Evaluation Board for Mono 2.8 W Class-D Audio Amplifier

EVAL-SSM2305

FEATURES

Single-ended and differential input capability
User-friendly interface connection
Optimized EMI suppression filter assembled on board

GENERAL DESCRIPTION

The SSM2305 is a fully integrated, single-chip, mono Class-D audio amplifier that is designed to maximize performance for mobile phone applications. The application circuit operates from a single 2.5 V to 5.5 V supply and is capable of delivering 2.8 W of continuous output power with less than 10% THD + N driving a 4 Ω load from a single 5.0 V supply.

The SSM2305 comes with a differential mode input port and a high efficiency, full H-bridge at the output that enables direct coupling of the audio power signal to the loudspeaker. The differential mode input stage allows for canceling of common-mode noise.

The part also features a high efficiency, low noise output modulation scheme that requires no 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 the nonlinearities of output LC filters.

This data sheet describes how to configure and use the SSM2305 evaluation board to test the SSM2305. It is recommended that this data sheet be read in conjunction with the SSM2305 data sheet, which provides more detailed information about the specifications, internal block diagrams, and application guidance for the amplifier IC.

EVALUATION BOARD DESCRIPTION

The SSM2305 evaluation board carries a complete application circuit for driving a loudspeaker. Figure 1 shows the top view of the evaluation board, and Figure 2 shows the bottom view.

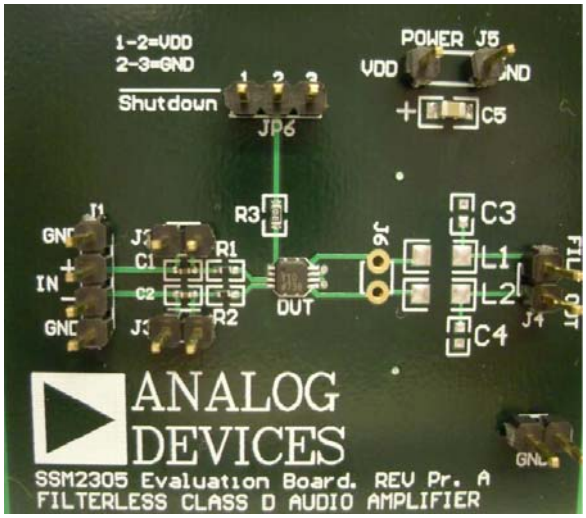


Figure 1. SSM2305 Evaluation Board, Top View

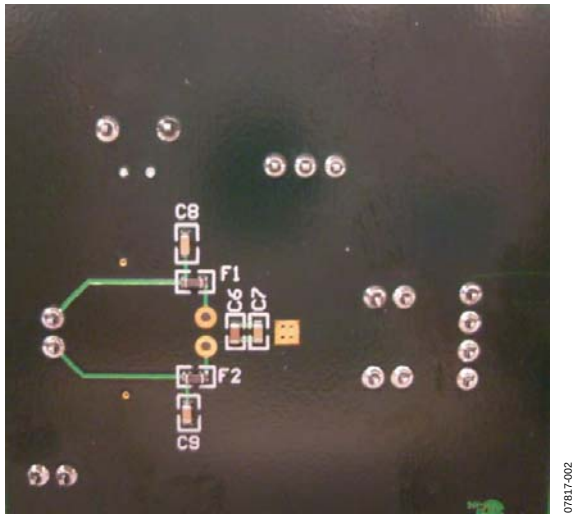


Figure 2. SSM2305 Evaluation Board, Bottom View

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.

