

---

## General Design Considerations for MEMS Microphones

---

### INTRODUCTION

Cirrus Logic MEMS microphones are widely used in consumer applications, offering high reliability and high performance in a miniature, low profile package compatible with Surface Mount Technology (SMT) processes. This application note offers recommendations in mechanical, PCB land pattern, and solder paste stencil design to deliver the best microphone performance in the final application design.

The design guidelines offered in this applications note should be considered as a starting point for further design optimisation. Adjustment and refinement in the mechanical, PCB land pattern and stencil design are necessary, according to specific mechanical, PCB constraints and manufacturing environments for zero defects in production.

### MECHANICAL DESIGN

The mechanical design is of prime importance and must be considered with care in the early design stage to avoid costly redesign of the casing, gasket and PCB. Correspondingly, it is essential for acoustic, PCB and mechanical designers to fully understand the mechanical structures involved, to be able to design a MEMS microphone into a final product successfully. To facilitate this, all the microphone dimensions can be obtained from the 'Package Dimensions' drawing in their respective datasheets.

Careful consideration should be given to the mechanical design configuration and stack up into the end product. This stack up typically includes the MEMS Microphone, PCB (or Flex PCB), gasket, mesh and casing as shown in Figure 1.

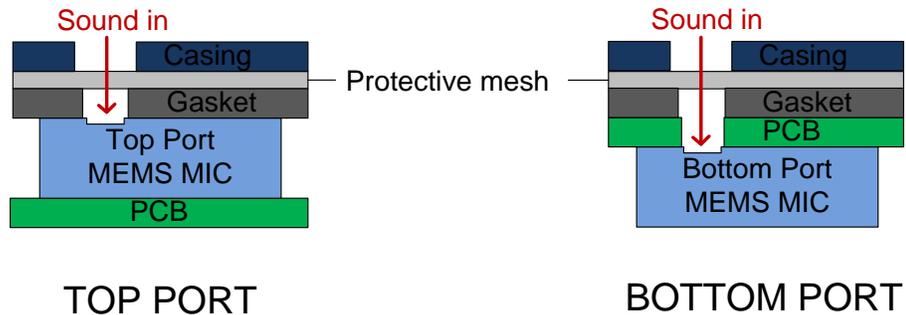


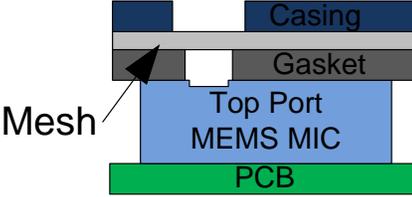
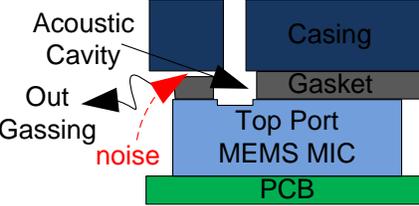
Figure 1 Typical Mechanical Mounting of Top and Bottom Port Microphones

Figure 1 illustrates the typical design configuration for top and bottom port microphones. In both configurations, the MEMS microphones are mounted on a PCB or Flex PCB to route the electrical connections. The microphone port hole should be positioned as close as possible to the outside environment. Thick layers of casing, gasket and PCB should be avoided, to maximise the sound performance.

For bottom port microphones, an acoustic port hole is required on the PCB for microphone sound pickup. The diameter of this PCB port hole should be larger than the port hole diameter of the bottom port microphone, smaller than the soldering ground ring of the microphone footprint and not plated to avoid excessive solder paste contamination into microphone port hole. A high standard of soldering at this ground ring is important to ensure a good acoustic seal. The requirement and suggestion for a high standard of soldering for this ground ring is discussed later in this document.

A rubber/poron gasket is generally required to provide a tight mechanical seal around the acoustic inlet. A tight seal by the gasket is important to block away any unwanted noise or acoustic coupling from other components in the system, and to reduce the vibration pickup in the overall design. For top port microphone configurations, it is important to make the acoustic inlet diameter on the gasket larger than the microphone port hole diameter. For bottom port microphone configurations, it is important to make the acoustic inlet diameter on the gasket larger, than the PCB port hole. In both configurations, the acoustic inlet diameter on the casing should be larger than the acoustic inlet on the gasket.

