transphorm

Application Note:

TPH3206LD and TPH3206LS Lead Free 2nd Level Soldering Recommendations for Vapor Phase Reflow

1. Package Description

Trasphorm'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 printed circuit board. It provides high lateral electrical isolation and excellent heat dissipation in applications similar to a D2Pak but in a thinner form factor.

Features

- Low profile & lead inductance
- RoHS Compliant

Figure 1 and Figure 2 show a photo of an 8x8 PQFN.

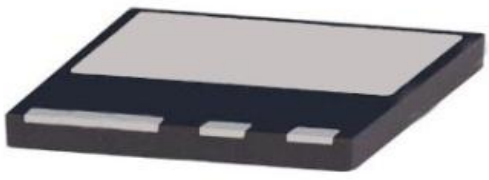

Figure 1: **TPH3206LD**
Bottom View

Figure 2: **TPH3206LS**
Bottom View

2. Printed Circuit Board

PCB Footprint

Like it is important for any PQFN package, the exposed solder pads should be 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 manufacture.

The following circuit board footprint design has demonstrated good self-alignment during reflow soldering process. Please note that this recommendation only offers dimensions for the solder mask opening. See Figure 3 & Figure 4.

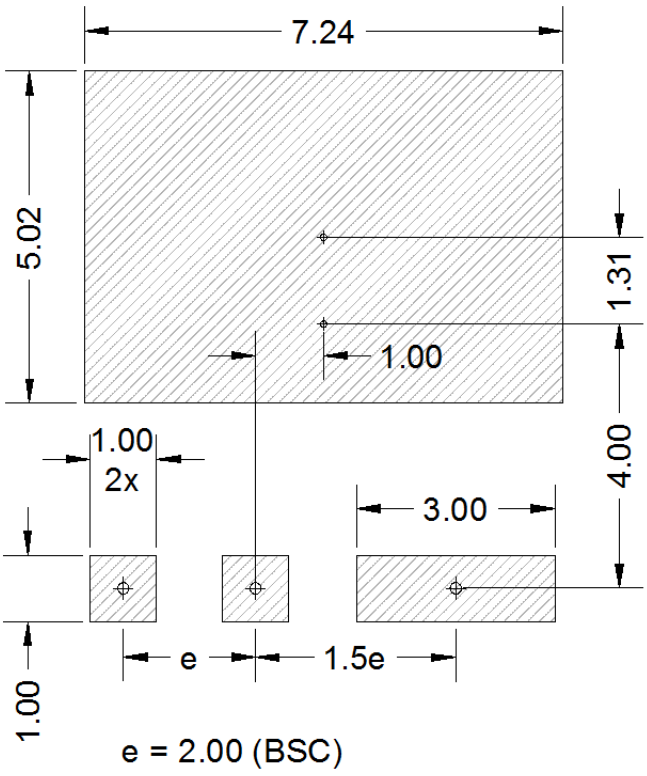

Figure 3: **TPH3206LD**
Recommended Solder Mask Opening [mm]

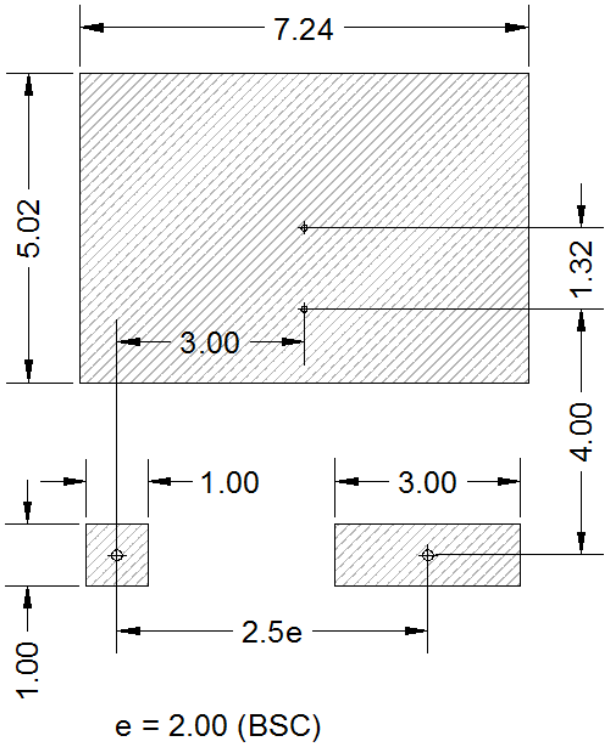

Figure 4: **TPH3206LS**
Recommended Solder Mask Opening [mm]

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 / Emmersion Au (ENIG) surface has the best solder wetting for Pb free solder.

