TECHNICAL NOTE

SMT RECOMMENDATIONS FOR BGA ASSEMBLY

INTRODUCTION

Ball grid array (BGA), fine pitch ball grid array (FBGA), and other chip-scale packages (CSP) require special consideration for successful assembly. As package geometry continues to shrink, processing margins decrease when compared to leaded packages. Surface mount assembly processes with tight control limits are needed to successfully manufacture boards containing these components.

A traditional BGA package has a ball grid pitch of 1.0mm pitch or greater. An FBGA package is defined as a ball grid array (BGA) with a ball-to-ball pitch of 0.80mm or less. In both cases, a substrate interposer is used to redistribute the IC I/Os to the grid of solder balls. Wire bond or flip chip interconnects are typically used to connect the IC to the substrate. Micron's BGA product offerings cover a range of ball pitches, including 1.25mm, 1.0mm, 0.8mm, and 0.75mm. Future BGA products will likely have a ball pitch as small as 0.5mm.

A CSP is defined as a package that is no more than 10 percent greater in size than the individual IC dice being used. Therefore, CSP components are available in a wide variety of different packages, a true flip chip bumped die, a die with a redistribution dielectric layer bump or ball array, an FBGA with a tape-based interposer, or even an FBGA.

The surface mount technology (SMT) recommendations in this technical note are made with the assumption that FR-4 is being used as the PCB base material. Refer to Table 1 for a list of typical material properties used in BGA packages.

PCB PAD LAYOUT

Proper pad layout is essential to achieving a balanced solder joint structure. An optimum solder joint is one that retains as much of the original ball height as possible and has equal cross-sectional areas at the BGA and the PCB interfaces. If one interface is significantly smaller than the other, that interface will be a higher stress region. Premature solder joint failure typically occurs in the higher stress region.

Micron® recommends a round non-solder mask defined (NSMD) pad and a PCB pad diameter with a 1:1 ratio to the component pad. If questions arise, Micron recommends that the PCB designer comply with IPC-SM-782A, "Surface Mount Design and Land Pattern Standard" requirements. For most ball sizes, the nominal land pattern size should be equal to 80 percent of the nominal ball diameter. For example, according to the IPC requirement, a 0.80mm pitch FBGA with a nominal ball diameter of 0.40mm should have a nominal land pattern size of 0.32mm.

The BGA pad diameter Micron uses is defined on the package outline drawing, which is part of the component data sheet. As the package drawings indicate, the PCB pad size should not be equal to the diameter of the solder ball, or the result will be a misshapen and collapsed solder joint. Refer to Table 2 for examples of pad sizes and tolerances used for various Micron components.

NOTE: The pad sizes specified in a data sheet should take precedence over pad sizes contained in this technical note.

Table 1
Typical BGA Material Properties

PROPERTY	SILICON	MOLD COMPOUND	SUBSTRATE	COPPER	SOLDER BALL
Coefficient of Thermal Expansion, CTE (PPM)	2.7	7–15	14–22	17	25
Glass Transition, T_g (ºC)	N/A	170	180	N/A	N/A
Melting Point, T_m (ºC)	1685	N/A	N/A	1,083	179–183

Table 2
SMT Design Recommendations – Nominal Values (mm)

BGA PITCH	BGA PAD SIZE	BALL DIAMETER*	PCB PAD SIZE	FOIL THICKNESS	SQUARE APERTURE
1.25	0.60	0.75	0.60	0.127	0.60
1.00	0.33	0.40	0.33	0.102	0.33
0.80	0.33	0.40	0.33	0.102	0.33
0.75	0.27	0.35	0.27	0.102	0.27

To convert mm to mils, multiply by 39.37
* Ball diameter measured prior to mounting and reflow

PCB SOLDERING FINISH

Use of a surface finish that is planar and has good solderability performance is important in achieving high assembly yields. The two most commonly available finishes that meet these requirements are copper with an organic solderability preservative (OSP) and electroless nickel (immersion gold [ENIG] over copper). The performance and reliability of these two finishes is widely documented.

Various electroless and electrolytic plating processes are available for gold, but only the ENIG process is self-limiting. Gold plating thickness can be an order of magnitude thicker than gold from an ENIG process. To prevent solder joint embrittlement due to an excessive weight percent of gold, only an ENIG process should be used.

Micron does not recommend using a standard hot air solder level (HASL) finish. The 63Sn37Pb or 60Sn40Pb solders used in HASL processes can provide good solderability, but the surface planarity and thickness control is much worse than that for OSP and ENIG finishes. In general, the negative attributes associated with a HASL finish are relatively poor surface planarity, unpredictable solder thickness, and a greater amount of PCB bow/twist.

MOISTURE SENSITIVITY

Plastic IC packages absorb moisture when exposed to atmospheric conditions. During surface mount reflow processing, absorbed moisture vaporizes. To prevent IC damage from internal stresses generated by the moisture vaporization process, it is critical that strict adherence to floor life exposure times be followed. The allowable floor life exposure time for any given part is identified on the label for each moisture barrier bag of components contained in a shipment.

If the floor life exposure time is exceeded, the components need to be baked dry. Refer to Micron's technical note on moisture absorption, TN-00-01, for recommended bake-out conditions.

SCREEN PRINT MATERIALS AND PROCESS

Solder pastes with Type IV (preferred) or Type III solder powder are recommended for improved printing performance. Two solder alloys will perform well, are widely used in the industry, and are commonly available are 63Sn37Pb and 63Sn36Pb2Ag. These two alloys are of widespread use in the industry and are commonly available.