A protective mesh layer is used to offer protection against dust and splashes at the acoustic inlet. Grounded conductive mesh may be used to provide protection for potential ESD damage to the MEMS microphone or other surrounding components. Care must be taken to implement a mesh solution that meets the application-specific requirements, as the mesh layer adds additional acoustic impedance to the incoming sound pressure level which can impact the overall acoustic performance in the system design.

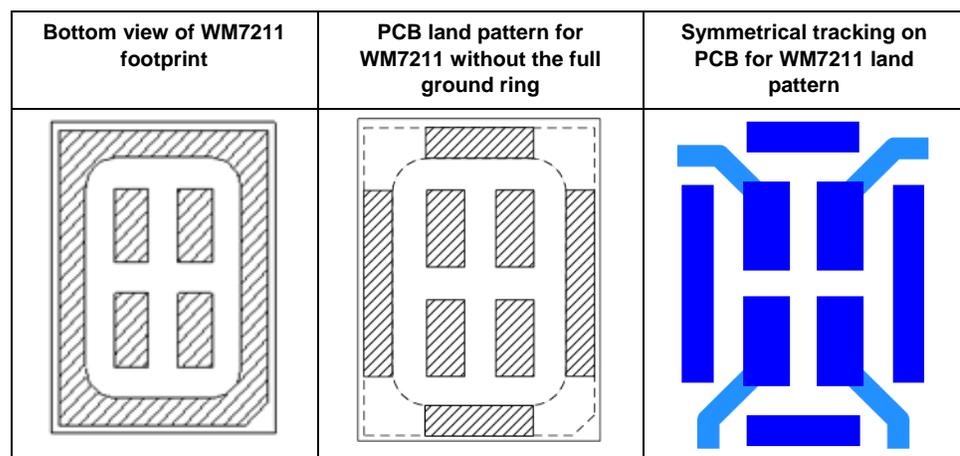
| <b>Good Design</b>                                                                                                                                                                                                         | <b>Bad Design</b>                                                                                                                                                |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------|
|                                                                                                                                          |                                                                               |
| <p>MEMS microphone should be positioned as close as possible to outside environment with larger port on gasket and casing to maximise the sound pickup.</p>                                                                | <p>Avoid long and narrow acoustic path formed by thick layers of casing and gasket to avoid Helmholtz's effect due to the trapped air in the acoustic cavity</p> |
| <p>Tight fitting gasket for good acoustic seal.</p>                                                                                                                                                                        | <p>Ensure tight fitting gasket to keep the noise coupling to minimum and out gassing issue with bad system frequency response performance</p>                    |
| <p>Grounded conductive mesh for protection against dust, splashes and ESD. The additional acoustic impedance helps to damp out any standing wave or Helmholtz's effect, extending the overall flat frequency response.</p> | <p>High risk of short product life with no protection from dust, splashes and ESD.</p>                                                                           |

**Figure 2 Cross Section View of Good and Bad Mechanical Design Examples**

**PCB FOOTPRINT DESIGN**

The following guidelines are recommended for PCB layout design:

- Each pad on PCB land pattern should be the same size as the respective pad on the package footprint. Details of the PCB footprint are provided in the “Recommended PCB Land Pattern” section of the respective datasheet. The mechanical tolerance of each dimension is specified in the “Package Dimensions” drawing.
- A 75um solder mask or solder resist clearance is recommended for typical PCB processes. A smaller clearance may be used, dependent on the capabilities of the PCB manufacture and assembly processes.
- Note that, for parts with a perimeter ground ring (e.g., WM7211, WM7210, WM7230), it is not critical to have the full ground ring on the PCB land pattern - the ground ring may be broken on the PCB land pattern if desired, as shown in Figure 3.
- Symmetrical arrangement of PCB tracks to pads underneath the microphone is recommended; this helps the microphone self-align to the PCB land pattern during the solder reflow process.
- Note that large VDD tracks and thermal pads are not required for microphone components.
- For bottom port microphones, a complete ground ring pad is required around the microphone port hole. This ring is important as it provides an acoustic seal around the port hole for the microphone input. The acoustic port hole in the PCB must be larger than the port hole of the microphone. It should also be un-plated and smaller than the ground ring pad, to avoid solder paste contamination in the microphone port hole.
- For bottom port microphones, an exposed metal ring should be provided on the opposite side of the PCB. This ring should be centred on the acoustic port hole and connected to the system ground. The exposed metal helps protect the microphone from ESD discharge and should not be covered by any housing or grommet material.
- For bottom port microphones, it is recommended that the thickness of the PCB should not exceed 2 times the diameter of the acoustic port hole in the PCB. The frequency response of the microphone is degraded if this limit is exceeded.
- A solid and unbroken ground plane should be provided under the microphone and connected to the microphone ground pad. For further information, refer to WAN0278 (Recommended PCB Layout for Microphone RF Immunity in Mobile Cell Phone Applications).
- Fiducials are recommended to assist with component alignment in automated assembly processes. An alignment marker around the edge of the microphone land pattern is not normally required but, if provided, it should be non-conductive or else sufficiently separated from the microphone land pattern to avoid electrical short circuit.
- The PCB area around the microphone must be kept sufficiently clear of other components to allow for the gasket that is required around the microphone.
- For bottom port microphones, it is recommended that no other components be positioned underneath the microphone (on the opposite side of the PCB). The acoustic performance may be compromised by the presence of other components in this area.



