminate Chloroflourocarbon (CFC) containing materials, many companies have moved to using a no-clean or aqueous based system. The proper cleaning of solder flux residues and other ionics left on the boards from the assembly process is necessary for long term reliability of the finished product. "NO clean" fluxes simply mean that there is no harmful residues left on the board that will cause any corrosion or damage to the components if left on the board. This residue has sometimes shown to be a collection point for outside contamination on the board surface. For any application, an evaluation needs to be done to see if the remaining residue still needs to be removed from the boards in final application.

Table 14-10. PBGA/HL-PBGA Example SMT Reflow

Zones	Characteristic Description	Windows/Limits
Preheat	Initial heating of lead/component Peak temperature	1-3 °C/second 100-140 °C
Pre-Reflow	Dryout and solder paste activation Soak Time	120 -170 °C 120 Seconds
Reflow	Time above 183 °C Peak component body temperature.	45 - 120 Seconds 205 - 225 °C 220 ° C Maximum
Cool Down	Cooling rate	2 -3 °C/second

14.9.7 SMT Process

Many factors contribute to a high yielding BGA assembly process. A few of the key focus areas and their contributing factors are highlighted in Table 14-11.

Table 14-11. Essentials for Assembly Quality

Solder paste quality	Uniform viscosity and texture. Free from foreign material. Solder paste should be used before the expiration date. Shipment and storage temperatures are maintained at the proper temperature. Paste is protected from drying out on the solder stencil.
Motherboard quality	Clean, flat, well-plated solder ball land area. Good soldermask coverage.
Placement accuracy	Tight tolerances are not usually required. The BGA can self-center itself as long as a major portion (more than 50%) of the solder ball is in contact with the fluxed solder paste land area on the board. Alignment marks (targets) on the PCB are helpful for placing parts.
Moisture Sensitivity Precautions	Know your components moisture sensitivity classification and adhere to IPC/JEDEC moisture control conditions or package delamination or cracking may occur.

14.9.8 Rework of BGA Packages

SMT yields of BGA packages are very high, but there may still be a possible need for rework of components. Component defects, SMT defects, or other functional problems require rework of the package from the PCB.

Rework is the process of removing a component from a PCB, and replacing it with a new component. The removed component is not immediately reusable. The shape and volume of the solder balls will not be the same as a new package. If component reuse is desired, a separate process for replacement of solder balls should be used.

14.9.8.1 Rework Tooling

There are several systems currently on the market for reworking BGA components. Some systems direct the heat under the package while other systems direct hot gas on the top of the package. Backside heating is a common feature of all BGA rework tools. While some systems attempt to backside heat with local hot gas behind the package to be reworked, this often results in large-scale PCB warpage during processing, especially if the PCB is large. The suggested method is to use a global heat that brings the entire PCB up to a specified temperature (100-125° C).

14.9.8.2 Rework Processing

The rework process is relatively simple. The biggest challenge of BGA rework is the requirement of a controlled process. Often SMT rework is accomplished with very coarse methods such as hand placement and soldering iron reflow. Such approaches do not work for BGA. If the process used to rework the package is not understood and controlled (similar to initial attach), the result is likely additional defects and additional rework.

To perform BGA rework, there are four basic steps: removal, site preparation, flux or solder paste application, and replacement/reflow. The following subsections describe each.

14.9.8.3 Component Removal

The BGA package is removed with a hot gas tool, usually fitted with a custom head that is sized to the BGA package. Proper sizing of the gas head reduces the thermal impact on adjacent packages. Correct tool settings depend on the tool being used, the package being removed, and the PCB. Determining the settings is an exercise in profiling, similar to initial reflow. By using a profile assembly with thermocouples mounted in solder joints, tool settings can be determined which assure that all solder joints reflow properly. Monitor both the top and bottom of the PCB.

The component is removed from the PCB by a vacuum nozzle within or integral to the hot gas head. When profiling, shut the vacuum off so the component is not removed to avoid damage to the profile card. Control the pressure of the head down onto the component during removal. If pressure is applied after the solder balls are melted, the solder is pressed between the “plates” of substrate and PCB, resulting in bridging that must be removed manually. Two possible solutions are:

- Establish the head height prior to reflow and control the stroke.
- Place shims under the edges of the component to prevent collapse

Consider the effects on other components. If other SMT packages are near the package being reworked, monitor their solder joint temperature. If the temperature approaches reflow temperatures, shielding may be necessary.

Preheating the PCB assembly is good practice when reworking BGA packages. Advantages are:

- It can reduce heating time required using the rework head. In cases where one PCB assembly can be preheated while another is reworked, cycle time can be greatly reduced.
- More uniform profiles are achieved. One challenge of profiling is the temperature spread between center and edge solder joints. Preheating reduces the spread by bringing the baseline temperature closer to the reflow temperature.

- PCB warpage is minimized. Warpage is caused by the higher local temperature at the site than the surrounding area. By raising the PCB assembly temperature, the mismatch is minimized and warpage is reduced.
- It reduces problems with adjacent components, as it allows shorter gas heat times or lower gas temperatures to be used, thus reducing risk of affecting neighboring solder joints.

14.9.8.4 Site Preparation

Once the component is removed, prepare the site to accept the new component. The removal process generally leaves varying amounts and shapes of solder on the mounting pads. To ensure maximum probability of success, it is suggested that all sites be made as uniform as possible prior to reattach of the new component. Solder removal is accomplished in a variety of methods — from solder wick to custom solder vacuum tools — the desired result is similar mounting surfaces across all mounting pads.

