By J. Lu, Y. Ding, S. Liu, J. Gong, C. Yue
July 2012

Molded Chip Scale Package Assembly Guidelines

Introduction to Molded Chip Scale Package

A chip scale package (CSP) has direct connection between the package and a printed circuit board (PCB) through solder balls, solder bumps or other metallic bumps. Since no bonding wires and metal leads are utilized in CSP, package parasitics can be minimized to boost the device's performance. As CSP has the smallest form factor in all three dimensions among all package types, it's becoming increasingly more important and popular in portable electronic devices. CSP parts can be mounted face-down onto PCB**s** using surface mount equipment without applying underfill material.

A true chip-scale package is one which has a footprint size that is exactly the same as the semiconductor chip it carries – i.e., the chip size is equal to the package footprint size. Alpha and Omega Semiconductor has developed a family of molded chip scale package (MCSP) devices, which have an additional mechanical protection that traditional CSPs do not have and are still categorized as true chip scale packages.

This application note provides the package structure, PCB design guideline, surface mount assembly guideline and rework procedure for MCSP.

Construction of MCSP

Figure 1 illustrates the top and bottom views of a typical MCSP part offered by Alpha and Omega Semiconductor in order to significantly improve the mechanical integrity of the entire chip scale package family. Figure 2 details the cross-section of the package. A polyimide layer is deposited on top of a passivation layer on the silicon chip. Metal 1 on the silicon can be either aluminum or aluminum copper. The purpose of under bump metal 2 is for solder bumping on the silicon chip. The silicon chip is sandwiched between two pieces of mold compound. The thickness of the solder plates on top of the silicon is the same as that of the mold compound on the same side such that the exposed solder surfaces flush with the surface of their mold compound counterpart. Solder plate material is Pb-free SnAgCu305. The thickness of the mold compound on either side of the silicon is 0.1mm and the total thickness of the package including the silicon is 0.3mm. The only portion of the silicon that is visible externally is the middle side walls with a thickness of only 0.1mm. This construction with mechanical protection drastically reduces silicon chipping and cracking problems that have been encountered in manufacturing, handling and board assembly of traditional CSP.

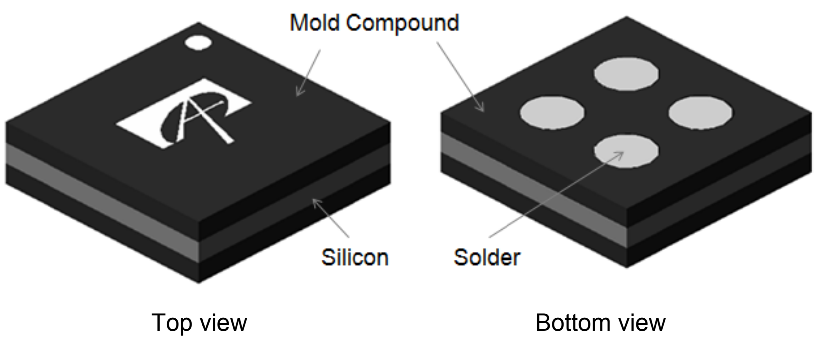

Figure 1. 3D Illustration of a MCSP

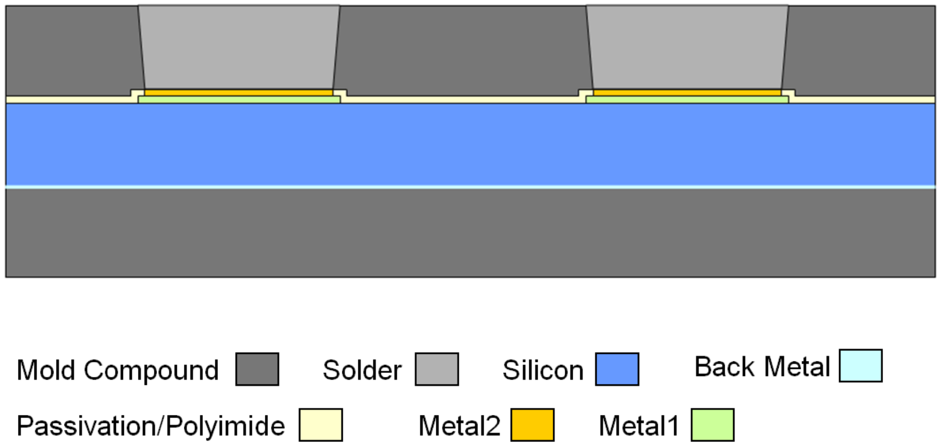

Figure 2. 2D Cross-Section of a Typical MCSP

PCB Materials

Standard FR4 or high-temperature FR4 PCB materials are acceptable for MCSP. PCBs which have higher grade organic substrate with higher glass transition temperatures and a lower coefficient of thermal expansion (CTE) will improve the package and overall system reliability.

