

SWAN EVT manufacturing Defects for MIC

SH Son

First Update

2020.11.19

Last Update

2020.00.00

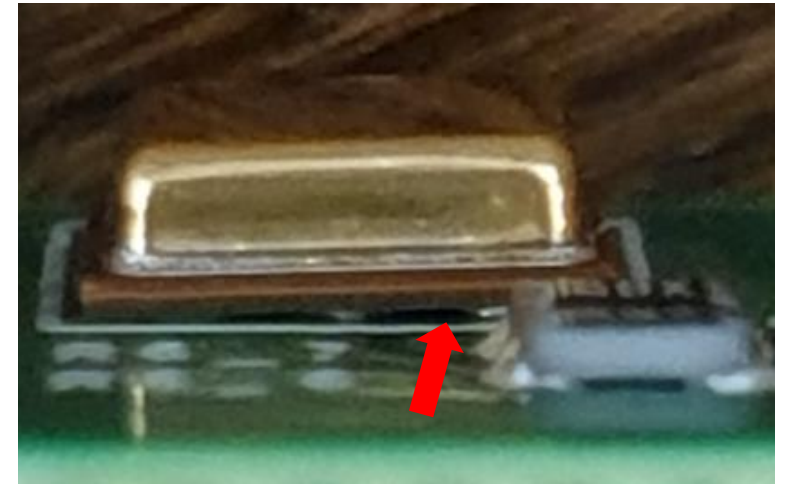
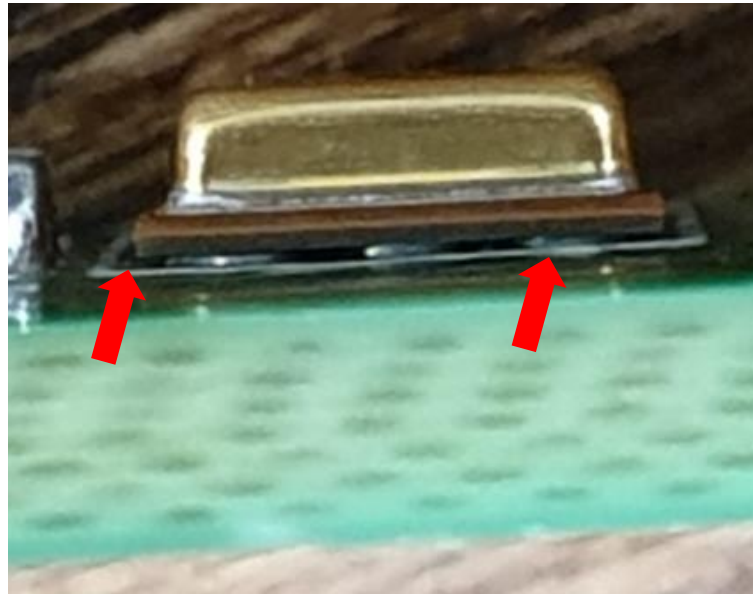
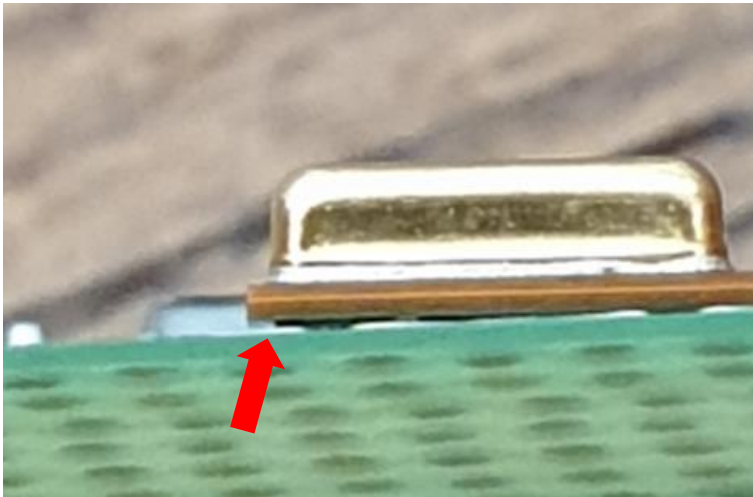
The ALT is under reviewing for MIC sealing solution.
And we found Mic PCBA manufacturing defect.

