Application Note 7001

March 2004

Guidelines for Using Fairchild's BGA Packages

Dennis Lang, Applications Engineer

Introduction

The development of MOSFETs in Chip Scale Package BGA packages was a technology breakthrough, producing a device that combined excellent thermal transfer characteristics, high-current handling capability, ultra-low profile packaging, low gate charge, and low $R_{DS(on)}$. It is also possible to heatsink Fairchild's BGA MOSFET's allowing industry leading performance in high power and high thermal load applications. Combining Fairchild's PowerTrench® process with state-of-the-art BGA packaging technologies, produced a significantly smaller package size, which is of prime concern when attempting to meet the power needs of today's smaller application requirements. This application note will address the special design factors required when creating board layouts, assembly information, and rework recommendations.

Fairchild's BGA Construction

The Fairchild BGA is constructed of a silicon die using high temperature solder attached to a copper lead frame (see Figure 1). The solder bumps on the Fairchild BGA are of a high lead composition (with a melting point above 300ºC) and do not melt when mounted to the PCB. This controls the height of the BGA, and will allow for a more predictable solder alloy composition when choosing from the wide variety of solder pastes, including lead free types. All Fairchild BGA's meet Jedec Standard No. 95-1 for Fine Pitch Ball Grid Array components. This specification relates to bump alignment for BGA components.

Figure 1. Fairchild BGA (FDZ202P)

Theory

The thermal mismatch of the mounted chip system creates loading in the solder joints as the BGA is operated. This loading leaves residual stresses in the joint, which if uncontrolled could cause fatigue failure. Creation of solder joints which have the highest resistance to fatigue loading, is formed by using solder of eutectic (63%Sn 37%PB) composition—a formulation readily available worldwide. Utilizing Stencils of the proper thickness will ensure that solder joints have the proper amount of solder paste. It is important to note that proper board design will also play a crucial role in producing reliable solder joints.

Board Design

Correct board design becomes even more critical in successful implementation of BGA devices than in standard surface-mounted components. When working with BGA packages solder joints serve a dual function as they form the electrical and mechanical connection for the device. This means that extra attention must be paid to the solder joints to ensure trouble-free operation of the part. When designing a PCB for a BGA, the designer should use a land pattern with individual pads for each bump (see Figure 2). Pads should be shorted together for best thermal and electrical performance, and should be non-solder mask defined for best reliability.

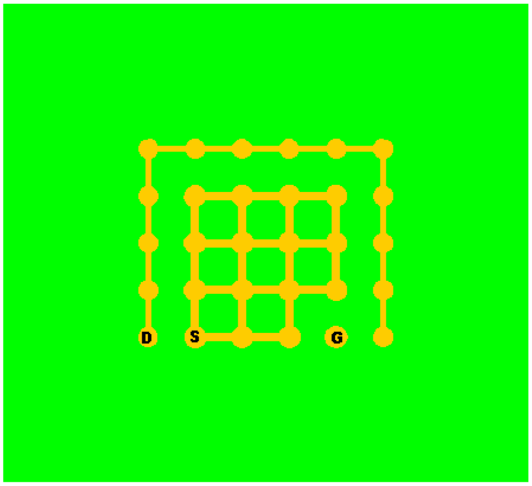

Figure 2. Individual Land Pattern

Non-solder mask defined pads offer (see Figures 3 and 4) solder joints that are approximately three times more reliable than is possible when using solder mask defined pads. When designing with a Fairchild BGA, it is best to devote slightly more copper area on the PCB to the drain than the source. This will yield the best thermal performance of the device for a given copper area. Surface finish of the PCB should be immersion tin, immersion silver or Organic Solderability Protection (OSP), such as ENTEK 106a. This will guarantee flatness of the mounting surface, allowing the BGA to self-align properly. Immersion gold should be avoided due to an unpredictable phenomenon known as black pad which may cause solder joint failures. Pads on the circuit board should use the size recommended on the individual part number's data sheet as a starting point for the PCB designer. It has been found that in large-scale manufacturing increasing the circular pad diameter 20% over the bump diameter requires less accuracy from pick and place equipment yielding a more robust manufacturing process.