Organic surface preservative (OSP) pad finish is recommended for optimum solder joint reliability.

1oz copper on PCB is acceptable; however, increasing copper trace thickness can help to reduce the device junction temperature. Figure 3 demonstrates the temperature difference for a 1x1mm MCSP (4 balls) device simply by increasing the copper layer thickness on a PCB from 1oz to 2oz with a device power loss of 0.8W.

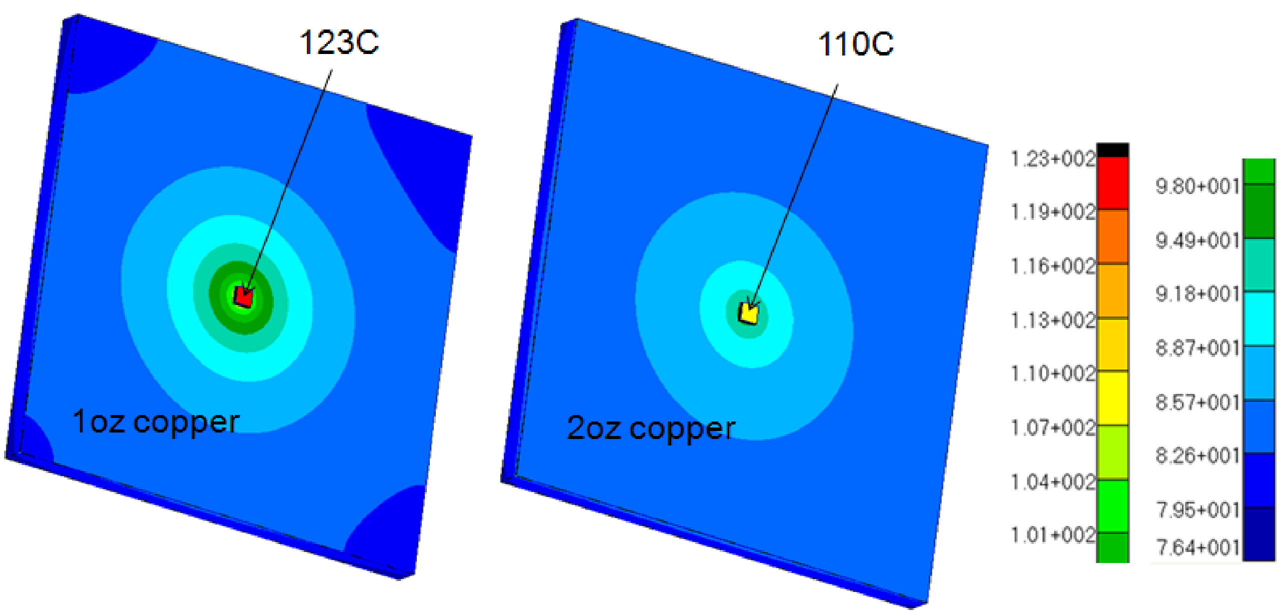

Figure 3. Temperature Difference (in degree C) Due to the Difference in Copper Thickness in PCB (power loss is 0.8W on a 25.4x25.4mm PCB)

PCB Design

Non-solder mask defined (NSMD, see Figure 4) pads are recommended for MCSPs because a chemical copper etching process has much tighter tolerance control – improving the reliability of solder joints, than a less accurate mechanical solder mask printing process. In addition, SMD pattern introduces more stress concentration near solder mask edge where the solder mask meets the solder layer and copper pad.

When using the NSMD pad, the width of the copper traces connecting the circular copper pads for MCSP solder openings should not be more than 60% of the pad diameter. The gap between the solder mask and the copper pad should be 0.05mm. Vias directly under the circular pads should be avoided whenever possible. If this is absolutely unavoidable, completely fill the vias to prevent a weak solder connection between MCSP and PCB. The copper pad size on PCB should be the same as the solder opening on the MCSP.

