XAPP426 (v1.3) March 3, 2006

Implementing Xilinx Flip-Chip BGA Packages

Summary

Xilinx flip-chip BGA package is offered for Xilinx high-performance FPGA products. Unlike traditional packaging in which the die is attached to the substrate face up and the connection is made by using wire, the solder bumped die in flip-chip BGA is flipped over and placed face down, with the conductive bumps connecting directly to the matching metal pads on the laminate substrate.

In this application note, guidelines on board design rules as well as board assembly parameters, rework process, and thermal management will be discussed. Note that the reflow and rework guidelines contained in this application note is applicable to standard packages only. For reflow and rework guidelines on Pb-free packages, refer to XAPP427, http://www.xilinx.com/bvdocs/appnotes/xapp427.pdf.

Introduction

Xilinx Flip-Chip Packages are assembled on high-density, multi-layer organic laminate substrates.

Since this package is used exclusively in high performance products, it is critical that the users know how to manage the implementation of Flip-Chip BGA packages to prevent costly replacements.

Package Construction

Figure 1 and Figure 2 show cross section views of the package constructions. Note that two types of lids are used to assemble flip-chip BGA packages, Type I Lids as shown in Figure 1 and Type II Lids as shown in Figure 2, depending on package type.

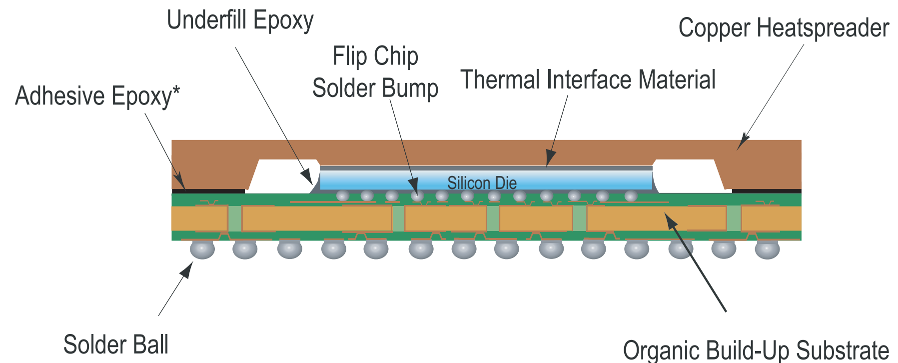

Figure 1: **Package Construction with Type I Lid**

© 2002-2006 Xilinx, Inc. All rights reserved. All Xilinx trademarks, registered trademarks, patents, and further disclaimers are as listed at http://www.xilinx.com/legal.htm. All other trademarks and registered trademarks are the property of their respective owners. All specifications are subject to change without notice.

NOTICE OF DISCLAIMER: Xilinx is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Xilinx makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Xilinx expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.

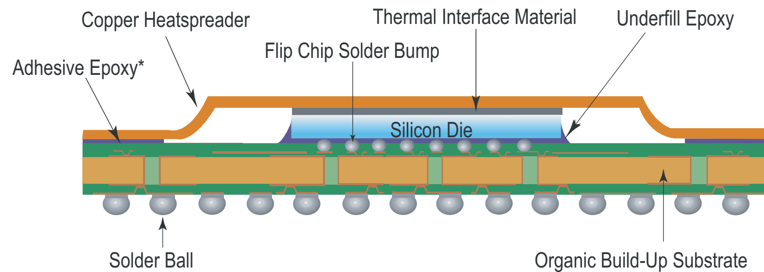

Figure 2: **Package Construction with Type II Lid**

*Xilinx flip-chip packages are not hermetically sealed and exposure to cleaning solvents/ chemicals or excessive moisture during boards assembly could pose serious package reliability concerns. Small vents are kept by design between the heatspreader (lid) and the organic substrate to allow for outgassing and moisture evaporation. Solvents or other corrosive chemicals could seep through these vents and attack the organic materials and components inside the package and hence are strongly discouraged during board assembly of Xilinx flip-chip BGA packages.

