PQFN88, PQFN56, and TO-263 Lead-free 2nd Level Soldering Recommendations for Vapor Phase Reflow

Contents

1.0 PQFN88: LS and LD Package Description 2
1.1 Printed circuit board 3
PCB footprint 3
Pad surface 4
1.2 Printed circuit board reflow solder and solder paste printing 4
Solder stencil 4
Solder paste 6
Reflow soldering 6
2.0 PQFN88: LSG and LDG Package description 8
2.1 Printed circuit board 8
PCB footprint 8
2.2 Solder Stencil 10
3.0 PQFN56: JSG Package description 11
3.1 Printed circuit board 11
PCB footprint 11
3.2 Solder Stencil 12
4.0 TO-263 Package Description 13
4.1 Printed circuit board 14
PCB footprint 14
4.2 Solder Stencil 15
Disclaimer 16
Electrostatic discharge (ESD) 16
Moisture sensitivity level 16

© 2024 Renesas Electronics

1.0 PQFN88: LS and LD Package Description

Renesas's PQFN (Power Quad Flatpack No Lead) package incorporates a DPC (Direct Plated Cu) substrate and a Cu lead frame encapsulated, in a green molding compound for bottom electrical connection and thermal contact to the PBC (printed circuit board). It provides high lateral electrical isolation and excellent heat dissipation similar to a D2Pak but in a thinner form factor.

Features

- Low profile and lead inductance
- RoHS-compliant

© 2024 Renesas Electronics

Figure 1. TPH3206LD – common drain package configuration (bottom view)

Figure 2. TPH3206LS – common source package configuration (bottom view)

1.1 Printed circuit board

PCB footprint

As with any PQFN package, it is important the exposed solder pads are designed to assure optimum thermal and electrical characteristics. This exposed pad must be connected to the circuit board with solder. Vias can be used as thermal paths between different layers of the circuit board. Copper dimension and thickness will depend on the capability of the board manufacturer.

The circuit board footprint in Figures 3 and 4 demonstrate good self-alignment during the reflow soldering process. Please note that this recommendation on offers dimensions for the solder mask opening

.

© 2024 Renesas Electronics

Figure 3. Recommended solder mask opening (mm) for PQFN88 (drain tab)

Figure 4. Recommended solder mask opening (mm) for PQFN88 (source tab)

Pad surface

Solder pad finish is important for good solder paste wetting. It is difficult to recommend a certain pad surface finish that will meet all requirements. The choice for pad finishes depends on the board design, pad geometry, components, solder type, and process condition. The specific choice will be according to the customer requirements.

Renesas's internal evaluation has demonstrated that Electroless Ni / Emmersion Au (ENIG) surface has the best solder wetting for Pb-free soldering.

1.2 Printed circuit board reflow solder and solder paste printing

Solder stencil

A solder stencil is used for applying solder paste onto the circuit board. The amount of solder paste that is applied to the circuit board is determined by the stencil apertures and thickness. A stainless steel stencil and laser cut is preferred. Another factor in achieving uniform and high solder paste transfer is the squeegee blade angle and speed. All components on the circuit board must be considered when choosing the stencil thickness. The stencil thickness will range between 100µm to 150µm (4 mils to 6 mils). Stencil aperture is given in Figures 5 and 6.

© 2024 Renesas Electronics

Renesas's internal investigation concluded that the best results were achieved with the PCB having vias in the drain or source pad and a stencil thickness of 127µm (5 mils).

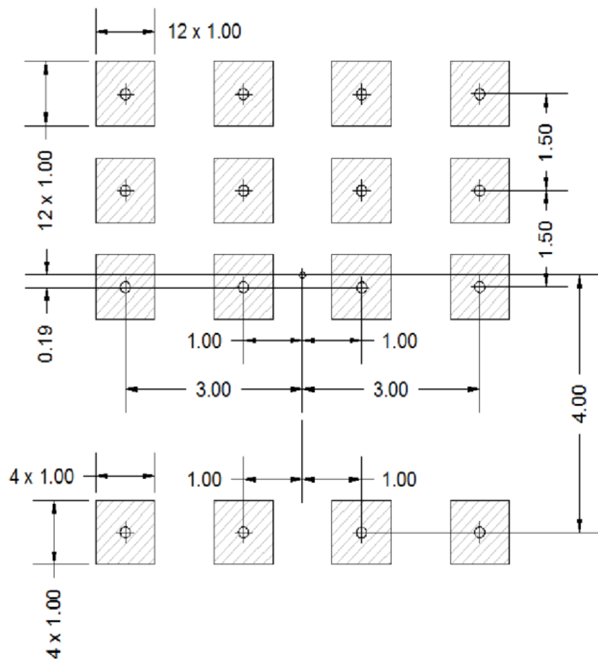


Figure 5. Recommended solder paste stencil (mm) for PQFN88 (drain tab)

