Leadless Leadframe Package (LLP)

National Semiconductor
Application Note 1187
September 2003

Table of Contents

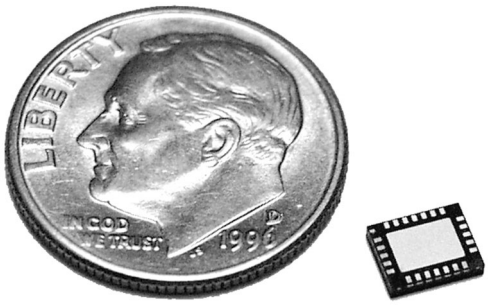


FIGURE 1. 24 Pin LLP

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 has the following advantages:

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

Package Overview

KEY ATTRIBUTES

- Construction of the LLP is Illustrated in *Table 1*, *Figure 2*, and *Figure 3*.
- Terminal contacts:
  - — The contact pads (or solder pad) are located peripherally in single row, dual rows or in array 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 7*.
  - — All LLP contacts are plated with 85Sn/15Pb solder for ease of surface mount processing.
  - — All Lead-Free LLP contacts are plated with matt tin solder for ease of surface mount processing.
- Printed Circuit Board (PCB) footprint:
  - — National recommends a one-to-one correlation between the PCB land patterns and the package 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.
- 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).
  - — All LLP packages are MSL 1 without the downbond. Specific package MSL can be confirmed via product application sheets.
  - — MSL of specific applications, requiring large packages, may vary depending on die size, exposed DAP design, and number of downbonds.

TABLE 1. Elements of the 24, 44 and 56 pin LLP

	24 Pin	44 Pin	56 Pin
Package Dimensions	5 x 4 x 0.8 mm	7 x 7 x 0.8 mm	9 x 9 x 0.8 mm
PCB Footprint Area (mm^2)	20	49	81
Standard	JEDEC	JEDEC	JEDEC
Pitch	0.5 mm	0.5 mm	0.5 mm
Weight	0.047 grams	0.104 grams	0.208 grams
Lead Frame	Copper	Copper	Copper
Lead Finish	Sn/Pb	Sn/Pb	Sn/Pb
Typical Thermal Resistance θ_{JA} (Note 1)	33˚C/W	20˚C/W	27˚C/W (Note 2)

Note 1: The typical data reported are measured values at still air and 1 watt input power using four layer FR4 substrate with Vias and copper thickness of 2.0/1.0/1.0/2.0 oz.

Note 2: Package option with limited exposed pad size due to incorporations of ground and power rings.

Package Overview (Continued)

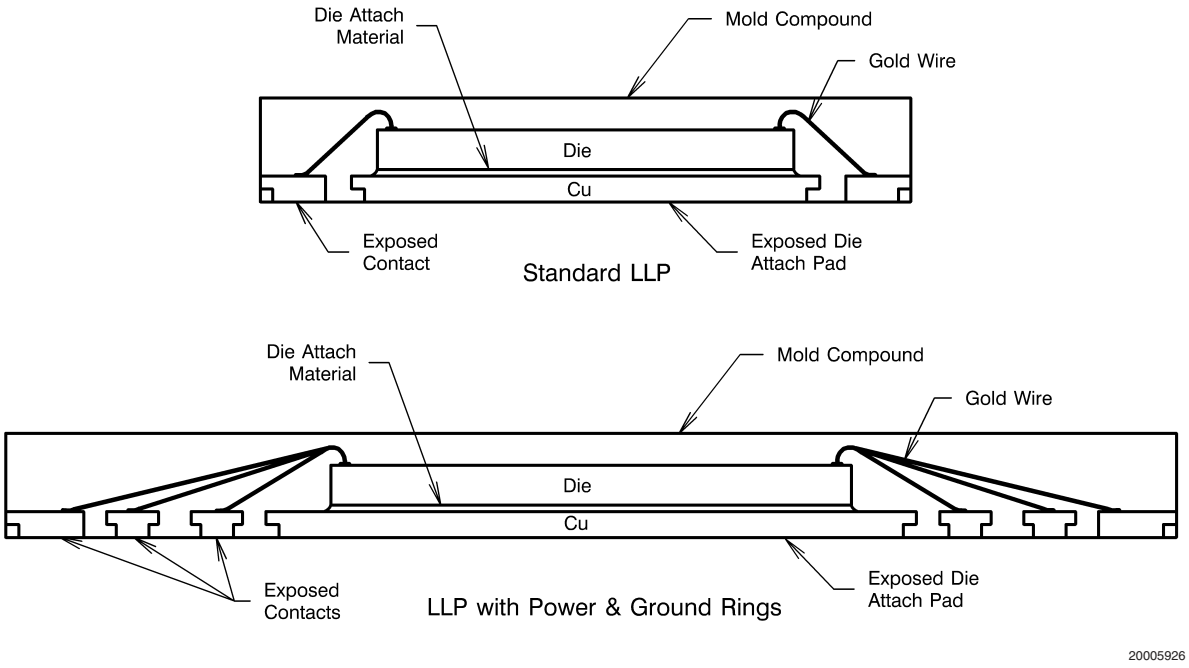


FIGURE 2. Construction of LLP

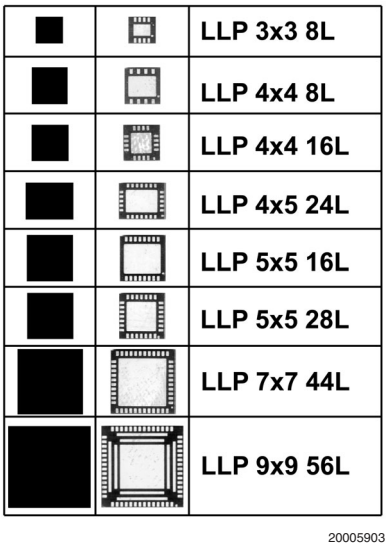


FIGURE 3. Examples

Package Overview (Continued)

PACKAGE OFFERING

Pin Count	Body size (mm)	Pitch (mm)	Max. Die Size (mm) (Note 5)	MKT DWG	MBS AD#	L/F Dwg #	QUAD/DIP (Note 4)	θ_{JA} (˚C/W) (Note 3)
6	2.2 x 2.5	0.65	1.04 x 1.27	ldb06a	ldb006AA	36-1058-01	DIP	NA
6	2.92 x 3.29	0.65	1.45 x 1.85	lde06a	lde006AA	36-1056-01	DIP	NA
6	3.0 x 4.0	0.8	1.85 x 2.15	ldc06d	ldc006AE	36-0989-01	DIP	45.6
8	2.5 x 2.5	0.5	1.45 x 0.65	lda08b	lda008AC	36-1006-01	DIP	63.9
8	2.5 x 3.0	0.5	1.45 x 1.15	lda08c	lda008AD	36-1007-01	DIP	58.2
8	3.0 x 3.0	0.5	1.85 x 1.15	lda08a	lda008AA	36-0977-01	DIP	55.3
8	4.0 x 4.0	0.8	2.69 x 1.88	ldc08a	ldc008AA	36-0990-01	DIP	38.8
10	3.0 x 3.0	0.5	1.85 x 1.15	lda10a	lda010AA	36-1000-01	DIP	54.1
14	4.0 x 5.0	0.5	2.69 x 2.90	lda14b	lda014AB	36-1010-01	DIP	34.6
14	5.0 x 6.0	0.8	2.69 x 4.00	ldc14a	ldc014AA	36-1043-01	DIP	NA
16	4.0 x 4.0	0.5	2.13 x 2.13	lqa16a	lqa016AB	36-0978-02	QUAD	39.8
20	4.0 x 4.0	0.5	2.13 x 2.13	lqa20a	lqa020AB	36-1020-01	QUAD	38.7
24	5.0 x 4.0	0.5	2.10 x 3.10	lqa24a	lqa024AA	36-0973-01	QUAD	34.7
24	6.0 x 6.0	0.8	3.89 x 3.89	lqc24a	lqc024AA	36-1033-01	QUAD	NA
28	5.0 x 5.0	0.5	2.13 x 2.13	lqa28a	lqa028AA	36-0993-01	QUAD	30.8
32	5.0 x 6.0	0.5	2.13 x 2.13	lqa32b	lqa032AB	36-1031-01	QUAD	28.2
32	6.0 x 6.0	0.5	3.89 x 3.89	lqa32a	lqa032AA	36-1034-01	QUAD	26.4
44	7.0 x 7.0	0.5	4.00 x 4.00	lqa44a	lqa044AC	36-0976-03	QUAD	24.2
56	9.0 x 9.0	0.5	4.50 x 4.50	lqa56a	lqa056AA	36-1045-01	QUAD	NA

Note 3: 4-layer board with Cu finished thickness 1.5/1/1/1.5 oz. Maximum die size used. 5x body length of Cu trace on PCB top. 50 x 50 mm ground and power planes embedded in PCB.

Note 4: DIP : Only two sides of the package have leads. QUAD: All four sides of the package have leads.

Note 5: Maximum die size without downbond.

PACKAGE HANDLING

The LLP is shipped in standard polycarbonate conductive carrier tape with pressure sensitive adhesive (PSA) cover tape. The LLP is shipped in 7"" reels. Samples can be shipped in carrier tape format and/or trays.

Pin Count	Body Size (mm)	Marketing Drawing	Reel S/N	Tape Carrier S/N	Tape Cover S/N	Tray S/N
6	2.2 x 2.5	ldb06a	017983	078156	025360	NA
6	2.92 x 3.29	lde06a	017983	076263	025360	NA
6	3.0 x 4.0	ldc06d	017983	075537	025360	NA
8	2.5 x 2.5	lda08b	017983	075535	025360	NA
8	2.5 x 3.0	lda08c	017983	075536	025360	NA
8	3.0 x 3.0	lda08a	017983	073104	025360	075393
8	4.0 x 4.0	ldc08a	017983	073105	025360	075391
10	3.0 x 3.0	lda10a	017983	073104	025360	075393
14	4.0 x 5.0	lda14b	070376	075538	025360	NA
14	5.0 x 6.0	ldc14a	017982	075539	025361	NA
16	4.0 x 4.0	lqa16a	017983	073105	025360	075391
20	4.0 x 4.0	lqa20a	017983	073105	025360	075391
24	5.0 x 4.0	lqa24a	017983	073106	025360	NA
24	6.0 x 6.0	lqc24a	017982	075540	025361	NA
28	5.0 x 5.0	lqa28a	070376	073896	025360	075392
32	5.0 x 6.0	lqa32b	017982	075539	025361	NA
32	6.0 x 6.0	lqa32a	017982	075540	025361	NA

Package Overview (Continued)

Pin Count	Body Size (mm)	Marketing Drawing	Reel S/N	Tape Carrier S/N	Tape Cover S/N	Tray S/N
44	7.0 x 7.0	lqa44a	017982	073456 073107(alt) 077373(alt)	030137 025361(alt)	073329
56	9.0 x 9.0	lqa56a	023815	076519	025361	073320

JEDEC REGISTRATIONS

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

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.