- Mic floating
- Mic solder ball

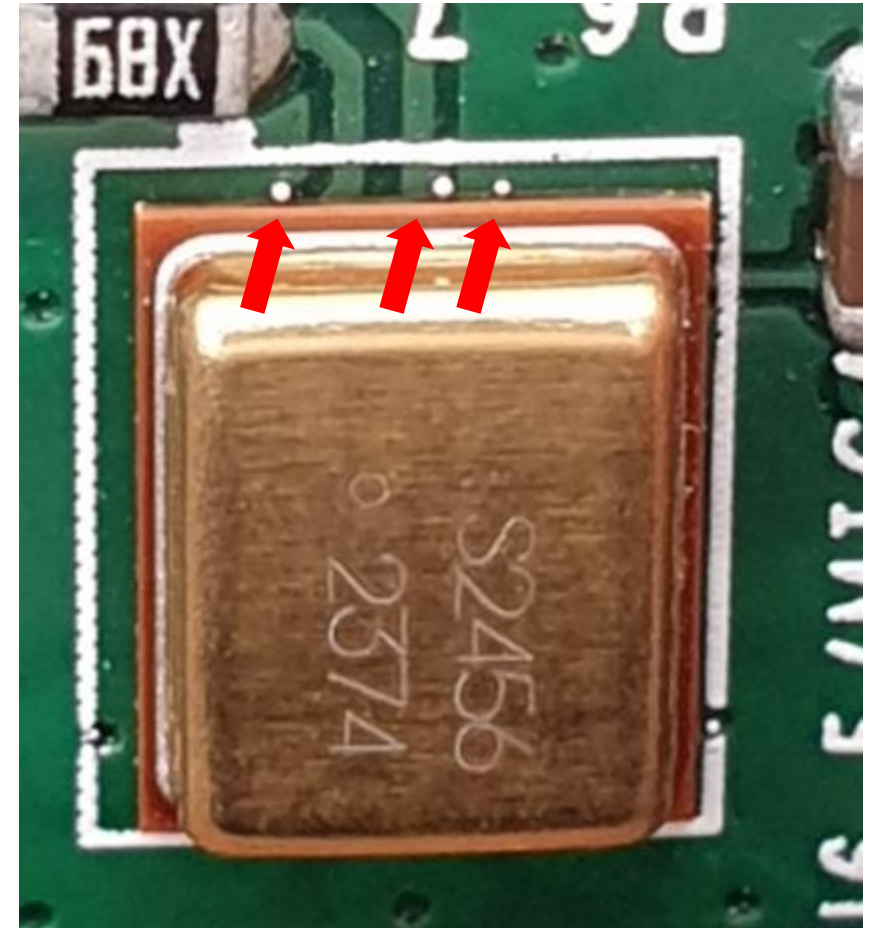
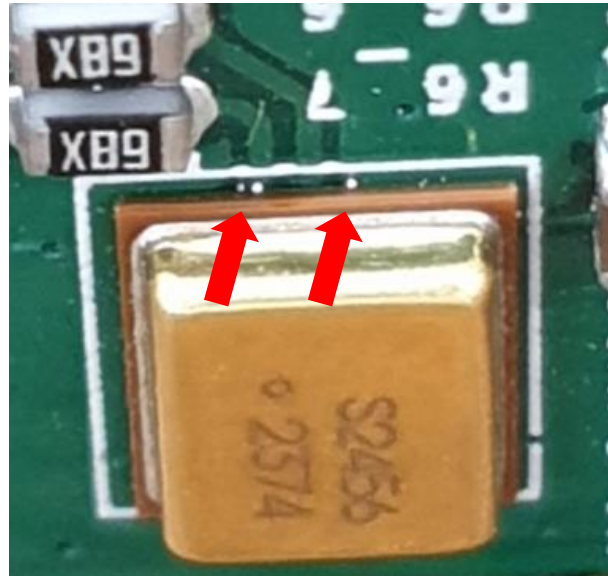
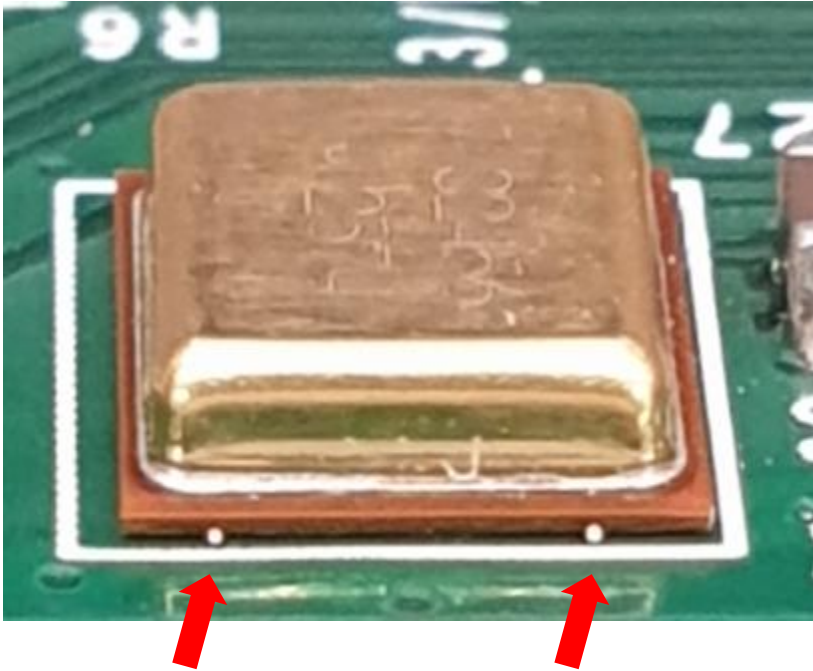
Also the solder ball is the defect defined by IPC-A-610