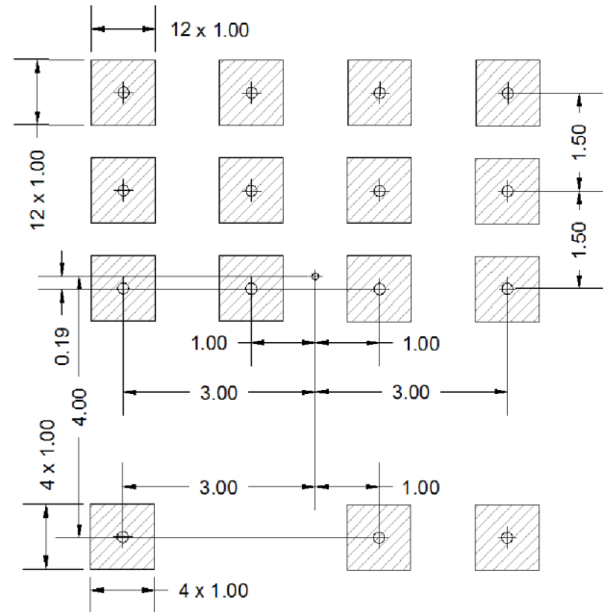


Figure 6. Recommended solder paste stencil (mm) for PQFN88 (source tab)

© 2024 Renesas Electronics

Solder paste

Solder paste is an important factor to consider. It is important to choose the correct solder paste to achieve high reliability and process repeatability. The characteristics of the solder paste will determine the temperature profile and other reflow parameters. It is important to follow the paste manufacturer suggested thermal profile. The solder paste will also determine the stencil aperture dimensions.

Renesas has achieved suitable results using SAC 305 (96.5Sn 3.0Au 0.5Cu) "No Clean" solder paste for Pb-free applications.

Reflow soldering

Soldering is the determining factor for high quality and good yields. It is important to check your temperature profile. High mass components will not heat up the same as lightweight components. If there are multiple components on the board, the profile temperature should be checked at different locations. Most boards will contain more than one package type; therefore, the reflow profile must meet all components requirements. It is important to follow the solder paste manufacturer profile specifications.

The recommended process is Vapor Phase Reflow, as it has improved solder joint quality (see Figure 7 for a profile example and Table 1 for example parameters). The recommended solder void criteria for Vapor Phase are 5% on a single void and 10% total voids. X-rays are included for Vapor Phase (Figure 8) and Standard Reflow (Figure 9).

