APPLICATION NOTE

Pulse a YAGEO company

# Recommended Design, Integration, and Rework Guidelines for Pulse's BGA Package

## Table of Contents




This Application Note discusses optimization of the layout and mounting recommendations for Pulse's BGA devices. Topics discussed include PCB layout placement, soldering, pick-and-place, reflow, cleaning, and reworking recommendations.

# APPLICATION NOTE

## Recommended Design, Integration, and Rework of Guidelines for Pulse's BGA Package

### Introduction

Pulse's BGA (Bump Grid Array) devices are high performance microelectronic devices designed to provide efficient and reliable operation. This application note discusses optimization of the layout and mounting recommendations for these devices. Topics include PCB layout, placement, soldering, pick-and-place, reflow, cleaning, and rework recommendations, which are provided to ensure that performance and reliability are not compromised.

### 1.0 PCB Layout and Design

To achieve maximum reliability and optimum electrical performance, the design of the PCB onto which the device is mounted should be considered. Specifically, the PCB routing, solder mask opening design, and termination should follow these guidelines.

### 1.1 PCB Routing and Land Pad Design

The PCB to which the devices are attached should meet the IPC-A-610 specifications. Figure 1 shows an example of the recommended pad and via design. As shown in this figure, Pulse recommends the use of NSMD (Non-Solder Mask Defined) pads for BGA lands. However, the use of SMD (Solder Mask Defined) pads would also be acceptable. The use of NSMD pads will provide the best reliability and electrical performance. Figure 2 shows the differences between SMD pads and NSMD pads.

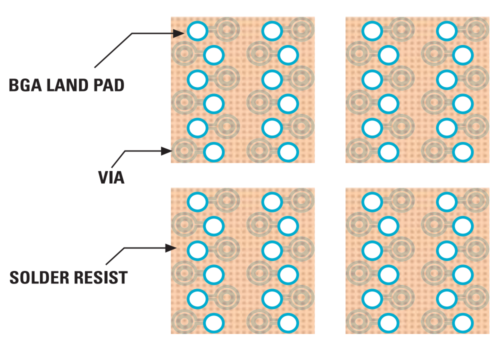


**Figure 1.** *Recommended PCB copper pattern for mounting Pulse BGA device.*

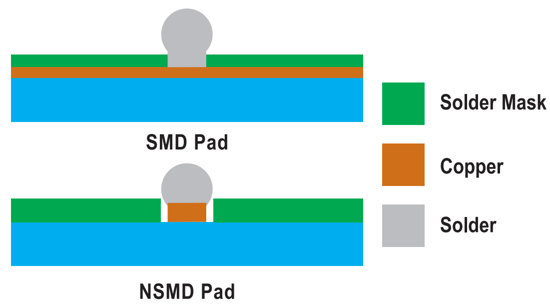


**Figure 2.** *Comparison between solder mask defined pad| and non- solder mask defined pad.*

It is recommended that the land pads on the motherboard should be Non-Solder Mask Defined. For Pulse BGA devices, the required land pad is 0.762mm in diameter (Figure 3). This diameter provides the optimum stand-off height and performance reliability.

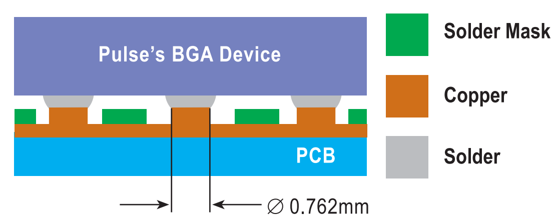


**Figure 3.** *Recommend Land Pad for BGA device*

### 1.2 Plating

For Pulse Lead-Free devices, it is recommended to use electrolytic nickel/gold or tin termination on the PCB. For Pulse Leaded devices, gold, tin, or HASL (Hot-Air Solder Leveling) terminations are recommended, although most other terminations are also acceptable. If using gold termination, a minimum of 5.0 microns of a nickel layer should be used as a barrier, and the gold should be 0.5-1.2 microns thick. It is strongly recommended that the gold thickness does not exceed 1.2 microns. If using tin finish, either white tin or 100% tin is acceptable.

## 2.0 Storage and Baking

It is recommended to follow the JEDEC standard IPC/JEDEC J-STD-033A [1] for handling Pulse devices. Pulse BGA devices are rated at MSL 1 and are supplied in trays. Given their MSL rating, exposure to ambient environment does not result in any moisture absorption at levels that would warrant the need to bake the parts prior to reflow solder processing.