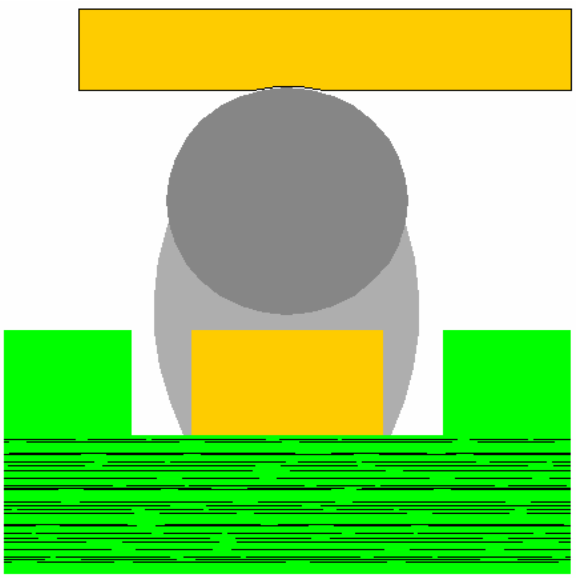

Figure 3. Non-Solder Mask Defined Joint

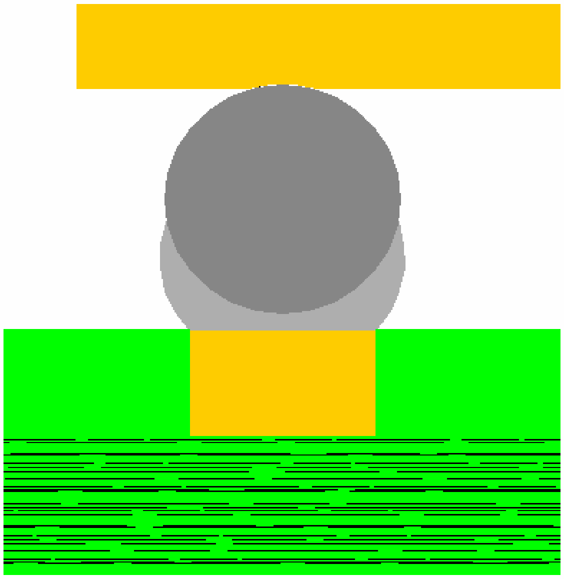

Figure 4: Solder Mask Defined Joint

Stencil Design

There are a several points to consider when designing the solder paste print stencil. First, the stencil should be 5-7 mils thick, with the apertures tapered to facilitate solder paste release. Now available, square aperture, laser cut stencils that yielded superior results in testing for solder paste release. The aperture should be approximately 12 mils square for 300 µm bump products, and 15 mils square for 400 µm bump products depending on pad size chosen for the PCB.

Solder Paste

We recommend Type 3 (mesh size -325 to +500 or 25 to 45 micron paste powder size) no clean solder paste from any major solder paste supplier. Testing at Fairchild has been performed using Kester, no-clean solder paste with a mesh size of –325 to +500, a composition of 63%Sn 37%Pb;described as type NC 840. Similar solder paste is readily available in using many different dispense methods, and is available through many suppliers. Fairchild BGA packages also perform well with lead-free solder. If solder bumps containing lead mix with lead-free solder paste, optimum performance could be affected by intermetallics, which could degrade the solder joint created. Fairchild's high temperature solder bump process creates bumps that do not fully alloy with the paste during reflow. This leads to superior performance with lead-free solder pastes when compared to BGA's and CSP's which utilize eutectic bumps. When determining the reflow profile, it is best to begin with the guidelines set by the solder paste manufacturer. Thermocouples should be used to make a thermal map of the oven to ensure that the temperatures in the oven are appropriate for the board, and its associated components.

The solder paste employed should be no-clean. No clean is necessary because of the difficulty in cleaning under the package once reflow is completed. Use a reflow profile as recommended by the solder paste vendor. It is critical that adequate solder paste is applied to the pad since the high temperature balls does not melt. To ensure adequate solder paste despost, the stencil apertures should be slightly larger than the pad when using about 6 mil thick stencil.

Underfill

Mounting a BGA device using the steps described above will produce good mechanical and thermal properties. However in some instances, strength of the solder bonds will be overcome by forces generated in, or on the device. Underfill can increase the strength of the chip-board bond by 5 to 10 times. In order to enhance the thermal cycling performance and shock resistance of the BGA bond, the process of underfill can be implemented. Underfill is illustrated in red in Figure 5.

