

### FEATURES

- Highly configurable DIP switch settings: gain adjustment, input configuration, and shutdown control**
- Optional dc power supply jack (accepts 2.5 V to 5 V)**
- Single-ended and differential input capability**
- Multiple output interface connection terminals**

### GENERAL DESCRIPTION

The *SSM2302* is a single-chip, single-channel, stereo Class-D 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 of 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 cancelling 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 providing a better sound quality and leading to savings in board space and overall cost.

This data sheet describes how to configure and use the *SSM2302* evaluation board. It is recommended that this data sheet be read in conjunction with the *SSM2302* data sheet, which provides more detailed information about the specifications, internal block diagrams, and application guidance for the amplifier IC.

### EVALUATION BOARD DESCRIPTION

The *SSM2302* evaluation board carries a complete application circuit for driving two loudspeakers. Figure 1 shows the top view of the PCB and Figure 2 shows the bottom view of the evaluation board. Figure 9 shows its layout. The silkscreen layer of the evaluation board is shown in Figure 7 with other top layers, including top copper, top solder mask and multilayer vias. Figure 8 shows the top silkscreen layer only. There is no component in the bottom side. Therefore, there is no bottom silkscreen layer. Figure 9 shows the top layers without the silkscreen layer. Figure 10 shows the bottom layers, including bottom copper, bottom solder mask, and multilayer vias. Figure 11 shows the mirrored bottom layers. The bill of materials is shown in Table 4.

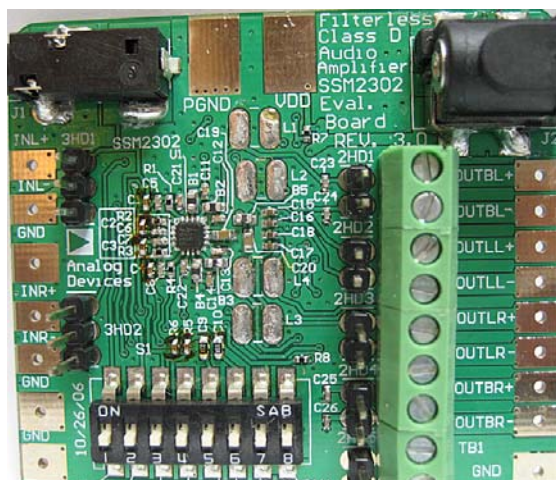


Figure 1. SSM2302 Evaluation Board Top View

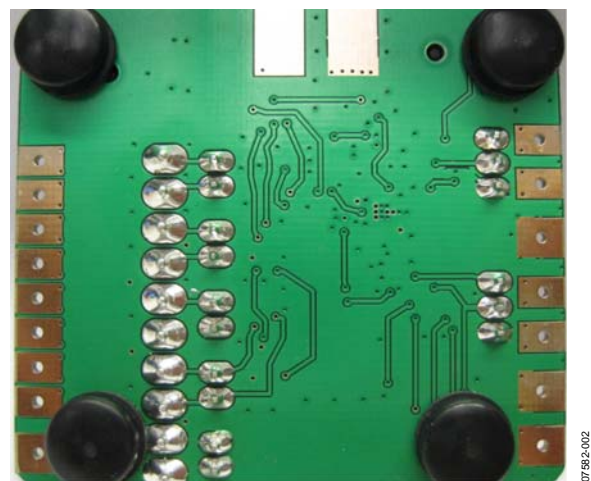


Figure 2. SSM2302 Evaluation Board Bottom View

### 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.

## TABLE OF CONTENTS

Features .....	1	Component Selections.....	4
General Description .....	1	Layout Guidelines.....	5
Evaluation Board Description.....	1	Evaluation Board Schematic and Artwork.....	6
Revision History .....	2	Ordering Information.....	8
Evaluation Board Hardware.....	3	Bill of Materials.....	8
Switches.....	3	Ordering Guide .....	8
Getting Started.....	3	ESD Caution.....	8
What to Test .....	4		

## REVISION HISTORY

8/08—Revision 0: Initial Version

## EVALUATION BOARD HARDWARE

Note that the SSM2302 evaluation board and layout guidelines were developed by Gang Liu of Analog Technologies, Inc., Sunnyvale, CA.

