SURFACE MOUNT ASSEMBLY OF PBGA/TEBGA PACKAGES

Issue 1: Sept. 1997

TABLE OF CONTENTS

OVERVIEW

Ball Grid Array Packages are rapidly replacing Plastic Quad Flat Pack (PQFP's) and PLCC's due to today's increased I/O demanded per unit board area. PQFP and PLCC surface mounting technology is a mature process with easily inspectable solder joints. Surface mount array packages such as the Plastic Ball Grid Array (PBGA) or Thermally Enhanced Ball Grid Array (TEBGA), when attached to the printed circuit board, do not present the user with easy access for visual inspection to all the solder connections being made. Only the peripheral solder joints can be inspected visually, although some use of X-Ray analysis of solder connections is being made. However through the implementation of proper process control, defect free soldering of BGA packages does provide high yielding board assemblies.

The objective of this document is to outline the processes which, if followed, will ensure that the surface mount assembly of the PBGA/TEBGA packages provides excellent board assembly yield and reliability.

BACKGROUND

PACKAGE CONSTRUCTION

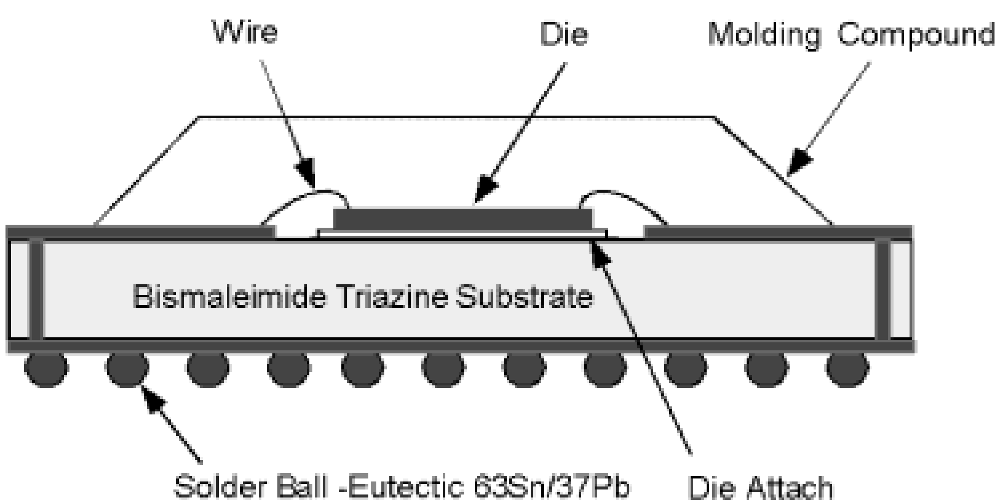


FIGURE 1 Plastic Ball Grid Array (PBGA)

Figure 1 shows the cross section of a Plastic Ball Grid Array (PBGA). The package is overmolded with an epoxy based molding compound (or plastic) which encapsulates the device mounted on a Bismaleimide Triazine (BT) laminate substrate.

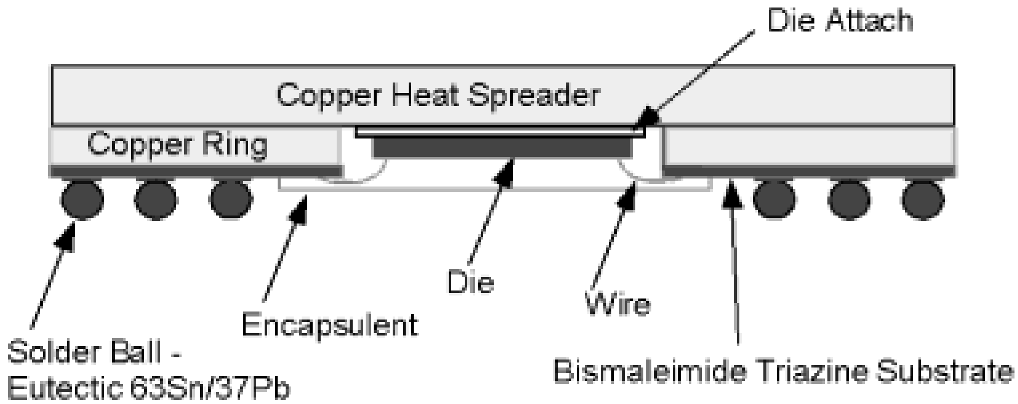


FIGURE 2 Thermally Enhanced Ball Grid Array(TEBGA)

Figure 2 shows a TEBGA which features direct attachment of the die to a copper heat spreader with silver filled conductive die attach to aid in thermal dissipation. If required the package top surface can easily support a heat sink. The die is protected with a dispensed encapsulent.