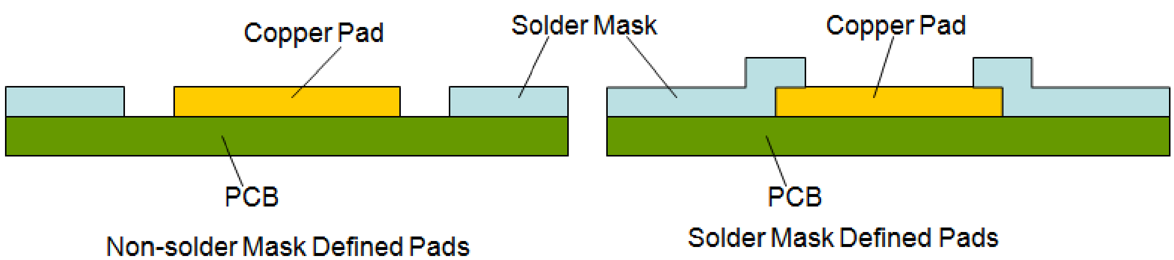

Figure 4. Cross-Section of Non-Solder Mask Defined (NSMD) and Solder Mask Defined (SMD) Land Patterns

The current MCSP family has two rows of solder openings with a pitch option of 0.4mm, 0.5mm or 0.8mm. As an end-user can have access on a PCB to all the solder openings from the outside of the package. It is not recommended that an end-user route copper trace through two adjacent copper pads on PCB considering added trace resistance, parasitic and current delivery capability, although it is possible to do so for MCSP with a pitch of 0.8mm (reference Figure 6).

The copper pad size should match the silicon pad opening size for optimized reliability performance. Figure 5 shows a typical design for copper pad and trace when the copper pad is 0.25mm. In this case the copper trace width adjacent the copper pad is 0.15mm and the solder mask opening diameter is 0.35mm. The design for different copper pad sizes can be adjusted accordingly using the design rule stated in this application note. Figure 6 shows a PCB design for 0.8mm pitch when a copper trace needs to pass through two adjacent copper pads.

As shown in Figures 5 and 6; it is also suggested that a small fillet with a radius of 0.1mm be added to where the copper trace meets the copper pad to minimize stress concentration at both solder and copper layers.

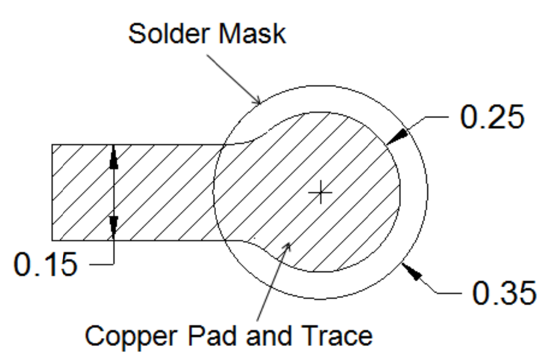

Figure 5. Copper Pad and Solder Mask for MSCPs with 0.4 and 0.5 mm Pitch (in mm)