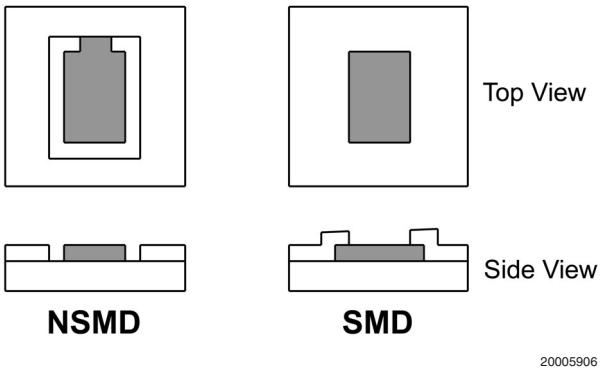


FIGURE 4. NSMD and SMD Pad Geometry

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 and solder mask this avoids overlap between the solder joint and solder mask and account 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 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 such that the requirement of 1:1 ratio of package pad to PCB pad is not violated. See *Figure 6* for PCB pad recommendations.

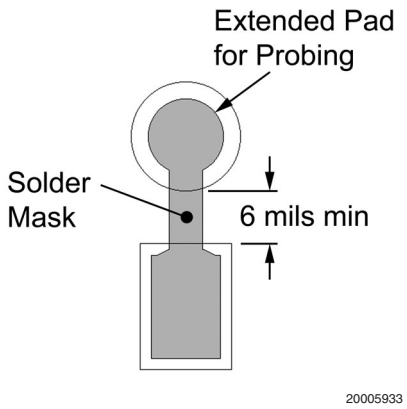
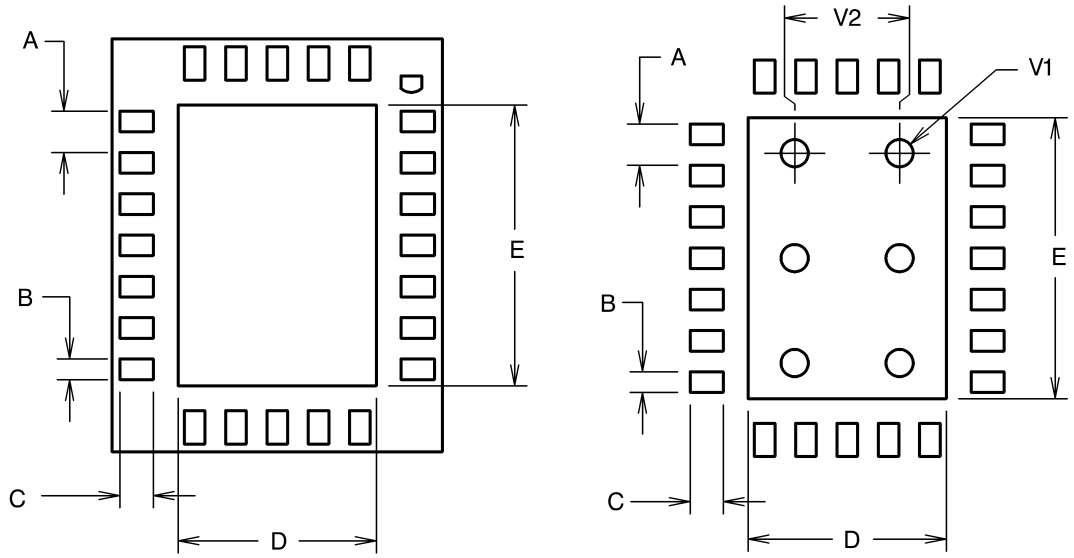


FIGURE 5. Recommended Pad Design for Probing

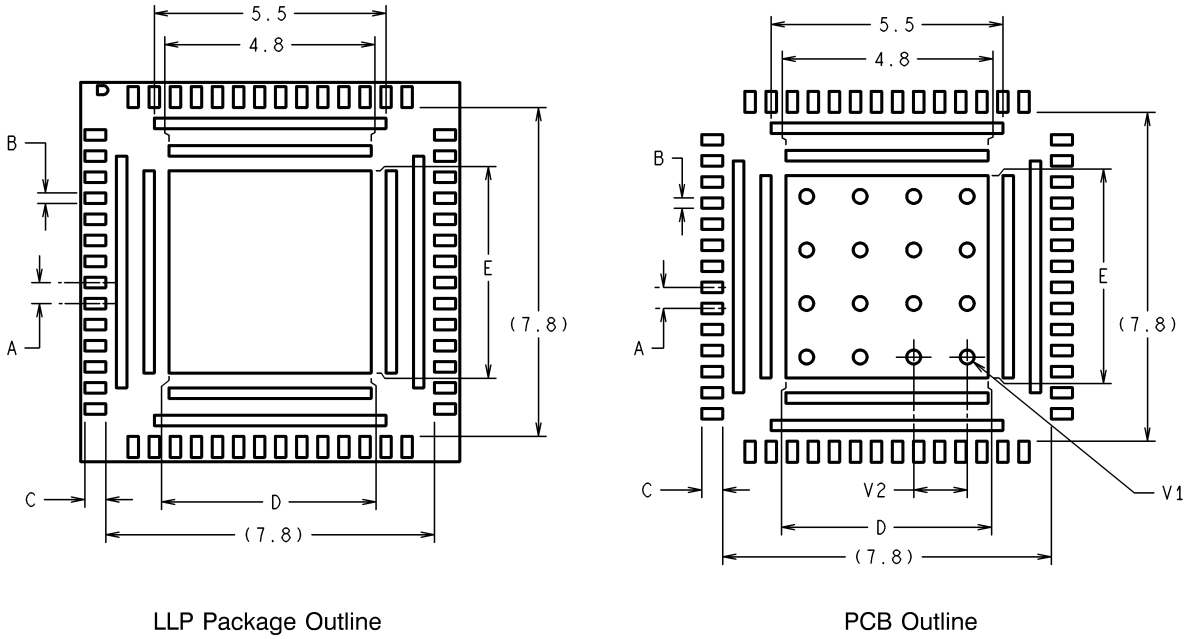
PCB Design Recommendations (Continued)



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.
A - LLP Terminal Pitch
B - LLP Terminal Width
C - LLP Terminal Length
D - Exposed DAP Width
E - Exposed DAP Length
V1 - Thermal Via Diameter. Recommended 0.2 - 0.33 mm
V2 - Thermal Via Pitch. Recommended 1.27 mm

FIGURE 6. Typical Recommended Printed Circuit Board Dimensions

PCB Design Recommendations (Continued)



20005927

Number of pins	56
Package Size (mm)	9 x 9
A - LLP Terminal Pitch (mm)	0.5
B - LLP Terminal Width (mm)	0.25
C - LLP Terminal Length (mm)	0.5
D - Exposed DAP Width (mm)	4.8
E - Exposed DAP Length (mm)	4.8
V1 - Thermal Via Diameter (mm)	0.2 - 0.33
V2 - Thermal Via Pitch (mm)	1.27

FIGURE 7. Recommended Printed Circuit Board Dimensions for LLP 56 L with Ground and Power Bars.

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).

PCB Design Recommendations

(Continued)

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 8*).

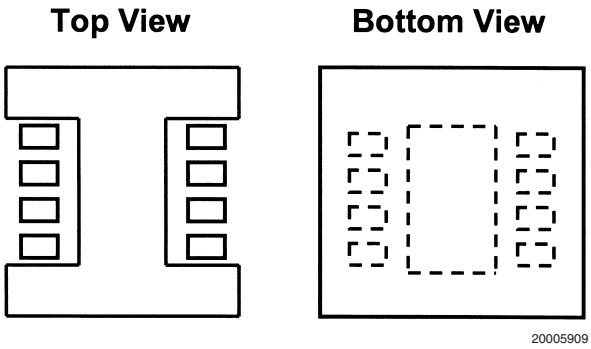


FIGURE 8. Dog Bone

THERMAL VIAS

Thermal vias are necessary. They conduct heat from the surface of the PCB to the ground plane. The number of vias is application specific and is dependent upon electrical requirements and power dissipation. A package thermal performance may be improved by increasing the number of vias. The improvement diminishes, however, as the number of vias increase. See *Figure 9*.

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. If the copper plating does not plug the via, 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 accross the entire PCB.

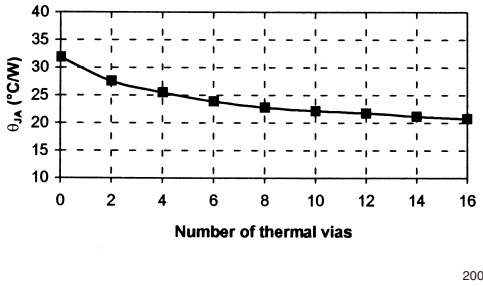


FIGURE 9. θ_{JA} vs. Number of Thermal Vias for the 44L LLP

EFFECTS OF THERMAL VOIDS

A void in the solder paste or die attach (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 10*). **NOTE:** voids typically do not have an impact on reliability.

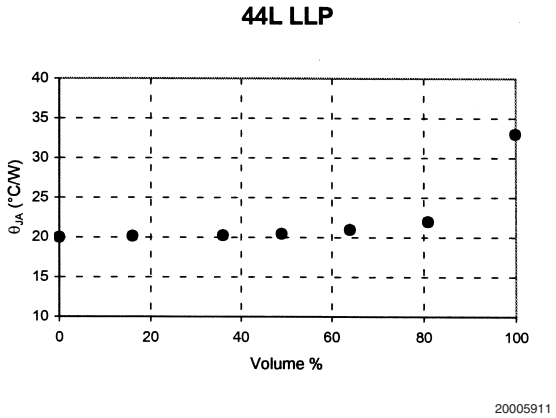


FIGURE 10. Thermal Voids Impact for the 44L LLP

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 achieving 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 to the thermal vias has significant effect on the thermal performance of the package. *Figure 11* 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.

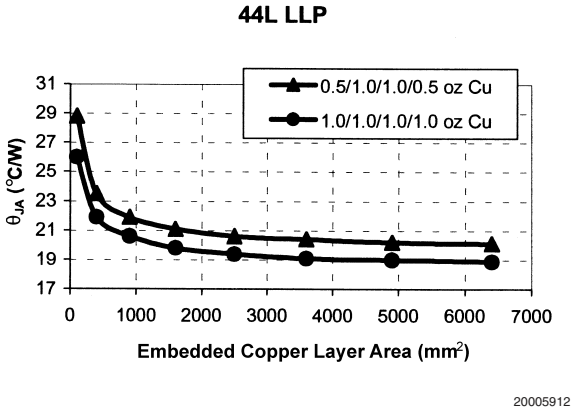


FIGURE 11. Effect of Thermal Layers on the 44L LLP's Junction-to-Ambient Thermal Resistance

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

SMT Assembly Recommendations

(Continued)

- X-ray pre reflow check - paste bridging
- Reflow and cleaning (dependent upon the flux type)
- X-ray post reflow check - solder bridging & Voids

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.

