Vesper Piezoelectric MEMS Voice Accelerometers
DESIGN GUIDELINES

Application Note 12
Rev 0.0.1
1. INTRODUCTION ...........................................................................................................................................3
2. LOCATION AND MOUNTING GUIDELINES .............................................................................................3
3. SCHEMATIC AND PCB GUIDELINES ........................................................................................................6
4. RF IMMUNITY GUIDELINES .....................................................................................................................10
5. REVISION HISTORY ..................................................................................................................................11
1. INTRODUCTION

The VA1200 is the world's first piezoelectric MEMS voice accelerometer. The VA1200 can be used to pick up the wearer's own voice through bone conduction. Using the VA1200 voice accelerometer in conjunction with a standard microphone, the application can achieve superior background and wind noise reduction.

The VA1200 has an ultra-small 2.9 mm X 2.76 mm X 0.9 mm package and is solder reflow compatible with no sensitivity degradation. It operates in environmentally harsh surroundings because it is dust and moisture resistant.

FEATURES

- Provides superior background and wind noise reduction
- Small Footprint – 2.9mm x 2.76 mm x 0.9mm
- Single Ended Analog Output
- High Frequency Bandwidth for user voice pickup

APPLICATIONS

- Truly Wireless Stereo (TWS) Headphones
- Background and Wind Noise Reduction
- Headsets
- Neckbands
- Earbuds
- Headsets / Hearables / Wearables

This Application Note is intended to describe the best practices that should be used to guarantee the VA1200 best performance in a customer application.

This document will cover:

- Location and Mounting guidelines.
- Schematic and PCB guidelines.
- RF immunity guidelines.

2. LOCATION and MOUNTING GUIDELINES

The VA1200 is best suited for TWS or headsets/neckbands with earbuds. Good mechanical coupling to the skull bone is essential for good voice signal pickup. Applications of VA1200 should consider these factors:

- Location of the VA1200 in the earbud:
  Vesper recommends the VA1200 be located in the main body of the earbud, and to avoid locations in the stem of the earbud.

- Orientation of the VA1200:
  The orientation is more important than the location of the voice accelerometer. Vesper's study of the skull vibration during voice production indicates the axis to pick up the strongest speech signal is the in-and-out of ear direction (Z axis in Figure 1). For designs that can more precisely control the orientation of the VA1200, the bone vibration direction depicted in Figure 2 can be referenced:
Figure 1: The Vesper Voice Accel senses vibrations in the Z direction so in order to get best results, the final location of the VA in the TWS should be parallel to the surface of the ear. Data on the right panel depict typical measurements from TWS mockups with 3 VA1200s mounted in orthogonal axes (Blue – VA1200 signal during speech at 81 dBSPL at the ear, Red – VA1200 signal during silence.)

Figure 2: Optimal angle against the head is 33 degrees down and 24 degrees forward relative to the axis represented by the line connecting the two ears.
• **Contact with earbud housing:**
The VA1200 can be mounted in flex PCB or rigid PCB as long as the FPCB or rigid PCB makes good contact with the housing of the TWS. Any rigid material (epoxy, lacquer, etc.) can be used to attach the voice accel to the side wall of the TWS. The more rigid the fixing material is, the better will be the coupling of vibrations from the ear into the TWS. If the VA1200 is mounted on the main PCB of the TWS, the PCB should be rigidly fixed to the housing with epoxy or screws as well.

• **TWS housing materials:**
Hard plastic TWS housing shells are preferred over soft ones.

• **Ear tips:**
Good fitting ear canal tips are important for voice capturing by voice accelerometers. Vesper recommends that the ear tips be designed to insert relatively deep into the ear canal. The silicone rubber ear tip cushion should be comfortable but not overly soft.
3. SCHEMATIC and PCB GUIDELINES

The VA1200 can be electrically connected in two configurations: single-ended or (pseudo) differential. It is recommended to use a differential connection to minimize the impact of any sources of noise, such as RF interference or supply/ground noise.

The VA1200 is a single-ended output device. However, customers have the option to use a single-ended or differential input configuration for their ADC inputs.

**Single-Ended Input Configuration:**

The Single-Ended Input Configuration below describes how to connect the VA1200 when customers have a single-ended input.

![Figure 3: Single-ended Input configuration schematic](image)

A ceramic capacitor of 100nF should be placed close to the \( V_{dd} \) pad of the VA1200 to adequately decouple the VA1200 from power supply noise.

A DC-blocking capacitor is required at the input of the ADC. The exact value needs to be determined based on the cut-off frequency required.