Recommended PCB Design Rules

For Xilinx BGA packages, NSMD (Non Solder Mask Defined) pads on the board are suggested. This allows a clearance between the land metal (diameter L) and the solder mask opening (diameter M) as shown in Figure 3. The space between the NSMD pad and the solder mask, and the actual signal trace widths depends on the capability of the PCB vendor. The cost of the PCB is higher when the line widths and spaces are tighter.

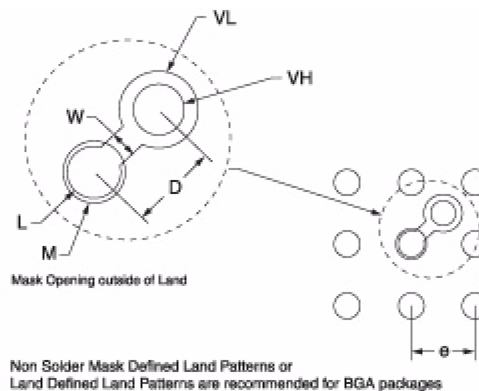

Figure 3: **Suggested Board Layout of Soldered Pads for BGA**

Table 1: **Recommended PCB Design Rules**

	1.00 mm Pitch Package	1.27 mm Pitch Package
Component Land Pad Diameter (SMD)[2]	0.48	0.61
Solder Land (L) Diameter	0.45	0.56
Opening in Solder Mask (M) Diameter	0.55	0.66
Line Width Between Via and Land (w)	0.13	0.203
Distance Between Via and Land (D)	0.70	0.90
Via Land Diameter (VL)	0.61	0.65
Through Hole (VH) Diameter	0.300	0.356

Notes:

1. 3 x 3 matrix for illustration only. One land pad shown with via connection.
2. Component land pad diameter refers to the pad opening on the component side (solder mask defined).

Assembling Flip-Chip BGAs

The Xilinx Flip-Chip BGAs conform to JEDEC body sizes and footprint standards. These packages follow the EIA moisture level classification for plastic surface mount components (PSMC). Standard surface mount assembly process should be used with consideration for the slightly higher thermal mass for these packages.

Like other SMT components, flip-chip BGA assembly involves the following process: screen printing, solder reflow, post reflow washing. The following will serve as a guideline on how to assemble flip-chip BGAs onto PCBs.

Screen Printing Machine Parameters

Below is an example of the parameters that were used for the screen printing process. Note that these may not be optimized parameters. Optimized parameters may depend on user's applications and setup.

- Equipment: MPM Ultraprint 2000
- Squeegee Type: Metal
- Squeegee Angle: 45°
- Squeegee Pressure: 24 lbs/sq. in.
- Squeegee Speed: 0.7 in/sec
- Print Cycle: One pass
- Stencil Snap Off: 0.10 inches
- Stencil Lift Off Speed: Slow

Screen Printing Process Parameters

- Solder Paste: Alpha Metals WS609 (Water Soluble)
- Stencil Aperture: 0.0177 inches Diameter
- Stencil Thickness: 0.006 inches
- Aperture Creation: Laser cut

It is highly recommended to use either a no-clean solder paste or a water soluble solder paste. If cleaning is required, then a water soluble solder paste should be used.

Reflow Profiling

An optimized profile is paramount in achieving successful reflow result. A good starting point is to refer to the solder paste manufacturer's suggested reflow profile. However, solder paste manufacturers only supply the basic time/temperature duration information. To get an optimized reflow, components and board characteristics should dictate the maximum temperature and proper ramp rate.

Profiles should be established for all new board designs using thermocouples at multiple locations on the component (top, bottom, and corners — see Figure 5 in the Appendix). In addition, if there are mixture of devices on the board, then the profile should be checked at different locations on the board to ensure that the minimum reflow temperature is reached to reflow the larger components and at the same time, the temperature does not exceed the threshold temperature that may damage the smaller, heat sensitive components. The minimum reflow temperature is the ideal thermal level at which the solder balls can be wetted to form the solder joints.

The solder paste manufacturers usually provide this information and it is typically 15-20°C above the solder's melting point. For eutectic (Sn63Pb37) solder, it is around 205-215°C.