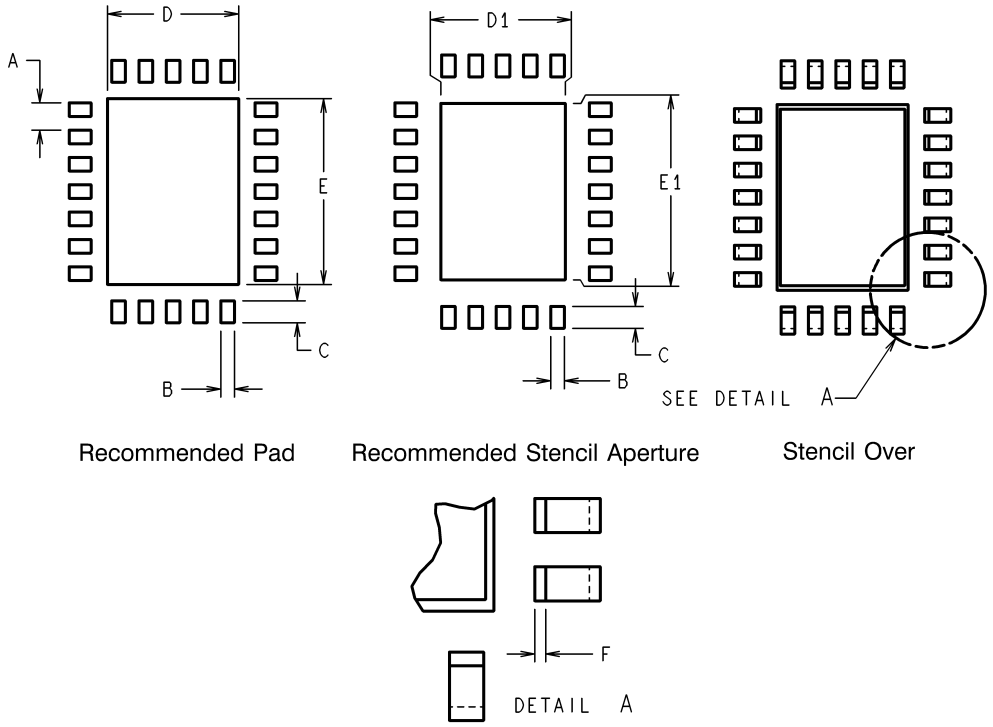
Stencils fabricated from chemical etching with elctro polished or laser cut is recommended. Tapered aperture walls (5˚ tapering) is recommended to facilitate paste release. Recommended stencil thickness is 127 µm. In order to prevent solder bridging the stencil aperture openings need to be modified as follows:

- The terminal contact aperture openings should be offset by 0.1 mm outward from the pads.
- For exposed pad aperture, up to 2 mm, the opening should be reduced to 95% of the corresponding PCB exposed DAP dimensions. See *Figure 12* and *Figure 13*.
- For exposed pad aperture with any side from 2 to 4 mm, the stencil opening should be split in two for any side. See *Figure 14*.
- For exposed pad aperature greater than 4 mm but without ground and power bars. See *Figure 15*.
- For exposed pad aperature greater than 4 mm with ground and power bars. See *Figure 16*.

TABLE 2. LLP Stencil Aperture Summary

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	Dimension (mm)		
							D1	E1	H
6	ldb06a	0.25 X 0.4	0.65	1.2 X 0.75	0.25 X 0.4	*Figure 13*	1.1	0.7	N/A
6	ldc06d	0.3 X 0.5	0.8	2 X 2.2	0.3 X 0.5	*Figure 14*	0.85	0.85	0.3
6	lde06a	0.35 X 0.5	0.95	1.92 X 1.2	0.35 X 0.5	*Figure 13*	1.8	1.1	N/A
8	lda08a	0.25 X 0.5	0.5	1.8 X 1.2	0.25 X 0.5	*Figure 13*	1.7	1.1	N/A
8	lda08b	0.25 X 0.5	0.5	1.5 X 0.7	0.25 X 0.5	*Figure 13*	1.4	0.6	N/A
8	lda08c	0.25 X 0.5	0.5	1.5 x 1.2	0.25 X 0.5	*Figure 13*	1.4	1.1	N/A
8	ldc08a	0.3 X 0.5	0.8	3 X 2.2	0.3 X 0.5	*Figure 14*	1.25	0.85	0.3
10	lda10a	0.25 X 0.5	0.5	2 X 1.2	0.25 X 0.5	*Figure 13*	1.9	1.1	N/A
14	lda14b	0.25 X 0.5	0.5	3 X 3.2	0.25 X 0.5	*Figure 14*	1.25	1.35	0.3
14	ldc14a	0.4 X 0.5	0.8	4.35 X 3	0.4 X 0.5	*Figure 14*	1.925	1.25	0.3
16	lqa16a	0.25 X 0.5	0.5	2.2 X 2.2	0.25 X 0.5	*Figure 14*	0.85	0.85	0.3
20	lqa20a	0.25 X 0.5	0.5	2.2 X 2.2	0.25 X 0.5	*Figure 14*	0.85	0.85	0.3
24	lqa24a	0.25 X 0.4	0.5	3.4 X 2.4	0.25 x 0.4	*Figure 14*	1.45	0.95	0.3
24	lqc24a	0.3 X 0.5	0.8	4.2 X 4.2	0.3 X 0.5	*Figure 15*	0.5	0.5	0.3
28	lqa28a	0.25 X 0.5	0.5	3.2 X 3.2	0.25 X 0.5	*Figure 14*	1.35	1.35	0.3
32	lqa32a	0.25 X 0.5	0.5	4.2 X 4.2	0.25 X 0.5	*Figure 15*	0.5	0.5	0.3
32	lqa32b	0.25 X 0.5	0.5	4.2 X 3.2	0.25 X 0.5	*Figure 15*	0.5	0.5	0.3
44	lqa44a	0.25 X 0.5	0.5	4.3 X 4.3	0.25 X 0.5	*Figure 15*	0.5	0.5	0.3
56	lqa56a	0.25 X 0.5	0.5	4.8 X 4.8	0.25 X 0.5	*Figure 16*	0.5	0.5	0.3

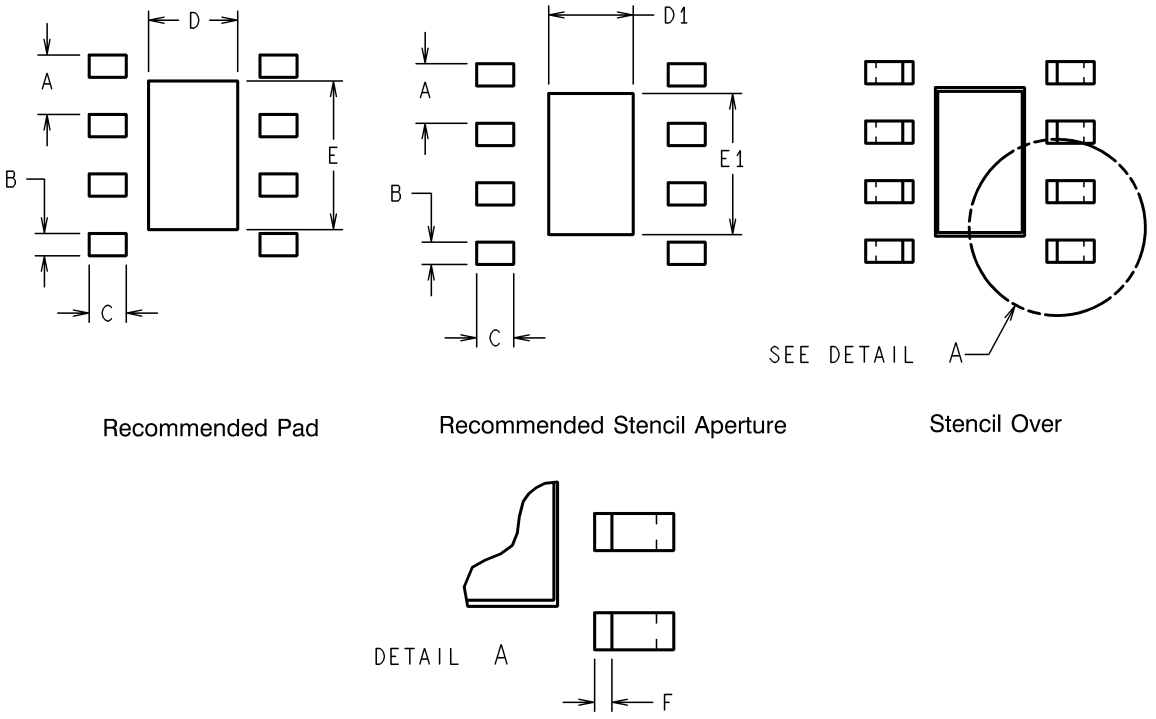
SMT Assembly Recommendations (Continued)



A, B and C of Stencil	1:1 ratio with A, B and C of PCB pad
D1	0.95 x D
E1	0.95 x E
F	0.1 mm
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 12. Typical Recommended PCB Dimensions vs. Stencil Apertures for Quad Packages with DAP < 2 mm.

SMT Assembly Recommendations (Continued)

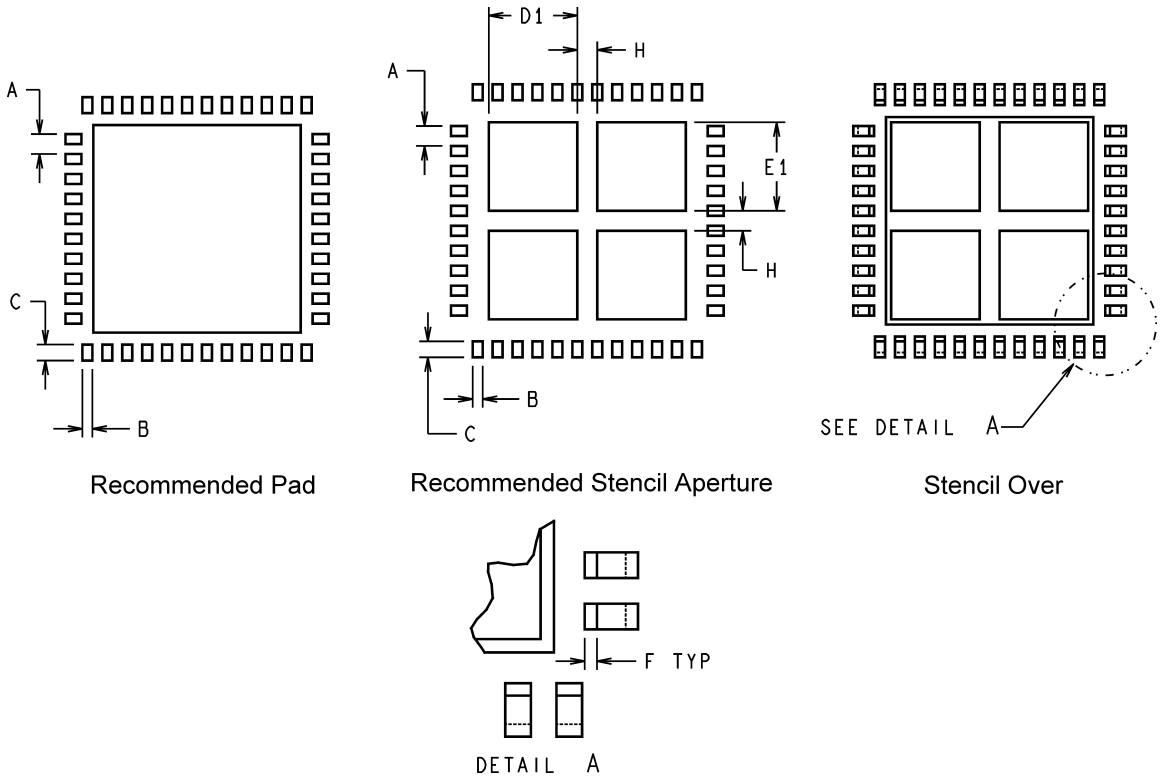


Recommended Pad | Recommended Stencil Aperture | Stencil Over

A, B and C of Stencil	1:1 ratio with A, B and C of PCB pad
D1	0.95 x D
E1	0.95 x E
F	0.1 mm
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 13. Typical Recommended PCB Dimensions vs. Stencil Apertures for Dual In-line Packages with DAP < 2 mm.

