AN79938

Design Guidelines for Cypress Ball Grid Array (BGA) Packaged Devices

Author: Cary Stubbles
Associated Project: No
Associated Part Family: All Cypress BGA Products
Software Version: N/A
Related Application Notes: None

AN79938 provides guidelines for designing, manufacturing, and handling Cypress's ball grid array (BGA) packages on printed circuit boards.

Contents

1 Introduction

This application note is for engineers who design and develop surface mount technology (SMT), printed circuit boards (PCB), or flexible printed circuits (FPC) for ball grid array (BGA) packaged devices.

These guidelines document the best practices for PCB/FPC design and assembly when using BGA-packaged devices. Because many factors influence manufacturing, performance, and reliability, validate these guidelines using your own product development and qualification processes. The following are the major influencing factors:

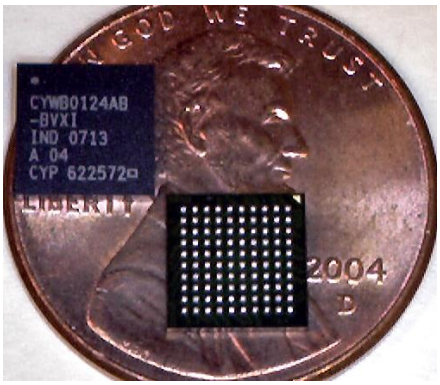
- PCB/FPC and solder/flux material
- Manufacturing equipment
- Application-specific requirements

2 Advantages of BGA Packages

Cypress BGA-packaged devices offer a near chip-size footprint with a high I/O count compared with standard leaded IC packages, such as SOP or QFP devices.

For example, the footprint of a 100-pin TQFP is 256 mm^2 (a 14.0 mm × 14.0 mm body with a 16.0 mm × 16.0 mm footprint, including package leads) while the equivalent very fine pitch BGA (VFBGA) with 100 balls (Figure 1) is only 36 mm^2 (a 6.0 mm × 6.0 mm body with no leads). As a result, you can reduce the component's PCB footprint by 85 percent.

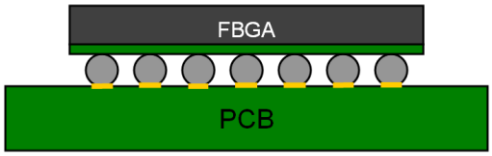
Figure 1. Example of a Cypress BGA Product



BGA packages provide a shorter conductor path between the die and the PCB/FPC because they use a substrate instead of a lead frame. The shorter path provides lower inductance, which results in better electrical performance, minimizing signal distortion in high-speed applications. The substrate also provides better heat dissipation by allowing the heat generated by the device to flow from the back of the die, through the solder balls, and to the PCB/FPC.

Assembling a BGA on to a PCB/FPC is more manageable when compared to a leaded IC with the same lead count. The majority of solder for reflow comes from the BGA's solder balls, which are factory-applied in precise form and size during BGA assembly. The solder balls also self-align to their corresponding land pads during reflow. Figure 2 shows a BGA package mounted on a PCB.

Figure 2. FBGA Package Mounted on a PCB



Cypress offers a wide range of BGA packages:

- Fine (FBGA)
- Very fine (VFBGA)
- Ultra fine (UFBGA)
- Plastic (PBGA)
- Heat slug (HSBGA)
- Flip chip (FCBGA)
- Heat slug flip chip (HFC-BGA)
- Heat sink cavity down (L2BGA)

Table 1. Cypress BGA Types

Type	Description	Ball Pitch (mm)
FBGA	BGA thickness ≥ 1.0 mm	0.75—1.27
VFBGA	1.0 mm > BGA thickness > 0.6 mm	0.5—0.80
UFBGA	BGA thickness ≤ 0.6 mm	0.5
PBGA	BGA thickness ≥ 2.0 mm	1.0—1.27
HSBGA	BGA with Heat Dissipation Slug	1.27
FCBGA	Flip Chip BGA	1.0
HFC-BGA	Flip Chip BGA with Heat Sink	1.0—1.27
L2BGA	Cavity Down BGA	1.27

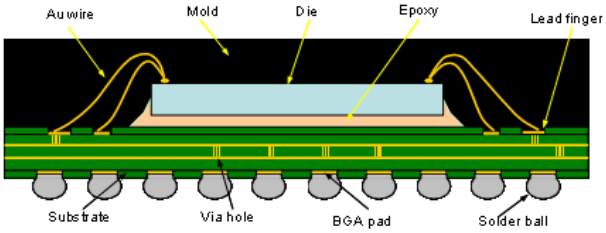
Cypress's BGA devices are available as 100 percent green and Pb-free in compliance with RoHS standards.

3 Cypress BGA Construction

3.1 Wire-Bonded BGA Packages

Cypress's wire-bonded BGA packages use a substrate made of bismaleimide-triazine (BT) epoxy glass laminate for the dielectric, and copper for the electrical contact from wire bonds to solder ball pads using plated through-holes. Depending on the complexity of the substrate, it can have two to eight layers. The outer layers of the substrate are protected from the environment by a coating of solder mask material. Figure 3 shows the internal construction of a wire-bonded BGA.

Figure 3. Wire-Bonded BGA X-Section View



The standard BGA packaging process begins by dispensing epoxy on the substrate where the die is to be attached. For more advanced substrate designs, an adhesive film is laminated to the back of the wafer instead of using die attach epoxy. Next, a pretested die is picked from a wafer that has been cut and mounted on a tape ring, and placed on the substrate. A UV or oven cure hardens the epoxy/film material before wire-bonding the die to the substrate.

