# Application Notes for Surface Mount Assembly of Amkor's Thermally / Electrically Enhanced Leadframe Based Packages

December 2001

For additional information please contact marketing@amkor.com

# Contents

**1.0 Introduction** 1

**2.0 PCB Design Requirements** 2

**3.0 Board Mounting Guidelines** 4
- 3.1 Stencil Design 4
- 3.2 Assembly Process Flow 5
- 3.3 Reflow Guidelines 6

**4.0 Rework Process** 6
- 4.1 Removal of Old Component 7
- 4.2 Site Redress and Cleaning 7
- 4.3 Application of Solder Paste 7
- 4.4 Placement and Attachment of New Component 7

## 1.0 Introduction

All thermally/electrically enhanced leadframe technology based packages (ExposedPad™ ), TQFP, ExposedPad TSSOP, ExposedPad SOP, ExposedPad SSOP, PSOP& PSSOP, LQFP PQ2 & LQFP PQ4 ) offered by Amkor have exposed paddle or Slug on the bottom of the package to provide the primary heat removal path as well as excellent electrical grounding to the Printed Wiring Board (PWB). An ExposedPad package is one in which the die attach paddle is down-set such that the pad is exposed during the mold process, as shown in Figure 1. A bottom view of ExposedPad 10x10 TQFP is shown in Figure 2.

Although the land pattern design for lead attachment on the PCB should be the same as that for conventional, non-thermally/electrically enhanced packages, extra features are required during the PCB design and assembly stage for effectively mounting thermally/electrically enhanced packages. Also, repair and rework of assembled packages may involve some extra steps, depending upon the current rework practice within the company.

