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 .......................................................................................................5
      Reflow soldering ...............................................................................................5

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 ..................................................................................................13

4.0 TO-263 Package Description..............................................................................14
  4.1 Printed circuit board..........................................................................................14
      PCB footprint .....................................................................................................14
  4.2 Solder Stencil ..................................................................................................16

Disclaimer ...............................................................................................................17

Electrostatic discharge (ESD).................................................................................17

Moisture sensitivity level ........................................................................................17
1.0 PQFN88: LS and LD Package Description

Transphorm’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

![Figure 1. TPH3206LD – common drain package configuration (bottom view)](image1)

![Figure 2. TPH3206LS – common source package configuration (bottom view)](image2)
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 offers dimensions for the solder mask opening.

![Figure 3. Recommended solder mask opening (mm) for PQFN88 (drain tab)](image1)

![Figure 4. Recommended solder mask opening (mm) for PQFN88 (source tab)](image2)
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.

Transphorm’s internal evaluation has demonstrated that Electroless Ni / Emersion 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.

Transphorm’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).
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.

Transphorm 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

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Pd-free alloy (SAC 305)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preheating temperature</td>
<td>170 °C</td>
</tr>
<tr>
<td>Preheating time</td>
<td>50 seconds</td>
</tr>
<tr>
<td>Soaking temperature</td>
<td>200 °C</td>
</tr>
<tr>
<td>Soaking time</td>
<td>75 seconds</td>
</tr>
<tr>
<td>Peak temperature (liquidus)</td>
<td>245 °C</td>
</tr>
<tr>
<td>Peak time (liquidus)</td>
<td>75 seconds</td>
</tr>
<tr>
<td>Vacuum pressure (liquidus)</td>
<td>125 mbar</td>
</tr>
<tr>
<td>Vacuum time</td>
<td>25 seconds</td>
</tr>
</tbody>
</table>

Table 1. Sample parameters for Vapor Phase Reflow
Figure 8. Vapor Phase Reflow x-ray

Figure 9. Standard Reflow x-ray
2.0  PQFN88: LSG and LDG Package description

![Figure 10. TPH320xLDG – common drain package configuration (bottom view)](image)

![Figure 11. TPH320xLSG – common source package configuration (bottom view)](image)

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

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

Transphorm’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)](image1)

![Figure 15. Recommended solder paste stencil (mm) for PQFN88 (source tab)](image2)
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.
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.

Transphorm’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 18. Recommended solder paste stencil (mm) for PQFN56 (source tab)](image-url)

Figure 18. Recommended solder paste stencil (mm) for PQFN56 (source tab)
4.0 TO-263 Package Description

![Image of TP65HxxxJSG package](image)

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

4.1 Printed circuit board

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)
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.

Transphorm’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)
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
Transphorm’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 Transphorm'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
2. DIMENSION b2 DOES NOT INCLUDE PROTRUSIONS.

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.

<table>
<thead>
<tr>
<th>SYMBOL</th>
<th>MILLIMETERS</th>
<th>INCHES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>MIN</td>
<td>NOM</td>
</tr>
<tr>
<td>A</td>
<td>4.06</td>
<td>4.57</td>
</tr>
<tr>
<td>A1</td>
<td>0.00</td>
<td>0.10</td>
</tr>
<tr>
<td>b</td>
<td>0.50</td>
<td>0.81</td>
</tr>
<tr>
<td>b1</td>
<td>0.50</td>
<td>0.53</td>
</tr>
<tr>
<td>b2</td>
<td>1.14</td>
<td>1.27</td>
</tr>
<tr>
<td>b3</td>
<td>1.14</td>
<td>-</td>
</tr>
<tr>
<td>c</td>
<td>0.38</td>
<td>0.38</td>
</tr>
<tr>
<td>c1</td>
<td>0.38</td>
<td>0.30</td>
</tr>
<tr>
<td>c2</td>
<td>1.14</td>
<td>1.27</td>
</tr>
<tr>
<td>D</td>
<td>8.38</td>
<td>9.00</td>
</tr>
<tr>
<td>D1</td>
<td>6.85</td>
<td>7.55</td>
</tr>
<tr>
<td>E</td>
<td>9.65</td>
<td>10.16</td>
</tr>
<tr>
<td>E1</td>
<td>6.22</td>
<td>8.89</td>
</tr>
<tr>
<td>e</td>
<td>2.54 BSC</td>
<td>0.100 BSC</td>
</tr>
<tr>
<td>H</td>
<td>14.60</td>
<td>15.28</td>
</tr>
<tr>
<td>L</td>
<td>1.77</td>
<td>2.54</td>
</tr>
<tr>
<td>L1</td>
<td>-</td>
<td>1.35</td>
</tr>
<tr>
<td>L2</td>
<td>-</td>
<td>1.50</td>
</tr>
<tr>
<td>L3</td>
<td>0.254 BSC</td>
<td>0.010 BSC</td>
</tr>
<tr>
<td>L4</td>
<td>4.78</td>
<td>4.93</td>
</tr>
<tr>
<td>R</td>
<td>0.46 TYP</td>
<td>0.018 TYP</td>
</tr>
<tr>
<td>R1</td>
<td>0.46 TYP</td>
<td>0.018 TYP</td>
</tr>
<tr>
<td>θ</td>
<td>0°</td>
<td>2°</td>
</tr>
</tbody>
</table>

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.