SOLDERING & REWORK GUIDELINES FOR MOLDED QFN PACKAGES

TN-002

02/13/2019

APPLICABLE TO TAGORE'S QFN PACKAGES

S/N	PACKAGE NO.	DESCRIPTION	NOTE
1	QFN-3X3-16-0.5	16 PIN 3x3mm QFN Pitch 0.5mm	**Pb-Free RoHS Compliance:** RoHS Directive 2011/65/EU – Annex II and Amending Directive 2015/863 **REACH Compliance:** REACH – Jan. 12, 2017 **Halogen Free Compliance:** IEC 61249-2-21:2003 Package Type: Molded Pin Finish: Tin (Sn) MSL: 1
2	QFN-4X4-32-0.4	32 PIN 4x4mm QFN Pitch 0.4mm	
3	QFN-5X5-32-0.5	32 PIN 5x5mm QFN Pitch 0.5mm	
4	QFN-5X7-22-0.5	22 PIN 5x7mm QFN Pitch 0.5mm	
5	QFN-6X3-32-0.4	32 PIN 6x3mm QFN Pitch 0.4mm	
6	QFN-6X6-48-0.4	48 PIN 6x6mm QFN Pitch 0.4mm	
7	QFN-8X10-30-0.5	30 PIN 8x10mm QFN Pitch 0.5mm	
8	Other QFN Package Types Provided by Tagore		

© 2018 Tagore Technology Inc. All rights reserved. Tagore, the Tagore words and logos are trademarks of Tagore Technology Inc. in the US and/or other countries. Other marks and brands may be claimed as the property of others. Tagore warrants performance of its Gallium Nitride (GaN) devices and other semiconductor products to current specifications in accordance with Tagore's standard warranty, but reserves the right to make changes to any products and services at any time without notice. Tagore assumes responsibility or liability arising out of the application or use of any information, product, or service described herein except as expressly agreed to in writing by Tagore. Tagore's customers are advised to obtain the latest version of device specifications before relying on any published information and before placing orders for products or services. Tagore makes no representations or warranties with respect to the accuracy or completeness of the contents of this document and also reserves the right to make changes at any time without notice. Tagore does not make any commitment to update the information contained herein.

1.0 Purpose

The purpose of this document is to provide recommended soldering and rework guidelines for Tagore's molded Quad Flat No-leads (QFN) packages.

2.0 Terms and Definitions

Ag	Silver
Cu	Copper
IEC	International Electrotechnical Commission
IPC	Institute for Printed Circuits
Pb	Lead
PCB	Printed Circuit Board
PTH	Plated Through Hole
QFN	Quad Flat No-leads
REACH	Registration, Evaluation, Authorization and Restriction of Chemicals
RoHS	Restriction of Hazardous Substances
SMD	Solder Mask Defined
Sn	Tin

3.0 Disclaimer

This document only states general soldering and rework guidelines for Tagore's molded QFN packages. Tagore does not take legal liability and responsibility for the information in this document. Please refer to the IPC website and/or solder manufacturers' recommendations for more specific information.

4.0 Solder Joint Requirement

(1) In order to achieve optimum and reliable solder joints on the perimeter pads, there should be about 50µm to 75µm (2 to 3 mils) standoff height (thickness) and a good side fillet on the outside. A joint with good stand-off height but no or low fillet will have reduced life but may meet the application requirement.
(2) To obtain 50µm to 75µm thick solder joints, it is recommended that the solder paste coverage to be at least 50% for plugged (Including filled) vias and 75% for non-plugged vias.
(3) The solder can wet down the via walls for some types of vias, QFN devices may have to be assembled on the top side (or final pass) if the solder protrusion cannot be avoided, as the protruded solder will impede acceptable solder paste printing on the other side of the PCB.

5.0 Solder Selection and Soldering Profile
5.1 Solder Paste Selection

For Tagore's QFN devices, it is recommended to use low-residue, no-clean, Type 3 or Type 4 per J-STD-005 Sn63Pb37 or Pb-free Sn96.5Ag3.0Cu0.5 solder paste for soldering. Other types of solder paste might also be used based on customers' experience.