We think these defects relate MIC sealing. So, the MIC manufacturer's guidelines have been applied first. Then we will apply the sealing solution for mechanical side.

FXN and VN factory must be fine the root cause and make manufacturing solution before DVT build.
We can not agree DVT current manufacturing quality.



More photo, please see the attachment zip file.



More photo, please see the attachment zip file.

Action item before DVT

: Report the evidence that design and process is followed Knowles design guide line.

https://www.knowles.com/docs/default-source/default-document-library/sisonic-design-guide.pdf?sfvrsn=ddda77b1_6

- PCB LAND PATTERN AND SOLDER STENCIL PATTERN_23
- MANUFACTURING INFORMATION_30
- PICK-AND-PLACE SETTINGS_31
- REFLOW_32
- REWORK_32
- HANDLING AND STORAGE_33
- QUALIFICATION TESTING_33
- SENSITIVITY MEASUREMENTS_34

PCB LAND PATTERN AND SOLDER STENCIL PATTERN

The PCB land pattern, the solder stencil pattern, the solder paste, and the reflow profile should be designed to yield reliable solder joints. The solder joints serve as the electrical connection, mechanical connection, and (for bottom-port mics) the acoustic seal between the mic and PCB. The recommended PCB land pattern for each SiSonic model matches the microphone solder pads dimensions. The solder stencil pattern must be optimized for production, and for bottom-port SiSonic models must use a broken solder ring such as that shown in the figure below.