### SWITCHES

On the upper left corner of the evaluation board is an audio stereo jack connector (3.5 mm), J1. It accepts standard stereo audio signals by using a conventional audio stereo signal connector/cable to receive audio signals from common appliances such as DVD players, personal computers, and TVs. Note that the two output signals from the J1 connector are combined by a resistor network, R1 and R2, into a single signal. When this input connector is utilized, turn the switches, S1E and S1F, to the upper positions, to ac short circuit the negative input ports to ground; this connector only provides single-ended audio signals. This signal path is shown in the schematic in Figure 6. Some appliances may have strong ground noise. In the event that the audio source has strong ground noise, do not use the J1 connector as the input connection point. A single-ended input signal with strong ground noise generates an audible hissing sound at the output. When this happens, differential mode connection is needed and the ground noise interference can be cancelled.

When differential mode audio signals are used as the input signal source, use either the 3HD1 and 3HD2 headers or the soldering pads located on the left side of the board. The top header is for the left channel signals and the bottom header is for the right channel. At the same time, turn the S1E and S1F switches to their off positions (the lower position).

There are two ground soldering pads on the lower left corner.

On the lower side of the board is a switch bank; the corresponding channels are shown in Table 1.

**Table 1. Switch Pin Nomenclature**

Switch Name	Corresponding Channel
S1A	AC couple left channel noninverting input
S1B	AC couple left channel inverting input
S1C	AC couple right channel inverting input
S1D	AC couple right channel noninverting input
S1E	AC ground left channel inverting input
S1F	AC ground right channel inverting input
S1G	Gain control pin
S1H	Shutdown

When the S1A, S1B, S1C, and S1D switches are placed in the up position, their corresponding coupling capacitors are shorted; when these switches are placed in the down position, the coupling capacitors are inserted in the signal paths.

The switches, S1E and S1F, as briefly described previously, are used to ac short circuit the left and right channel negative input ports, respectively, to ground. This function is generally recommended, but only necessary when driving the input ports in single-ended mode. After shorting the negative input ports to ground, the noise picked up by the input port connections is conducted to the ground.

S1G controls the gain. Place S1G in the upper position for the higher gain and place it in the lower position for the lower gain.

S1H is for the shutdown function. The upper position is for shutdown and the lower position turns the amplifier on.

The upper right corner has a dc power jack connector. The center pin is for the positive terminal, which accepts 3 V to 5 V. When the input voltage is 5 V, the maximum peak current is approximately 0.6 A when driving an 8  $\Omega$  load.

There are two solder pads in the upper center edge area for connecting the power supply voltages by clipping or soldering.

All the output ports are located on the right side of the board and marked properly with labels. See the legend on the evaluation board in Figure 1 and the schematic in Figure 6.

There are three ways to connect the output signals to the loads (the loudspeakers): using the four 2-pin headers, the terminal block, or the soldering pads.

### GETTING STARTED

To ensure proper operation, follow these steps carefully:

- Verify that the control switches are at the proper positions.
  - Shutdown control: S1H down. This activates the amplifier.
  - Gain setting: S1G. Select the position based on the gain needed.
  - Audio source: S1E and S1F. For single-ended mode (audio jack), put S1E and S1F in the upper position.
  - Input coupling: S1A, S1B, S1C, and S1D. For most applications, ac couple the inputs by setting these switches in the lower position.
- Connect the loads to the proper output ports. Depending on the application, different nodes should be used. To connect the loads right after the beads, use the OUTBL+, OUTBL-, OUTBR+, and OUTBR- nodes. To connect the loads after the inductors, use the OUTLL+, OUTLL-, OUTLR+, and OUTLR- nodes.
- Connect the power supply with the correct polarity and proper voltage.

# EVAL-SSM2302

## WHAT TO TEST

The EVAL-SSM2302 tests the following:

- Electromagnetic interference (EMI). Connect wires for the speakers in the same length as for the actual application situation and complete the EMI test.
- Signal-to-noise ratio.
- Total harmonic distortion (THD).
- Output noise. Make sure to use an A-weighted filter to filter the output before the measurement meter.
- Maximum output power.
- Efficiency.

## COMPONENT SELECTIONS

Selecting the right components is the key to achieving the performance required at the cost budgeted. The bill of materials for the SSM2302 evaluation board is shown in Table 4.

