

PAM8302A-EV Board User Guide
AE Department

1. Revision Information

Date	Revision	Description	Comment
2008/06/12	V1.0	Initial Release	
2008/11/14	V2.0	PCB Change	

2. EV Board Schematic

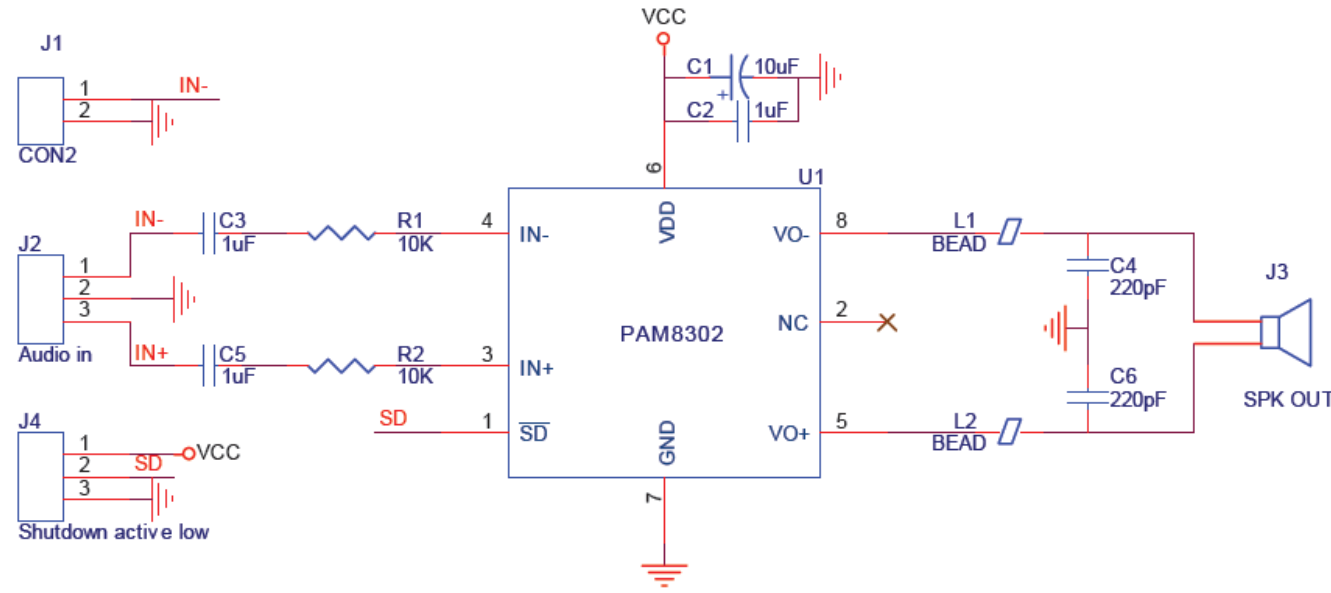


Figure 1 Demo Board Schematic

3. PAM8302A EB85AA Description

PAM8302A CB81AS is design for PAM8302A demo and evaluation, targeted to be used in providing a simple and convenient evaluation environment for the PAM8302A. Requires parts, potentiometer for standard RCA jacks for audio inputs, pin jacks for power supply and signal outputs, low-pass RC output filter for each channel, etc. on the board make it easy to be evaluated.

