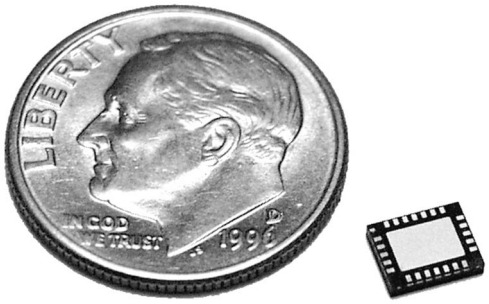
Leadless Leadframe Package (LLP)

National Semiconductor
Application Note 1187
February 2006

Table of Contents



20005901

Introduction

The Leadless Leadframe Package (LLP) is a leadframe based chip scale package (CSP) that may enhance chip speed, reduce thermal impedance, and reduce the printed circuit board area required for mounting. The small size and very low profile make this package ideal for high density PCBs used in small-scale electronic applications such as cellular phones, pagers, and handheld PDAs. The LLP package is offered in the Pullback and no Pullback configuration. In the Pullback configuration the standard solder pads are offset from the edge of the package by 0.1 mm. In the no Pullback configuration the standard solder pads extend and terminate at the edge of the package. This feature offers a visible solder fillet after board mounting.

The LLP has the following advantages:

- Low thermal resistance
- Reduced electrical parasitics
- Improved board space efficiency
- Reduced package height
- Reduced package mass

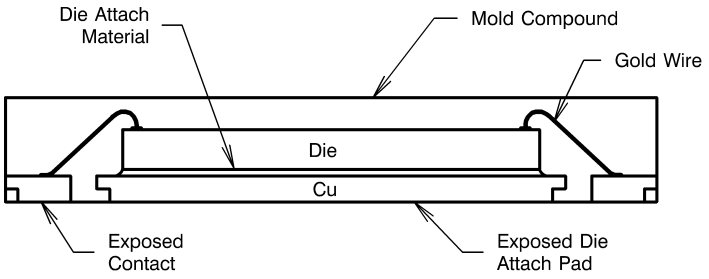
JEDEC Registration Information:

- Quad LLP Packages: MO-220
- Dual-in-line LLP Packages: MO-229

Package Overview

KEY ATTRIBUTES

- Construction of the LLP is Illustrated in *Figure 1*, *Figure 2* and *Figure 3*.
- Terminal contacts:
  - The contact pads (or solder pad) are located peripherally in single row format depending on the specific number of pins and body size.
  - For certain specific applications the packages are incorporated with common power and/or ground pins as illustrated in *Figure 8*.
  - All LLP contacts are plated with 85Sn/15Pb solder for ease of surface mount processing.
  - All Lead-Free LLP contacts are plated with matte tin solder for ease of surface mount processing.
- Printed Circuit Board (PCB) footprint:
  - Soldering the exposed die attach pad (DAP) to the PCB provides the following advantages:
    - Optimizes thermal performance.
    - Enhances solder joint reliability.
    - Facilitates package self alignment to the PCB during reflow.
- The LLP is offered in either dual-in-line (DIP) or quad configuration, and Pullback or No Pullback terminal contact designs.
- The LLP package also comes in different thicknesses. 0.8mm is the most prevalent thickness but the package is selectively available also in 0.6mm thickness.
- Coplanarity is not an area of concern for this package.
  - All LLP contacts are flush with the bottom of the package.
- Moisture Sensitivity Level (MSL).
  - MSL of specific applications, requiring large packages, may vary depending on die size, exposed DAP design, and number of downbonds.



20005946

FIGURE 1. Pullback LLP Configuration

Package Overview (Continued)

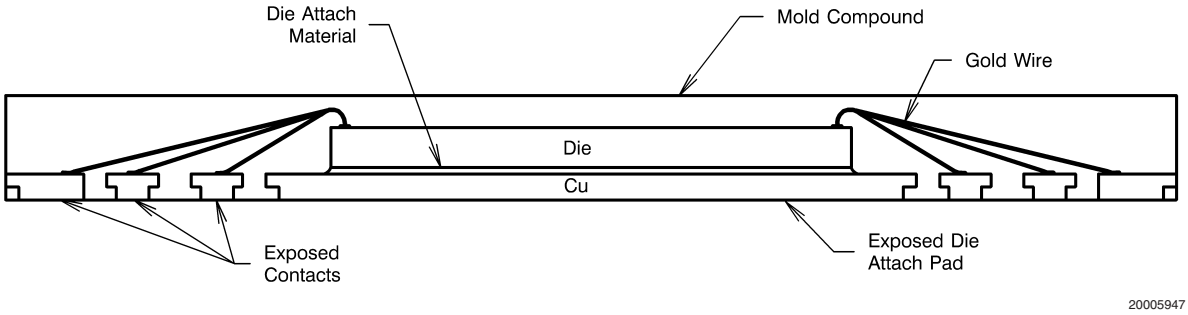


FIGURE 2. Pullback LLP Configuration with Power & Ground Rings

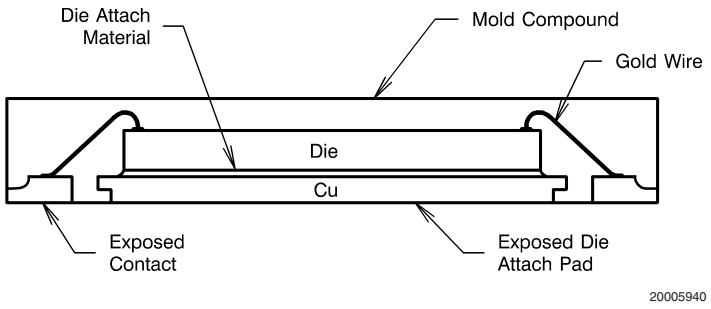


FIGURE 3. No Pullback LLP Configuration

PACKAGE OFFERING

Number	I/O Count	Body size (mm)	Terminal Pitch (mm)	Marketing Drawing
Pullback (DUAL)				
1	6	2.2 X 2.5	0.65	LDB06A
2	6	3 X 4	0.8	LDC06D
3	6	2.92 X 3.29	0.95	LDE06A
4	8	3 X 3	0.5	LDA08A
5	8	2.5 X 2.5	0.5	LDA08B
6	8	2.5 X 3	0.5	LDA08C
7	8	4 X 4	0.8	LDC08A
8	10	3 X 3	0.5	LDA10A
9	10	3 X 4	0.5	LDA10B
10	10	4 X 4	0.8	LDC10A
11	14	4 X 4	0.5	LDA14A
12	14	4 X 5	0.5	LDA14B
13	14	4 X 3	0.5	LDA14C
14	16	5 X 5	0.5	LDA16A
Pullback (QUAD)				
1	8	2 X 2	0.65	LQB08A
2	10	5 X 4	0.8	LQC10A
3	16	4 X 4	0.5	LQA16A
4	20	4 X 4	0.5	LQA20A
5	24	5 X 4	0.5	LQA24A
6	24	6 X 6	0.8	LQC24A
7	28	5 X 5	0.5	LQA28A

Package Overview (Continued)

Number	I/O Count	Body size (mm)	Terminal Pitch (mm)	Marketing Drawing
8	32	6 X 5	0.5	LQA32B
9	32	6 X 6	0.5	LQA32A
10	36	6 X 6	0.5	LQA36A
11	36	6 X 6	0.5	LQA36B
12	44	7 X 7	0.5	LQA44A
13	44	7 X 7	0.5	LQA44B
14	48	7 X 7	0.5	LQA48B
15	56	9 X 9	0.5	LQA56A
Pullback (QUAD, Dual Row) QUALIFICATION IN PROGRESS				
1	80	8 X 8	0.5	LQA80A
Pullback 0.6 mm Thin (QUAD)				
1	28	5 X 5	0.5	LPA28A
2	44	7 X 7	0.5	LPA44A
No Pullback (DUAL)				
1	6	2.2 X 2.5	0.65	SDB06A
2	6	3 X 3	0.95	SDE06A
3	8	2.5 X 2.5	0.5	SDA08B
4	8	2.5 X 3	0.5	SDA08C
5	8	3 X 3	0.5	SDA08A
6	8	4 X 4	0.8	SDC08A
7	10	3 X 3	0.5	SDA10A
8	10	4 X 4	0.8	SDC10A
9	14	4 X 3	0.5	SDA14A
10	14	4 X 4	0.5	(Note 1)
11	14	4 X 5	0.5	(Note 1)
12	16	5 X 5	0.5	SDA16A
No Pullback (QUAD)				
1	10	5 X 4	0.5	(Note 1)
2	16	4 x 4	0.5	SQA16A
3	24	4 X 4	0.5	SQA24A
4	24	5 X 4	0.5	SQA24B
5	28	5 X 5	0.5	SQA28A
6	32	5 X 5	0.5	(Note 1)
7	32	6 X 5	0.5	(Note 1)
8	36	6 X 6	0.5	SQA36A
9	40	5 X 5	0.4	SQF40A
10	48	6 X 6	0.4	SQF48A
11	48	7 X 7	0.5	SQA48A
No Pullback 0.6 mm Thin (QUAD)				
1	28	5 X 5	0.5	SPA28A
2	48	7 X 7	0.5	SPA48A
No Pullback 0.4 mm Ultra Thin (QUAD)				
1	28	5 X 5	0.5	SNA28A
2	40	6 X 6	0.5	SNA40A
3	48	7 X 7	0.5	SNA48A

Note 1: Currently Under Development

PCB Design Recommendations

NSMD VS. SMD LAND PATTERN

Two types of land patterns are used for surface mount packages: (1) Non-Solder Mask Defined Pads (NSMD) and (2) Solder Mask Defined Pads (SMD). NSMD has an opening that is larger than the pad, whereas SMD pads have a solder mask opening that is smaller than the metal pad. *Figure 4* Illustrates the two different types of pad geometry.