To make electrical connections between the bond pads on the die and the substrate bond pads, bond wires (made of gold or copper) are attached between the die pad and its corresponding substrate bond pad. For Cypress BGA devices, the wire diameter ranges from 0.0008—0.001 inch (0.02—0.025 mm), depending on the electrical or mechanical requirements of the device.

After wire bonding, a transfer mold machine injects a plastic molding compound, heated between 175—185 °C, into a mold cavity around the wire-bonded die and substrate.

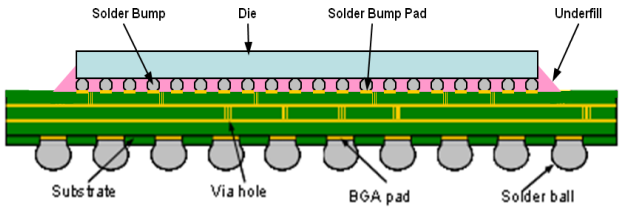
After the mold compound has cured, the molded substrates are removed from the mold machine and laser marked with the product part number, lot number, date code, and Cypress logo.

Next, individual devices are singulated from the manufacturing strip using either punch-singulation (a mechanical trimming machine) or saw-singulation (a spindle sawing machine with a circular blade) depending on the selected substrate.

3.2 Flip Chip BGA Packages

Inside a flip chip BGA package the die is connected to the substrate face down, which is flipped compared to the face up wire-bonded package. Figure 4 shows the internal construction of a flip chip BGA.

Figure 4. Bare Die Flip Chip BGA X-Section View



Wafer bumping is an essential process in flip chip packaging. The bumping process is an advanced packaging technique where bumps made of solder are formed on the wafers before being cut into the individual die.

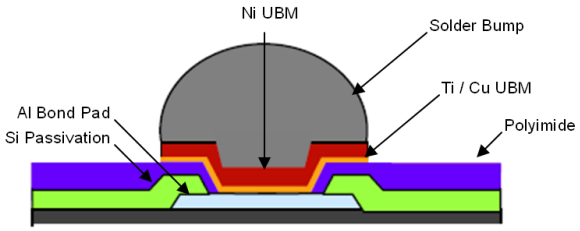
The wafer bumping process starts by applying a layer of polyimide over the wafer. This layer has openings to allow access to the aluminum pads on the die surface.

The first part of the under-bump metallization (UBM) of titanium and copper is then sputtered on to the wafer. The UBM is described as the adhesion layer because it adheres to both the bond pad metal and the surrounding passivation. The UBM provides a strong, low-stress mechanical and electrical connection.

The final UBM layer of nickel and solder is then applied directly over the aluminum pads on the die surface using photo-patterning and plating.

After solder plating, the UBM that is not covered by the solder plating is etched away. Finally, the wafer is reflowed, which causes the solder bumps to form, as shown in Figure 5.

Figure 5. Flip Chip Bump Structure



The flip chip BGA packaging process begins by dispensing flux on the substrate pads where the die is to be attached. Next, the bumped and pretested die is picked from a wafer that has been cut and mounted on a tape ring and placed on the substrate. The solder bumped die is attached to the substrate by a solder reflow process very similar to the process used to attach BGA balls to a PCB.

After the die is soldered in place, underfill is injected between the die and the substrate. Underfill is an epoxy designed to reduce stress in the solder joints connecting the die to the substrate. This stress is caused by the difference in thermal expansion between the die and the substrate and can cause broken or intermittent electrical connections.

3.3 Final BGA Packaging

Whether flip chip or wire-bonded, the final packaging process step, prior to electrical test, is solder ball placement. This process begins by screen printing or transferring solder flux on to the BGA pads. Most advanced solder ball attach machines use a pin matrix to transfer flux to the BGA pads. The flux provides adhesion sites for solder ball loading. Preformed solder balls are placed on the substrate and connect the BGA package to the PCB. The most common ball placement processing techniques are gravity transfer and vacuum transfer. Most automated ball placement machines accomplish all of these steps in a single platform. Table 2 gives the material properties for Cypress BGA solder balls.

Table 2. Material Properties of Cypress BGA Solder Balls

Properties	SAC105	SAC305	SAC405	Eutectic	
Ag (%)	1.0	3.0	4.0	-	2.0
Cu (%)	0.5	0.5	0.5	-	-
Sn (%)	98.5	96.5	95.5	63.0	62.0
Pb (%)	-	-	-	37.0	36.0
Liquidus	228°C	~217°C	~217°C	183°C	179°C

4 BGA PCB/FPC Layout Guidelines

4.1 Land Pad Pattern

PCB/ FPC fabrication uses two types of land pad patterns during surface mount assembly as shown in Figure 6.

- Non-solder mask defined (NSMD): The metal pad on the PCB (to which the package ball is attached) is smaller than the solder mask opening.
- Solder mask defined (SMD): The solder mask opening is smaller than the metal pad.