**Figure 1: Cross-Section of *e*Pad Package with Heat Transfer Schematic**

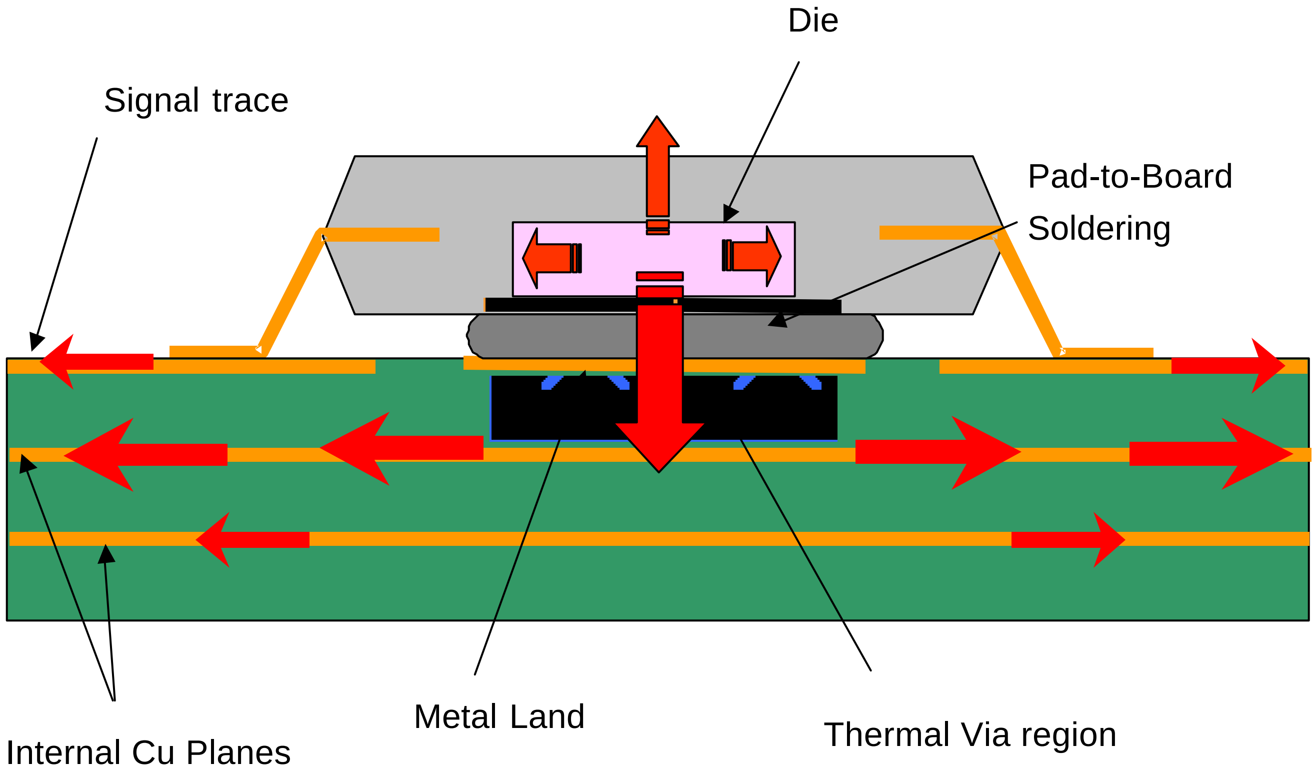
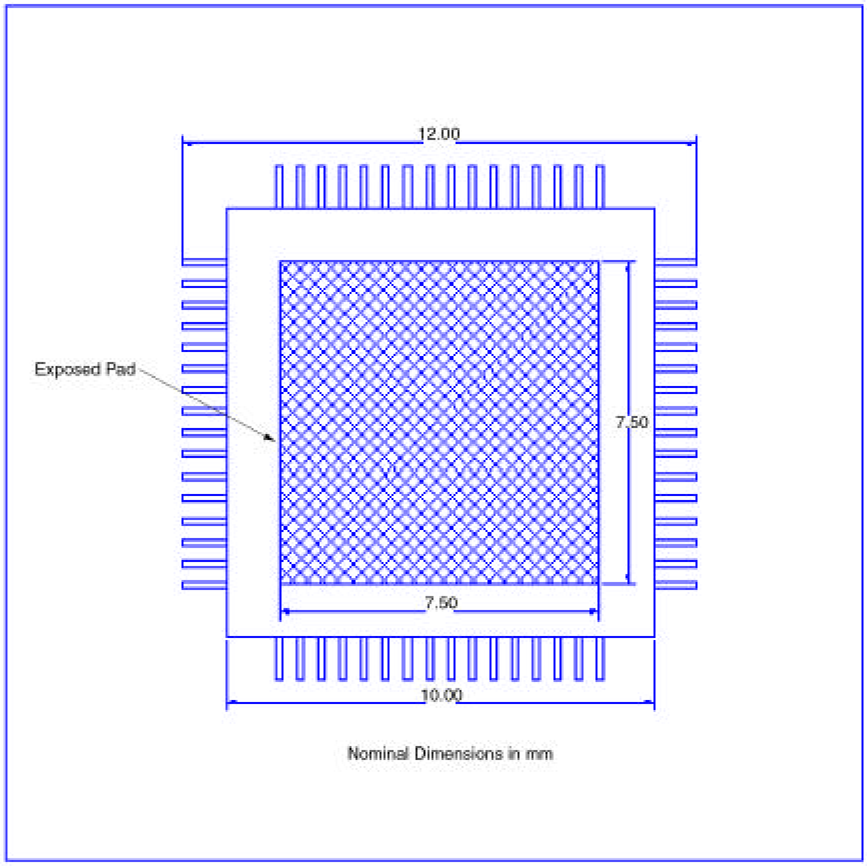


