Exposed-Heatslug Ball Grid Array Package Design and Soldering Guidelines

REVISION 0.1

16215 Alton Parkway • P.O. Box 57013 • Irvine, CA 92619-7013 • Phone: 949-450-8700 • Fax: 949-450-8710

REVISION HISTORY

Revision	*Date*	*Change Description*
Packaging-AN400-R	03/09/05	Initial release.

Broadcom Corporation
P.O. Box 57013
16215 Alton Parkway
Irvine, CA 92619-7013

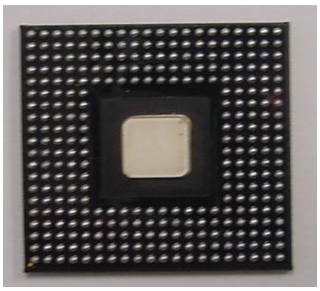
TABLE OF CONTENTS

OVERVIEW

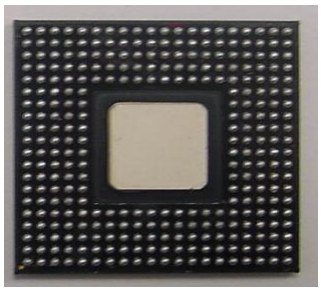
This application note outlines the Printed Circuit Board (PCB) layout and soldering guidelines for Exposed-Heatslug Ball Grid Array (BGA) packages. It is recommended that these guidelines be followed to ensure proper thermal and electrical connections.

EXPOSED-HEATSLUG BGA PROPERTIES

The figures below show the bottom view of the exposed-pad for packages with a 5x5mm Exposed-Heatslug BGA and 7x7mm Exposed-Heatslug BGA.



5x5mm Exposed-Heatslug BGA Package



7x7mm Exposed-Heatslug BGA Package

Figure 1: Bottom View of the Exposed-Heatslug BGA Package

The Exposed-Heatslug BGA has an exposed heatslug which is soldered to the PCB in parallel with the BGA solder joints, see Figures 2 and 3. The Exposed-Heatslug BGA package has the following attributes:

- The exposed-heatslug surface is silver plated for excellent solderability.
- The exposed-heatslug lowers the effective thermal resistance between the package and the PCB.
- The exposed-heatslug provides additional electrical ground connection (additional current return path for signals) for enhanced electrical performance.