3. 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 to achieve 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 Figure 5 and Figure 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 127um (5mils).

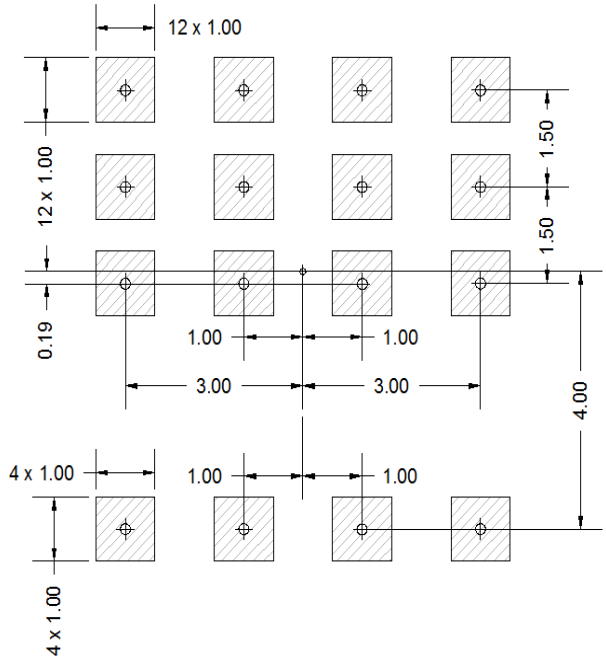

Figure 5: **TPH3206LD**
Recommended dimension for Solder Paste Stencil [mm]

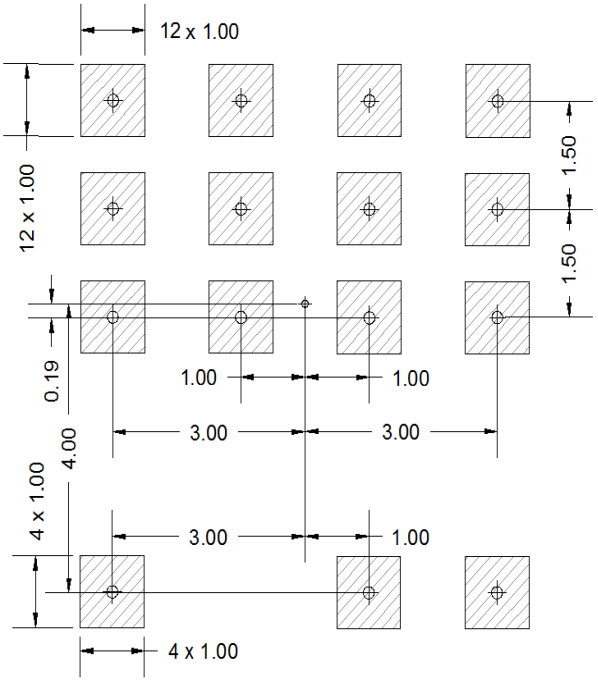

Figure 6: **TPH3206LS**
Recommended dimension for Solder Paste Stencil [mm]

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 obtain satisfactory 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 reflow profile must meet all components requirements. It is important to follow the solder paste manufacture profile specifications.

The recommended process is "Vapor Phase Reflow". See Figure 7 for an example profile and Table 1 for Parameters. This process has improved solder joint quality. The recommended solder void criteria for "Vapor Phase" are 5% on a single void and 10% total voids. Figure 8 is the X-ray for "Vapor Phase" and Figure 9 for "Standard Reflow"