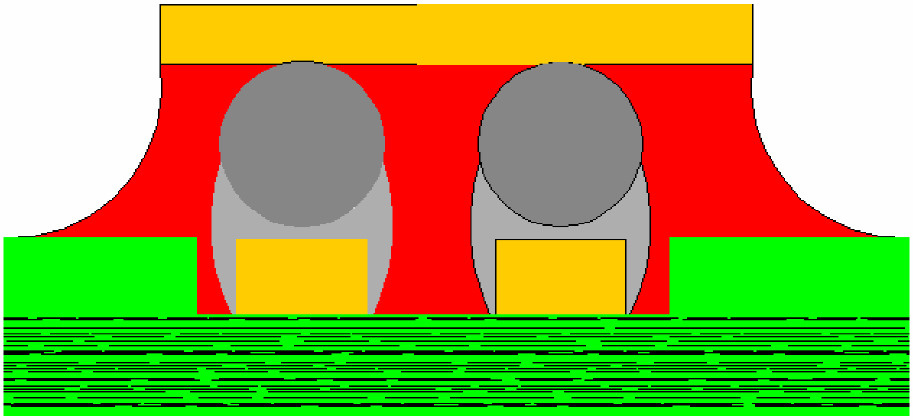

Figure 5: Underfilled BGA Assembly

Underfill is a polymer dispensed on the board after the device has undergone reflow. To begin the process of underfill, pre-bake of the substrate is recommended to minimize moisture outgas, and facilitate the flow of the underfill (typical pre-bake between 75-100ºC). Underfill is a liquid that is usually dispensed on a corner, or in a line along the edge of the BGA. The chip and board are then heated to the recommended temperature, which is in the range of 125-165ºC, to flow the underfill. The underfill is drawn under the chip by capillary action. Temperature is held until the underfill has cured. Fast-curing underfills can be done in 5 minutes, while harder, stronger, underfills take up to 90 minutes, producing optimal strength.

When selecting an underfill the designer must be conscious of several factors:

1) glass transition temperature (T_g) that represents the temperature at which the underfill becomes soft and rubbery,
2) generic requirements for underfills (low ionics), low moisture absorption, and low alpha-particle emission.

If the designer chooses to allow rework of the board after the chip has been underfilled, it is advised that Loctite 3567 be used as an underfill. This underfill can be reworked on a standard BGA rework station by heating the device to 220ºC for 100 seconds. This will allow the chip to be taken off using a rotating motion with tweezers. If the application places extreme demands on the chip, use an underfill that is stronger, such as Loctite 3563, which is designed for the high strength, and high production speeds required by consumer electronics. This type of encapsulant does not allow rework of the part.

Rework

Using appropriate techniques, BGA yields during manufacture will be very high. However, if there is a need to do rework due to component, or process failure then follow these few directions.

1. Remove the BGA package using one of the many types of hot air, or laser rework systems on the market. Most systems use air on the front and backside of the board, with some of these systems using backside heating across the entire board to yield a uniform temperature. Other types of systems will employ localized heating only at the backside of the part. For larger boards, systems that heat across the entire backside of the board are preferred as there is less chance of board warpage during rework. Laser systems may be appropriate if the adjacents components are very close to BGA (under 100 mils) since they may be damaged by hot air if they are not properly shielded.
2. Apply heat directly to the device with a nozzle to reflow the solder. Nozzles typically have an integral tube to create a vacuum, used to remove the part after the solder has been melted. It is important to note that utilizing excessive heat to melt the solder could adversely affect the other parts in the SMT assembly, especially if they are very close to the BGA.
3. After the part is removed it should be discarded. While it is possible to re-ball some BGAs, it is not recommended for Fairchild’s BGA products due to cost, and durability concerns. After the part is removed the mounting pads need to be prepared for rework. To ensure proper alignment during rework, the solder should be pulled from the pads using solder braid or vacuum solder removal systems. Then, solder paste should be printed on the pads using a rework stencil. Using the rework system, align the new BGA, and reflow to the PCB.
4. It is critical that adequate solder paste is applied to the pad since the high temperature balls do not melt. To ensure adequate solder paste deposition, the stencil apertures should be slightly larger than the pad when using a stencil that is about 6 mil thick.