**Figure 3 Bottom View of WM7211 Footprint, PCB Land Pattern and Symmetrical Tracking**

**PCB STENCIL DESIGN**

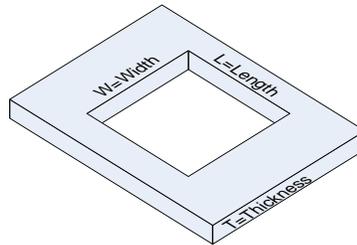
The stencil is a thin sheet of metal with apertures for uniform solder paste transfer onto the PCB land pattern. The design of the stencil is important to ensure high quality of the solder joint without electrical short or open connection for component pads on PCB land pattern.

Bad soldering on the ground ring for bottom port microphones can compromise on the acoustic performance. Therefore, adjustment of the stencil thickness and aperture size are critical for a high standard of soldering, good electrical connection and good acoustic seal for overall system performance with MEMS microphones.

Design Guidelines:

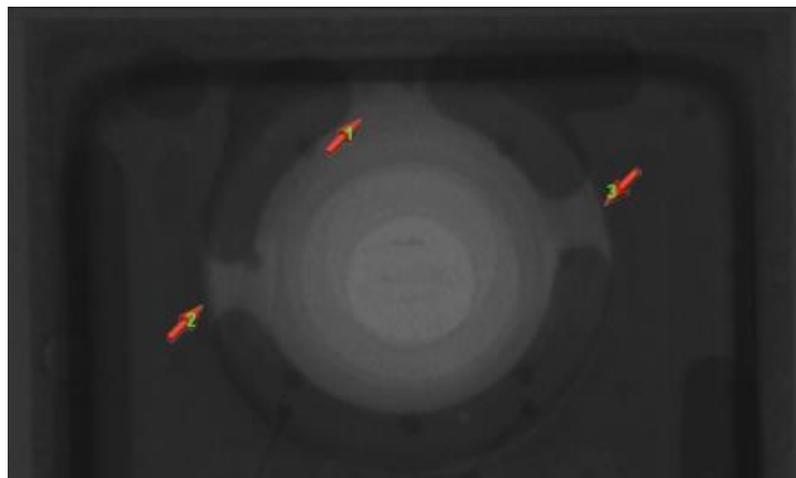
- A Laser-cut, stainless steel stencil is highly recommended as it not easy to deform and ensure more uniform and accurate solder paste deposition.
- Avoid Electroformed stencils.
- The aperture area should be 80% to 90% of the Pad area on PCB for better stencil alignment accuracy and repeatability. The smaller stencil aperture should be centred on the pad from each side.
- Stencil thickness of 75µm to 125µm is recommended. Smaller apertures require thinner stencil to maintain good area ratio.
- Ensure area ratio > 0.66 for all apertures

$$\text{Area Ratio} = \frac{\text{Area of the Pad}}{\text{Area of Aperture Wall}} = \frac{L \times W}{2 \times T \times (L + W)} > 0.66$$