Care should be taken to apply the proper volumes of solder paste. To enhance solder paste release, the walls of the apertures should be as smooth as possible. Our general recommendation for the aperture geometry is a square aperture with sides equal in length to the diameter of the PCB pad. The corners of the square should also be slightly rounded, and the aperture should be slightly tapered or trapezoidal in the vertical axis. Smooth walls, rounded corners, and a trapezoidal cross-section enhance the release of solder paste from aperture. See Figure 1 for examples of these apertures.

For 1.0mm pitch and less BGAs, a 100 percent post-print 3D inspection is highly recommended for verifying that solder paste volumes are within process control limits. For 1.25mm pitch, a sample-based, post-print 3D inspection is recommended.

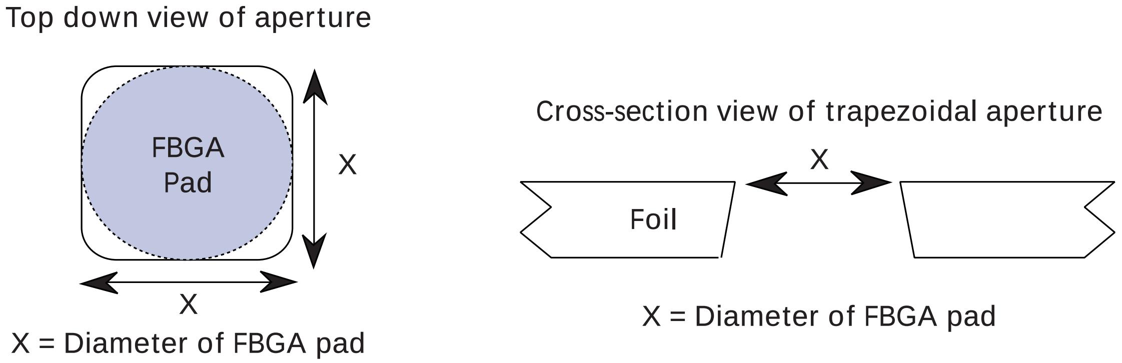

Figure 1
Stencil Design Recommendations

COMPONENT PLACEMENT

As component ball pitch decreases, accurate commponent placement becomes more important. Many different types of pick-and-place assembly equipment are readily available to ensure accurate placement.

Ideally, the goal is to have perfect component alignment with the PCB. Because this is not always possible, it is recommended that a minimum of 80 percent of the ball diameter intersect with the PCB land. This will help maximize the inherent self-alignment properties of the BGA package.

BGA packages self-align during the reflow process due to the surface tension of molten solder.

REFLOW PROCESS

As might be expected, establishing and controlling the parameters of the board reflow process is critical. General recommendations include the use of forced-air convection reflow, an N2 environment with an O2 content <100 PPM, and a peak reflow temperature of 215°C.

The advantage of a forced-air convection reflow process is that a temperature uniformity of ±5°C or less can be maintained across even complex PCBs.

An inert N2 environment provides well known and documented advantages over an oxidizing air environment for soldering. An oxidizing air environment has a deleterious effect on wetting, void content, and solder joint standoff due to formation of oxides on soldering surfaces. The inert N2 environment results in improved wetting, reduced voiding, and an increase in solder joint standoff (due to lack of oxide film on surface on molten solder). All of these are positive attributes that improve time zero yields and solder joint reliability.

The maximum reflow temperature should be minimized to prevent unnecessary damage to any plastic package ICs. When using 63Sn37Pb or 63Sn36Pb2Ag, successful soldering can be achieved with peak temperatures of 215°C to 220°C. Again, for any plastic package ICs, exposure times to high temperatures need to be minimized. Only a short spike to the maximum temperature and a brief dwell of 10 to 20 seconds is necessary for soldering.

Other reflow profile parameters must also be established. Heating rates, dwell times, and cooling rates are primarily a function of the flux chemistry being used. Parameters specified by the solder paste supplier should be closely adhered to.

INSPECTION

X-ray analysis tools should be available for reflow process monitoring and as a failure analysis tool. As previously mentioned, for 1.0mm pitch and finer BGAs, a 100 percent post-print 3D inspection is highly recommended for verifying that solder paste volumes are within process control limits. For 1.25mm pitch, a sample-based, post-print 3D inspection is recommended.

HANDLING OF ASSEMBLED PRODUCT

FBGA and CSP components are particularly susceptible to mechanical damage from mishandling. All processes where bending, flexing, impacting, or dropping—or where stresses could potentially be transferred to a component upon loading into tooling, test fixtures, sockets, cases, packaging for shipment, etc.—should be closely scrutinized and be included in routine preventative maintenance checks.

REWORK

Rework processes should be monitored as closely as, if not more closely then, any of the more automated processes. Moisture sensitivity guidelines also apply to a component mounted on PCB, as do the reflow guidelines. If the exposure time exceeds the floor life for a component that is going to be removed/replaced or is in the immediate vicinity to a component that is going to be removed/replaced, moisture absorbed by the component will need to be baked out prior to rework. Refer to Micron's technical note on moisture absorption, TN-00-01, for recommended bake-out conditions.

If a component is to be returned for failure analysis, it is imperative that proper handling procedures be followed prior to rework. Damage caused during removal of a component can mask the true cause of a failure and render a component useless.

SUMMARY

Reliable BGA assemblies can be manufactured with high yields if the necessary process controls or monitors are in place. Micron's recommendations will help ensure a higher degree of success in the manufacture of assemblies that contain these types of devices.

8000 S. Federal Way, P.O. Box 6, Boise, ID 83707-0006, Tel: 208-368-3900
E-mail: prodmktg@micron.com, Internet: http://www.micron.com, Customer Comment Line: 800-932-4992