Heat Sinking

One of the advantages of having a metal backed package is the ability to utilize a heat sink to dramatically improve power handling of the device. Fairchild BGA's used in this configuration have current carrying capabilities that set new standards in the industry. While it is not easy to predict the performance of a heat sink system a customer may use, Fairchild testing revealed that a small heatsink on a BGA was able to quadruple its power handling. When heat sinking a Fairchild BGA it is important to remember that the backside of the package is an electrically active drain connection. Therfore it is strongly suggested that the thermal interface material chosen to fill the gap between the heat sink and the BGA is an electrical insulator. If not, there may be significant electrical interference.

References

[1] S. Greathouse, “Surface Tension and the Self-Centering of BGA’s”, SMT, May, 1997

[2] Intel 1998 Packaging Data Book, “Plastic Ball Grid Array Packaging”

[3] J.H. Lau and Y.H. Pao, Solder Joint Reliability of BGA, CSP, Flip Chip and Fine Pitch SMT Assemblies, McGraw-Hill, NY, 1997

[4] JEDEC Solid State Products Engineering Council, JEDEC Standard, No. 95-1, “Design Requirements for Outlines of Solid State and Related Products.” Fine Pitch Ball Grid Array Packages (FBGA) Square Outlines, Dec. 1999, Rev. #B.

[5] M. P. McCurdie, “Flux Residue Cleaning for High-Lead Bumping”, Semiconductor International, October, 2000

[6] E. Strafstrom, “Unraveling the Final Finishing Mystery”, Circuits Assembly, November, 2000, pp. 56-62

TRADEMARKS

The following are registered and unregistered trademarks Fairchild Semiconductor owns or is authorized to use and is not intended to be an exhaustive list of all such trademarks.

ACEx™
ActiveArray™
Bottomless™
CoolFET™
CROSSVOLT™
DOME™
EcoSPARK™
E^2CMOS™
EnSigna™
FACT™
Across the board. Around the world.™
The Power Franchise™
Programmable Active Droop™

FACT Quiet Series™
FAST®
FASTr™
FPS™
FRFET™
GlobalOptoisolator™
GTO™
HiSeC™
I^2C™
ImpliedDisconnect™

ISOPLANAR™
LittleFET™
MICROCOUPLER™
MicroFET™
MicroPak™
MICROWIRE™
MSX™
MSXPro™
OCX™
OCXPro™
OPTOLOGIC®
OPTOPLANAR™
PACMAN™

POP™
Power247™
PowerSaver™
PowerTrench®
QFET®
QS™
QT Optoelectronics™
Quiet Series™
RapidConfigure™
RapidConnect™
SILENT SWITCHER®
SMART START™
SPM™

Stealth™
SuperFET™
SuperSOT™-3
SuperSOT™-6
SuperSOT™-8
SyncFET™
TinyLogic®
TINYOPTO™
TruTranslation™
UHC™
UltraFET®
VCX™

DISCLAIMER

FAIRCHILD SEMICONDUCTOR RESERVES THE RIGHT TO MAKE CHANGES WITHOUT FURTHER NOTICE TO ANY PRODUCTS HEREIN TO IMPROVE RELIABILITY, FUNCTION OR DESIGN. FAIRCHILD DOES NOT ASSUME ANY LIABILITY ARISING OUT OF THE APPLICATION OR USE OF ANY PRODUCT OR CIRCUIT DESCRIBED HEREIN; NEITHER DOES IT CONVEY ANY LICENSE UNDER ITS PATENT RIGHTS, NOR THE RIGHTS OF OTHERS.

LIFE SUPPORT POLICY

FAIRCHILD'S PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF FAIRCHILD SEMICONDUCTOR CORPORATION. As used herein:

1. Life support devices or systems are devices or systems which, (a) are intended for surgical implant into the body, or (b) support or sustain life, or (c) whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in significant injury to the user.

2. A critical component is any component of a life support device or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.

PRODUCT STATUS DEFINITIONS

Definition of Terms

Datasheet Identification	Product Status	Definition
Advance Information	Formative or In Design	This datasheet contains the design specifications for product development. Specifications may change in any manner without notice.
Preliminary	First Production	This datasheet contains preliminary data, and supplementary data will be published at a later date. Fairchild Semiconductor reserves the right to make changes at any time without notice in order to improve design.
No Identification Needed	Full Production	This datasheet contains final specifications. Fairchild Semiconductor reserves the right to make changes at any time without notice in order to improve design.
Obsolete	Not In Production	This datasheet contains specifications on a product that has been discontinued by Fairchild semiconductor. The datasheet is printed for reference information only.