If the VA1200 is used near high-frequency signals (such as Bluetooth, WiFi, or fast digital signals), three 15pF ceramic capacitors should be added to suppress RF noise. These capacitors should be 0201 size (0603 metric) or smaller to maintain high-frequency performance (see Table 1).

- One 15pF capacitor close to the \( V_{dd} \) pad, in parallel with 100nF. This should be placed closer than the 100nF capacitor because it operates at higher frequencies.
- One 15pF capacitor close to the \( V_{out} \) pad, from \( V_{out} \) to ground.
- One 15pF capacitor near the ADC input pin, from the ADC input pin to ground. (ADC may be part of the SoC)
In a single-ended configuration, care should be taken when routing the ground signal. We recommend a separate route for the VA1200 ground. This special ground path (GND VA in Figure 3) should branch off as close to possible to the ground pin of the MICBIAS bypass capacitor. Wherever possible, GND VA should be used to shield the VOUTP and MICBIAS traces (as labeled in Figure 3). The shielding should include, if possible, a GND VA plane above and below the VOUTP and MICBIAS traces, and GND VA traces shielding the sides of those signals. Ideally, there should be GND VA vias near the sensitive traces, every few millimeters. VOUTP and MICBIAS should be buried in-between ground planes as much as possible, with minimal trace area exposed at the PCB surface.

**Differential Input Configuration:**

The Differential Input Configuration below describes how to connect the VA1200 when customers have differential inputs.

![Differential Input configuration schematic](image)

As in the single-ended case, a ceramic capacitor of 100nF should be placed close to the Vdd pad of the VA1200 to adequately decouple the VA1200 from power supply noise.

Two DC-blocking capacitors are required, one at each input of the ADC. The exact value needs to be determined based on the cut-off frequency required.

One 400Ω resistor should be used to generate the pseudo-differential negative output (VOUTN in Figure 4). This resistor should be near the ground pads of the VA1200. This resistor value is chosen to match the output impedance of the VA1200.

If the VA1200 is used near high-frequency signals (such as Bluetooth, WiFi, or fast digital signals), five 15pF ceramic capacitors should be added to suppress RF noise. These capacitors should be 0201 size (0603 metric) or smaller to maintain high-frequency performance (see Table 1).
• One 15pF capacitor close to the \( V_{dd} \) pad, in parallel with 100nF. This should be placed closer than the 100nF capacitor because it operates at higher frequencies.
• One 15pF capacitor close to the \( V_{out} \) pad, from \( V_{out} \) to ground.
• One 15pF capacitor close to the 400\( \Omega \) resistor (and electrically in parallel).
• Two 15pF capacitors near the ADC input pins, one from each ADC input pin to ground. (ADC may be part of the SoC)
• We recommend adding an extra footprint for a capacitor between \( VOUTP \) and \( VOUTN \) (see DNP capacitor in Figure 4). This footprint can help solve unexpected RF interference issues.

In a differential configuration, extensive ground shielding should be used to protect the \( VOUTP \), \( VOUTN \) and \( MICBIAS \) traces (as labeled in Figure 4). These traces should be buried in-between ground planes as much as possible, with minimal trace area exposed at the PCB surface. The shielding should include ground traces on the sides of these sensitive traces, and ground vias near the sensitive traces every few millimeters.

**RF capacitor selection:**

While we recommend 15pF RF capacitors as a starting value, the best choice of capacitor value is influenced by its physical size and the frequency of the interfering signal. We recommend that the RF capacitor be 0201 size (0603 metric) at the most. See Table 1 for more details:

<table>
<thead>
<tr>
<th>Interfering RF source</th>
<th>Capacitor size:</th>
<th>Capacitor size:</th>
<th>Capacitor size:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Imperial 008004</td>
<td>Imperial 01005</td>
<td>Imperial 0201</td>
</tr>
<tr>
<td></td>
<td>Metric 0201</td>
<td>Metric 0402</td>
<td>Metric 0603</td>
</tr>
<tr>
<td></td>
<td>0.25mm x 0.125mm</td>
<td>0.4mm x 0.2mm</td>
<td>0.6mm x 0.3mm</td>
</tr>
<tr>
<td>DECT (1.9GHz)</td>
<td>56pF or 62pF</td>
<td>39pF</td>
<td>27pF</td>
</tr>
<tr>
<td>Bluetooth, WiFi (2.4GHz)</td>
<td>33pF or 39pF</td>
<td>22pF</td>
<td>15pF</td>
</tr>
<tr>
<td>WiFi (5GHz)</td>
<td>8.2pF</td>
<td>5.2pF</td>
<td>3.9pF</td>
</tr>
</tbody>
</table>

*Table 1: RF capacitor recommendations*

**Example PCB Layout:**

Below is an example of single-ended (upper) and differential (lower) VA1200 layouts on a four-layer PCB. It is crucial to minimize the loop area of the RF capacitors (indicated with yellow circles in Figure 5). More information about properly RF-shielded routing can be found in Section 4: RF Immunity Guidelines.
Figure 5: Layer 1 (Top), PCB layout for VA1200. Single-Ended layout (upper) and Differential layout (lower).