It is critical to keep the temperature gradient across the board as minimal as possible (maintain less than 10×C) to prevent warpage of the components and the board. This is accomplished by using a slower rate in the warm-up and preheating stages. A heating rate of less than 1×C/sec during the initial stage, in combination with a heating rate of not more than 3×C/sec throughout the rest of the profile is recommended.

Aside from the board, it is also important to minimize the temperature gradient on the component, between top surface and bottom side, especially during the cool down phase. In fact, cooling is a crucial part of the reflow process and must be optimized accordingly. While a slow cooling rate may result in high assembly yields, it could lead to formation of thick intermetallic layers with large grain size; thereby, reducing the solder joint strength. On the other hand, faster cooling rate leads to smaller solder joint grain size and hence resulting in higher solder joint fatigue resistance. However, overly aggressive cooling on stiff packages with large thermal mass such as flip-chip BGAs may lead to cracking or package warpage, caused by the differential cooling effects between the top surface and bottom side of the component and between the component and the PCB materials.

The key is to have an optimized cooling with minimal temperature differential between the top surface of the package and the solder joint area. The temperature differential between the top surface of the component and the solder joint area should be as minimal as possible, preferably below 7° C during the critical region of the cool down phase of the reflow process. This critical region occurs at the phase in which the balls are not completely solidified to the board yet, usually between the 180°C and down to 160°C range. The best solution may be to divide the cooling section into multiple zones, with each zone operating at different temperatures to efficiently cool the parts. For a graphical representation of the typical reflow conditions for BGA, see Figure 6 in the appendix.

Post Reflow Washing

Most major PCB assembly subcontractors today have successfully developed the no-clean process in which post assembly washing is not required. That would be an ideal process. If cleaning is required as part of the process, then it is recommended to use a water soluble paste and then wash with deionized water in a washer, such as a Westek Triton III at 140°F-145°F.

Cleaning solutions or solvents are not recommended as some cleaning solutions may contain chemicals that could attack the heatspreader adhesive, thermal compound, or the components inside the package.

Reworking Flip-Chip BGAs

Since the devices packaged in a flip-chip BGA package are typically high performance and high priced devices, it is essential that proper procedures are followed to ensure successful rework of flip-chip BGAs.

Prebaking

As the printed circuit board and the BGA packages are quite moisture sensitive, one should always bake the PCBs and the BGA devices prior to any rework operations. The recommended temperature and duration is 125°C for at least four hours.

BGA Removal

An accurate thermal profile needs to be established for the component removal process. This will determine the exposure duration and the maximum component/board temperatures.The profile should be adapted to each board and component to be removed. Although the typical profile should provide a peak temperature between 205 to 215°C (at the solder joint) for a maximum of 75 seconds, it is best, however, to consult with equipment manufacturer for the recommended profile.

Research has also indicated that a short delta T and a short dwell time above 183°C are preferred to minimize intermettallic growth and control board warpage. Also of importance is a need to assure that the component and the board are not overheated, and that all balls are reflowed on the specific component being removed. In general, preheat the entire board to a minimum of 85°C to avoid large temperature differentials and potential board warpage.

In terms of the equipment and tools available, automatic hot gas rework systems with vacuum suction are recommended. The nozzle should be designed such that most of the heat is applied at the solder joint area and not on the package. Excess heat can cause the lid attach epoxy to soften, which can cause the lid to come off. Apply heat from the topside using the rework profile developed (ramp the temperature for 45-60 secs with a maximum temperature between 205-215°C). When the solder balls are fully liquidus, remove the component using a vacuum tip. Do not attempt to remove partially reflowed component from a board by prying it off, as this would likely damage the component and can cause the lid to come off. **Note:** To avoid package delamination, the temperature at the top of the package must not exceed 225°C (245°C, 250°C, or 260°C for Pb-free flip-chip BGA packages, depending on package size).

Site Preparation

The excess solder that remains on the board can be removed using a vacuum desoldering system or a soldering iron with a solder wick. Special care must be taken to avoid damaging the solder mask material and the solder pads. As a final step, alcohol may be used with a brush to clean the rework area. Allow the board to dry and inspect to ensure a clean solderable surface. The specific steps used here may be different from board to board and from company to company. As a minimum, the removal of the excess solder is an essential requirement.