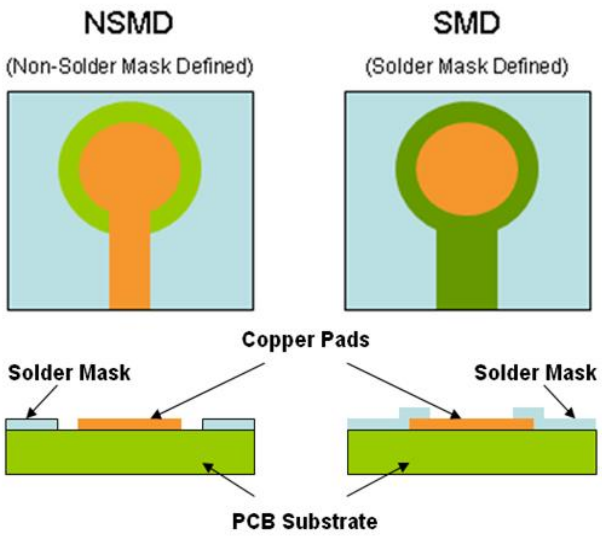
Typically, NSMD land pads are more reliable for board-level mechanical performance; however, NSMD pads are prone to peeling off during rework processing.

For board applications where underfill is required, Cypress recommends SMD land pads to minimize underfill issues. Eliminating the exposed laminate surrounding the solder ball land pads will ensure uniform flow and prevent voids.

Whether you choose NSMD or SMD land pads for your design will depend on your product reliability and manufacturing requirements.

Details of package outlines for Cypress BGA devices are available in each product datasheet.

Figure 6. NSMD and SMD Land Patterns



4.2 Board Material and Thickness

Standard glass/epoxy substrates are compatible with Cypress BGA devices. High-temperature FR4 laminate is preferred over standard FR4 for enhanced package reliability. This is because the coefficient of thermal expansion (CTE) of high-temperature FR4 laminate (12—16 ppm/°C) is lower than that of standard FR4 (14—18 ppm/°C) and is closer to that of the silicon (~2.5 ppm/°C).

The actual CTE of a board depends on its design. Numerous factors, such as the number of metal layers of the PCB, trace density, laminate material, component density, and the operating environment, affect thermal expansion. For enhanced reliability, the PCB laminate glass transition temperature should be above the operating range of the intended application (Tg > 170°C recommended).

Board thickness typically ranges from 0.016—0.093 inch (0.4—2.3 mm). Thinner boards are more flexible, resulting in improved thermal fatigue during thermal cycling in comparison to thicker boards. Similarly, thinner packages also contribute to improved package thermal fatigue performance at the board level.

4.3 Pad Finish

Cypress BGA devices are compatible with a variety of different pad finishes. Nickel-based finishes are popular due to higher substrate shelf life, improved corrosion resistance, better thermal stability of solder joints, and ability to be reworked. Among nickel-based finishes, Electroless Ni Immersion Gold (ENIG) has become popular for lead-free or RoHS applications.

Other finish options include immersion Au, immersion Ag, organic solder preserves (OSP), and electrolytic Ni-Au. Among these, the most promising is electrolytic Ni-Au.

4.4 Pad Geometry

For optimum board-level reliability, the recommended ratio between the solder ball pad and the PCB/FPC pad size is 1:1. The PCB/FPC pad design is to be the same diameter as the BGA package solder ball pad.

Figure 7. BGA SMD Solder Ball Pad Design

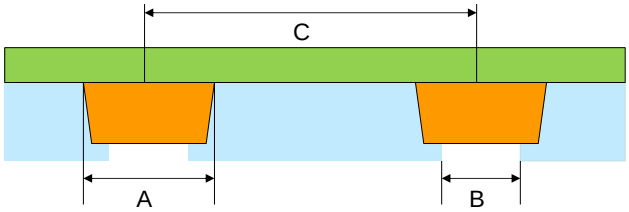


Table 3: BGA SMD Solder Ball Pad Design (mm)

Solder Ball Diameter	0.25 0.30	0.40	0.45	0.50	0.60	0.63	0.80
Ball Land Pad (A)	0.35	0.50	0.55	0.55	0.55	0.58	0.75
Solder Mask Opening (B)	0.25	0.35	0.40	0.40	0.40	0.48	0.60
Ball Pitch (C)	Refer to Associated Device Datasheet						

Figure 8. BGA NSMD Solder Ball Pad Design

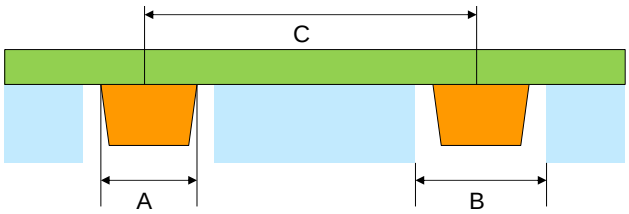


Table 4: BGA NSMD Solder Ball Pad Design (mm)

165 FBGA 13 × 15 or 15 × 17 mm	
Solder Ball Diameter	0.50
Ball Land Pad (A)	0.40
Solder Mask Opening (B)	0.55
Ball Pitch (C)	1.00

Cypress recommends that customers contact Cypress Customer Support to determine the BGA solder ball pad design.

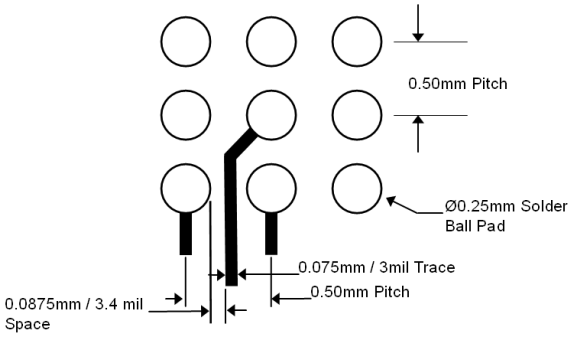
4.5 Via Layout and Dimension

The size and layout of vias and via capture pads affect the amount of space available for escape routing (a method used to route signals from one component to another on the PCB/FPC). For 0.5-mm pitch devices, a micro-via in the center of the surface land pad (via-in-pad technology) is recommended. The micro-via drill hole diameter should be equal to or less than 0.150 mm. The drill hole should be plugged and planarized to create a flat surface, which prevents the solder from wicking through the holes during board-level reflow.