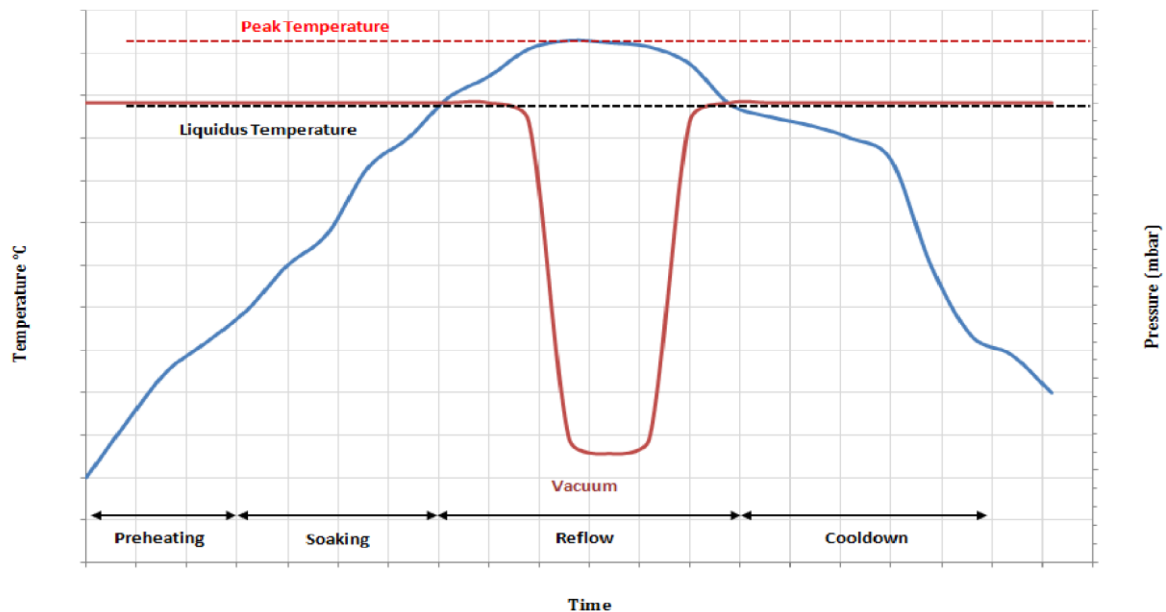


Figure 7. Sample profile for Vapor Phase Reflow

Parameter	Pd-free alloy (SAC 305)
Preheating temperature	170°C

© 2024 Renesas Electronics

Preheating time	50 seconds
Soaking temperature	200°C
Soaking time	75 seconds
Peak temperature (liquidus)	245°C
Peak time (liquidus)	75 seconds
Vacuum pressure (liquidus)	125 mbar
Vacuum time	25 seconds

Table 1. Sample parameters for Vapor Phase Reflow

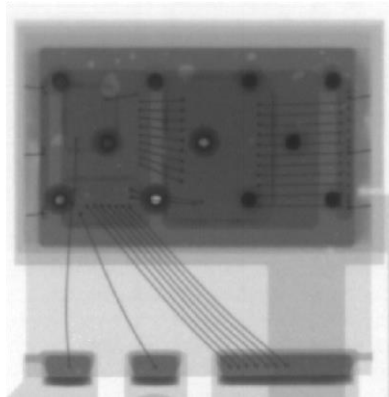


Figure 8. Vapor Phase Reflow x-ray

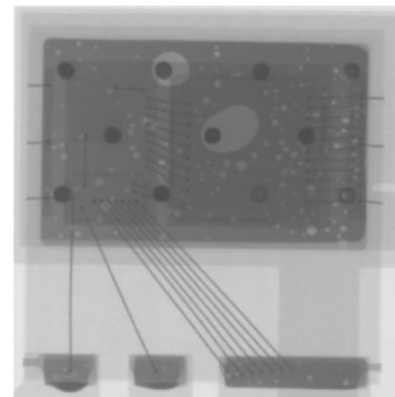


Figure 9. Standard Reflow x-ray

© 2024 Renesas Electronics

2.0 PQFN88: LSG and LDG Package description

Figure 10. TPH320xLDG – common drain package configuration (bottom view)

Figure 11. TPH320xLSG – common source package configuration (bottom view)

Note: TP65H0xxLxG uses the same package landing pattern as Figure 10 and 11

2.1 Printed circuit board

PCB footprint

As with any PQFN package, it is important the exposed solder pads are designed to assure optimum thermal and electrical characteristics. This exposed pad must be connected to the circuit board with solder. Vias can be used as thermal paths between different layers of the circuit board. Copper dimension and thickness will depend on the capability of the board manufacturer.

The circuit board footprint in Figures 12 and 13 demonstrate good self-alignment during the reflow soldering process. Please note that this recommendation on offers dimensions for the solder mask opening.

© 2024 Renesas Electronics

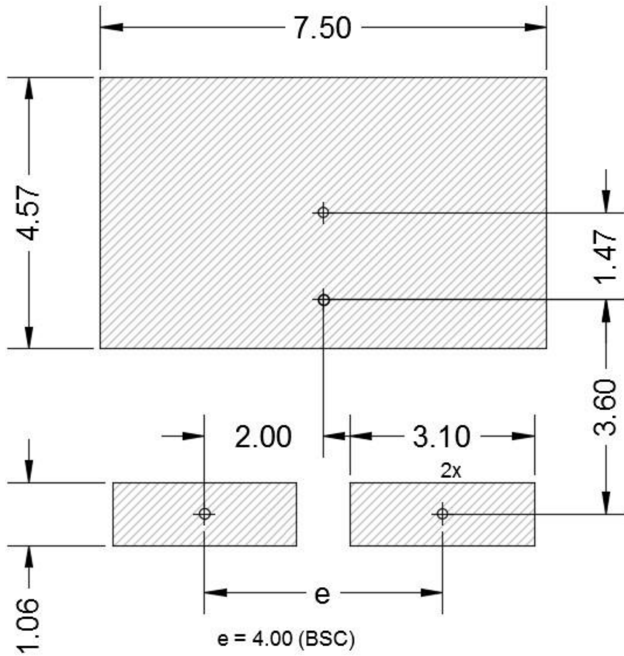


Figure 12. Recommended solder mask opening (mm) PQFN88 (drain tab)