### Input Coupling Capacitor Selection—C1, C2, C3, and C4

The 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 cutoff frequency is often chosen between 20 Hz and 50 Hz. The value of the input capacitor is calculated by

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

where:

$$R = 150 \text{ k}\Omega + R_{\text{EXT.}}$$

$f_c$  is the cutoff frequency.

### Input Serial Resistors—R1, R2, R3, and R4

R1, R2, R3, and R4 are not necessary for amplifier operation and are only needed when special gain values are required. To determine the proper gain value, use the following formula:

$$\text{External Gain Settings} = 20 \log [2/(1 + R/150 \text{ k}\Omega)]$$

## Warning

Using large value resistors can increase the input noise.

Table 2. Recommended Output Bead

Part No.	Manufacturer	Z ( $\Omega$ )	I <sub>MAX</sub> (mA)	DCR ( $\Omega$ )	Size (mm)
BLM18EG221SN1	Murata	220	2000	0.05	1.6 × 0.8 × 0.8

Table 3. Recommended Output Inductors

Part No.	Manufacturer	L ( $\mu$ H)	I <sub>MAX</sub> (mA)	DCR ( $\Omega$ )	Size (mm)
LQH32CN4R7M53	Murata	4.7	650	0.15	3.2 × 2.5 × 1.55
LQH32CN3R3M53	Murata	3.3	710	0.12	3.2 × 2.5 × 1.55
LQH32CN2R2M53	Murata	2.2	790	0.1	3.2 × 2.5 × 1.55
SD3118-100-R	Cooper Bussmann	10	900	0.3	3.1 × 3.1 × 1.8
ELL4LM100M	Panasonic	10	690	0.18	3.8 × 3.8 × 1.8
LBC2518T2R2M	Taiyo Yuden	2.2	630	0.13	2.5 × 1.8 × 2
1033AS-4R7M	Toko	4.7	680	0.31	3.8 × 3.8 × 1

### Output Beads—B1, B2, B3, and B4

B1, B2, B3, and B4 are necessary components for filtering out the EMIs caused at the switching output nodes. Make sure that these beads have enough current conducting capability while providing sufficient EMI attenuation. The current rating needed for an 8  $\Omega$  load is about 600 mA, and the impedance at 100 MHz must be  $\geq 220 \Omega$ . In addition, the lower the dc resistance (DCR) of these beads, the better for minimizing their power consumptions. The recommended bead is detailed in Table 2.

### Output Shunting Capacitors for the Beads

There are two groups of output shunting capacitors. C11, C12, C13, and C14 are for filtering out the lower frequency EMIs of up to 250 MHz. C23, C24, C25, and C26 are for filtering out the higher frequency EMIs of >250 MHz. Use small size (0603 or 0402) multilayer ceramic capacitors that are made of X7R or COG (NPO) 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. Using 500 pF to 1 nF values for the first group of capacitors and using 100 pF to 200 pF for the second group of capacitors is recommended.

### Output Inductors

Some users do not want high frequency EMIs in their systems and prefer to use inductors to filter out the high frequency components at the output node. Select inductance >2.2  $\mu$ H for these inductors. The higher the inductance, the lower the EMI at the output, and the lower the quiescent current at the power supply, but the higher the power consumption by the inductors when the output power level is high. Using 2.2  $\mu$ H to 10  $\mu$ H inductors is recommended and the current rating needs >600 mA (saturation current) for an 8  $\Omega$  load. Table 3 shows the recommended inductors. Note that these inductors are not populated on the evaluation board.

## LAYOUT GUIDELINES

To keep the EMI under the allowable limit and to ensure that the amplifier chip operates under the temperature limit, PCB layout is critical in application designs. Follow these guidelines carefully when setting up the PCB layout designs.