4.6 Signal Line Space and Trace Width

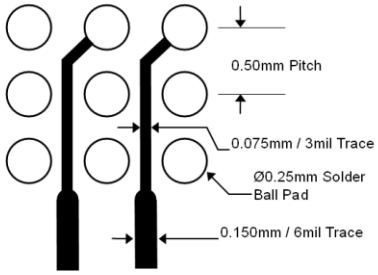
The ability to perform escape routing is defined by the width of the traces and minimum space required between the traces. For 0.5-mm pitch devices, 0.075-mm trace width is the minimum. Only one trace can be routed between SMT land pads or via capture pads. The recommended line space and trace width are shown in Figure 9.

Figure 9. Signal Line Space and Trace Width



0.075-mm traces should only be used in BGA areas that have routing space constraints. Necking down traces can be used to increase manufacturability and ensure better yields. After the trace exits the BGA land pads, return to standard trace and space geometry as shown in Figure 10.

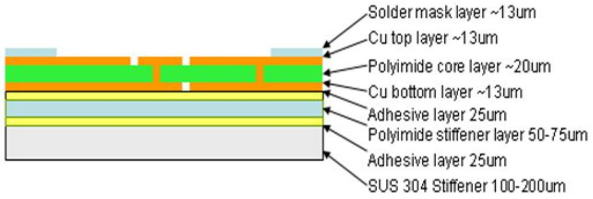
Figure 10. Necking Down Traces



4.7 FPC Stiffener

Stainless steel (SUS304) is a recommended stiffener to enhance the thermal and mechanical strength of BGA packages in FPC applications. However, you must evaluate the reflow process to determine whether a SUS304 stiffener will hurt solder joint quality due to warpage. When you use a metallic stiffener in capacitive sensing applications, you must consider the additional parasitic capacitance of the stiffener.

Figure 11. Typical FPC Structures



Cypress recommends that customers contact Cypress Customer Support for FPC layout design rules and review.

5 BGA SMT Guidelines

Cypress's SMT guidelines will help you achieve the highest possible yield, performance, and reliability for each step in your BGA SMT assembly process:

- Stencil design
- Solder paste
- Package placement
- Reflow
- Head-in-Pillow
- Underfill
- SMT rework
- Component replacement

5.1 Stencil Design

Follow the IPC-7525 Stencil Design Guidelines standard for all assemblies. You must use high-quality stencils to achieve quality solder paste printing. The thickness of the stencil determines the amount of solder paste deposited on to the PCB land pads. Too much paste causes solder bridging during reflow, while too little paste results in poor or open connections. You can achieve better solder stencil performance by using laser-cut or electro-formed stencils, instead of chemically etched stencils.

To prevent unbalanced solder height, the solder stencil openings should be identical for all land pads in the BGA array. It is best to use a design that results in good paste release. To accomplish this, the aperture size and pitch should be 1:1 ratio with the PCB land pad. Typical BGA stencils are designed with round apertures; however, Cypress recommends square apertures for better paste release. To improve paste release, a positive taper (5° angle) with a top opening smaller than the bottom can be used. Also, maintain a diameter-to-stencil thickness ratio of at least 3:1, larger openings providing better print quality. For example, the minimum aperture opening in a 0.006-inch thick stencil is 0.018 inch. Printing a small amount of paste on to the solder mask that surrounds the land pad has not shown to decrease yield or reliability.

Periodic stencil cleaning is also required for consistent paste printing. Avoid manual cleaning because it can cause dents and damage to the stencil and degrade paste-printing quality.

5.2 Solder Paste

The solder paste printing process transfers the solder paste to the PCB/FPC by squeezing it over the stencil. Cypress recommends that auto stencil underside cleaning is done periodically during screen printing. This improves solder paste volume uniformity and release.

BGA devices should be mounted to the PCB/FPC with the maximum device standoff possible under the BGA, by using the maximum allowable solder paste volume. Solder paste volume is the best predictor of finished board quality and reliability. Perform a thorough inspection to ensure solder volume uniformity before package placement.

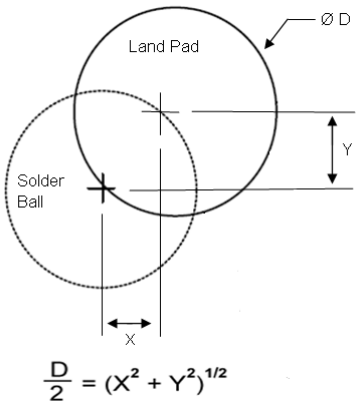
Do not use solder paste with active or acid-based flux. The residue from these fluxes is difficult to remove and can degrade the solder joints over time because of the low clearance under a mounted BGA package.

5.3 Package Placement

Good setup and process control are required when pick-and-place machines are used with BGA devices to avoid damaging the thin structures in the package. A typical surface mount placement equipment uses either a vision system that optically recognizes and positions the component or a mechanical system that physically aligns the component. Either method is acceptable since BGA packages self-align during solder reflow.

