

FEATURES

DC power supply accepts 2.5 V to 5.5 V

Single-ended and differential input capability

Extremely small board size allows integration (0.34 in × 0.34 in)

GENERAL DESCRIPTION

The SSM2306 is a single-chip, dual-channel, Class-D stereo audio amplifier. It is a fully integrated chip, which means that the application circuit requires a minimum of external components. It comes with a differential mode input port and a high efficiency H-bridge at the output. When compared to a half-bridge output stage, a full H-bridge enables direct coupling of the audio power signal to the loudspeaker, doubling the output voltage swing and eliminating the need for a large output coupling capacitor. Another benefit of a full H-bridge is an increase in the maximum output power by 4× when compared to a half bridge under the same load impedance. These benefits are particularly useful for low voltage, battery-powered, portable electronics where energy and space are limited. The differential mode input stage allows for canceling of common-mode noise leading to a superior CMRR.

Moreover, the part features a high efficiency, low noise output modulation scheme that does not require external LC output filters when attached to an inductive load. The modulation provides high efficiency even at low output power. Filterless operation also helps to decrease distortion due to nonlinearities of output LC filters, thereby delivering better sound quality and leading to savings in board space and overall cost.

This data sheet describes how to configure and use the SSM2306 mini evaluation board. It is recommended that this data sheet be read in conjunction with the [SSM2306](#) data sheet, which provides more detailed information about the specifications, internal block diagrams, and application guidance for the amplifier IC.

EVALUATION BOARD DESCRIPTION

The SSM2306 mini evaluation board carries a complete application circuit for driving a stereo loudspeaker. Figure 1 shows the top view of the PCB, and Figure 2 shows the bottom view of the PCB. The top silkscreen layer of the evaluation board is shown in Figure 4 with other top layers, including top copper, top solder mask, and multilayer vias. The bottom silkscreen layer of the evaluation board is shown in Figure 5 with other bottom layers, including bottom copper, bottom solder mask, and multilayer vias.

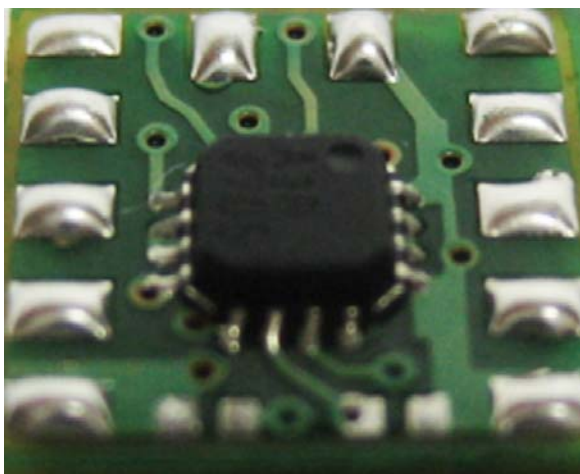


Figure 1. SSM2306 Mini Evaluation Board, Top View

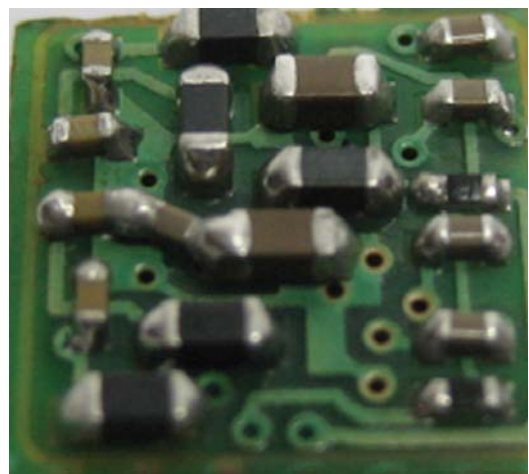


Figure 2. SSM2306 Mini Evaluation Board, Bottom View

Rev. 0

Evaluation boards are only intended for device evaluation and not for production purposes. Evaluation boards are supplied "as is" and without warranties of any kind, express, implied, or statutory including, but not limited to, any implied warranty of merchantability or fitness for a particular purpose. No license is granted by implication or otherwise under any patents or other intellectual property by application or use of evaluation boards. Information furnished by Analog Devices is believed to be accurate and reliable. However, no responsibility is assumed by Analog Devices for its use, nor for any infringements of patents or other rights of third parties that may result from its use. Analog Devices reserves the right to change devices or specifications at any time without notice. Trademarks and registered trademarks are the property of their respective owners. Evaluation boards are not authorized to be used in life support devices or systems.