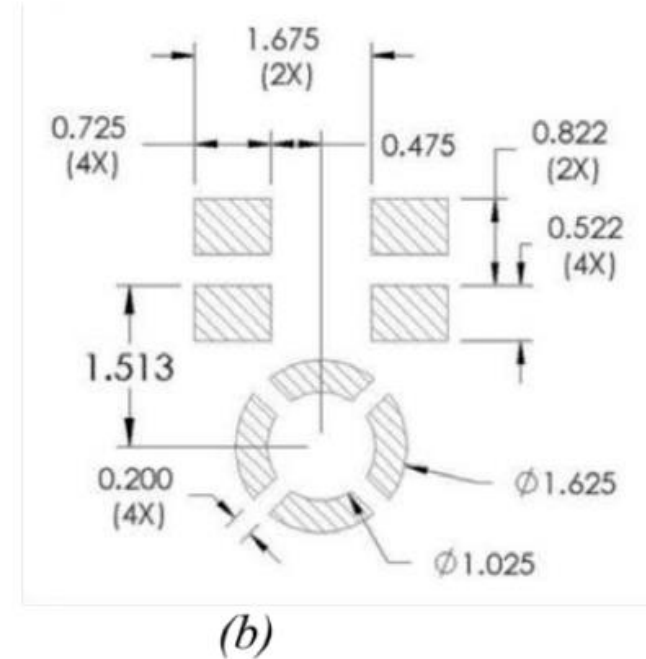
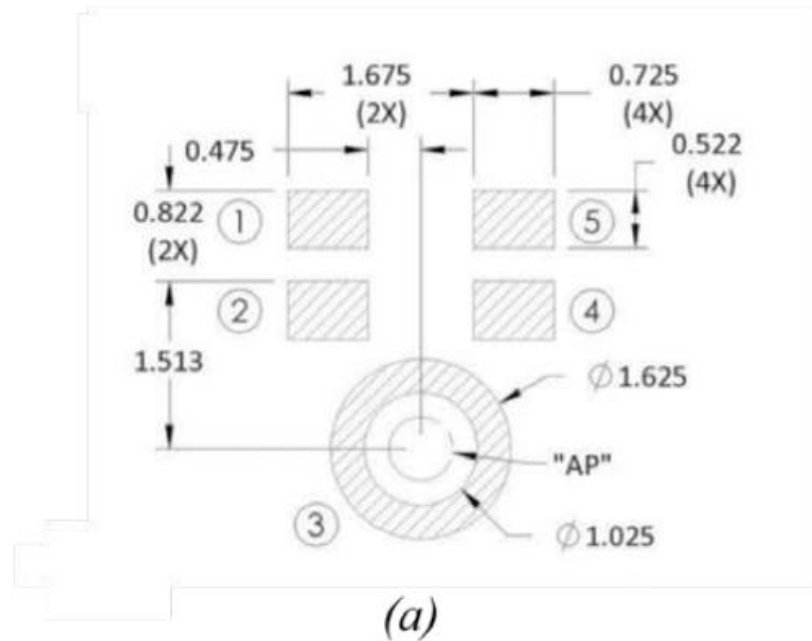


Figure 23: Comparison of SPH SiSonic (a) microphone solder pads and (b) a non-optimized reference solder stencil pattern with a broken solder ring.

MECHANICAL DESIGN CONSIDERATIONS

The solder stencil and land patterns should be designed while considering PCB design rules, solder type, reflow profile, solder stencil thickness, etc. Design optimizations could include:

1. Increasing the land pattern size symmetrically to extend beyond the edge of the mic to allow for visual inspection of the solder joint.
2. Splitting round pads in the land pattern into two semicircles to allow for better out-gassing during reflow and reduce the occurrence of bubbles.
3. For bottom-port mics, reducing the PCB hole diameter or increasing the solder ring diameter to meet PCB design rule requirements.
4. Optimizing the solder reflow profile for each unique board design to ensure good solder joints between the mic and PCB.
5. For bottom-port mics, reducing the solder flux content of solder paste to prevent excessive flux from entering the mic port hole during reflow.
6. Increasing the solder stencil thickness to ensure adequate solder volume for good solder joints.
7. Reducing the solder stencil thickness to reduce solder volume to minimize the occurrence of solder balls.