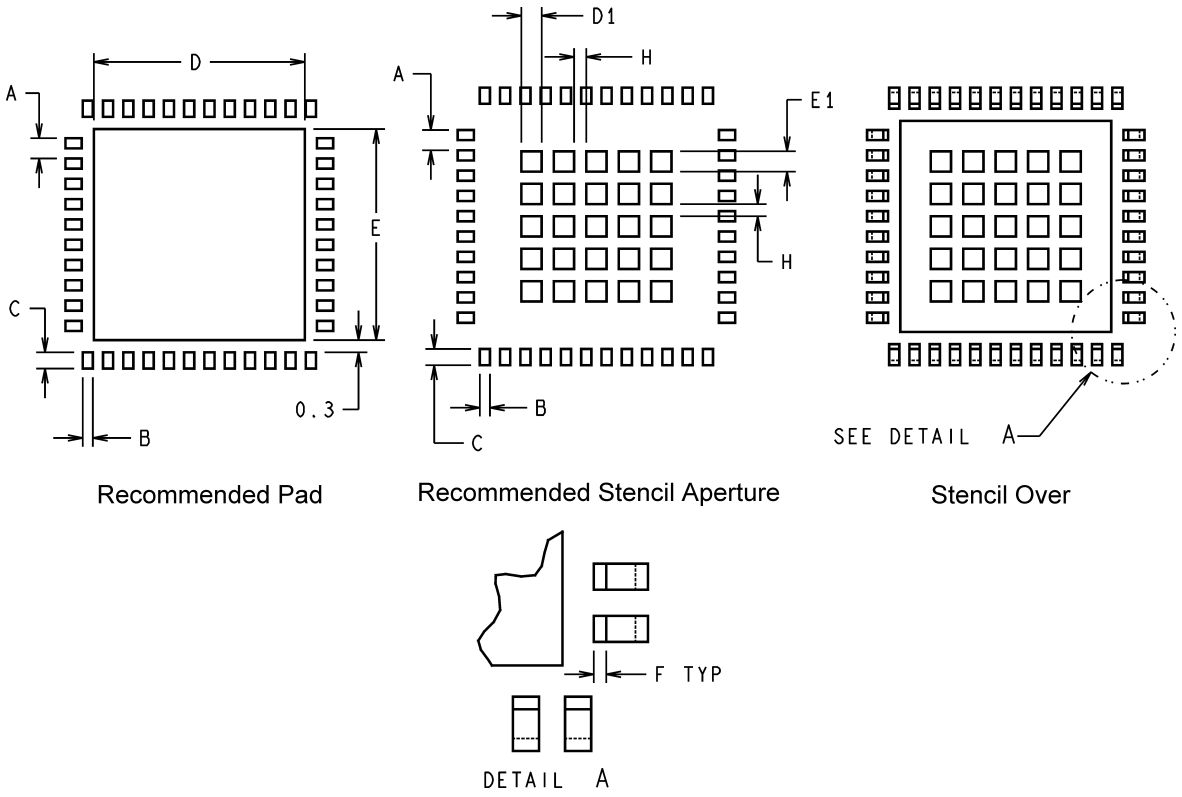
SMT Assembly Recommendations (Continued)



A, B and C of Stencil	1:1 ratio with A, B and C of PCB pad
D1	See *Table 2*
E1	
H	
F	0.1 mm
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 14. Typical Recommended Stencil Openings for Exposed DAP from 2 mm to 4 mm.

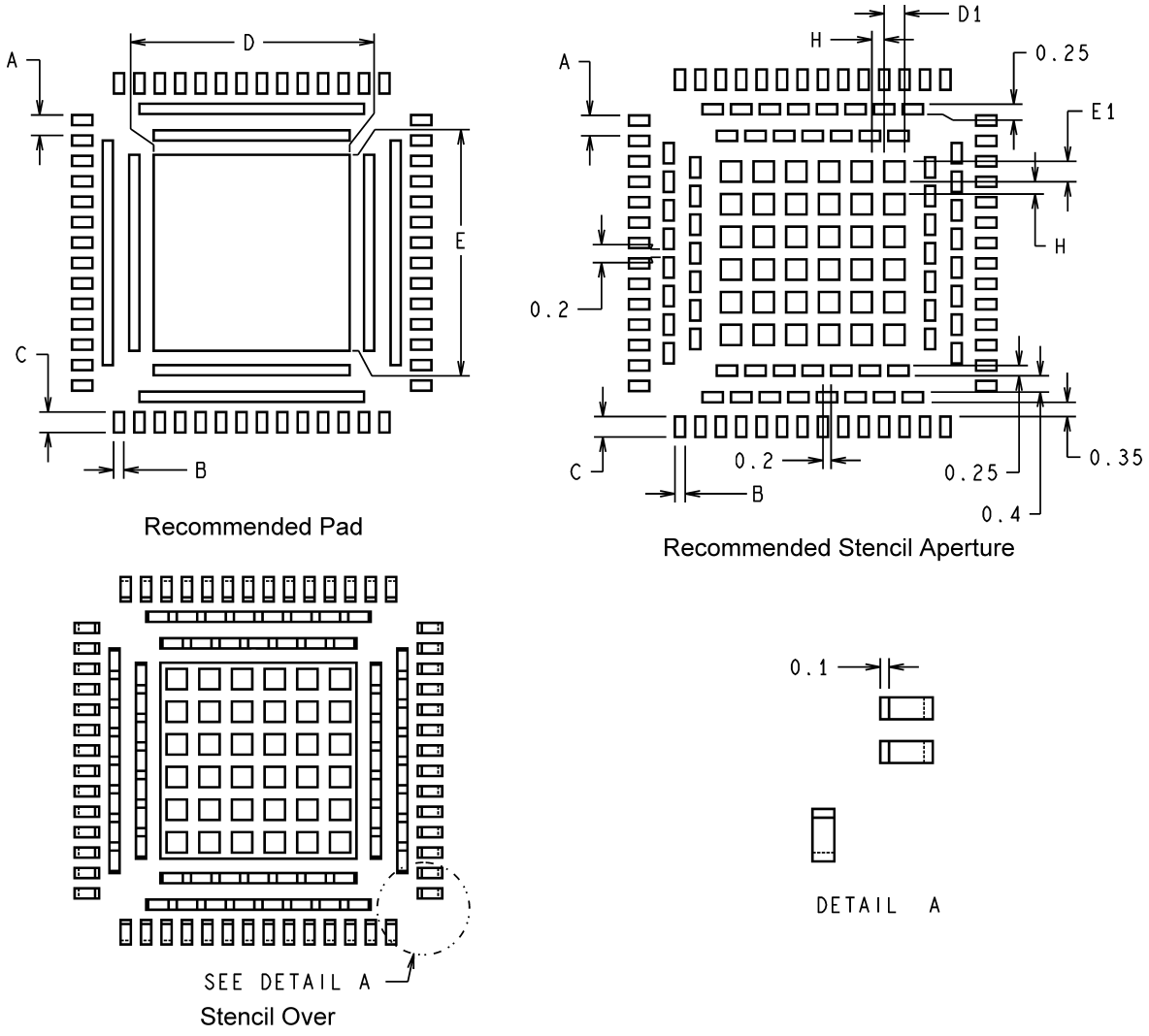
SMT Assembly Recommendations (Continued)



A, B and C of Stencil	1:1 ratio with A, B and C of PCB pad
D1	See *Table 2*
E1	
H	
F	0.1 mm
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 15. Typical Recommended Stencil Openings for Exposed DAP > 4 mm on any side without Ground and Power Bars.

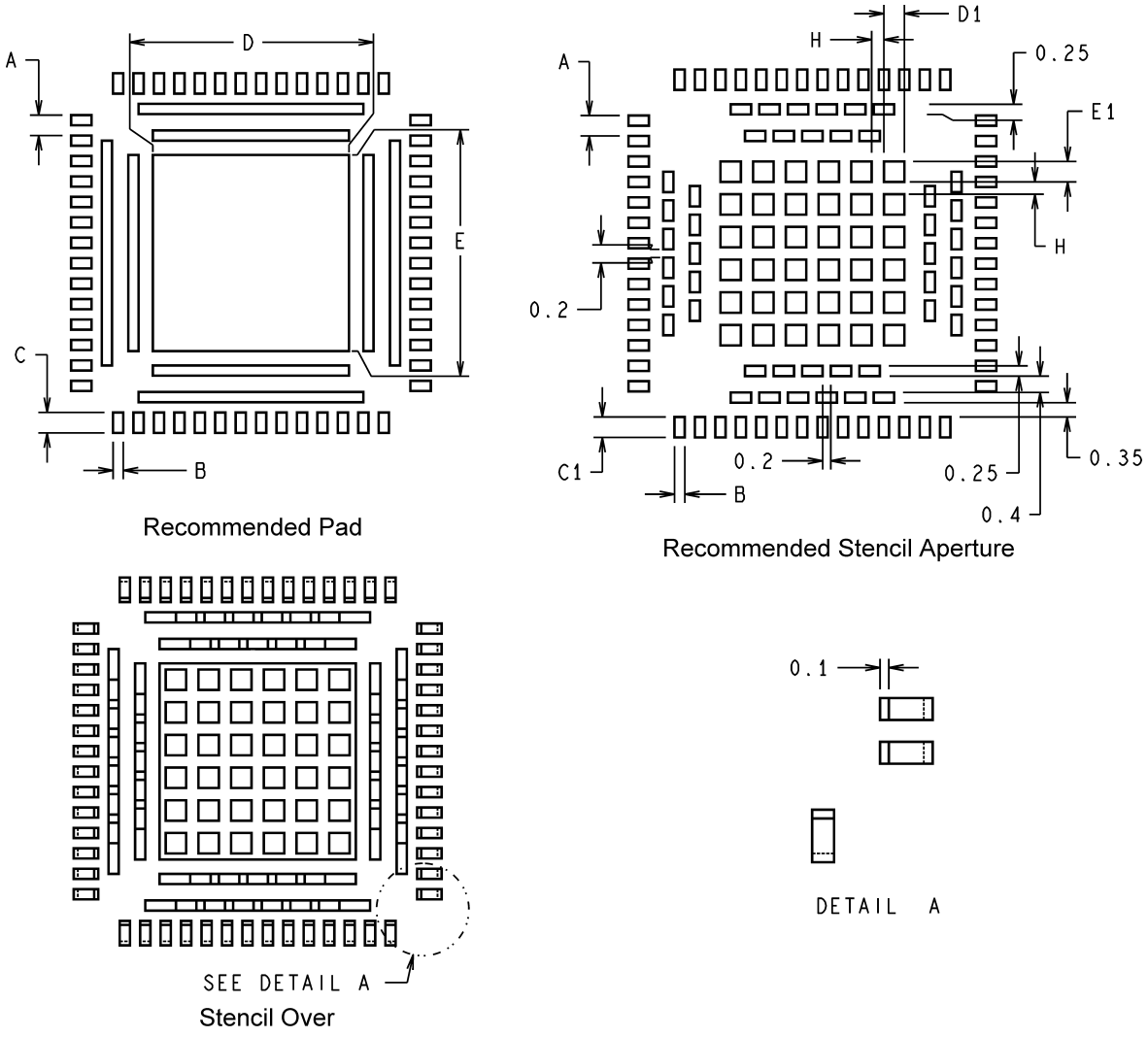
SMT Assembly Recommendations (Continued)



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 16. Typical Recommended Stencil Openings for LLP with Exposed DAP, Ground and Power Bars.