Solder Paste Application

There are several options available to apply the solder paste to the component site. The BGA package itself may be screened with paste prior to placement. In addition, the site may receive solder paste with a dispensing method. Finally the application of flux to a prepared pre-tinned site can produce acceptable results in most situations.

BGA Placement and Reflow

The next step is to replace the component on the board. The replacement component should be baked prior to assembly if the component has been exposed to the environment for more than the allotted time. Place the component on the site, observing all the alignment precautions. Reflow the balls using hot air in a manner similar to the removal process. Again observe total board temperature to avoid any thermal gradients that can result in board warpage. It is recommended to heat the PCB from the underside to a given temperature (depending on the board size and properties), preferably in the 80°C-145°C range.

Heating the underside of the board can help to minimize the temperature gradient on the board.

Additionally, larger BGA components such as Flip-Chip BGAs are quite sensitive to heat; therefore, extra precautions are necessary to ensure successful result. It is critical to minimize

the temperature gradient on the part. High temperature gradient will create thermal shock that leads to package warpage. The temperature delta between the following locations should be 7°C or less: the solder balls on the corners, the solder balls at the center of the package, and the top surface side of the package. To achieve minimal temperature gradient, a slower ramp up rate (0.5°C/sec) and a lower peak reflow temperature (200°C as measured at the solder balls) is recommended. Additionally, cooling should be optimized to minimize the temperature differential as described under the **Reflow Profiling**, page 4.

BGA Reballing

Xilinx does not recommend reballing. Xilinx parts that are reballed will not be guaranteed by Xilinx. A maximum of three reflow cycles are allowed.

Conformal Coating

Xilinx has no experience or reliability data on flip-chip BGA packages on board after exposure to conformal coating. It is recommended that the end-user should characterize the board level reliability performance of Xilinx packages before production use.

Post Assembly Handling

When assembling mechanical connectors or fixtures to the PB board, users should be careful not to create excessive bowing or flexing on the PB board as this might weaken or cause damage to the solder joints interface.

Thermal Management

All the packages can use thermal enhancements, which can range from simple airflow to schemes that can include passive as well as active heatsinks. This is particularly true for the high performance flip-chip packages where system designers have the option to further enhance the packages to handle in excess of 20 watts with arrangements that take system physical constraints into consideration.

The accompanying flip-chip thermal management chart shown in Figure 4 illustrates simple but incremental power management schemes that can be brought to bear on a flip-chip package. Similar concept can apply to the other packages in the family.