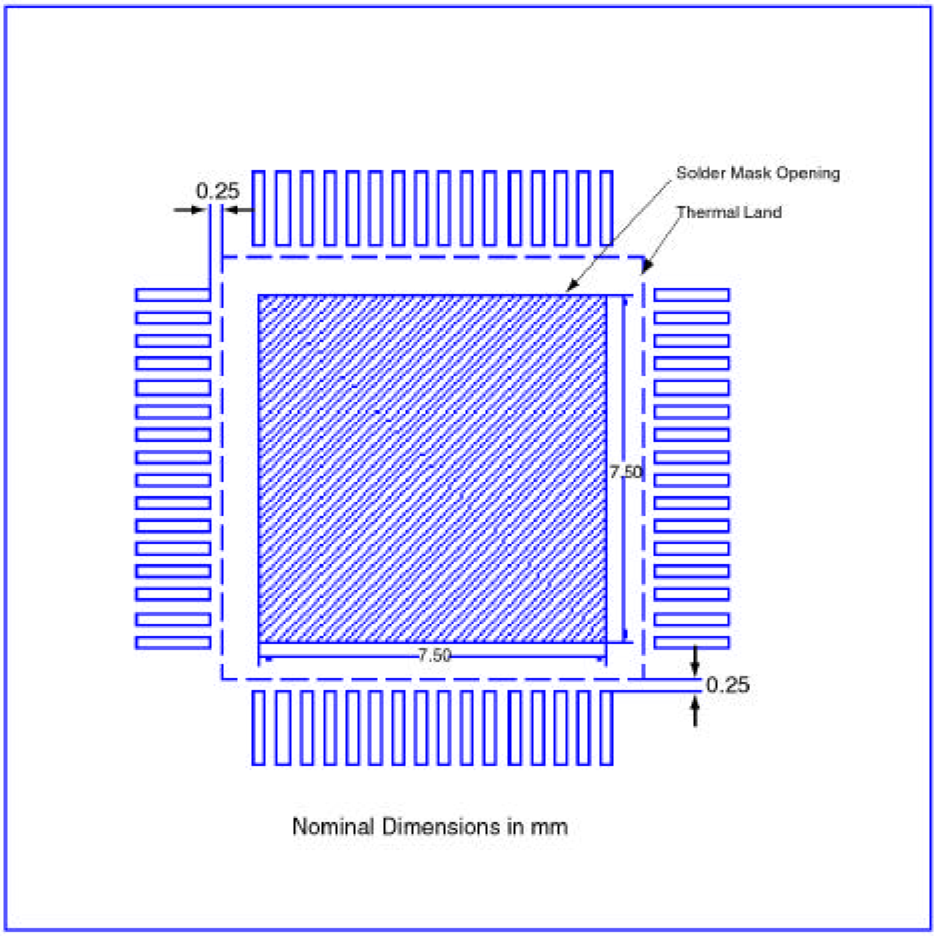
**Figure 2: Bottom View of Exposed Pad™ TQFP**



## 2.0 PCB Design Requirements

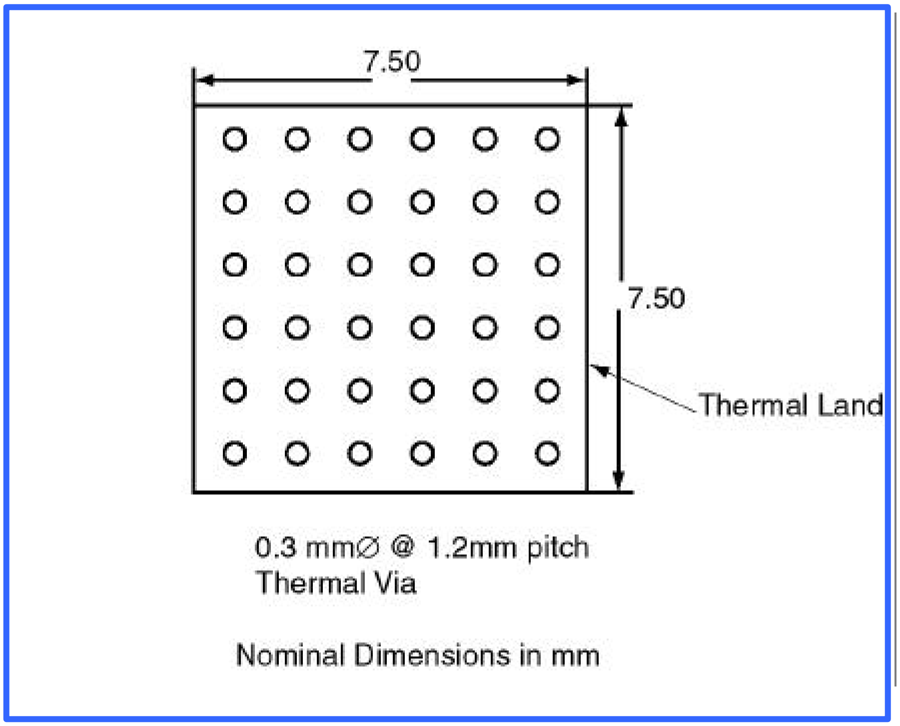
In order to maximize both the removal of heat from the package and the electrical performance, a land pattern must be incorporated on the PWB within the footprint of the package corresponding to the exposed metal pad or exposed heat slug on the package, as shown in Figure 3. The size of this land pattern can be larger, smaller, or even take on a different shape than the exposed pad on the package. However, the solderable area, as defined by the solder mask, should be at least the same size/shape as the exposed pad/slug area on the package to maximize the thermal/electrical performance. A clearance of at least 0.25mm should be designed on the PWB between the outer edges of the land pattern and the inner edges of pad pattern for the leads to avoid any shorts.

