Design Considerations for ISD1700 Family

AN-CC1002

1. Introduction

Winbond's ISD1700 family is another ChipCorder® product, which has a Class D PWM speaker output with wide operating voltage, ranging from 2.4V to 5.5V. This PWM speaker driver maximizes audio volume and power efficiency. However, without adequate system power supply and distribution design, can result in higher noise levels than a typical speaker driver. Also, as usual, higher operating voltage induces more power noise on the system. As a result, it increases the challenges on optimizing the voice quality on an end-product.

This Application Note discusses the crucial factors for considerations while implementing ISD1700 Series. Then, proposes some simple and cost-effective recommendations.

2. Decoupling Capacitors

In addition to the typical 0.1μF capacitor on the power lines, when a 10μF Aluminum Electrolytic capacitor is added to each V_{CCA}, V_{CCD} and V_{CCP} power line with respect to the related ground path, it can reduce the noise from the power supplies. The locations of these capacitors should be as close to the device as possible. By doing so, it enhances the voice quality. Sometimes, a 4.7μF Aluminum Electrolytic capacitor may be sufficient for certain applications. Furthermore, we have experienced that the SMT capacitor reduces the noise, but the result may not be as good as that using the Aluminum Electrolytic type.

3. Layout Techniques

A good practice is to separate each power line (V_{CCA}, V_{CCD} & V_{CCP}) and each ground path (V_{SSA}, V_{SSD}, V_{SSP1} & V_{SSP2}) individually from the device to the system. Meanwhile, the ISD1700 device has isolated V_{CCP}, V_{SSP1} & V_{SSP2} pads for PWM speaker driver. To minimize the noise from the PWM speaker driver, it is vital to route an independent trace from each related pin to the supply and ground terminals directly. The following aspects should be taken into accounts:

- Place V_{SSA} and V_{SSD} planes/paths on one side of PCB and the V_{SSP1} & V_{SSP2} planes/paths on the opposite side of PCB.
- Make each V_{SSA} and V_{SSD} as one big ground plane and as large as possible. Size the V_{SSA} and V_{SSD} planes in approximate equivalent area.
- Layout the V_{SSP1} & V_{SSP2} traces as large as possible and connect them to the system ground in the shortest distance.
- Funnel each ground path back to ground terminal for better grounding effect.
- Similarly, apply independent path technique on V_{CCA}, V_{CCD} and V_{CCP} signals.
- The decoupling capacitors should be as close to the device as possible.

- Also, avoid running the signal traces close to the V_{CCP}, V_{SSP1} and V_{SSP2} traces.

3.1 Layout Example

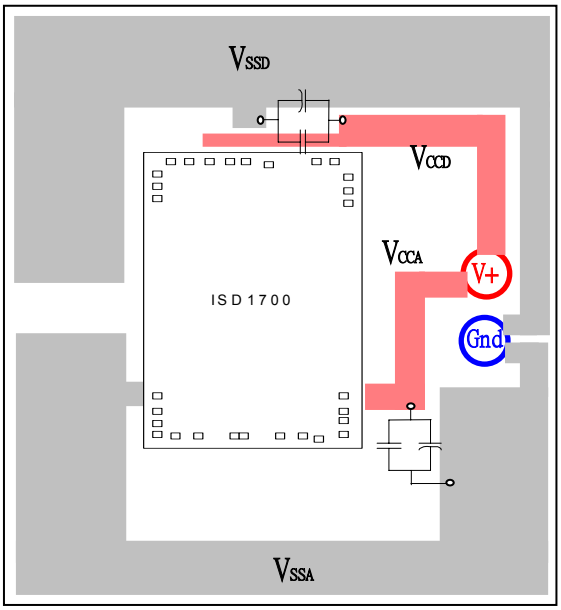

Fig. 1: One side of PCB

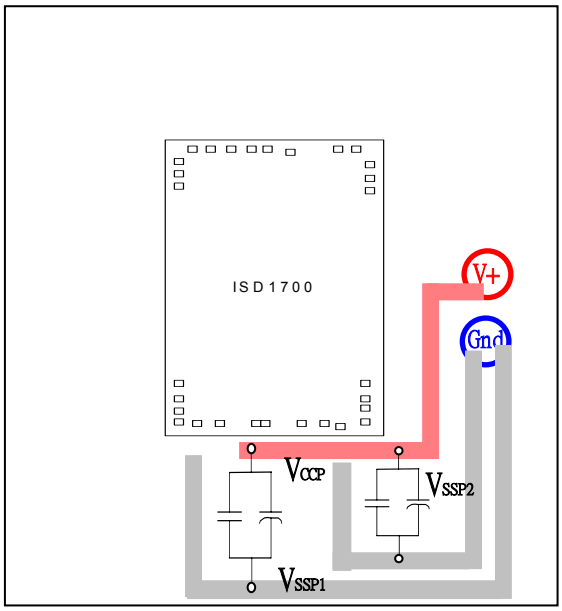

Fig. 2: Another side of PCB

The above diagrams represent a generic layout for ISD1730 or ISD1760 device as an example. In the diagrams, the die is magnified in order to review clearly the locations of the related power and ground pads, as well as the components. Hence, the illustrations are not in 1:1 ratio.