The Single-Ended layout (upper) has an independent ground (GND VA) which is separate from the main ground plane.

Figure 6: Layer 2, PCB layout for VA1200. Single-Ended layout (upper) and Differential layout (lower).

Figure 6 shows the routing of the most sensitive signals on Layer 2. Ground traces are used to shield the sides of these signals. The Single-Ended layout (upper) has an independent ground (GND VA) which is separate from the main ground plane. Ground vias are included every few millimeters, to provide more lateral shielding.

Figure 7: Layer 3, PCB layout for VA1200. Single-Ended layout (upper) and Differential layout (lower).

Figure 7 shows additional ground planes below the sensitive signals to provide further shielding. As in Layer 1 and 2, the Single-Ended layout (upper) has an independent ground (GND VA) which is separate from the main ground plane.
Some additional PCB layout guidelines:

- We recommend using Non-Soldermask Defined (NSMD) pads for the VA1200 footprint. NSMD pads have tighter manufacturing tolerances than SMD pads. This is because the boundary of the exposed pad area is defined by the copper layer, not the soldermask layer. This tighter tolerance is necessary for the small pads on the VA1200.

- Avoid placing vias underneath the VA1200 footprint. This is because the VA1200 should be mounted on a very flat PCB surface, to ensure good assembly yield during reflow soldering. The presence of a via underneath the VA1200 footprint can reduce PCB flatness, especially at extreme temperatures.

- It is acceptable to route signal traces on the copper layer directly underneath the VA1200. Fill the area around the traces with ground plane to maintain planarity of the PCB surface (see Figure 5).

4. RF IMMUNITY GUIDELINES

The VA1200 itself is relatively insensitive to interference from most RF devices. This includes interference in the 2.4GHz band (Bluetooth and WiFi) and similar bands (1.9GHz DECT and 5GHz WiFi). Many of these recommendations apply not only to the VA1200 but to analog sensors in general.

However, the signal traces of the VA1200 are vulnerable to RF pickup. These traces (such as $\text{VOUTP}$, $\text{VOUTN}$ and $\text{MICBIAS}$ in Figure 2) connect the VA1200 to the audio receive circuit (ADC or audio buffer) and the microphone supply source. If these traces are routed close to an RF transmitter, they can pick up the RF signal. This can result in audio interference, because the RF signal can interact with non-linearities in the pins connected to these traces (such as the ADC input). One example of such a non-linearity is the ESD protection that is built into most integrated circuits. As a result, care is needed in routing the VA1200 signals close to an RF transmitter.

The following precautions are recommended:

- Bury the sensitive traces to reduce their exposed area on the PCB surface. Wherever possible, run these traces on internal PCB layers, with ground planes above and below. There should also be ground on either side of these traces to provide lateral shielding. Tie the grounds together with frequent vias (every few millimeters, if possible).
• If the VA1200 will be connected to a single-ended input (see Figure 3), use a separate route for the VA1200 ground. This special ground path (GND_VA) should branch off as close to possible to the ground pin of the MICBIAS bypass capacitor.

• If the VA1200 will be connected to differential inputs (see Figure 4), create a pseudo-differential negative output (VOUTN) for the VA1200 using a 400 Ω resistor. Run this trace adjacent to the positive output (VOUTP) so that the two traces pick up similar levels of RF. This configuration will help the differential inputs to reject RF interference.

• Place small RF shunt capacitors near the VA1200 Vdd and Vout pins, and near the ADC input pins. See Figures 3 and 4 for details. Table 1 provides more information about appropriate capacitor value and sizing. When routing the signal to these capacitors, avoid stubs on the connections (see Figure 9). Instead, track through the capacitor pad.

![Figure 9: Diagram showing PCB pad connection with stub (left) and without stub (right).](image)

5. REVISED HISTORY

<table>
<thead>
<tr>
<th>Revision</th>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0.1</td>
<td>02/15/2021</td>
<td>Initial Revision</td>
</tr>
</tbody>
</table>