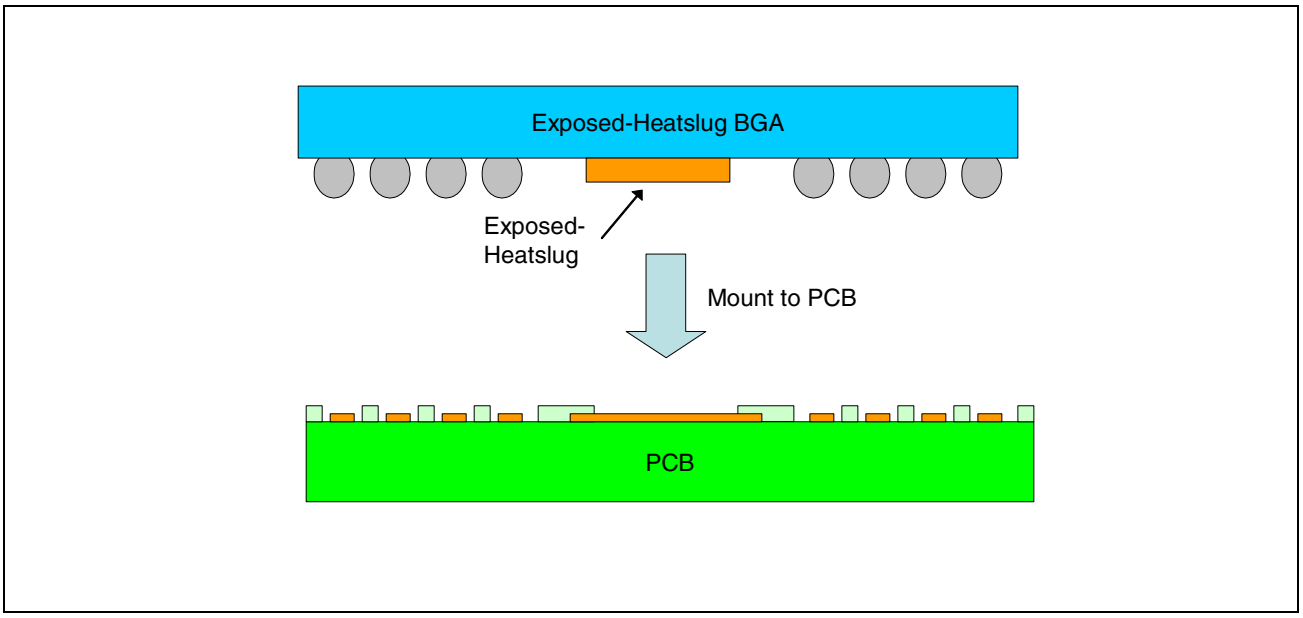


Figure 2: Surface Mounting of Exposed-Heatslug BGA

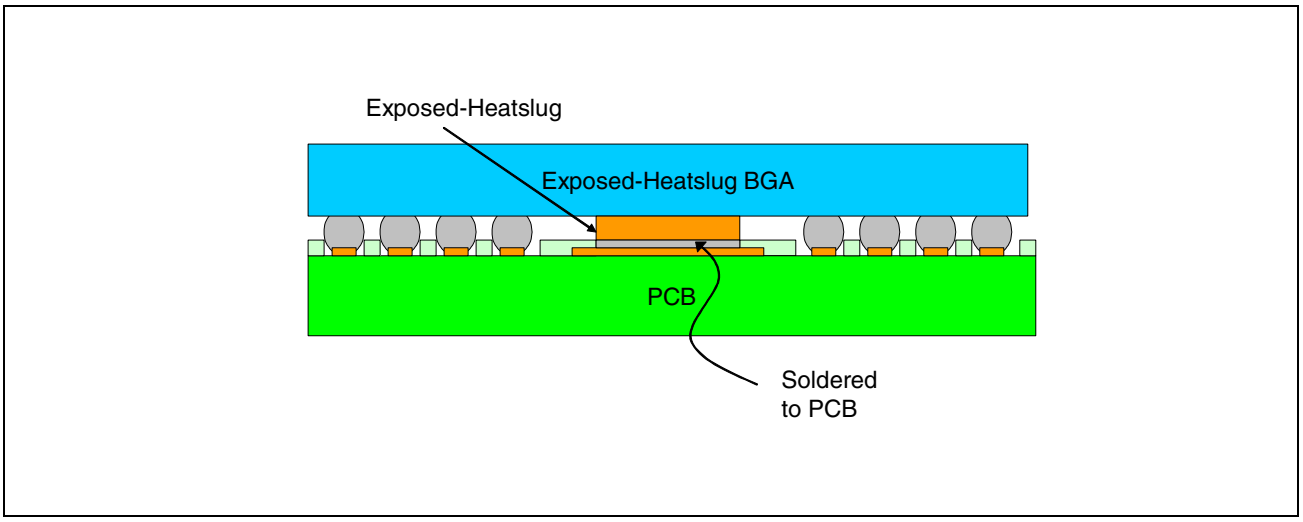


Figure 3: Surface Mounted Exposed-Heatslug BGA

PCB Design Recommendations

This section covers the PCB design guidelines for the landing area which connects to the exposed-heatslug of the BGA.

The landing area for the exposed-heatslug should be a solid metal plane covered with an array of soldermask openings as shown in Figure 4. Table 1 gives the recommendations for landing and soldermask dimensions for the different exposed-heatslug sizes.

It is recommended that the landing area be filled with a matrix of thermal vias (shown in Figure 4 and Figure 5) to connect to internal PCB ground planes. Table 1 gives the recommended via array design parameters. All vias should be tented with soldermask on the component side.

Table 1: Dimension Recommendations for PCB Design

Exposed-Heatslug Size	*Landing Pad Size (A)*	*Soldermask Opening (E)*	*Soldermask Web Width (F)*	*Soldermask Opening Array (FxG)*	*Via Array (MxN)/ Via Pitch (P)*
3.5x3.5mm	195x195mil	36mil	9mil	3x3	4x4/45mil
5x5mm	280x280mil	36mil	9mil	4x4	7x7/45mil
6x6mm	340x340mil	36mil	9mil	5x5	8x8/45mil
7x7mm	390x390mil	36mil	9mil	6x6	9x9/45mil
Note: For the exposed-heatslug size of a given product, refer to the *Package Outline Drawing* in the product data sheet.					