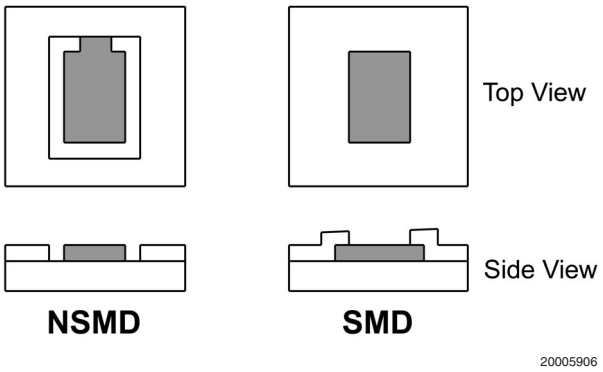


FIGURE 4. NSMD and SMD Pad Geometry

NSMD is preferred because the copper etch process has tighter control than the solder masking process. Moreover, the smaller size of the copper pad in the NSMD definition facilitates escape routing on the PCB when necessary.

NSMD pads require a ±0.075 mm (3 mils) clearance around the copper pad for mask registration tolerances.

SMD pad definition can introduce stress concentration points near the solder mask on the PCB side. Extreme environmental conditions such as large temperature variations may cause fatigue that leads to cracked solder joints and reliability problems.

For optimal reliability, National recommends a 1:1 ratio between the package pad and the PCB pad for the Pullback LLP. If probing of signal pad is required, it is recommended to design probe pads adjacent to signal pads as shown in *Figure 5*. The trace between the signal pad and the probe pad must be covered by solder mask.

For No Pullback LLP, we recommended the PCB terminal pads to be 0.2mm longer than the package pads to create a solder fillet to improve reliability and inspection.

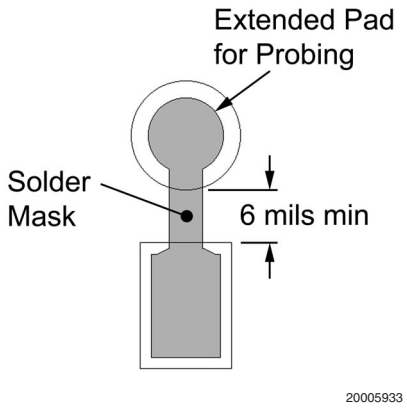
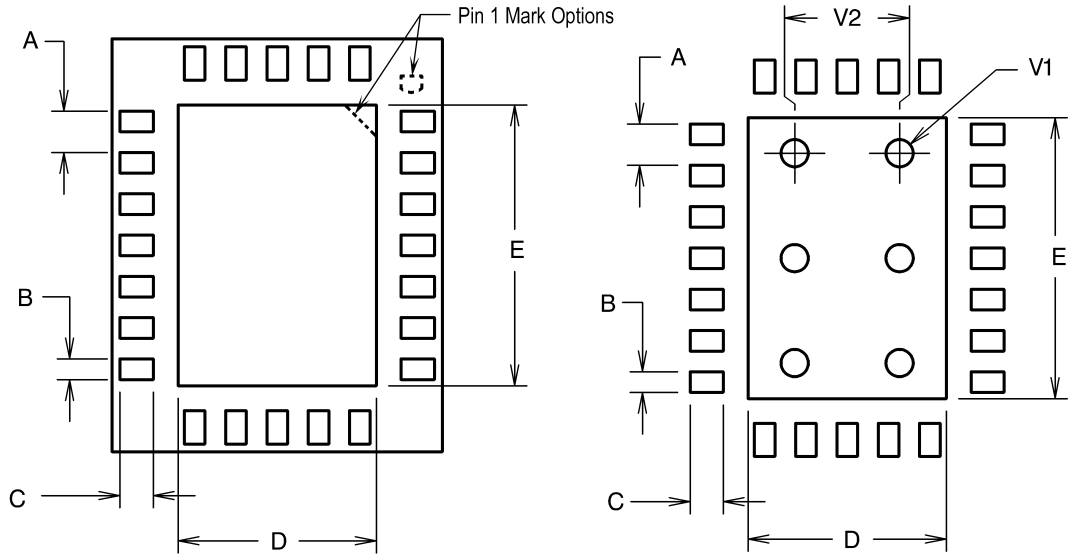


FIGURE 5. Recommended Pad Design for Probing

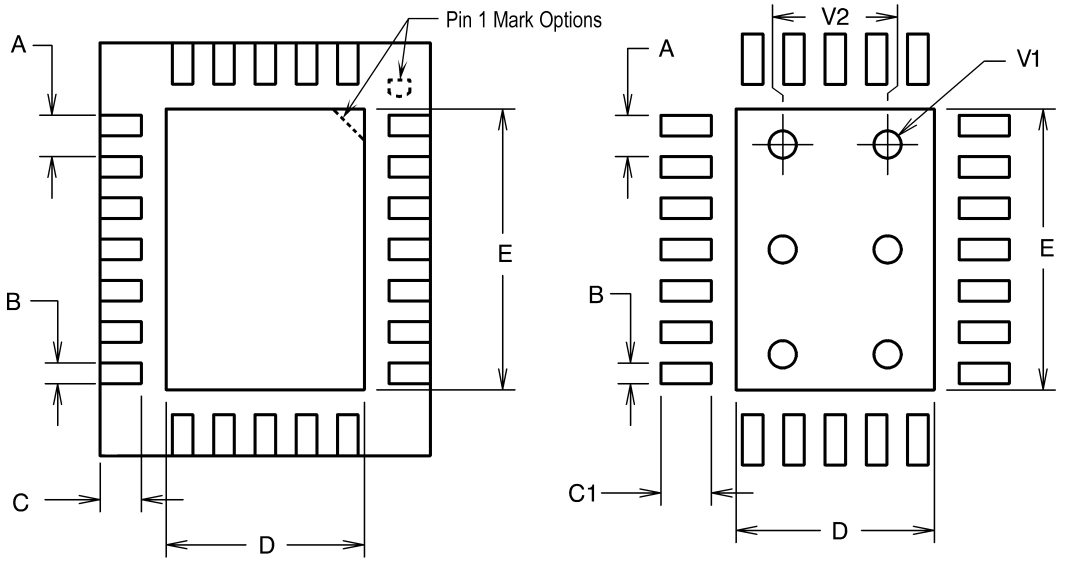
PCB Design Recommendations (Continued)



Dimensions A, B, C, D, and E of PCB are 1:1 ratio with package pad dimensions. For specific detailed package dimensions refer to respective marketing outlines.	
Terminal Pitch	A
Terminal Width	B
Terminal Length	C
Exposed DAP Width	D
Exposed DAP Length	E
Thermal Via Diameter. Recommended 0.2 - 0.33 mm	V1
Thermal Via Pitch. Recommended 1.27 mm	V2

FIGURE 6. Typical Recommended Printed Circuit Board for Pullback Packages

PCB Design Recommendations (Continued)



LLP Package Outline

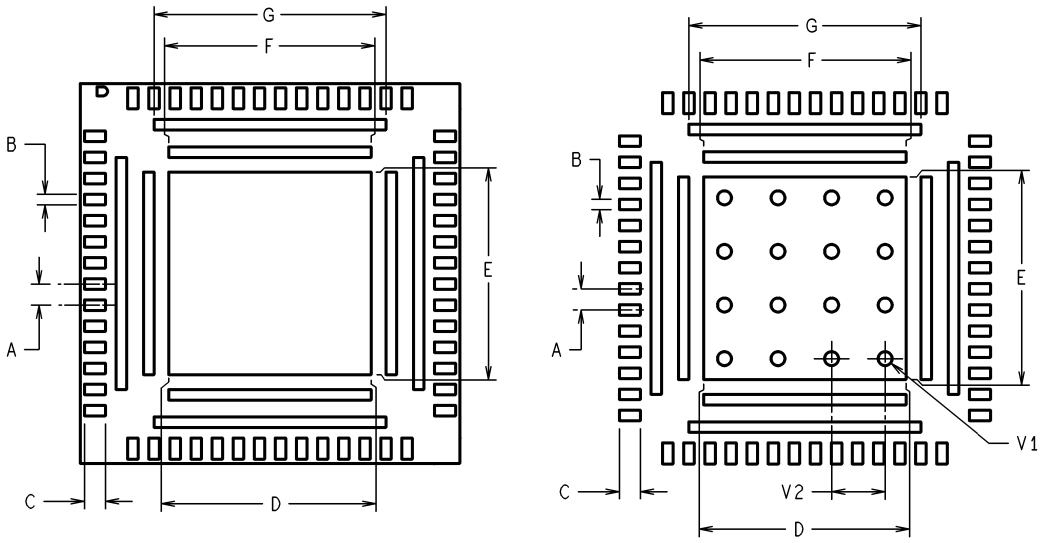
PCB Outline

20005950

Dimensions A, B, D, and E of PCB are 1:1 ratio with package pad dimensions. For specific detailed package dimensions refer to respective marketing outlines.	
Terminal Pitch	A
Terminal Width	B
Terminal Length	C
PCB Pad Length C + 0.2 mm	C1
Exposed DAP Width	D
Exposed DAP Length	E
Thermal Via Diameter. Recommended 0.2 - 0.33 mm	V1
Thermal Via Pitch. Recommended 1.27 mm	V2

FIGURE 7. Typical Recommended Printed Circuit Board for No Pullback Packages

PCB Design Recommendations (Continued)



LLP Package Outline PCB Outline

20005927

Dimensions A, B, C, D, E, F and G of PCB are 1:1 ratio with package pad dimensions. For specific detailed package dimensions refer to respective marketing outlines.	
Terminal Pitch	A
Terminal Width	B
Terminal Length	C
Exposed DAP Width	D
Exposed DAP Length	E
Ground Bar	F
Power Bar	G
Thermal Via Diameter	V1
Thermal Via Pitch	V2

FIGURE 8. Typical Recommended Printed Circuit Board for Pullback Packages with Ground and Power Bars.

PCB Design Recommendations

(Continued)

THERMAL DESIGN CONSIDERATIONS

THERMAL LAND

The LLP thermal land is a metal (normally copper) region centrally located under the package and on top of the PCB. It has a rectangular or square shape and should match the dimensions of the exposed pad on the bottom of the package (1:1 ratio).

For certain high power applications, the PCB land may be modified to a "dog bone" shape that enhances thermal performance. The packages used with the "dog bone" lands will be a dual inline configuration. (See *Figure 9*).