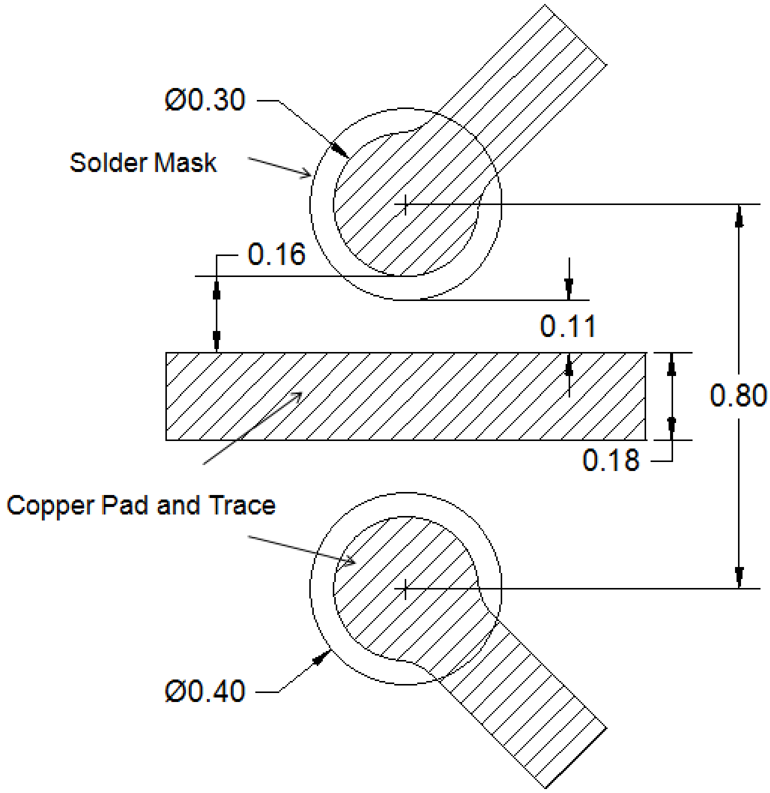

Figure 6. Copper Pad and Solder Mask for MSCP with 0.8 mm Pitch (in mm)

PCB Assembly Process

It is recommended that a nickel-plated stencil of 0.125mm thick by either laser cutting with electropolishing or electroforming be used for solder paste printing on PCB. It is also recommended that apertures be tapered from PCB side of the stencil to print side of the stencil by 5 degrees to improve solder release capability (see Figure 7). Stencil aperture can be 0.25x0.25mm square with small fillets for MCSP of 0.4mm and 0.5mm pitches and 0.30x0.30mm square with small fillets for MCSP of 0.8mm pitch. Local fiducials are required on PCB for MSCP part placement with controlled accuracy.

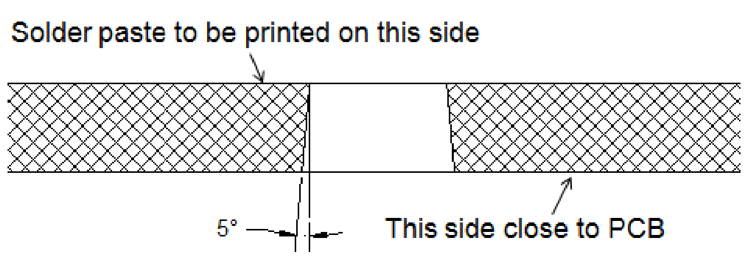

Figure 7. Taper Aperture Design in Stencil

Type 3 or finer Pb-free SAC305 non-clean type solder pastes (comply with ANSI/J−STD−005) are recommended. Solder pastes with water soluble or RMA flux may be used for MCSP as well.

Standard pick-and-place equipment with vision system to locate package silhouette or individual solder openings can be used to place MCSP on PCB. The contact force between pick-up nozzle and MCSP should always be minimized during pick-and-place process, although the MCSP can withstand a minimum uniform compressive force of 10N. It is recommended that the solder opening surface be dipped into the solder paste block such that the solder opening surface sits 0.025mm (1 mil) below the originally printed solder paste surface before reflow. Part placement accuracy should be 0.05mm (2 mils).

Figure 8 lists a typical process flow for mounting MCSP parts onto PCB.