## Figure 3:ExposedPad™ TQFP Land Pattern



While the land pattern on the PWB provides a means of heat transfer/electrical grounding from the package to the board through a solder joint, thermal vias are necessary to effectively conduct from the surface of the PWB to the ground plane(s). These vias act as “heat pipes”. The number of vias (i.e. “heat pipes”) are application specific and dependent upon the package power dissipation as well as electrical conductivity requirements. Thus, thermal and electrical analysis and/or testing are recommended to determine the minimum number needed. Maximum thermal and electrical performance is achieved when an array of vias is incorporated in the land pattern at 1.2mm grid, as shown in Figure 4. It is also recommended that the via diameter should be 12 to 13mils (0.30 to 0.33mm) with 1 oz copper via barrel plating. This is desirable to avoid any solder wicking inside the via during the soldering process which may result in voids in solder between the exposed pad/slug and the thermal land. If the copper plating does not plug the vias, the thermal vias can be “tented” with solder mask on the top surface of the PWB to avoid solder wicking inside the via during assembly. The solder mask diameter should be at least 4mils (0.1 mm) larger than the via diameter. Note: These recommendations are to be used as a guideline only.

## Figure 4: ExposedPad™ TQFP Via Grid



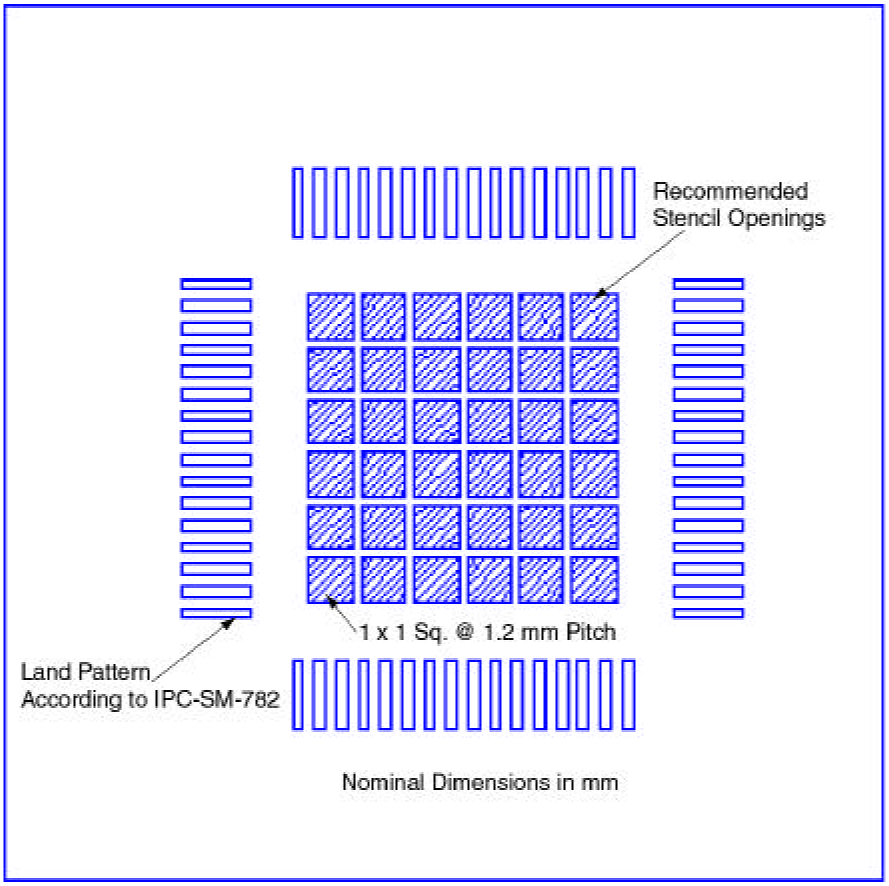