## 3.0 Stencil Design

For Pulse BGA devices, a 0.15mm thick stencil with 0.762mm diameter openings is recommended. Figure 4 shows a typical stencil for a Pulse BGA device. Green areas represent the stencil openings for solder bumps.

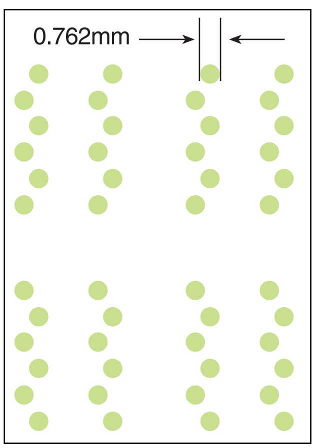


**Figure 4.** *BGA device stencil example*

## 4.0 Surface Mounting Guidelines

The use of solder paste is required for mounting Pulse devices. An automatic or manual stencil/screen printer can be used to distribute the solder onto the PCB pads. Tin-silver-copper SN96.5Ag3Cu0.5 or Sn 95.5Ag4Cu0.5 solder paste, or similar, must be used for Pulse Lead-Free devices. Eutectic tin-lead SN63/Pb37 solder paste, or similar, must be used for Pulse Leaded devices. For Pulse BGA devices, a placement accuracy of ±0.1mm is required.

## 5.0 Reflow Recommendations

For Pulse devices, the solders listed in Section 4.0 should be used. Table 1 and Figure 5 show a typical reflow profile and conditions for lead-free solders and tin-lead eutectic solders. However, it is strongly recommended that the solder paste manufacturer's guidelines be followed. The profile probe should be located on the PCB in the immediate vicinity of the device's perimeter pads. Figure 6 is an example of a Sn96.5Ag3Cu0.5 solder reflow profile and a Sn63Pb37 solder reflow profile.

| Profile Feature | | Lead-free Solder* | Leaded Solder* (Sn-Pb Eutectic Solder) |
|---|---|---|---|
| Pre-heat | Temperature Min ($T_{Smin}$) | 150°C | 100°C |
| | Temperature Max ($T_{Smax}$) | 200°C | 150°C |
| | Time ($t_{Smin}$ to $t_{Smax}$) | 60-180 seconds | 60-120 seconds |
| Reflow | Liquids Temperature ($T_L$) | 217°C | 183°C |
| | Time ($t_L$) | 60-150 seconds | 60-150 seconds |
| Peak Temperature ($T_P$) | | 245°C (±5°C) | 225°C (±5°C) |
| Time within 5°C of peak ($T_P$) | | 20-30 seconds | 10-30 seconds |
| Average Ramp Up Rate ($T_{Smax}$ to $T_P$) | | 3°C/second max | 3°C/second max |
| Ramp Down Rate | | 6°C/second max | 6°C/second max |
| Time 25°C to Peak Temperature ($T_P$) | | 8 minutes max | 6 minutes max |
| Do not exceed . . . | | 245°C | 225°C |

* *See Figure 5*

**Table 1.** *Solder reflow profile conditions*

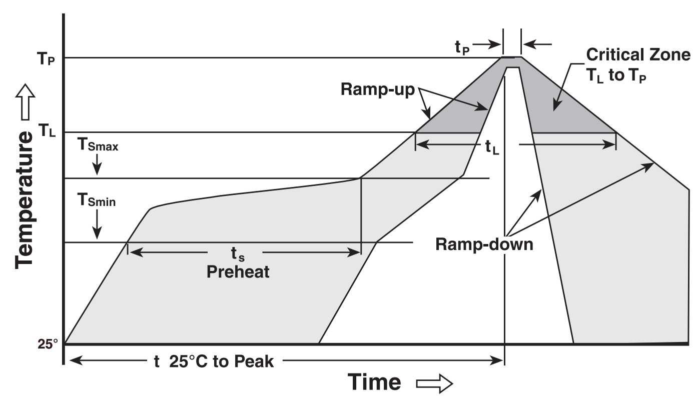


**Figure 5.** *Typical reflow profile*

All reflow profiles should conform to the IPC/JEDEC J-STD-020C standard. The peak reflow temperature for lead-free devices should not exceed 245°C. For Pulse leaded devices, peak reflow temperature should not exceed 225°C.