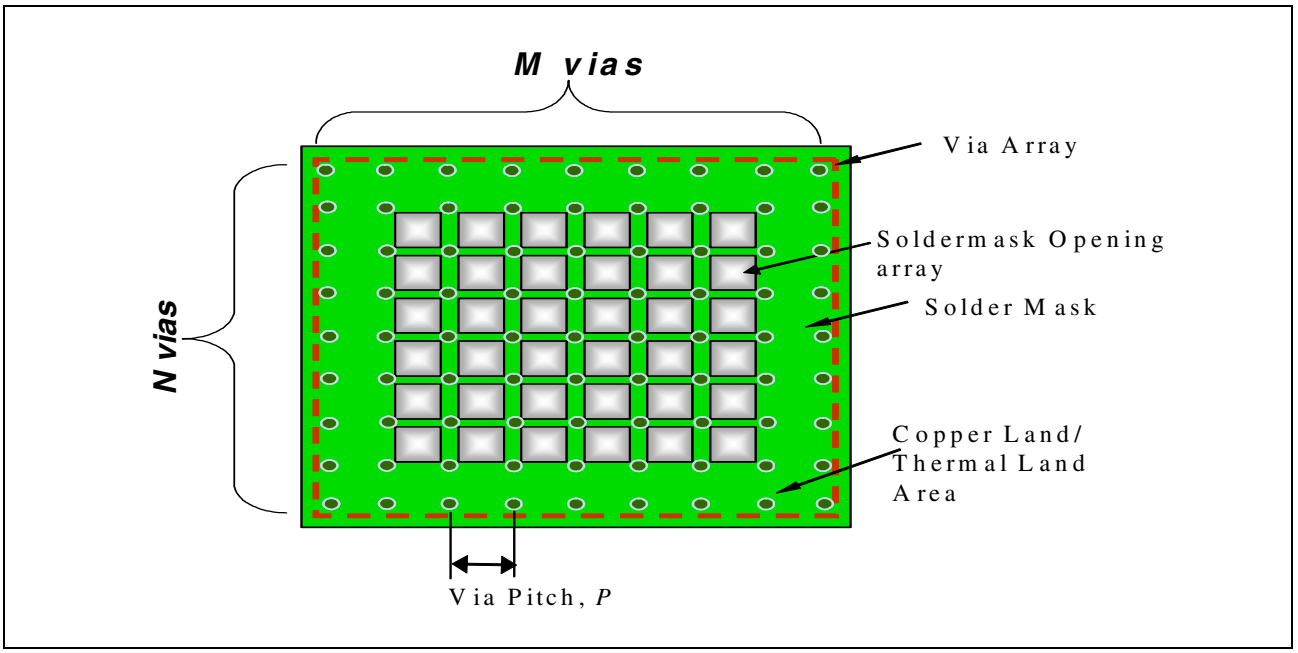


Figure 4: - Thermal Via Layout in Exposed-Heatslug Landing Area

SURFACE MOUNT PROCESS RECOMMENDATIONS

Printed solder paste should be used to create the interconnection between the exposed-heatslug and the PCB landing. To achieve the optimum printing characteristics and solder paste coverage, an array of stencil apertures should be used. A single stencil aperture covering the entire landing area is not recommended.

Figure 5 shows the stencil design and Table 2 gives the design parameters as a function of exposed-heatslug size.