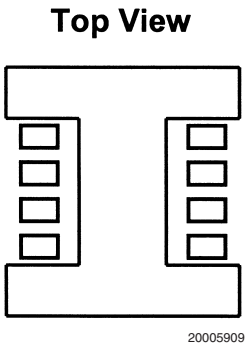


FIGURE 9. Dog Bone

THERMAL VIAS

Thermal vias are necessary. They conduct heat from the exposed pad of the package to the ground plane. The number of vias is application specific and is dependent upon electrical requirements and power dissipation.

An array of vias with a 1.27 mm pitch is shown in *Figure 6*. The via diameter should be 0.2 mm to 0.33 mm with 1oz. copper via barrel plating. It is important to plug the via to avoid any solder wicking inside the via during the soldering process. The thermal vias can be tented with solder mask on the top surface of the PCB. The solder mask diameter should be at least 75 microns (or 3 mils) larger than the via diameter. The solder mask thickness should be the same across the entire PCB.

A package thermal performance may be improved by increasing the number of vias. *Figure 10* shows such effect for a 36L LLP with a 9 x 9 mm body and 7 x 7 mm pad. Two via diameters are illustrated, namely, 0.2 and 0.33 mm. Different patterns are also shown to depict possible layouts for specific numbers of vias. Two die sizes are shown in this example, 2.1 x 2.1 mm and 6.4 x 6.4 mm. For a given number of vias, placing the vias toward the periphery of the pad provides up to 5% improvement over centrally placed vias. There is diminishing improvement, however, as the number of vias increases beyond a critical number – 16 in this example.

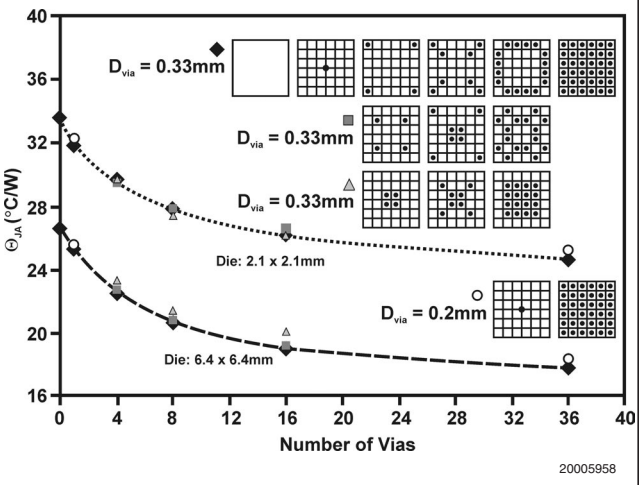


FIGURE 10. θ_{JA} vs. Number , Distribution and Diameter of Thermal Vias and Die Sizes for 36L LLP with 9x9 mm Body and 7x7 mm Pad

EFFECTS OF SOLDER VOIDS

A void in the solder joint (generated during the manufacturing process) could have a direct impact on heat dissipation. The effect is not significant unless the void volume exceeds a certain percentage of the corresponding material volume (see *Figure 11*).

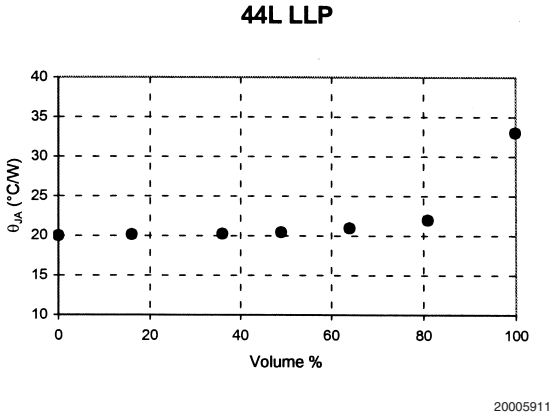


FIGURE 11. Thermal Voids Impact

THERMAL LAYERS IN THE PCB

Because of the small size and low profile, the majority of heat generated by the die within the LLP is dissipated through the exposed pad to PCB. Consequently, the PCB configuration and metal layers embedded in the PCB become important to achieve good thermal performance. In a 4-layer PCB (2 layers for signals and 2 layers for power/ground), the area of the embedded copper layer connecting

PCB Design Recommendations

(Continued)

to the thermal vias has significant effect on the thermal performance of the package. *Figure 12* shows simulation data of θ_{JA} vs. the embedded copper layer area for the 44L LLP. Increasing the copper layer area reduces the thermal resistance. However, in the similar manner, as the number of vias increases, the amount of thermal resistance improvement diminishes as the embedded copper area increases.

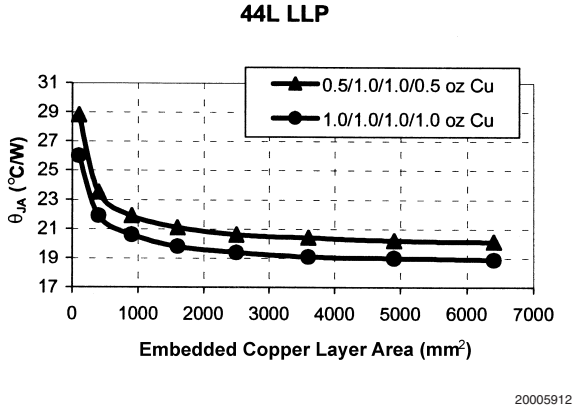


FIGURE 12. Effect of Thermal Layers on the LLP's Junction-to-Ambient Thermal Resistance

THETA JA ON 4L AND 2L JEDEC BOARDS

θ_{JA} is strongly dependent on the board on which packages are mounted. θ_{JA} of LLP packages on 4L JEDEC board with vias and 2L JEDEC boards are simulated and illustrated in *Figure 13 — Figure 14* and *Figure 15 — Figure 16*, respectively.

The maximum via number is used when doing the simulation. LLP packages are classified into different groups (shown as rectangular bands) according to body size. For each body size, two pad sizes with largest and smallest areas are selected. For each pad size, two die sizes with the same as and one fourth of the exposed pad area are selected. This is shown by the inserts in *Figure 13 — Figure 16*.

As an example, consider *Figure 13* depicting θ_{JA} values for six body sizes (2.5 x 2.5 mm, 2.5 x 3 mm, 3 x 3 mm, 2.92 x 3.29 mm, 3 x 4 mm, and 4 x 4 mm). The 2.5 x 3 body size shows the following θ_{JA} values at the four corners: 70.5˚C/W for a die of the same size as a pad of 1.75 mm^2 (lower left corner of 2.5 x 3 block); 98˚C/W for a die ¼ the area of a pad of 1.75 mm^2 (upper left corner of 2.5 x 3 block); 84˚C/W for a die ¼ the area of a pad of 2.4 mm^2 (upper right corner of 2.5 x 3 block); 62.5˚C/W for a die of the same size as the area of a pad of 2.4 mm^2 (lower right corner of 2.5 x 3 block). Die and pad size combinations falling within this 2.5 x 3mm body block will exhibit θ_{JA} values covered by this block.

When the body, pad and die sizes of a LLP package are given, the corresponding block can be used to get a quick evaluation for its θ_{JA} by interpolation according to pad and die sizes. It is noted that due to the nonlinear variation of θ_{JA} with pad sizes, die sizes, and I/O number, the θ_{JA} values in *Figure 13 — Figure 16* may involve up to 15% error. If more accurate data are expected, a separate simulation or experimental test is necessary.

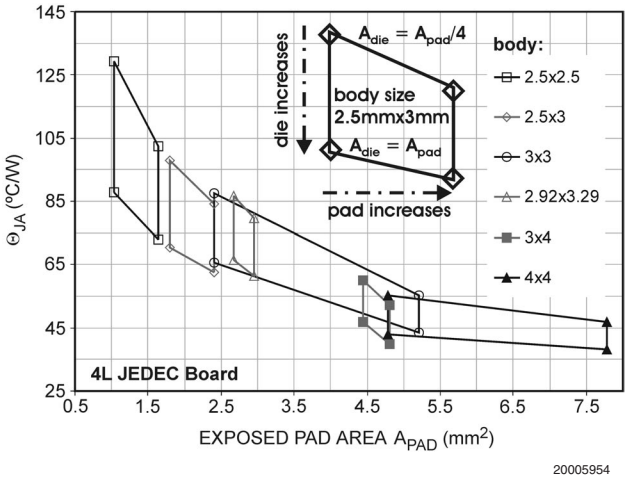


FIGURE 13. θ_{JA} vs. Body, Pad and Die Sizes for 4L JEDEC Board.

PCB Design Recommendations

(Continued)

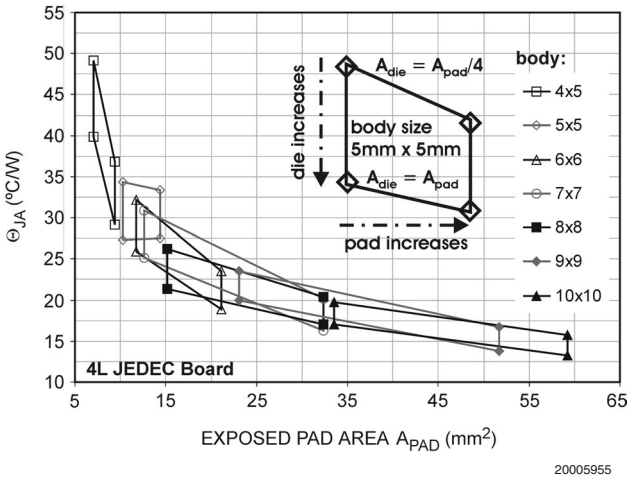


FIGURE 14. θ_{JA} vs. Body, Pad and Die Sizes for 4L JEDEC Board.

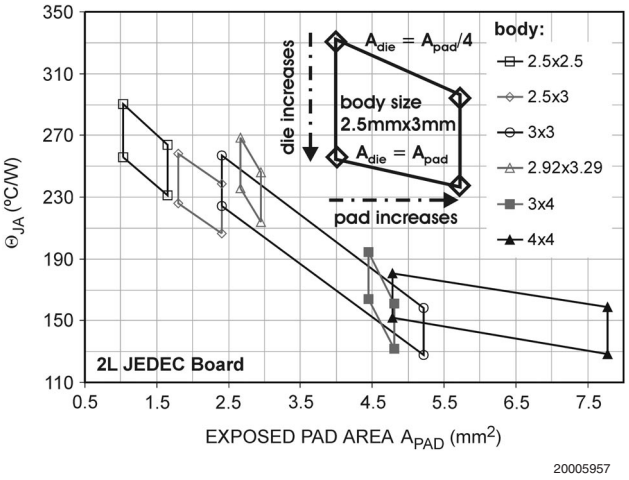


FIGURE 15. θ_{JA} vs. Body, Pad and Die Sizes for 2L JEDEC Board.