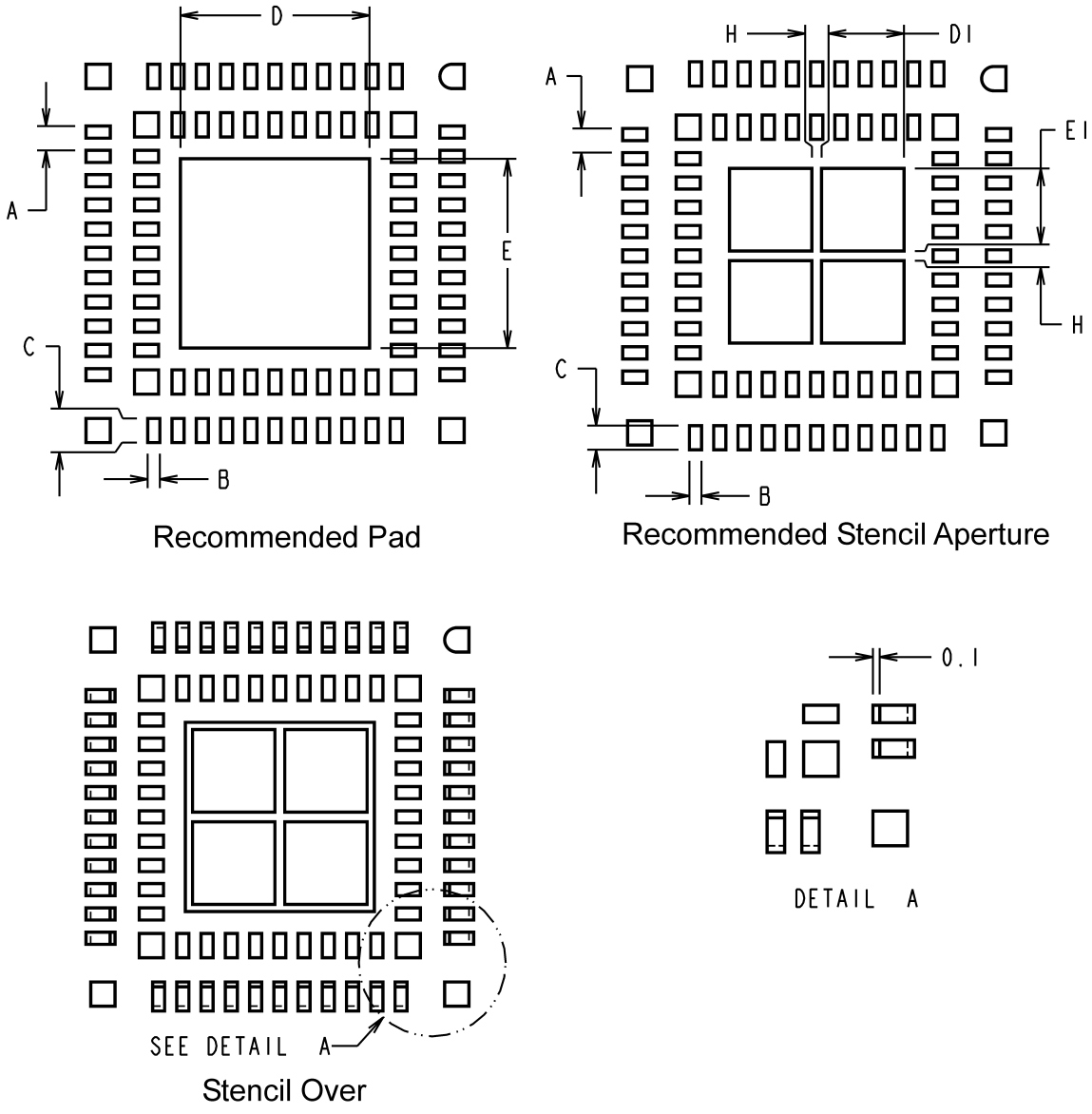
SMT Assembly Recommendations (Continued)



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 Terminal Length (mm)	0.5
C1 - LLP Stencil Terminal Length (mm)	0.45
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 17. Typical Recommended Stencil Openings for LLP with Exposed DAP, Ground and Power Bars for PCB with HASL Finish.

SMT Assembly Recommendations (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 18. Typical Recommended Stencil Openings for LLP with Exposed DAP and Dual Row.

SMT Assembly Recommendations (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 19* for solder stencil openings.
- For new board design, it is recommended to use *Figure 13* for PCB pad and stencil openings.

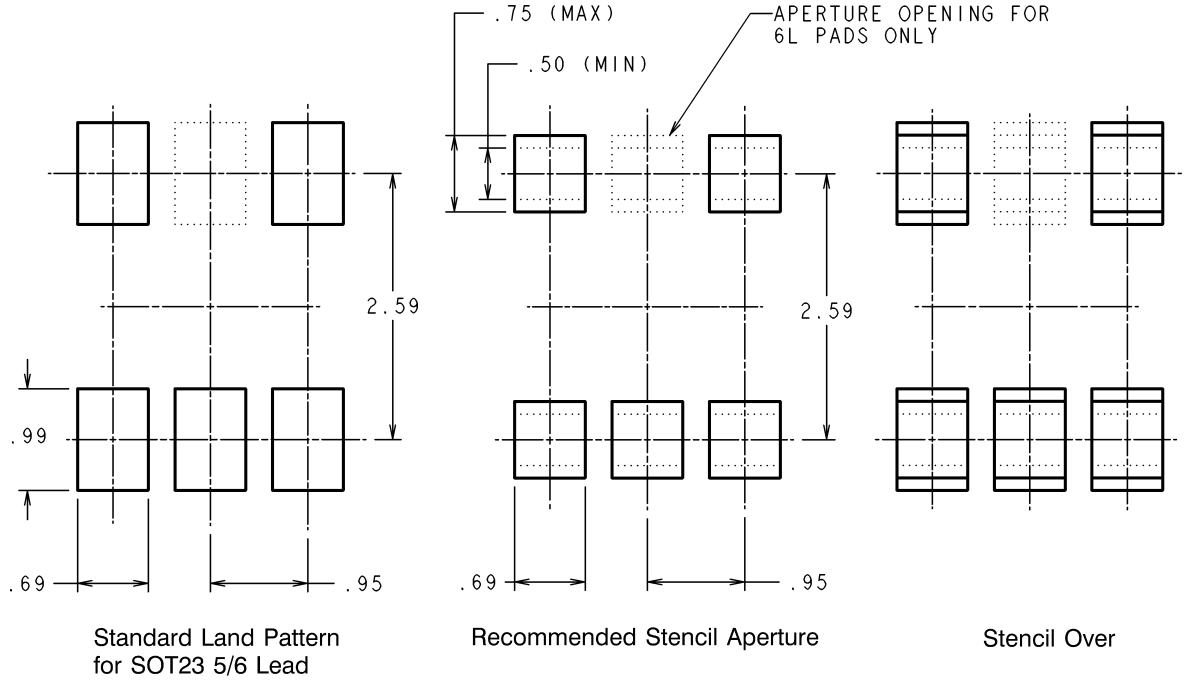


FIGURE 19. Recommended Stencil Apertures for SOT23 5/6 Lead Footprint Compatible LLP

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 bumps 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.

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 20* and *Figure 21* illustrate typical reflow profiles.

SMT Assembly Recommendations (Continued)

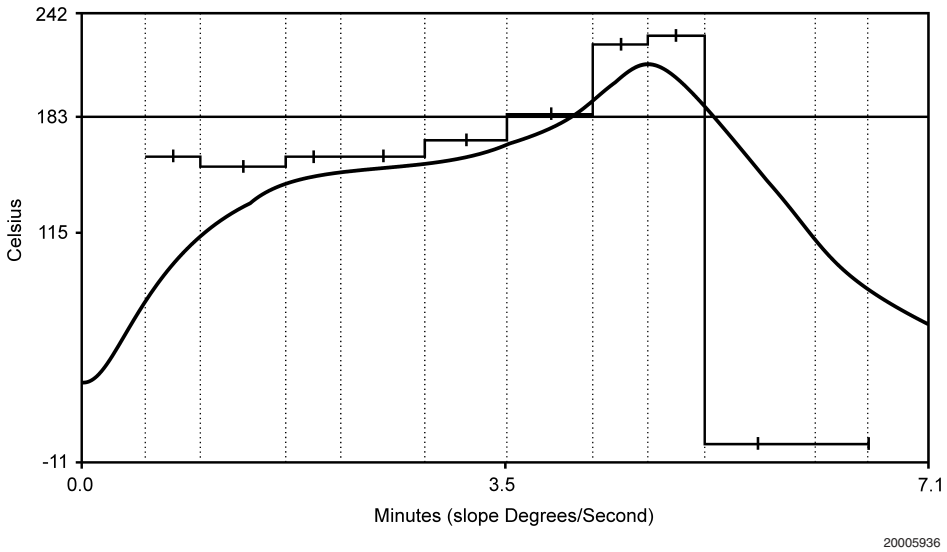


FIGURE 20. Typical Reflow Profile

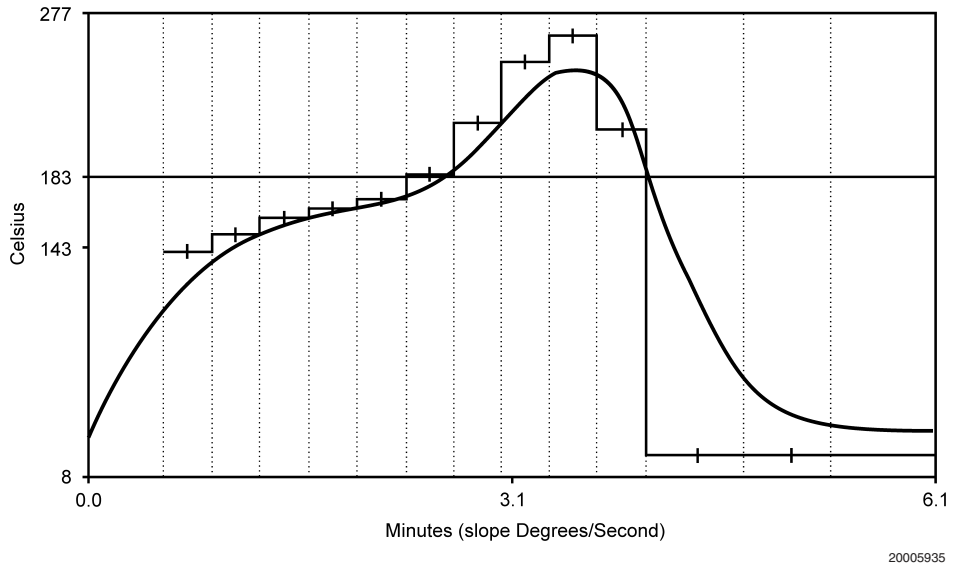
		Convection / IR
Ramp Up ˚C/sec (Note 8)	Maximum	4˚C/sec
	Recommended	**2˚C/sec** (Note 6)
	Minimum	(Note 7)
Dwell Time ≥ 183˚C (Note 8)	Maximum	85 seconds
	Recommended	**75 seconds** (Note 6)
	Minimum	(Note 7)
Peak Temperature (Note 8)	Maximum	240˚C
	Recommended	**215˚C**
	Minimum	(Note 7)
Dwell Time Max. (within 5˚C of peak temperature)	Maximum	10 seconds
	Recommended	**5 seconds**
	Minimum	1 second
Ramp Down ˚C/sec (Note 8)	Maximum	4˚C/sec
	Recommended	**2˚C/sec**
	Minimum	(Note 7)

Note 6: Will vary depending on board density, geometry, and package types. May vary depending on solder paste manufactures recommendations.