JEDEC MOISTURE LEVEL CLASSIFICATION

Because these packages use plastic molding compounds as environmental protection they are subject to the same sensitivity to moisture uptake as their leaded counterparts the PQFP and PLCC. They can present the user with the same unpleasant “popcorn” failure modes if proper precautions are not followed.

PMC-Sierra has qualified the TEBGA package following JEDEC standard JESD22-A113 Level 3 moisture level preconditioning. The plastic ball grid array package is currently undergoing qualification (PMC-970840 Qualification Plan for the 256L PBGA) following JESD22-A113 Level 3 preconditioning and, given assembler subcontract qualification data we have analysed to date, no difficulties are expected. JEDEC Level 3 Moisture Sensitivity Level means PMC-Sierra will ensure the devices will be sealed in a moisture barrier bag along with dessicant and a humidity indicator card (HIC); the user will have a maximum of 48 hours (2 days) out of the bag exposure under 30 deg C and 60 % R.H. conditions in which to complete assembly of the units on the card.

SMT ASSEMBLY FLOW

The assembly flow using BGA packages is similar to the conventional SMT assembly flow and the process flow recommended is as shown in Figure 3.0 below.

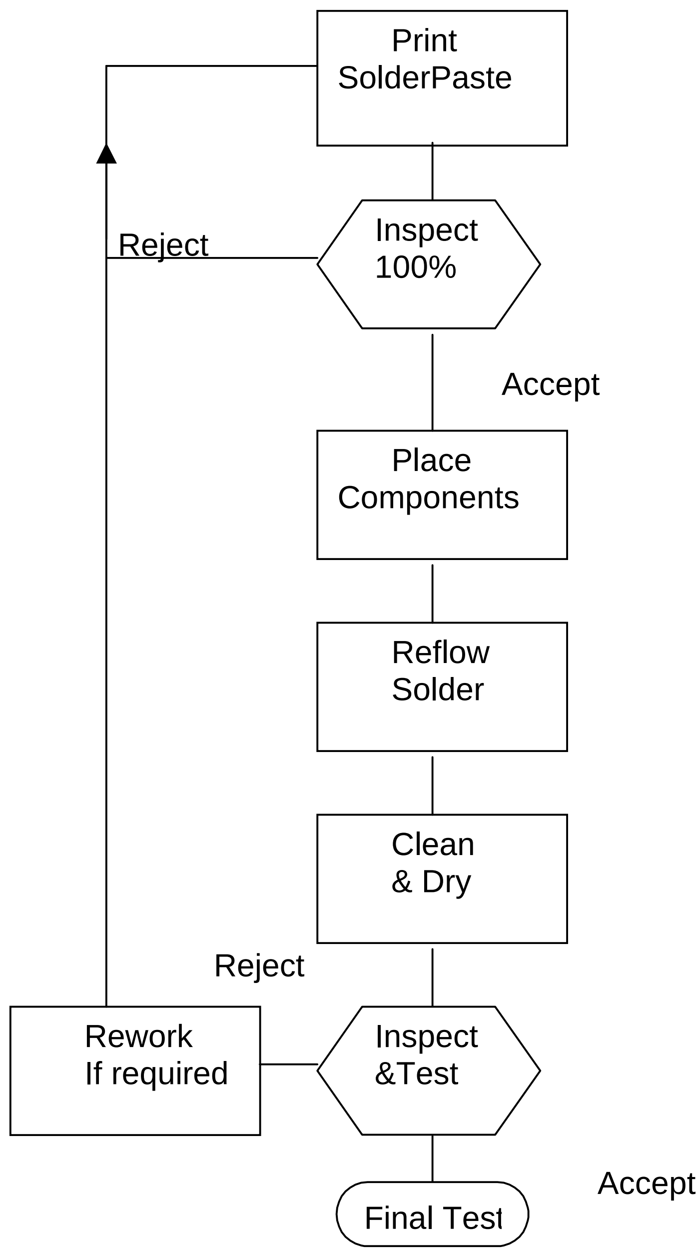


FIGURE 3 PBGA/TEBGA SMT PROCESS FLOW

PRINTED CIRCUIT BOARD(PCB) REQUIREMENTS

The factors pertaining directly to the PCB which most affect the assembly performance of PBGA/TEBGA are the solderability and the lack of planarity in the PCB itself. The solderability is an important factor due to the lack of visual inspection possible of the finished solder joints other than on the package periphery.