- 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 is 0.33 mm. If there are internal layers available in the PCB, allocate an area as large as possible as the ground planes and let these vias be connected to the planes (see Figure 3). If internal layers are available, allocate a certain area as a heat sink, and make sure to connect the vias conducting the heat to the internal layers.
- Place the EMI filtering beads, B1, B2, B3, B4, and B5, as close to the amplifier chip as possible. The same principle applies to the output inductors, L1, L2, L3, and L4, if they exist in the application design.
- Place the decoupling capacitors for the beads, C11, C12, C13, C14, C23, C24, C25, and C26, as close to the amplifier chip as possible, and connect all their ground terminals together as close as possible, as shown in Figure 4. The same principle applies to the decoupling capacitors for the inductors, C15, C16, C17, and C18, if they exist in the application design. Ideally, solder their ground terminals together; do not rely on PCB traces or ground planes for connecting their ground terminals together.
- Place the decoupling capacitor for the power supply, C19, as close to the amplifier chip as possible, as shown in Figure 4, and connect its ground terminal directly to the IC ground pins, Pin 13 and Pin 16.
- Place the decoupling capacitor for the power supply, C20, as close to the amplifier chip as possible, and connect the ground terminal to the PCB ground plane where it shares a common ground with the power supply.
- 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 where the chip stays.
- The 1 nF capacitor and the ferrite bead can block the EMI for up to 250 MHz. To eliminate EMIs higher than the 250 MHz, place a low value small size capacitor, such as 100 pF 0402 size, in parallel with the decoupling capacitors, C11, C12, C13, and C14. However, place this small capacitor a short distance away, at least 20 mm, from the 1 nF decoupling capacitor. Ideally, the ground terminals of these capacitors are connected to the ground terminals or the PCB traces, which are placed as close to the output loads (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, as shown in Figure 4.
- Decouple the input port nodes and the digital pins, Pin 3, Pin 4, Pin 5, Pin 8, Pin 9, and Pin 10, with small capacitors, such as 100 pF. The capacitors are not necessary, but can lower the EMI from these pins. The ground terminals of

these capacitors should be connected to the chip ground as close as possible. See Figure 7, Figure 8, and Figure 9.

- Ground the unconnected pins, Pin 6 and Pin 7.
- Connect the ground pins, Pin 6, Pin 7, Pin 13, and Pin 16, to the thermal pad and place grounding vias, as shown in Figure 7, Figure 8, and Figure 9.
- Use a solid polygon plane on the other side of the PCB for the area of the vias, which are placed on the thermal pad of the chip (see Figure 10 and Figure 11). This polygon serves as both the EMI shielding ground plane and the heat sink for the SSM2302.
- Keep the PCB traces of high EMI nodes on the same side of the PCB and as short as possible. Pin 1, Pin 2, Pin 11, and Pin 12 are the high EMI nodes.
- The incoming and outgoing PCB tracks to the output decoupling capacitors should not be connected to each other. The correct layout is shown in Figure 4. An example of an incorrect layout is shown in Figure 5.

The SSM2302 works well only if these techniques are implemented in the PCB design to keep EMI and the amplifier temperature low.