EVAL-SSM2306-MINI

TABLE OF CONTENTS

Features	1	Component Selection	3
General Description	1	PCB Layout Guidelines.....	4
Evaluation Board Description.....	1	Evaluation Board Schematic and Artwork.....	5
Revision History	2	Ordering Information.....	7
Evaluation Board Hardware.....	3	Ordering Guide	7
Getting Started.....	3	ESD Caution.....	7
What to Test	3		

REVISION HISTORY

7/08—Revision 0: Initial Version

EVALUATION BOARD HARDWARE

Note that the SSM2306 evaluation board and layout guidelines were developed by Gang Liu of Analog Technologies, Inc., Sunnyvale, CA.

The SSM2306 mini evaluation board features the SSM2306 filterless, Class-D stereo amplifier, designed to drive a bridge-tied load (BTL) stereo loudspeaker in portable audio applications. The SSM2306 mini evaluation board operates from a dc power supply that can provide 2.5 V to 5.5 V. The audio input source is amplified to drive 1.4 W into an 8 Ω loudspeaker at 5.0 V supply. The SSM2306 mini evaluation board accepts differential or single-ended input signals.

On the left side of the SSM2306 mini evaluation board are five solder pads for the input audio signal: INR+ and INR– for the right channel, INL+ and INL– for the left channel, and GND (see Figure 4). The evaluation board uses INx+ and INx– to receive audio signals from common appliances, such as DVD players, PCs, and TVs.

When the input signal is sensed in single-ended mode, the single audio signal is transmitted from INR+ and/or INL+ into the IC. For single-ended mode, the negative input nodes (INR– and/or INL–) must be short-circuited to a neighboring ground (see the schematic in Figure 3). Because some appliances have strong ground noise, using only INR+ and/or INL+ as the inputs may allow this noise to be converted into audible hissing sounds at the output. If this problem occurs, a differential mode connection is needed to cancel the ground noise interference.

GETTING STARTED

To ensure proper operation, connect the loads immediately after the ferrite beads using the OUTBL+, OUTBL–, OUTBR+, and OUTBR– nodes.

WHAT TO TEST

When implementing the SSM2306 mini evaluation board in your design, test the evaluation board for the following items:

- Electromagnetic interference (EMI). Connect wires for the speakers in the same length as for the actual application and perform the EMI test.
- Signal-to-noise ratio (SNR).
- Total harmonic distortion + noise (THD + N).
- Output noise. Use an A-weighting filter to filter the output before the measurement meter.
- Maximum output power.
- Efficiency.

COMPONENT SELECTION

Selecting the right components is the key to achieving the performance required at the budgeted cost.

Input Coupling Capacitors

The input coupling capacitors, C1, C2, C3, and C4, should be large enough to couple the low frequency signal components in the incoming signal and small enough to filter out unnecessary low frequency signals. For music signals, the selected cutoff frequency is often between 20 Hz and 30 Hz. The value of the coupling capacitor is calculated as follows:

$$C = 1/(2\pi \times R \times f_c)$$

where $R = 43 \text{ k}\Omega + R_{\text{EXT}}$ and f_c is the cutoff frequency.

Input Series Resistors

Resistors in series with the input pins are not necessary components for amplifier operation and are needed only when special gain values are required. Using resistors with too high a value increases the input noise.

Output Beads

The output beads, B1, B2, B3, and B4, are necessary components for filtering out the EMI caused at the switching output nodes when the length of the speaker wire is greater than 10 cm. The penalty for using ferrite beads for EMI filtering is slightly worse noise and distortion performance at the system level due to the nonlinearity of the beads.

Ensure that these beads have enough current conducting capability while providing sufficient EMI attenuation. The current rating needed for an 8 Ω load is approximately 600 mA, and impedance at 100 MHz must be > 220 Ω . In addition, the lower the dc resistance (DCR) of these beads, the better for minimizing their power consumption. Table 1 describes the recommended beads.

Table 1. Recommended Output Bead

Part Number	Manufacturer	Z (Ω)	I _{MAX} (mA)	DCR (Ω)	Size (mm)
MPZ1608S221A	TDK	220	2000	0.05	1.6 × 0.8 × 0.8

Output Shunting Capacitors for the Beads

The output shunting capacitors for the beads, C5, C6, C7, and C8, filter out lower frequency EMI ≤ 250 MHz. Use small size (0603 or 0402) multilayer ceramic capacitors made of X7R or COG (NP0) materials. The higher the value of these capacitors, the lower the residual EMI level at the output, and the higher the quiescent current at the power supply. It is recommended that 500 pF to 1 nF capacitors be used.