PCB VISUAL INSPECTION

Prior to assembly it is advisable to conduct a visual inspection to ensure that all solder lands are free of any foreign particles, grease, solder mask residue due to incomplete etching to aid in the complete wetting of the pad surface. All pad surfaces should be bright and shiny not dull or heavily oxidized.

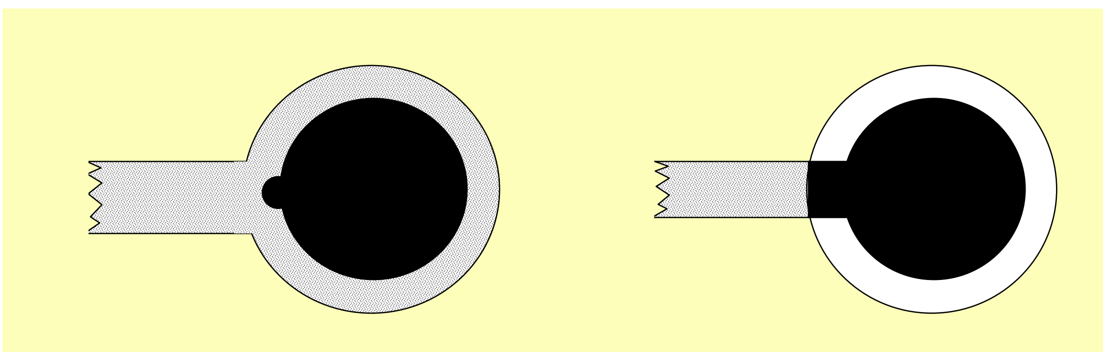
SOLDERABILITY

Prior to assembly PCB’s should meet the design requirements in IPC-D-275 and must conform to the solderability test for printed circuit boards in ANSI/J-STD-003.

PRINTED CIRCUIT BOARD(PCB) DESIGN

The basic pad design recommended for the PCB is an array of circular pads which provides the mounting surface for the package. It is very important that the mounting pads be isolated from any plated through holes(PTH) required for connection to other planes by solder mask. Stacking of the PTH under the active area of the mounting pad can result in the free solder being drawn into the plated through hole and this can effectively starve the solder joint of a suitable volume of solder to establish a properly soldered joint.

The pad design can be of one of two types either solder mask defined (SMD) or non-solder mask defined (NSMD) as shown in Figure 4.0 below.



SMD Pad　　　　Non-SMD Pad

FIGURE 4 Solder Pad Variations

SOLDER MASK DEFINED PAD (SMD)

The Solder Mask Defined (SMD) pad design offers increased pad-laminate adhesion and solder-joint definition. The SMD pad does not provide the highest routing density on the PCB due to the size of the pad. A typical design might use a 0.028” circular copper pad with a 0.003” diameter solder mask overlap defining a 0.025” pad opening with the increased use of through holes the boards will be less economical with this style of pad.

NON-SOLDER MASK DEFINED PAD (NSMD)

The NSMD pad design offers increased routing density and is more economical. A typical board design may use a 0.025” copper pad for PBGA to match the pad size on the package and a 0.003” annular ring solder mask clearance. Specific pad sizes for the two styles of packages are shown in table 1.0 below and generally speaking it is desirable to match the PCB pad to the package pad.

Package Type	Pitch	Pad Size	Ball Diameter
PBGA	1.27 MM (0.050”)	0.63 MM (0.025”)	0.76 MM (0.030”)
TEBGA	1.27 MM (0.050”)	0.58MM (0.022”)	0.76 MM (0.030”)

TABLE 1 Ball Grid Array Pad Designs

SOLDER PASTE

The solder paste selected for PBGA/TEBGA placement should be based on the requirements of the entire component mix of the PCB under assembly and not based solely on the PBGA/TEBGA requirements due to the wide process latitude which the PBGA/TEBGA can accommodate, in fact soldering can be accomplished with just flux applied to the card since the balls are eutectic (63%Sn/37%Pb) with a melting temperature of 183 degrees C, have good planarity ,(PBGA<0.006 inch and TEBGA<0.008 inch) and solder balls which melt/collapse during reflow.