Half-on-pad is a rule that defines the placement accuracy as no less than 50 percent of the solder ball aligned with the PCB/FPC land pad as shown in Figure 12.

Figure 12: Half-On-Pad Rule



Avoid overdriving the device into the solder paste by managing the placement height (Z) of the device on the pick-and-place equipment. Optimum height is one-half the printed solder paste height. Maintaining board flatness (coplanarity) is important in keeping the Z-height under control. Thin board technologies, such as FPC, require more caution.

5.4 Reflow

Cypress BGA devices are available with different solder ball compositions. For example, if the package is designed for a handheld application, the solder ball composition will most likely be SAC105, whereas if the package is designed for networking equipment, the solder ball composition will be SAC305 or SAC405. Cypress also offers some legacy devices with non–Pb-free solder balls, as shown in Table 2. Cypress recommends that customers refer to the corresponding Package Material Declaration or contact Cypress Customer Support to determine the actual solder ball composition.

The reflow furnace should have a nitrogen purge with oxygen content below 50 ppm.

Actual reflow temperatures are based on thermal loading effect measurements within the furnace. Other factors include the complexity of the components on the board and the board size and thickness.

For Pb-free solder, the reflow profile is critical. SAC305 and SAC405 solder melt at ~217 °C. As a result, the reflow peak temperature, measured at the solder joint, should be 15—20 °C higher than melting temperature. See Table 2 for more details about solder ball melting point.

Higher reflow temperatures than the qualified temperature can cause delamination within the package.

BGA packages are sensitive to moisture-induced stress. To avoid damage during board-level reflow follow the guidelines listed in IPC/JEDEC J-STD-020D.1.

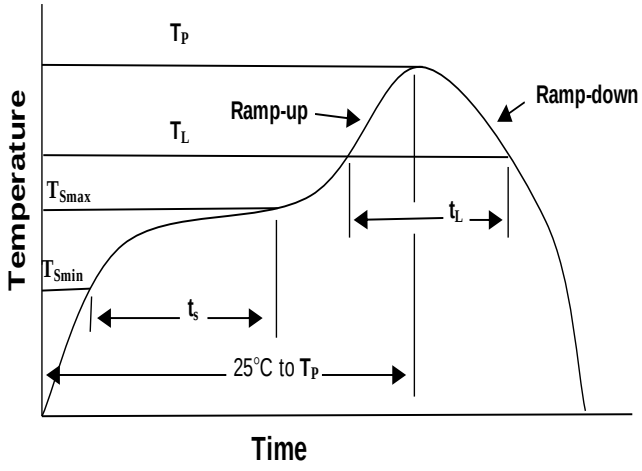
Contact your solder manufacturer for its recommended reflow profile parameters.

Cypress's Pb-free BGA devices are qualified at 260°C reflow with Moisture Sensitivity Level 3. Cypress's non–Pb-free BGA devices are qualified at 220 °C reflow with Moisture Sensitivity Level 3. The user maximum temperature profile and critical reflow parameters are shown in Table 5 and Figure 13.

Table 5: Reflow Profiles (per JEDEC J-STD-020D.1)

Process Step		Pb-Free Solder	Non–Pb-Free Solder (Eutectic Sn-Pb)
Pre-heat	Temperature min (T_{Smin})	150 °C	100 °C
	Temperature max (T_{Smax})	200 °C	150 °C
	Time (t_S) (T_{Smin} to T_{Smax})	60-120 seconds	60-120 seconds
Reflow	Ramp-up rate (T_L to T_P)	3 °C/second max	3 °C/second max
	Liquidus temperature (T_L)	217 °C	183 °C
	Time (t_L)	60-150 seconds	60-150 seconds
Peak temperature (T_P)		See Table 8	See Table 7
Time within 5°C of peak (T_P)		30 seconds	20 seconds
Ramp-down rate (T_P to T_L)		6 °C/second max	6 °C/second max
Time 25 °C to peak temperature (T_P)		8 min max	6 min max
Maximum number of reflow cycles		3	3

Figure 13. Reflow Temperature Profile

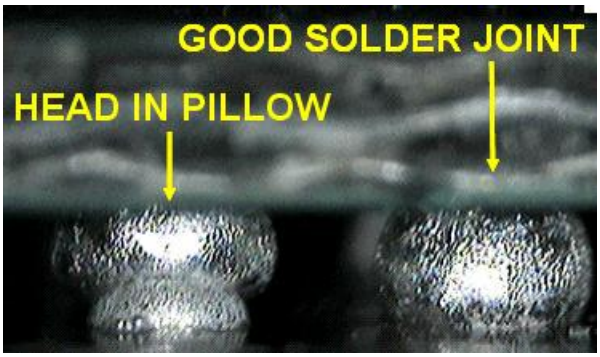


A good quality solder joint is formed when all of the solder paste on the PCB land pad joins the solder ball of the BGA. The surface of the joint should be smooth and the shape symmetrical. Voids in the solder joint after reflow can occur if the reflow profile is not properly tuned.

Solder joint inspection can be performed with an x-ray to monitor defects such as bridging, opens, and voids.

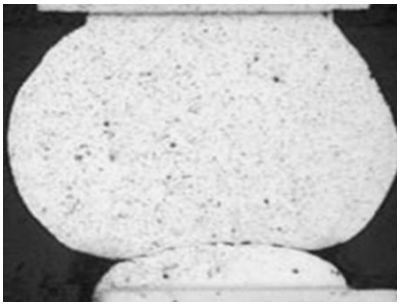
5.5 Head-In-Pillow

Figure 14. Post Reflow Solder Joints



Head-in-pillow is a solder joint defect where the solder paste wets to the PCB land pad, but does not fully wet to the BGA solder ball. Although mechanical contact is made between the land pad and the BGA solder ball, the solder joint is not formed at a metallurgical level as shown in Figure 15.