4. EV Board View

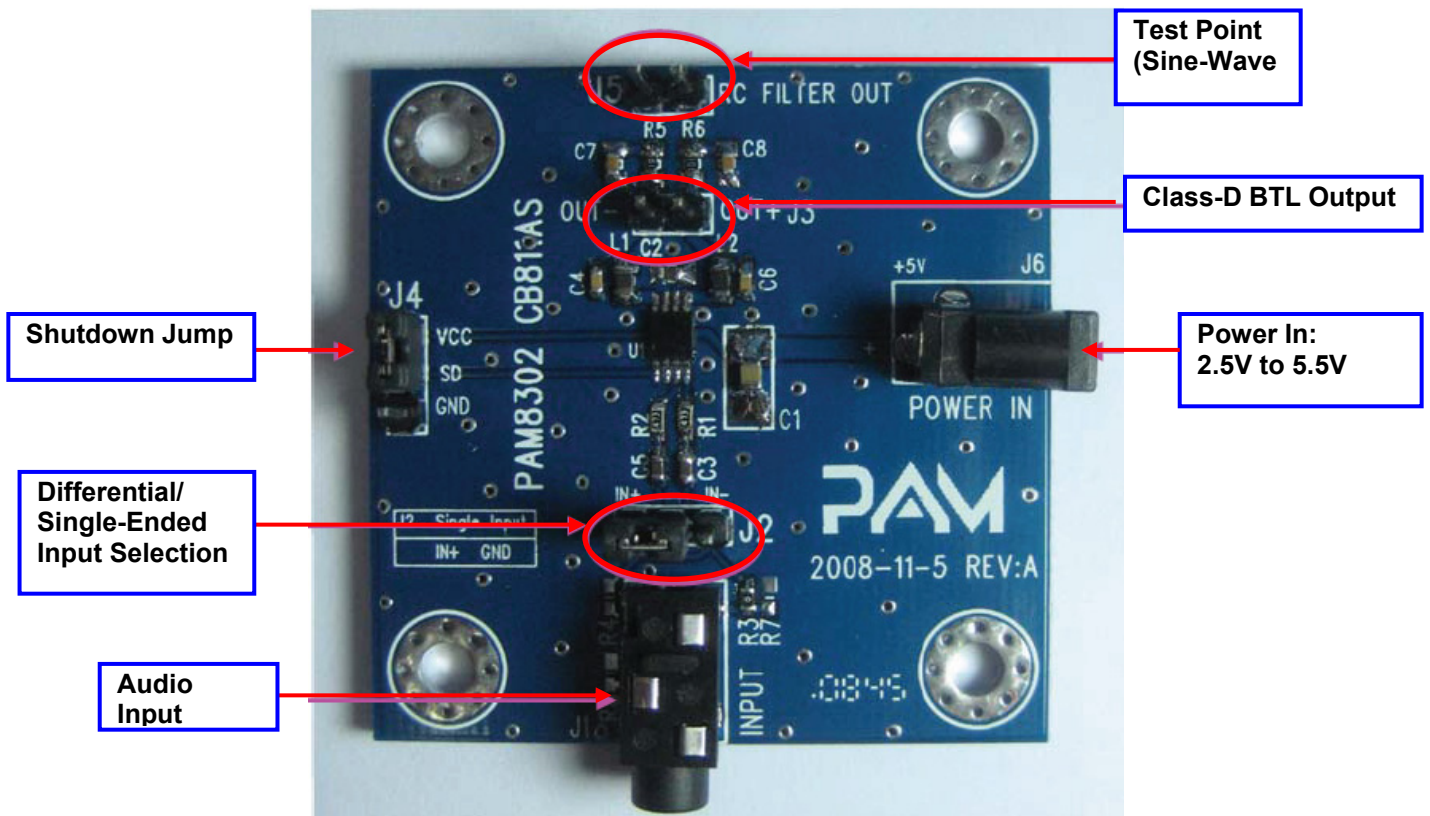


Figure 2 Demo Board Top View

EV Board Operational Sequence:

- Preset the power supply to between 2.5V and 5.5V.
- Connect power supply to EV board power.
- Connect audio input from audio input jack.
- Connect the SPKs to the BTL output jack, 8ohm speaker recommend.
- Turn on the power supply and verify that the sound quality of speaker.

EV Board BOM List

Item	Value	Type	Rating	Description	Vender and Part Number
C3, C5	100nF	X5R/X7R, Ceramic/0603	10V	Input CAP	Murata GRM21R71C104B
C1	10μF	X5R/X7R, Ceramic/1210	10V	VDD coupling CAP	Murata GRM21R71C106B
C2	1μF	X5R/X7R, Ceramic/0603	10V	VDD coupling CAP	Murata GRM21R71C105B
R1, R2	10K		1%	Input Resistor	
Bead1/2	120Ω/1A	0603		EMI	
C4, C6	220pF	X5R/X7R, Ceramic/0603	10V	EMI	

5. External Components Selection

Input Capacitors (C3, C5)

- (1) Form a high pass filter with R_i , and the cut off frequency is $f_c = 1/2\pi R_i C_i$.
- (2) Have a tolerance of 10% or better for matching: any mismatch in capacitance causes an importance mismatch at the corner frequency.
- (3) Low leakage current needed 0.1μF.