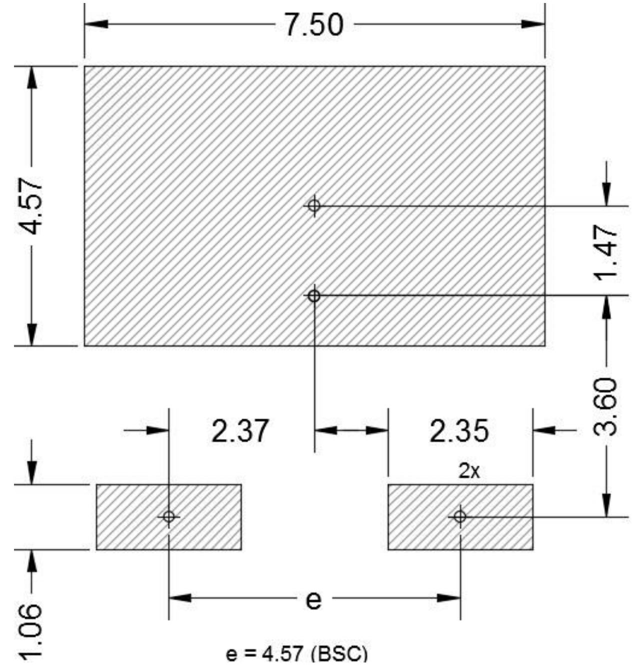


Figure 13. Recommended solder mask opening (mm) for for PQFN88 (source tab)

© 2024 Renesas Electronics

2.2 Solder Stencil

A stainless steel stencil and laser cut is preferred. Another factor in achieving uniform and high solder paste transfer is the squeegee blade angle and speed. All components on the circuit board must be considered when choosing the stencil thickness. The stencil thickness will range between 100μm to 150μm (4 mils to 6 mils). Stencil aperture is given in Figures 14 and 15.

Renesas's internal investigation concluded that the best results were achieved with the PCB having vias in the drain or source pad and a stencil thickness of 127μm (5 mils).

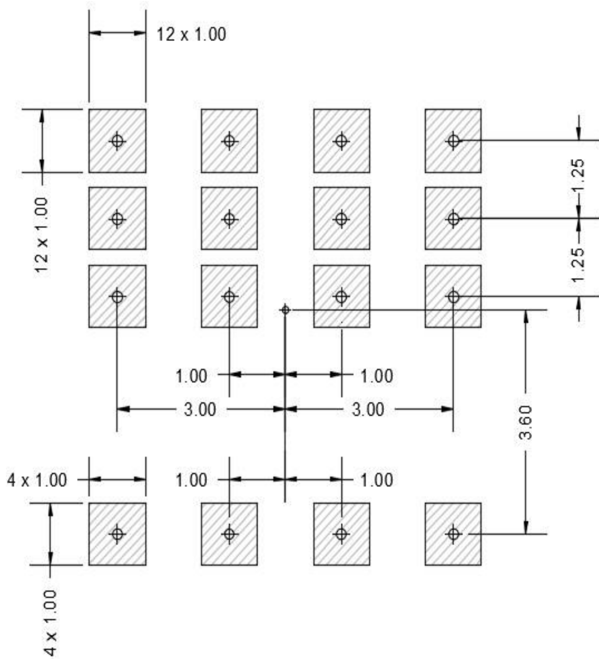


Figure 14. Recommended solder paste stencil (mm) for PQFN88 (drain tab)

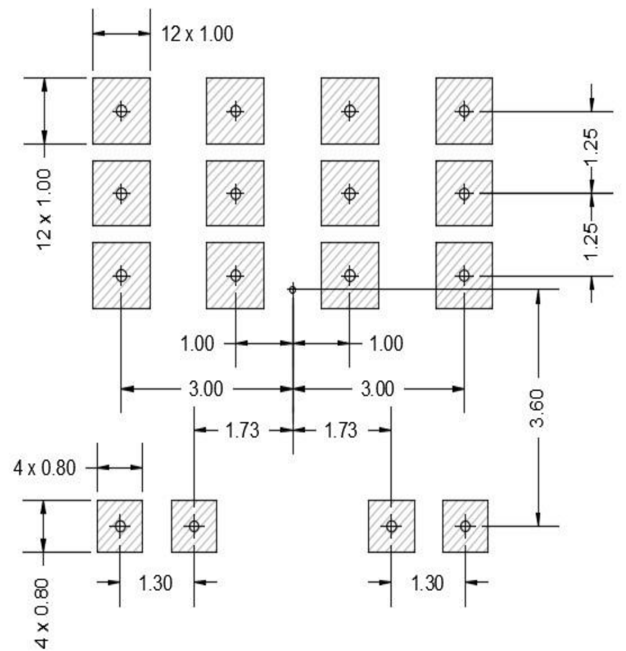


Figure 15. Recommended solder paste stencil (mm) for PQFN88 (source tab)

© 2024 Renesas Electronics

3.0 PQFN56: JSG Package description

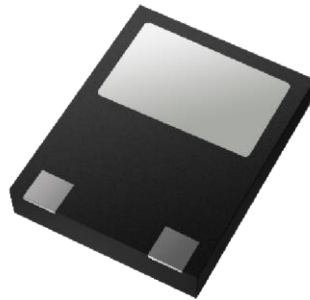


Figure 16. TP65HxxxJSG – common source package configuration (bottom view)

3.1 Printed circuit board

PCB footprint

As with any PQFN package, it is important the exposed solder pads are designed to assure optimum thermal and electrical characteristics. This exposed pad must be connected to the circuit board with solder. Vias can be used as thermal paths between different layers of the circuit board. Copper dimension and thickness will depend on the capability of the board manufacturer.