Note 7: Will vary depending on package types, and board density.

Note 8: All Temperatures are measured at the PCB surface.

SMT Assembly Recommendations (Continued)



Note : For detail settings, please refer to solder paste manufacturer's recommendation.

FIGURE 21. Typical Reflow Profile - Lead Free

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 22* shows a typical X-ray photograph after assembly.

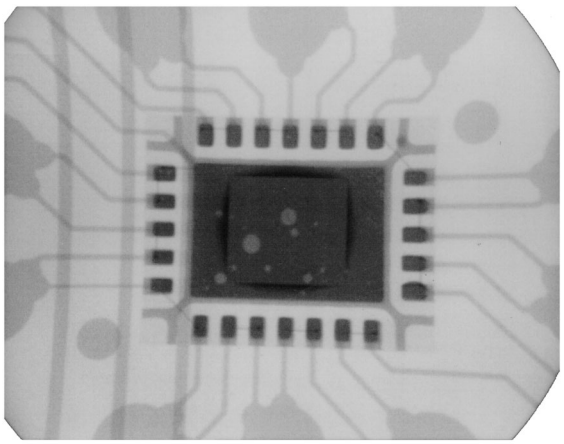


FIGURE 22. Typical X-ray after process

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.

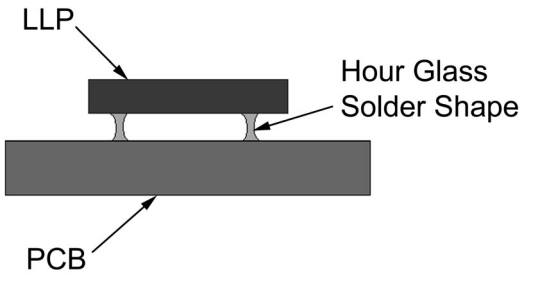


FIGURE 23.

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.
- 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.

SMT Assembly Recommendations

(Continued)

- 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.

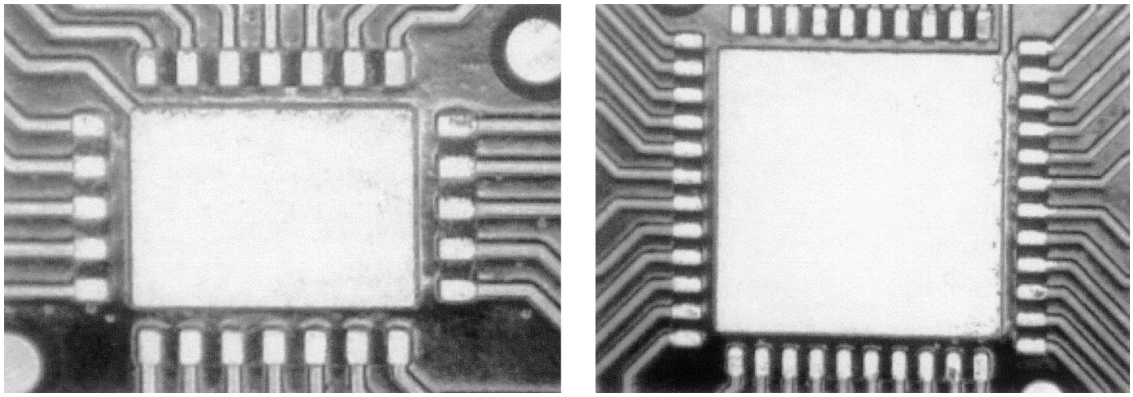
It is recommended that the reflow profile used to reflow the LLP be as close to the PCB mount profile as possible. Preheat the PCB area, through the bottom side of the board, to 100˚C before heating the LLP to ensure a controlled process. Once the liquidus temperature is reached, nozzle vacuum is automatically 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 24*.

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 25*.

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.

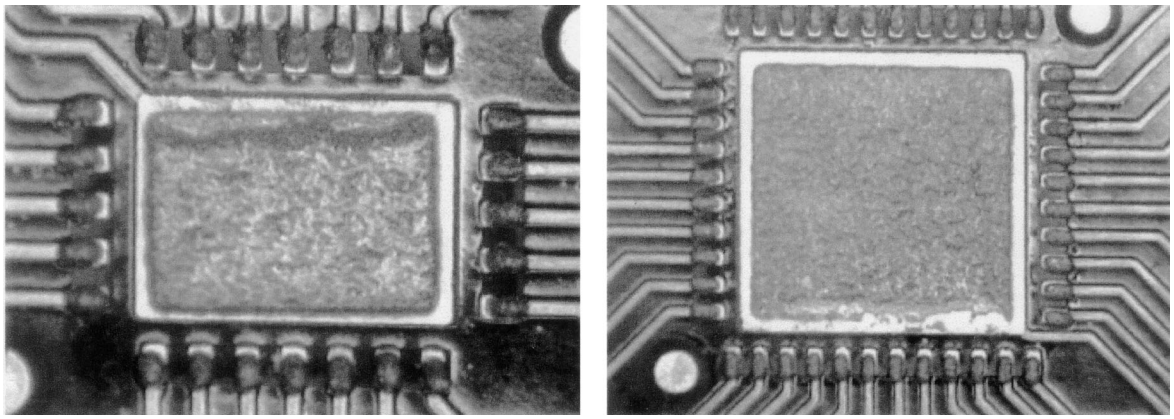
Component Reflow It is recommended that the reflow profile used to reflow the LLP be as close to the PCB mount profile as possible. Preheat the PCB area, through the bottom side of the board, to 100˚C before heating the LLP to ensure a controlled process. Once the liquidus temperature is reached, the solder will reflow and the LLP will self align. *Figure 26* shows a cross section of a solder joint after rework.



20005917

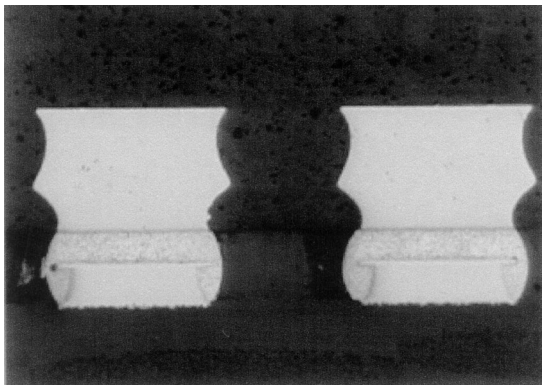
FIGURE 24. Pads After Removing Components and Cleaning

SMT Assembly Recommendations (Continued)



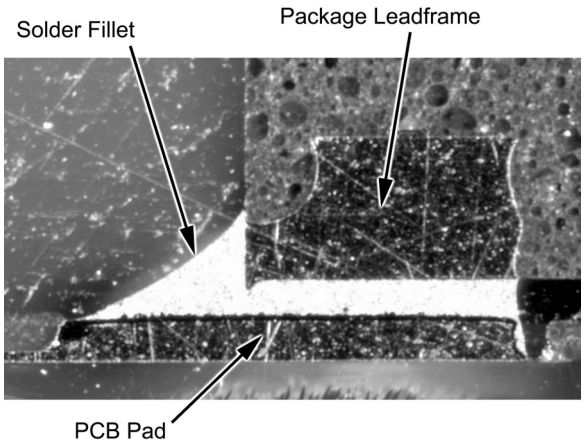
20005918

FIGURE 25. Solder Paste Printing of LLP 24 and LLP 44 Using 127 µm (5 mil) Thick Stencil



20005920

FIGURE 26. X-section Across Solder Joints



20005939

FIGURE 27. X-section of NO Pull Back Solder Joint with fillet

No Pull Back LLP

The No Pullback 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 No Pullback LLP has the following advantages:

- Low thermal resistance
- Reduced electrical parasitics
- Improved board space efficiency
- Reduced package height
- Reduced package mass
- Visible solder fillets
- Probing pads

ELEMENTS OF NO PULLBACK LLP

TABLE 3. Elements of the 10, 16 and 48 pin No Pull back LLP

	10 pin	16 pin	48 pin
Package Dimensions	4 x 4 x 0.8 mm	5 x 5 x 0.8 mm	7 x 7 x 0.8 mm
PCB Footprint Area (mm^2)	17.64	27.04	51.84
Standard	JEDEC	JEDEC	JEDEC
Pitch	0.8 mm	0.5 mm	0.5mm
Weight	0.037 grams	0.062 grams	0.125 grams
Leadframe	Copper	Copper	Copper
Lead Finish	Sn/Pb	Sn/Pb	Sn/Pb
Typical Thermal Resistance θ_{JA} (Note 9)	38˚C/W	32˚C/W	20˚C/W (Note 10)

Note 9: The typical data reported are measured values at still air and 1 watt input power using four layer FR4 substrate with Vias and copper thickness of 2.0/1.0/1.0/2.0 oz.

Note 10: Package option with limited exposed pad size due to incorporations of ground and power rings.

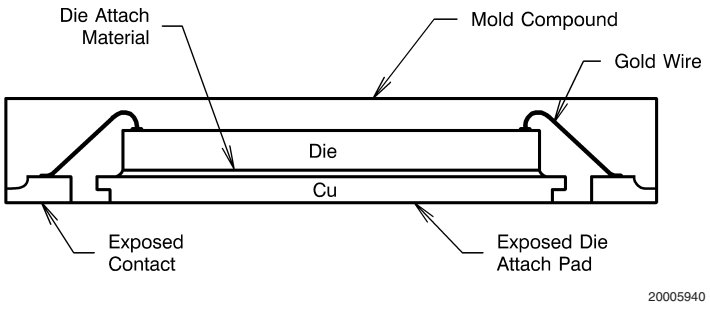


FIGURE 28. Construction of No Pull Back LLP

NO PULL BACK LLP PACKAGE OFFERING

Pin Count	Body size (mm)	Pitch (mm)	Max. Die Size (mm) (Note 13)	MKT DWG	MBS AD#	L/F Dwg #	QUAD/DIP (Note 12)	θ_{JA} (˚C/W) (Note 11)
6	3.0 x 3.0	0.95	1.96 x 1.55	sde06a	sde006aa	36-1113	DIP	55.3
8	4.0 x 4.0	0.8	2.29 x 2.7	sdc08a	sdc008aa	36-1109	DIP	38.8
10	4.0 x 4.0	0.8	2.29 x 2.7	sdc10a	sdc010aa	36-1106	DIP	38.8
16	5.0 x 5.0	0.5	3.69 x 3.3	sda16a	sda016aa	36-1108	DIP	N/A
24	4.0 x 4.0	0.5	2.29 x 2.29	sqa24a	sqa024aa	36-1092	QUAD	39.0
28	5.0 x 5.0	0.5	3.30 x 3.30	sqa28a	sqa028aa	28-LPCC-0008	QUAD	30.8
36	6.0 x 6.0	0.5	4.3 x 4.3	sqa36a	sqa036aa	36-1095	QUAD	N/A
48	7.0 x 7.0	0.5	4.54 x 4.54	sqa48a	sqa048aa	36-1096	QUAD	N/A

Note 11: 4-layer board with Cu finished thickness 1.5/1/1/1.5 oz. Maximum die size used. 5x body length of Cu trace on PCB top. 50 x 50 mm ground and power planes embedded in PCB.