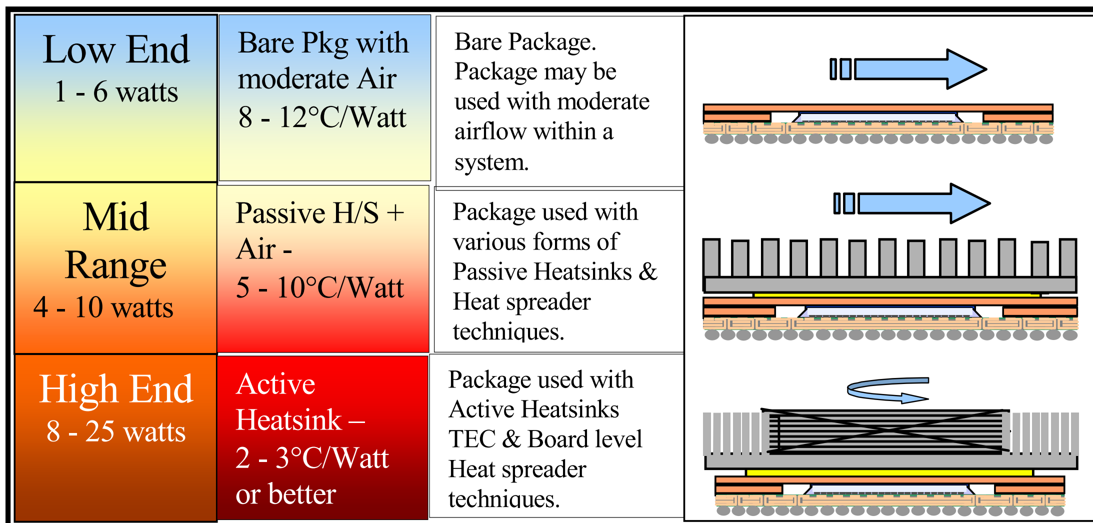

Figure 4: **Thermal Management Options for Flip-Chip BGA Packages**

For moderate power dissipation (less than 6 watts), the use of passive heatsinks and heatspreaders attached with thermally conductive double-sided tapes or retainers can offer quick thermal solutions in these packages.

The use of lightweight finned external passive heatsinks can be effective for dissipating up to 10 watts in the bigger packages. The more efficient external heatsinks tend to be tall and heavy. To

help prevent component joint from heatsink induced stress cracks, the use of spring loaded pins or clips that transfer the mounting stress to a circuit board is advisable whenever a bulky heatsink is considered. The diagonals of some of these heatsinks may be designed with extensions to allow direct connection to the board.

All Flipchip packages offered are thermally enhanced BGAs with die facing down. They are offered with exposed metal heatsink at the top. These are considered high-end thermal packages and they lend themselves to the application of external heatsinks (passive or active) for further heat removal efficiency. Again precaution should be taken to prevent component damage when a bulky heatsink is attached.

Active heatsinks may include simple heatsink incorporating a mini fan or even Peltier Thermoelectric Cooler (TECs) with a fan to carry away any heat generated. Any consideration to apply TEC in heat management should require consultation with experts in using the device since these devices can be reversed and cause damage to components. Also condensation can be an issue.

Outside the package itself, the board on which the package sits can have a significant impact on thermal performance. As much as 80% of the heat generated can go through the BGA balls and thus the board. Board designs may be implemented to take advantage of the board's ability to spread heat. The effect of the board will be dependent on the size and how it conducts heat. Board size, the level of copper traces on it, the number of buried copper planes all lower the junction-to-ambient thermal resistance for a package mounted on it.

Heat Sink Removal Procedure

The heatspreader on the package provides mechanical protection for the die and serves as the primary heat dissipation path. It is attached with an epoxy adhesive to provide the necessary adhesion strength to hold the package together. For an application in which an external heatsink subjects the lid adhesion joint to continuous tension or shear, extra support may be required.

In addition, if the removal of an attached external heat sink subjects the joint to tension, torque, or shear, care should be exercised to ensure that the lid itself does not come off. In such cases, it has been found useful to use a small metal blade or metal wire to break the lid to heatsink joint from the corners and carefully pry the heatsink off. The initial cut should reach far in enough so that the blade has leverage to exert upward pressure against the heat sink. Please contact the heat sink and heat sink adhesive manufacturer for more specific recommendations on heat sink removal.

Package Pressure Handling Capacity

For the flip-chip BGAs that have been surface mounted, a sustained, direct compressive (non varying) force applied NORMALLY to the lid with a tool head that coincides with the lid or is slightly bigger will not cause any damage to the flip-chip solder bumps or external balls provided the force does not exceed 4.0 grams per external ball and the device and board are supported to prevent any flexing or bowing. Forces greater than this may work, however if the forces will exceed this limit, a careful finite element analysis should be performed.

The PC board needs to be properly supported to prevent any flexing resulting from such a force. Any bowing resulting from such a force can likely damage the package to board connections. It is recommended that that any complicated mounting arrangements need to be modeled with respect to the thermal stresses that can result from the heating of the device.