The circuit board footprint in Figures 17 demonstrate good self-alignment during the reflow soldering process. Please note that this recommendation on offers dimensions for the solder mask opening.

© 2024 Renesas Electronics

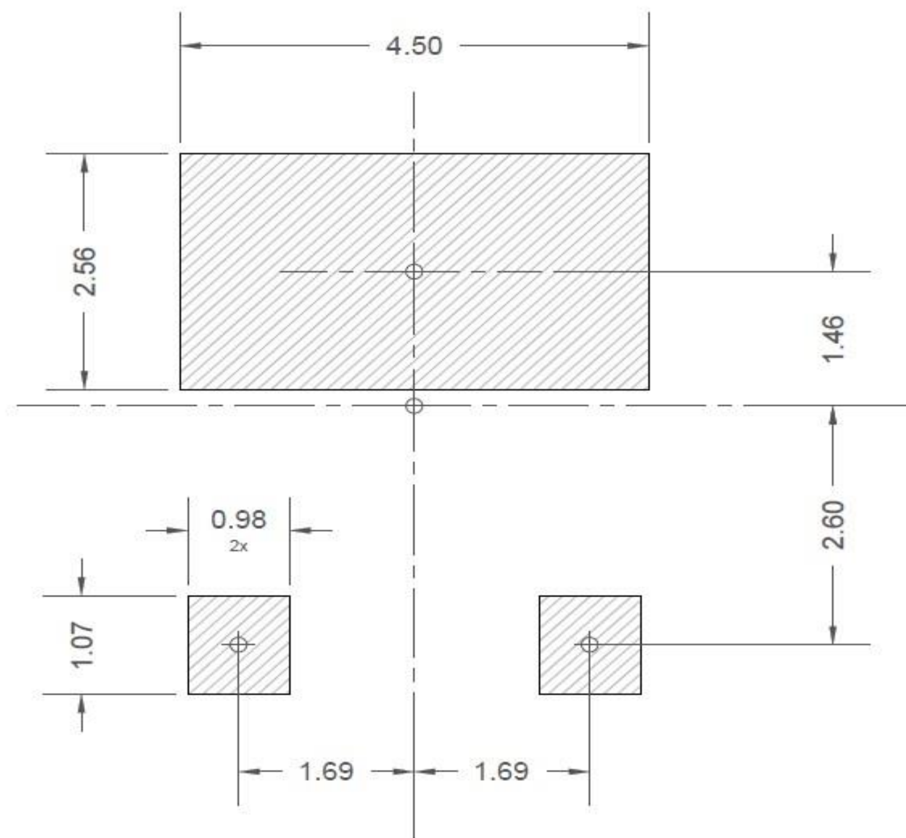


Figure 17. Recommended solder mask opening (mm)
for PQFN56 (source tab)

3.2 Solder Stencil

A stainless steel stencil and laser cut is preferred. Another factor in achieving uniform and high solder paste transfer is the squeegee blade angle and speed. All components on the circuit board must be considered when choosing the stencil thickness.
The stencil thickness will range between 100μm to 150μm (4 mils to 6 mils). Stencil aperture is given in Figures 18.

Renesas's internal investigation concluded that the best results were achieved with the PCB having vias in the drain or source pad and a stencil thickness of 127μm (5 mils).