One Technology Way, P.O. Box 9106, Norwood, MA 02062-9106, U.S.A.
Tel: 781.329.4700 **www.analog.com**
Fax: 781.461.3113 **©2009 Analog Devices, Inc. All rights reserved.**

TABLE OF CONTENTS

Features 1
General Description 1
Evaluation Board Description 1
Revision History 2
Evaluation Board Hardware 3
Input Configuration 3
Output Configuration 3
Power Supply Configuration 3
Component Selection 3
PCB Layout Guidelines 4
Getting Started 5
What to Test 5
Evaluation Board Schematic and Artwork 6
Ordering Information 8
Bill of Materials 8
Ordering Guide 8
ESD Caution 8

REVISION HISTORY

5/09—Revision 0: Initial Version

EVALUATION BOARD HARDWARE

INPUT CONFIGURATION

On the left side of the PCB is a 4-pin header (J1). This is used to feed the audio signal into the board. If the input audio signal is differential (IN+ and IN−), use the two center pins of J1. In this case, either the top or bottom pin should be connected to the source/signal ground.

For a single-ended audio input only two pins of J1 are used. One pin is for the ground and the other pin is for either IN+ or IN−. If IN+ is used, place a jumper between Pin 3 and Pin 4 of J1, shorting IN− to ground. If IN− is used, place the jumper between Pin 1 and Pin 2 of J1, connecting IN+ to the ground.

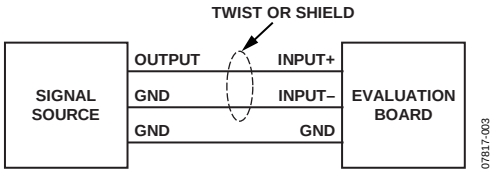


Figure 3. Cancelling Ground Noise by Differential Input Mode

The SSM2305 evaluation board has two ground connection header pins adjacent to the audio input terminals. One valuable use for these headers is for connection to the ground of the signal source, as indicated in Figure 3. This is the proper technique for cancelling ground noise in differential input mode.

The 3-pin header, JP6, is used to turn the SSM2335 amplifier on or off. Placing a jumper across Pin 2 and Pin 3 of JP6 shuts down the SSM2335 so that only a minimum current (about 20 nA) is drawn from the power supply. Placing the jumper across Pin 1 and Pin 2 of JP6 puts the SSM2335 in normal operating mode.

OUTPUT CONFIGURATION

To connect the output ports (located on the right side of the board) to a load, use the J4 header port. J4 is located on the far right side of the board after the signal path of the optional LC (top) or ferrite bead (bottom) EMI filters. A direct connection to the SSM2305 is established on J6. If EMI filtering is used, insert only one style of EMI filtering on the board.

The SSM2305 does not require any external LC output filters due to a low noise modulation scheme. However, if the speaker length is >10 cm, it is recommended that a ferrite bead (F1 and F2) be placed near each output pin of the SSM2305 to reduce electromagnetic interference (EMI), as shown in the schematic in Figure 7.

On the board, there are two inductors, L1 and L2, that are not loaded and are not required for normal operation. Some users may replace the ferrite beads with these inductors to evaluate applications with specific EMI vs. audio performance constraints. If L1 and L1 are loaded, the ferrite beads F1 and F2 must be removed.

As an aid, a properly tuned ferrite bead based EMI filter is assembled at the output terminals of the device. For optimal performance, as specified in the SSM2305 data sheet (in particular, for THD and SNR), remove the entire EMI filter, short across the ferrite bead terminals, and open the capacitor terminals.

POWER SUPPLY CONFIGURATION

The upper right section of the SSM2305 evaluation board has a power supply terminal, J5 (see Figure 8). The SSM2305 can operate with a 2.7 V to 5.5 V supply voltage. The maximum peak current is approx-imately 1.2 A when driving a 4 Ω load and 0.6 A when driving an 8 Ω load, when the input voltage is 5 V. Care must be taken to connect the dc power with correct polarity and voltage.

Polarity and Voltage

The wrong power supply polarity or overvoltage may permanently damage the board.

COMPONENT SELECTION

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

Input Coupling Capacitor Selection—C1 and C2

The C1 and C2 capacitors should be large enough to couple the low frequency components in the incoming signal but small enough to filter out unnecessary lower frequency signals. For music signals, the cutoff frequency is, typically, between 20Hz and 80Hz. The cutoff frequency is calculated by

$$C = 1/(2\pi Rf_c),$$

where:
R = 37 kΩ + R_{EXT} (the external resistor used to fine-tune the desired gain; on the schematic (see Figure 7), this is the 0 Ω resistor at the input pins).
f_c is the cutoff frequency.