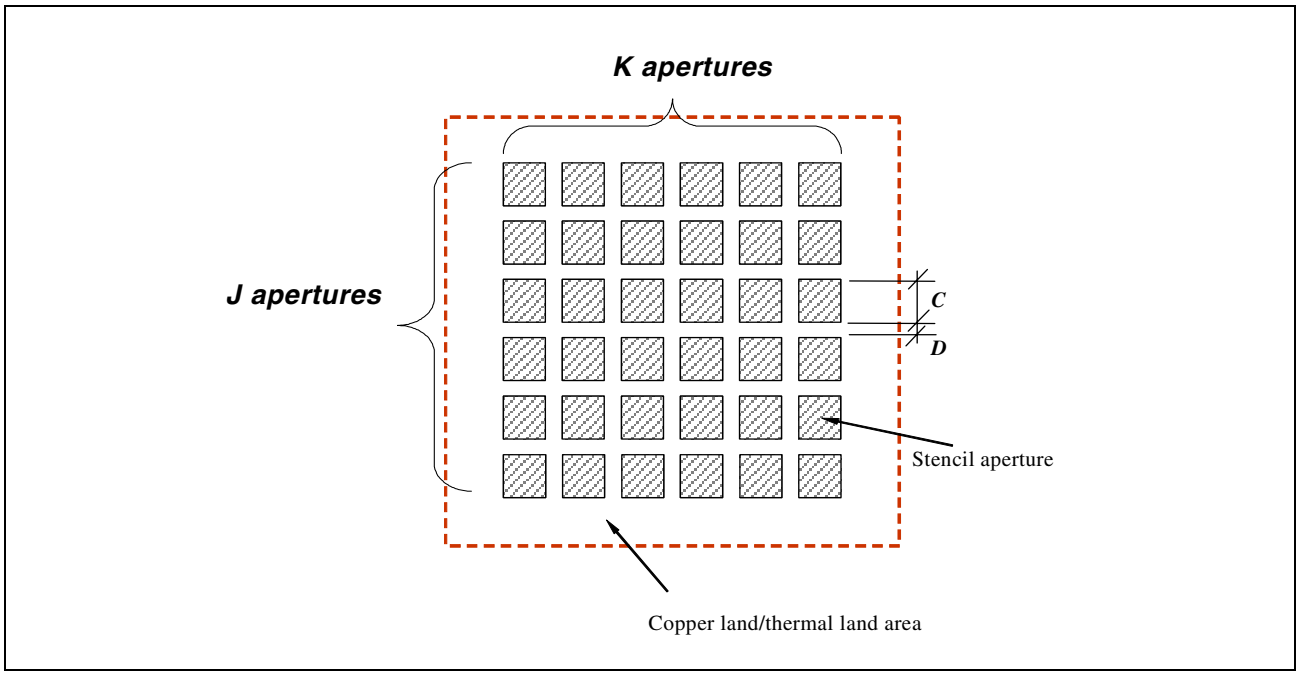


Figure 5: Stencil Design for Exposed-Heatslug Landing Area

Table 2: Exposed-Heatslug BGA Stencil Design Parameters

Exposed-Heatslug Size	*Aperture Array (KxJ)*	*Aperture Opening (C)*	*Distance Between Apertures (D)*	*Stencil Thickness*
3.5x3.5mm	3x3	36mil	9mil	6mil
5x5mm	4x4	36mil	9mil	6mil
6x6mm	5x5	36mil	9mil	6mil
7x7mm	6x6	36mil	9mil	6mil

Note: For the exposed-heatslug size of a given product, refer to the *Package Outline Drawing* in the product data sheet.

PCB LAND PATTERN RECOMMENDATIONS FOR BGA PACKAGES

Figure 6 shows the BGA solder joint geometry prior to surface mount assembly and after surface mount assembly. The recommendations for PCB pad size given in Table 3 are based on the BGA ball size (in accordance with IPC-SM-782).