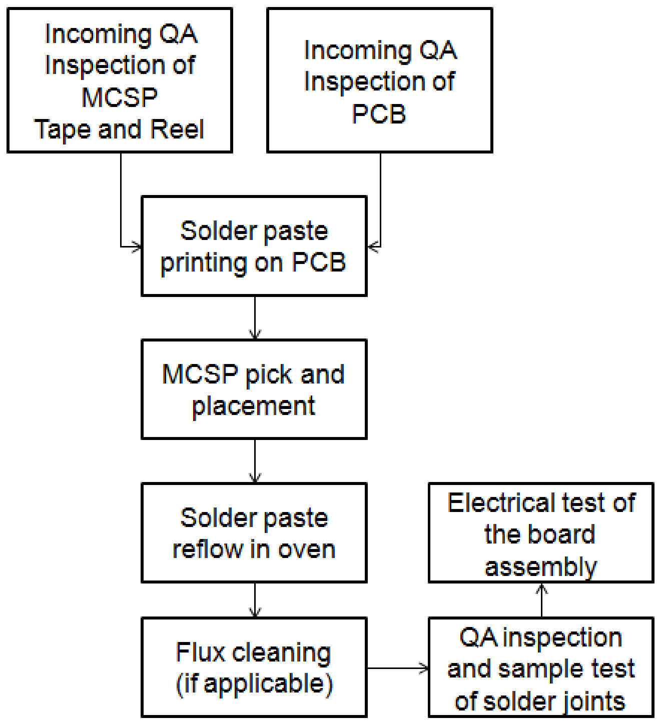

Figure 8. A Typical PCB Assembly Process Flow

Solder Reflow Profile

We recommend the following solder reflow profile (comply with J-STD-020) as shown in Figure 9 which consists of 5 reflow regions: initial heating-1, soaking-2, ramp-up-3, peak reflow-4, and cool-down-5. The initial heating rate shall be controlled within 1-4ºC/sec and temperature at the soaking stage is between 150ºC~200ºC for a period of 60-180 seconds. Ramp-up rate at region 3 shall not exceed 1-4ºC/sec. The peak reflow temperature for SAC solder is controlled between 245ºC and 260ºC and the duration of the peak reflow shall not exceed 10 seconds. Final cooling rate cannot be more than 4ºC/sec.

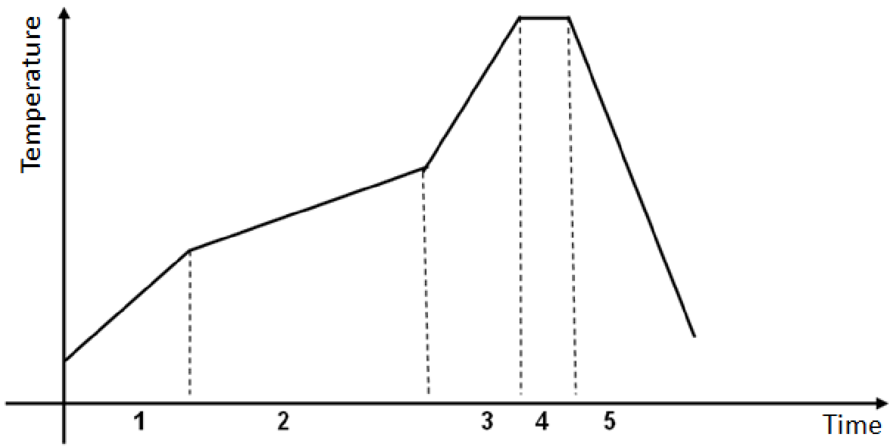

Figure 9. SAC Solder Reflow Profile

Rework Guidelines

It is recommended that parts and PCB are basked for 6 hours in oven at 120ºC to remove excess moisture. Standard SMT rework systems are recommended for this procedure as the temperature gradient can be well controlled to avoid thermal stress induced damage to the part. Care must be taken when removing or mounting a part on PCB such that no physical damage is induced to part or PCB.

1. Preheat the PCB to 100ºC as a first step to reduce temperature gradient before removing the MCSP part to be replaced.
2. Apply local convection heat to the part to raise the temperature slightly above the solder reflow temperature. The temperature gradient across the heating zone should not exceed 5ºC.
3. Thoroughly clean the solder paste and flux residue on the PCB. Ensure no damage is made to the solder resist layer which is crucial for solder reflow confinement for the part.
4. After cleaning the copper pads, reapply appropriate amount of solder paste with a mini stencil and squeegee and make sure solder paste is evenly distributed on all copper pads. Position and align the pads of the part onto the copper pads of the PCB using a pick-and-place device.
5. Use local convection heating tool to reflow the solder paste at the solder reflow temperature. Excessive local heat can drastically increase the part temperature resulting in permanent mechanical and functional damages to the part. Therefore, it is strongly recommended that either a convection heating tool with reflow profiling capability or a local part temperature monitoring device be used to monitor the part temperature.
6. Let the board and the part cool down to room temperature prior to electrical testing and evaluation.
7. Repeat steps 1-6 if additional rework is needed.

LEGAL DISCLAIMER

Alpha and Omega Semiconductor makes no representations or warranties with respect to the accuracy or completeness of the information provided herein and takes no liabilities for the consequences of use of such information or any product described herein. Alpha and Omega Semiconductor reserves the right to make changes to such information at any time without further notice. This document does not constitute the grant of any intellectual property rights or representation of non-infringement of any third party's intellectual property rights.

LIFE SUPPORT POLICY

ALPHA AND OMEGA SEMICONDUCTOR PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS.

As used herein:

1. Life support devices or systems are devices or systems which, (a) are intended for surgical implant into the body or (b) support or sustain life, and (c) whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in a significant injury of the user.

2. A critical component in any component of a life support, device, or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.