December 8, 1997

Revision 1.0

THERMAL, ELECTRICAL AND MECHANICAL CONSIDERATIONS IN APPLYING BGA TECHNOLOGY TO A DESIGN

ABSTRACT

This document briefly discusses the thermal, mechanical and electrical design considerations associated with using ball-grid array components.

INTRODUCTION

Current wire-bonding technology limits today's smaller package sizes to pin counts of 256 or less. The introduction of ball-grid array (BGA) package technology allows a device to have up to 1000 interconnects on a 33 mil package with a ball pitch of 1.5 mils. BGA package material falls into two classes: *organic*, which includes plastic and Tape Automated Bonding BGA (T-BGA), and *inorganic* or ceramic BGA. These package materials do have some limitations.

This document briefly discusses the thermal, mechanical and electrical issues associated with both organic-based and inorganic-based ball-grid array components. Considerations for designing a printed circuit board that employs ball-grid array packages are also discussed.

WHAT IS BGA?

A BGA package consists of a wire-bonded die on a substrate. The substrate is usually made of two metal layers of copper clad bismaleimide (BT) laminates, but can have up to four metal layers. Four metal-layer substrate designs often contain additional power and ground planes. The die and bonds are encapsulated within a protective molding compound. Via holes drilled through the board provide routing from the lead fingers to eutectic solder balls on the underside. The array of solder balls on the underside of the BGA package provides a high density interconnection of quality solder joints.

Ball-grid array packages provide a low cost alternative to fine-pitch leaded surface mount packages. Other advantages over traditional packages include:

- Higher interconnect densities
- Reduced placement problems
- Better thermal performance
- Better electrical performance
- Higher board assembly yields
- Reduced handling issues

BALL GRID ARRAY DESIGN CONSIDERATIONS

Thermal considerations:

The primary method of heat dissipation in a BGA device is conduction to the printed circuit board (PCB). The PCB acts as a heat sink, which makes the thermal performance of the BGA device dependent on the design of the PCB, and the air flow across the PCB. The ground plane is the most convenient thermal conductor and should be placed as close as possible to the surface attached to the BGA package. The design should use thick copper power and ground planes and it should have a large surface area available to better dissipate heat.

Connections from the BGA package to the power and ground planes should be made with as many through-hole connections as possible. This fills the area under and around the BGA package with copper, which couples the device to the "heatsink" layers and transfers the heat through redundant through-holes. Many BGA packages are designed with thermal vias integrated into the package. These vias provide a thermal conduction path from the back side of the die to the eutectic fingers and then into the PCB. The thermal vias are formed from plated through-holes which are designed in during the PCB layout process.

Summary of Key Points

- Provide good air flow across the printed circuit board.
- Use the ground plane as a heat sink.
- Take advantage of thermal vias on the BGA device.

Electrical considerations:

The limitations on the electrical performance of a semiconductor device depend primarily on the degree to which the distributed inductance and capacitance of the device leads limits the frequency of operation. The overall frequency response of a device is the combination of the response of the package interconnection system and the active device. Three electrical parameters are always present in interconnect systems:

- **Resistance**. Resistance can cause DC voltage drops on signal lines and contribute to RC charging delays. Resistance loading should be minimized in a good system design.

- **Capacitance**. Capacitance is a major cause of signal delay in a device. Capacitance loading of signal lines can be decreased by reducing node and trace lengths. Propagation delay is the time delay for a signal's rising edge to propagate along a given length of interconnect in a device, or on a printed-circuit board. Propagation delay can be the most significant characteristic and must be considered during circuit design. Capacitance loading should be minimized in a good system design.

- **Inductance**. Crosstalk is the coupled noise from active lines onto quiet lines due to mutual capacitive and inductive coupling. Inclusion of a ground trace between adjacent signal lines reduces crosstalk by up to 50 percent. Critical circuits should be designed in a strip line format by taking advantage of the internal power and ground planes. This allows BGA devices to be used in high-end applications which demand low crosstalk while operating at high frequencies.

Switching noise occurs when a CMOS driver switches on or off. This creates a transient current spike that is passed through the power distribution network that supplies energy to the device.

The magnitude of the noise depends on the current slew rate. Switching noise limits the achievable system performance and can create many problems in digital electronic designs when it is not handled correctly.

Summary of Key Points

- Always consider propagation delays in circuit design.
- Reduce signal delay from capacitance loading by keeping trace lengths short.
- Minimize switching noise and cross talk through decoupling.

Mechanical considerations:

Reliability is a major factor in the selection of a package type. To help ensure the reliability of a BGA device, proper handling and placement techniques must be used. Two important factors to consider are component placement and the presence of moisture.

Moisture

Moisture in the liquid form is one of worst enemies of an integrated circuit. Proper material selection, fabrication design, and processing conditions minimizes the amount of moisture intrusion, which extends the life of the device and improves resistance to delamination or package cracking during the assembly process. Moisture-caused problems include:

- *Popcorning* is a delamination of a crack that reaches the outside of the plastic package. It is caused by the rapid expansion of moisture to water vapor inside the plastic device.
- *Delamination* is the separation of interior layers. It is very important to check for delamination in BGA devices.
- Moisture trapped between the solder balls underneath the package can result in AC performance issues and cause pin-to-pin leakage. Special care should be taken to avoid this.

Component Placement

The board should be designed in such a way that major stress points are not located near BGA packages.

Summary of Key Points

- Observe proper SMT placement and handling procedures to reduce delamination and device cracking.
- The presence of moisture and mechanical stress may affect reliability.

BGA routing considerations:

Using BGA packages in a design can increase the level of complexity of a design because it allows higher pin counts. Failing to consider all elements of the BGA package technology can result in a increase in the number of layers required in the PCB and can make PCB layout more challenging. In all BGA designs, careful attention to the placement of power and ground pins simplifies PCB routing.

Depopulated array options are becoming more popular for high-pin-count BGA solutions. The center of the array is the best region to locate power and ground pins to simplify the routing to these pins. In most cases, it may be possible to connect a high-pin-count BGA to a four-layer design with two signal layers, depending on the number of power and ground pins, their location, and the complexity of the design.

Summary of Key Points

- Proper placement of power and ground pins simplifies routing.
- Proper routing techniques can help reduce the number of board layers.

CONCLUSION

BGA packages provide a reliable solution for high-pin-count designs as long as thermal, electrical and mechanical issues are considered. BGA devices can have a positive impact on the overall PCB technology. Using BGA packaging technology can result in a more robust and cost effective assembly process.

REFERENCES

Ball Grid Array Technology
John H. Lau
McGraw Hill, Inc.

Intel Packaging Databook
Intel Corporation
http://developer.intel.com/design/flcomp/packdata/packbook.htm

Ball Grid Arrays: Questions & Answers
Intel Corporation
http://developer.intel.com/design/i960/packdata/2451.htm