However since solder paste is a convenient vehicle for holding the device in place during reflow and does not require an additional step as it is being used for other components on the PCB it’s use is recommended, both Type 3 and Type 4 mesh pastes have been reported with equal success. It is important however that all of the thermal and environmental requirements suggested by the paste vendor are met. It is recommended that the user follow closely the paste vendors recommended profiles in terms of ramp rates and peak temperatures.

SOLDER PASTE APPLICATION-SCREEN PRINTING

Stainless steel stencils fabricated using either chemical etch with electropolishing or laser cut fabrication are recommended. Stencil thickness of 0.15-0.20 mm (0.006-0.008 inch) with circular apertures are recommended. Metal and/or polymer blades with a minimum durometer of 90 are recommended for usage with PBGA/TEBGA packages as they have proven to provide higher and more consistent deposits.

STENCIL DESIGN

As with PCB design, having the proper stencil design is much of the battle in paste printing. The usual design is to use circular openings over circular holes. As the majority of the eutectic solder will be provided by the package, the exact solder paste volume is not critical, and thus the aperture sizes can be varied somewhat without affecting the final product quality. However, it is best to use a design that results in good paste release and therefore results in consistent paste deposit shapes and volumes. Maintaining a diameter to stencil thickness ratio of at least 3 to1 to assures good BGA print quality, with larger openings providing better print quality. It is also good to use an opening at least as large as the mounting pad to give a wide placement window. The recommended minimum opening in a .006” thick stencil is .024”, in a .008” stencil it would be .024”. A typical design might be a .028” opening in a .006” thick stencil, going over a .024” pad on the PCB. The printing of small amounts of paste onto the soldermask surrounding the pad has not proven to be a problem in either yield or reliability.

PASTE VOLUME

As the majority of the eutectic solder will be provided by the package, the amount of paste deposited is not critical. It is suggested that the finished paste dispense represent the shape of a trapezoidal hockey puck with a flat top surface (see Figure 5 below). An excessively coned deposit usually indicates a paste release problem and could lead to plugging and lower yields.

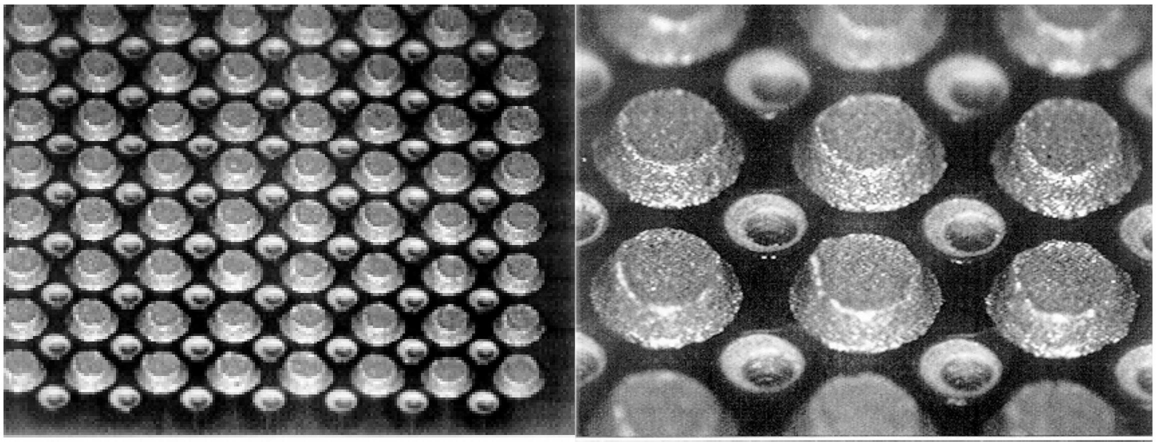


FIGURE 5 Proper Solder Paste Deposit *(courtesy AMKOR/ANAM/IBM)*

PRINT QUALITY INSPECTION

The quality of the paste print is the single most important determining factor affecting BGA assembly yield. Therefore it is imperative that during the initial manufacturing runs using BGA's that visual inspection of the paste print be incorporated into the process flow. Any situation that causes problems such as plugged stencil holes which in turn create voids in the dispense and or excessively slumped deposits can be visually detected prior to assembly and the PCB stripped and rescreened thus trapping any gross defects. Once process control has been adequately demonstrated through consistently acceptable yields the paste inspection could be reduced or eliminated for cost and throughput.