Input Gain Setting Resistor Selection—R1 and R2

The R1 and R2 resistors can be used to drop the gain lower than the default 18 dB setting. The evaluation board, by default, is populated with 0 Ω resistors. You must remember to factor the extra resistance of the inserted series resistors when calculating the cutoff frequency. Using external resistors, the gain is calculated by

$$Gain = 296\ k\Omega/(37\ k\Omega + R_{EXT})$$

Output Ferrite Beads—F1 and F2

The F1 and F2 output beads 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.

Table 1. Recommended Output Beads (L1 and L2)

Part No.	Manufacturer	Z (Ω)	I_{MAX} (mA)	DCR (Ω)	Size (mm)
BLM18PG121SN1D	Murata	120	2000	0.05	1.6 × 0.8 × 0.8
MPZ1608S101A	TDK	100	3000	0.03	1.6 × 0.8 × 0.8
MPZ1608S221A	TDK	220	2000	0.05	1.6 × 0.8 × 0.8
BLM18EG221SN1D	Murata	220	2000	0.05	1.6 × 0.8 × 0.8

Table 2. Recommended Output Inductors (L3 and L4)

Part No.	Manufacturer	L (µH)	I_{MAX} (mA)	DCR (Ω)	Size (mm)
LQM31PNR47M00	Murata	0.47	1400	0.07	3.2 × 1.6 × 0.85
LQM31PN1R0M00	Murata	1.0	1200	0.12	3.2 × 1.6 × 0.85
LQM21PNR47MC0	Murata	0.47	1100	0.12	2.0 × 1.25 × 0.5
LQM21PN1R0MC0	Murata	1.0	800	0.19	2.0 × 1.25 × 0.5
LQH32CN2R2M53	Murata	2.2	790	0.1	3.2 × 2.5 × 1.55

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 420 mA, and impedance at 100 MHz must be ≥120 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.

Output Shunting Capacitors—C3, C4, C8 and C9

There are two output shunting capacitors, C8 and C9, that work with the F1 and F2 ferrite beads. There are also two output shunting capacitors, C3 and C4, that work with the L1 and L2 inductors, if they are used. Use small size (0603 or 0402) multilayer ceramic capacitors that are made of X7R or COG (NPO) materials. Note that the capacitors can be used in pairs: a capacitor with small capacitance (up to 100 pF) plus a capacitor with a larger capacitance (less than 1 nF). This configuration provides thorough EMI reduction for the entire frequency spectrum. For BOM cost reduction and capable performance, a single capacitor of approximately 470 pF can be used.

Output Inductors

If you prefer using inductors for the purpose of EMI filtering at the output nodes, choose inductance that is <2.2 µH. The higher the inductance, the lower the EMI is at the output. However, the cost and power consumption by the inductors are higher. Using 0.47 µH to 2.2 µH inductors is recommended, and the current rating needs >600 mA (satur-ation current) for an 8 Ω load. Table 2 shows the recommended inductors. Note that these inductors are not populated on the evaluation board.

PCB LAYOUT GUIDELINES

To lay out the board, do the following:

1. In the LFCSP package, place at least nine vias on the solder pad for the thermal pad of the amplifier for proper conduction of heat to the opposite side of the board.

   The outer diameter of the vias should be 0.5 mm, and the inner diameter should be 0.25 mm to 0.3 mm.

2. Use a PCB area of at least 2 cm^2 equivalent area on the opposite side of the layer of the amplifier chip as a heat sink. Also, extend the ground pad (for the thermal pad) as much as possible on the amplifier side of the PCB as a heat sink (see Figure 4).
3. If internal layers are available, allocate a certain area as a heat sink; make sure to connect the vias conducting the heat to the internal layers.