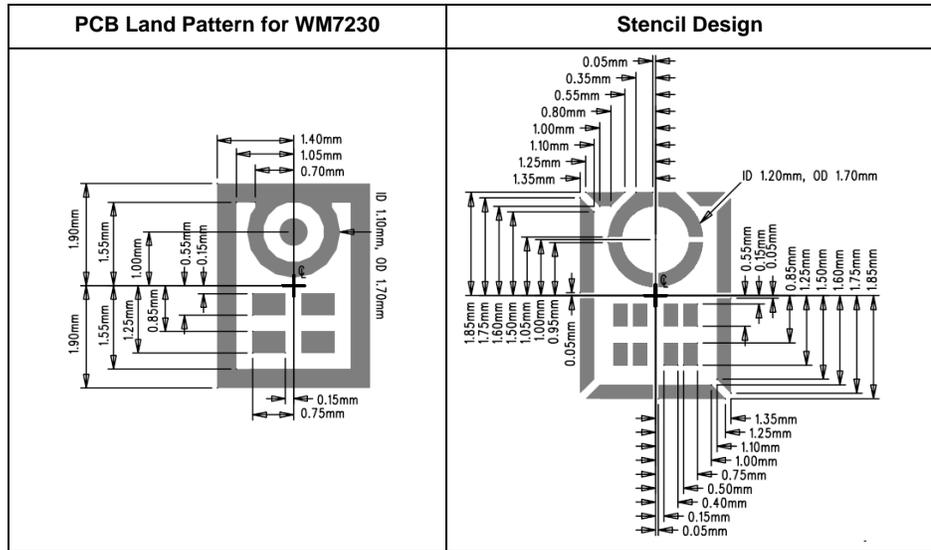
**Figure 4 Stencil Aperture Area Ratio**

- For pads with long edges in the direction of squeegee blade travel, it is recommended to have multiple smaller apertures with shorter edge for better solder paste control and more uniform printing.
- Avoid pasting on the acoustic port hole to avoid contamination for bottom port microphones. For the ground ring around the bottom port microphone port hole, it is recommended to have 4 apertures around the acoustic port hole separated by small relief gaps in between. It is essential to keep this gap small to avoid solder void or bad acoustic seal as shown in Figure 5.



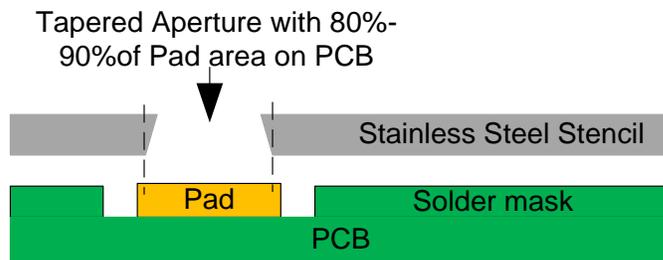
**Figure 5 Solder Void around the Bottom Port Microphone Port Hole with Bad Acoustic Seal**

- Figure 6 shows a typical stencil design recommendation. Note that, any recommendation of the stencil dimension is only a starting point for further optimisation with stencil thickness to achieve area ratio > 0.66 for each aperture or good solder joint on the PCB pad



**Figure 6 Typical Stencil Recommendation for WM7230**

- The apertures should be tapered approximately 5° to facilitate better solder paste release as shown in Figure 7.
- The stencil should be electroplated to improve the side wall smoothness for paste release.
- Avoid pasting on solder mask and bare substrate areas to minimise any risk of solder short.



**Figure 7 Cross Section View of Stencil Placement and Alignment on PCB (not to scale)**

**Trouble shooting:**

- Ensure correct stencil placement and alignment on PCB for the paste transfer as shown in Figure 7.
- Visual check to confirm good solder pasting before components placement.
  - Ensure every pad is pasted with accurate stencil alignment
  - Ensure accurate volume of solder paste deposition on the pad.
  - Ensure smaller area of solder paste than the Pad area as discussed.
  - Ensure no pasting on solder mask and bare substrate.
- Visual check after the soldering:
  - Avoid solder ball out at the edges of the microphone due to excessive solder paste in the printing process.
  - The soldered microphone is properly aligned in position and level evenly to the PCB.
- X-ray check to ensure good solder joint:
  - No significant voiding on solder joint.
  - It is essential to confirm the ground ring on PCB provides a good seal for the bottom port microphone as discussed.
- Electrical check to avoid short/open of all electrical signals.
  - No short with respect to ground for signal pads
  - Diode check to confirm the electrical connection for signal pads
- Review the following for further optimisation of the solder joints:
  - Aperture size with respect to stencil thickness as discussed.
  - Choice of lead-free paste.
  - PCB technology and mix of components on board.
  - Squeegee blade pressure and speed.
  - PCB planarity
- Ensure regular stencil cleaning for good paste release and avoid issues caused by paste/flux build-up and bleed on to PCB.