Figure 15. Cross Section of Head-in-Pillow Defect

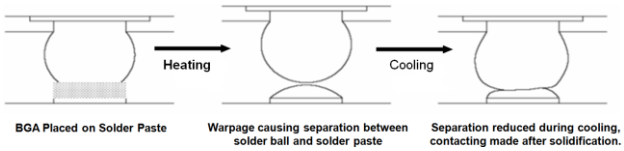


Because the solder joint lacks strength, components with head-in-pillow defects may fail with very little mechanical or thermal stress. Unfortunately, this defect is usually not found during functional testing but rather in the field after the assembly is exposed to physical or thermal stress.

Head-in-pillow defects are caused by the following chain of events:

1. During pick-and-place, the BGA solder balls are depressed into the solder paste.
2. As heating is initiated the BGA and/or PCB start to flex or warp.
3. The BGA's solder balls separate from the solder paste as a result of the warpage.
4. The unprotected solder balls then form a new oxide layer.
5. As heating continues, it is possible for the warpage to flatten out and allow the solder balls to regain contact with the solder paste.
6. When the solder reaches the liquidus phase, there is not sufficient fluxing activity to break down the new oxide layer.
7. The final result is head-in-pillow defects as shown in Figure 16.

Figure 16. Head-in-Pillow Failure Mechanism



Several things can prevent head-in-pillow defects:

1. BGA component and PCB storage conditions

- When the BGA and/or PCB absorb moisture, the warpage during reflow will be more severe. If moisture abortion is a concern, bake PCBs based on the supplier's recommendations and follow the BGA requirements called out in the BGA Handling during Packing, Shipping, and SMT section of this document.

2. Optimize the print parameter for uniform and maximum solder paste volume.

- Slow down print speed to 25 mm/s
- Increase pressure to 10 kg
- Slow down separation speed to 0.75 mm/s
- Implement good stencil cleaning practice
- Calculate the process capability (CPK) for the paste volume to understand paste print quality

3. Solder paste selection

- Solder paste can be a major contributor to head-in-pillow. Select a solder paste with an enhanced oxidation barrier, longer tack life, and better wetting performance.
- The electronics industry is moving towards halogen-free. When halogens are used in solder paste flux, it enhances the activity without reducing reliability. If a halogen-free paste is used, the reflow profile must be within the solder paste manufacturer's guidelines and a nitrogen purge implement during reflow.
- No-clean solder pastes typically perform better than water-soluble pastes, with regard to head-in-pillow. No-clean chemistries are generally rosin-based materials. Water-soluble pastes contain other chemicals and are not as effective as rosins in preventing re-oxidation and do not hold up well in elongated reflow profiles.

4. Solder paste handling

- Storage of solder paste is best achieved by refrigeration in the range of 0 to 10°C. The material should be placed in storage upon receipt. The solder paste should never be stored at room temperature (19 to 25 °C) for an extended amount of time.
- The solder paste should be allowed to reach room temperature prior to use. This typically takes 3 to 4 hours, depending on packaging size.
- During solder paste printing, it is best to maintain the temperature inside the printer between 22°C and 29 °C and 40% and 60% RH.

5. Stencil design

- Stencil should be laser-cut.
- Stencil thickness should be 0.005—0.006 inch.
- A typical stencil aperture should be 1:1 ratio with the PCB land pad pattern. Depending on the solder paste flux activity the aperture may need to be increased to break down the oxide film that is created on the solder balls during SMT reflow. An increase of up to 25% is acceptable as long as there is no bridging.

6. Verify the pick and place process

- Ensure that the placement registration and force are sufficient and that the parts are not tilted after they are mounted on the PCB.

7. Optimize the reflow process.

The reflow profile parameters have a significant impact on head-in-pillow. The ideal situation is for all the components on the board to reflow at the same time. However, PCBs during reflow have some level of temperature variation between different components on the board and within a single component. If there is a temperature delta between the two sides of the BGA, one side will reflow sooner than the other causing the BGA to tilt. In this case, oxidation will increase on the balls that separate from the solder paste. As the package is pulled back to flat by the paste, it wets the solder balls. How quickly it pulls back depends on the wetting force of the paste. Head-in-pillow will develop in the areas where the flux is not active enough to reduce the oxides on the solder ball surface.

- Ensure the reflow furnace has a nitrogen atmosphere with oxygen content below 50 ppm. Using a nitrogen purge during reflow has a large impact in preventing solder ball surface oxidation.
- Verify that the reflow profile conditions meet the solder paste manufacturer's recommendations for a BGA package.
- A fast ramp rate increases solder paste slumping.
- A linear ramp profile prevents exhausting the flux activity prior to reaching liquidus. Flux activity will be reduced by a long preheat time. Oxidation will also occur during preheat and soak stages of the profile.
- Ensure that the peak temperature is as low as possible and the TAL as short as possible. Peak temperature affects the amount of warpage whereas TAL has an impact on the amount of oxides that form on the molten solder surface.