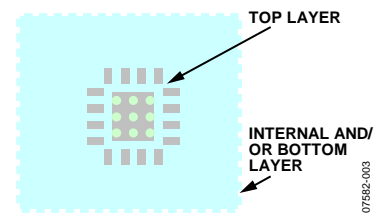


Figure 3. Heat Sink Layout

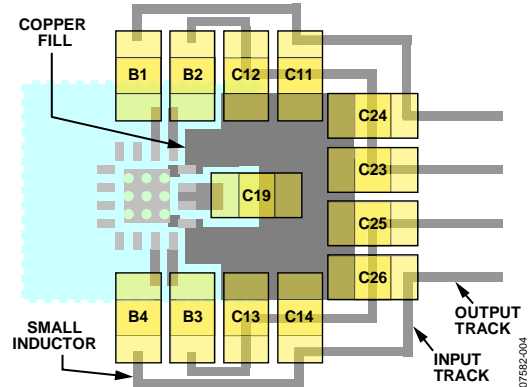


Figure 4. Placement and Routing for the Decoupling Capacitors

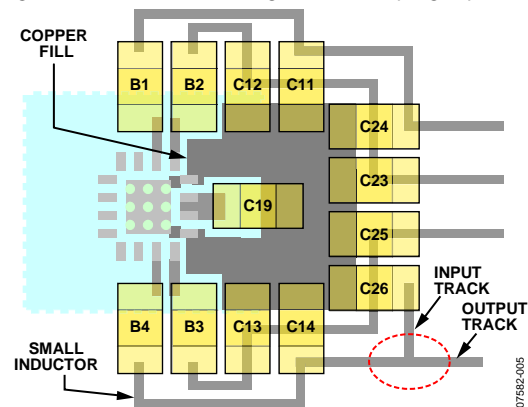


Figure 5. Incorrect Routing for the Output Decoupling Capacitor

EVALUATION BOARD SCHEMATIC AND ARTWORK

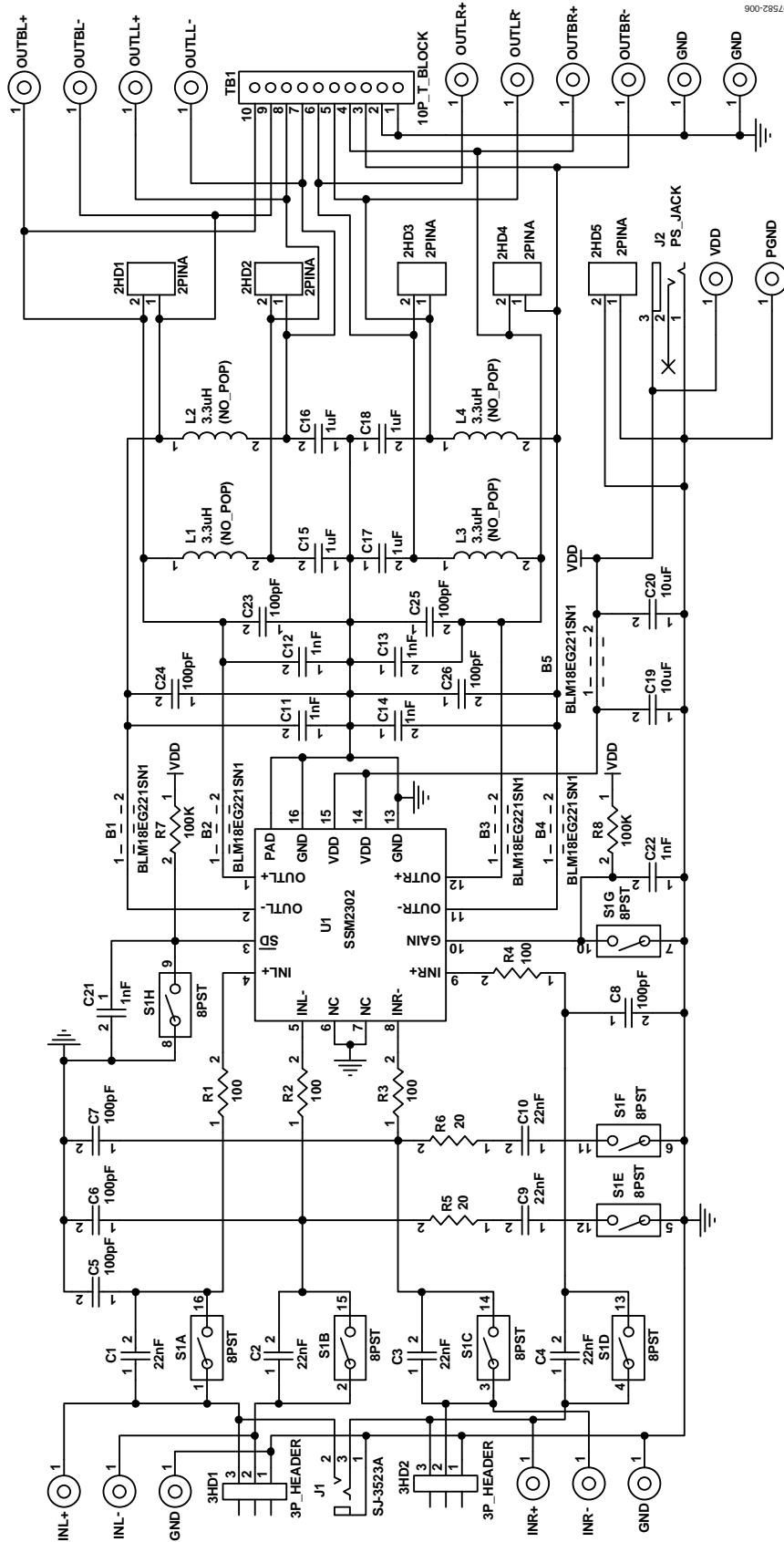


Figure 6. Schematic of SSM2302 Evaluation Board

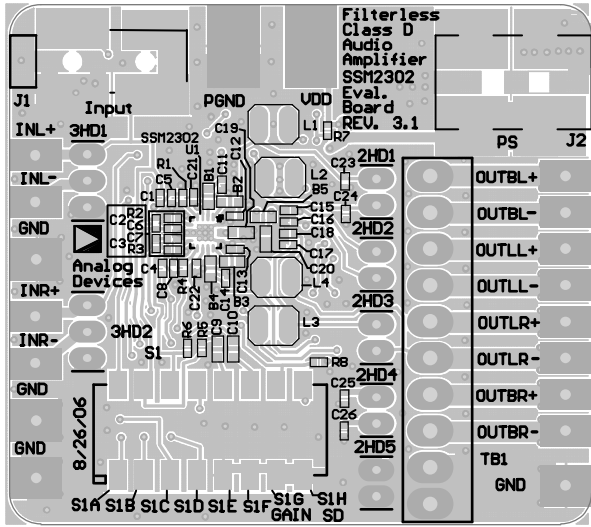


Figure 7. Top Silkscreen Layer with Other Top Layers

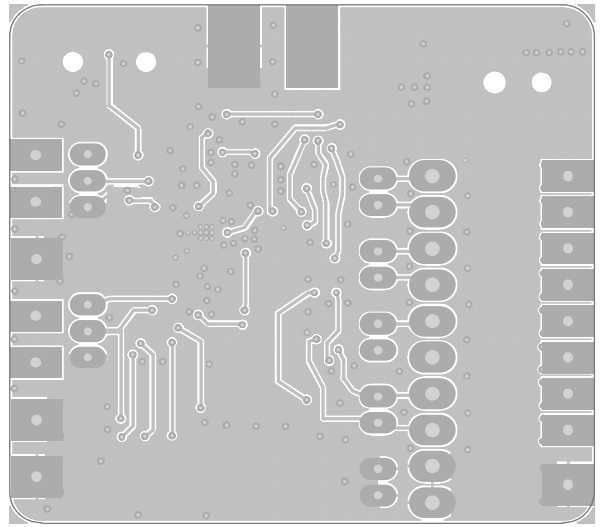


Figure 10. Bottom Layers

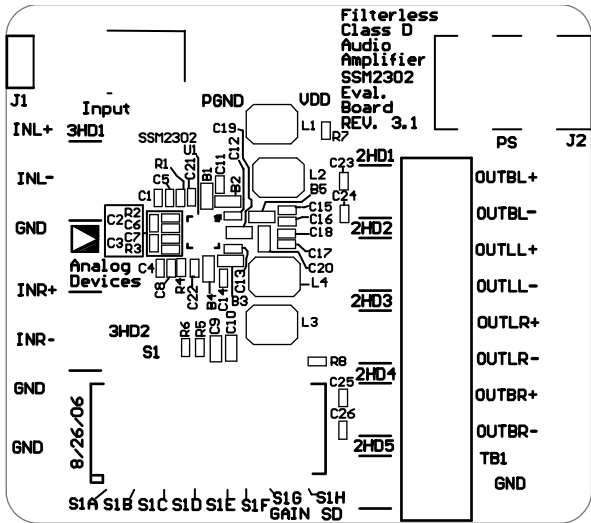


Figure 8. Top Silkscreen

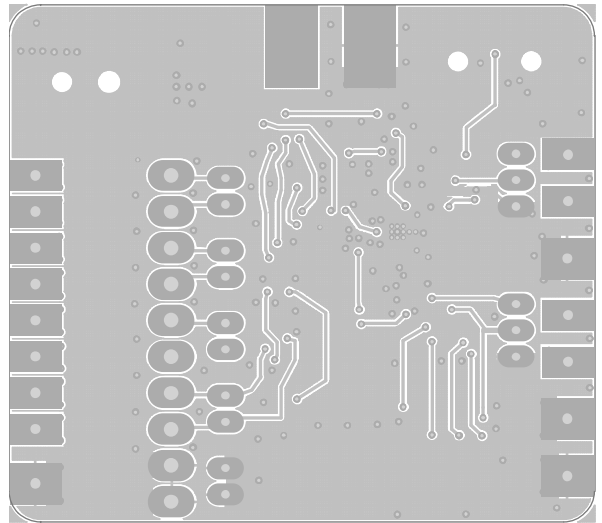


Figure 11. Mirrored Bottom Layers

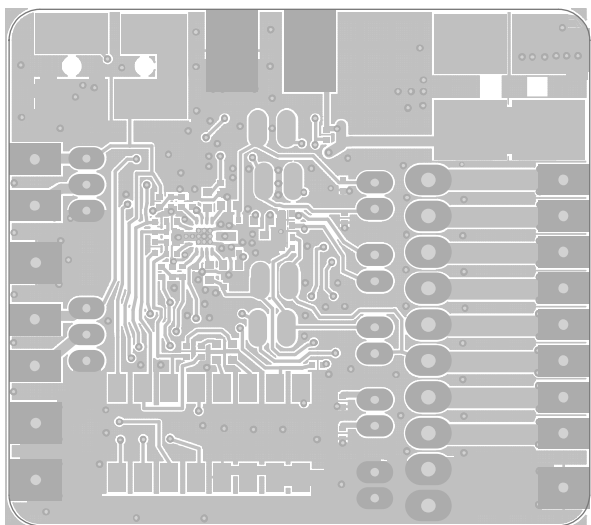


Figure 9. Top Layers Without Top Silkscreen Layer