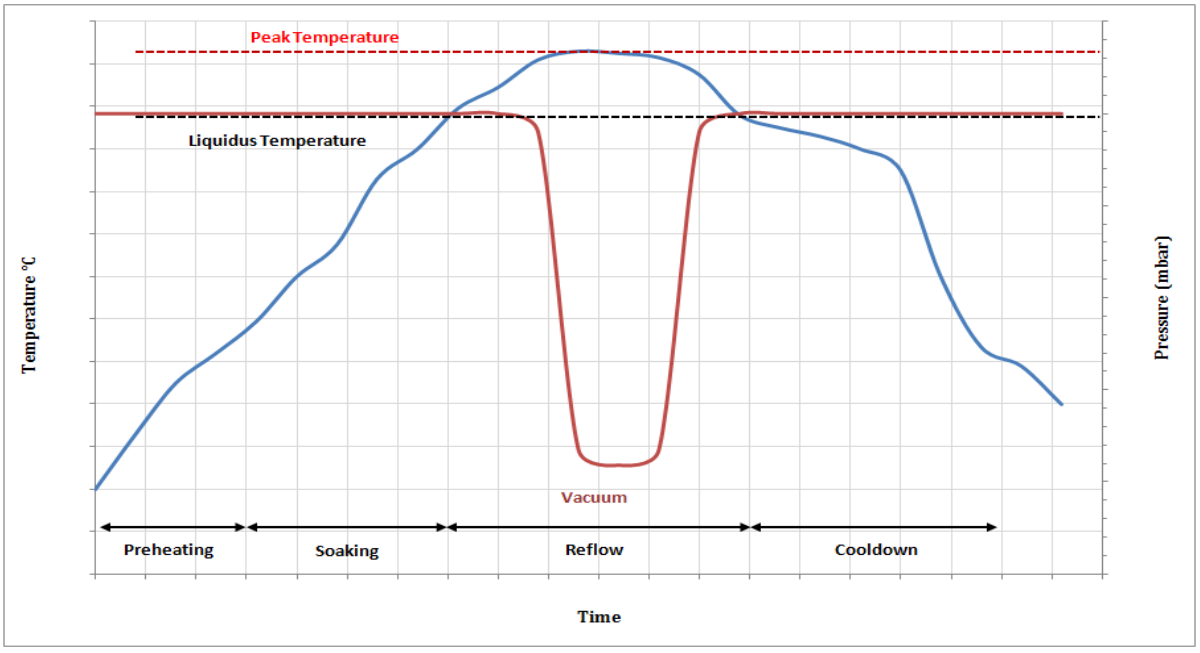

Figure 7: Example of Profile for Vapor Phase Reflow

Parameters	Pd-Free alloy (SAC 305)
Preheating Temperature	170°C
Preheating Time	50 seconds
Soaking Temperture	200°C
Soaking Time	75 seconds
Peak Temperature (liquidus)	245°C
Peak Time (liquidus)	75 seconds
Vacuum Pressure (liquidus)	125mbar
Vacuum Time	25 seconds

Table 1: Example of Parameters for Vapor Phase Reflow

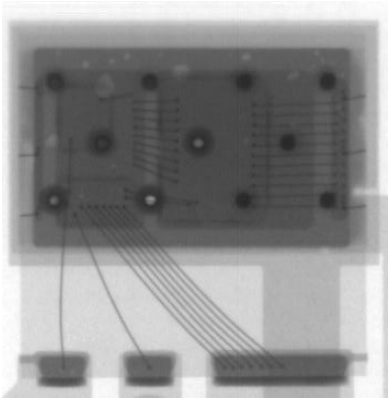

Figure8: Vapor Phase Reflow X-ray

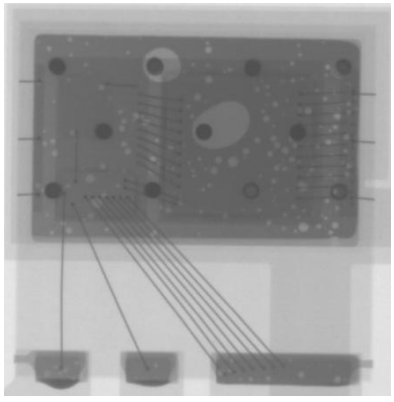

Figure 9: Standard Reflow X-ray

Disclaimer

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

For this application note, a 2 layer Printed Circuit Board with 1oz copper was used.

Electrostatic Discharge (ESD)

Devices are ESD sensitive. Use proper precaution.

Moisture Sensitivity Level

Transphorm 8x8 PQFN meet all MSL3 requirements. These devices are moisture sensitive and should be handled within proper MSL 3 guidelines to avoid damages from moisture absorption and exposure to solder reflow temperature that can result in yield and reliability degradation.

Reference

Customer may refer to the following IPC/JEDEC standards for more details.

- J-STD-020D.1 Moisture/Reflow Sensitivity Classification for Non-hermetic Solid State Surface Mount Devices