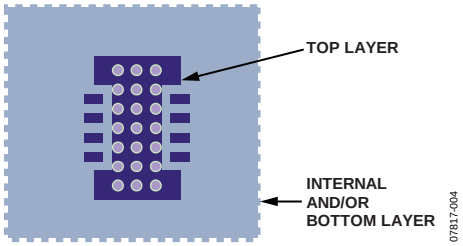


Figure 4. Heat Sink Layout

4. Place the EMI filtering beads, F1 and F2, as close to the amplifier chip as possible.
5. Place the decoupling capacitors for the C8 and C9 beads as close to the amplifier chip as possible, and connect all their ground terminals together, as close as possible.

   Ideally, solder their ground terminals together, as shown in Figure 5. Do not rely on PCB tracks or ground planes for connecting these ground terminals.

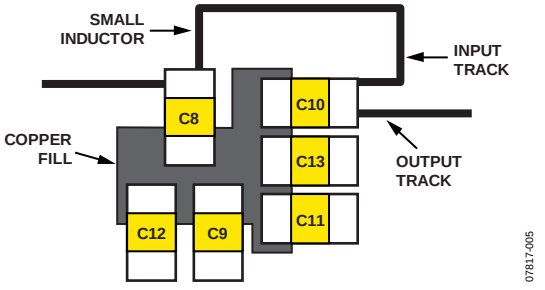


Figure 5. Correct Placement and Routing for Decoupling Capacitors

6. A 1 nF capacitor and the ferrite bead can block the EMI for up to 250 MHz. To eliminate EMI higher than 250 MHz, place a low value, small size capacitor, such as a 100 pF, 0402 size capacitor, in parallel with the 1 nF decoupling capacitor.

   Place this small capacitor a small distance away from the 1 nF capacitor and use the PCB connection track as an inductor to form a PI shape, low-pass filter, as shown in Figure 5.

7. If implementing a PCB track PI filter, do not connect the arriving input PCB track and the leaving output PCB track to the decoupling capacitor.

   The correct layout is shown in Figure 5. The incorrect layout is shown in Figure 6.

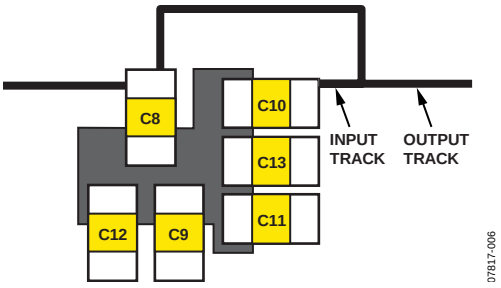


Figure 6. Incorrect Routing for the Inductive Track Output Decoupling Capacitor

8. Decouple the input port nodes with small capacitors, such as 100 pF capacitors. They are not necessary but can lower the input EMI.

GETTING STARTED

To ensure proper operation, carefully follow Step 1 through Step 4.

1. Ensure proper connection of the input terminals. For single-ended input, be sure to ground the unused audio input terminal; otherwise, you will encounter poor audio quality.
2. To activate the SSM2305, place a jumper across Pin 1 and Pin 2 (the left and center terminals) of JP6 and make sure that the SD pin is pulled up to VDD.
3. Connect the load across the output ports.
4. Connect the power supply with the correct polarity and proper voltage.

WHAT TO TEST

When implementing the SSM2305 evaluation board, test the board for the following:

- Electromagnetic interference (EMI)—connect wires for the speakers, making sure they are the same length as the wires required for the actual application environment; then complete the EMI test.
- Signal-to-noise ratio (SNR).
- Output noise—make sure to use an A-weighted filter to filter the output before reading the measurement meter.
- Maximum output power.
- Distortion.
- Efficiency.