### 3.0 Board Mounting Guidelines

The following are general recommendations for mounting exposed pad/slug leadframe devices on the PWB. This should serve as the starting point in assembly process development and it is recommended that the process should be developed based on past experience in mounting standard, non-thermally/electrically enhanced packages.

### 3.1 Stencil Design

For maximum thermal/electrical performance, it is required that the exposed pad/slug on the package be soldered to the land pattern on the PWB. This can be achieved by applying solder paste on both the pattern for lead attachment as well as on the land pattern for the exposed pad. While for standard (non-thermally/electrically enhanced) leadframe based packages the stencil thickness depends on the lead pitch and package coplanarity, the package standoff must also be considered for the thermally/electrically enhanced packages to determine the stencil thickness. For a nominal standoff of 0.1mm, the stencil thickness of 5 to 8mils (depending upon the pitch) is recommended. The aperture openings should be the same as the solder mask opening on the land pattern (i.e. 1:1). Since a large stencil opening may result in poor release, the aperture opening should be subdivided into an array of smaller openings, similar to the thermal land pattern shown in Figure 5. The above guidelines will result in the solder joint area to be about 80 to 90% of the exposed pad/slug area.

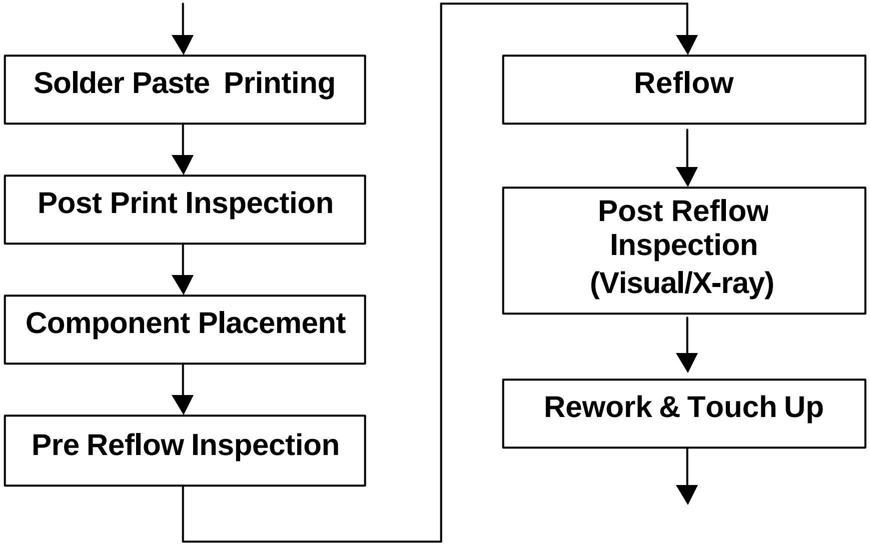
**Figure 5:  Recommended Stencil Design**



### 3.2 Assembly Process Flow

Figure 6 shows the typical process flow for mounting surface mount packages to the printed circuit boards.  The same process can be used for mounting exposed pad/slug packages without any modifications.