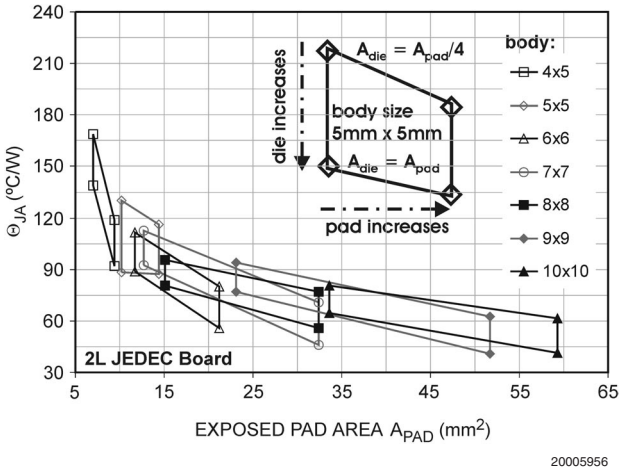


FIGURE 16. θ_{JA} vs. Body, Pad and Die Sizes for 2L JEDEC Board.

SMT Assembly Recommendations

The LLP surface mount assembly operations include:

- PCB plating requirements
- Screen printing the solder paste on the PCB
- Monitor the solder paste volume (uniformity)
- Package placement using standard SMT placement equipment
- X-ray pre reflow check - paste bridging
- Reflow and cleaning (dependent upon the flux type)
- X-ray post reflow check - solder bridging & Voids

SMT Assembly Recommendations (Continued)

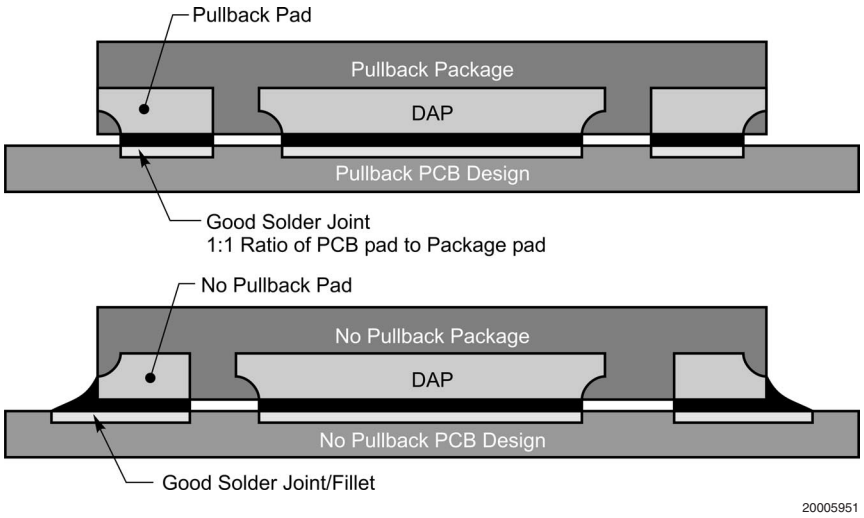


FIGURE 17. Recommended PCB Design

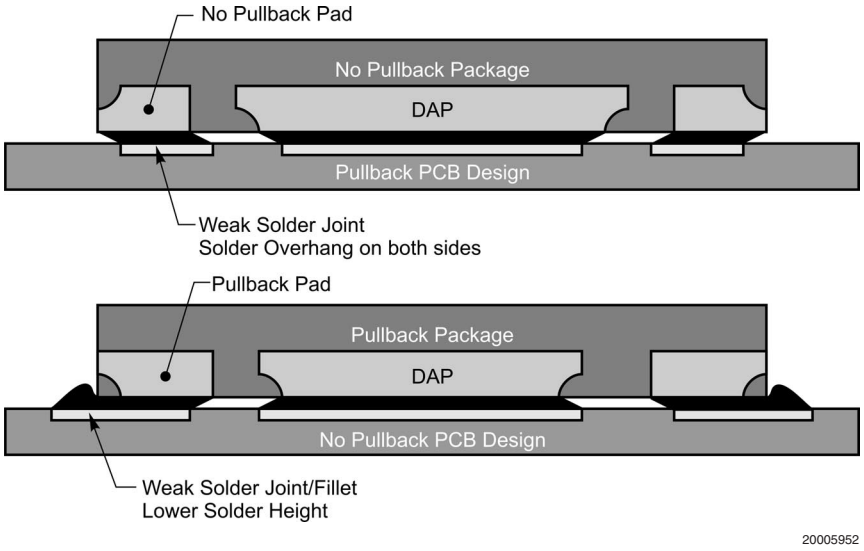


FIGURE 18. Not Recommended PCB Design

PCB SURFACE FINISH REQUIREMENTS

A uniform PCB plating thickness is key for high assembly yield.

- For an electroless nickel immersion gold finish, the gold thickness should range from 0.05 µm to 0.20 µm to avoid solder joint embrittlement.
- Using a PCB with **O**rganic **S**olderability **P**reservative coating (OSP) finish is also recommended, as an alternative to Ni-Au.
- For a PCB with Hot Air Solder Leveling (HASL) finish, the surface flatness should be controlled within 28 micron.

SOLDER STENCIL

Solder paste deposition using a stencil-printing process involves the transfer of the solder paste through pre-defined apertures with the application of pressure. Stencil parameters such as aperture area ratio and the fabrication process have a significant impact on paste deposition. Inspection of the stencil prior to placement of the LLP package is highly recommended to improve board assembly yields.

Laser cut stencil with electro polish is recommended. Tapered aperture walls (5˚ tapering) is recommended to facilitate paste release. Recommended stencil thickness is 127 µm for pitches ≥ 0.5 mm. The recommended stencil thickness is 100 µm for 0.4 mm pitch. In order to prevent solder bridging the stencil aperture openings need to be modified as follows:

- The terminal contact aperture openings should be offset by 0.1mm outward from the center of the package. For package with dual row terminal contacts, only the outer row terminal contacts require the 0.1mm offset.

SMT Assembly Recommendations

(Continued)

- For exposed pad aperture, up to 2 mm, the opening should be reduced to 95% of the corresponding PCB exposed DAP dimensions.
- For exposed pad aperture with any side from 2 to 4 mm, the stencil opening should be split in two for any side.
- For exposed pad aperature greater than 4 mm but without ground and power bars, refer to Table 2.
- For exposed pad aperature greater than 4 mm with ground and power bars, refer to *Figure 29*.
- For Pullback LLP with single row terminal contacts, stencil aperture is 1:1 with 0.1 mm offset.
- For dual row packages, only the outer row will require the 0.1 mm offset.
- For No Pullback packages with single row terminal contacts, stencil aperture is 0.1mm longer than PCB pads with 0.1 mm offset away from the center of the package.
- For No Pullback packages with dual row terminal contacts, the inner row stencil aperture is 1:1 to the package pads without 0.1 mm offset. The outer row will be 0.1 mm longer than the package pads with 0.1 mm offset away from the center of the package.
- For better control of package flatness after reflow, we recommend the stencil aperture for the DAP as shown in Table 1.

TABLE 1. Recommended DAP Stencil Aperture

Package DAP Size	Number of Openings	Gap Between Openings	Percentage of Solder Coverage
Less than 2mm X 2mm	1	N/A	80 - 90%
2.1mm - 4.4mm	4	0.2 - 0.3mm	70 - 85%
> 4.4mm	Vary	0.2 - 0.3mm	65 - 85%

TABLE 2. Pullback LLP Stencil Aperture Summary, See *Figure 19* .

Pin Count	MKT Dwg	PCB I/O Pad Size (mm)	PCB Pitch (mm)	PCB DAP size (mm)	Stencil I/O Aperture (mm)	Stencil DAP Aperture (mm)	Number of DAP Aperture Openings	Gap Between DAP Aperture (Dim A mm)
6	ldb06a	0.25 X 0.4	0.65	1.2 X 0.75	0.25 X 0.4	1.1 X 0.7	1	N/A
6	ldc06d	0.3 X 0.5	0.8	2 X 2.2	0.3 X 0.5	1.8 X 1.98	1	N/A
6	lde06a	0.35 X 0.5	0.95	1.92 X 1.2	0.35 X 0.5	1.8 X 1.1	1	N/A
8	lda08a	0.25 X 0.5	0.5	1.8 X 1.2	0.25 X 0.5	1.7 X 1.1	1	N/A
8	lda08b	0.25 X 0.5	0.5	1.5 X 0.7	0.25 X 0.5	1.4 X 0.6	1	N/A
8	lda08c	0.25 X 0.5	0.5	1.5 X 1.2	0.25 X 0.5	1.4 X 1.1	1	N/A
8	ldc08a	0.3 X 0.5	0.8	3 X 2.2	0.3 X 0.5	1.3 X 0.9	4	0.2
10	lda10a	0.25 X 0.5	0.5	2 X 1.2	0.25 X 0.5	1.9 X 1.1	1	N/A
14	lda14b	0.25 X 0.5	0.5	3 X 3.2	0.25 X 0.5	1.3 X 1.4	4	0.2
14	ldc14a	0.4 X 0.5	0.8	4.35 X 3	0.4 X 0.5	1.9 X 1.3	4	0.2
16	lqa16a	0.25 X 0.5	0.5	2.2 X 2.2	0.25 X 0.5	1.9 X 1.9	1	N/A
20	lqa20a	0.25 X 0.5	0.5	2.2 X 2.2	0.25 X 0.5	1.9 X 1.9	1	N/A
24	lqa24a	0.25 X 0.4	0.5	3.4 X 2.4	0.25 x 0.4	1.5 X 1.0	4	0.2
24	lqc24a	0.3 X 0.5	0.8	4.2 X 4.2	0.3 X 0.5	1.9 X 1.9	4	0.2
28	lqa28a	0.25 X 0.5	0.5	3.2 X 3.2	0.25 X 0.5	1.4 X 1.4	4	0.2
32	lqa32a	0.25 X 0.5	0.5	4.2 X 4.2	0.25 X 0.5	1.9 X 1.9	4	0.2
32	lqa32b	0.25 X 0.5	0.5	4.2 X 3.2	0.25 X 0.5	1.9 X 1.4	4	0.2
44	lqa44a	0.25 X 0.5	0.5	4.3 X 4.3	0.25 X 0.5	1.9 X 1.9	4	0.2

SMT Assembly Recommendations (Continued)

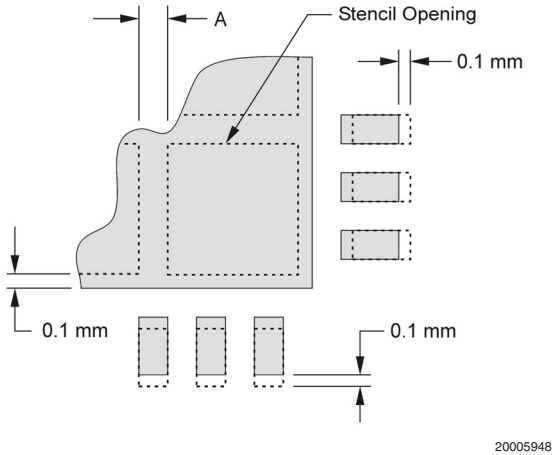


FIGURE 19. Pullback LLP, Single Row

TABLE 3. No Pullback LLP Stencil Aperture Summary, See *Figure 20*.

