International
IOR Rectifier

Application Note
AN-1028

Recommended Design, Integration and Rework Guidelines for International Rectifier's *iPOWIR*™ Technology BGA Packages

Introduction

The International Rectifier iPOWIR line of BGAs provides several advantages over other packages such as QFPs or SO packages. Some of these include:

- Smaller package footprint
- Better electrical and thermal performance
- Larger pitch than fine-pitch QFPs which improves the ease of assembly and testability
- Ability to self-align over PCB solder-lands

These packages are high performance microelectronic devices designed to provide efficient and reliable operation. Because these are high performance packages, it is important to closely adhere to the following guidelines to ensure that their performance and reliability are not compromised.

Motherboard Design

A well-designed and manufactured printed circuit board is required for optimum performance. To achieve maximum reliability and optimum thermal performance, the design of the PCB onto which the BGA is mounted should be considered.

Motherboard Design

The board to which the devices are attached should be constructed of FR-4 or Polyimide and meet IPC–A-610 specifications. The board should be routed in such a manner as to maximize thermal conductivity (amount of copper) in the region of the BGA.

Figure 1 shows an example of the recommended motherboard design. The PCB has been designed with large copper power planes and solder-mask defined openings for the ball lands instead of individual metal land pads. This will provide the best thermal and electrical performance. Please refer to the data sheet for individual device performance under differing application conditions.

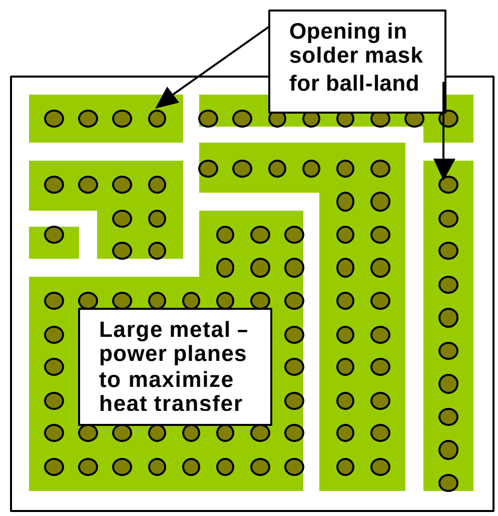

Figure 1: PCB design showing large Cu power planes instead of individual ball lands for better thermal performance

Land Design -Land Diameter

The land pads on the motherboard should be solder mask defined (SMD). The solder balls are 20 mil diameter and the recommended solder mask opening, for most applications, is 80-100% (16-20 mils) of the solder ball diameter. This provides the optimum stand-off height.

SMD pads have some advantages over non solder mask defined (NSMD) pads. Solder mask size definition (photo-imaged process) is typically better than copper pad definition. Also the overlap of the solder mask onto the copper enhances the copper adhesion to the laminate surface.

Thickness of metal layers on PCB

The underlying material should be minimum 2.0 oz. copper terminated in 3 microns of Ni and 0.3 microns of Au, for best thermal performance.

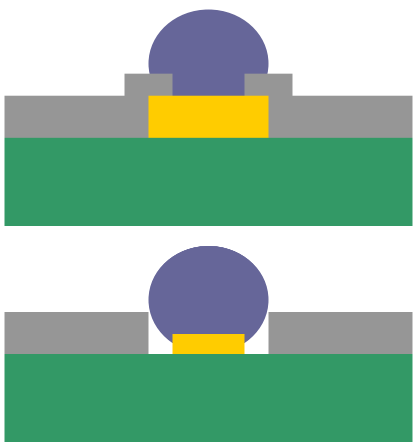

Figure 2: Cross-section of solder-mask-defined (SMD) vs. Non solder-mask-defined (NSMD) pads

Package Integration

General

International Rectifier's iPOWIR BGAs are designed for soldering to printed circuit boards or other similar organic substrates (e.g. FR-4, polyimide and other standard PCB material). These devices are designed to be completely compatible with standard surface-mounting equipment like pick-and-place machinery and furnace reflow machines.

Storage / Baking

International Rectifier's BGAs are supplied in T&R form, and are vacuum sealed in moisture proof packaging. These devices should be stored in the original packaging or in a Nitrogen environment. If these devices are exposed to the ambient environment for more than 168 hours, they will need to be baked at 150°C for 16 hours before mounting to a motherboard. This ensures that any moisture absorbed during storage is expelled prior to board mounting.

Soldering / Board Mounting

The use of a solder-paste is recommended for mounting these BGA devices, although it is possible to omit the paste and use only a flux.

Advantages of using a paste are:

- The paste serves as the vehicle to provide the flux necessary to both the PCB and the solder ball surfaces to allow soldering

- Paste helps to overcome any minor variations in the planarity of the solder balls

- Paste holds the BGA in place during reflow better than flux.

- Paste contributes to the final volume of solder in the joint and provides a higher stand-off, which results in a more reliable joint. Also, using paste allows the volume of the solder to be optimized by varying the stencil opening and stencil thickness.