© 2024 Renesas Electronics

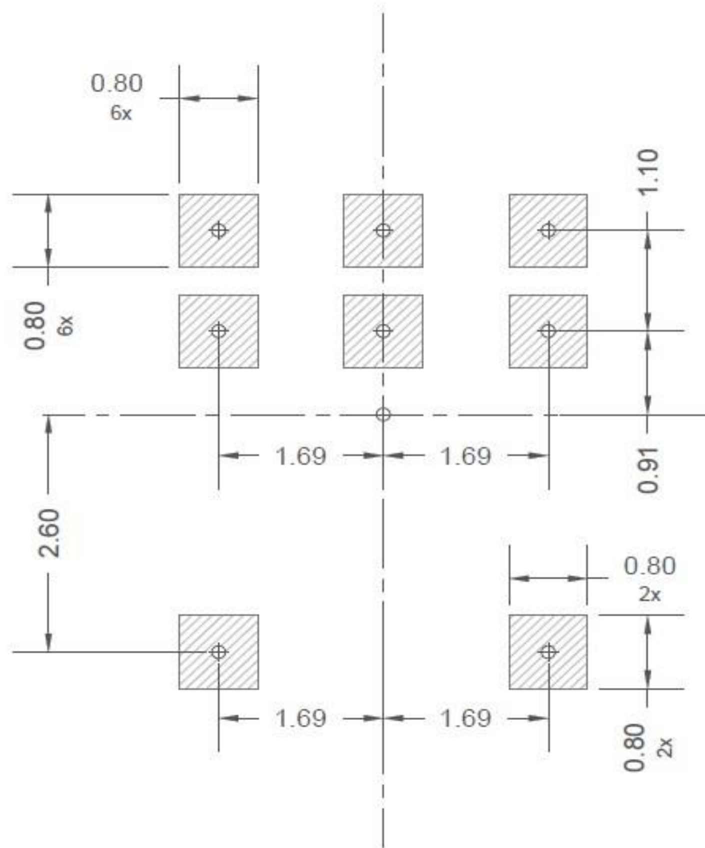


Figure 18. Recommended solder paste stencil (mm)
for PQFN56 (source tab)

4.0 TO-263 Package Description

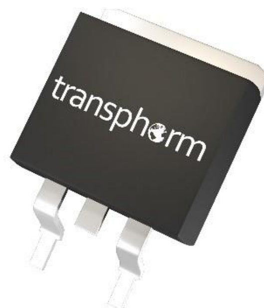


Figure 19. TP65HxxxJSG – common source package
configuration (top view)

© 2024 Renesas Electronics

4.1 Printed circuit board

PCB footprint

As with any TO-263 package, it is important the exposed solder pads are designed to assure optimum thermal and electrical characteristics. This exposed pad must be connected to the circuit board with solder. Vias can be used as thermal paths between different layers of the circuit board. Copper dimension and thickness will depend on the capability of the board manufacturer.

The circuit board footprint in Figures 20 demonstrate good self-alignment during the reflow soldering process. Please note that this recommendation on offers dimensions for the solder mask opening.

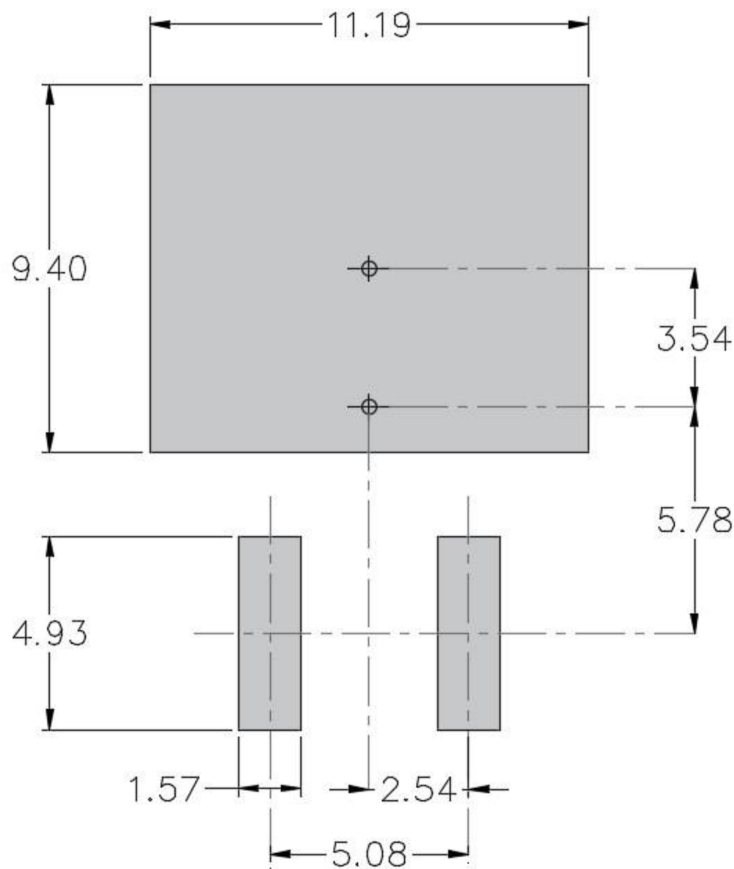


Figure 20. Recommended solder mask opening (mm)

© 2024 Renesas Electronics

4.2 Solder Stencil

A stainless steel stencil and laser cut is preferred. Another factor in achieving uniform and high solder paste transfer is the squeegee blade angle and speed. All components on the circuit board must be considered when choosing the stencil thickness.
The stencil thickness will range between 100μm to 150μm (4 mils to 6 mils). Stencil aperture is given in Figures 21.