4. Physical Dimension of PCB

Experiment 1:

The first experiment uses a large PCB. The layout follows the above guidelines with both 0.1μF and 10μF capacitors installed on all V_{CCA}, V_{CCD} and V_{CCP} power lines. It produces excellent voice quality on the recorded message.

If 10μF capacitors are removed from all power lines, the voice quality slightly degrades. However, it may still be acceptable for certain kinds of applications. The PCB dimension is approximately 4 ½ inches x 3 inches. Fig. 3 shows the 1:1 ratio of the size of the PCB.

Experiment 2:

In the second experiment which tries to simulate the practical case for mass production environment, a much smaller PCB is used. When both 0.1μF and 10μF capacitors are installed on all V_{CCA}, V_{CCD} and V_{CCP} power lines; a very good voice

quality is produced.

However, if the 10μF capacitors are changed to lower value or removed, the voice quality obviously degrades with distortions or noise, which may only be suitable for limited types of applications. Therefore, the existence of the Aluminum Electrolytic capacitor plays an important role to the quality required. In this case, the dimension of the PCB is about 1 ½ inches x 2 1/8 inches, which is substantially smaller than that of Expt. 1. Fig. 4 shows the 1:1 ratio of the size of the PCB.

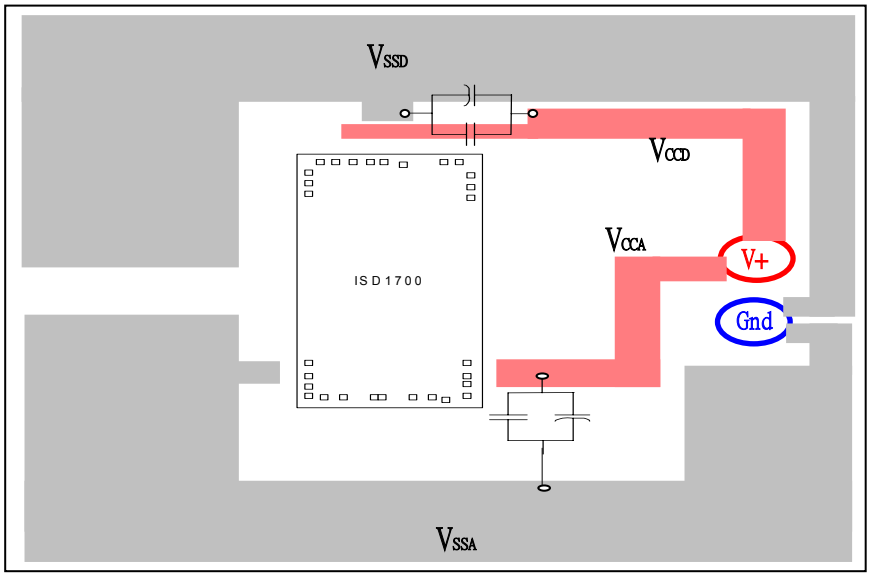

Fig. 3: PCB Dimension of Expt. 1 (1:1 ratio – PCB size)

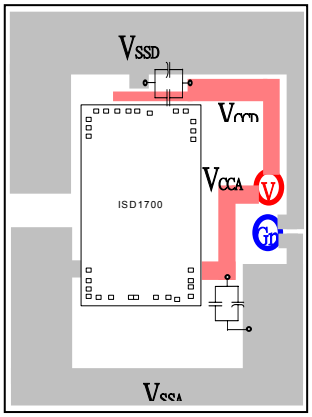

Fig. 4: PCB Dimension of Expt. 2 (1:1 ratio – PCB size)

5. Recommendations

In general, the quality of speaker engaged and the design of speaker housing are essential to the voice quality. The experimental results confirm that the voice quality can also be influenced by any or all of the following factors:

- The presence of the Aluminum Electrolytic decoupling capacitors on the power lines,
- Appropriate layout techniques,
- The dimension of the PCB.

Ideally implementing all above is strongly recommended. However, depending upon the specifications of the system and what level of voice quality are required for the applications, one can select an appropriate solution accordingly. Experiments are suggested in order to optimize for the best voice quality on the end-product. Nevertheless, there are trade-offs on which approach should be employed, pending upon the final products. Therefore, compromise may be necessary between the cost effectiveness and the voice quality.