**SUMMARY**

This application note offers design notes and good practices on mechanical mounting, PCB land pattern and stencil design for MEMS microphones. For further information on MEMS microphone assembly and handling guidelines, please refer to apps note WAN0273 (MEMS MIC Assembly and Handling Guidelines).

---

## Contacting Cirrus Logic Support

For all product questions and inquiries, contact a Cirrus Logic Sales Representative.

To find one nearest you, go to [www.cirrus.com](http://www.cirrus.com).

---

### IMPORTANT NOTICE

The products and services of Cirrus Logic International (UK) Limited; Cirrus Logic, Inc.; and other companies in the Cirrus Logic group (collectively either "Cirrus" or "Cirrus Logic") are sold subject to Cirrus's terms and conditions of sale supplied at the time of order acknowledgement, including those pertaining to warranty, indemnification, and limitation of liability. Software is provided pursuant to applicable license terms. Cirrus reserves the right to make changes to its products and specifications or to discontinue any product or service without notice. Customers should therefore obtain the latest version of relevant information from Cirrus to verify that the information is current and complete. Testing and other quality control techniques are utilized to the extent Cirrus deems necessary. Specific testing of all parameters of each device is not necessarily performed. In order to minimize risks associated with customer applications, the customer must use adequate design and operating safeguards to minimize inherent or procedural hazards. Cirrus is not liable for applications assistance or customer product design. The customer is solely responsible for its selection and use of Cirrus products.

CERTAIN APPLICATIONS USING SEMICONDUCTOR PRODUCTS MAY INVOLVE POTENTIAL RISKS OF DEATH, PERSONAL INJURY, OR SEVERE PROPERTY OR ENVIRONMENTAL DAMAGE ("CRITICAL APPLICATIONS"). CIRRUS PRODUCTS ARE NOT DESIGNED, AUTHORIZED OR WARRANTED FOR USE IN PRODUCTS SURGICALLY IMPLANTED INTO THE BODY, AUTOMOTIVE SAFETY OR SECURITY DEVICES, NUCLEAR SYSTEMS, LIFE SUPPORT PRODUCTS OR OTHER CRITICAL APPLICATIONS. INCLUSION OF CIRRUS PRODUCTS IN SUCH APPLICATIONS IS UNDERSTOOD TO BE FULLY AT THE CUSTOMER'S RISK AND CIRRUS DISCLAIMS AND MAKES NO WARRANTY, EXPRESS, STATUTORY OR IMPLIED, INCLUDING THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR PARTICULAR PURPOSE, WITH REGARD TO ANY CIRRUS PRODUCT THAT IS USED IN SUCH A MANNER. IF THE CUSTOMER OR CUSTOMER'S CUSTOMER USES OR PERMITS THE USE OF CIRRUS PRODUCTS IN CRITICAL APPLICATIONS, CUSTOMER AGREES, BY SUCH USE, TO FULLY INDEMNIFY CIRRUS, ITS OFFICERS, DIRECTORS, EMPLOYEES, DISTRIBUTORS AND OTHER AGENTS FROM ANY AND ALL LIABILITY, INCLUDING ATTORNEYS' FEES AND COSTS, THAT MAY RESULT FROM OR ARISE IN CONNECTION WITH THESE USES.

This document is the property of Cirrus and by furnishing this information, Cirrus grants no license, express or implied, under any patents, mask work rights, copyrights, trademarks, trade secrets or other intellectual property rights. Any provision or publication of any third party's products or services does not constitute Cirrus's approval, license, warranty or endorsement thereof. Cirrus gives consent for copies to be made of the information contained herein only for use within your organization with respect to Cirrus integrated circuits or other products of Cirrus, and only if the reproduction is without alteration and is accompanied by all associated copyright, proprietary and other notices and conditions (including this notice). This consent does not extend to other copying such as copying for general distribution, advertising or promotional purposes, or for creating any work for resale. This document and its information is provided "AS IS" without warranty of any kind (express or implied). All statutory warranties and conditions are excluded to the fullest extent possible. No responsibility is assumed by Cirrus for the use of information herein, including use of this information as the basis for manufacture or sale of any items, or for infringement of patents or other rights of third parties. Cirrus Logic, Cirrus, the Cirrus Logic logo design and SoundClear are among the trademarks of Cirrus. Other brand and product names may be trademarks or service marks of their respective owners.

Copyright © 2014–2016 Cirrus Logic, Inc. All rights reserved.