PLACEMENT

Due to the ability of the molten solder on the package to exert self-centering forces during reflow the placement accuracy demanded of the pick-place equipment is not challenging for today's manufacturing equipment. The general rule of thumb for BGA placement is that the placement must be “half on pad”. Current typical placement accuracies and repeatability of pick-place machines are on the order of +/-0.004 inches (+/- 0.1 mm) which is required for fine pitch PQFP and the accuracy required for BGA placement is +/-0.0085 inches.

Several placement techniques can be used including body recognition, mechanical alignment using a mechanical nest and vision recognition. Vision recognition is the fastest method and utilizes a camera to look directly at the balls for registration.

Manual placement and/or repositioning is not recommended

REFLOW

Reflow of PBGA/TEBGA packages is in itself nothing new from standard SMT reflow. No “magic profiles” exist to achieve best yields, nor is there a “right” type of furnace or heat to produce the best joint. Both IR and convection furnaces are used for PBGA/TEBGA assembly. Convection generally provides more even heating, and may be necessary if thermal spread in IR is too great. Likewise, both air and nitrogen atmospheres can be used. Nitrogen reduces probability of soldering problems, but final joint shape and reliability are not necessarily enhanced by using nitrogen. The profile and other reflow parameters are determined by the choice of solder paste. Every paste manufacturer provides a recommended thermal profile for their products. This guide, or any other internal specifications used for general SMT profiles within a factory, should be applied to BGA packages as well.

SPECIAL CONSIDERATIONS-PBGA/TEBGA

There are several special considerations relative to reflow that do come into play with BGA packages, the first is process control. While the profile does not need to be custom for BGA packages, the control of the reflow does. It is not possible to inspect the final solder joints to insure that all of them have reflowed properly. Thermal profiling is critical, as it is the ONLY method that will insure that all joints on a BGA package are completely reflowed. The thermal profile should be checked for EVERY new board design. If there are multiple BGAs on a single board, it is recommended to check the profile of all of the BGA sites. Differences in surrounding components, board cross-section, part density, and BGA part design all will result in varying temperatures under the BGA packages. There will also be a temperature difference between the edge of the part and the center, usually with the center balls lower in temperature. It is therefore recommended that more than one thermocouple per part be used to insure part temperature. As a minimum a BGA must be profiled at its center and leading edge. The thermocouples should be located within the solder joints themselves for most accurate readings. The simplest method for achieving this is to drill a hole through the laminate and into center of the pad prior to reflow, then reflow the board with the BGA in place, after reflow the pilot hole is used as a guide for a deeper hole (0.010”-0.020”) into the solder ball of the pad, thermally conductive epoxy may then be used to secure the thermocouple in place.
The maximum PBGA/TEBGA body temperature reached during the profile of the PCB must be 220 deg. C. It is also recommended that the printed circuit board

laminate temperature not exceed 245 deg.C .

CLEANING

Cleaning of BGA’s has proven to be within the capabilities of most acqueous systems. Conveyor speed and temperature should be set such that after cleaning there has been complete removal of any excess moisture which may reside trapped underneath the package. A quick test can be performed with a blast of CDA at the solder lands of the package to see if any moisture remains after the drying cycle and, if so, adjustments may be made to the duration or temperature.

CONCLUSION

In summary, with proper process control implementation it is possible to produce high reliability, high yielding surface mount assemblies using BGA packages. Most users who have taken the time to understand the processes and have accepted the loss of visual inspection have embraced BGA’s as the package of choice for high density applications.

REFERENCES AND FURTHER READING

PMC-951041	PMC-Sierra Application Note : Handling Moisture Sensitive IC Packages
PMC-940101	Procedure for the Bake Out of Moisture Sensitive Plastic IC Packaged Devices
AMKOR/ANAM	Application Notes on Surface Mount Assembly of AMKOR/ANAM PBGA and SuperBGA Packages
AEIC :	Mastering & Implementing BGA Technology
	Advanced Electronics Interconnect Center (AEIC) 25 Iris Way Haverhill, MA 01830 Tel/Fax 508 373-2600
IPC-D-275	Design Standard for Rigid Printed Boards and Rigid Printed Board Assemblies
IPC J-STD-015	Implementation of Ball Grid Array and Other High Density Technology
ANSI J-STD-003	Solderability Tests for Printed Circuit Boards
JESD22 - A113-A	Preconditioning of Plastic Surface Mount Devices Prior to Reliability Testing
contact :	Global Engineering Documents 15 Inverness Way East Englewood, CO 80112-5074 Tel 1-800-854-7179 or 303-397-7956

NOTES

PMC-970951R1 Issue date: September, 1997.

Printed in Canada