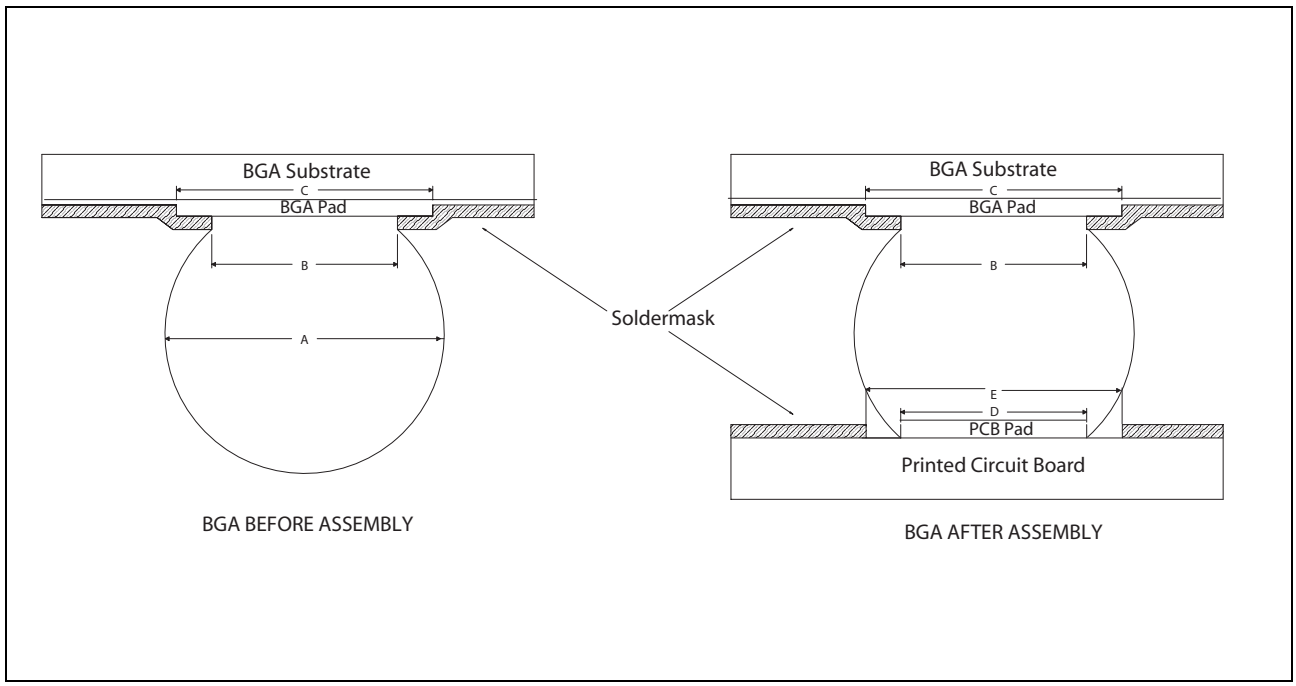


Figure 6: BGA Solder Joint Geometry

Table 3: PCB Land Pattern Recommendations

Ball Pitch	*Ball Size (A)*	*BGA Soldermask Opening (B)*	*PCB Dimension Recommendations*		
			Minimum Allowable PCB Pad Diameter (DMIN)	**Nominal PCB Pad Diameter (D)**	**Maximum Allowable PCB Pad Diameter (DMAX)**
1.27 mm	0.76 mm	0.63 mm	0.50 mm	0.55 mm	0.60 mm
1.00 mm	0.60 mm	0.50 mm	0.40 mm	0.45 mm	0.50 mm
1.00 mm	0.50 mm	0.45 mm	0.35 mm	0.40 mm	0.45 mm
0.80 mm	0.40 mm	0.30 mm	0.25 mm	0.30 mm	0.35 mm
0.50 mm	0.30 mm	0.30 mm	0.20 mm	0.25 mm	0.30 mm
0.40 mm	0.25 mm	0.25 mm	0.15 mm	0.20 mm	0.25 mm

Note: For the ball size of a given product, refer to the *Package Outline Drawing* in the product data sheet.

Note: In general, Non-Soldermask Defined (NSMD) pads are recommended for BGA package land pattern and the dimensions provided here are referenced to the pad diameter of an NSMD pad. Soldermask Defined (SMD) pads can be used based on the user's experience. For the case of SMD pads, dimension D in Table 3 should refer to the PCB soldermask opening.

Additional notes:

- For Non-Soldermask Defined pads, the soldermask opening Figure 6: “BGA Solder Joint Geometry,” on page 5 should account for any manufacturing tolerances such that the soldermask will not overlap the copper pad.
- All Broadcom package outlines are in accordance with the appropriate specification furnished by JEDEC design standard JEP95.

REFLOW PROFILE GUIDELINES

Exposed-Heatslug BGA packages can be treated as a standard surface mount ball grid array using a conventional surface mount process (screen print paste->place component->reflow). The following summarizes general reflow profile guidelines for Exposed-Heatslug BGA packages. The customer should conduct reflow tests to optimize the reflow profile for the specific application board. Further information is available in the Broadcom package reflow process guidelines application note (Packaging-AN101-R, *Reflow Process Guidelines for Surface Mount Assemblies)*.

Figure 7 shows a reference convection reflow profile. The parameter values for the reference reflow profile are provided in Table 4 on page 7.