Renesas's internal investigation concluded that the best results were achieved with the PCB having vias in the drain or source pad and a stencil thickness of 127μm (5 mils).

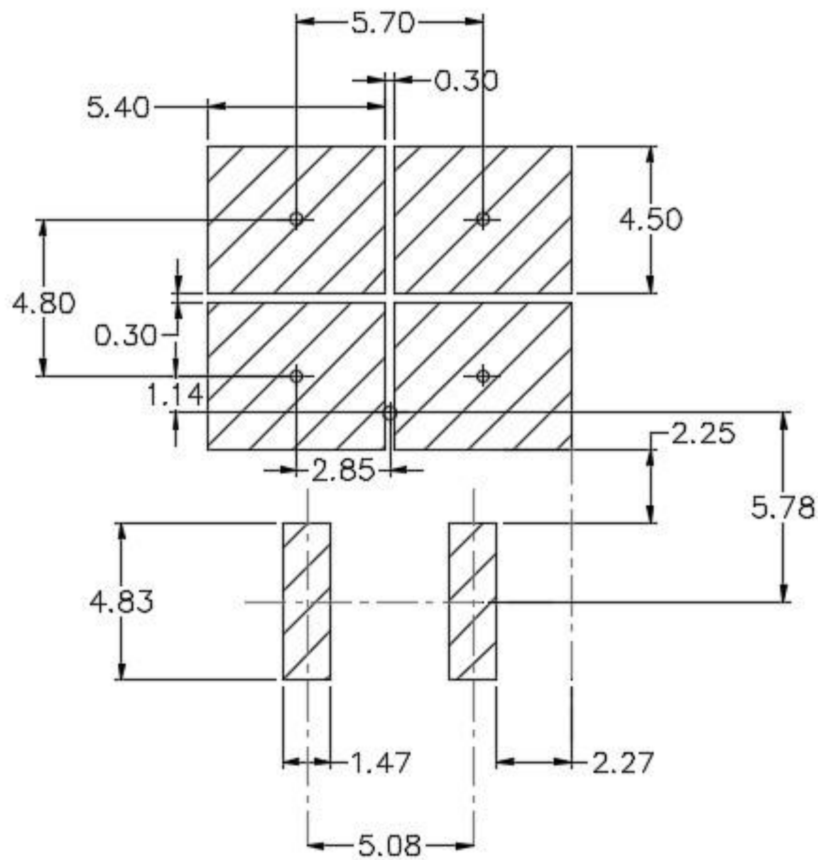


Figure 21. Recommended solder paste stencil (mm)

© 2024 Renesas Electronics

Disclaimer

Please note that these are recommendations only and are to be used as a guideline. PCB size, layer count, copper thickness, and component count will need to be considered.

For this application note, a two-layer printed circuit board with 1oz copper was used.

Electrostatic discharge (ESD)

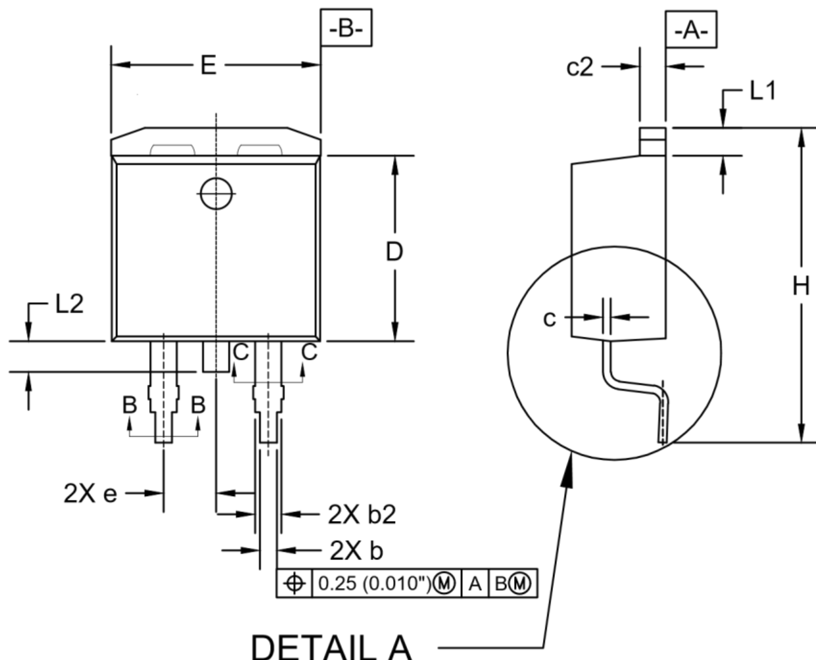
Devices are ESD-sensitive. Use proper precaution.