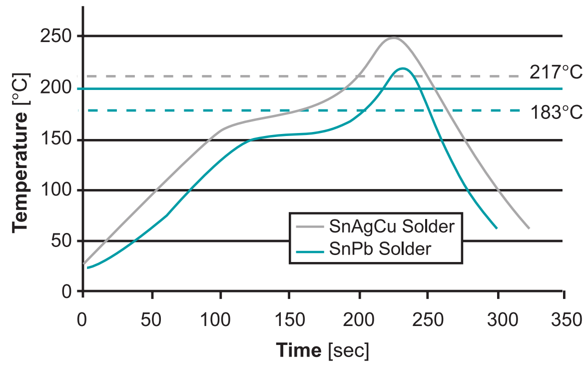


**Figure 6.** *Typical solder reflow profile*

### 6.0 Post Reflow Cleaning

Post reflow flux cleaning is recommended for Pulse BGA devices. 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 used to ensure that no water is trapped beneath the device upon the completion of cleaning. It is important to closely adhere to both the solvent and the solder paste supplier's recommendations.

### 7.0 Rework Guidelines

Rework is the process of removing a component from the PCB and replacing it with a new component. Removal and replacement of the Pulse device is most easily achieved with the assistance of specialized rework equipment. This equipment heats a very localized region of the motherboard while applying a lifting force to the component. Most rework systems utilize hot air to locally heat the device and IR to locally heat the PCB, simultaneously reflowing the solder joint for removal and replacement. Global preheat of the entire PCB should be preformed prior to heating the Pulse device. Preheating will decrease the heating time required during rework and will also reduce potential PCB warpage and damage.

A computer controlled reflow system is recommended. Once an acceptable rework profile has been created, it can be used for future rework needs. The reflow profile should very closely match the applicable profiles depicted in Section 5.0. It is important not to exceed the maximum recommended temperature for these devices.. Before device removal, the assembly should be free of moisture to prevent PCB delamination. It is recommended to dispense flux around the device to maximize heat transfer. The PCB should be rigidly mounted in a retaining frame that allows access to the component to be reworked. The device and the PCB should be horizontally leveled during the reflow to prevent solder bridging. Follow the manufacturer's operating instructions for proper machine operation. After the component is removed, the pads should be thoroughly de-soldered and cleaned with alcohol. The PCB pads can be de-soldered quickly and safely using the "Site Cleaning Micro Tip Nozzle" (PN: ONYX) by Air-Vac. After cleaning, a complete visual inspection of the pads on the PCB should be made with the aid of a microscope. This is to ensure that all land pads are in good condition and the solder resist is not damaged.

Once the pads have been properly cleaned, the solder paste recommended in Section 4.0 should be used to remount a new device. For stencil printing, a mini-stencil specially designed for the device is recommended. For BGA devices, the solder paste can be screened or dispensed on the part or PCB. A rework machine with split field optics or a vision system is recommended for accurate component placement. The new part should be reflowed onto the PCB using the profiles recommended by the solder paste manufacturer or similar to those listed in Section 5.0. After the part has been reflowed, the recommended cleaning instructions in Section 6.0 be followed. A video, "Air-Vac DRS25 BGA Rework Process Video," demonstrating the rework procedure with the nozzle is available upon request.

### 7.1 Machine and Thermal Profiles

The Air-Vac DRS25 "Semi-Automated SMT Rework & Repair" system and the ONYX29 "Robotic SMT Rework, Repair, and Low Volume Assembly," with Air-Vac nozzle N19.3EZ30.5G10-G (Figure 7) are examples of systems that can be used for the Quattro rework process. This same system, along with Air-Vac nozzle N16.6EZ19.4G11-G (Figure 8) can be used for the Dos rework process. The general rework process for this equipment follows. Preheat the appropriate nozzle to 100ºC. Ramp the nozzle heater up during the soak, ramp, and reflow stages. The thermal plot and summary analysis for the Quattro/Dos device is below (Figures 6 and Table 2).