Note 12: DIP : Only two sides of the package have leads. QUAD: All four sides of the package have leads.

Note 13: Maximum die size without downbond.

No Pull Back LLP (Continued)

PCB DESIGN RECOMMENDATIONS

The No Pullback LLP PCB layout is similar to the LLP PCB layout, except the PCB pad length is extended by 0.2 mm compared to the component pad length. See *Table 4*

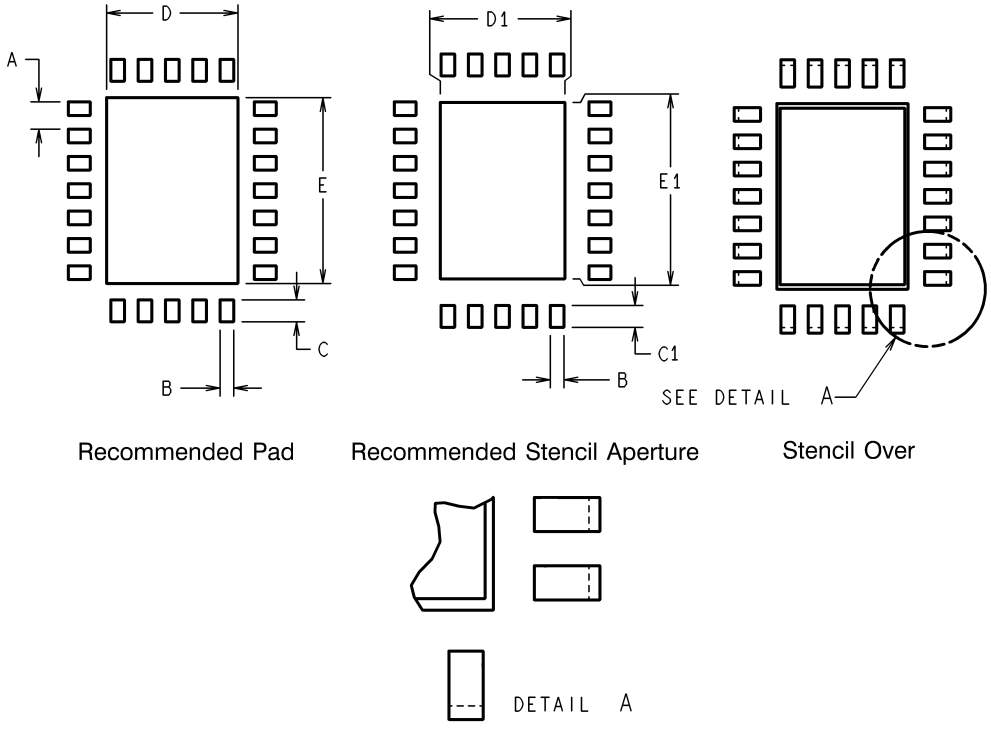
TABLE 4. Component Pad to PCB Pad Comparison

Component pad size (mm)	PCB I/O Pad Size (mm)
0.25 X 0.4	0.25 X 0.6
0.30 X 0.4	0.30 X 0.6

STENCIL APERTURE DESIGN FOR NO PULLBACK LLP

TABLE 5. No Pull Back LLP Stencil Aperture Summary.

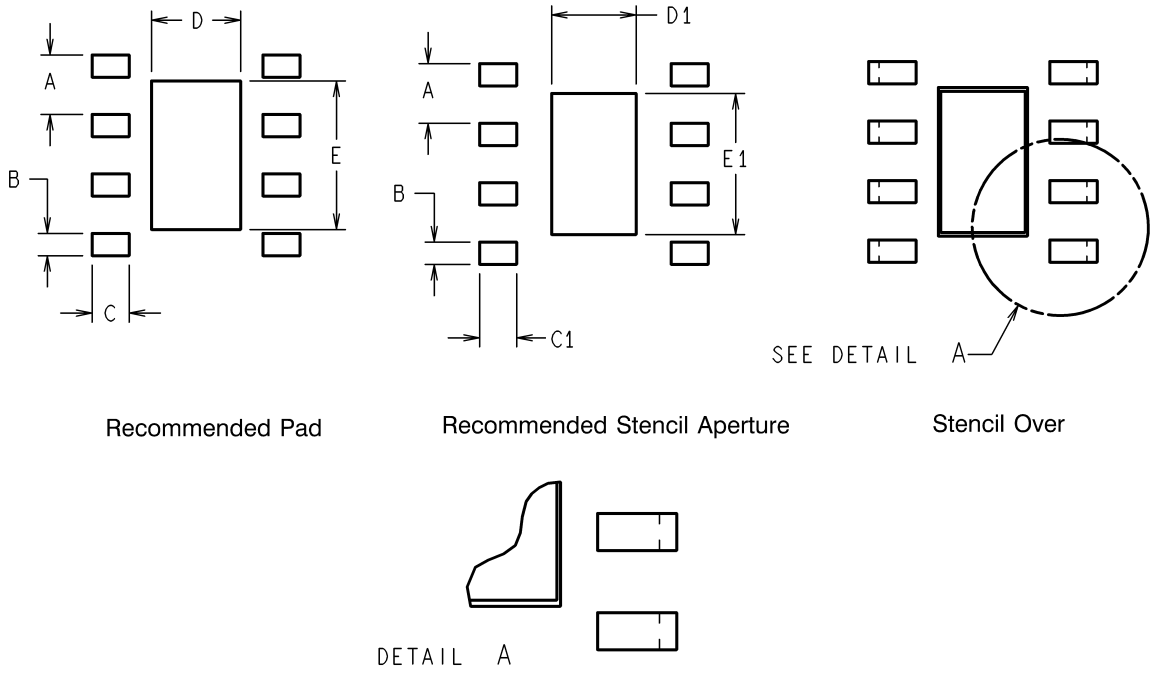
Pin Count	MKT Dwg	PCB I/O Pad Size (mm)	PCB Pitch (mm)	PCB DAP size (mm)	Stencil I/O Aperture (mm)	Stencil DAP Aperture	Dimension (mm)		
							D1	E1	H
6	sde06a	0.25 x 0.6	0.95	2.1 x 1.6	0.25 x 0.7	*Figure 30*	2.1	1.6	N/A
8	sdc08a	0.30 x 0.6	0.8	3.0 x 2.6	0.30 x 0.7	*Figure 31*	1.35	1.15	0.3
10	sdc10a	0.30 x 0.6	0.8	3.0 x 2.6	0.30 x 0.7	*Figure 31*	1.35	1.15	0.3
16	sda16a	0.25 x 0.6	0.5	4.0 x 3.6	0.25 x 0.7	*Figure 31*	1.85	1.65	0.3
24	sqa24a	0.25 x 0.6	0.5	2.6 x 2.6	0.25 x 0.7	*Figure 31*	1.15	1.15	0.3
28	sqa28a	0.25 x 0.6	0.5	3.6 x 3.6	0.25 x 0.7	*Figure 31*	1.65	1.65	0.3
36	sqa36a	0.25 x 0.6	0.5	4.6 x 4.6	0.25 x 0.7	*Figure 32*	1	1	0.2
48	sqa48a	0.25 x 0.6	0.5	5.1 x 5.1	0.25 x 0.7	*Figure 32*	1.10	1.10	0.2



A and B of Stencil	1:1 ratio with A and B of PCB pad
C1	C + 0.1
D1	0.95 x D
E1	0.95 x E
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 29. Typical Recommended PCB Dimensions vs. Stencil Apertures for Quad Packages with DAP < 2 mm.

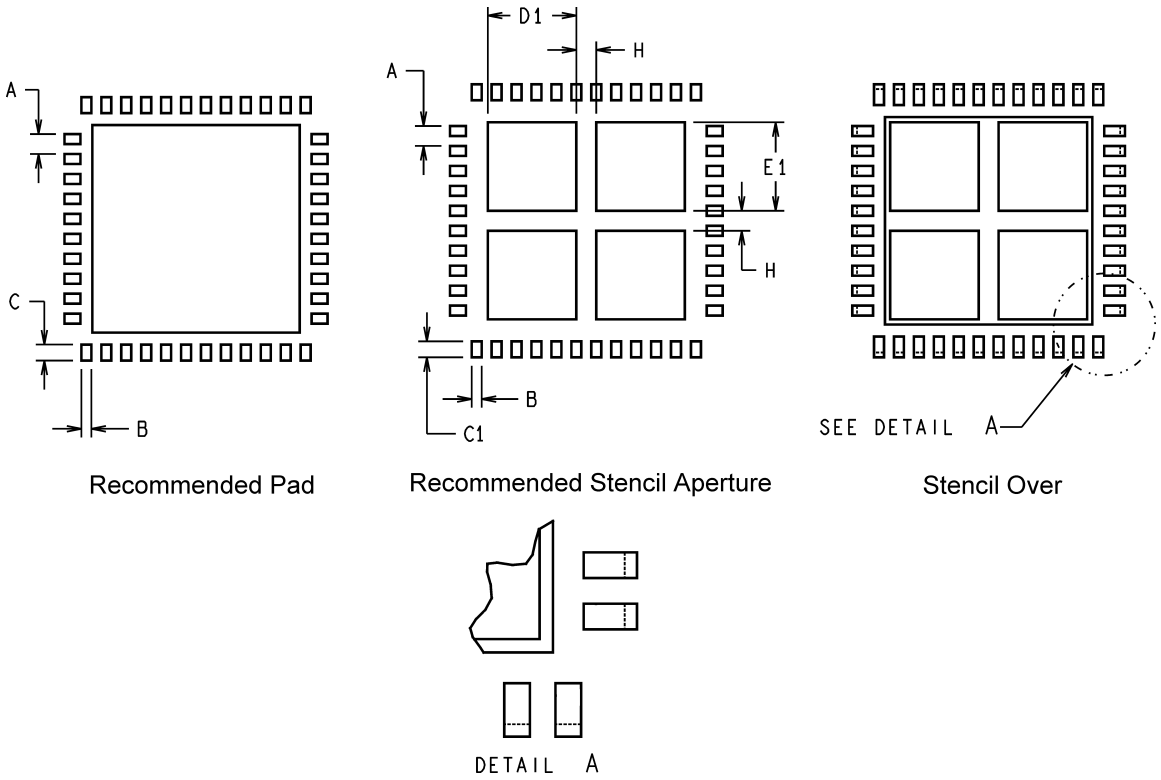
No Pull Back LLP (Continued)



A and B of Stencil	1:1 ratio with A and B of PCB pad
C1	C + 0.1
D1	0.95 x D
E1	0.95 x E
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 30. Typical Recommended PCB Dimensions vs. Stencil Apertures for Dual In-line Packages with DAP < 2 mm.