Pin Count	MKT Dwg	PCB I/O Pad Size (mm)	PCB Pitch (mm)	PCB DAP size (mm)	Stencil I/O Aperture (mm)	Stencil DAP Aperture (mm)	Number of DAP Aperture Openings	Gap Between DAP Aperture (Dim A mm)
6	sde06a	0.25 X 0.6	0.95	2.1 X 1.6	0.25 X 0.7	1.9 X 1.4	1	N/A
8	sdc08a	0.3 X 0.6	0.8	3 X 2.6	0.3 X 0.7	1.3 X 1.1	4	0.2
10	sdc10a	0.3 X 0.6	0.8	3 X 2.6	0.3 X 0.7	1.3 X 1.1	4	0.2
16	sda16a	0.25 X 0.6	0.5	4 X 3.6	0.25 X 0.7	1.8 X 1.6	4	0.2
24	sqa24a	0.25 X 0.6	0.5	2.6 X 2.6	0.25 X 0.7	1.1 X 1.1	4	0.2
28	sna28a	0.25 X 0.6	0.5	3.6 X 3.6	0.25 X 0.7	1.6 X 1.6	4	0.2
28	sqa28a	0.25 X 0.6	0.5	3.6 X 3.6	0.25 X 0.7	1.6 X 1.6	4	0.2
36	sqa36a	0.25 X 0.6	0.5	4.6 X 4.6	0.25 X 0.7	1.0 X 1.0	16	0.2
40	sna40a	0.25 X 0.6	0.5	4.6 X 4.6	0.25 X 0.7	1.0 x 1.0	16	0.2
40	sqf40a	0.20 X 0.6	0.4	3.6 X 3.6	0.2 X 0.8	1.6 X 1.6	4	0.2
48	sna48a	0.25 X 0.6	0.5	5.1 X 5.1	0.25 X 0.7	1.1 X 1.1	16	0.2
48	sqf48a	0.20 X 0.6	0.4	4.6 X 4.6	0.2 X 0.8	0.95 X 0.95	16	0.2
48	sqa48a	0.25 X 0.6	0.5	5.1 X 5.1	0.25 X 0.7	1.1 X 1.1	16	0.2

SMT Assembly Recommendations (Continued)

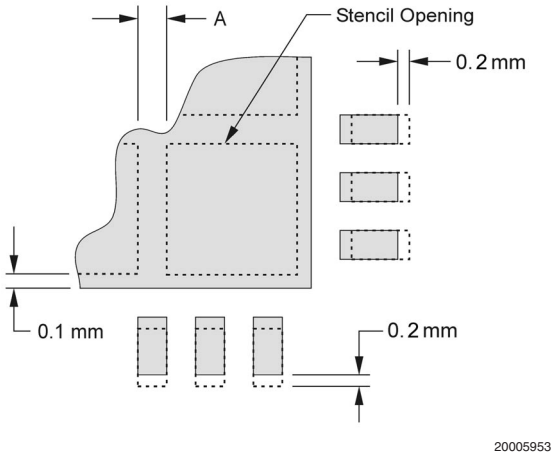


FIGURE 20. No Pullback LLP, Single Row

PACKAGE PLACEMENT

LLP packages can be placed using standard pick and place equipment with an accuracy of ±0.05 mm. Component pick and place systems are composed of a vision system that recognizes and positions the component and a mechanical system which physically performs the pick and place operation. Two commonly used types of vision systems are: (1) a vision system that locates a package silhouette and (2) a vision system that locates individual pads on the interconnect pattern. The latter type renders more accurate place but tends to be more expensive and time consuming. Both methods are acceptable since the parts align due to a self-centering feature of the LLP solder joint during solder reflow.

It is recommended to release the LLP package 1 to 2 mils into the solder paste or with minimum force to avoid causing any possible damage to the thinner packages.

SOLDER PASTE

Type 3, water soluble, no clean, and leadfree solder pastes are acceptable.

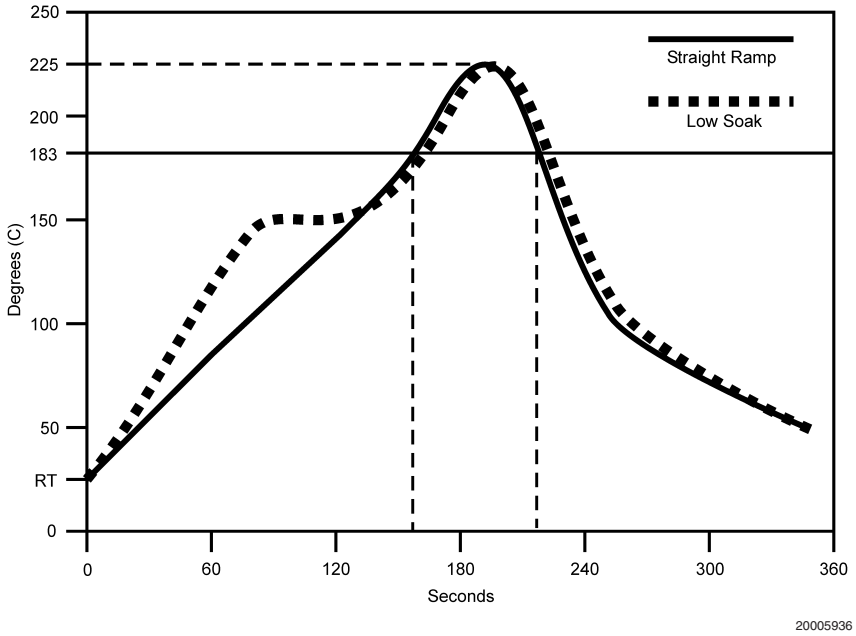
REFLOW AND CLEANING

The LLP may be assembled using standard IR / IR convection SMT reflow processes without any special considerations. As with other packages, the thermal profile for specific board locations must be determined. Nitrogen purge is recommended during solder for no-clean fluxes. The LLP is qualified for up to three reflow cycles at 235˚C peak (J-STD-020). The actual temperature of the LLP is a function of:

- Component density
- Component location on the board
- Size of surrounding components

It is recommended that the temperature profile be checked at various locations on the board. *Figure 21* and *Figure 22* illustrate typical reflow profiles.

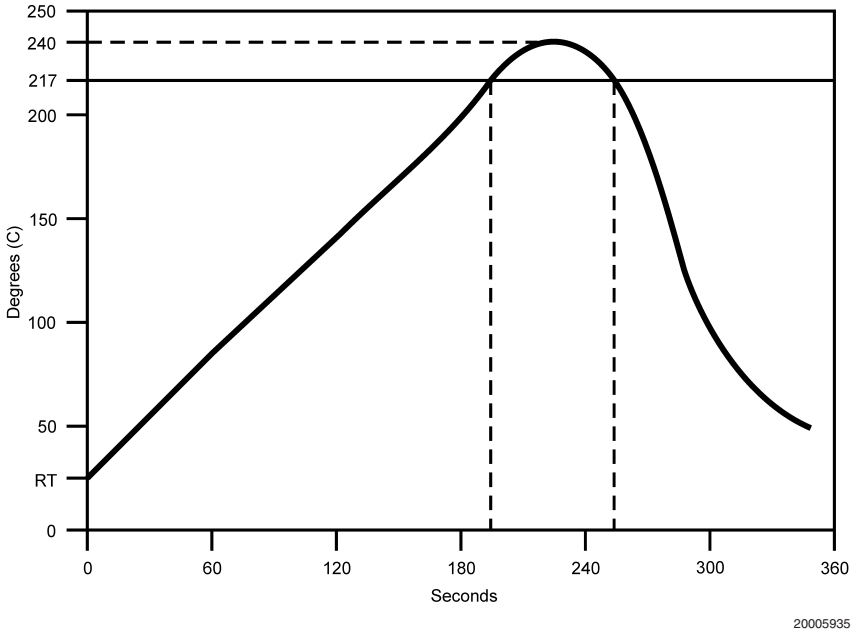
SMT Assembly Recommendations (Continued)



Profile Elements	Straight Line Profile	Low Soak Profile
Ramp rate	0.8 - 1.2 ˚C/s (RT to Peak temp)	1.5 - 2.0 ˚C/s (RT to 145 ˚C)
Dwell @ 145 to 160 ˚C	N/A	30 - 120 s
2nd Ramp rate	N/A	1.5 - 2.0 ˚C/s (to Peak Temp)
Time above liquidus (183 ˚C)	45 - 75 seconds	
Peak temperature range	210 - 225 ˚C typical, (240 ˚C max)	
Ramp-down rate to RT	1 - 3 ˚C/s typical, (6 ˚C/s max)	
Note: For details, please refer to solder paste manufacturer's recommendation.		