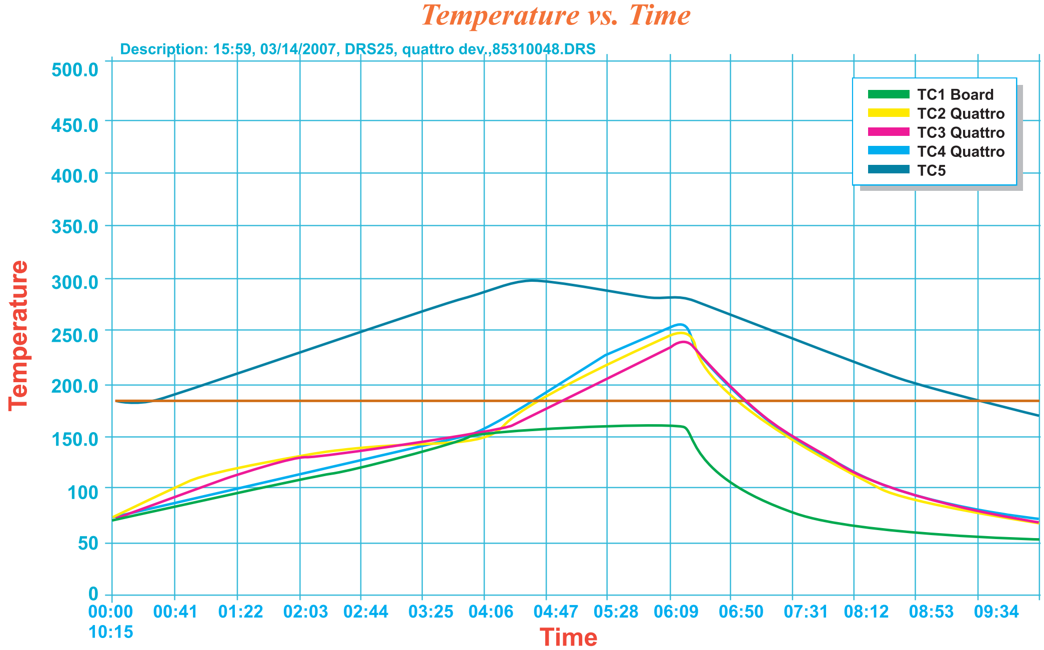


**Figure 7.** *Quattro/Dos Lead-free, Thermal Plot (lead-free devices)*

| | Max Temp (°C) | Min Temp (°C) | Temp Slope (°C/sec) | Time Over 217°C (sec) |
|---|---|---|---|---|
| **TC 1 Board** (Top Side) | 159 | 044 | -0.44 | 000 |
| **TC 2 Quattro** (Joint) | 245 | 066 | -0.78 | 067 |
| **TC 3 Quattro** (Joint) | 237 | 066 | -0.75 | 048 |
| **TC 4 Quattro** (Package) | 257 | 068 | -0.81 | 076 |
| **TC 5** (Adjacent) | 180 | 057 | -0.54 | 000 |

**Table 2.** *Quattro/Dos Lead-free Thermal Summary Analysis*

The key to reworking the Quattro/Dos on a production board is the ability to preheat the assembly to 150ºC prior to localized topside reflow. Both the DRS25 and the ONYX29 are capable of providing this preheated requirement on large, high thermal mass assemblies. Preheating to 150ºC will reduce the amount of heat that needs to be transferred through the device, thereby minimizing package and PCB temperature.

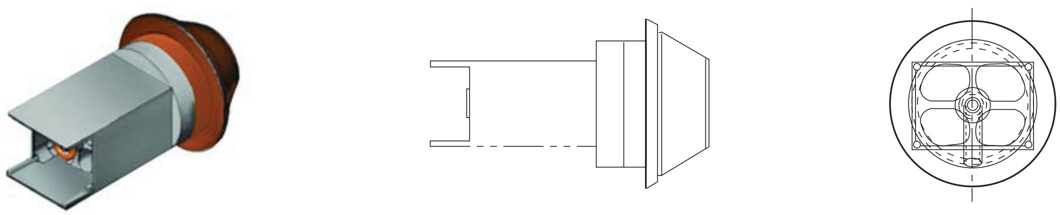

**Figure 8.** *Air-Vac N19.3EZ30.5G10-G: Quattro Nozzle*

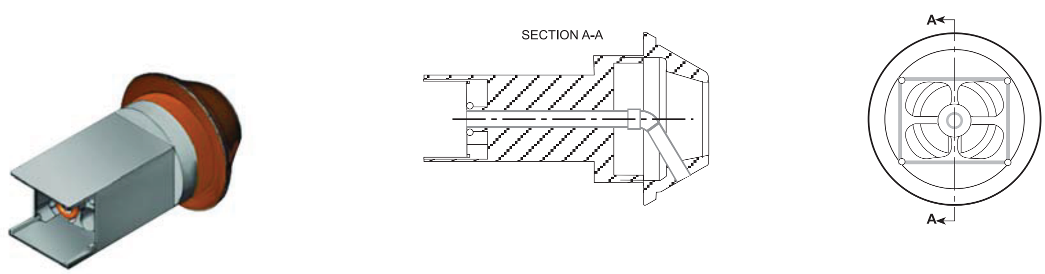


**Figure 9.** *Air-Vac N16.6EZ19.4G11: Dos Nozzle*

Pulse a YAGEO company

## References

[1] Joint Industry Standard, Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices, IPC/JEDEC J-STD-033A, July 2002.