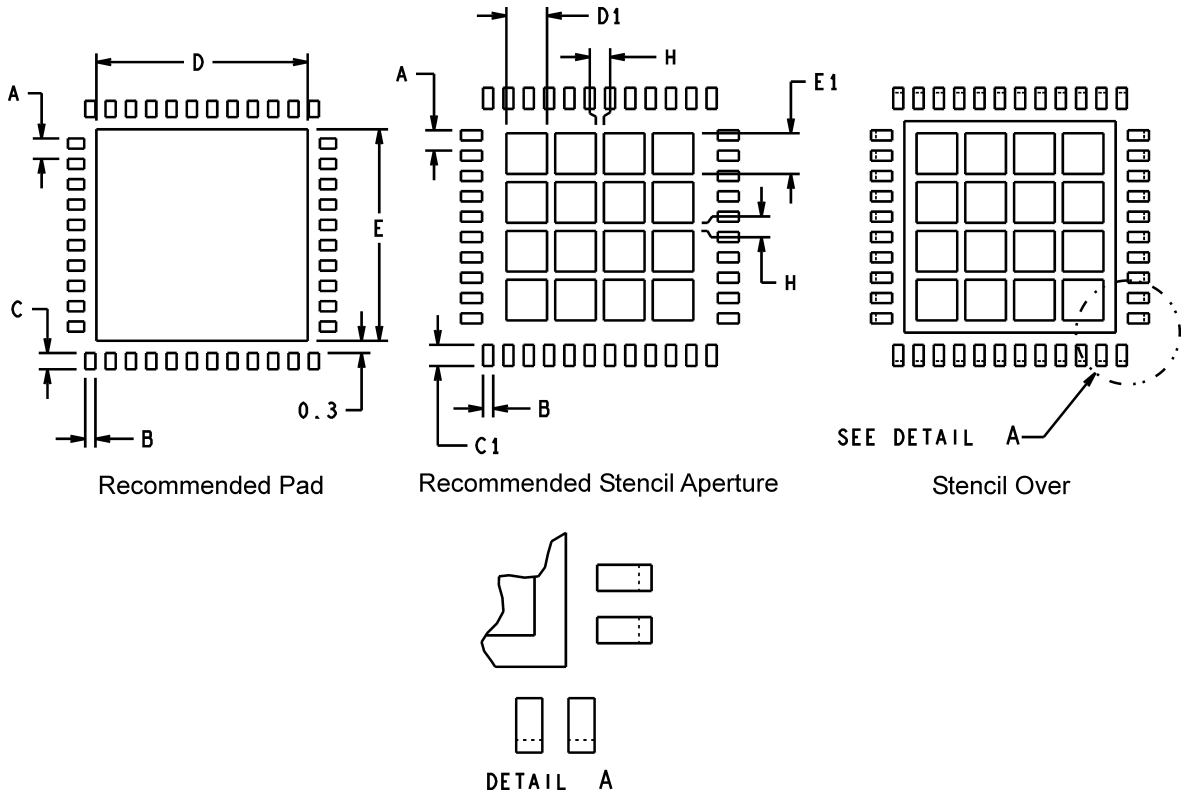
No Pull Back LLP (Continued)



A, B and C of Stencil	1:1 ratio with A, B and C of PCB pad
C1	C + 0.1
D1	See *Table 5*
E1	
H	
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 31. Typical Recommended Stencil Openings for Exposed DAP from 2 mm to 4 mm.

No Pull Back LLP (Continued)



A, B and C of Stencil	1:1 ratio with A, B and C of PCB pad
C1	C + 0.1
D1	See *Table 5*
E1	
H	
Note: For specific detailed package dimensions refer to respective Marketing Outlines.	

FIGURE 32. Typical Recommended Stencil Openings for Exposed DAP > 4 mm on any side.

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 0.127 µm thickness
- Dummy die in package
- Package is bonded with a Daisy Chain Circuit

Failure Determination: Change of 10% in Net Resistance

Results:

24L 4 mm x 5 mm LLP Package
(Package Die Attach Pad soldered to the PCB)

Timepoint	Lot A	Lot B	Lot C
0 Cycles	0/41	0/84	0/83
500 Cycles	0/41	0/84	0/83
1050 Cycles	0/41		

24L 4 mm x 5 mm LLP Package
(Package Die Attach Pad NOT soldered to the PCB)

Timepoint	Lot A	Lot B	Lot C
0 Cycles	0/81	0/78	0/76
500 Cycles	0/81	0/78	0/76
950 Cycles	0/81	0/78	0/76
1050 Cycles	0/81	0/78	0/76

44L 7 mm x 7 mm LLP Package
(Package Die Attach Pad soldered to the PCB)

Timepoint	Lot A	Lot B	Lot C
0 Cycles	0/33	0/69	0/88
500 Cycles	0/33	0/69	0/88

Appendices (Continued)

Timepoint	Lot A	Lot B	Lot C
1050 Cycles	0/33	0/69	0/88

44L 7 mm x 7 mm LLP Package
(Package Die Attach Pad NOT soldered to the PCB)

Timepoint	Lot A	Lot B	Lot C
0 Cycles	0/81	0/78	0/76
500 Cycles	0/81	0/78	0/76
950 Cycles	0/81	0/78	0/76
1050 Cycles	0/81	0/78	0/76

Standard 56L 9 mm x 9 mm Package
(Power and Ground Ring not soldered to the PCB)

Timepoint	Results
0 Cycles	0/75
500 Cycles	0/75
1050 Cycles	0/75

Lead-Free 56L 9 mm x 9 mm Package
(Power and Ground Ringsoldered to the PCB with SnAgCu solder paste and Sn lead finish)

PCB Finish	750 TMCL	1050 TMCL
NiAu	0/98	0/98
OSP	0/96	0/96

Standard & Lead-Free 56L LLP Board Level TMCL Comparison

Lead Finish	Solder Paste	PCB Finish	750 TMCL	1050 TMCL
Sn	SnPb	NiAu	0/100	0/100
SnPb	SnPb	NiAu	0/100	0/100
Sn	SnPb	OSP	0/99	0/99
SnPb	SnPb	OSP	0/95	0/95
Sn	SnAgCu	NiAu	0/98	0/98
SnPb	SnAgCu	NiAu	0/99	0/99
Sn	SnAgCu	OSP	0/96	0/96

SOT23 5/6L Footprint Compatible LLP

Timepoint	DAP soldered to PCB	DAP not soldered to PCB
0 cycles	0/126	0/84
500 cycles	0/126	0/84
1050 cycles	0/126	0/84

14 Lead Power LLP

Timepoint	Results
0 Cycles	0/80
500 Cycles	0/80
1050 Cycles	0/80

No Pull Back 48L 7 mm x 7 mm Package

Timepoint	Results
0 Cycles	0/64
500 Cycles	0/64
750 Cycles	0/64
1050 Cycles	0/64

Board Drop Test

Test Conditions:

- Test Board Dimension: 142.5 mm x 142.5 mm x 1.6 mm
- Printed Circuit Board Finish: Ni-Au 2 - 5 micro inches 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

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 to 0.127 µm thickness
- Dummy die in package
- Package is bonded with a Daisy Chain Circuit
- Die attach pad soldered to PCB

Appendices (Continued)

- 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

APPENDIX 2: THERMAL SIMULATION DATA FOR POWER LLP

Thermal Simulation Conditions

All dimensions are in millimeters

Die Size	4.09 x 2.67 x 0.216
DAP Size	4.35 x 3.00
Package Size	6.00 x 5.00 x 1.00
Thermal Vias	0, 2, 4, 8, 12. See *Figure 33*.
Board Size	101.6 x 76.2 x 1.6 (4 layer JEDEC)
Copper Thickness	2.0/1.0/1.0/2.0 oz. (1 oz. = 36 µm)
Copper Coverage	Top layer: traces (27.5 x 0.25) plus metalization area as shown in *Figure 34*. Middle layers: 60.0 x 60.0 Bottom layer: 15% of the board area.

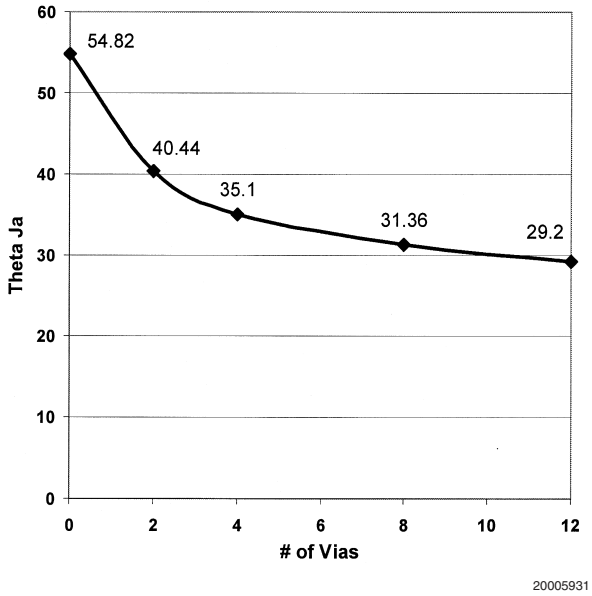


FIGURE 33. θ_{JA} as a Function of Number of Vias Placed in PCB

Appendices (Continued)

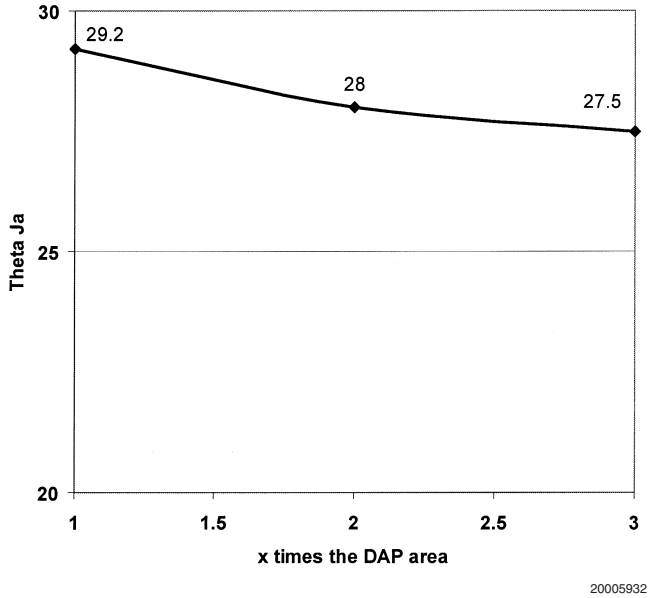


FIGURE 34. θ_{JA} as a Function of Top Metalization Area

LIFE SUPPORT POLICY

NATIONAL'S PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF THE PRESIDENT AND GENERAL COUNSEL OF NATIONAL SEMICONDUCTOR CORPORATION. As used herein:

1. Life support devices or systems are devices or systems which, (a) are intended for surgical implant into the body, or (b) support or sustain life, and whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in a significant injury to the user.
2. A critical component is any component of a life support device or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.

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