EVALUATION BOARD SCHEMATIC AND ARTWORK

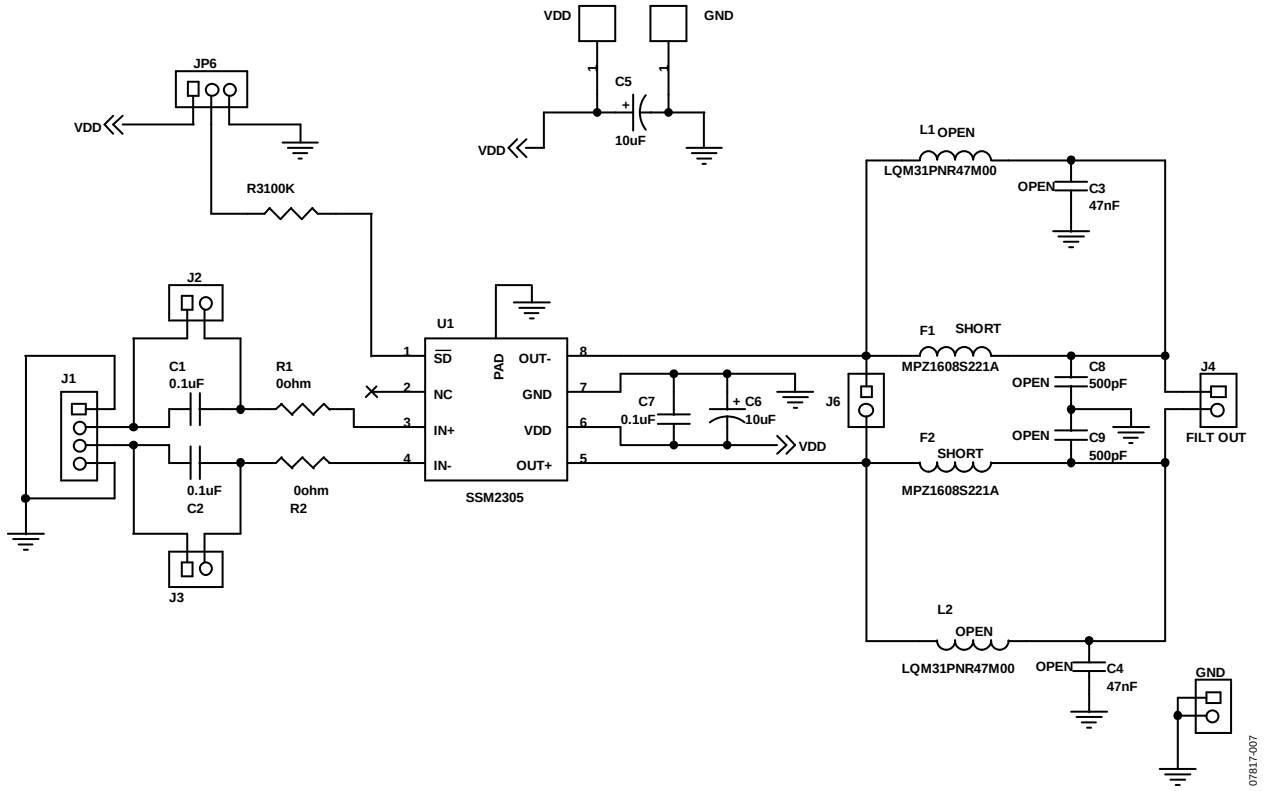


Figure 7. Schematic of the SSM2305 Evaluation Board

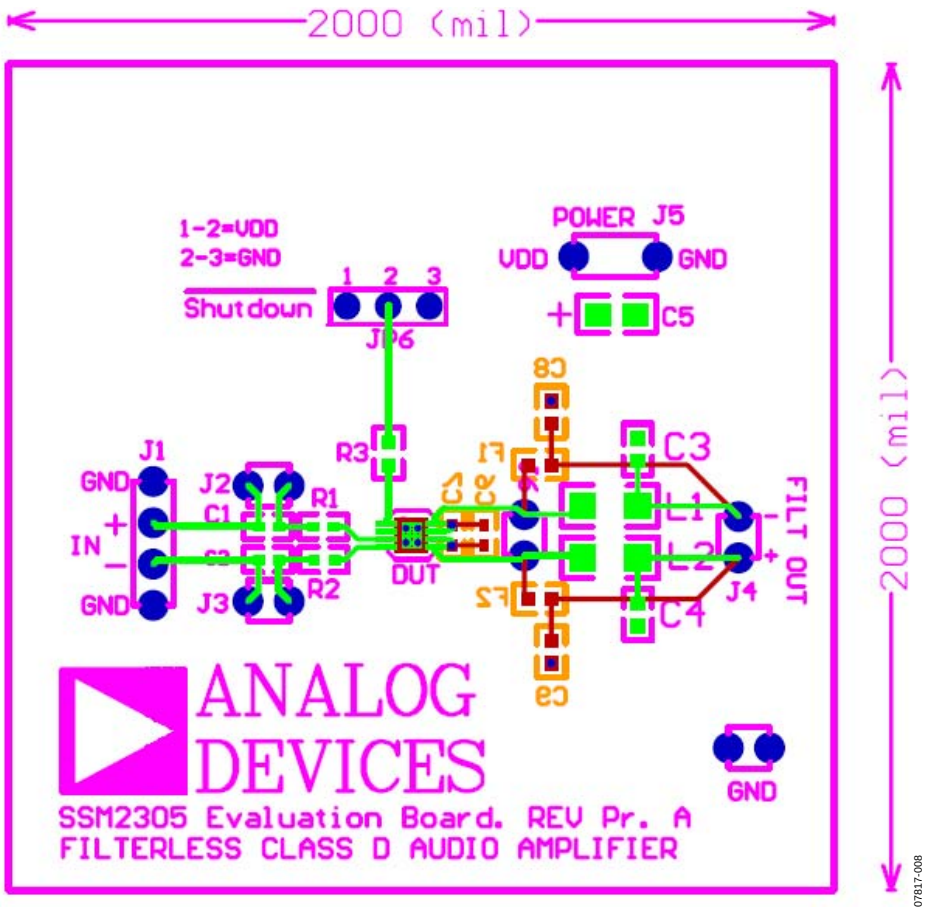


Figure 8. Layout of the SSM2305 Evaluation Board

ORDERING INFORMATION

BILL OF MATERIALS

Table 3.

Qty.	Reference Designator	Description	Supplier/Part No.
2	C3, C4	Ceramic capacitor, 1 nf, 10%, 5 V	Kemet, C0603C102J5GACTU
2	L1, L2	Inductor, 0.47 µH	Murata, LQM31PNR47M00
2	C1, C2	Ceramic capacitor, 0.1 µF	Panasonic, ECJ-1VB1C104K
4	J2, J3, J4, J5, J6	100 mil straight header, JUMPER100	Tyco, 4-103747-0-01
1	C5	Ceramic capacitor, 10 µF	Murata, GRM21BR61A106KE19L
1	C6	Ceramic capacitor, 10 µF	Panasonic, ECJ-1VB0J104M
1	R3	Resistor, 100 kΩ	Panasonic, ERJ-3EKF1003V
1	JP6	100 mil straight header, SIP 3 × 2	Tyco, 4-103747-0-03
1	U1	SSM2305, LFCSP 3 × 3 8LD	Analog Devices, SSM2305
1	C7	Ceramic capacitor, 0.1 µF	Panasonic, ECJ-1VB1C104K
2	R1, R2	Resistor, 0 Ω	Yaego, 9C06031A0R00JLHFT
2	F1, F2	Ferrite bead 220 Ω, 2 A, 0603	TDK, MPZ1608S221A
2	C8, C9	Ceramic capacitor, 510 pF	Murata, GRM1885C2A511JA01D
1	J1 (IN)	100 mil straight header, HEADER4PIN	Tyco, 640452-4

ORDERING GUIDE

Model	Description
SSM2305-EVALZ[1]	Evaluation Board

[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.

©2009 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners.
EB07817-0-05/09(0)