Input Resistors (R1, R2)

- (1) Limit the closed-loop gain.
- (2) Form a high pass filter with C_i , and the cut off frequency is $f_c = 1/2\pi R_i C_i$.
- (3) 1% tolerance needed for resistor matching to improve CMRR, PSRR.

Power Supply decoupling Caps (C1, C2)

- (1). Low ESR for good THD, PSRR.
- (2) C2, 1.0μF ceramic for higher frequency transients, spikes.
- (3) C1, Additional 10uF or greater for low frequency noise filtering and serves as a local storage capacitor for supplying current during large signal transients on the amplifier outputs.
- (4) Need place very closed to the IC.

6. PCB Layout Guidelines

Grounding

- (1) Use plane grounding or separate grounds
- (2) Do not use one line connecting power GND and analog GND
- (3) Output noise grounds must tie to system ground at the power in exclusively.
- (4) Signal currents for the inputs need to be returned to quiet ground. This ground only ties to the signal components and the GND pin.

Others

- (1) The power supply de coupling capacitors (C1, C2) need to place very close to the PAM8302A's pins.
- (2) The output route should be far away from audio input route.

7. PCB Layout Example

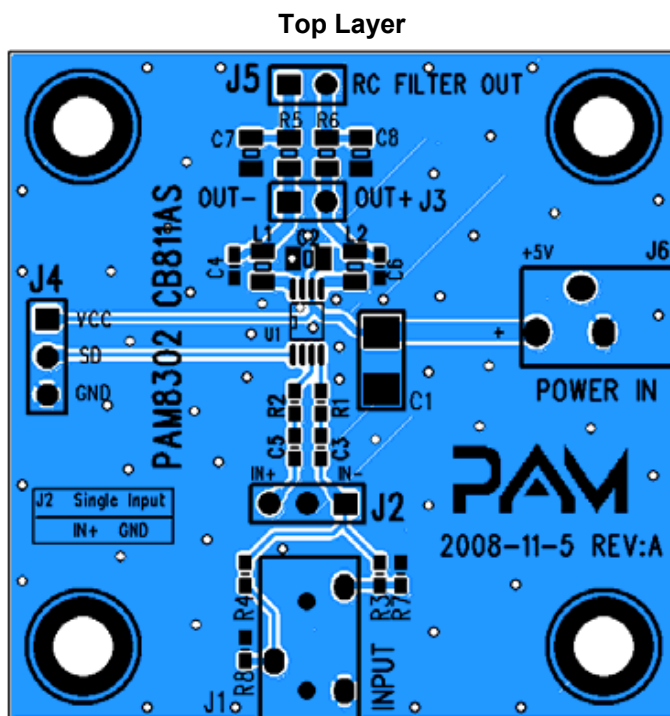


Figure 3

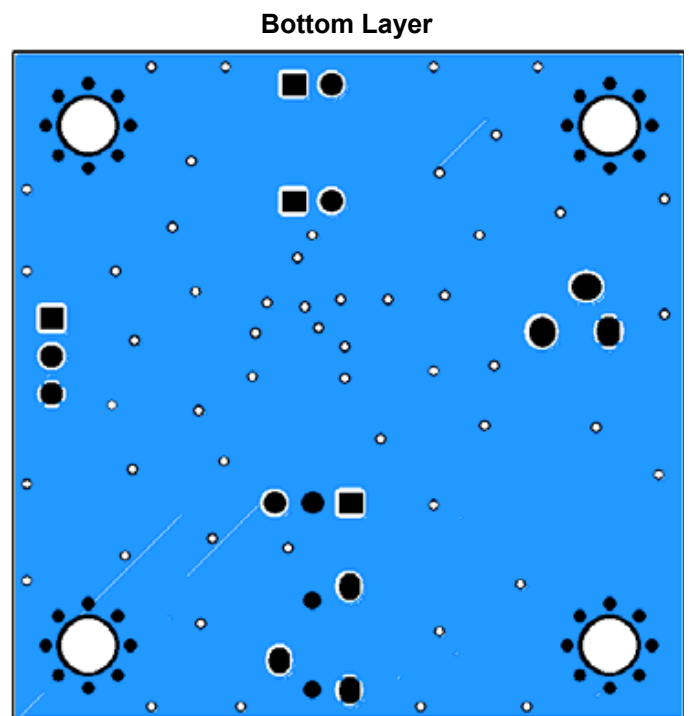


Figure 4