Moisture sensitivity level

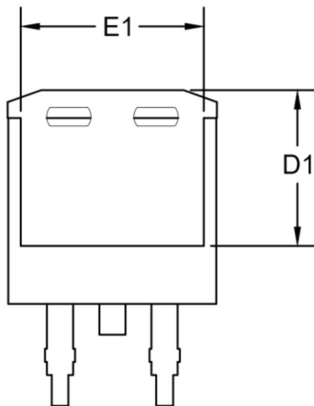
Renesas's PQFN88, PQFN56, and TO-263 are classified to MSL Level 3 per the Joint Industry Standard "Moisture/Reflow Sensitivity Classification Nonhermetic Surface Mount Devices" IPC/JEDEC J-STD-20.

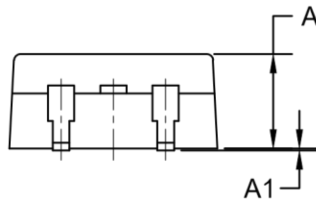
Handling, packing, shipping and use of Renesas's PQFN should conform to the Joint Industry Standard "Handling, Packing, Shipping and Use of Moisture, Reflow, and Process Sensitive Devices" IPC/JEDEC J-STD-033

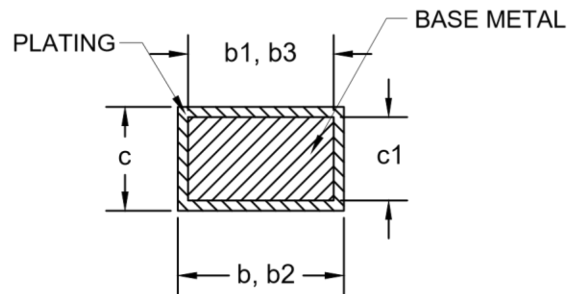
© 2024 Renesas Electronics



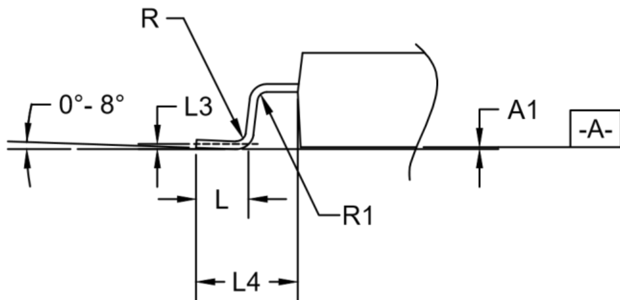
SYMBOL	MILLIMETERS			INCHES		
	MIN	NOM	MAX	MIN	NOM	MAX
A	4.06	4.57	4.83	0.160	0.180	0.190
A1	0.00	0.10	0.26	0.000	0.004	0.010
b	0.50	0.81	1.00	0.020	0.032	0.039
b1	0.50	0.53	0.89	0.020	0.021	0.035
b2	1.14	1.27	1.78	0.045	0.050	0.070
b3	1.14	-	1.73	0.045	-	0.068
c	0.38	0.38	0.74	0.015	0.015	0.029
c1	0.38	0.30	0.59	0.015	0.012	0.023
c2	1.14	1.27	1.66	0.045	0.050	0.065
D	8.38	9.00	9.66	0.330	0.355	0.380
D1	6.85	7.55	-	0.270	0.297	-
E	9.65	10.16	10.67	0.380	0.400	0.420
E1	6.22	8.89	-	0.245	0.400	-
e	2.54 BSC			0.100 BSC		
H	14.60	15.28	15.88	0.575	0.602	0.625
L	1.77	2.54	2.80	0.070	0.100	0.110
L1	-	1.35	1.68	-	0.053	0.066
L2	-	1.50	1.78	-	0.059	0.070
L3	0.254 BSC			0.010 BSC		
L4	4.78	4.93	5.29	0.188	0.194	0.208
R	0.46 TYP			0.018 TYP		
R1	0.46 TYP			0.018 TYP		
Θ	0°	2°	8°	0°	2°	8°







SECTION A-A



DETAIL A

NOTES:

1. DIMENSION D and E DOES NOT INCLUDE MOLD FLASH.
2. DIMENSION b2 DOES NOT INCLUDE PROTRUSIONS.
3. OUTLINE CONFORMS TO JEDEC TO-263F-AB.

TO-263 2L (D2Pak)

transphorm

SCALE: 1:1	SHEET 1/1	DRAWING NO. Eng000509	VER. 1