5.6 Underfill

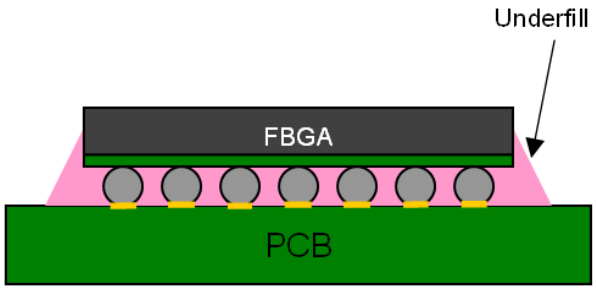
Underfill is not typically used at the board level. However, in some applications mechanical stresses can induce early failures in board-level solder joints. These failures are often related to CTE mismatch between the BGA package and board.

Applying underfill will encapsulate the solder joints and reduce the CTE mismatch between the BGA package and the board. The use of underfill also protects the solder joints from unwanted package moisture, ionic contamination, radiation, solder bump extrusion, thermal and mechanical shock, and vibration, which can all cause reliability failures for SMT assemblies.

Underfill is often considered undesirable because it adds manufacturing steps and cost. However, significant improvement in product reliability can offset these negative factors.

There are significant differences in performance between underfill types and suppliers. Cypress strongly recommends using optimized underfill material derived from an actual DOE. Cypress will review your underfill material selection and make recommendations. Contact Cypress Customer Support.

Figure 17. BGA Mounted to a PCB with Underfill



5.6.1 Rigid Multilayered FR4 PCB

Underfill is recommended for Cypress BGA devices used in handheld (cell phones and PDAs), automotive, and military applications where solder joint performance under drop and bend tests, and thermal cycling is a reliability concern.

5.6.2 Flex PC

Underfill is required for BGAs mounted on FPC regardless of the application.

5.7 SMT Rework

Use a controlled and qualified process when you rework BGA devices to prevent mechanical and ESD damage.

Prior to attempting any rework, you must ensure that the assembly is free of moisture, which can damage the board or other components.

The ability to rework components with underfill depends on the characteristics of the underfill. There are underfills in the market that can be reworked using heat and proper cleaning.

An accurate thermal profile needs to be established for the component removal process. This will determine the maximum temperature and duration for the component and PCB. A profile should be created for each board and component requiring removal. The typical profile should provide a peak temperature between 235 and 250°C for a maximum of 30 seconds, for a Pb-free solder, or between 210 and 220°C for a maximum of 30 seconds, for a non–Pb-free solder. It is always best to consult with your equipment and paste manufacturers for their recommended profile.

It is also important to ensure that the component and the board are not overheated and that all solder balls are reflowed on the BGA prior to removal.

Preheating of the PCB to a minimum of 85 °C assembly is recommended. Some of the advantages are:

- Reduced heating time when using a rework head
- Reduced temperature spread, bringing the baseline temperature closer to the reflow temperature
- Reduced PCB warpage during processing
- Less risk of causing solder shorts on adjacent components

With regard to equipment and tools, hand-held as well as automatic hot gas rework systems with vacuum suction have been developed for BGA removal. Most rework systems use hot air to locally heat the BGA device and IR to locally heat the PCB, simultaneously reflowing the solder joint for BGA removal.

Reusing a BGA that has been removed from a board is not advised. Although it is possible to re-ball a BGA, it is not recommended to reuse a Cypress BGA due to reliability concerns.

5.8 Component Replacement

Clean the rework site with a solvent prior to replacing the component in order to remove any surface contamination. For solder paste/flux application, use a mini-stencil with a squeegee as wide as the stencil. Align the apertures with the solder pads under 50x to 100x magnification. The placement machine should allow fine adjustment in the X, Y, and rotation axes.

Follow the solder paste manufacturer's recommendation for reflow profile, and make sure the maximum temperature does not exceed the package qualification level. Reflow profiles developed for initial placement or rework can be used. A three-stage (ramp up, hold, ramp down) profile may result in smaller temperature distribution across the site.

6 Cypress BGA Reliability Test Data

6.1 Component Level Reliability Test

Cypress performs all component-level reliability tests according to internal specification 25-00112. Cypress qualification procedures and requirements comply with various industry standards including JEDEC/IPC and MIL-STD-883.

Major stress tests for component-level reliability are listed in Table 6.

Table 6. BGA Qualification Stress Tests

Test Method	Test Conditions	MSL	Duration
Temp Cycles Cond C	-65°C to +150°C	MSL 3	500 cycles
Temp Cycles Cond B	-55°C to +125°C	MSL 3	1000 cycles
HAST	130°C/85% RH	MSL 3	128 hours
Pressure Cook Test	121°C/100% RH	MSL 3	168 hours
High Temp Storage	150°C	NA	1000 hours

Detailed qualification reports for specific Cypress BGA products are available online at www.cypress.com or through your local Cypress sales representative.

6.2 Board-Level Reliability Test

Board-level reliability of BGA devices can be heavily affected by the board's material, design parameters, and thickness.

Typically, BGA-packaged devices are highly reliable when assembled on rigid PCBs. The volume of solder paste dispensed when mounting a BGA package on to a PCB is critical for the proper package standoff height and for acceptable solder joints of the BGA package. This will improve board-level reliability.

The reliability of BGA products on FPC is not well known in the industry. However, when you optimize the FPC structure and SMT process, the performance can be sufficient to meet the application requirements for typical handheld products, such as cell phones. Use a SUS304 stainless steel stiffener to improve mechanical strength of BGA devices mounted on an FPC.

Contact your local Cypress sales representative for more detailed information.