PCB LAYOUT GUIDELINES

Correct PCB layout is critical in application designs to prevent EMI from exceeding allowable limits and to ensure that the amplifier chip operates below the temperature limit. Follow these guidelines carefully when designing the PCB layout.

1. Place nine vias onto the thermal pad of the amplifier. The outer diameter of the vias should be 0.5 mm, and the inner diameter should be 0.33 mm. Use a PCB area of at least 2 cm² or an equivalent area on the back side of the PCB layer as the heat sink (see Figure 4 and Figure 5). If internal layers are available in the PCB, allocate an area as large as possible to the ground planes and connect these vias to the ground planes.
2. Place the EMI filtering beads (B1, B2, B3, and B4) as close to the amplifier chip as possible.
3. Place the decoupling capacitors for the beads (C5, C6, C7, and C8) as close to the amplifier chip as possible, and connect all their ground terminals together.
4. Place the first decoupling capacitor for the power supply (C9) as close to the amplifier chip as possible, and connect its ground terminal directly to the IC GND pins (Pin 13 and Pin 16).
5. Place the other decoupling capacitor for the power supply (C10) as close to the amplifier chip as possible, and connect its ground terminal to the PCB ground area from which the power supply traces come.
6. Place the bead for the power supply (B5) as close to the amplifier chip as possible, and keep it on the same side of the PCB as the chip.
7. The ferrite beads can block EMI up to 160 MHz. To eliminate EMI higher than 160 MHz, place a low value, small size capacitor, such as a 100 pF, 0402 size capacitor, in parallel with the decoupling capacitors, C5, C6, C7, and C8. Place this small capacitor at least 20 mm away from the 1 nF decoupling capacitor. Ideally, the ground terminals of these small capacitors should be connected to the ground terminals of the 1 nF decoupling capacitors or to the PCB traces, which are placed as close to the output loads (the loudspeakers) as possible. In this way, the PCB connecting trace between these two capacitors serves as an inductor for filtering out the high frequency component.
8. Ground the unconnected pins, Pin 6 and Pin 7.
9. Connect the GND pins, Pin 13 and Pin 16, to the thermal pad and place grounding vias, as shown in Figure 3 and Figure 4.
10. Use a solid polygon plane on the other side of the PCB for the area of the vias that are placed on the thermal pad of the chip. See Figure 4 or Figure 5. This polygon serves as both the EMI shielding ground plane and the heat sink for the SSM2306.
11. Keep the PCB traces of high EMI nodes on the same side of the PCB and as short as possible. The high EMI nodes on the SSM2306 are Pin 1, Pin 2, Pin 11, and Pin 12.

The SSM2306 mini evaluation board works well only if these techniques are implemented in the PCB design to keep EMI and amplifier temperature low.