References

1. Adams, Jeff, "Xilinx FF1152 Assembly Report", March 27, 2001, Samina Corporation.
2. Gilleo, Ken, "Area Array Packaging Handbook", copyrighted 2002 by McGraw-Hill Co., pages 14.14-14.16.
3. Hall, James, "Concentrating on Reflow's Cooling Zones", EP&P, 3/01/2001
4. Narrow, Phil, "Soldering", SMT Magazine, Aug. 2000

5. O'Donnell, Dennis, "BGA Rework Practices", Precision PCB Services Inc, 2001

Appendix

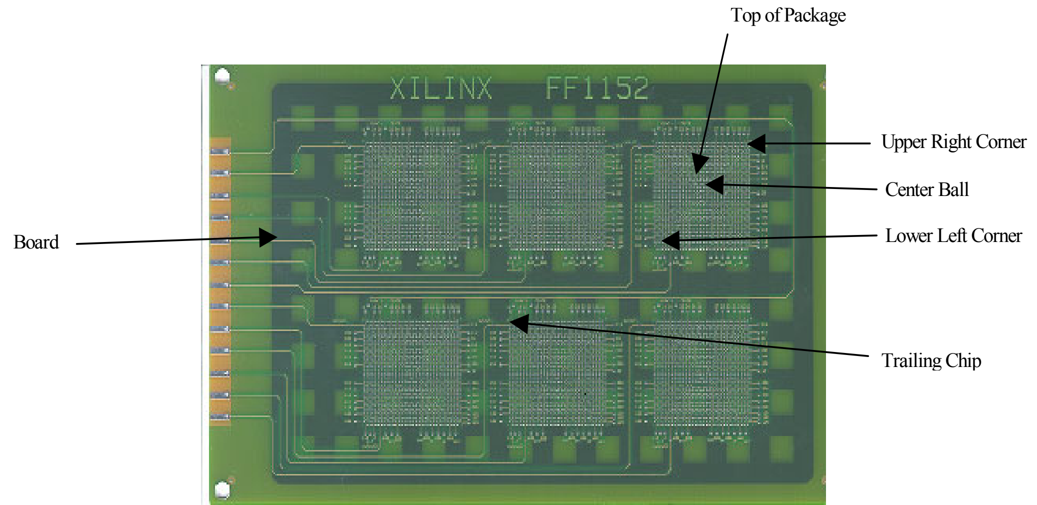

Figure 5: **Temperature Measurement Locations**

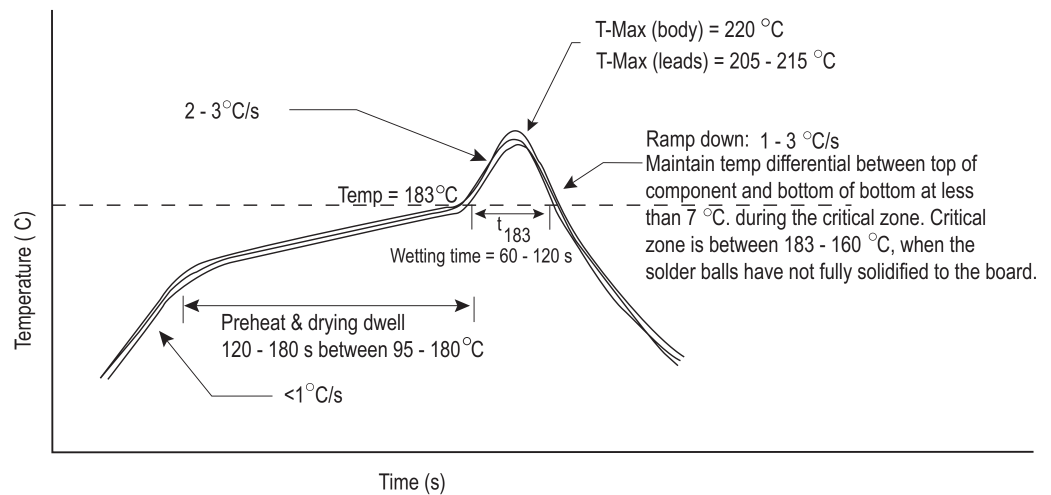

Figure 6: **Typical Solder Reflow Profle for BGA**

Revision History

The following table shows the revision history for this document.

Date	Version	Revision
12/09/02	1.0	Initial Xilinx release.
05/27/03	1.1	Added **BGA Reballing** section.
01/15/04	1.2	Changed Peak Reflow Temperature from 200-210ºC to 2050215ºC.
03/03/06	1.3	Included package construction with Type II Lids (Figure 2), added note about nonhermiticity and conformal coating.