# EVAL-SSM2302

## ORDERING INFORMATION

### BILL OF MATERIALS

Table 4.

Qty.	Designator	Description	Supplier	Supplier Part No.
4	R1, R2, R3, R4	Resistor, 100 Ω, 1/16 W, 0.5%, 0402 SMD	Digi-Key	RR05P100DTR-ND
2	R5, R6	Resistor, 20 Ω, 1/16 W, 0.5%, 0402 SMD	Digi-Key	RR05R20DTR-ND
2	R7, R8	Resistor, 100 kΩ, 1/16 W, 0.5%, 0402 SMD	Digi-Key	RR05P100DTR-ND
6	C1, C2, C3, C4, C9, C10	Capacitor, ceramic, 22 nF, 50 V, X7R 0402	Digi-Key	490-3884-2-ND
8	C5, C6, C7, C8, C23, C24, C25, C26	Capacitor, ceramic, 100 pF, 6.3 V, X7R 0402	Digi-Key	04026C101KAT2A-ND
6	C11, C12, C13, C14, C21, C22	Capacitor ceramic, 1 nF, 5%, 50 V, X7R 0402	Digi-Key	478-3661-2-ND
4	C15, C16, C17, C18	Capacitor, ceramic, 1 μF, 10 V, X5R 0402	Digi-Key	490-3890-2-ND
2	C19, C20	Capacitor, ceramic, 10 μF, 6.3 V, X5R 0603	Digi-Key	PCC2395TR-ND
4	L1, L2, L3, L4	Inductor, 3.3 μH, 710 mA, 0.12 Ω DCR, 3.2 × 2.5 × 1.55	Murata	LQH32CN3R3M53
5	B1, B2, B3, B4, B5	Filter chip, 220 Ω, 2 A, 0603	Digi-Key	490-3992-2-ND
1	S1	Switch DIP top slide, 8-position, SMD, SDA08H1SBDA	Digi-Key	CKN6092-ND
1	J1	Audio connectors, 3.5 mm, SMT, stereo, black, 3-position, all plastic, SJ-3523A	Mouser	806-STX-3500-3N
1	J2	DC power connector, 2 mm, SMT power jack	Mouser	806-KLDX-SMT2-0202-A
1	U1	Filterless, high efficiency, Class-D, stereo audio amplifier	Analog Devices	SSM2302CPZ
5	2HD1, 2HD2, 2HD3, 2HD4, 2HD5	Connector header, 2-position, 0.100 inch SGL gold	Digi-Key	SAM1029-02-ND
2	3HD1, 3HD2	Connector header, 3-position, 0.100 inch SGL gold	Digi-Key	SAM1029-03-ND
1	TB1	Terminal block, 3.5 mm, 10-position PCB	Digi-Key	ED1522-ND

### ORDERING GUIDE

Model	Description
SSM2302Z-EVAL <sup>1</sup>	Evaluation Board

<sup>1</sup> 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.