7 Package Thermal Resistance

Cypress lists the thermal resistance of each BGA product in the company's associated device datasheet.

Typical simulation conditions for thermal resistance require you to mount the device to either a two-layer or four-layer PCB, depending on the application of the product.

Theta J_A, thermal resistance of junction-to-ambient (Θ_{JA}) specifies the thermal resistance between a die in its package and the ambient air surrounding the packaged part when mounted to a PCB or FPC, according to JEDEC EIA/JESD51-2.

8 BGA Handling during Packing, Shipping, and SMT

The moisture sensitivity level (MSL) of a component indicates its floor life and storage conditions after the original container is opened. All Cypress BGA products are classified as MSL3 and support the reflow peak temperature called out in Table 7 and Table 8, according to JEDEC standard J-STD-020.

Table 7. Non–Pb-free Eutectic Process – Peak Temperature Classification

Package Thickness (mm)	Volume (mm^3)	
	< 350	≥ 350
< 2.5	235 °C	220 °C
≥ 2.5	220 °C	220 °C

Table 8. Pb-Free Process – Peak Temperature Classification

Package Thickness (mm)	Volume (mm^3)		
	< 350	350—2000	> 2000
< 1.6	260 °C	260 °C	260 °C
1.6—2.5	260 °C	250 °C	245 °C
> 2.5	250 °C	245 °C	245 °C

Note All temperatures refer to the top side of the package, measured on the package body surface.

The use of a 5%-10%-60% Humidity Indicator Card is required by IPC/JEDEC J-STD-033 during the dry packing process. Units shall undergo baking after the 10 percent relative humidity spot turns a shade of pink.

The shelf life in a sealed container is 12 months at < 40°C and < 90% relative humidity from the bag-seal date. After the container is opened, devices that will be subjected to reflow solder or other high temperature process must be:

- Mounted within one year if exposed to conditions of equal or less than 30 °C / 60% RH.
- Stored at equal or less than 10% RH.

If any of the following conditions apply, the device requires baking before mounting:

- Humidity indicator card is equal or greater than 10% when read at 23 °C +/- 5 °C.
- Not mounted on the board in one year in equal or less than 30 °C / 60% RH.
- Not stored in equal or less than 10% RH.

If baking is required, devices must be baked for 24 hours at 125 °C +5/-0 °C. Use baking trays rated at greater than 125 °C.

After the board-level assembly, moisture can be trapped between the solder balls underneath the package, which can result in pin-to-pin leakage. Take special care to avoid this.

9 References

1. Applications Notes on Surface Mount Assembly of Amkor Technology BGA Packages – Amkor Technology, September 1999, http://www.amkor.com/index.cfm?objectid=A136555A-C286-2A98-A0CC7ACF6025BDAC
2. Chapter 17: Printed Circuits Handbook (McGraw Hill Handbooks) – Sixth Edition, Darwin Edwards, Clyde F. Coombs, Jr., 2008
3. Head-in-Pillow BGA Defects – AIM Manufacturing and Distribution Worldwide, http://www.aimsolder.com/Portals/0/PDFs/technical/Head-in-pillow%20BGA%20defects.pdf
4. Addressing the Challenges of Head-In-Pillow Defects in Electronic Assembly – Scalzo Mario, IPC APEX EXPO Technical Conference 2010
5. Cypress application notes:

   AN5093 - Cypress MoBL® Dual-Port 100-Ball VFBGA Printed Circuit Board (PCB) Layout Guidelines

   AN15456 - Guide to a Successful EZ-USB® FX2LP™ Hardware Design

Document History

Document Title: AN79938 - Design Guidelines for Cypress Ball Grid Array (BGA) Packaged Devices

Document Number: 001-79938

Revision	ECN	Orig. of Change	Submission Date	Description of Change
**	3711070	CS	08/13/2012	New Application Note
*A	3869842	CS	01/15/2013	Added 0.25 mm Solder Ball Diameter to Table 3
*B	4199297	CS	11/22/2013	Added Ag values to Table 2. Updated Figures 7 and 8 Corrected items called out in the BGA SMT Guidelines section Fixed stencil aperture ratio to match 3:1 example Updated Table 5 and 8 and rewrote the head-in-pillow section.
*C	4346105	CS	04/14/2014	Added Max number of reflow cycles to Table 5.
*D	4883427	DEVM	08/13/2015	Updated template.
*E	5713733	AESATMP9	04/26/2017	Updated logo and copyright.

Worldwide Sales and Design Support

Cypress maintains a worldwide network of offices, solution centers, manufacturer's representatives, and distributors. To find the office closest to you, visit us at Cypress Locations.

Products

ARM® Cortex® Microcontrollers	cypress.com/arm
Automotive	cypress.com/automotive
Clocks & Buffers	cypress.com/clocks
Interface	cypress.com/interface
Internet of Things	cypress.com/iot
Memory	cypress.com/memory
Microcontrollers	cypress.com/mcu
PSoC	cypress.com/psoc
Power Management ICs	cypress.com/pmic
Touch Sensing	cypress.com/touch
USB Controllers	cypress.com/usb
Wireless Connectivity	cypress.com/wireless

PSoC® Solutions

PSoC 1 | PSoC 3 | PSoC 4 | PSoC 5LP | PSoC 6

Cypress Developer Community

Forums | WICED IOT Forums | Projects | Videos | Blogs | Training | Components

Technical Support

cypress.com/support

Cypress Semiconductor
198 Champion Court
San Jose, CA 95134-1709