5.2 Soldering Profile
5.2.1 Sn63Pb37 Solder

The recommended soldering profile for generic Sn63Pb37 solder paste is shown in Figure 5.1, ensure that solder solidifies at exit of last heated zone/area to avoid disturbed joint defects:

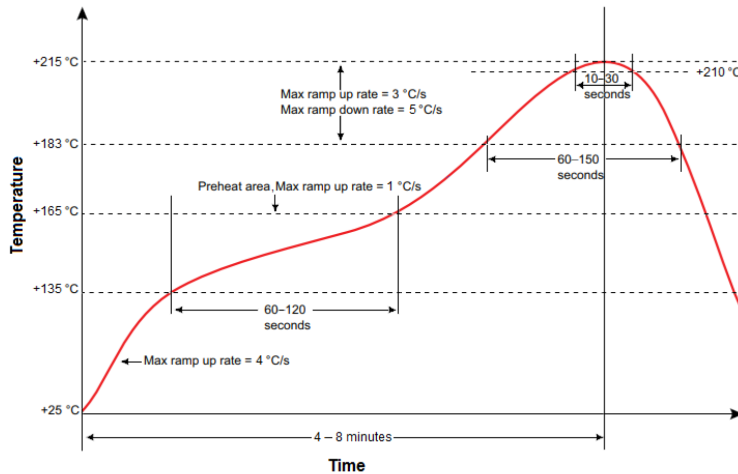

Figure 5.1 Recommended Soldering Profile for Sn63Pb37 Solder

5.2.2 Sn96.5Ag3.0Cu0.5 Solder

The recommended soldering profile for generic Pb-free Sn96.5Ag3.0Cu0.5 solder paste is shown in Figure 5.2, ensure that solder solidifies at exit of last heated zone/area to avoid disturbed joint defects.

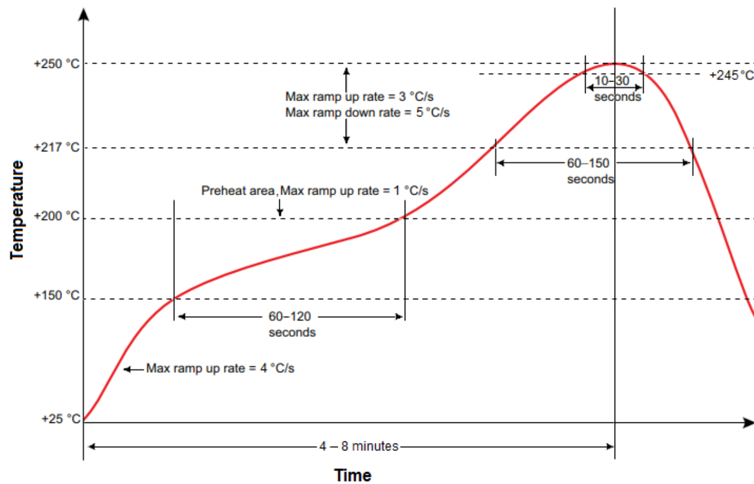

Figure 5.2 Recommended Soldering Profile for Sn63Pb37 Solder

In any case, the soldering temperature should not exceed the maximum temperature the package is qualified for according to the moisture sensitivity level.

After reflow, the mounted package should be inspected in the transmission x-ray for the presence of voids, solder balls, or other defects. Cross-sectioning may also be required to determine the fillet shape, size and the joint standoff height during process development.

6.0 Rework Guidelines

Since solder joints are not fully exposed in case of QFNs, any retouch is limited to side fillets. If defects exist underneath the package, the whole package has to be removed.

Prior to any rework, it is strongly recommended that the PCB assembly be baked for at least 4 hours at +125°C to remove any residual moisture from the assembly.

Because of the product dependent complexities, automated rework system is recommended to be used for rework. For manual rework, following procedures can serve as a starting point for the development of a successful rework process.

(1) Remove component
It is recommended that the PCB should be heated from the bottom side using convective heaters if possible, and hot gas or air should be used on the top side of the component. Special nozzles should be used to direct the heating in the component area and the heating of adjacent components should be minimized. Excessive airflow should also be avoided since this may cause the package to skew. Air velocity of 15-20 liters per minute is a good starting point.

Once the joints have reflowed, the vacuum lift-off should be automatically engaged during the transition from the reflow to cool down. Because of their small size the vacuum pressure should be kept below 15 inches of Hg. This will allow the component not to be lifted off if all joints have not been reflowed and avoid pad damage.

(2) Clean site
After the component has been removed, the site needs to be cleaned properly. It is best to use a combination of a blade-style conductive tool and a de-soldering braid. The width of the blade should be matched to the maximum width of the footprint and the blade temperature should be low enough not to cause any damage to the circuit board.

Once the residual solder has been removed, the lands should be cleaned with a solvent. The solvent is usually specific to the type of paste used in the original assembly and the paste manufacturer's recommendations should be followed.

(3) Print solder paste
It is recommended to use a miniature stencil specific to the component to print solder paste, the printing can be done either on the PCB lands or on the QFN package side.

(4) Place component
A split-beam optical system should be used if possible to align the component on the board, this will form an image of leads overlaid on the mating footprint and aid in proper alignment. Manual operation can also be done with the help of an optical microscope at 50 to 100X magnification.

(5) Attach component
The reflow profile developed during original attachment should be used to attach the new component. Since all reflow profile parameters have already been optimized, using the same profile will eliminate the need for thermocouple feedback and will reduce operator dependencies.