An automatic or manual stencil/screen printer can be used to distribute the solder onto the printed circuit board ball-lands. The usual stencil design is to use circular openings over circular holes. Recommended stencil thickness is 6 mils. Preferred stencil openings should match 100% in size with solder pads on the printed circuit board. It is recommended that a "no-clean" eutectic lead-tin (Sn63/Pb37) solder paste be used. The solder paste should cover 100% of each land pad and be between 5 and 8 mils thick.

Pick and Place

The placement radial accuracy required is +/- the radius of the pads on the board i.e. +/-10 mils for a 20 mil pad. This will ensure that the center of each solder ball is always placed on its corresponding land pad. Due to the surface tension of the solder in its liquidus state, devices will self align during reflow to give symmetrical joints. Most placement tools have much better accuracy than required and placement variability is rarely an issue with BGA devices.

Reflow

Once the solder has been stenciled, the BGA component should be placed and convection reflowed within 4 hours. The reflow atmosphere should be minimum 80% nitrogen. See graph (Figure 3) for recommended reflow profiles. The peak temperature is 220°C - 225°C. The total furnace time is approximately 5 minutes with approximately 10 seconds at the peak temperature.

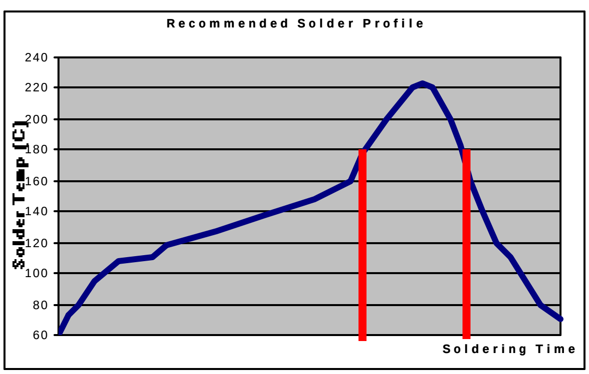

Figure 3: Standard SMT Reflow profile with 220°C peak
Time to get to 180°C: 175 secs
Time above 180°C: 50 secs

Inspection

Unlike peripherally leaded packages, the final solder joints of a BGA cannot be visually inspected to determine joint quality. BGAs, after attachment to the board effectively hide the joints between the package body and the PCB. Only the outer row can be visually inspected and that too will depend on the proximity and location of the neighboring components.

These devices can be X-rayed if required to look for solder bridging. No other inspection step need be performed after board mounting. Figure 4 shows solder-bridging between solder balls.

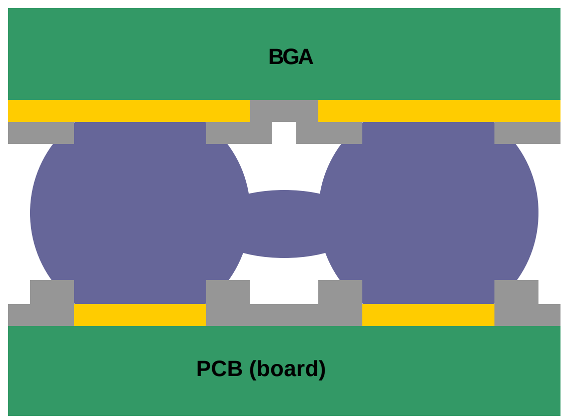

Figure 4: Solder bridging

Cleaning

Cleaning may be required depending on application requirements. Ultrasonic cleaning is permissible. Alcohol based solvents and other properly controlled water cleaning systems are acceptable. Most solvents that are acceptable to other components on circuit assemblies are equally acceptable for use with these devices. Surfactants can be introduced to improve water penetration and flow. An adequate drying profile should be introduced to ensure that no water is trapped beneath the package upon the completion of the cleaning step.

Package Rework

Rework is the process of removing a component from the PCB and replacing it with a new component. The removed part is not reusable as the shape and volume of the solder balls will not be the same as a new package.

Removal and replacement of the BGA packages is not recommended as a routine practice, however if required, it is most easily achieved with the assistance of special equipment such as the Zephyrtroncis Airbath ZT-7. This equipment basically heats a very localized region of the motherboard while applying a tensile "plucking" force to the component. Most rework systems utilize hot gas to locally heat the BGA and reflow the solder balls for removal and replacement. Global preheat (to about 80-100°C) of the entire PCB can be done before local heating of the BGA. Preheating will decrease the heating time required during rework and will also reduce potential PCB warpage.

The following guidelines are prepared around the use of such standard BGA rework equipment.

Removal

The assembly should be free of moisture prior to rework. The motherboard should be rigidly mounted in a retaining frame that allows the component to be accessed by the heating and plucking heads. The device and the motherboard should be kept horizontally level during the reflow to prevent solder bridging. Follow the manufacturer's operating instructions for proper machine operation.

Cleaning

After the component is removed, the land pads should be thoroughly desoldered and cleaned with alcohol. After cleaning, a complete visual inspection of the land grid array should be made with the aid of a microscope. This is done to ensure that all mounting pads are in good condition.

Replacement

Once the land pads have been properly cleaned, eutectic lead-tin "no-clean" solder paste should be applied 5 to 8 mils thick. Using the recommendations mentioned above, a new part should be reflowed onto the motherboard following the manufacturer's instructions.

Cleaning

After the part has been reflowed, the recommended cleaning instructions above can be followed.