FIGURE 21. Typical Reflow Profile for Eutectic (63Sn/37Pb) Solder Paste

SMT Assembly Recommendations (Continued)



Profile Elements	Convection or IR
Ramp rate (RT to Peak temp)	0.8 - 1.2 ˚C/s
Time above liquidus (217 ˚C)	35 - 80 seconds
Peak temperature range	235 - 240 ˚C typical, (260 ˚C max)
Ramp-down rate to RT	1 - 2˚C/s typical, (6 ˚C/s max)
Note: For details, please refer to solder paste manufacturer's recommendation.	

FIGURE 22. Typical Reflow Profile for Lead-Free (SAC305 or SAC405) Solder Paste

SOLDER JOINT INSPECTION

After surface mount assembly, transmission X-ray should be used for **sample** monitoring of the solder attachment process. This identifies defects such as solder bridging, shorts, opens and voids. **NOTE:** voids typically do not have an impact on reliability. *Figure 23* shows a typical X-ray photograph after assembly.

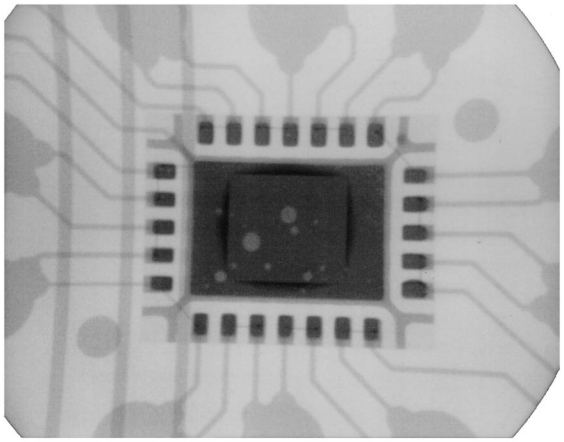


FIGURE 23. Typical X-ray after process

In the process setup, it is recommended to use side view inspection in addition to X-ray to determine if there are 'Hour Glass' shaped solder existing. The 'Hour Glass' solder shape is not a reliable joint. 90˚ mirror projection can be used for side view inspection. See *Figure 24*.

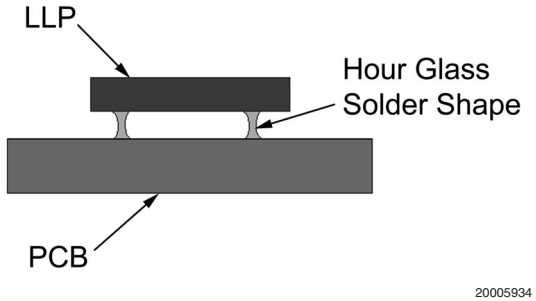


FIGURE 24. Pullback LLP Hour Glass Solder Joint

REPLACEMENT/ REWORK

The quality of the rework is controlled by:

- Directing the thermal energy through the component body to solder without over-heating the adjacent components.

SMT Assembly Recommendations

(Continued)

- Heating should occur in an encapsulated, inert, gas-purged environment where the temperature gradients do not exceed ±5˚C across the heating zone.
- Using a convective bottom side pre-heater to maximize temperature uniformity.
- Interchangeable nozzles designed with different geometries will accommodate different applications to direct the airflow path

NOTE: Standard SMT rework systems are capable of these elements.

Removal of the LLP Removing the LLP from the PCB involves heating the solder joints above the liquidus temperature of eutectic (63Sn-37Pb) solder using a vacuum gas nozzle. Baking the PCB at 125˚C for 4 hours is recommended PRIOR to any rework. Doing this removes any residual moisture from the system, preventing moisture induced cracking or PCB delamination during the demount process.

A 1.27 mm (50 mil) keep-out zone for adjacent components is recommended for standard rework processing. If the adjacent components are closer than 1.27 mm, custom tools are required for the removal and rework of the package.

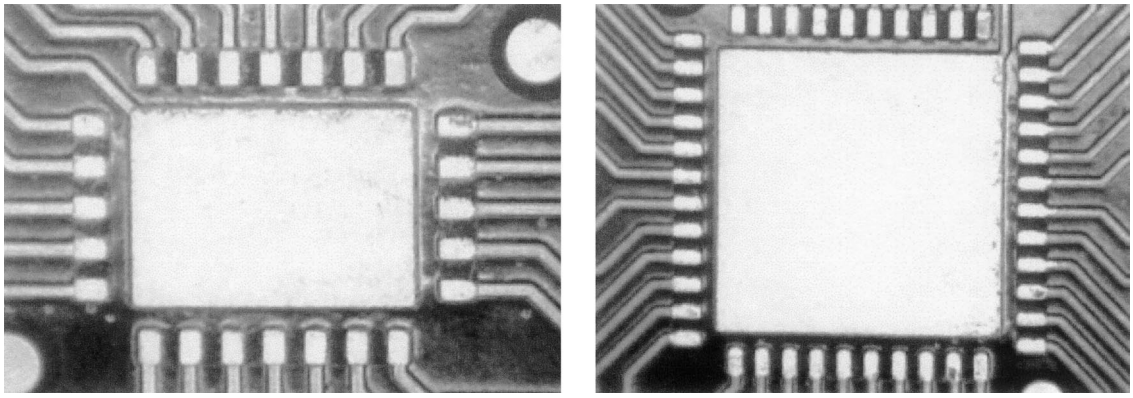
It is recommended that the reflow profile used to reflow the LLP be as close to the PCB mount profile as possible. Preheat the PCB area, through the bottom side of the board, to 100˚C before heating the LLP to ensure a controlled process. Once the liquidus temperature is reached, nozzle vacuum is automatically activated and the component is removed. After removing the package, the pads may be heated with the nozzle to reflow any residual solder, which may be removed using a Teflon tipped vacuum wand.

Site Preparation Once the LLP is removed, the site must be cleaned in preparation for package attachment. The best results are achieved with a low-temperature, blade-style conductive tool matching the footprint area of the LLP in conjunction with a de-soldering braid. No-clean flux is needed throughout the entire rework process. Care must be taken to avoid burn, lift-off, or damage of the PCB attachment area. See *Figure 25*.

Solder Paste Deposition Because the LLP is a land area type package, solder paste is required to insure proper solder joint formation after rework. A 127 µm (5 mil) thick mini-stencil is recommended to deposit the solder paste patterns prior to replacement of the LLP. See *Figure 26*.

Component Placement Most CSP rework stations will have a pick and place feature for accurate placement and alignment. Manual pick and place, with only eye-ball alignment, is not recommended. It is difficult or impossible to achieve consistent placement accuracy. Improper handling or excessive pressure may cause damage to some of the thinner packages.

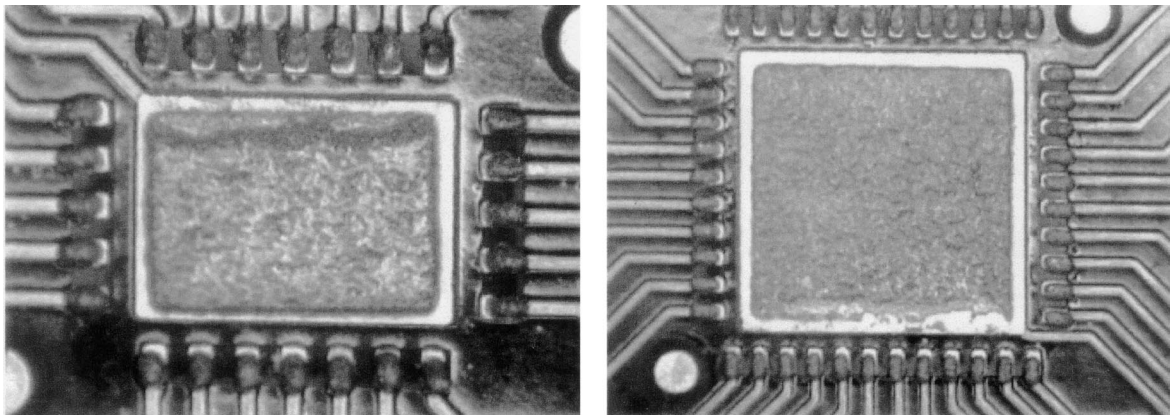
Component Reflow It is recommended that the reflow profile used to reflow the LLP be as close to the PCB mount profile as possible. Preheat the PCB area, through the bottom side of the board, to 100˚C before heating the LLP to ensure a controlled process. Once the liquidus temperature is reached, the solder will reflow and the LLP will self align. *Figure 27* shows a cross section of a solder joint after rework.



20005917

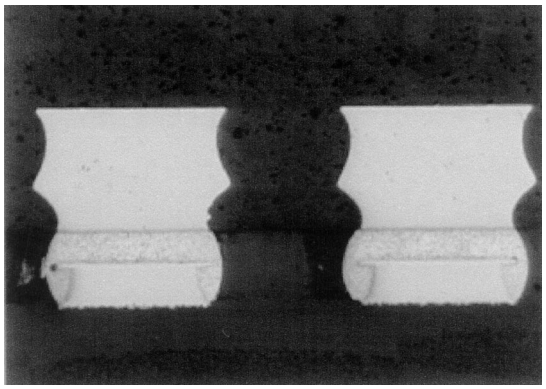
FIGURE 25. Pads After Removing Components and Cleaning

SMT Assembly Recommendations (Continued)



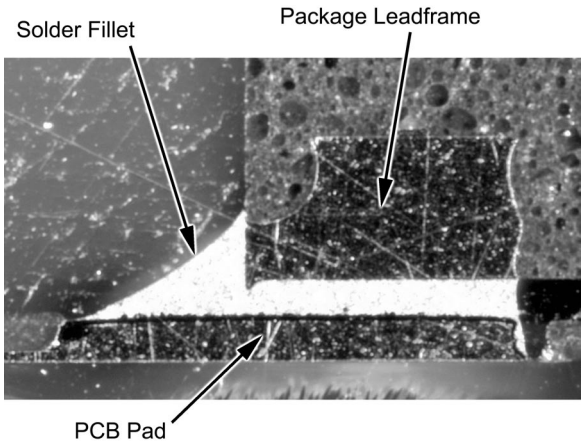
20005918

FIGURE 26. Solder Paste Printing of LLP 24 and LLP 44 Using 127 µm (5 mil) Thick Stencil