EVALUATION BOARD SCHEMATIC AND ARTWORK

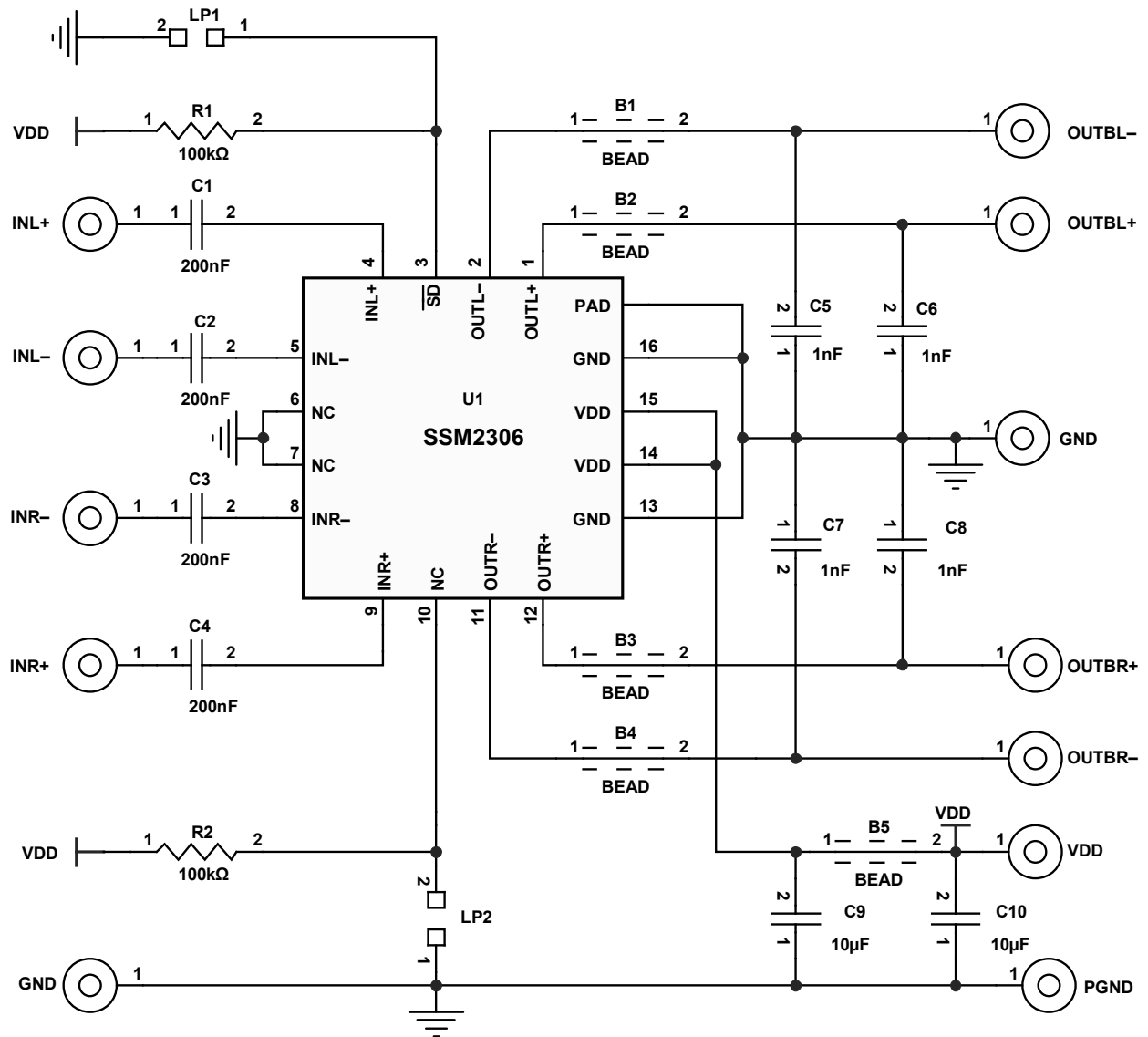


Figure 3. Schematic of the SSM2306 Mini Evaluation Board

07577-003

EVAL-SSM2306-MINI

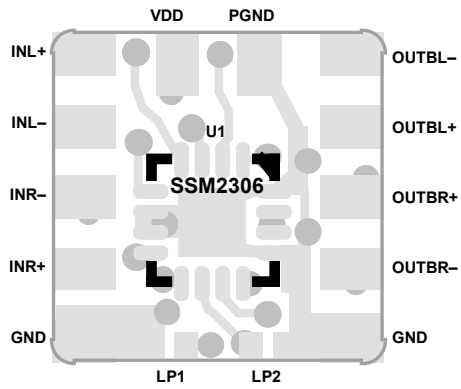


Figure 4. Top Layers of the Evaluation Board

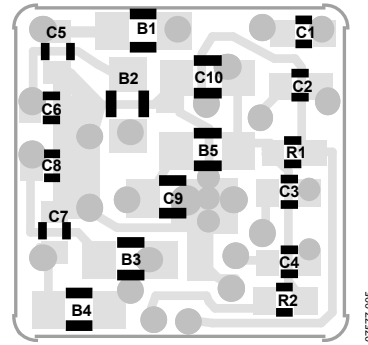


Figure 5. Bottom Layers of the Evaluation Board

ORDERING INFORMATION

ORDERING GUIDE

Model	Description
SSM2306-MINI-EVALZ ¹	Evaluation Board without DIP switch

¹ Z = RoHS Compliant Part.

ESD CAUTION



ESD (electrostatic discharge) sensitive device. Charged devices and circuit boards can discharge without detection. Although this product features patented or proprietary protection circuitry, damage may occur on devices subjected to high energy ESD. Therefore, proper ESD precautions should be taken to avoid performance degradation or loss of functionality.

NOTES