Tagore Technology, Inc.

SOLDERING & REWORK GUIDELINES FOR AIR CAVITY PACKAGES

TN-003

03/20/2018

APPLICABLE TAGORE'S PACKAGES

S/N	PACKAGE NO.	DESCRIPTION	NOTE
1	QFN-5X5x1.25-32-0.5	32 PIN 5x5x1.25mm Air Cavity QFN Package, Pitch 0.5mm	**Pb-Free RoHS Compliance** Package Type: Air Cavity Pin Finish: Tin (Sn) MSL: 1
2	Other Air Cavity Packages		

1.0 Purpose

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

2.0 Terms and Definitions

ACP	Air Cavity Package
Ag	Silver
Cu	Copper
IPC	Institute for Printed Circuits
Pb	Lead
PCB	Printed Circuit Board
QFN	Quad Flat No-leads
RoHS	Restriction of Hazardous Substances
Sn	Tin

3.0 Disclaimer

This document only states general soldering and rework guidelines for Tagore's Air Cavity 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 Air Cavity Package (ACP) Description

As shown in Figure 4.1, ACP uses a pre-molded open cavity assembly formed by lead frame (or substrate) and plastic epoxy as the foundation of the package, a lid is then attached to the assembly wall using adhesive to seal the assembly after die attach and other internal procedures.

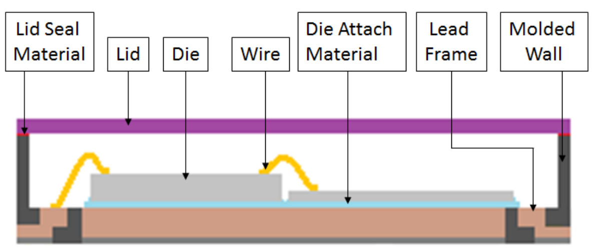

Figure 4.1 Cross Section of A Typical ACP

ACP package is appropriate to a number of applications. Many Micro-Electro-Mechanical System (MEMS) devices cannot function if embedded in epoxy, so air cavity packaging is optimum. RF and microwave semiconductors perform better in an air cavity rather than embedded in a lossy solid dielectric. Free space optical and imaging devices cannot be surrounded with an opaque material that attenuates or diffracts light. Many sensors need to sense their environment to function as opposed to being isolated in a solid block of material.

5.0 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 along 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.

6.0 Solder Selection and Soldering Guidelines

6.1 Solder Paste Selection

For Tagore's Air Cavity QFN devices, it is highly 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 to attach to PCBs. Other types of solder pastes might also be used based on customers' experience.

It is strongly recommended that the PCBs with devices in Air Cavity Packages are not washed to remove flux especially not washed with water after soldering process, since solvent residue might get into the cavity and is left inside resulting in reliability problems.

6.2 Flux Selection

During manual soldering or rework process, if flux has to be used to achieve better solder attachment, no-clean flux is recommended to avoid subsequent cleaning procedure.

6.3 Soldering Profile

6.3.1 Sn63Pb37 Solder

The recommended soldering profile for generic Sn63Pb37 solder paste is as follow:

Option 1: Straight ramp profile for low density assemblies
(1) Ramp @ 60~120°C/min to 220±15°C peak
(2) Soak time above 183°C is 30~60sec.
(3) Ramp down to room temperature @ 90~120°C/min.
(4) Ensure that solder solidifies at exit of last heated zone to avoid disturbed joint defects

Option 2: Soak profile for high density assemblies
(1) Ramp @ 60~120°C/min to 150±15°C
(2) Dwell @ 150±15°C for 60~120 sec.
(3) Ramp @ 60~120°C/min to 220±15°, time over 183°C for 30~60 sec.
(4) Ramp down to room temperature @ 90~120°C/min.
(5) Ensure that solder solidifies at exit of last heated zone to avoid disturbed joint defects

6.3.2 Sn96.5Ag3.0Cu0.5 Solder

The recommended soldering profile for generic Sn96.5Ag3.0Cu0.5 solder paste is as follow:

Option 1: Straight ramp profile for low density assemblies
(1) Ramp @ 60~120°C/min to 245±5°C peak
(2) Soak time above 220°C is 30~60sec.
(3) Ramp down to room temperature @ 90~120°C/min.
(4) Ensure that solder solidifies at exit of last heated zone to avoid disturbed joint defects

Option 2: Soak profile for high density assemblies

(1) Ramp @ 60~120°C/min to 165±15°C
(2) Dwell @ 165±15°C for 60~120 sec.
(3) Ramp @ 60~120°C/min to 245±5°, time over 220°C for 30~60 sec.
(4) Ramp down to room temperature @ 90~120°C/min.
(5) Ensure that solder solidifies at exit of last heated zone to avoid disturbed joint defects

Although Tagore's Air Cavity QFN packages can withstand +260°C reflow temperature, it is recommended that the peak soldering temperature should not exceed +250°C.