20005920

FIGURE 27. X-section Across Solder Joints



20005939

FIGURE 28. X-section of NO Pull Back Solder Joint with fillet

Appendices

APPENDIX 1: BOARD LEVEL RELIABILITY TEST DATA

Temperature Cycle Test

Test Conditions:

- Temperature Range: –40 to 125˚C
- Cycle Duration: 1 hour (15 minute ramp/15 minute Dwell)
- Test Board Dimension: 142.5 mm x 142.5 mm x 1.6 mm
- Test Board Finish: Ni-Au 0.05 µm to 0.2 µm gold thickness
- Dummy die in package
- Package is bonded with a Daisy Chain Circuit

Failure Determination: Change of 10% in Net Resistance

Results:

I/O Count	Body Size (mm)	Exposed DAP Size (mm)	Die Size (mm)	Solder	DAP Soldered	PCB Surface Finish	15/15/15/15		
							Failure Cycle		Beta
							First	63.2%	
LLP - Pullback, Dual Row (Note 2)									
80	8 x 8 x 0.8	3.9 x 3.9	3.8 x 3.8	63Sn/37Pb	Yes	NiAu	1469	2350	8.6
LLP - Pull Back, Single Row (Note 2)									
56	9 x 9 x 0.8	4.8 x 4.8	4.06 x 3.76	63Sn/37Pb	Yes	NiAu	Pass 1050x No Failure		
					No	NiAu	Pass 1050x No Failure		
					Yes	OSP	Pass 1050x No Failure		
				SAC305	No	NiAu	Pass 1050x No Failure		
					No	OSP	Pass 1050x No Failure		
44	7 x 7 x 0.8	4.3 x 4.3	3.25 x 3.08	63Sn/37Pb	Yes	NiAu	Pass 1050x No Failure		
					No	NiAu	Pass 1050x No Failure		
24	5 x 4 x 0.8	3.4 x 2.4	2.31 x 1.4	63Sn/37Pb	Yes	NiAu	Pass 1050x No Failure		
					No	NiAu	Pass 1050x No Failure		
LLP - Pull Back (Single Row) (Note 3)									
14	6 x 5 x 1.0	4.35 x 3.0	4.29 x 2.95	63Sn/37Pb	Yes	NiAu	Pass 1050x No Failure		
SOT-23 Footprint Compatible LLP (Note 4)									
6	2.92 x 3.29 x 0.8	1.92 x 1.2	1.45 x 1.14	63Sn/37Pb	Yes	NiAu	Pass 1050x No Failure		
					No	NiAu	Pass 1050x No Failure		
LLP - No Pull Back (Note 5)									
40	5 x 5 x 0.8	3.6 x 3.6	3.6 x 3.6	SAC305	Yes	NiAu	Pass 1050x No Failure		
48	6 x 6 x 0.8	4.6 x 4.6	4.3 x 4.3	SAC305	Yes	NiAu	950	2730	3.3
48	7 x 7 x 0.8	5.1 x 5.1	2.4 x 2.2	63Sn/37Pb	Yes	NiAu	Pass 1500x No Failure		
48	7 x 7 x 0.6	5.1 x 5.1	4.7 x 4.7	63Sn/37Pb	No	NiAu	Pass 1500x No Failure		
48	7 X 7 X 0.4	5.2 X 5.2	4.4 X 4.1	SAC305	No	NiAu	Pass 1600x No Failure		
48	7 X 7 X 0.4	5.2 X 5.2	2.2 X 2.2	SAC305	No	NiAu	Pass 1600x No Failure		
LLP - No Pull Back (Note 6)									
48	7 X 7 X 0.4	5.2 X 5.2	4.4 X 4.1	SAC305	No	NiAu	Pass 1700x No Failure		

Note 2: Terminal Pad Size: 0.5 x 0.25 mm; Terminal Pad Pitch: 0.5 mm; Die Thickness: 0.216 mm; PCB Thickness: 1.6 mm

Note 3: Terminal Pad Size: 0.5 x 0.4 mm, Terminal Pad Pitch: 0.8 mm, Die Thickness: 0.216 mm, PCB Thickness: 1.6 mm

Note 4: Terminal Pad Size: 0.5 x 0.35 mm, Terminal Pad Pitch: 0.96 mm, Die Thickness: 0.216 mm, PCB Thickness: 1.6 mm

Note 5: Terminal Pad Size: 0.4 x 0.25 mm, Terminal Pad Pitch: 0.5 mm, Die Thickness: 0.216 mm, PCB Thickness: 1.6 mm

Note 6: Cycle Duration: 34 minutes (3 minute ramp/14 minute Dwell)

Appendices (Continued)

Board Drop Test

Test Conditions:

- Test Board Dimension: 142.5 mm x 142.5 mm x 1.6 mm
- Printed Circuit Board Finish: Ni-Au0.05 µm - 0.2 µm gold thickness
- Dummy die in package
- Package is bonded with a Daisy Chain Circuit
- Cumulative Dead weight of the board: 150 Grams
- Drop Height: 1.5 meters
- Drop Surface: Non cushioning vinyl tile
- Number of Drops: 30 total
  - — 7 drops: along the length of the PCB
  - — 7 drops: along the width of the PCB
  - — 8 Drops: Along the diagonal of the board
  - — 8 Drops: With the components on the top of the board

Failure Determination: Change of 10% in Net Resistance

Results:

Package Type	Drop Test Results
24L 4 mm x 5 mm LLP (DAP soldered to PCB)	0/20
24L 4 mm x 5 mm LLP (DAP NOT soldered to PCB)	0/20
44L 7 mm x 7 mm LLP (DAP soldered to PCB)	0/20
44L 7 mm x 7 mm LLP (DAP NOT soldered to PCB)	0/20
56L 9 mm x 9 mm LLP (DAP soldered, Power/Ground Rings soldered to PCB)	0/25
14L Power LLP	0/32
48L 7 mm x 7 mm LLP (Note 7) (DAP solder to PCB)	0/15

Note 7: Per Jedec Standard JESD22-B111, 1500G's 0.5mS Half-Sine Pulse

Vibration Test

Test Conditions:

- Test Board Dimension: 142.5 mm x 142.5 mm x 1.6 mm
- Printed Circuit Board Finish: Ni-Au 0.05 µm - 0.2 µm gold thickness
- Dummy die in package
- Package is bonded with a Daisy Chain Circuit
- Die attach pad soldered to PCB
- Vibration test conditions:
  - — Sinusoidal excitation performed for 1 hour at 20G force followed by 3 hours at 40G force
  - — Random Vibration with variable frequencies ranging from 20Hz to 2,000Hz for 3 hours with a force of 2G RMS

Results: DAP Soldered to PCB

Package Type	Test Results
24L 4 mm x 5 mm LLP	0/24
44L 7 mm x 7 mm LLP	0/20
56L 9 mm x 9 mm LLP	0/25
14L 6 mm x 5 mm Power LLP	0/32

Flex Test

Test Conditions:

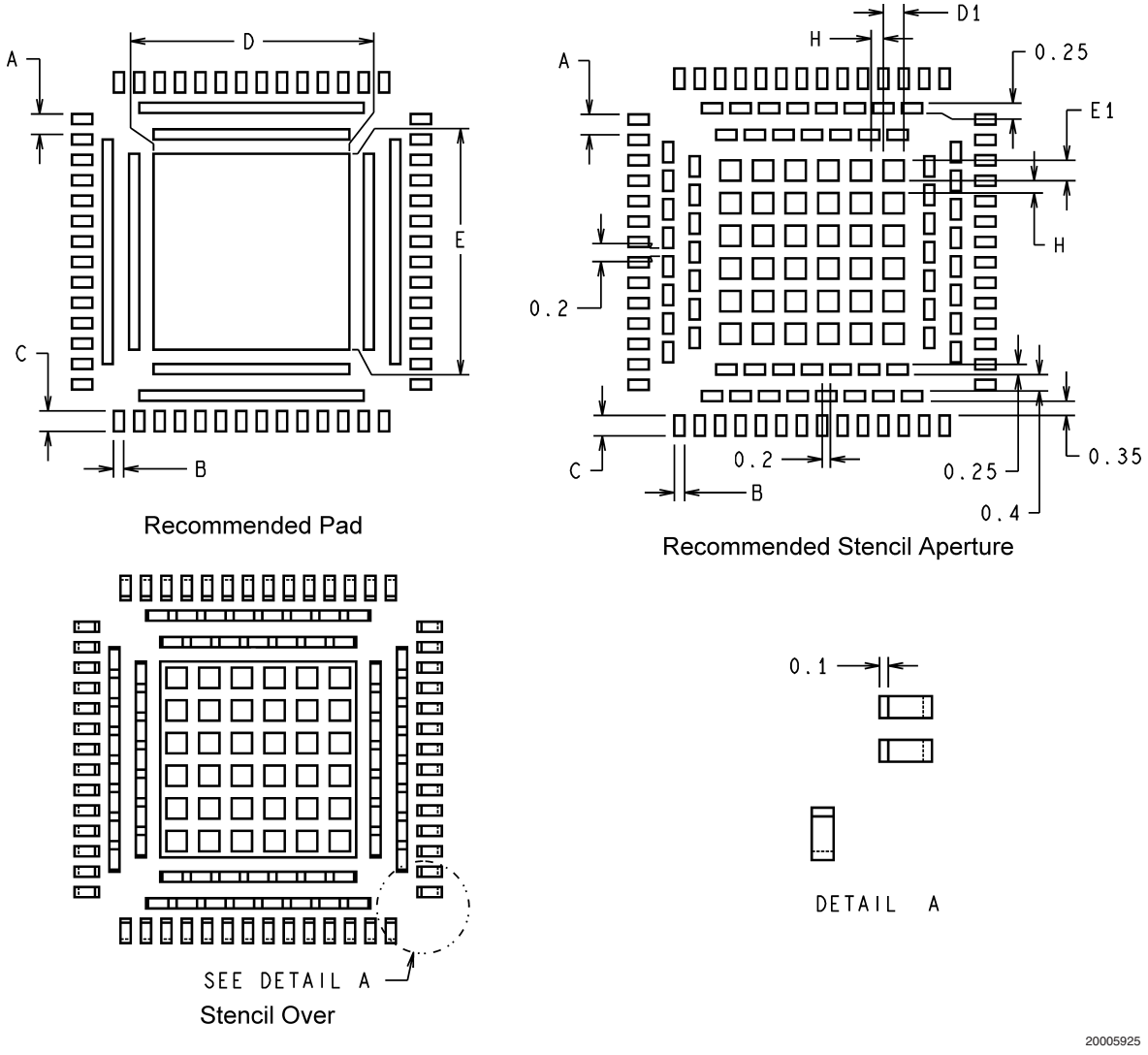
- PCB Size: 75 x 55 x 0.8 mm
- Surface Finish: Electroless Nickel Immersion Gold
- Solder Paste: 63Sn/37Pb
- Stencil: 0.127 mm
- Deflection: 1 mm with 12.7 mm puncher offset
- Span: 50 mm
- Frequency: 1Hz