14.9.9 Removal and Replacement Process

A removal and replacement procedure for a PBGA package is as follows:

1. Plastic Ball Grid Array Package Removal from Board.
    a. Preheat the board to a minimum temperature of 80° C (max. temperature is 220° C). Any part of the board over 160° C-170° C is close to the solder melting temperature of 183° C and risks damaging the joints of other components on the board, especially the bottom side parts which are closer to the heat source. It is recommended that each manufacturer conducts time and temperature experiments to determine the optimum conditions to minimize board/package warpage. Monitor both top and bottom-side board temperatures.
    b. Dispense liquid, no-clean flux between the package and the board.
    c. Attach a vacuum pick-up tip onto the package and apply hot air (preheat, ramp time, and temperature to be determined by manufacturer's own experimentation). Note that some cases of mother-board pad lifting problems have been reported possibly due to the machine type used and how much upward tension there is on the vacuum pick-up. This problem may be solved by the manufacturer determining the typical time to release when using the vacuum pick-up, adding 30 seconds and then not applying the vacuum pick-up during removal until this amount of time had passed.
    d. Lift the package from the board.
    e. Turn off the hot air and carefully remove the board from the heat source and allow to cool to a safe, handling temperature. Inspect the board to determine no damage has occurred to adjacent components or to the board itself.
2. Inspection, Preparation and Replacement of New Package
    a. Remove any excess solder from the PBGA solder pads using a solder wick or vacuum.
    b. Clean the PBGA solder pads with alcohol and brush. Allow the board to dry and inspect to ensure a clean surface.
    c. Preheat the board to a minimum temperature of 80° C. Any part of the board over 160° C-170° C is really close to the solder melting temperature of 183° C and risks damaging the joints of other components on the board, especially the bottom side parts which are closer to the heat source. It is advised that each manufacturer conduct temperature experiments to determine optimum conditions to minimize board/package warpage. Monitor both top and bottom-side board temperatures.

d. Use of stenciled solder paste is optional (and as PBGA packages get larger, may be required). For applications where a solder stencil is not possible or not desired, acceptable results may be obtained by only applying flux to the pre-tinned pads. This is done by using a non-metallic spreader and applying a no-clean flux paste to the pads on the board. Be careful not to scratch the pads or the board.

e. Apply liquid flux to the solder balls of the new package. Once the liquid flux is applied, within two minutes, place the package on the board and then reflow (be sure to use the board's alignment features fudicials and then place component using either a mechanical or manual means).

f. Reflow solder balls using hot air directed at the edge and under the package body. It is recommended that temperature experiments be conducted to determine optimum conditions.

g. Remove the board from the heat source and allow to cool to a safe handling temperature.

h. Inspect the board and package to verify proper solder ball collapse and observe any defects that may have been caused by the rework procedure.

14.10 Package Performance

14.10.1 Thermal Performance

In general, three factors affect the thermal performance of the BGA: package and board materials, package geometry, and use environment. Obviously, the more thermally conductive the materials, the better the package dissipates heat. On a PBGA package, the molding compound that surrounds the chip, which provides mechanical protection as well as a surface for marking, is typically 45mils in thickness and is the main barrier to the elimination of the heat from the BGA package.

In general design-related factors have greater thermal effect on the PBGA than material-related variables. A large die spreads heat easier, as does a larger package size and thus a higher ball count. Substrate design features have tremendous effect on the package's ability to dissipate heat. Vertical vias running through the substrate help to transfer heat from the die to the solder balls. Four-layer designs often incorporate conductive 2-ounce copper ground planes, which have a significant, positive effect on thermal performance. The Enhanced PBGA has thermal balls under the die while H-PBGA and the HL-PBGA utilize a heat spreader or slug across the top of the package to dissipate heat even more efficiently. PBGAs with center thermal balls dissipate considerable heat into the board. A considerable increase in thermal effectiveness of a BGA package can be obtained by using boards that are thermally efficient, increasing the airflow, or providing thermal paths from the board. Remember, with PBGAs, the board is your primary heatsink.

Environmental conditions play a critical role in the thermal performance of PBGAs. Ambient conditions, junction and case temperatures, the device's placement and orientation on a board, in conjunction with the volume and temperature of air flowing past the unit present a broad range of possible thermal solutions and problems for IC packaging. Typically a package cannot be capable of handling a given power requirement unless the environmental conditions allow heat to dissipate.

When environmental or geometric constraints limit a BGA's ability to dissipate heat, a copper or aluminum heatsink is often used to provide an additional method for heat transfer. As with other types of packages, the heatsinks for BGAs vary in design and methods of attachment. Most applications recommend a maximum case temperature for the package. Various factors effect the case temperature including ambient conditions and airflow. If the case temperature exceeds the recommended rating, a heatsink may be required. Contact an Intel applications engineer for the product to determine if a heatsink has been developed for the particular package or application.

Refer to Chapter 4 of this handbook for more information on thermal characteristics.

14.10.2 Electrical Performance

Generally, due to the fine-line laminate base and to the lack of long pins, the electrical performance of BGAs are typically better than the pinned packages. The shortening of the electrical paths by the solder balls through the plated via-holes to the conducting plane/ground plane reduces electrical parasitics. This can be improved further by optimizing the shortening of the overall trace length. For electrical performance details of the package, please consult the product data sheet or call your local Intel sales office.

14.11 Revision History

- General review & edit of the chapter
- Added new package proliferations
- Added FC Style, H-PBGA information