After solder 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.

7.0 Manual Soldering and Rework Guidelines

NOTE: Due to reliability concerns, for Air Cavity Packages, manual soldering and rework can only be used for engineering and/or evaluation purposes, they shall not be adopted for production.

7.1 Manual Soldering

Since the mass of ACP lid is very small, if a heat gun is used during manual soldering process and the heat gun output air temperature is higher than +260°C, it is extremely not recommended to heat the ACP directly from the top as shown in Figure 7.1 without any protection on the ACP. The reason is that the lid and even the ACP wall will be heated up first very fast to above +260°C, the adhesive used to attach the lid will deteriorate quickly resulting in weak adhesion, the air inside the ACP will expend, the lid will fall off as the consequence of these effects.

Figure 7.1 Prohibited Soldering Method

Figure 7.2 Photo of a Typical Preheater
(Such as Hakko's preheater FR-870)

In order to prevent the ACP lid adhesion from being damaged during manual soldering process, following options are recommended:

(1) Use a heat gun to heat from bottom side of the PCB as shown in Figure 7.3, the heat can be transferred from bottom to top of the PCB through vias causing solder reflow without damaging the ACP. The heat gun temperature can be higher (i.e., +300°C for Pb-free solder) if this method is used, but procedures need to be taken to protect surrounding components at the bottom side of the PCB, Kapton tape (or other shield material) can be used to cover the affected components.

(2) Use Kapton tape (or other shield material which can isolate the heat) to cover the ACP lid and wall (four sides) as shown in Figure 7.4, use a preheater similar as shown in Figure 7.2 to heat the PCB from bottom side up to +150°C for more than 60 seconds (customers can adjust the preheat temperature and time depending on the PCB structure and solder paste used), then use a heat gun to heat from top to complete the soldering process. The heat gun exit air temperature can be up to +300°C in this case, but the heating time shall be controlled to be <10 seconds after the solder starts to melt.

(3) If somehow the PCB cannot be heated from the bottom, for instance, there are some sensitive components which cannot withstand high temperature at the bottom of the PCB, use Kapton tape (or other shield material which can isolate the heat) to cover the ACP lid and wall (four sides) as shown in Figure 7.5, then use a heat gun to heat from top to complete the soldering process. The heat gun exit air temperature shall be ≤+250°C, but the heating time can be longer to ensure good solder attachment.

Figure 7.3 Manual Soldering Option 1

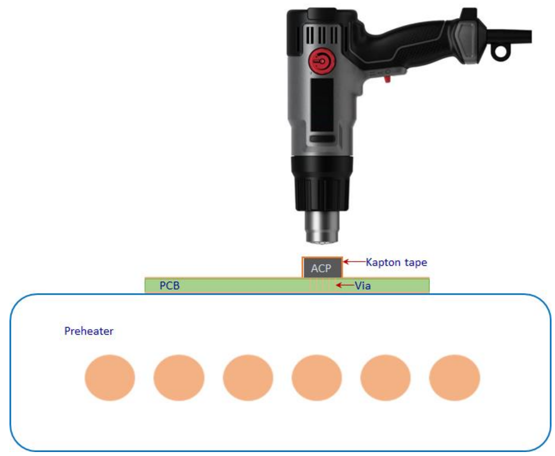

Figure 7.4 Manual Soldering Option 2

Figure 7.5 Manual Soldering Option 3

7.2 Rework

Since solder joints are not fully exposed, 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
Use one of the 3 manual soldering options to remove the ACP from PCB, if the device can be disposed, the shield on ACP is not required.

(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 ACP side.

(4) Place component
Place and align the new component on the board with the help of an optical microscope at 50 to 100X magnification.

(5) Attach component
Use one of the 3 manual soldering options to attach the new component.