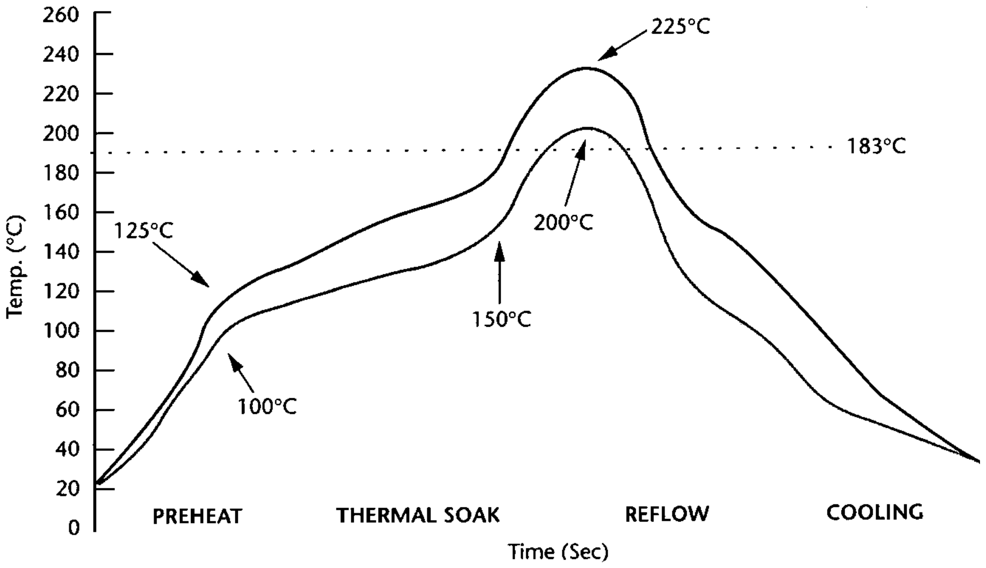
**Figure 6.**



## 3.3 Reflow Guidelines

The typical reflow profile consists of four sections. In the preheat section, the PWB assembly should be preheated at the rate of 2 to 3°C/sec to start the solvent evaporation and to avoid thermal shock. The assembly should then be thermally soaked for 60 to 120 seconds to remove solder paste volatiles and for activation of flux. In the reflow section of the profile, the time above liquidus (183°C for Eutectic Sn/Pb solder) should be between 30 to 60 seconds with a peak temperature in the range of 205 to 220°C. Finally, the assembly should undergo cooldown in the fourth section of the profile. A typical profile band is shown in Figure 7. The actual profile parameters depend upon the solder paste used and recommendations from paste manufacturers should be followed.

**Figure 7: Typical Profile Band for Sn63/Pb37 Alloy Solder Paste**



The reflow profile for exposed pad packages need not be any different than the one used for non-thermally/electrically enhanced packages. For parts with heat slugs (PSOPs and PSSOPs), however, the oven settings may need to be adjusted to achieve a reflow temperature between 205 to 220°C with time above liquidus of 30 to 60 seconds.

## 4.0 Rework Process

The following are the steps involved in the rework process of ExposedPad ,PSOP, and Power Quad packages:

? Removal of old component
? Site redress and cleaning
? Application of solder paste
? Placement and attachment of new component

### 4.1 Removal of Old Component

For proper removal of part, special nozzles should be used for local heating of the part to avoid reflow of adjacent parts. The PWB assembly should be preheated to about 75°C from the bottom using convective preheaters and the preheated nozzle should be lowered on the part. The proper nozzle should also heat the component leads by either hot gas or hot bar. The ideal reflow profile should be the same as the one used for mounting the part and the reflow zone should be between 20 to 60 seconds, depending upon the paste used. The reflow zone can be shortened as long as the reflow is complete. The part should then be lifted off automatically during the transition from reflow to cooldown cycles using a vacuum.

### 4.2 Site Redress and Cleaning

Once the part is removed, the site needs to be cleaned for attachment of a new package. This may be done by vacuum desoldering or wick. Low-temperature, blade style, conductive tools in conjunction with desoldering braids can also be used. Once all residual solder is removed, the site should be cleaned with appropriate solvent such as alcohol and a lint-free swab.

### 4.3 Application of Solder Paste

For a precise and uniform solder paste deposition on the redressed site, it is recommended that a miniaturized stencil for the individual component be used. The stencil should be aligned under 50x to 100x magnification, depending on the part. After precise alignment, the stencil should be lowered onto the PWB and the paste should be applied using a small metal squeegee blade. To avoid any inadvertent overprinting, the blade width should be the same as the stencil width and the paste should be applied in one pass.

### 4.4 Placement and Attachment of New Component

The new part should be placed on the site using split-beam alignment where a dual image of the part leads and the land pattern on the PCB can be viewed on a high-resolution monitor. Once aligned, the part should be placed on the site and attached to the board using the reflow profile used for the part removal.