Results:

Package	16L	60L
Body Size (mm)	4 x 4	9 x 9
1mm @ 1Hz with 0.5 inch offset	+5000 cycles	+5000 cycles

Appendices (Continued)

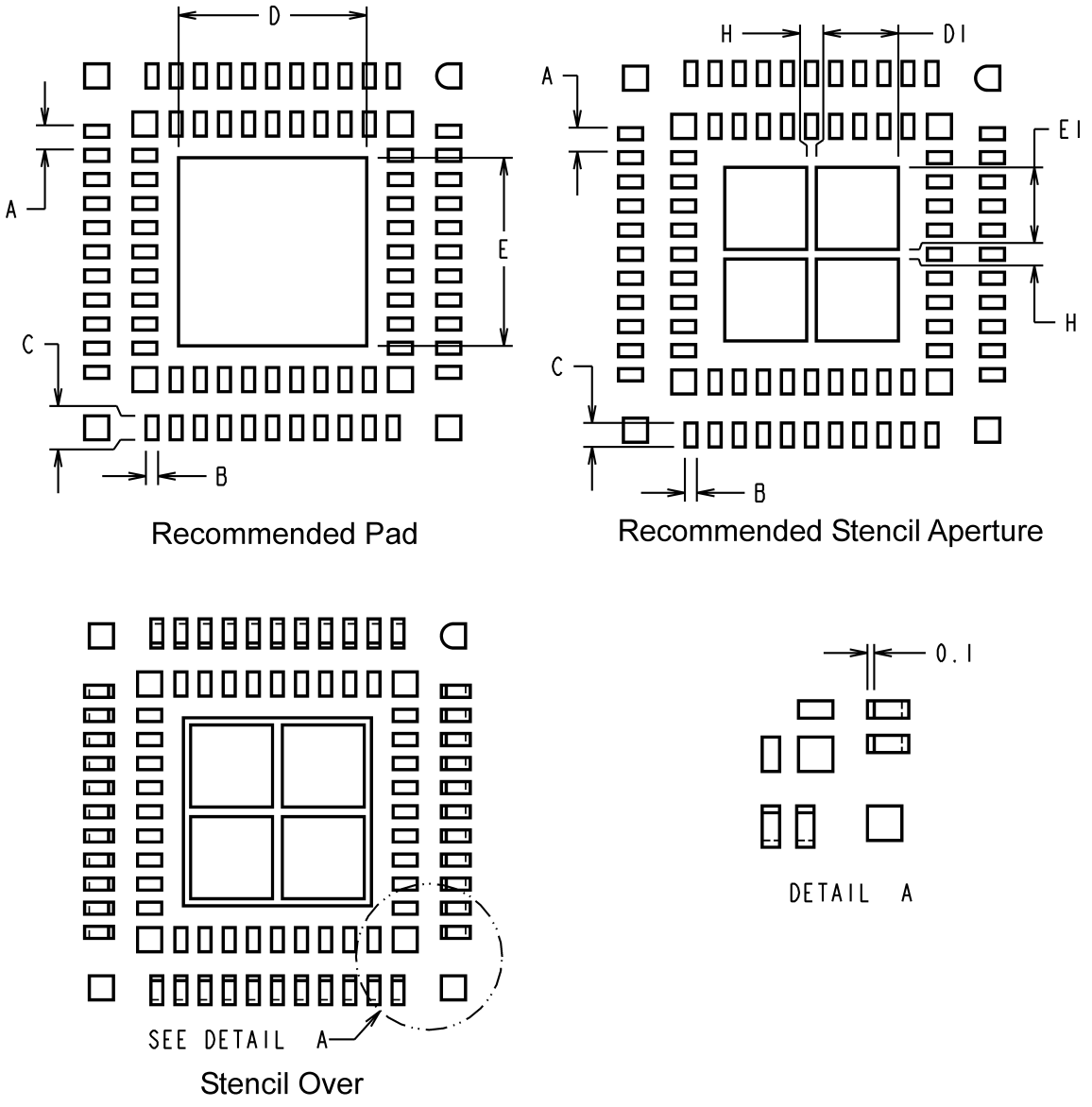
APPENDIX 2: CUSTOM PACKAGE DESIGN



Number of pins	56
A - LLP, PCB, Stencil Terminal Pitch (mm)	0.5
B - LLP, PCB, Stencil Terminal Width (mm)	0.25
C - LLP, PCB, Stencil Terminal Length (mm)	0.5
D - LLP, PCB Exposed DAP Width (mm)	4.8
D1 - Exposed DAP Aperture Width (mm)	0.5
H - Aperture split width, centered (mm)	0.3
E - LLP, PCB Exposed DAP Length (mm)	4.8
E1 - Exposed DAP Aperture Length (mm)	0.5
F - Stencil Aperture opening offset (mm)	0.1

FIGURE 29. Typical Recommended Stencil Openings for 56 Pin LLP with Exposed DAP, Ground and Power Bars.

Appendices (Continued)



Number of pins	80
A - LLP, PCB, Stencil Terminal Pitch (mm)	0.5
B - LLP, PCB, Stencil Terminal Width (mm)	0.25
C - LLP, PCB Terminal Length (mm)	0.5
D - LLP, PCB Exposed DAP Width (mm)	3.9
D1 - Exposed DAP Aperture Width (mm)	1.7
H - Aperture split width, centered (mm)	0.2
E - LLP, PCB Exposed DAP Length (mm)	3.9
E1 - Exposed DAP Aperture Length (mm)	1.7
F - Stencil Aperture opening offset (mm)	0.1

FIGURE 30. Typical Recommended Stencil Openings for 80 Pin LLP with Exposed DAP and Dual Row.

Appendices (Continued)

STENCIL OPENINGS FOR SOT23 5/6L FOOTPRINT COMPATIBLE LLP

- For the SOT23 5/6L footprint compatible LLP for which the PCB has been designed for the SOT23 package, refer to *Figure 31* for solder stencil openings.

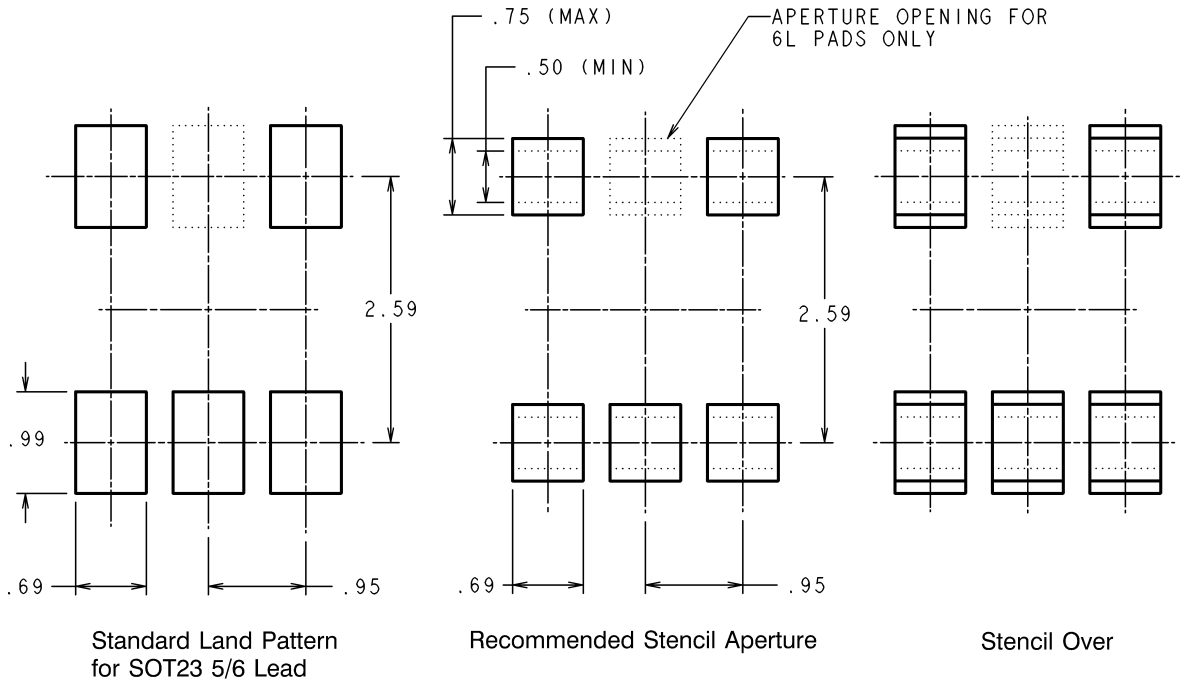


FIGURE 31. Recommended Stencil Apertures for SOT23 5/6 Lead Footprint Compatible LLP

Revision Log

Revision Date	Description
September 2005	Added 0.4 mm LLP information.
February 2006	Updated the section on Thermal Design Considerations

Notes

National Semiconductor
Americas Customer
Support Center
Email: new.feedback@nsc.com
Tel: 1-800-272-9959

www.national.com

National Semiconductor
Europe Customer Support Center
Fax: +49 (0) 180-530 85 86
Email: europe.support@nsc.com
Deutsch Tel: +49 (0) 69 9508 6208
English Tel: +44 (0) 870 24 0 2171
Français Tel: +33 (0) 1 41 91 8790

National Semiconductor
Asia Pacific Customer
Support Center
Email: ap.support@nsc.com

National Semiconductor
Japan Customer Support Center
Fax: 81-3-5639-7507
Email: jpn.feedback@nsc.com
Tel: 81-3-5639-7560