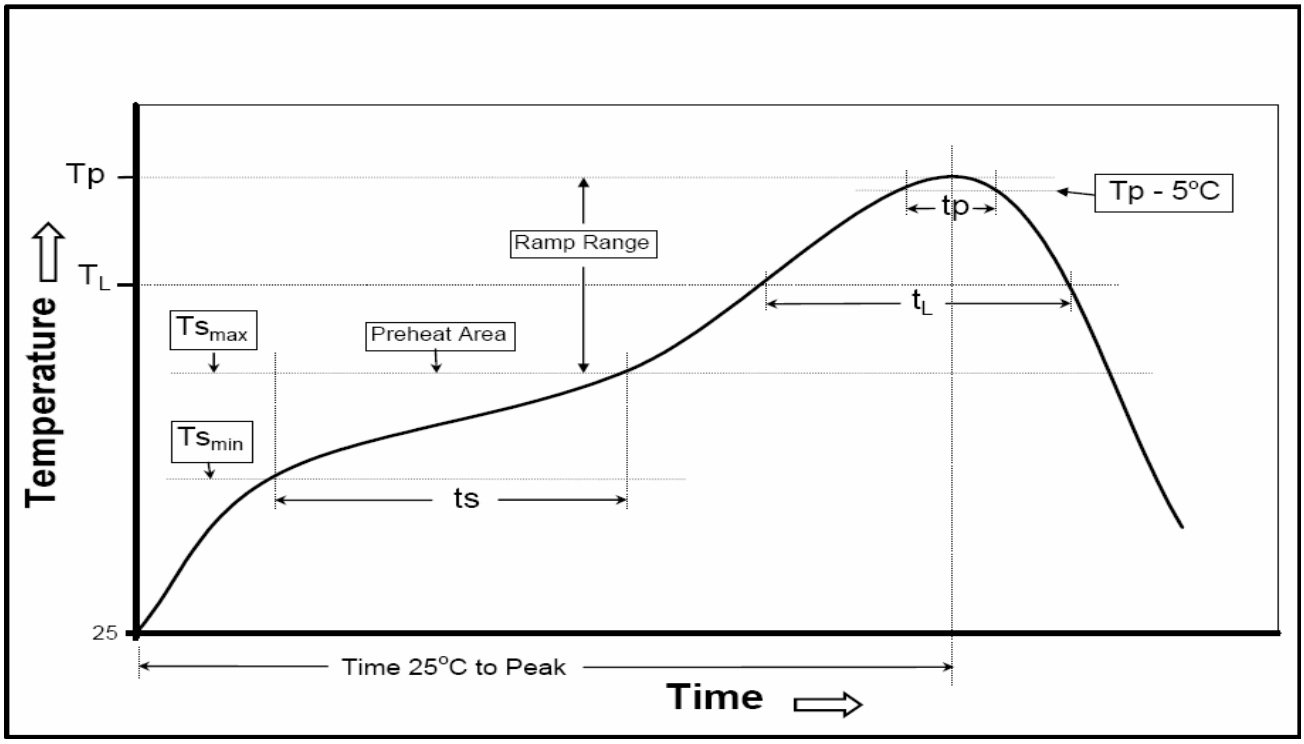


Figure 7: Reference Reflow Profile

Table 4: Reference Reflow Profile Parameters

Profile Feature	*Sn-Pb Eutectic Assembly*	*Pb-Free Assembly*
Average ramp-up rate (T_L to Tp)	2°C/second max.	2°C/second max.
Preheat:		
Temperature Min (Ts_{min})	100°C	150°C
Temperature Max (Ts_{max})	150°C	200°C
Time (min. to max) (ts)	60-120 seconds	60-180 seconds
Ts_{max} to T_L:		
Ramp-up Rate	2°C/second max	2°C/second max
Time maintained above:	183°C	217°C
Temperature (T_L)	60-90 seconds	60-90 seconds
Time (t_L)		
Peak temperature (Tp)	215-220°C	245-250°C
Time within 5°C of actual peak temperature (Tp)	20 seconds max.	20 seconds max.
Ramp-down rate	6°C/second max.	6°C/second max.
Time 25°C to peak temperature	6 minutes max.	8 minutes max.

Note: All temperatures refer to topside of the package, measured on the package body surface.
Note: Heatslug reflow temperature profile should be measured to check for proper soldering.

REWORK RECOMMENDATIONS FOR EXPOSED-HEATSLUG BGA PACKAGES

Note: Broadcom strongly recommends that components in Exposed-Heatslug BGA packages should not be reused after removal from PCB.

The exposed-heatslug soldered on PCB provides excellent heat sinking capability to the Exposed-Heatslug BGA packages. The exposed-heatslug also adds additional considerations to the package rework procedures. The exposed-heatslug is hidden underneath the package body. In order to heat up the exposed-heatslug above solder melting point during part removal process, the package mold body is typically subjected to higher temperatures than the same size BGA packages without an exposed-heatslug. Broadcom conforms to the JEDEC standard JESD22-A113C which dictates the reliability requirements based on a maximum of three reflow cycles. A package that has been attached to a board and then removed has already been subjected to two reflow cycles. Removal, re-balling and re-attaching a BGA part back to board will exceed the allowable reflow cycles adding three additional reflow cycles (removal, re-balling and re-mounting). Therefore, it is recommended that Exposed-Heatslug BGA packages be discarded after removal from PCB.

Broadcom Corporation

16215 Alton Parkway
P.O. Box 57013
Irvine, CA 92619-7013
Phone: 949-450-8700
Fax: 949-450-8710